USER GUIDE

VERSION 15.0v3

FOUNDRY.

Foundry
5 Golden Square
London
W1F 9HT
UK

Rev: Thursday, January 18, 2024

Contents

Preface

Hiero, in conjunction with HieroPlayer, is designed to provide shot management, conform, and playback capabilities for people creating visual effects, especially those using Foundry's Nuke Compositor. Hiero combines well with Nuke to deliver visual effects sequences without resorting to other third party applications.

Key Features

The following are key features supported by Hiero and HieroPlayer:

- Conform multi-track timelines from CMX 3600 Edit Decision Lists (EDLs), OTIO, Final Cut Pro XML, and AAFs including cuts, transitions and retimes.
- Soft import any image sequence type supported by Nuke, meaning the media is not copied multiple times saving disk space. In addition, Hiero and HieroPlayer can also ingest audio and multi-channel images.
- Playback clips and sequences in real-time (dependent on hardware) on 2-up source/record Viewers, including multi-view or stereo footage.
- Version and snapshot clips and sequences to record progress and manage your creative options.
- Add real-time GPU soft effects directly on the timeline before export.
- Round-trip through Nuke or export general purpose shots using soft exports in most cases - no bakes are required.
- Accomplish most tasks through Python scripting that you can with the Hiero and HieroPlayer user interfaces.

About this Guide

Throughout this User Guide, we assume you have a basic knowledge of video and audio theory, as well as proficiency with the operating system on which the application is installed.

For the most up-to-date information, please see the relevant product page and the latest User Guide on our website at www.foundry.com.

Getting Help

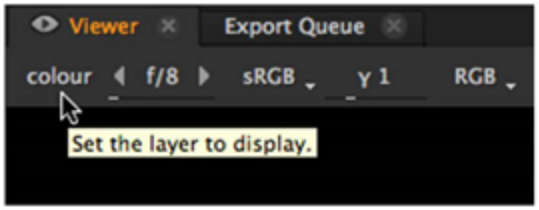
If you can't find what you need in this document, there are other sources of help available to you for all aspects of Hiero and its operation.

Customer Support

Should questions arise that this help fails to address, you can contact Customer Support by visiting support.foundry.com.

Online Help

Most controls offer concise instructions in the form of tooltips. To display the tooltips, hover your mouse pointer over an interface control or parameter.



Hiero's Help menu includes links to the latest Release Notes and www.nukepedia.com, the online knowledge base maintained by experienced Nuke users, containing downloads, tutorials, interviews and more.

There is also a user forum at community.foundry.com/discuss/hiero, set up as a hub for users to ask questions and share information.

Installation and Licensing

Installing and licensing new applications 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 a default workspace in front of you and are ready to start viewing and editing your media with the minimum of effort.

System Requirements

Before you do anything else, ensure that whichever platform you intend to run Hiero or HieroPlayer on meets the minimum requirements listed in the release notes for your version of the application. See the Hiero Release Notes page for more information.

Installing Hiero or HieroPlayer

Hiero and HieroPlayer are included in the Hiero application bundle, so you'll need to download the required Hiero version from www.foundry.com/products/ by choosing the correct platform from the list. For example, if you intend to install Hiero on Mac, download the Hiero **.dmg** file.

Follow the on-screen instructions to download the correct installer.

Mac

1. Double-click on the **.dmg** archive to extract the installation package.
2. Double-click on the **.pkg** file.
3. Follow the on-screen instructions to install the application. By default, Hiero (contained within the Hiero bundle) is installed to:
   `/Applications/Nuke<version>`

Linux

1. Extract the application from the **.run** archive with the following terminal command:
   `sudo ./Nuke<version>-linux-x86-release-64.run`
   The installer displays the End User Licensing Agreement (EULA) and prompts you to accept it.

2. If you agree with the EULA, enter **y** and press **Enter** to accept the EULA. (If you don't agree with the EULA and press **n** instead, the installation is canceled.)

   The selected application is installed in an appropriately named folder in the current working directory.

   If you leave out **sudo** from the terminal command, you need to ensure that you have sufficient permissions to install the application under your current working directory.

Note: After the application files have been installed, the installer also runs a post-installation script that creates the following directory:
`/usr/local/foundry/RLM`
If you don't have sufficient permissions on the **/usr/local** folder for this directory to be created, the post-installation script prompts you for your **sudo** password as necessary.

Windows

1. Download the .zip file from our website at https://www.foundry.com/products/nuke/download and extract the contents.
   In the extracted directory, you'll find an .msi executable and several .cab files, which are required for installation on Windows.
2. Double-click on the .msi file to start the installation. Follow the on-screen instructions. By default, the application is installed to:
   `C:\Program Files\Nuke<version>`

Launching Hiero or HieroPlayer

Launch Hiero or HieroPlayer using one of the following methods, depending on which platform you're using.

Mac

- Click the required dock icon.
- Using the Finder, open the Hiero application directory (by default, **/Applications/Nuke15.0v3/** and double-click the required **.app** icon (or list item).
- Open a terminal and enter the following command:

```
/Applications/Nuke15.0v3/Nuke15.0v3.app/Contents/MacOS/Nuke15.0v3 --hiero
```

OR

```
/Applications/Nuke15.0v3/Nuke15.0v3.app/Contents/MacOS/Nuke15.0v3 --player
```

Linux

- Double-click the required icon on the Desktop.
- Open a terminal, navigate to the **~/Nuke15.0v3** directory and enter:

```
./Nuke15.0 --hiero
```

OR

```
./Nuke15.0 --player
```

Note: The **~/** in the file path indicates the directory from which the installer was executed.

Windows

- Double-click the required icon on the Desktop.
- Using a command prompt, navigate to the application directory (by default, **\Program Files\Nuke15.0v3** or **\Program Files (x86)\Nuke15.0v3**), and enter:

```
Nuke15.0.exe --hiero
```

OR

```
Nuke15.0.exe --player
```

Hiero 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 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
- Memory speed
- Application version string
- Session duration (in seconds)
- Model
- MAC address
- GPU model name
- GPU driver version

Startup Options

If you choose to launch the application from a command line, you can append arguments to the command as follows:

Argument	Result
--help (-h)	Displays the available arguments with examples.
--version	Displays version and copyright information.
--script	Opens the specified script. For example: `./Hiero15.0v3 --hiero --script myscript.py` If the script resides in a different directory to the application, specify the file path as well.
--player (Hiero only)	Launches Hiero in HieroPlayer mode.
--log-file	Sets the location of any logfiles created. For example: `./Hiero15.0v3 --hiero --log-file /Desktop/log.txt` See to specify the logfile using an environment variable.
--log-level	Sets the level of logging produced during operation. For example: `./Hiero15.0v3 --hiero --log-level warning`

Argument	Result
	Log messages are output to screen unless you specify a **--log-file**. There are four levels of detail, on a sliding scale from minimal to verbose: • **error** • **warning** (default) • **message** • **verbose** **Note:** Setting the logging level to **verbose** can produce large log files when **--log-file** is specified. See to set the log level using an environment variable.
--quiet (-q)	Launch the application without displaying the splash screen or startup dialog.
--safe (-s)	Launch the application without loading any plug-ins, Export presets, and so on.
--single-threaded-playback	Launch the application in single-thread mode. This option can solve playback issues on various Linux Fedora flavors.

On Windows, you can also use the same arguments on executables (or their shortcuts) by adding them to the program's Properties. To use a startup argument:

1. Right-click the **C:\Program Files\Hiero15.0v3** icon or the equivalent icon on the Desktop.
2. Click **Properties**.
3. Append the required argument to the **Target** or **.exe** field.

Licensing Foundry Applications

Foundry applications support two types of license, cloud-based login licensing and local RLM licensing. Login licenses require an internet connection, but local RLM licenses are locked to a single machine or license server and don't require an internet connection.

Login Licensing

Cloud-based login licensing provides quick and easy access to all your licenses regardless of which machine you happen to be using. You can use your login license on any machine with internet access, giving you the freedom to work anywhere.

Note: Local RLM licensing is available alongside login licensing. You are not obliged to switch to the login-based model unless you're using an Indie or Non-Commercial license.

As well as **Individual** licenses, login licensing also supports **Team** licenses that give administrators greater control over who uses the licenses available in the team pool. You can quickly invite a team of artists to collaborate on a project with nothing more than their name and email address.

Individual Login Licenses

Individual Login licenses allow you to run Foundry applications on any machine using your Foundry account email and password. There's no need to manually install a license, the application takes care of all your license administration. If you don't have a Foundry account, you can create one by clicking **Sign up** on this page: https://www.foundry.com/user/register

Once you have a Foundry account, you can request an Individual 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 a Foundry application without a license, selecting Try or Buy a License, and then selecting:
  - **Try** - opens a web browser directly to our website's 30-day trial license screen. Enter your Foundry account details or create a new account and follow the on-screen instructions to receive a trial license.
  - **Buy** - opens a web browser directly to our website to purchase a license.

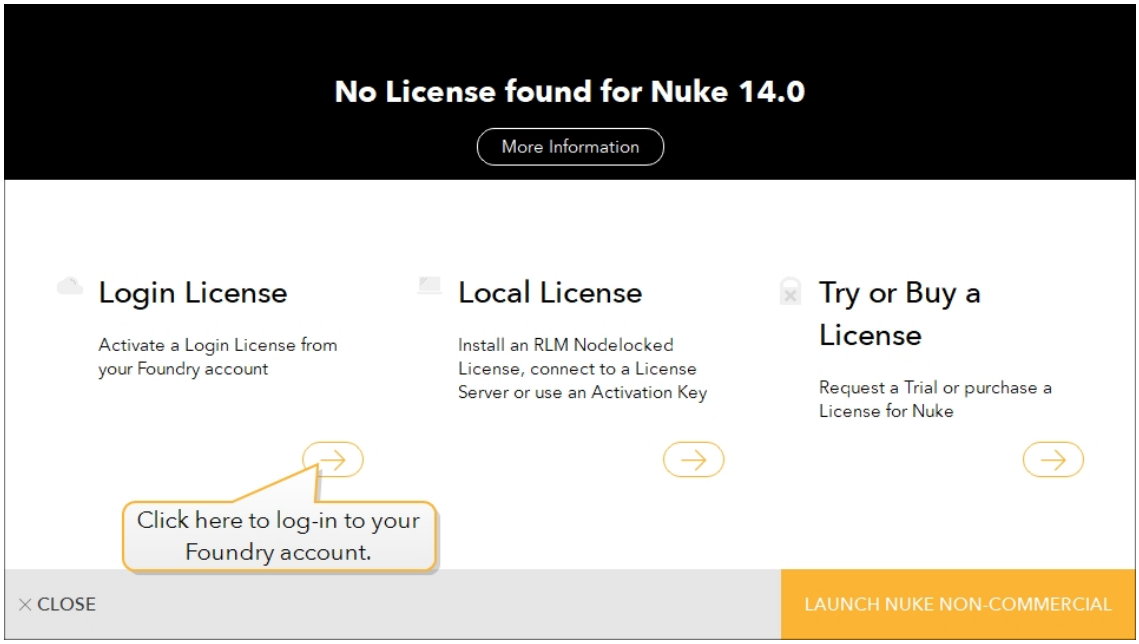
After purchasing a license, download your Foundry application from https://www.foundry.com/product-downloads

Note: If you were given an Activation Key, navigate to https://www.foundry.com/licensing/activate-product, enter the key, and follow in the onscreen instructions.

1. Launch your Foundry application as described in the Install Guide for your Foundry application by operating system.

   The Licensing dialog displays.

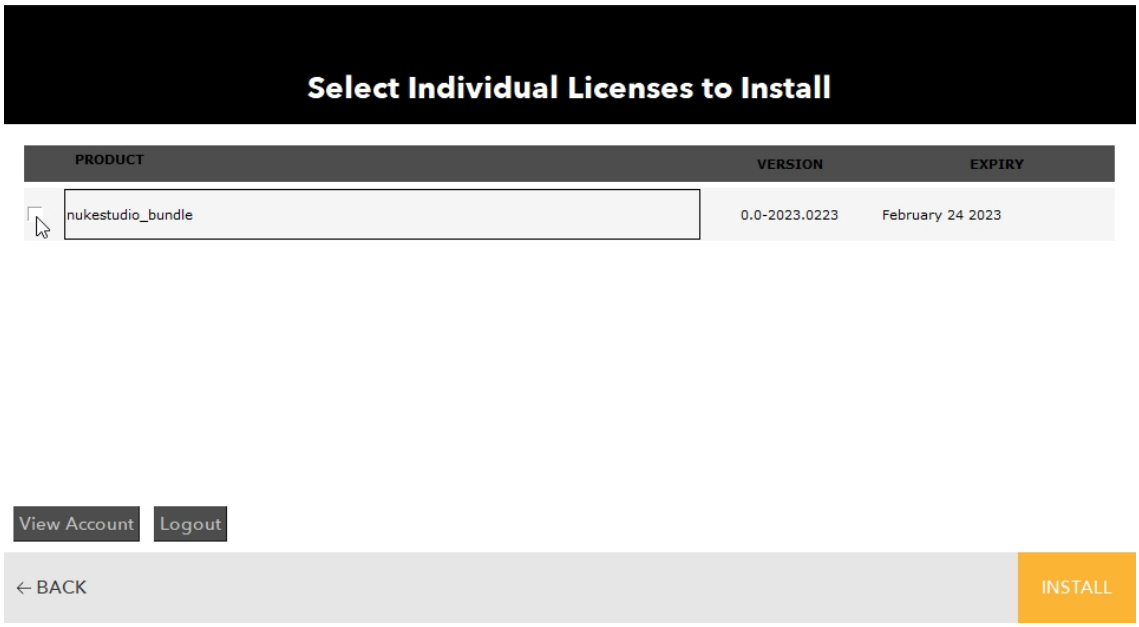
2. Select Login License to continue.



3. If this is the first time you've logged in to a Foundry application on this machine, enter your email address and Foundry account password, and then click Sign in.

   You don't need to enter your Foundry account information again unless you log out of an application.

4. Select the license you want to install from the list and click Install.



5. Click **Done** to close the Licensing dialog.

   You can now use your Foundry application.

Note: A license is activated for a period of 37 days. Make sure that you connect to the internet at least once during the activation period to extend your access. You can check the activation period of your license in your Foundry application's **Help** menu.

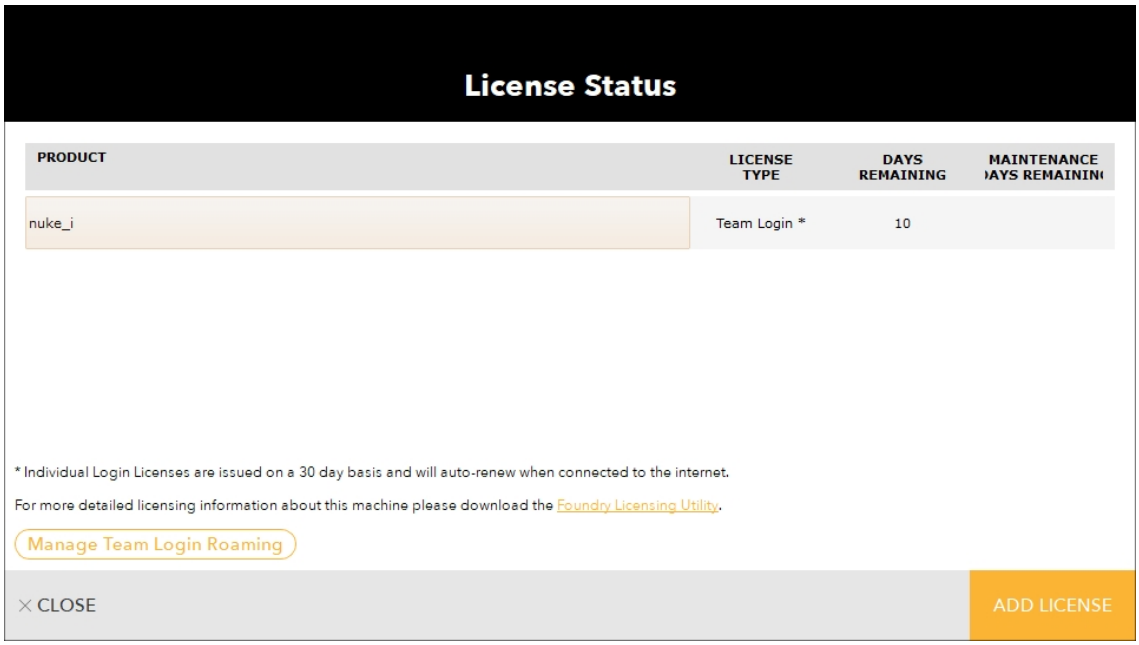
Deactivating an Individual License

An Individual license is active until you deactivate it or sign out of the Foundry Licensing application. Deactivation means you can activate the license again without needing to enter your email address and password.

Note: Individual licenses can be active on up to two machines concurrently. If you want to activate a license on a third machine, you must return an active license first.

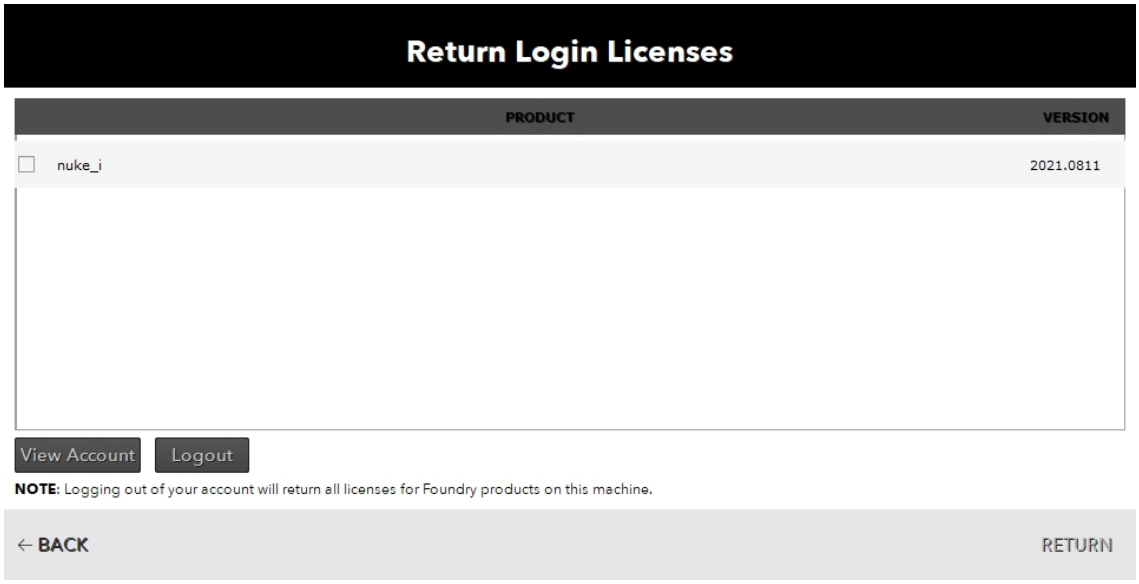
To deactivate a license:

1. Go to your application's Help menu and click Licenses.
2. The **License Status** dialog displays.



3. Click **Manage Login Licenses**.

   The Return Login Licenses dialog displays.



4. Select the license you want to deactivate and then click **Return**,

   OR

   Click Logout to return all of the activated licenses on this machine and clear your login details.

Tip: Returning a license allows you to activate it again without entering your Foundry login details.

Team Login Licenses

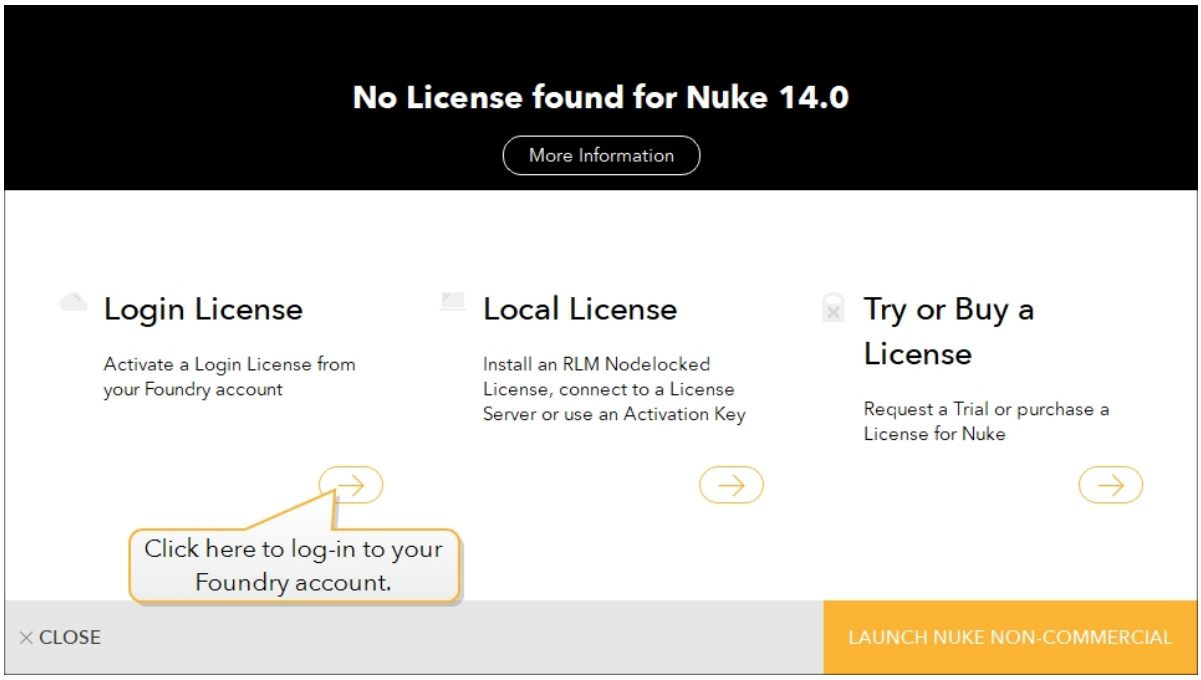
Team licenses also allow you to run Foundry applications using your Foundry account email and password, but the license is tied to team entitlements within the cloud-based login licensing system, rather than an individual's Foundry account. A license is taken from the pool of available licenses when you launch your Foundry application and the license is returned automatically when you close the application. Members of a team don't need to log-in and log-out to take and return licenses.

Team administrators can add and remove users from the team and control license access. See the Foundry Licensing website for more information on Team license administration.

1. Launch your Foundry application as described in the Install Guide for your Foundry application by operating system.

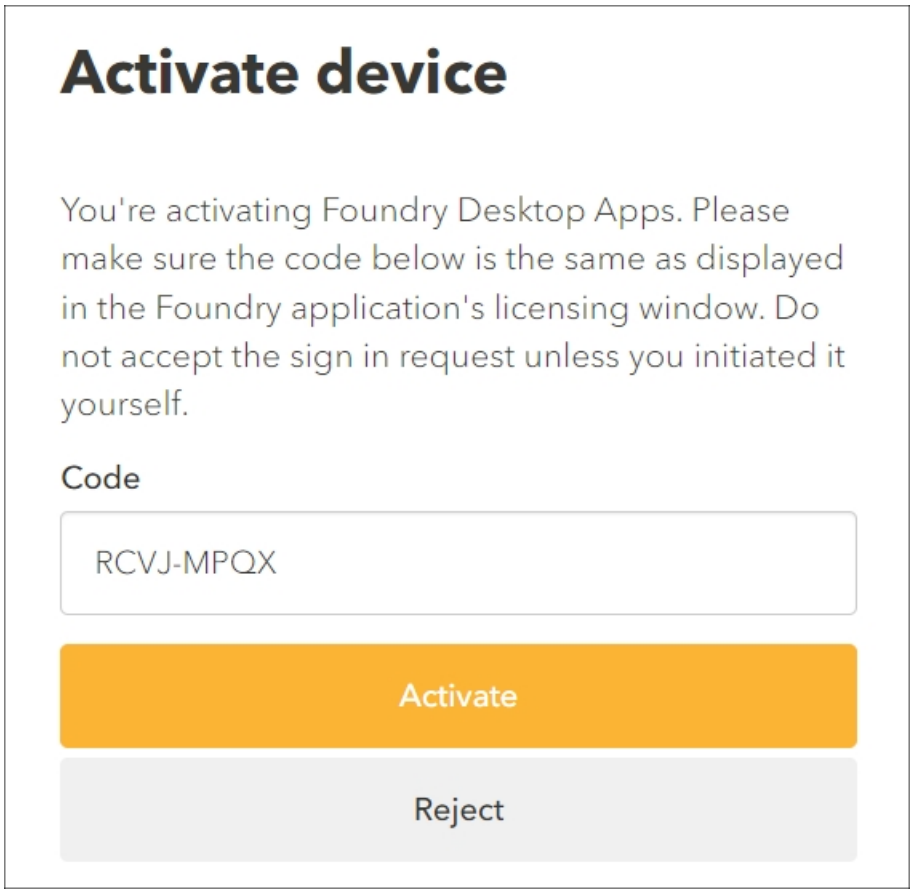
   The Licensing dialog displays.

2. Select Login License to continue.



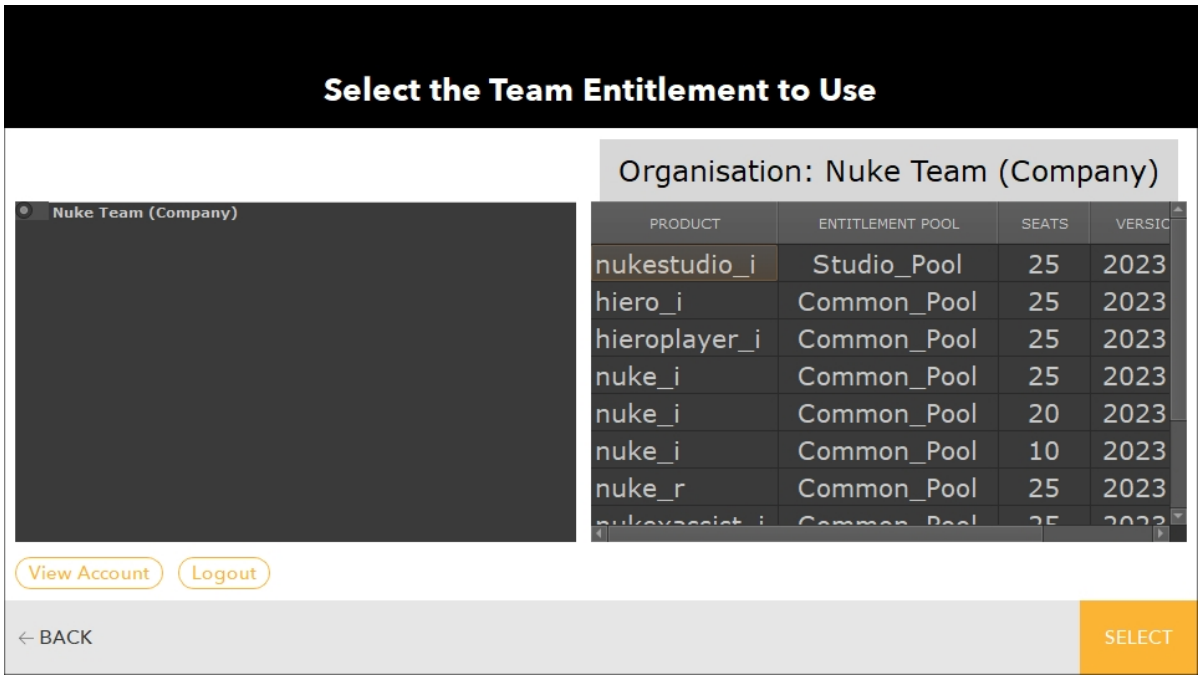
3. Enter your Foundry account information and click Sign in.

   You'll be asked to verify an activation code in a web browser.



4. Check that the code in the browser matches the code in the Licensing dialog and then click Activate.
5. Select Team to access your team entitlements.
6. Click the entitlement from which you want to take a license and click Select.

> **Note:** You can use any of the licenses in the selected entitlement, except those reserved by your administrator. See Foundry Licensing for more information on reserving licenses.



7. Click **Done** to close the Licensing dialog.

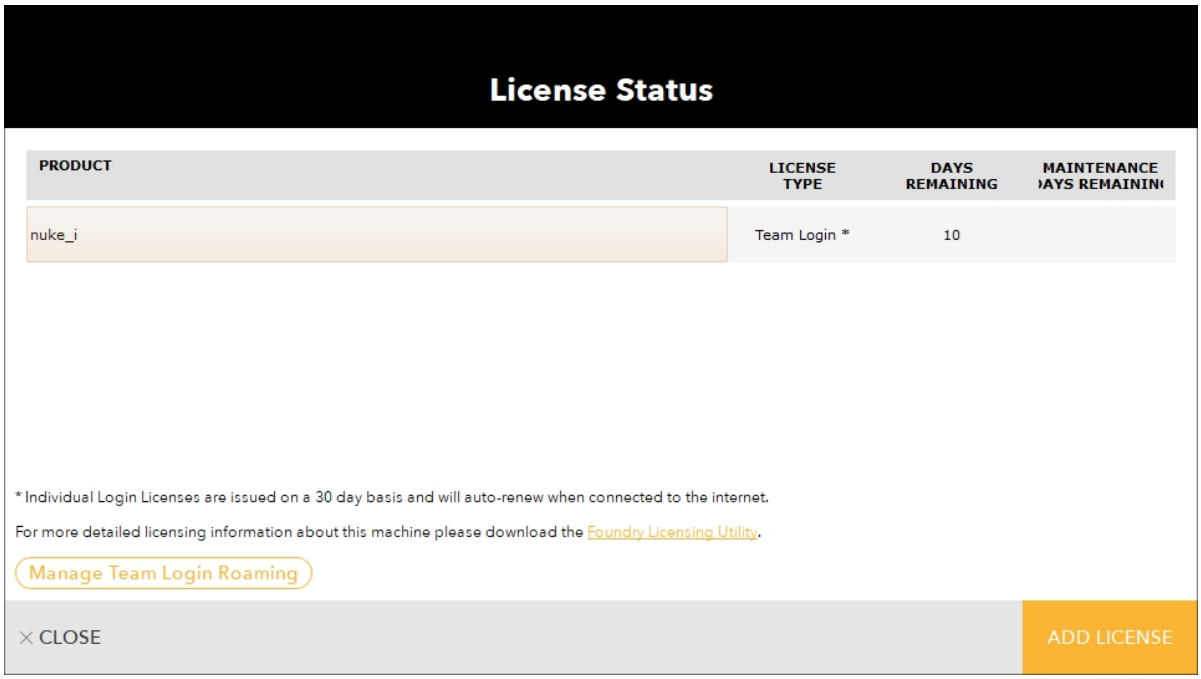
   You can now use your Foundry application.

Using a Different Team License or Logging Out

A Team license is active until you close your Foundry application, you don't need to return the license manually. You can use a license from a different Team on the same machine or you can logout to stop any licenses running on your machine.

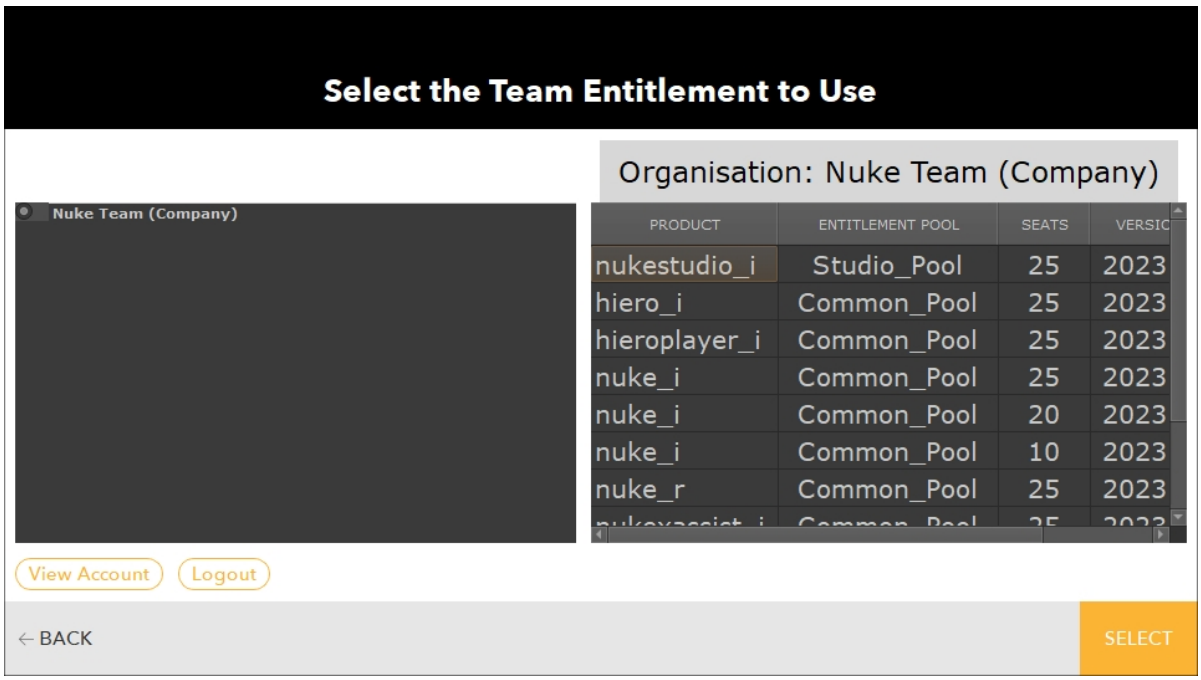
To use a different Team license:

1. Go to your application's Help menu and click Licenses.
2. The **License Status** dialog displays.



3. Click **Add License** and select **Login License**.

   The **Select the Team Entitlement to Use** dialog displays.



4. Click the entitlement from which you want to take a license and click Select.

> **Note:** You can use any of the licenses in the selected entitlement, except those reserved by your administrator. See Foundry Licensing for more information on reserving licenses.

5. Click **Done** to close the Licensing dialog.

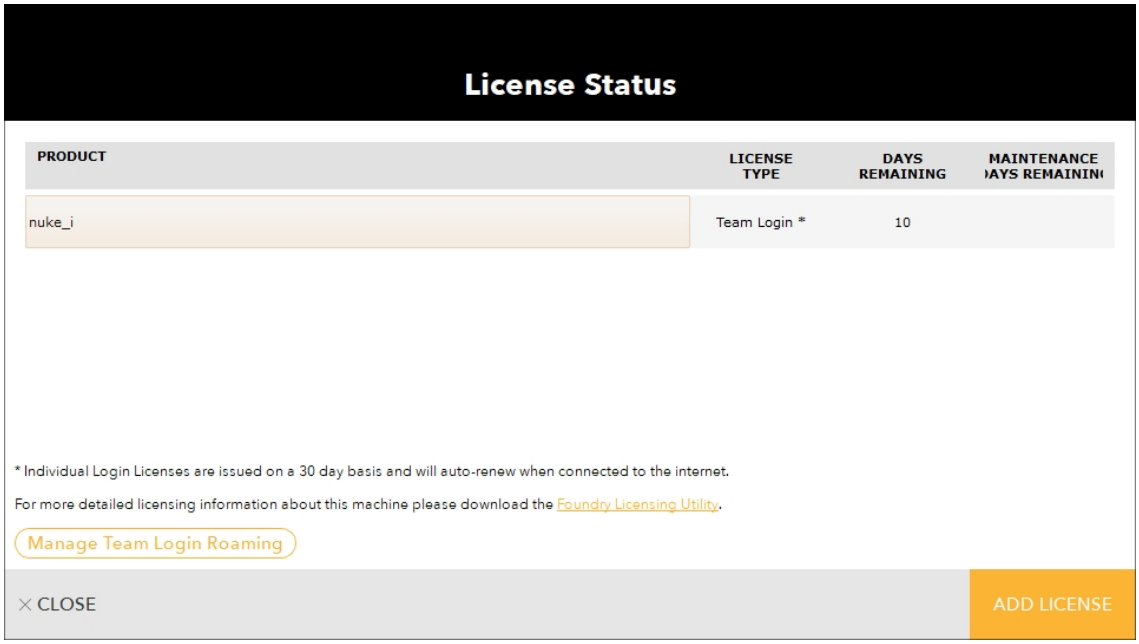
   You can now use your Foundry application with the new Team license.

   When you're finished working, close the application to automatically return the license to the pool.

To logout, return all of the activated licenses on this machine and clear your login details:

1. Go to your application's Help menu and click License.

   The **License Status** dialog displays.



2. Click **Add License** and then click Login License.
3. Click Team and then click **Logout** to return all of the activated licenses on this machine and clear your login details.

Local RLM Licensing

Local licensing using RLM software does not required internet access and so it is well suited to individuals that move around a lot and to site licensing where security considerations mean that there is no access to the internet.

Note: Local RLM licensing is available alongside login licensing. You are under no obligation to switch to the login-based model unless you're using an Indie or Non-Commercial license.

The following local licensing methods are available:

- **Activation Keys** - activation keys allow you to 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 application's **Licensing** dialog. Go to the Activate a Product page to use your Activation Key: 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 supplied as part of the Foundry Licensing Utility can be downloaded at no extra cost from https://www.foundry.com/licensing/tools

Obtaining Local Licenses

To obtain a local license, you'll need your machine's System ID (sometimes called the MAC address, Host ID, or rlmhostid). Just so you know what a System ID looks like, here's an example: `000ea641d7a1`.

Note: For local 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/tools and run it. Click **System ID** to display your computer's unique identifier.

When you know your System ID, you can request a local 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 Try or Buy a License and then selecting:
  - **Try** - opens a web browser directly to our website's 30-day trial license screen. Enter your Foundry account details or create a new account and follow the on-screen instructions to receive a trial license.
  - **Buy** - opens a web browser directly to our website to purchase a 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.

To use local license roaming you require a Roaming License. Licenses issued Before July 10th 2020 do not support roaming. New roaming licenses will be issued to customers at Maintenance Renewal time. If you are in Current Maintenance and would like to upgrade your licenses to take advantage of this new feature, please submit a request using this form: https://www.foundry.com/nuke-roaming-license

See Local License Roaming for more information on roaming.

Installing Local Licenses

We recommend using the Foundry Licensing Utility to install local licenses, available from https://www.foundry.com/licensing/tools free of charge. See https://learn.foundry.com/licensing/Content/install.html for more information on installing licenses.

However, you can also install local licenses after starting a Foundry application.

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 Local License from Disk or Local License Text 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 for more information.
- **A floating license** - also known as a server license. if you requested a server 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 Use a Floating Server License for more information.

To Install a Local License from Disk or Local License Text

1. Save the license file to a known location on disk.
2. Launch your Foundry application.

   The **Licensing** dialog displays.
3. Click Local License to display the available license installation options and then click License File.
4. Click **Open File**, browse to the location of the license file, and then click **Open** to install the license automatically in the correct directory,

   OR

   Copy the license text, paste it over the Drag & Drop or paste license texta License here string, and then click Install License.

   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="60P0450MJRP97E3DP
```

```
B42C99Y5UAPRMEMGNQ39PG22H4WGH3WFK2KPTXFWJTYR0GYASJBXC0PU8"
```

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

To Install an Activation Key

1. Launch your Foundry application.

   The **Licensing** dialog displays.
2. Click **Local License** to display the available license installation options.
3. Click **Activation Key** and then enter the Activation Key string into the key field.

   A license key typically looks something like this:
   `nuke-0101-77d3-99bd-a977-93e9-8035`
4. Click **Activate Key**.

   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 Use a Floating Server License

If you requested a server license from Foundry, you receive your license key (foundry.lic) in an email or download. You should also receive a link to 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.

To install a connection to your local license server:

1. Launch your Foundry application.

   The **Licensing** dialog displays.
2. Click **Local License** to display the available license installation options.

3. Click Server and then enter the port and server name or IP address of the license server.

   The format for the server name is: `<port>@<servername>`, for example, `30001@red`.
4. Click Connect.
5. 

   Note: You must perform these steps on each client machine that requires a license from the server. The client machines do not need the server tools installed.

Local License Roaming

License roaming allows you to temporarily check out a server license so you can work offline. Licenses are checked out and returned through the **Help** > **License** menu or by setting environment variables.

Note: License Roaming is not currently supported by Katana and Mari or with Production Collective and EDU Collective licenses.

There are a few important things to take into consideration with license roaming:

- Roaming is only available to Maintenance customers and requires a new license from Foundry.
- Roaming and returning licenses requires a connection to the license server.
- Roamed licenses are valid for a maximum of 30 days.
- Roamed licenses are only valid on the machine that took the license from the server.
- The number of roaming licenses available on the server decreases by one until you return the license.

Installing the Local Roaming License

To activate roaming licenses you need to request a new roaming license and install the license on your license server.

See Licensing Foundry Applications and To Use a Floating Server License for more details on requesting and installing a server license.

How to Roam a Local License

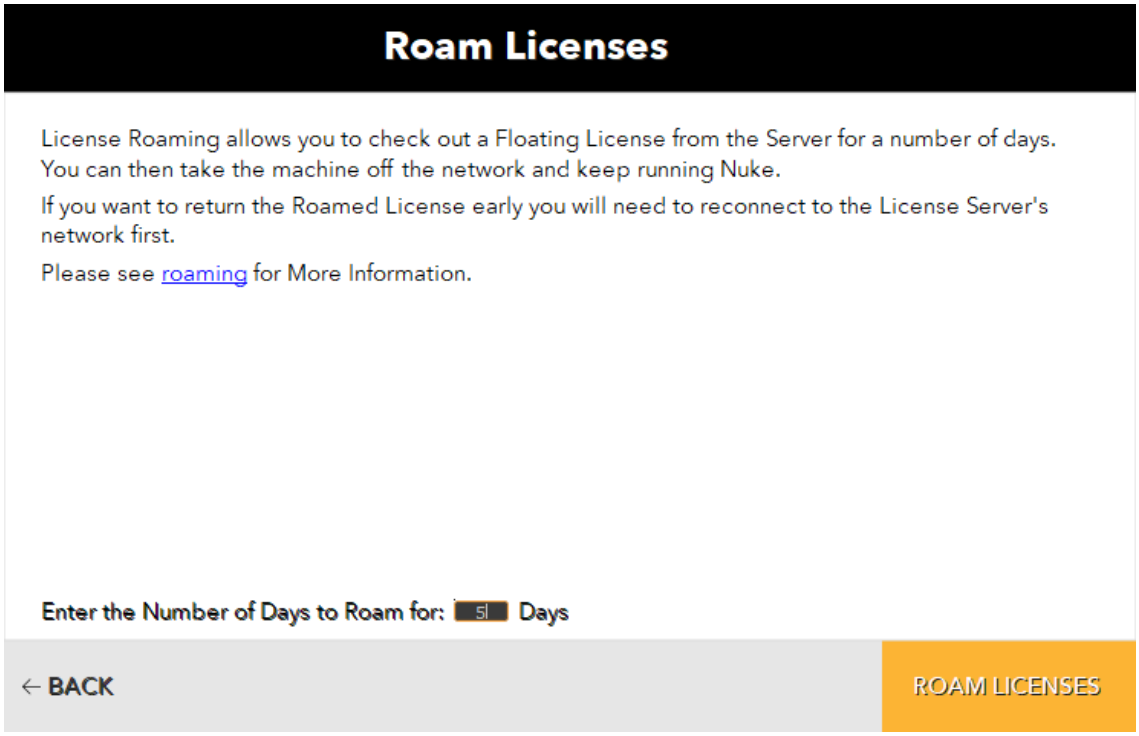
1. Launch the application you want to roam, making sure it has a connection to your license server.

2. Go to **Help** > **License**.

   The **Licensing** dialog displays.
3. Click **Manage Roaming**.

   The **Roam Licenses** panel displays.
4. Enter the number of days you want the roamed license to last, from 1 to 30 days.

> **Note:** The maximum number of roam days may be less than 30 depending on your license.



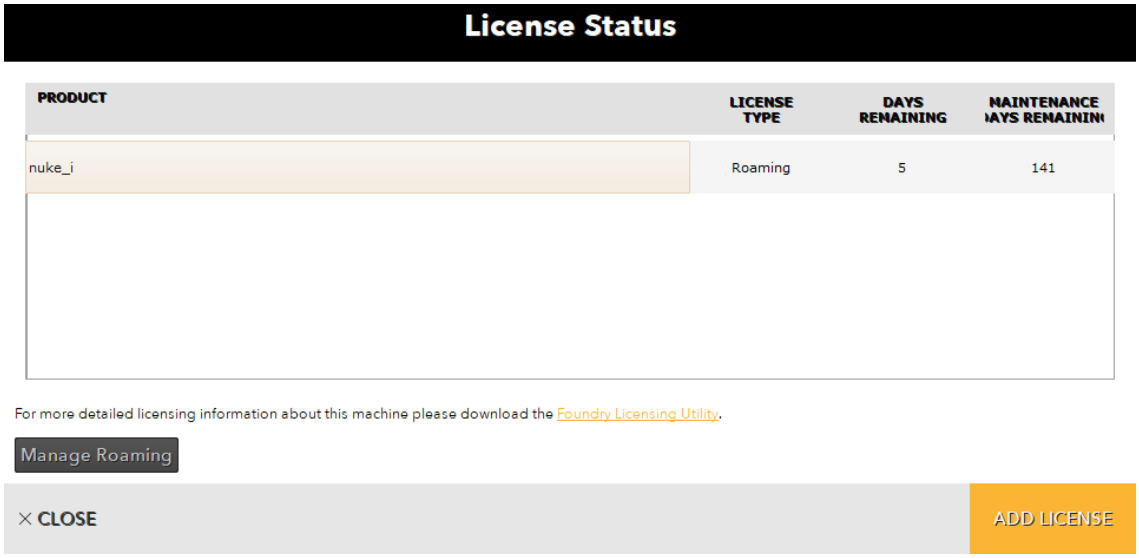
5. Click **Roam Licenses**.

   The panel displays the number of roaming days remaining.
6. Click **Close**.

How to Return a Roamed Local License

1. Launch the application using the roamed license, making sure it has a connection to your license server.
2. Go to **Help** > **License**.

The **Licensing** dialog displays the license type and number of roam days remaining.



3. Click **Manage Roaming**.
4. Click **Return Roam**.
5. Exit the application to return the roaming license to the server.

Roaming Using Environment Variables

You can automate some of the aspects of roaming licensing using the **RLM_ROAM** environment variable supplied by Reprise Software, the third-party vendor of our licensing administration software.

For more information see *How to use Roaming Licenses* in the *RLM Administration Guide*: https://www.reprisesoftware.com/RLM_License_Administration.pdf

Further Reading on Licensing

There is a lot to learn about licenses, much of which is beyond the scope of this manual. For more information on cloud-based login licensing and local RLM licensing, go to our Licensing website: https://learn.foundry.com/licensing/

Uninstalling Hiero or HieroPlayer

To uninstall Hiero or HieroPlayer, following the instructions below, depending on what platform you're running.

Mac

1. Navigate to **Applications** and delete the **Hiero 15.0v3** folder.
2. Delete, rename, or move your **.nuke** folder, if it exists.
   The **.nuke** folder is found in your home directory, by default:
   `/Users/<login name>/.nuke`

Note: The **.nuke** folder may be a hidden folder on your machine. To allow your Mac to display hidden files and folders, 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 Hiero Environment Variables for more information.

Linux

1. Navigate to **/usr/local/** and delete the **Hiero 15.0v3** folder.
2. Delete, rename, or move your **.nuke** folder, if it exists.
   The **.nuke** folder is 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 Hiero Environment Variables for more information.

Windows

1. Navigate to the install directory for the required version. By default, the application is installed to:
   `C:\Program Files\Nuke15.0v3`
2. Double-click uninstall.bat.

   The Uninstall dialog displays.
3. Click Yes to uninstall.

> **Tip:** You can also uninstall silently, with no prompts or popups, by running the Command Prompt as an Administrator and entering:
> `"C:\Program Files\Hiero15.0v3\uninstall.bat" /passive`

4. Delete, rename, or move your .nuke directory, if it exists.

   The .nuke directory is usually found under the directory pointed to by the HOME environment variable. If this variable is not set (which is common), the directory is under the directory specified by the USERPROFILE environment variable, which is generally one of the following:
   `C:\Documents and Settings\login name\`
   `C:\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.
5. 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.

6. Remove any application-related environment variables. See Environment Variables for more information.

Hiero and HieroPlayer Workflow

This section describes the main differences between Hiero and HieroPlayer, and an example collaborative workflow using Nuke.

Hiero

In a typical collaborative project environment, Hiero sits in between edit and grading, and finishing and delivery, acting as a hub for visual effects. It is used to locate, preview, and then farm media out to shot-based effects applications (such as Nuke), and take their results and re-integrate them into the sequences for review and comment.

HieroPlayer

HieroPlayer sits next to your compositing and other shot creation software, allowing artists to view their work in the context of a timeline. This collaborative workflow allows multiple artists to work on the same timeline, viewing colleague's work as well as their own. See Collaborative Project Case Study for more information.

HieroPlayer is a free annual subscription with your Nuke or NukeX license. Simply renew your existing Nuke or NukeX license to receive your HieroPlayer entitlement or fill in the on-demand form.

See HieroPlayer Free with Nuke for more information.

Feature Comparison

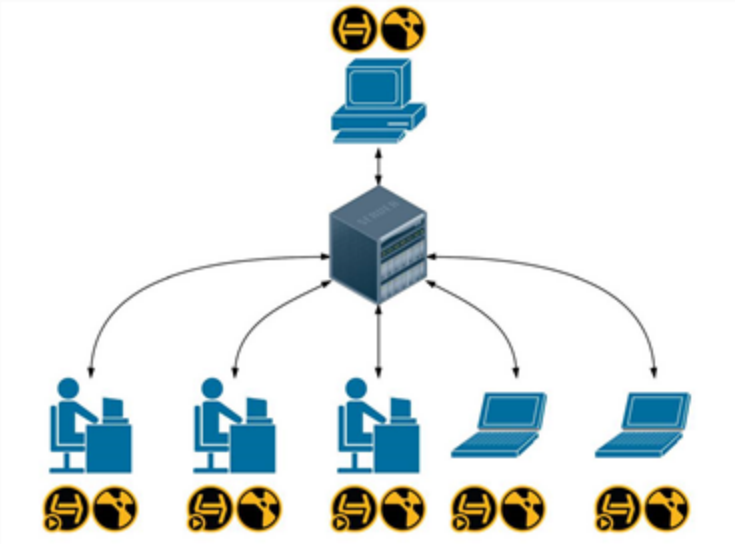
Designed to work hand-in-hand in different scenarios, Hiero and HieroPlayer share a number of features, but as the hub, Hiero has a richer feature set than HieroPlayer.

Feature	Hiero	HieroPlayer
Review and Playback - Robust playback & review engine for post production		
Multi-track timeline viewing and editorial workspace, including industry standard tools	●	●
Nuke equivalent format support including OpenColorIO management	●	●
Multi-channel audio import and playback	●	●
Extensive review toolset including A/B compare, masks, and color picker	●	●
Realtime image scopes including histogram, waveform, and vector	●	●
Media localization and caching	●	●
Standard jog and shuttle controls	●	●
Track Blending for merging of tracks on the timeline	●	●
Annotations in the Viewer and exports for reviewing work in collaborative projects	●	●
Broadcast Monitor support	●	
Editing - Standard tools and layouts for editorial.		
Modal editorial tools including equivalent keyboard hotkey interface	●	●
Source/record 2-up Viewer workspace and 3-point editing	●	●

Feature	Hiero	HieroPlayer
Conform - Simple and flexible ingest of cuts from editorial.		
Conform multi-track timelines from CMX 3600 Edit Decision Lists (EDLs), OTIO, Final Cut Pro XML, and AAFs including cuts, transitions and retimes	●	
Session-wide spreadsheet containing all sequence events	●	
Shot Management and Export - Manage, distribute and ingest media with ease		
Export sequences, timeline shot structures, clip hierarchies, and EDL/OTIO/XMLs	●	
Soft Effects directly on the timeline, such as Crop, LUT, and Transform.	●	
Transcode, copy, and soft link media	●	
Create template Nuke scripts for easy distribution of work	●	
Automatically update VFX shots in context	●	●
Version shots and snapshot timelines for review and editing	●	●
Hierarchical project and element library management	●	●
Tag shots, frames, and sequences for easy filtering of content including custom metadata	●	●
Open and Scriptable Playback Framework - Flexible enough to fit any pipeline		
Extensive Python API to accomplish most tasks through Python scripting that you can with the Hiero and HieroPlayer user interfaces	●	●
Integrated Pyside enabling fine-grained customization of interface tasks	●	●

Collaborative Project Case Study

Hiero can work effectively as the hub for managing content in a post pipeline. The diagram below shows a typical workgroup scenario, with Hiero and Nuke at the top as the hub managing a number of Nuke workstations employing HieroPlayer as a review tool.



Preparing a Project in Hiero

The first step in Hiero is to conform the EDL, OTIO, or XML from the editor. The conform process pulls together the required source clips and matches them to events in the EDL, OTIO, or XML. Hiero displays these events in a spreadsheet which is linked to a timeline showing the corresponding shots. See Conforming Using Hiero for more information.

Once the conform is complete, you can view your timeline to confirm that the correct clips are present, check the content using Hiero's scopes, add reference tracks, and pass on any pertinent information to the next step in production using tags and notes. See Ingesting Media for more information.

The next step is to make any minor edits to the timeline, including transitions and retimes, swap or rename shots, or examine different versions of clips on the timeline. See Managing Timelines and Versions and Snapshots for more information.

The final step before HieroPlayer steps in is to export and save your project. Exporting from Hiero and saving the project creates placemarker shots for VFX work in a read-only Hiero project (**.hrox** file), preserving the conform work from Hiero. Multiple artists can then work on shots, creating versions which are then automatically updated in HieroPlayer. See Round-Tripping Using the Export Dialog for more information.

Loading a Hiero Project in HieroPlayer

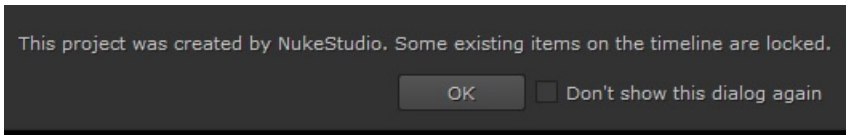
HieroPlayer is most commonly used to open Hiero projects as read-only copies to preserve conform work from Hiero. This way, multiple artists can work on a project without worrying about overwriting a colleagues work.

To open a Hiero project in read-only form:

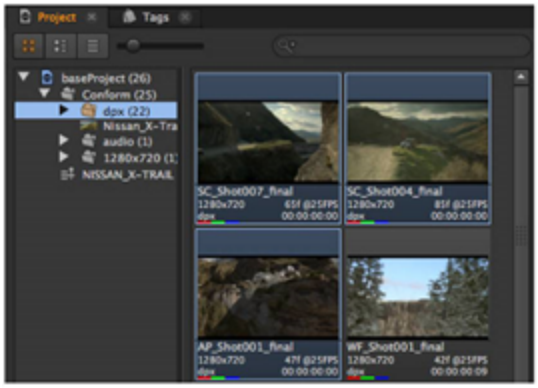
1. Launch HieroPlayer and navigate to **File** > **Open**.

> **Tip:** You may find that the project you require is displayed in the startup dialog under **Recent Projects**.
> Select the project and click **Open** to quickly access the project.

2. Locate the required Hiero project using the browser and click **Open**.
   A warning displays informing you that the project is read-only, as it was created in Hiero.



3. Click **OK** to dismiss the prompt.
4. Opening a Hiero project in HieroPlayer changes the interface highlight color to light blue to indicate that the project is read-only.



You can't delete bins, clips, or shots, or make edits on the timeline when the highlight is light blue. You can, however, drag source clips on to new tracks on the timeline.

5. To save your work, navigate to **File** > **Save as Player Project**. HieroPlayer cannot overwrite Hiero projects.

Opening a Writable Hiero Project

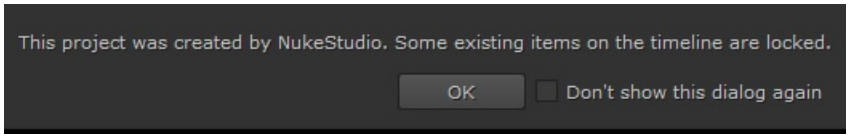
If you want to open a writable version of a Hiero project, you can open the project and save it as a HieroPlayer project. HieroPlayer projects are always readable in Hiero, so don't worry about backward compatibility.

To open a Hiero project in writable form:

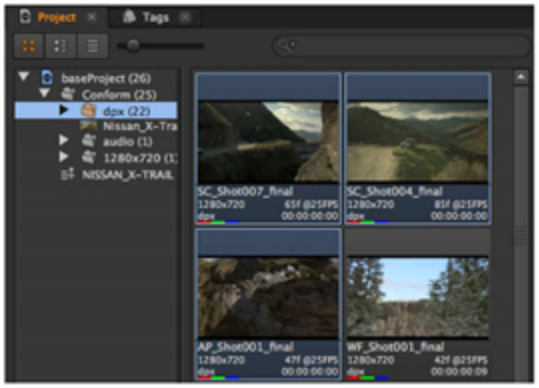
1. Launch HieroPlayer and navigate to **File** > **Open**.

> **Tip:** You may find that the project you require is displayed in the startup dialog under **Recent Projects**.
> Select the project and click **Open** to quickly access the project.

2. Locate the required Hiero project using the browser and click **Open**.
   A warning displays informing you that the project is read-only, as it was created in Hiero.

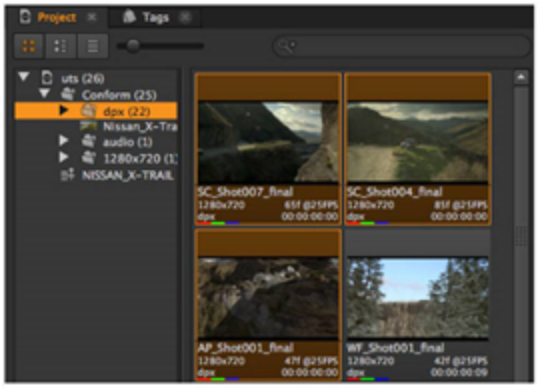


3. Click **OK** to dismiss the prompt.
4. Opening a Hiero project in HieroPlayer changes the interface highlight color to light blue to indicate that the project is read-only.



   You can't delete bins, source clips, or shots or make edits on the timeline when the highlight is light blue.

5. Navigate to **File** > **Save as Player Project** and select a save location.
6. The highlight color changes to orange to indicate that the project is now writable.


Project
Tags
uts (26)
Conform (25)
dpx (22)
audio (1)
SC_Shot007_final
SC_Shot004_final
AP_Shot001_final
WF_Shot001_final

Customizing Your Workspace

The application interfaces are customizable using floating panes and show/hide functionality, but several default workspaces are supplied for your convenience.

Workspace Overview

Hiero and HieroPlayer ship with a number of different workspaces. These are specific layouts that can be loaded as the current tasks change.

Hiero ships with the following workspaces:

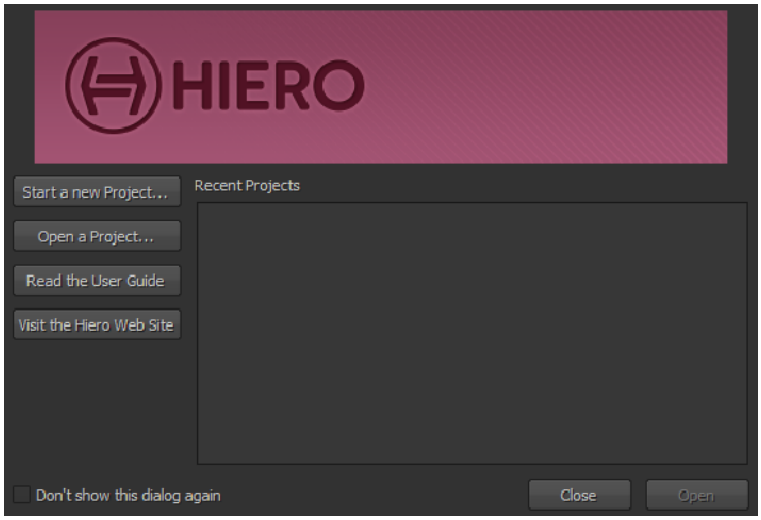
- Hiero's **Conforming** workspace is where the bulk of the work you undertake with Hiero is done. The linked spreadsheet and timeline panes are where you'll really see the power of Hiero.
- The **Editing** workspace provides the tools you'll need to fine-tune your media in Hiero, including 2-up source/record Viewers.
- The **Reviewing** workspace is shared by both applications and is used to confirm your media intent is fully realized. You can add tags and notes, mark In and Out points on clips and timelines, and organize your ingested media into bins as required.
- The **Timeline** workspace is similar to the Conforming workspace, but doesn't include the spreadsheet pane.

HieroPlayer ships with the following workspaces:

- The **Editing** workspace provides the tools you'll need to fine-tune your media, including 2-up source/record Viewers.
- The **Flipbook** workspace consists of a Viewer and Project tab and is used primarily as a review tool in HieroPlayer.
- The **Reviewing** workspace is shared by both applications and is used to confirm your media intent is fully realized. You can add tags and notes, mark In and Out points on clips and timelines, and organize your ingested media into bins as required.
- The **Timeline** workspace is similar to the Editing workspace, but only include a single source Viewer.

Note: HieroPlayer doesn't include any Conform functionality and projects loaded from Hiero can't be edited unless they're resaved using **File** > **Save as Player Project**.

When you launch either application, you're presented with a startup dialog.

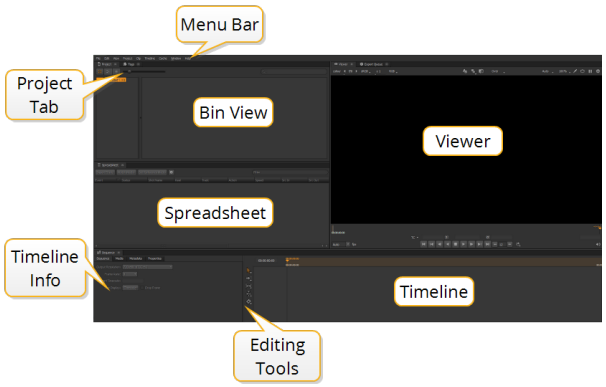


The startup dialog allows you to create a new project, open an existing project, or view the User Guide and website dedicated to Hiero.

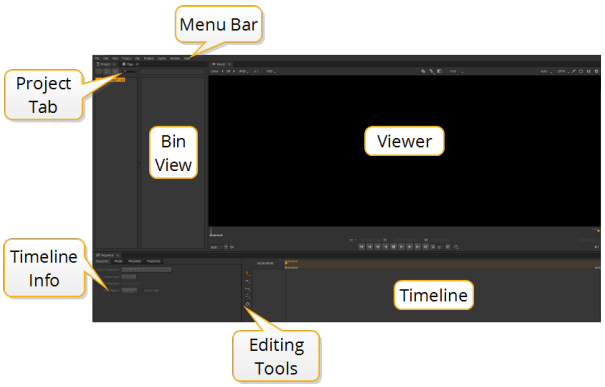
Tip: You can disable this dialog by checking **Don't show this dialog again** or by toggling it off in the dialog.

Select the required option or click **Close** to access the application. Take a moment to familiarize yourself with the default workspaces.

Hiero's default **Conforming** workspace:



HieroPlayer's default **Timeline** workspace:



Feature	Description	Hiero	HieroPlayer
Project Tab	You can manage all aspects of your projects and bins	●	●

Feature	Description	Hiero	HieroPlayer
	in the Project tab.		
Menu Bar	Use the Menu bar to access Hiero'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.	●	●
Editing Tools	There is a comprehensive set of editing tools provided in Hiero and HieroPlayer. See Timeline Editing Tools for more information.	●	●
Spreadsheet Tab	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.	●	
Timeline	The timeline displays the current track including all shots and any effects that have been added.	●	●

Menu Bar Components

The Menu Bar contains various controls, many of which are available through right-click menus, keyboard shortcuts, and toolbars.

For a full list of keyboard shortcuts, buttons, and menu functions, seeAppendix B: Keyboard Shortcuts.

You can switch between workspaces by navigating to **Window** > **Workspace** and selecting **Conforming**, **Editing**, **Reviewing**, **Flipbooking**, or **Timeline**. Alternatively, you can toggle between the default workspaces using **Shift**+**F1**, **Shift**+**F2**, **Shift**+**F3**, and so on.

Any custom workspaces you create are added to the end of the list. You can access the first custom workspace using **Shift+F5**, and so on.

Creating Your Own Workspace

Hiero's interface is comprised of any number of panels. Panels act like containers with different tabs; you can configure panels to contain specific tools, such as the timeline or a Viewer, depending on the task you are presently performing. The default panel configurations are saved as Workspaces (referred to as Layouts in Nuke) and include: Conforming, Editing, Reviewing, Flipbooking, and Timeline. You can easily create your own custom workspace that suits your needs, and then save it as a custom Workspace for easy recall.

During the customization process, you can:

- Resize your entire workspace or panes independently.
- Split panes to create new work areas, for example two Viewers side-by-side.
- Add and remove panes and tabs as required.
- Float and nest tabs to group similar functions together, for example Histogram, Vector, and Waveform in the same pane.
- Maximize the pane under the mouse cursor by pressing ` (back tick). Press ` again to return to the regular interface.

You can create as many workspaces as you require, one for each project you undertake if necessary. In practice though, you may find that two or three custom workspaces are sufficient.

Tip: You can return to the saved version of a workspace by navigating to **Window** > **Workspace** > **Reset Workspace**.

Resizing Your Workspace

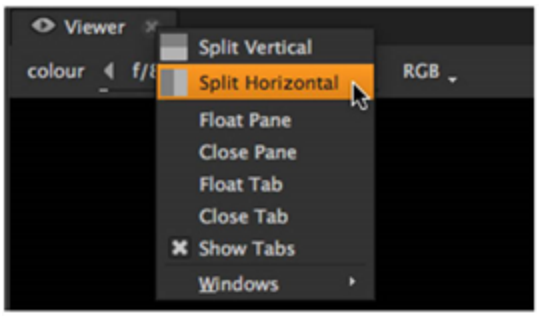
Resizing the interface, panes, and tabs works in much the same way as other software applications:

- To resize the entire interface, drag the resize handle at the bottom right of the application to the required width and height.
- To resize individual panes, hover the mouse over the edges of the required area until the cursor changes to the resize icon. Click and drag the cursor to resize the pane or tab.

Splitting Panes

Splitting panes allows you to organize your workspace into distinct areas delineated by well-defined borders.

• To split panes, right-click the required pane and select **Split Vertical** or **Split Horizontal**.

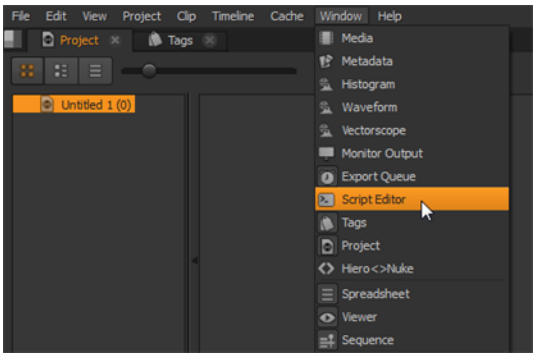


Adding and Removing Panes and Tabs

Adding and removing tabs gives you the freedom to personalize your workspace depending on your current project.

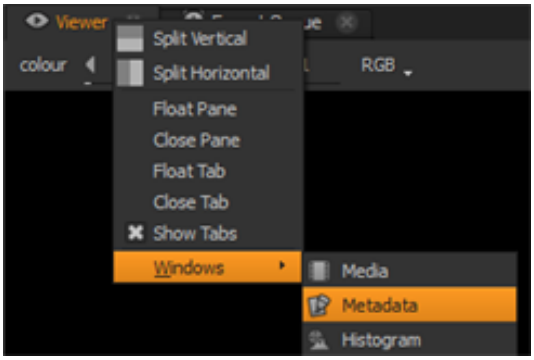
• Add new tabs by:

  • Navigating to **Window** and selecting the required tab. The tab is embedded in its default position or added to the interface as a floating window,



  OR

  • Right-clicking the pane in which you want the tab to appear and selecting it from the **Windows** dropdown.

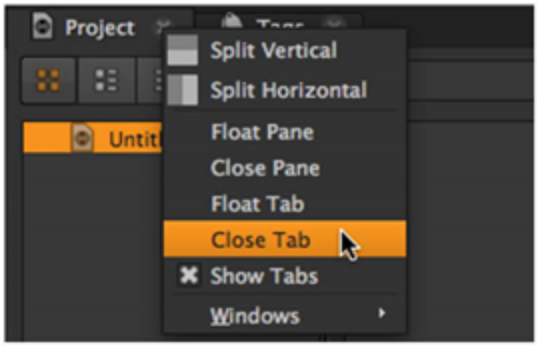


• Close any tab that is not required by:

- Clicking the **x** on the tab's right-hand side,

OR

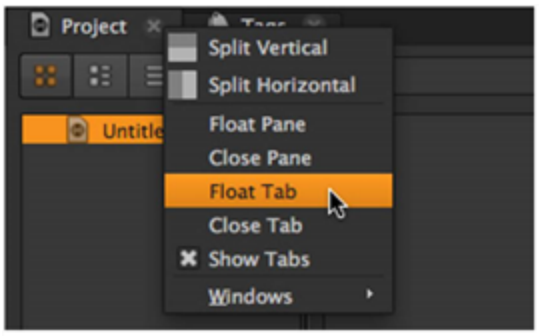
- Right-clicking the tab name and selecting **Close Pane** or **Close Tab**.



Floating and Nesting

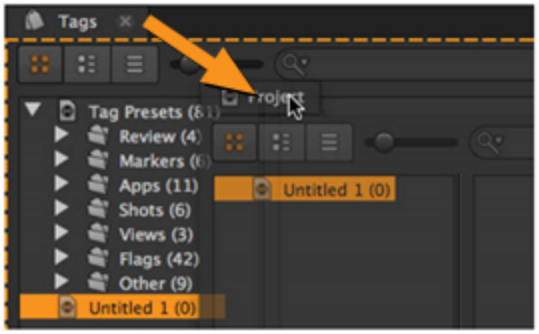
You can move existing panes and tabs to new locations in the interface (to help you organize your project).

- Float a pane or tab by:
  - Right-clicking the tab name and selecting **Float Pane** or **Float Tab**.

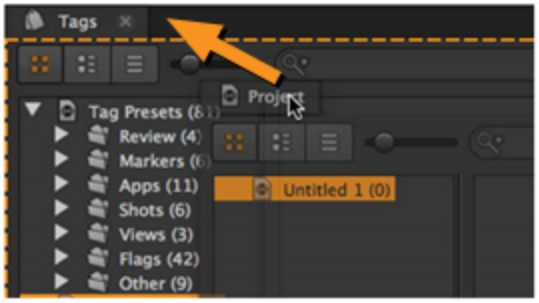


OR

- Dragging the required tab from its current location.



• Nest any floating pane or tab by dragging it to any existing pane.

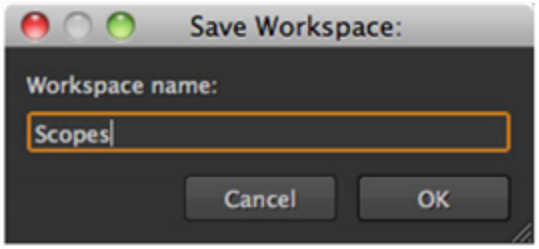


Tip: Use the orange highlight around the destination pane to help you determine where the tab nests.

Saving Workspaces

Once you're happy with the changes you've made to the workspace, navigate to **Window** > **Workspace** > **Save Workspace**.

Enter a name for the workspace, then click **OK** to preserve your workspace as a workspace file (extension **.hws**).



Tip: If you save a workspace called **Conforming**, **Editing**, **Reviewing**, **Flipbooking**, or **Timeline** you can override the defaults supplied with either application by clicking **Save** in the dialog box.

Delete the workspace you created to restore the default layout.

Loading Workspaces

To use a previously saved workspace, navigate to **Window** > **Workspace** and select the custom workspace name you require.

Tip: Alternatively, you can use the workspace keyboard shortcut **Ctrl/Cmd**+**n** (where **n** is equal to the number of the custom workspace).

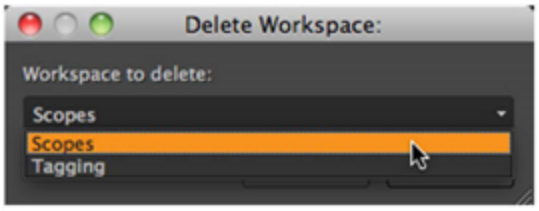
Your pre-saved workspace is loaded.

Deleting Workspaces

You may find that after using Hiero or HieroPlayer on a number of projects, some of the workspaces you created earlier have been superceded by more useful entries. **Delete Workspace** allows you to clear up your saved files, although you can remove the files manually from the save location.

To delete custom workspaces:

1. Navigate to **Window** > **Workspace** > **Delete Workspace**.
2. Select the required workspace from the dropdown menu.



3. Click **OK** to delete the selected entry.

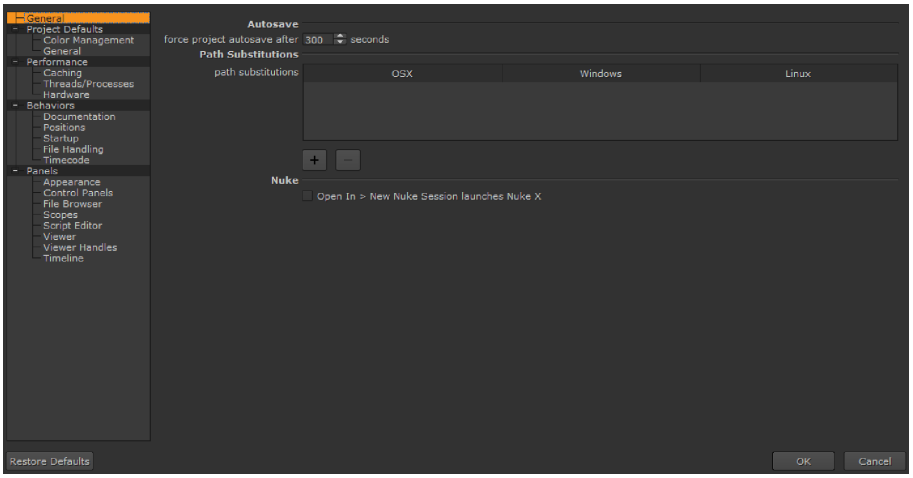
Workspace Preferences

The **Preferences** dialog allows you to make global changes to the way the interface behaves every time you start Hiero or HieroPlayer. Changes you make in the **Preferences** override any local changes when you restart the application. See Appendix A: Preferences

Note: Workspace **Preferences** are saved in the **~/.nuke/uistate.ini** file.

To access the **Preferences** dialog, either:

- Navigate to **Hiero** (**HieroPlayer**) > **Preferences** (OS X) or **Edit** > **Preferences** (Linux and Windows), or
- Use the Preferences hotkeys **Shift**+**S**.



Once you've made your selections in the **Preference** dialog, click **OK** to save your changes. Bear in mind that changing some preferences, for example **Scopes**, require you to restart Hiero before they're applied.

Note: Click **Restore Defaults** to return to the default **Preference** settings.

Optimizing Read and Decode Performance

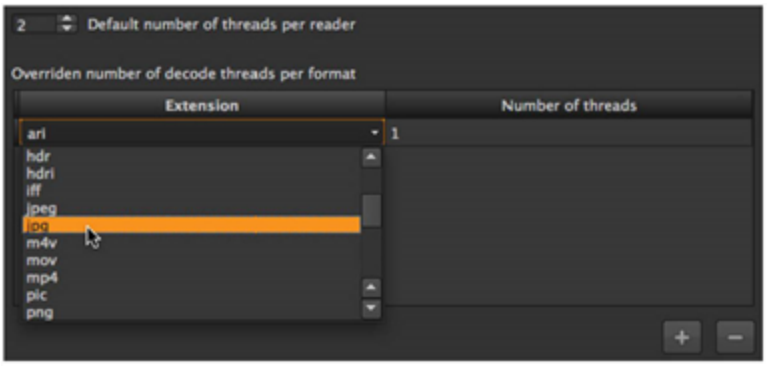
The **Performance** preferences enable you to get the best the results from the hardware you're using by specifying the number of "threads" used for reading and decoding media.

A thread is, at its most basic level, a series of operations (in this case reading and decoding frames) that operates concurrently with other sets of operations. Most media is read intensive, but highly compressed formats like **.jpg** are decode intensive. Increasing the number of threads can decrease the time it takes to perform an instruction. For example, decoding a **.dpx** file using 2 threads, instead of 1, decodes 2 frames in parallel.

Hiero defaults to 4 threads per reader, but as a general rule of thumb, you shouldn't increase the number of threads above the number of cores in your machine.

You can override the **Default number of threads per reader** on a format-by-format basis using the override table in the **Preferences**.

1. Click the + icon to add an entry to the table.
2. In the **Extension** column, click the dropdown to select the file extension to override.



3. Enter the **Number of threads** to use for the selected format.

   For example, **.jpg** files rely heavily on decoding so an override setting between 6 and 8 is recommended. Setting the default threads per helper to 6 is not advisable, except on high specification hardware.
4. Click **OK** to close the **Preferences** and apply the setting immediately, there is no need to restart the application, so feel free to experiment to achieve the best combination for your setup.

Using Helper Threads

Hiero and HieroPlayer also include two additional helper thread settings dedicated to OpenEXR and ARRI files. These helper threads assist the reader threads to improve performance for **.exr** and **.ari** files.

You can change the number of available helpers using the up and down arrows, or by simply entering the number of required helpers in the fields provided under **Preferences** > **Performance** > **Threads/Processes**.

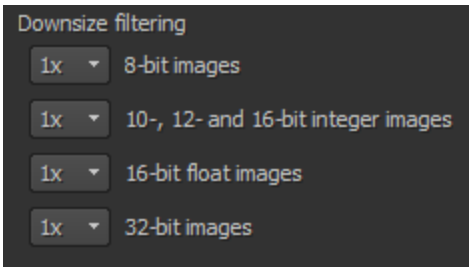
In general, the default setting **0** should automatically assign helper threads correctly. If, however, your project is **.exr**/**.ari** heavy you might consider increasing the number of helper threads.

Note: The application must be restarted to apply changes to the ARRI helper setting.

Using Downsize Sampling

The Viewer **Image Quality** dropdown affects the decode rate and resolution of clips displayed in the Viewer. Lower resolutions decode faster and vice versa.

You can customize this behavior by bit-depth using the **Preferences** > **Performance** > **Threads/Processes** > **Downsize filtering** controls.



Ingesting Media

Hiero handles many file formats, including embedded audio and **.wav** files. For a full list of supported file formats , see Appendix C: Supported File and Camera Formats.

About Clips and Shots

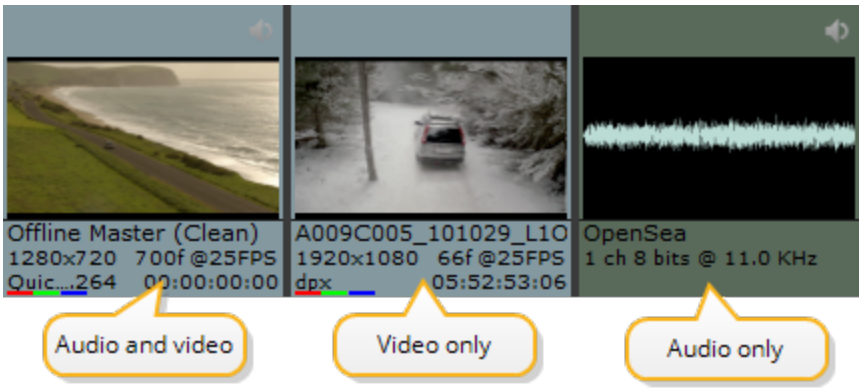
The interface sorts your clips into three broad categories: Audio and Video, Audio Only, and Video Only. Clips are displayed differently depending on their content, location, and in the case of the Viewer, the current mode.

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

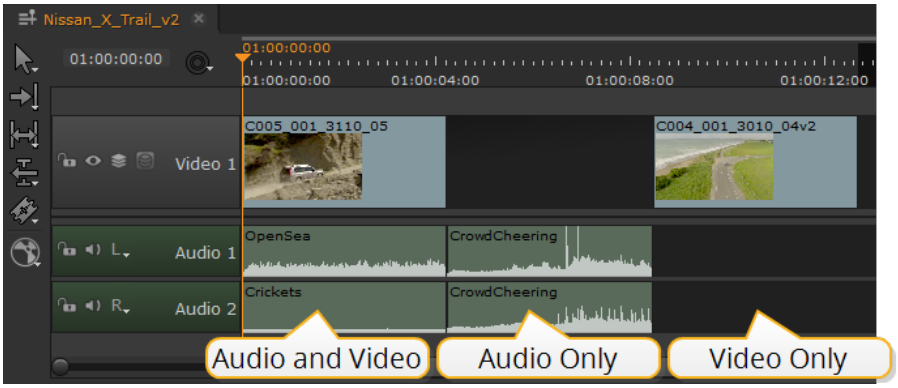
Source clips are representations of files on disk. Shots on the timeline refer to source clips, so making changes to a clip in a bin affects all shots that refer to that source clip.



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.

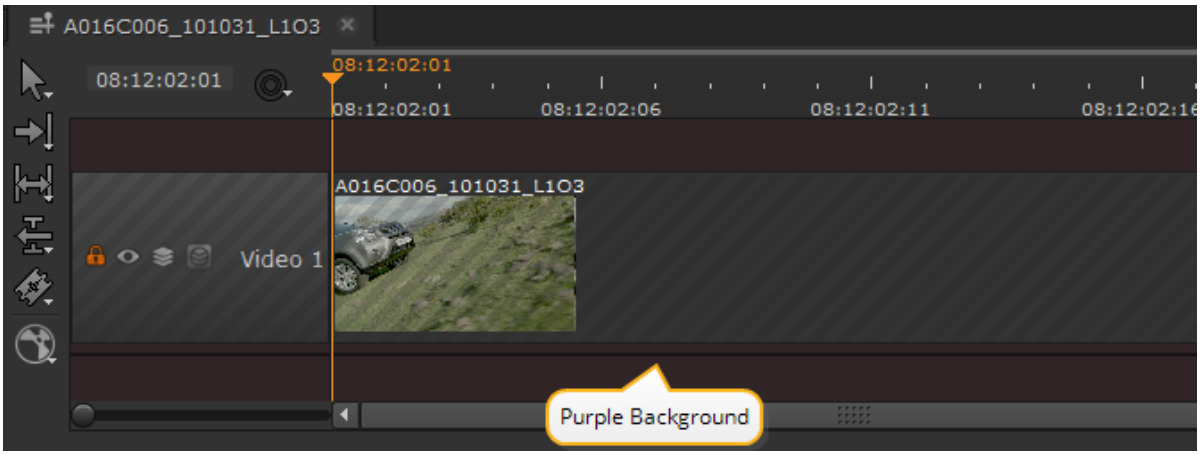
Shots

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 Adding Effects on the Timeline for more information.


A016C006_101031_L1O3
08:12:02:01
08:12:02:01
08:12:02:01
08:12:02:06
08:12:02:11
08:12:02:16
A016C006_101031_L1O3
Video 1
Purple Background

Clip and Shot Properties

The **Properties** contain standard controls similar to Hiero'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.

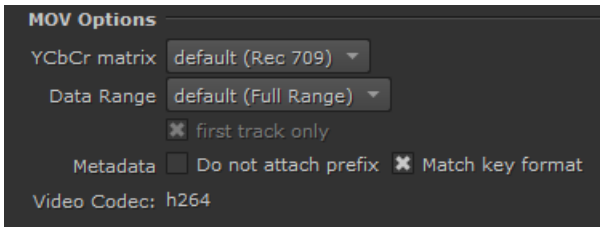
See Localizing Media for more information on localization.

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 Project Settings for more information.

As an example, some **.mov** files allow you to control the **YCbCr matrix** and **Data Range** 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.



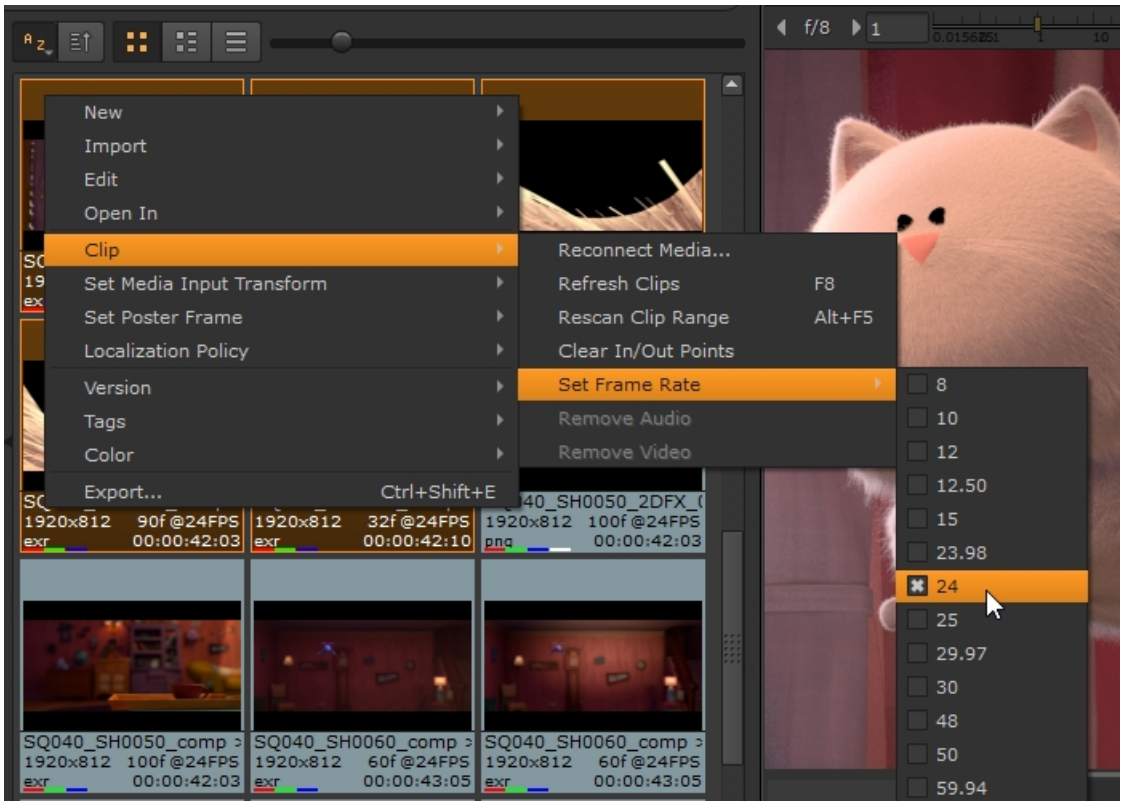
In addition, source clip properties are accessible through the same Python API as Hiero, improving scripting capabilities and integration into existing pipelines.

Setting Clip Frame Rates

You can change a clip's frame rate, or frames per second (FPS), from the Project bin. This allows you to set or change the FPS of clips before adding them to a sequence in case the clip's FPS is set incorrectly or badly defined in the file's metadata. You can set a single clip's FPS or multiple selections simultaneously.

To set a clip's frame rate:

1. Select the target clip or clips in the Project bin.
2. Right-click a selected clip and navigate to Clip > Set Frame Rate.
3. Select the required frame rate from the menu.



The selected clip or clips are updated to playback at the chosen FPS.

> **Note:** Changing a clip's frame rate does not change the FPS of related shots in a sequence. This is because a shot's FPS is set by the sequence Frame Rate control and does not rely on the source clip's frame rate.

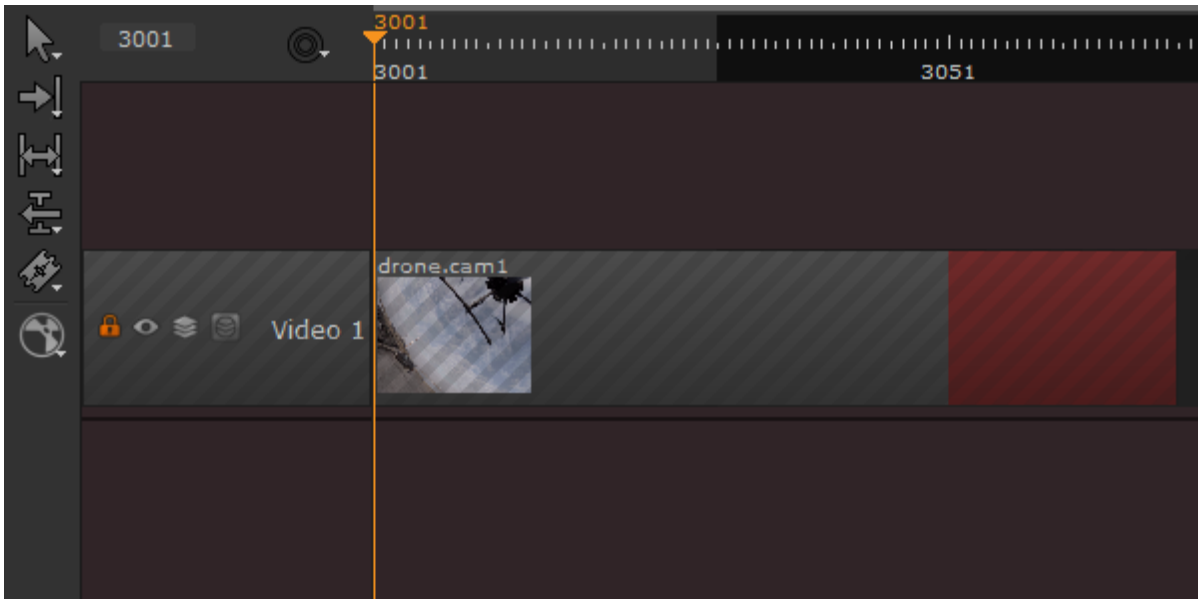
Setting Source Clip Ranges

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.

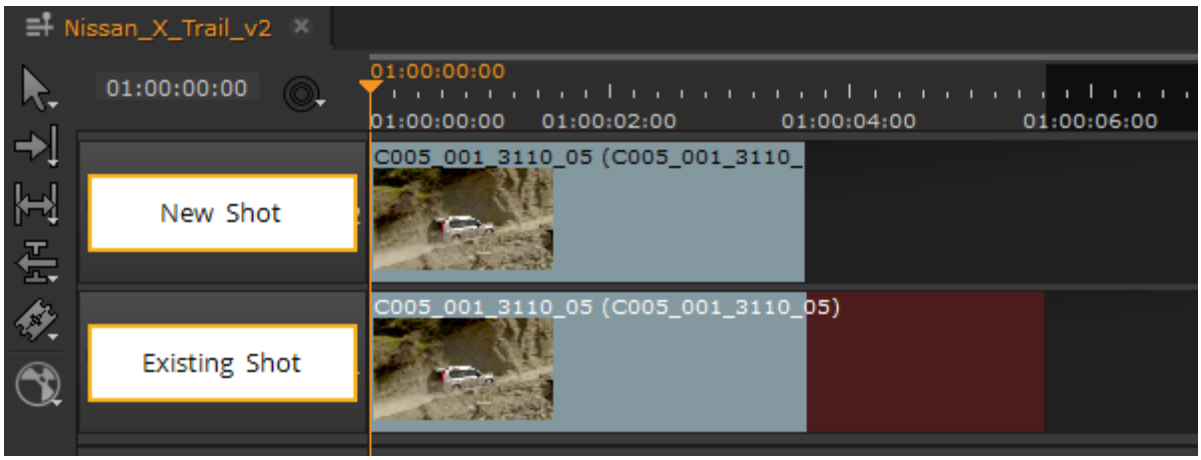
Note: You can reset source clip to its original frame range at any time by clicking **rescan**. Rescanning also appends new frames as they become available.

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.

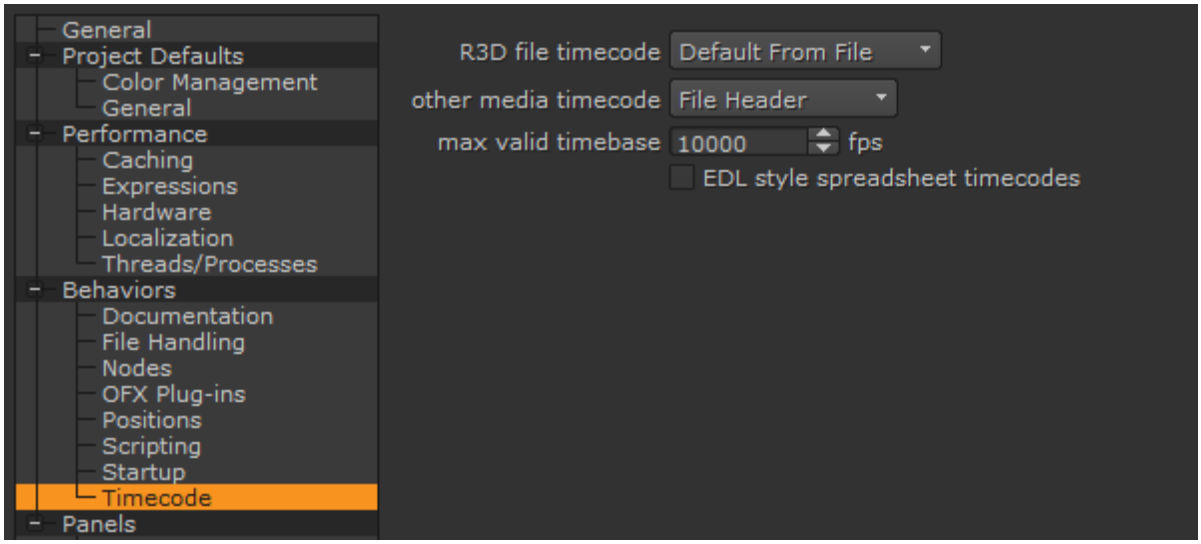
Note: Projects containing large amounts of movie files (for example **.r3d** and **.mov**) may exceed the number of available file handles per process, causing problems opening new files or projects and exporting.

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.

- **Edge Timecode** - force the use of the Edge Timecode as specified in the clip metadata.

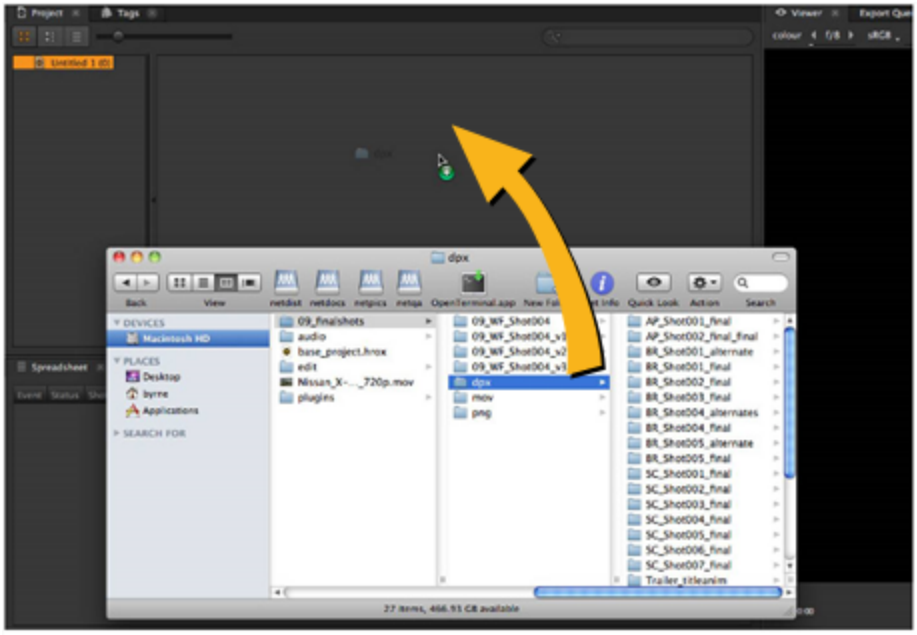
4. Use the **other media timecode** dropdown to determine clip behavior for all other clips:
   - **File Header** - the file metadata header is used to derive the timecode, if it exists. This option defaults to **Frame Number** if the header is missing.
   - **Frame Number** - ignores the metadata header, 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.

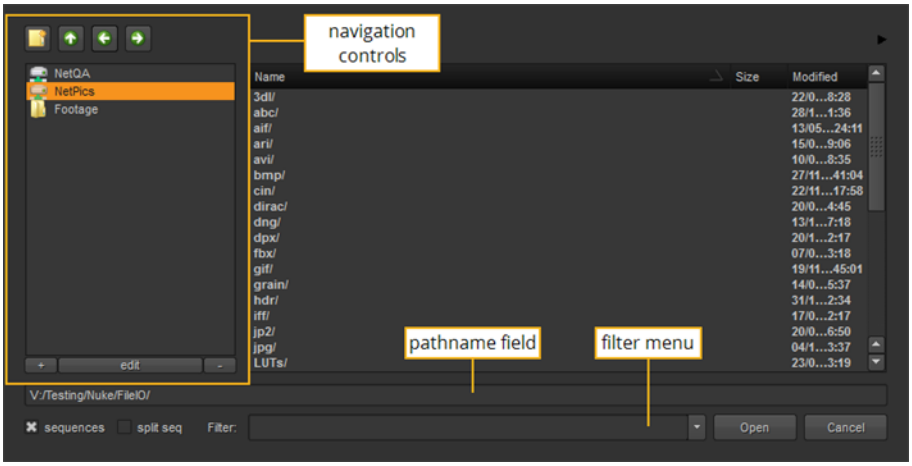
    For example, dragging frames 1-5 and 11-20 ingests the entire frame range.
  - **Disabled** - only the dragged frame or range is imported into the bin.

    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.

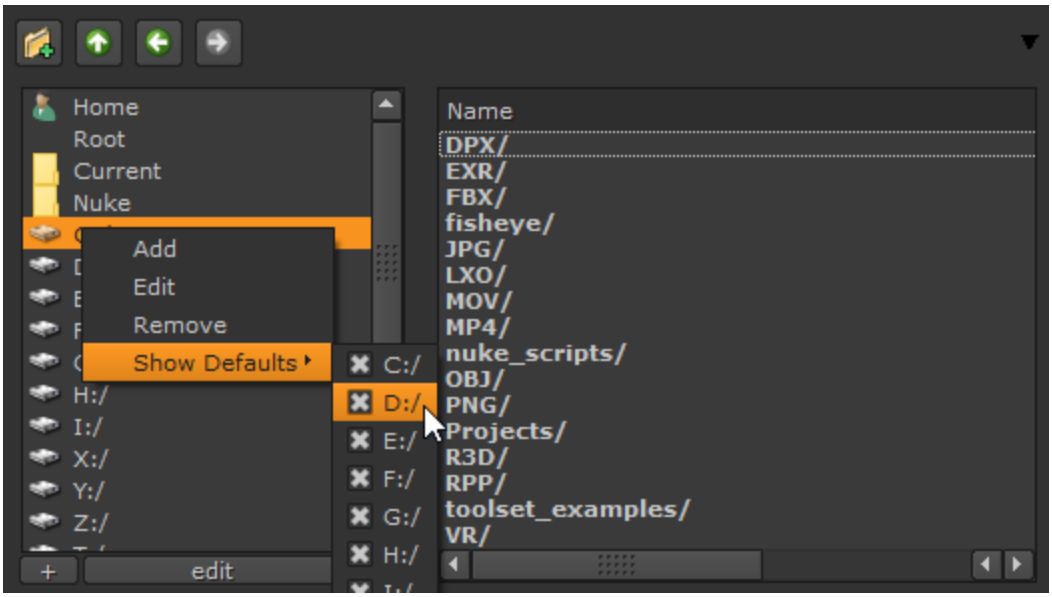
Whenever you load or save files, a browser similar to the one shown below is displayed. The directory navigation buttons let you create or access the directory from which you wish to read or write data.



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.

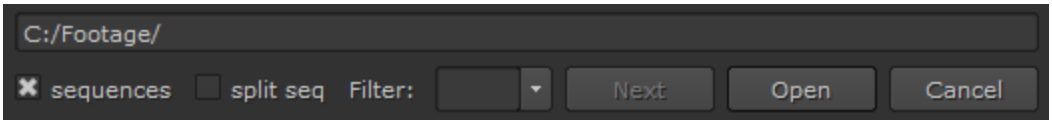


To Use the Navigation Controls

- 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

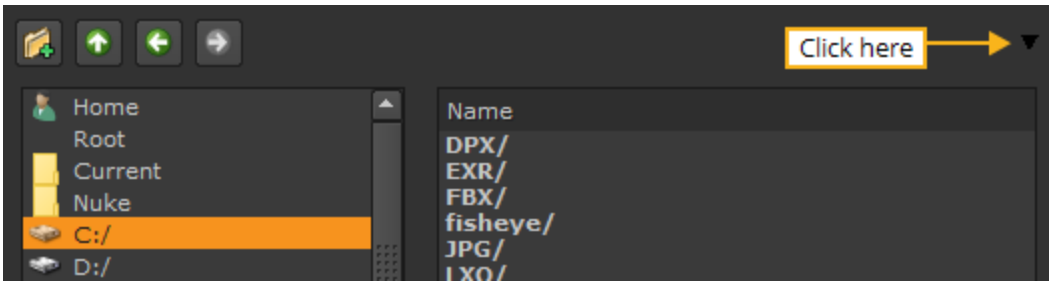
• Select ***.nk** to display only Nuke script files.
• Select ***** to display all files (except hidden files), regardless of what they're associated with.
• Select **.* *** to display all files, including hidden files.
• Select ***/** to display directory names, but not their contents.
• Check **sequences** 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.

• You can also split incomplete sequences into separate Read nodes using the **split seq** checkbox.

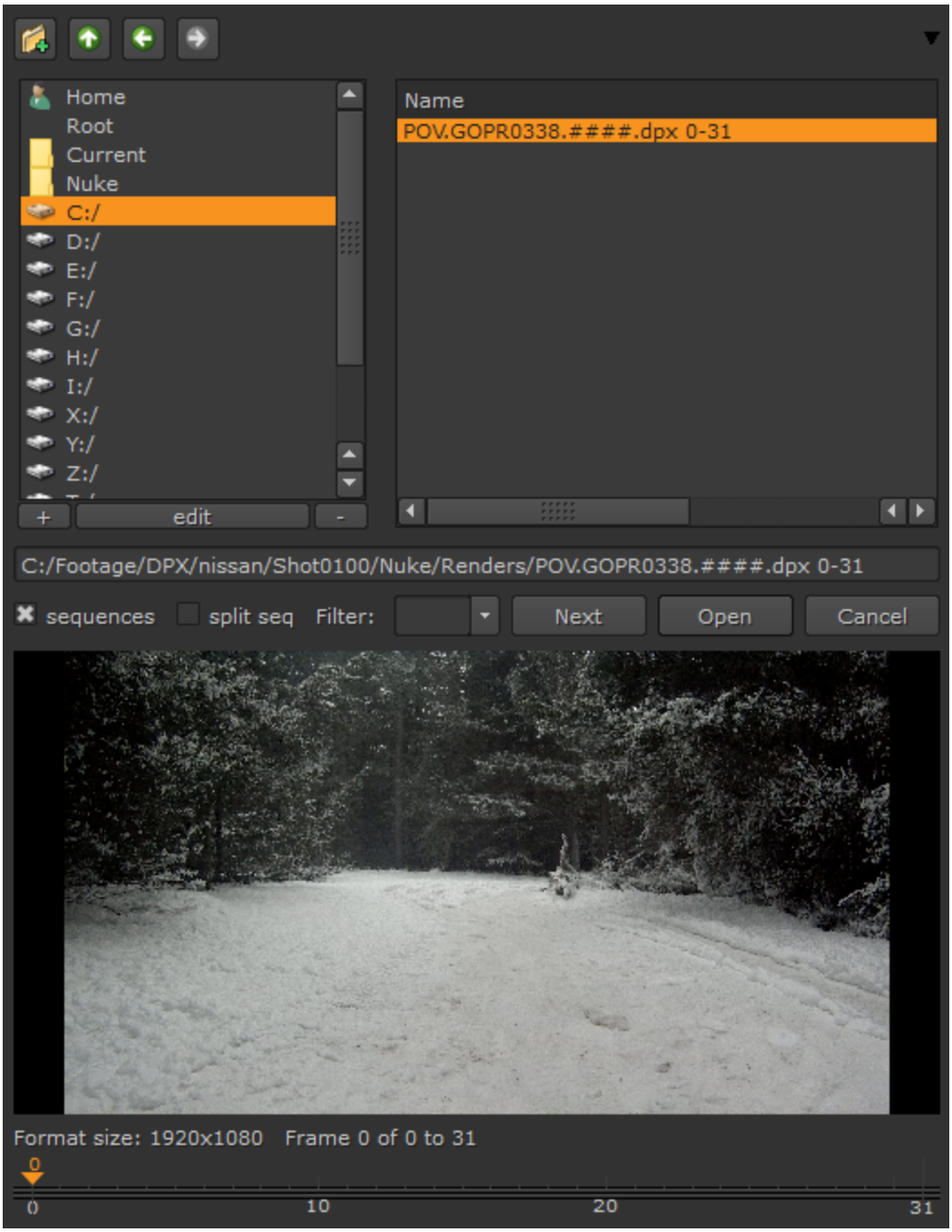
To Preview Files in the File Browser

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.



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.

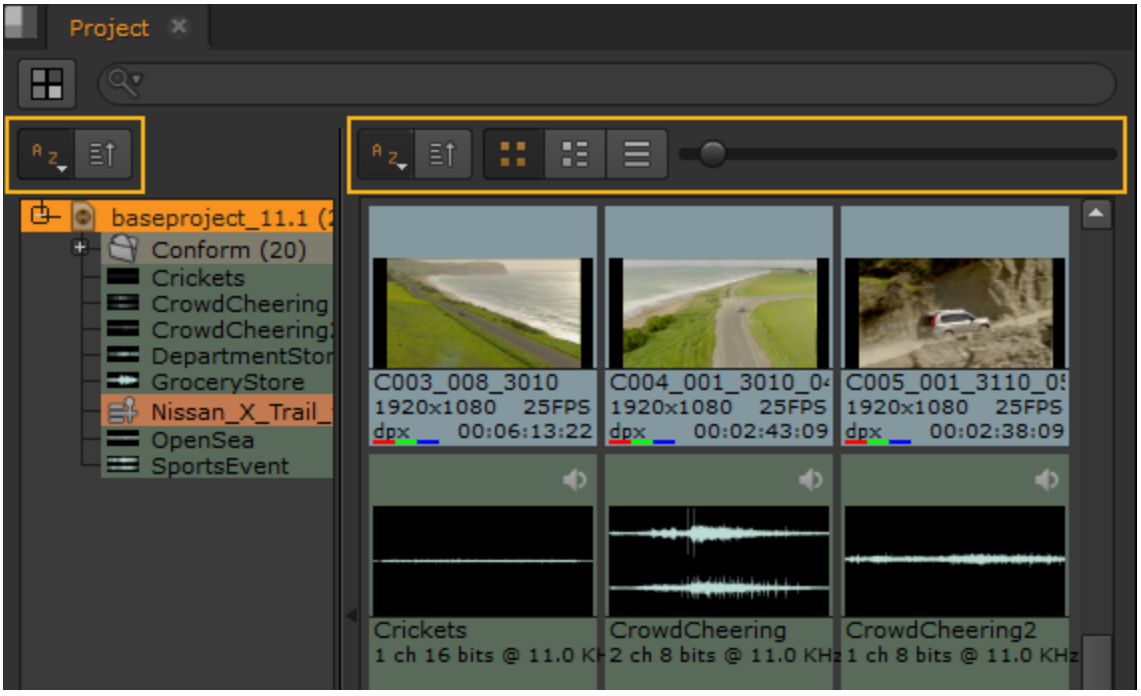
Sorting and Searching Media

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.

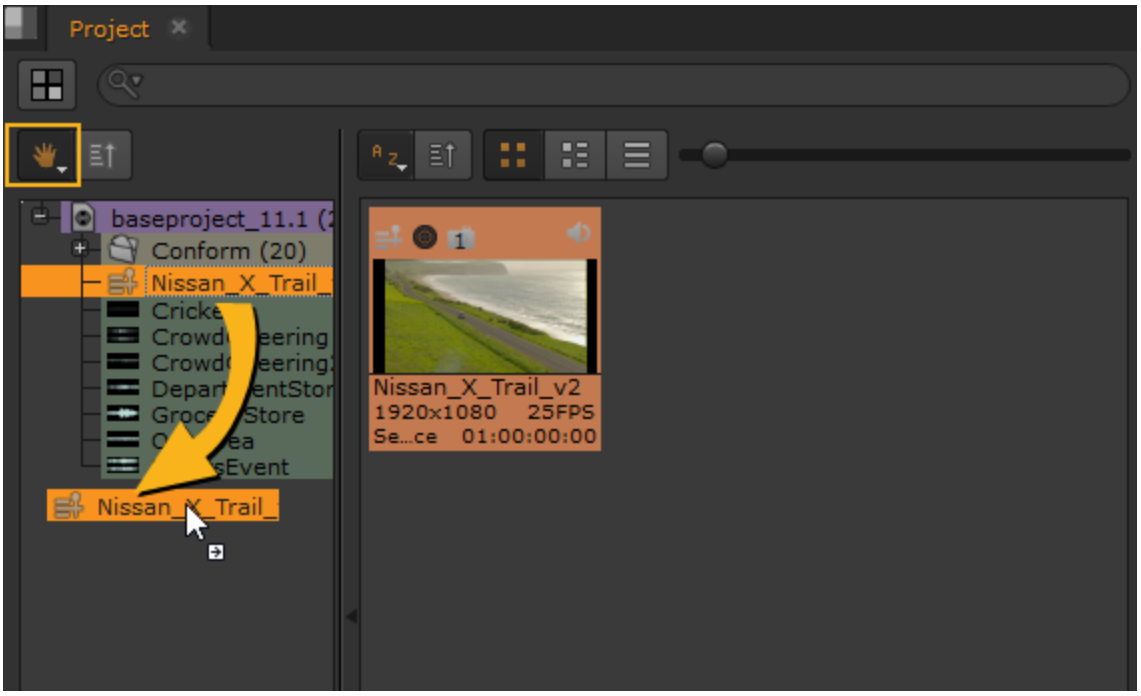
Sorting the Project Panel

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.

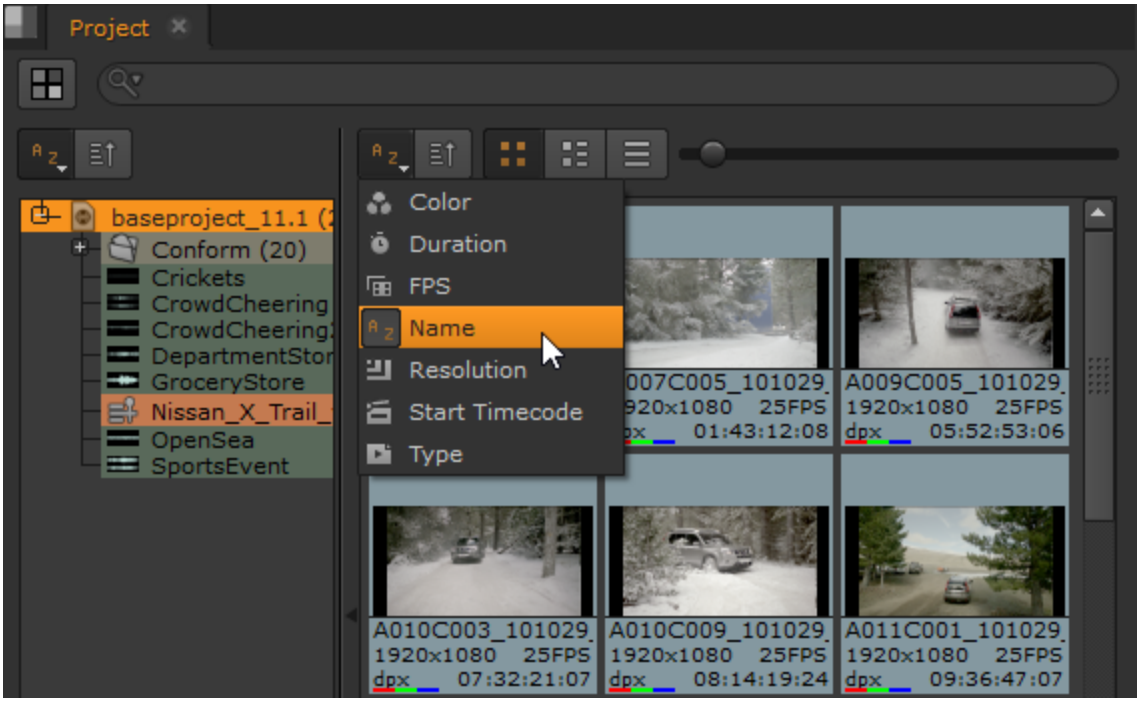


In the left-hand directory pane, items are sorted alphabetically in ascending order by default. To sort project items manually:

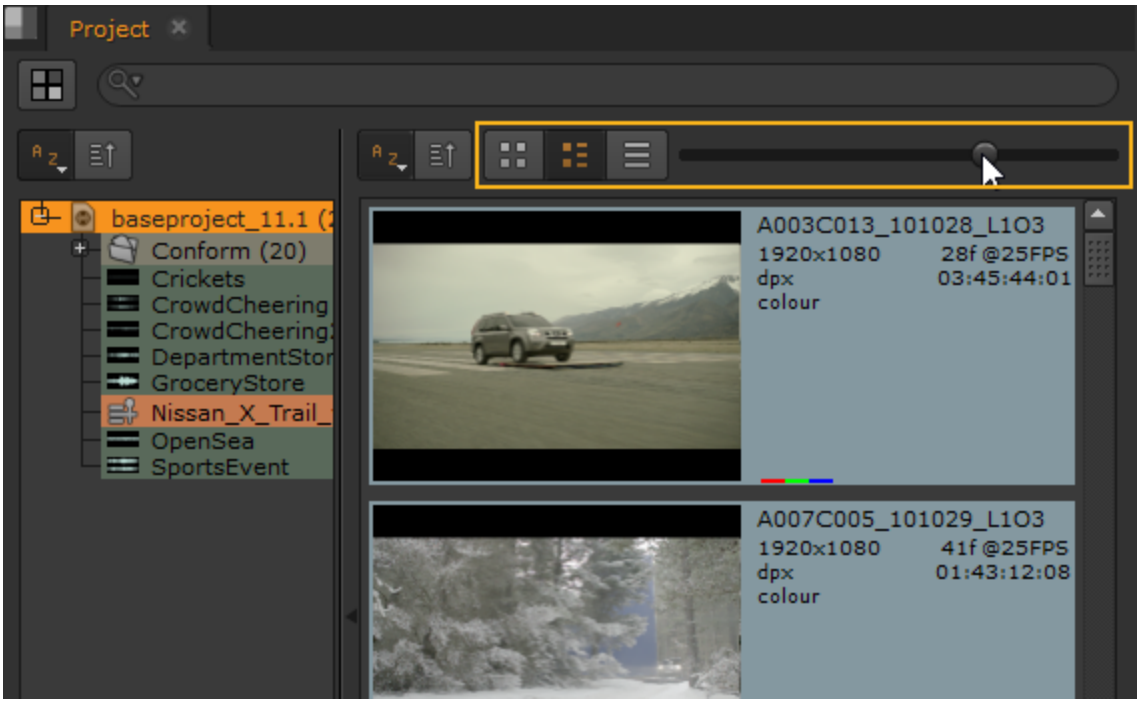
1. Click and hold, or right-click, the sorting dropdown and select **Manual**.
2. Drag-and-drop items into your preferred order.



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**.



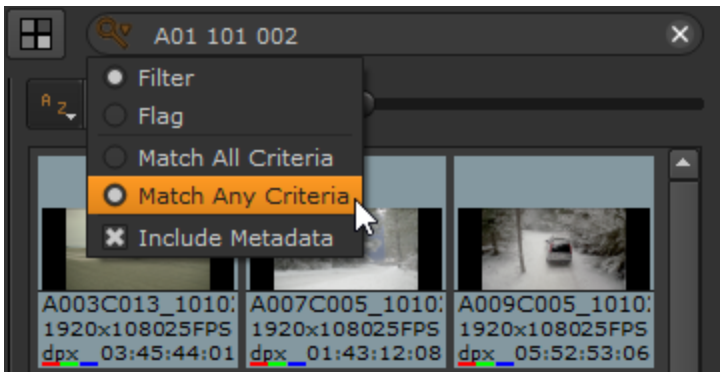
You can also change the layout and size of items in the right-hand bin pane. Choose a layout from thumbnails, list, and details and use the slider to determine the size of the items. The slider is disabled in details mode.



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.

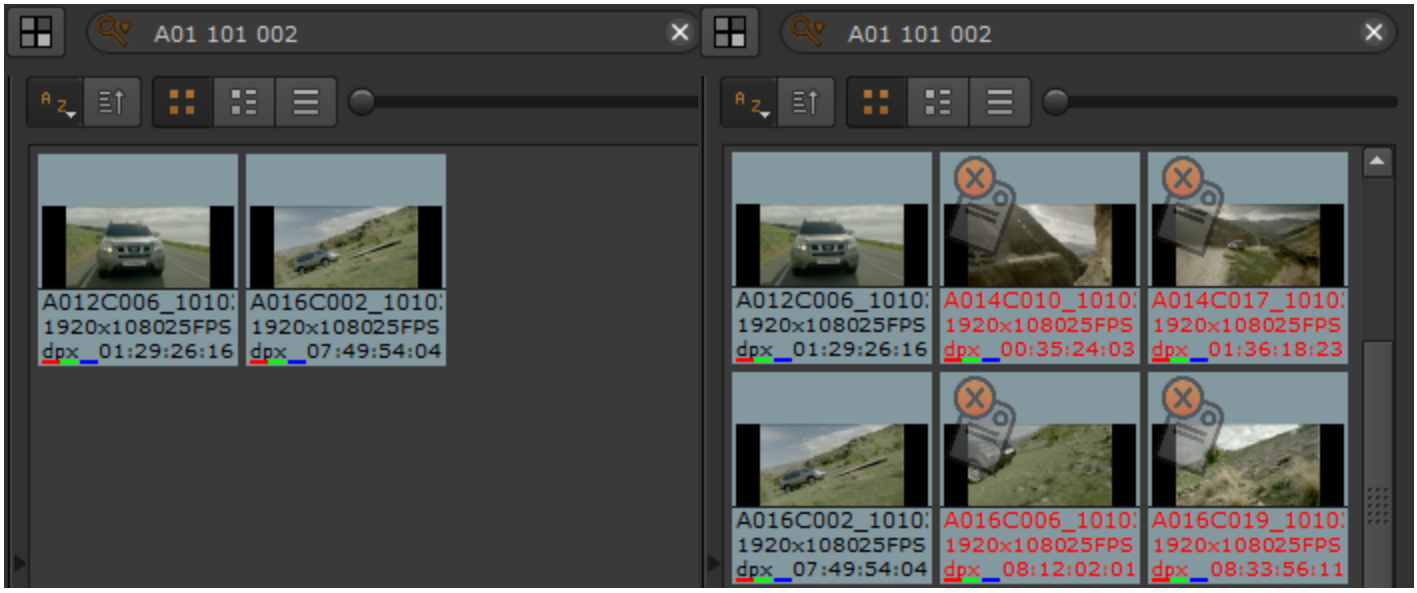


- **Match All Criteria** - media that matches **all** the entries in the search string are matched. For example, entering A01 DPX only matches media containing both A01 and DPX.

- **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: 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.



Filtering ... ... and Flagging

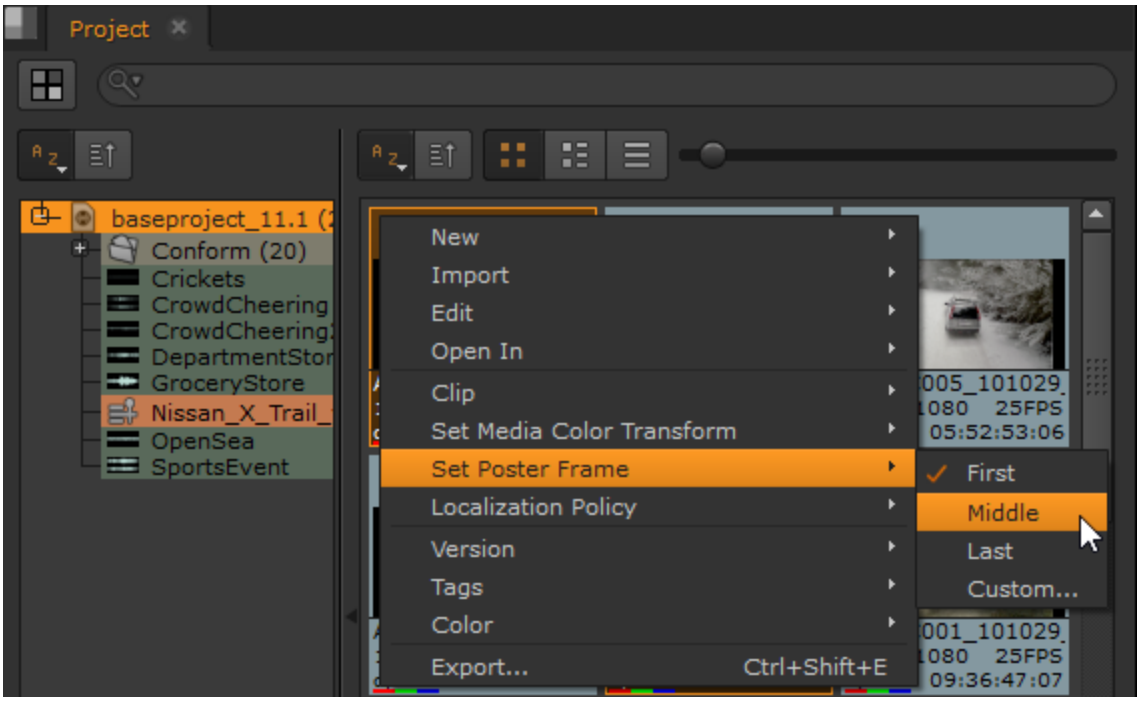
You can also filter and flag media by tag. See Filtering and Flagging Media Using Tags for more information.

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 **.dpx** sequence **myClip.####.dpx 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 **.dpx** sequence **myClip.####.dpx 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

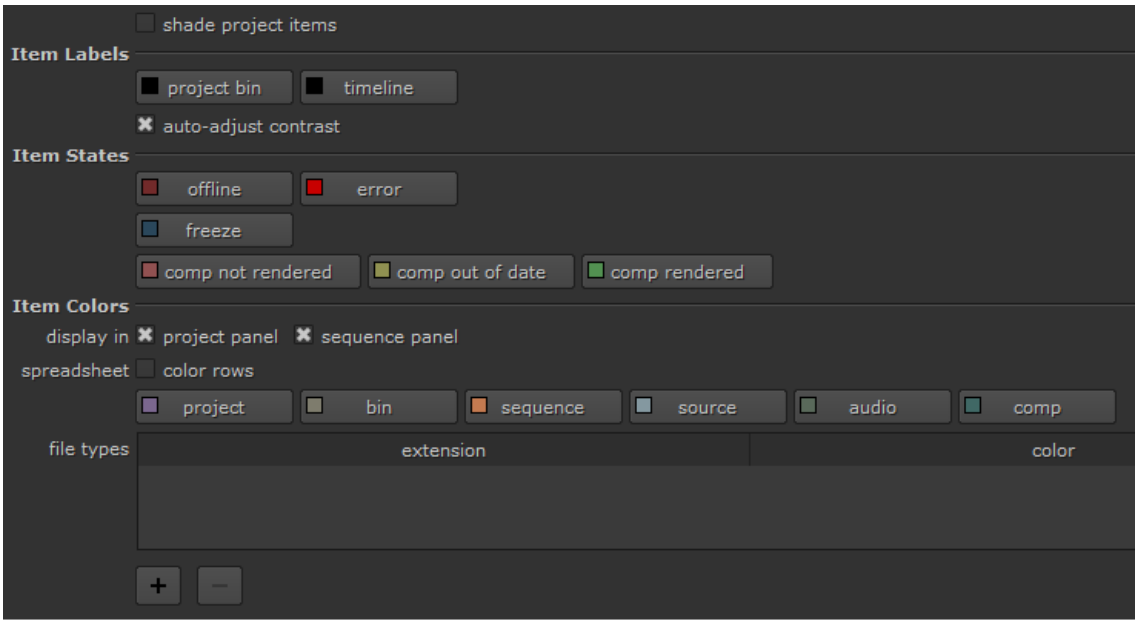
In large projects, the **Project** bin and timeline can quickly become busy and difficult to manage. Adding colors to items or file types can help you find what you're looking for in bins and sequences. You can quickly enable and disable colors in the **Project** panel and timeline in the **Preferences** 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.

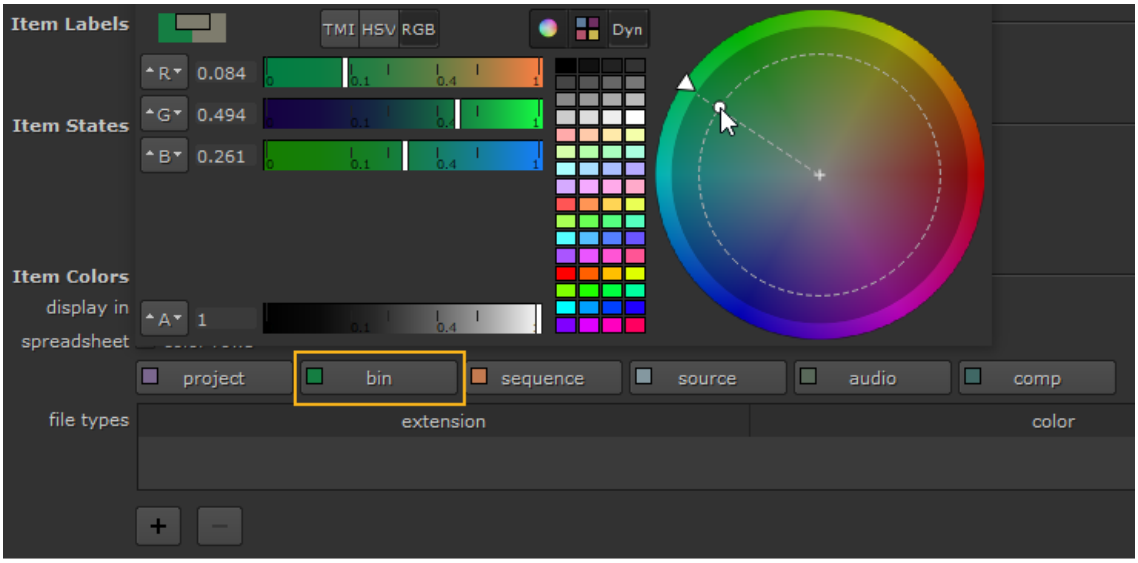
Setting Default Colors

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**.

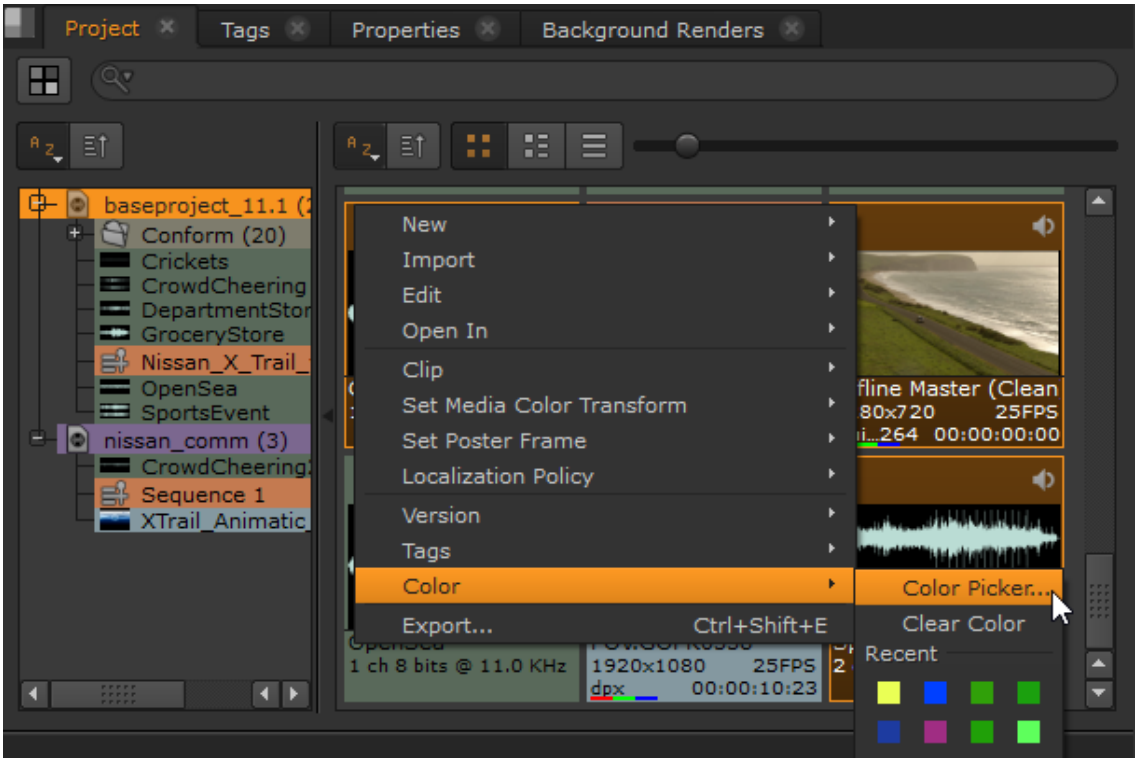
Setting Colors by Selection

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**.



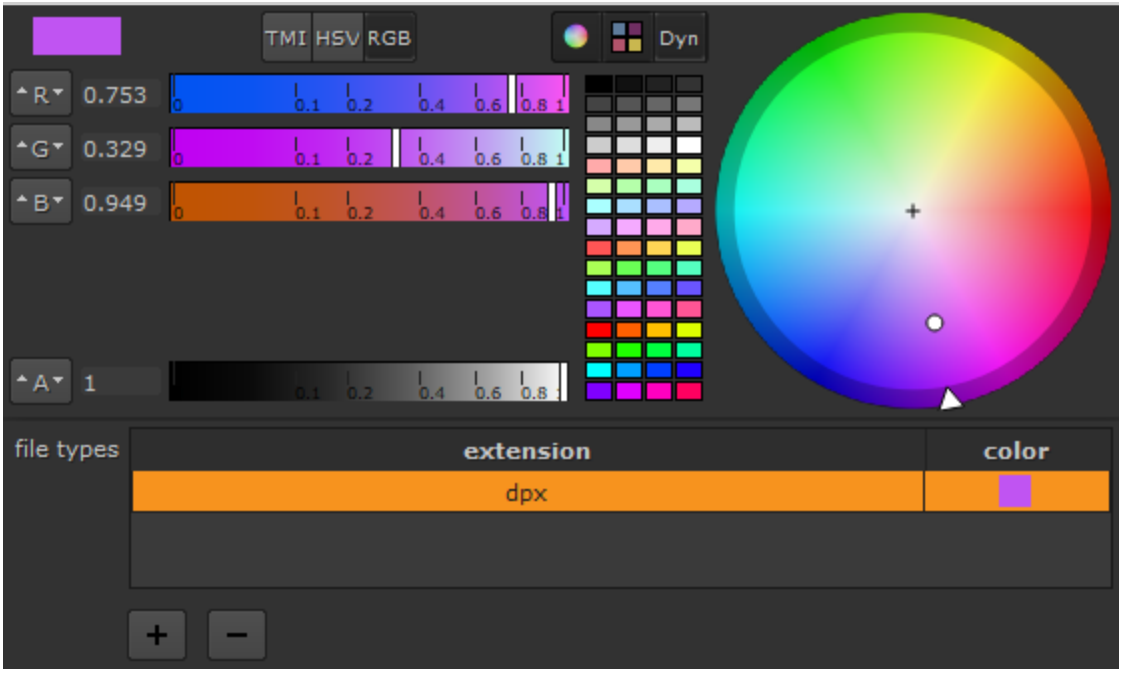
3. Click **Clear Color** to remove any custom colors applied to the selection. The selection reverts to the default item color set in the **Preferences**.

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.

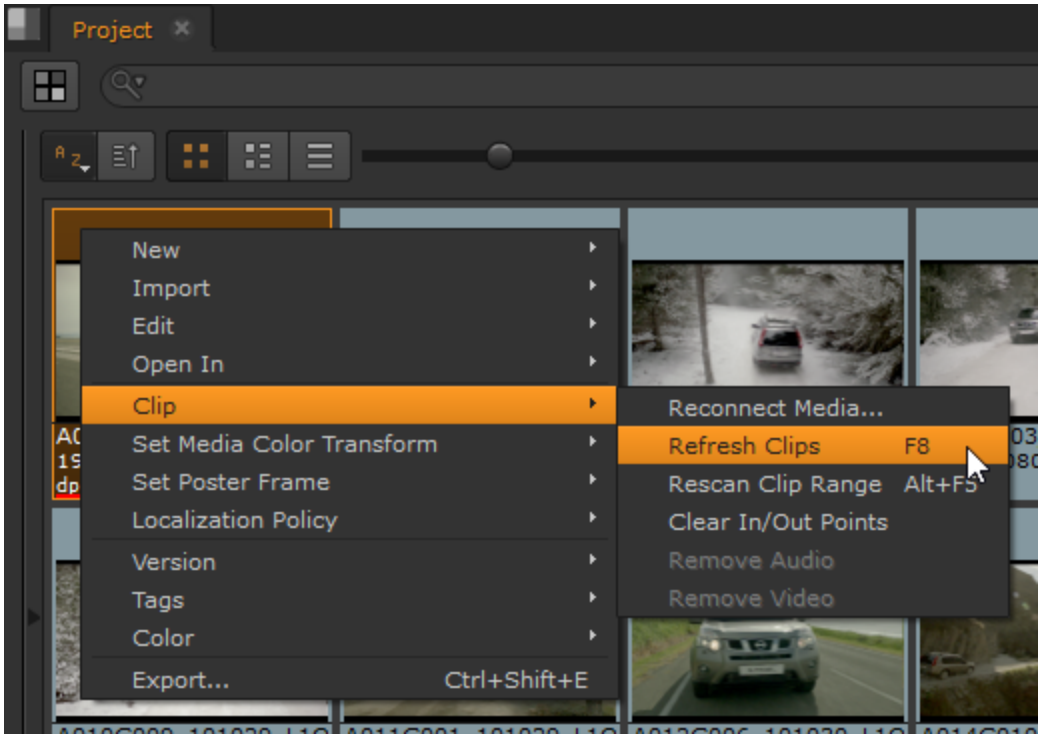
   A new entry is added to the file types table.
4. Click the **extension** dropdown and select the required file type.
5. Double-click the **color** swatch and select the new color using the color wheel.



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.

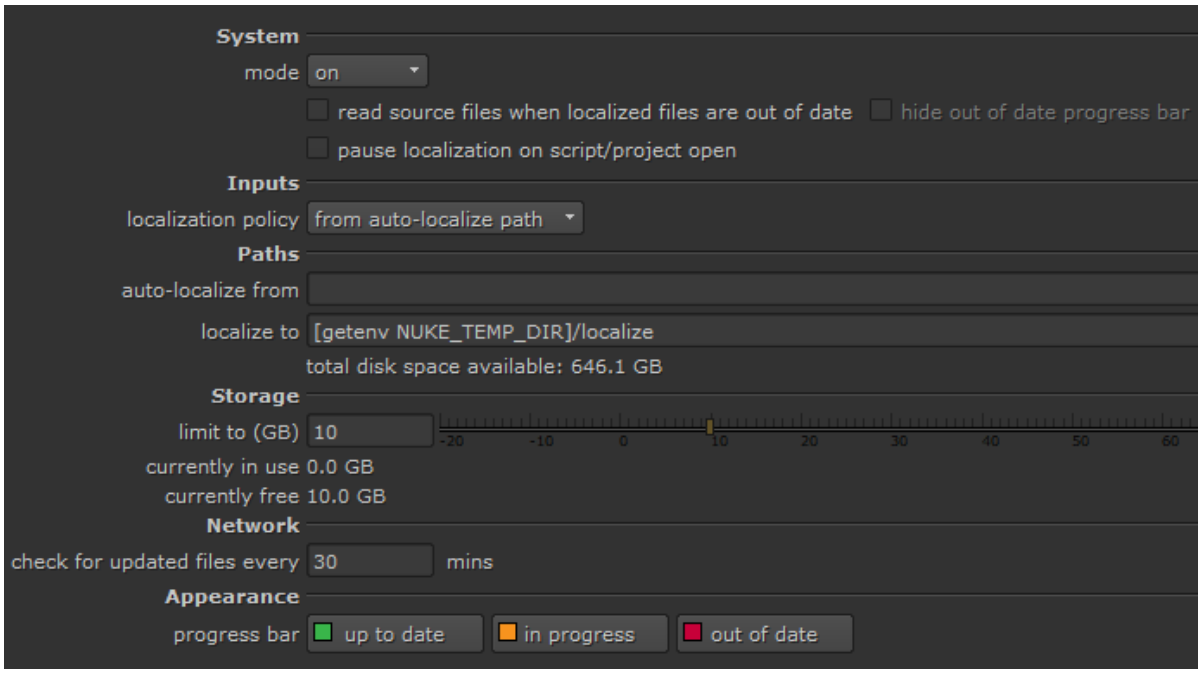
Localizing Media

Hiero 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

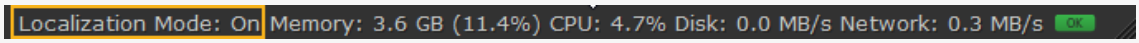
The **Preferences** control how Hiero deals with new source clips as they are ingested, but does not affect existing source clips in the project. If you plan to localize source clips in your project, it's a good idea to set the **Preferences** before ingesting your source clips.

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.
   - **off** - no source clips are localized, regardless of the their localization policy.

Note: The current localization **mode** is displayed in the status bar at the bottom-right of the interface.



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 Hiero, providing that the **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.

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**

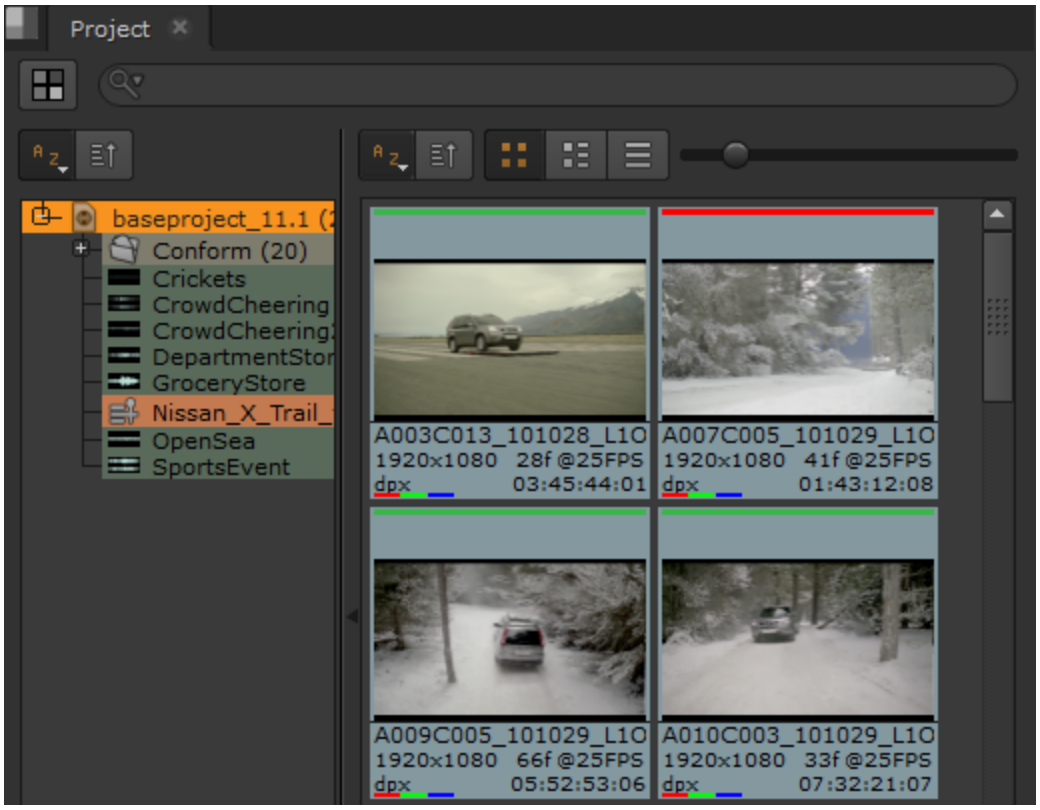
6. Enter a value for **limit to (GB)** to control how much disk space is available in the cache directory.

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.

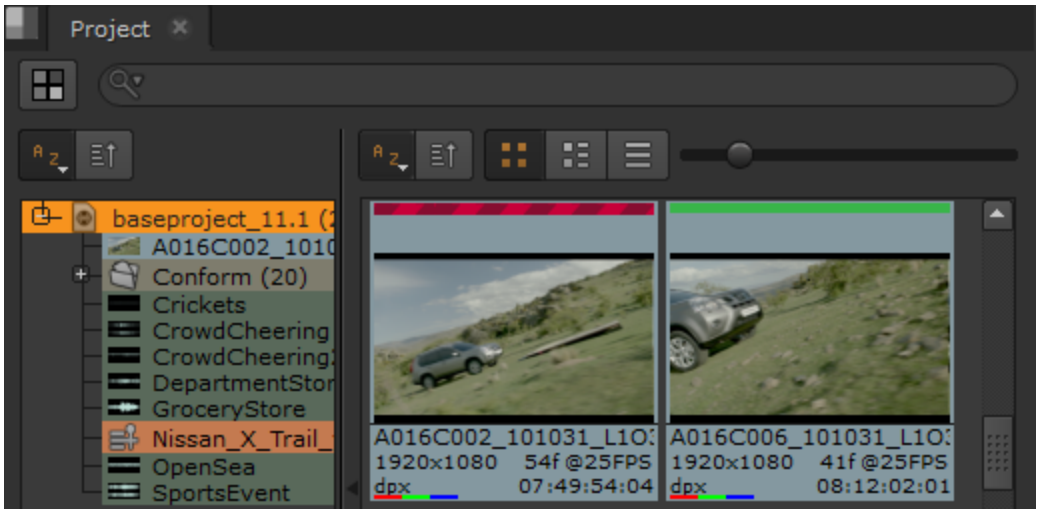
7. You can specify the time interval (in minutes) before localized files are checked for updates using the **check for updated files every ## mins** control.

The default setting checks for new versions of files every 30 minutes, but you can set this control to any value. Any 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 Hiero uses to process jobs. Set the NUKE_LOCALIZATION_NUMWATCHERS environment variable to the number of threads you want to use. See the Nuke Online Help for more information.

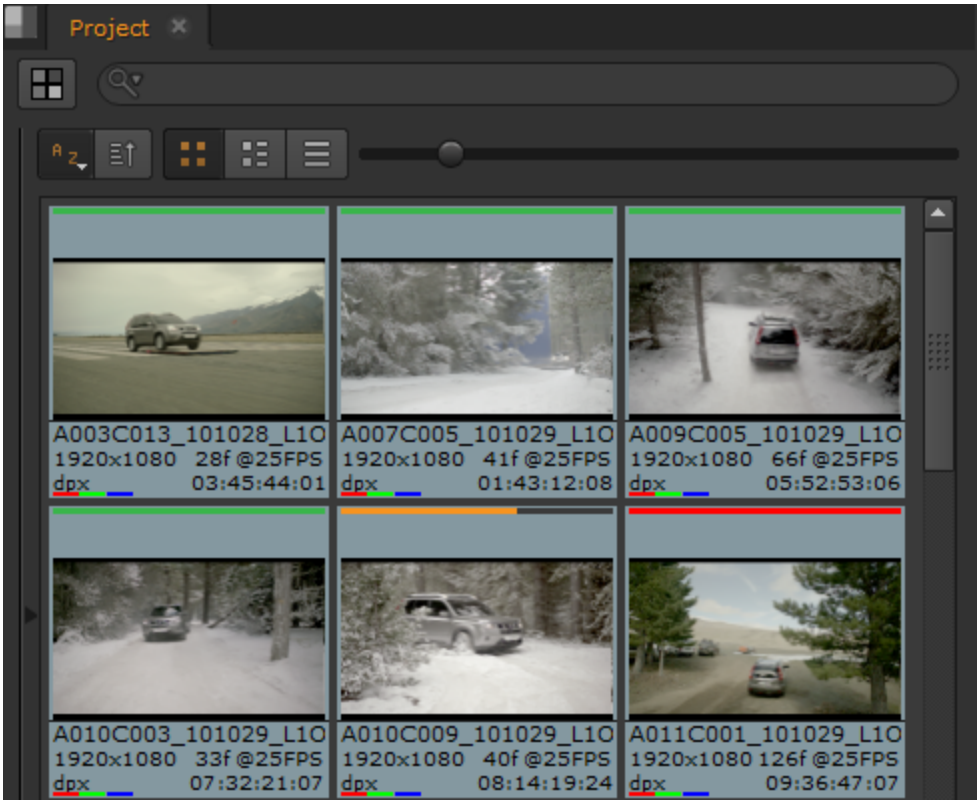
To set the localization policy for source clips:

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.

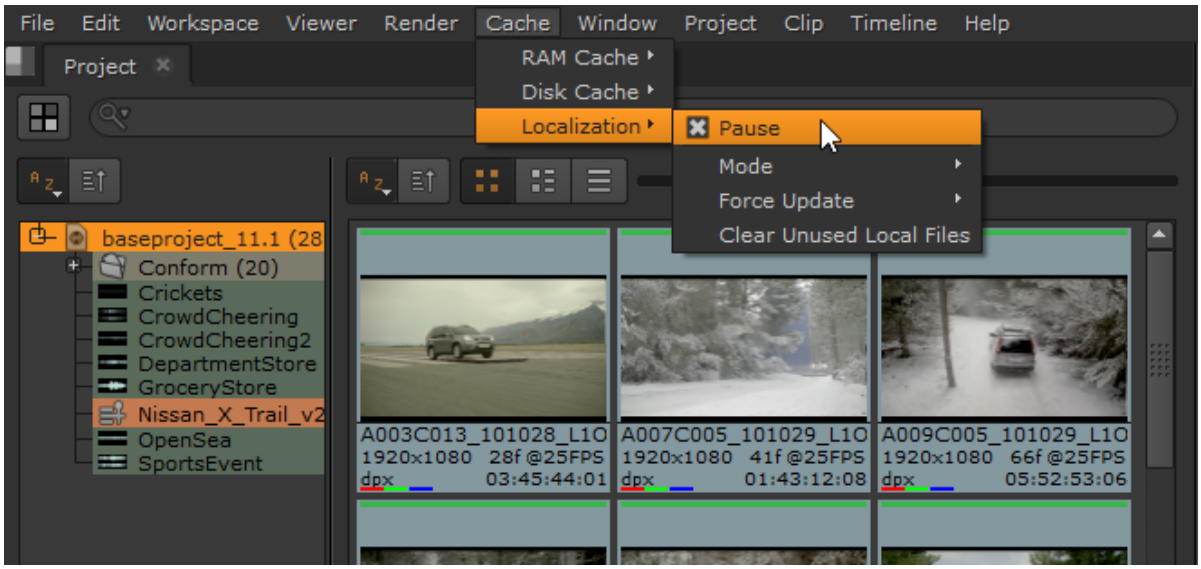
As clips are localized, an amber progress bar displays in the thumbnail. Fully cached clips are marked with a green bar at the top of the thumbnail and out-of-date clips are marked in red.

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.



Note: Hiero 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 Hiero 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**.

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.

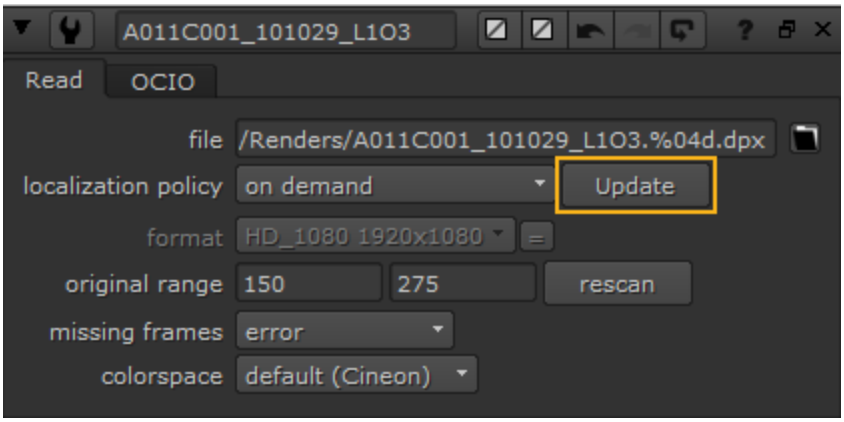
System Preference	Source Clip Preference			
	on	auto-path	on demand	off
on				
manual				
off				

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**.

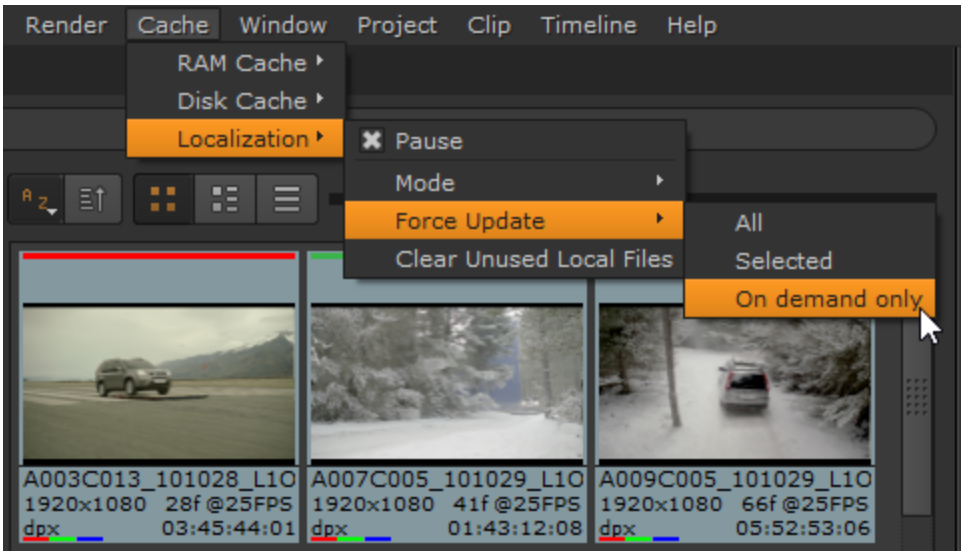


Note: If a Read node's localization policy is set to **on demand** and it hasn't been localized previously, clicking the **Update** button localizes the file.

The local copy of the source clip is updated from the remote source clip.

To update all **on demand** source clips:

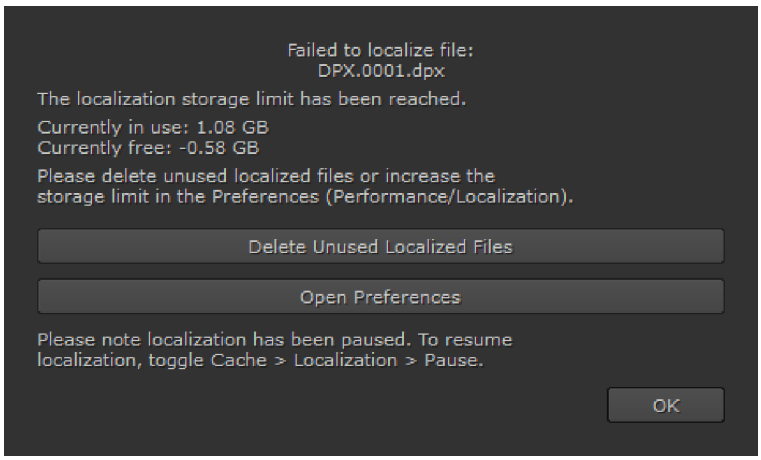
1. Navigate to **Localization** > **Force Update**.
2. Click **On demand only**.



The local copy of all **on demand** source clips is updated from the remote source clip.

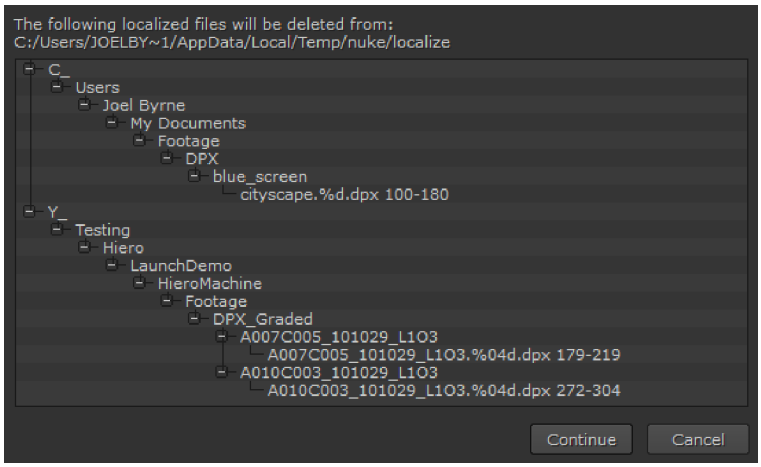
Clearing Localized Files

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**). Hiero 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 Viewer

Hiero 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.

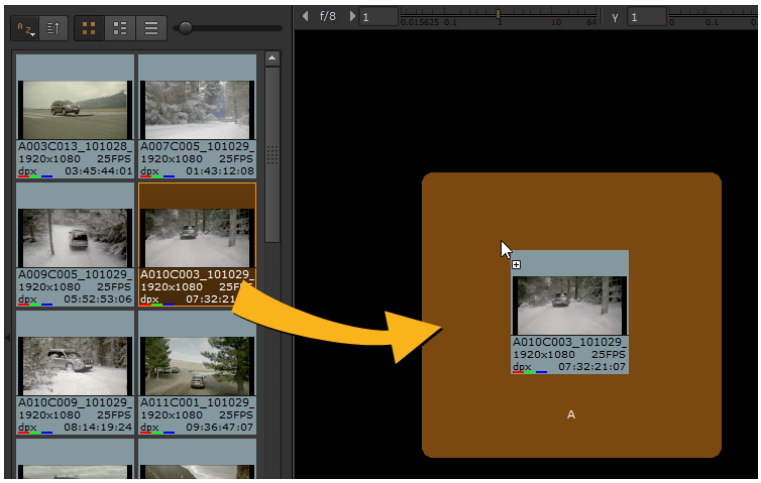
Sequence Viewers, also known as record Viewers, are marked with the  icon and deal with sequences and shots on the timeline. You can set In and Out points and apply tags here too, but you can also edit the shots on the timeline by trimming, retiming, and so on. See Timeline Editing Tools for more information.

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**.

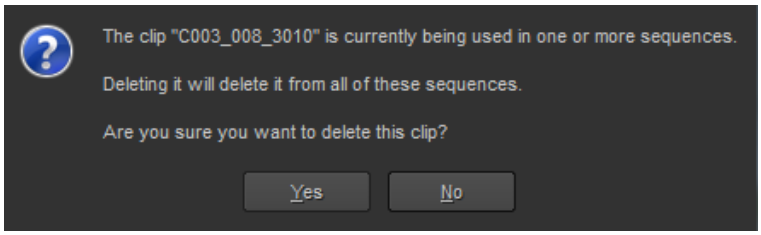
See Appendix A: Preferences for more information.



Deleting Media

To remove media from the bin view, select the clip(s) or bin and press **Backspace** or **Delete**.

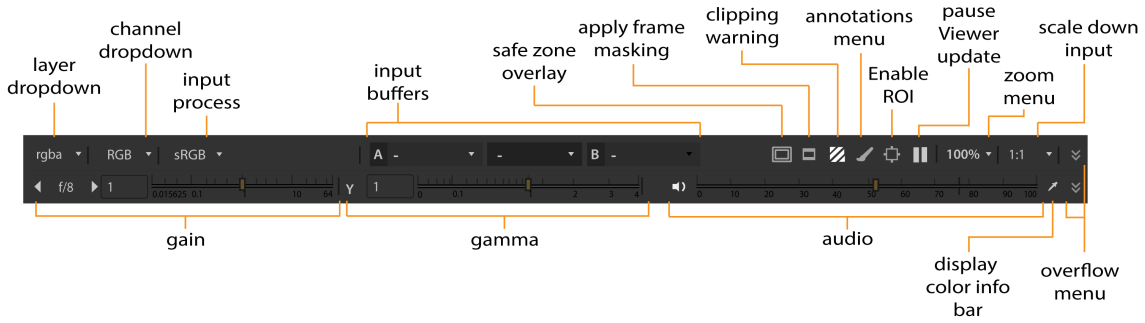
If any of the media is in use in a sequence, the following warning displays:



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.

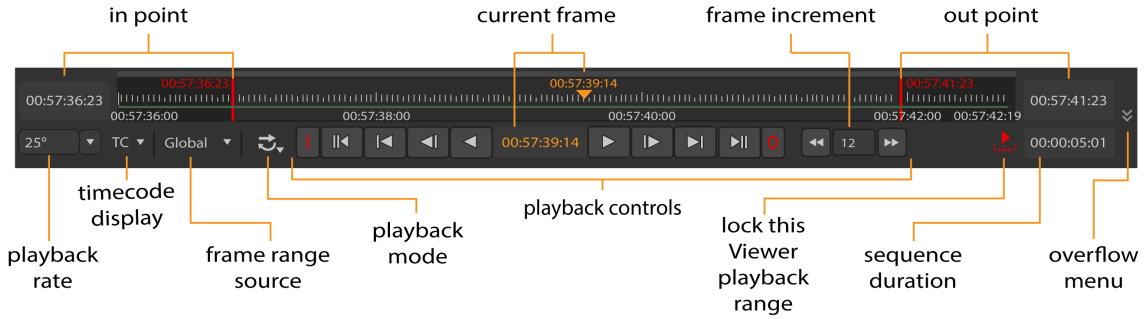


For more information about the tools above the Viewer, see Viewer Tools.

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 an 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, Hiero 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. Hiero 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.

Playback Controls

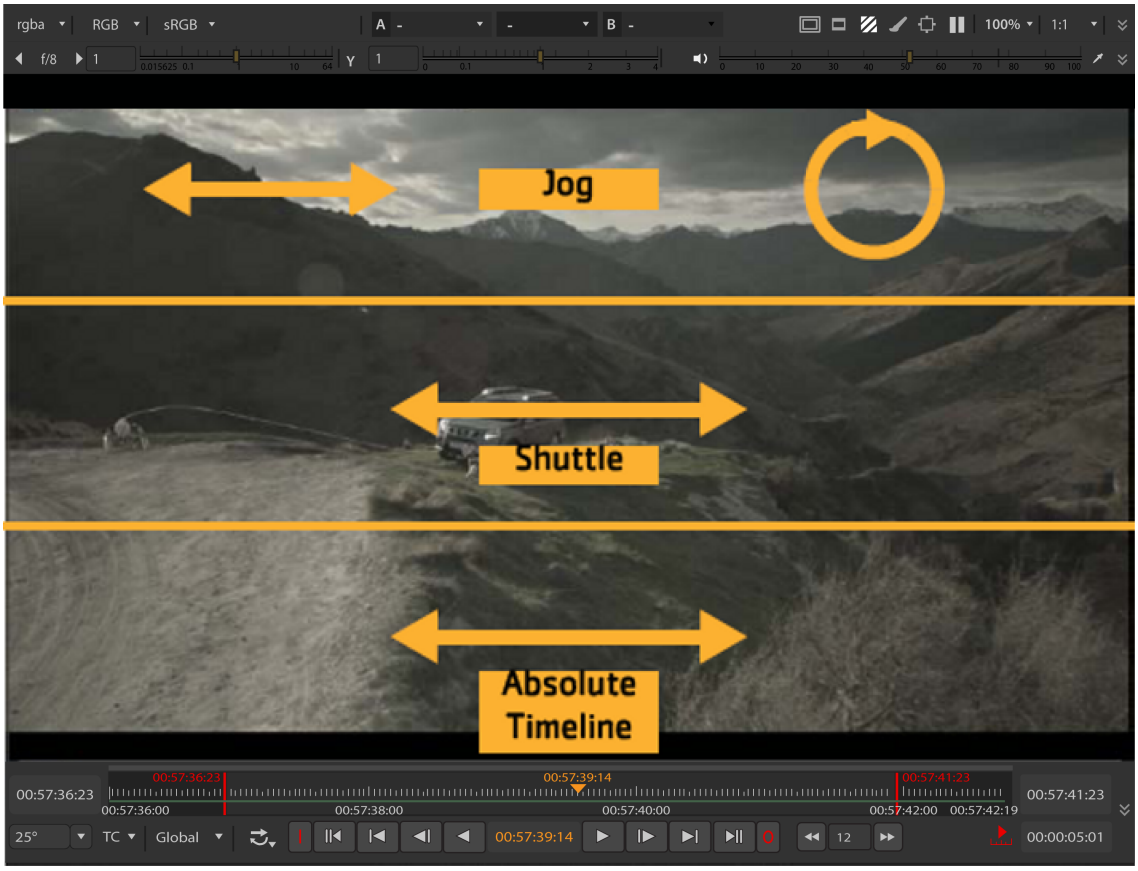
The **playback rate** field (frames-per-second) initially displays the project's playback speed. The Viewer attempts to maintain this speed throughout playback, although this adjusts depending on the resolution of the imagery and your hardware configuration.

The following table lists the functions of the playback buttons:

Buttons	Functions
	The **Play backward** and **Play forward** 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.
	The **Back 1 Frame** and **Forward 1 Frame** buttons cue the sequence to the previous or next frame.
	The **Previous keyframe** and **Next keyframe** buttons cue the sequence to the script's previous or next keyframe.
	The **First frame** and **Last frame** buttons cue the sequence to the first and last frame.
	The **Frame Increment** 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.

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:

Button	Function
Repeat	Repeatedly plays the sequence in a loop.

Button	Function
Bounce	Repeatedly plays the image back and forth from head to tail.
Stop	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.
Continue	Plays once from the beginning to the end of the sequence, ignoring any in and out points.

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 Viewer, are used to affect the mouse pointer as you move over the Viewer, and to select Viewer preferences:

Icon	Function	Description
rgba	Layers	Select the layer to output to the Viewer, for example **forward** motion vectors or **disparityL**. Only layers available in the clip are displayed - check the clip's thumbnail to see at a glance which layers are present:  - red color layer. - green color layer. - blue color layer. - alpha layer. - depth layer. - forward motion vector layers.

Icon	Function	Description
		 - backward motion vector layers.  - all other custom layers, such as disparity.  **Note:** You can scroll through available layers using **PgUp** or **PgDn**.
	Channels	Select the channel(s) to output to the Viewer, for example **RGB**, single channel, **Alpha**, or **Luma**.
	Viewer color transform	Set the view transform 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	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	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

Icon	Function	Description
		information.
	Mask	Enable or disable a range of Viewer masks, for example **16:9** or **1.85:1**.
	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.
	Annotations	Click to enable the Annotations tool bar. Annotations allow you draw and add text to clips in the Viewer. See Annotations for more information. **Note:** The Annotations button also controls existing annotation visibility.
	ROI	Click and drag to define a Region of Interest (ROI) in the Viewer. The scopes only display information within the ROI, when active.
	Pause playback caching	Pause or release Viewer playback caching, indicated by the green bar under the Viewer.
Scale		Set the scale applied to the clip in the Viewer, for example **25%**, **75%**, or **Fit**.
Image Quality		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. **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

Icon	Function	Description
		proxy value is equal to 2K, 1:2 is equal to 1K, and so on.
	Non RT Playback	Sets 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 maintain the frame rate. • **Play All Frames, Buffering** - plays all frames by buffering and playing frames back as they become available.
	See through missing media	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. **Note:** This control only affects the current Viewer.
	View	Select the Viewer display mode, for example **Audio and Video** or **Video Only**.
	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.
	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.
	Gain	Adjusts the gain applied to the linear input image before viewing, but doesn't affect your exported image.

Icon	Function	Description
	Gamma	Adjusts the gamma applied to the image after the viewing transform, but doesn't affect your exported image.
	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.
	Color Sample	Enable or disable the RGBA color information bar in the Viewer. **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 Colorspaces for more information.

Using In and Out Markers

In and Out markers enable you to alter the duration of a shot 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.

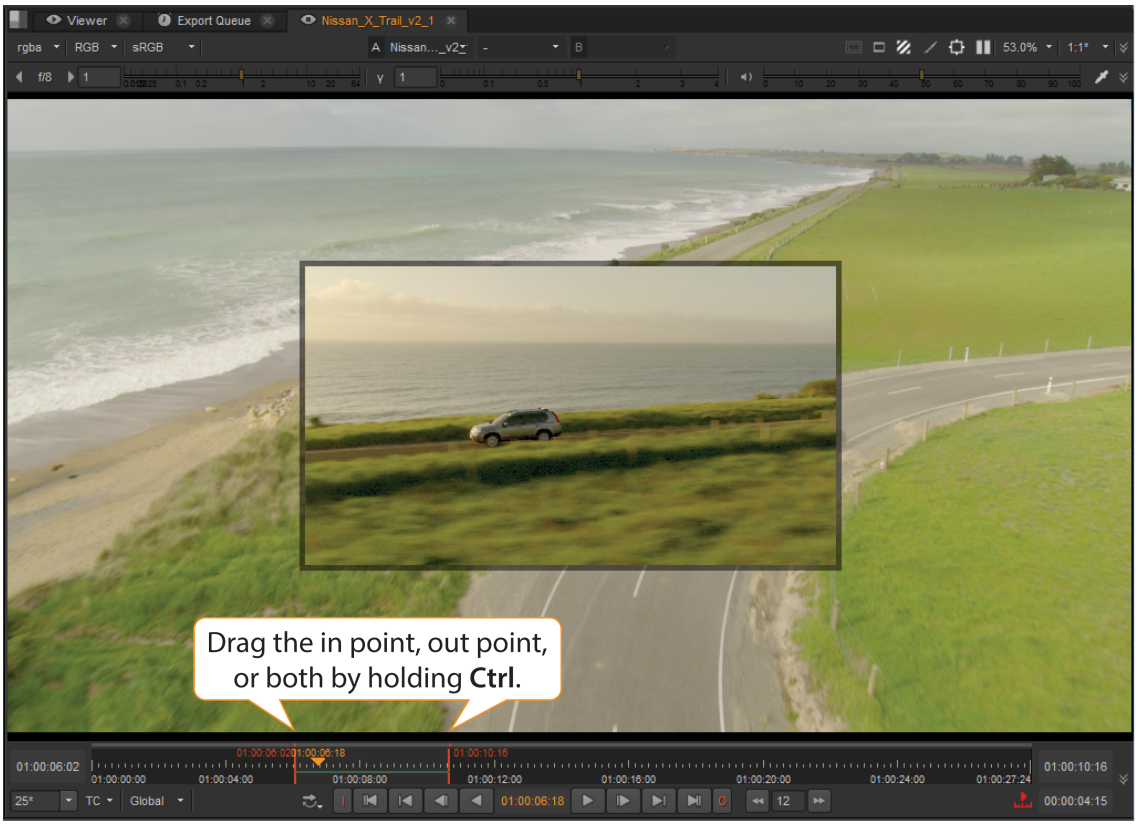
   The In point is marked by the In tab and the time is recorded in the playback controls.

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**.

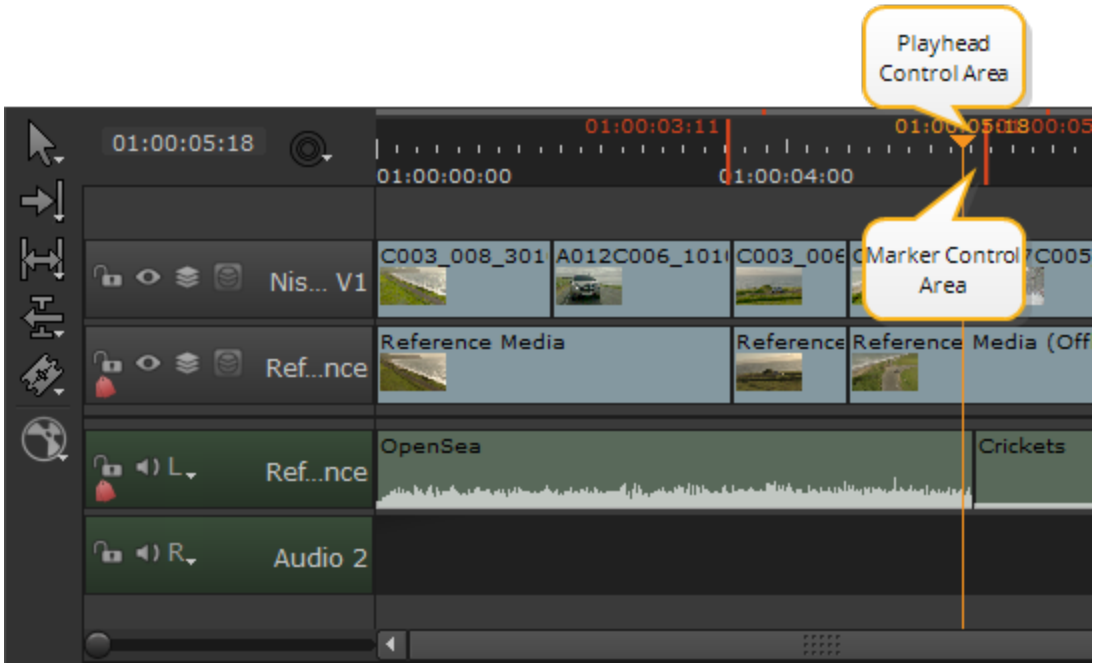
Click and drag the markers to adjust their position, or hold **Ctrl**/**Cmd** to move both markers at once, retaining their relative positions. A Viewer preview shows the current frame for the selected marker(s) and a timecode/frame popup helps to set the new position.



Clear the markers from your clip by navigating to **View** > **ClearIn Point** (Alt+I) and **Clear OutPoint** (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.

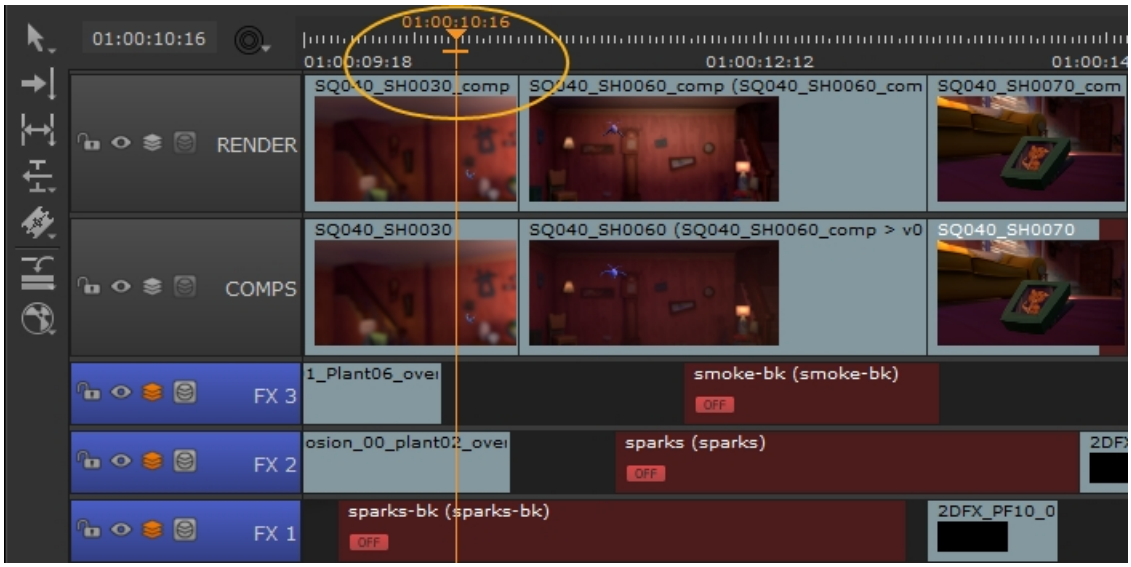
When the playhead is positioned near In and Out markers, the top half of the timecode scale controls the playhead and bottom half controls the markers.



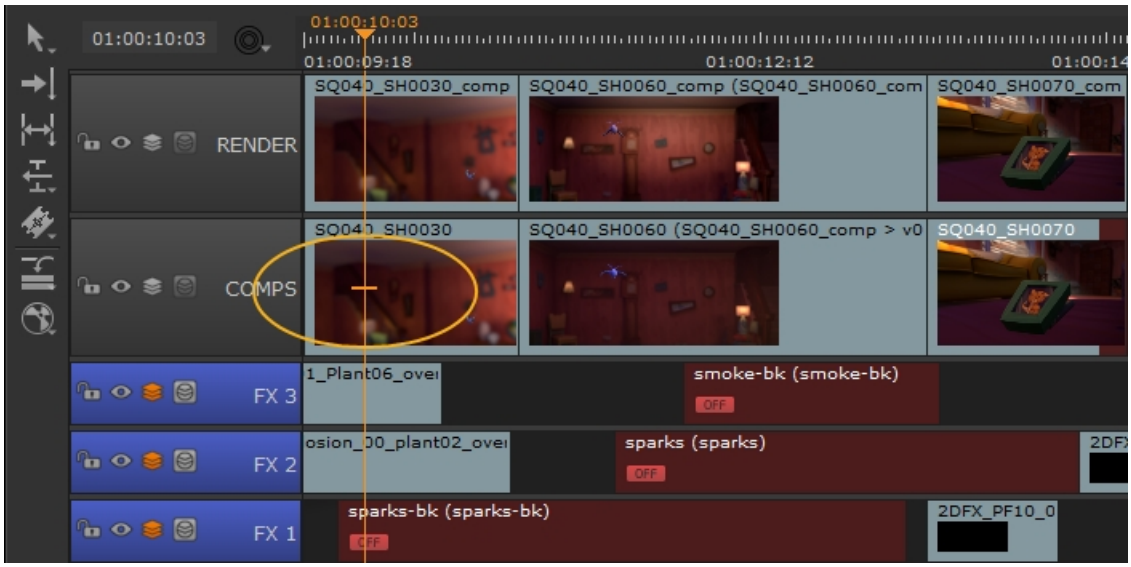
Playhead A/B Indicators

The timeline playhead indicates what is currently displayed in the Viewer using a number of different display markers. Swapping tracks between buffers automatically updates the playhead so that you always know which tracks you are looking at in the Viewer.

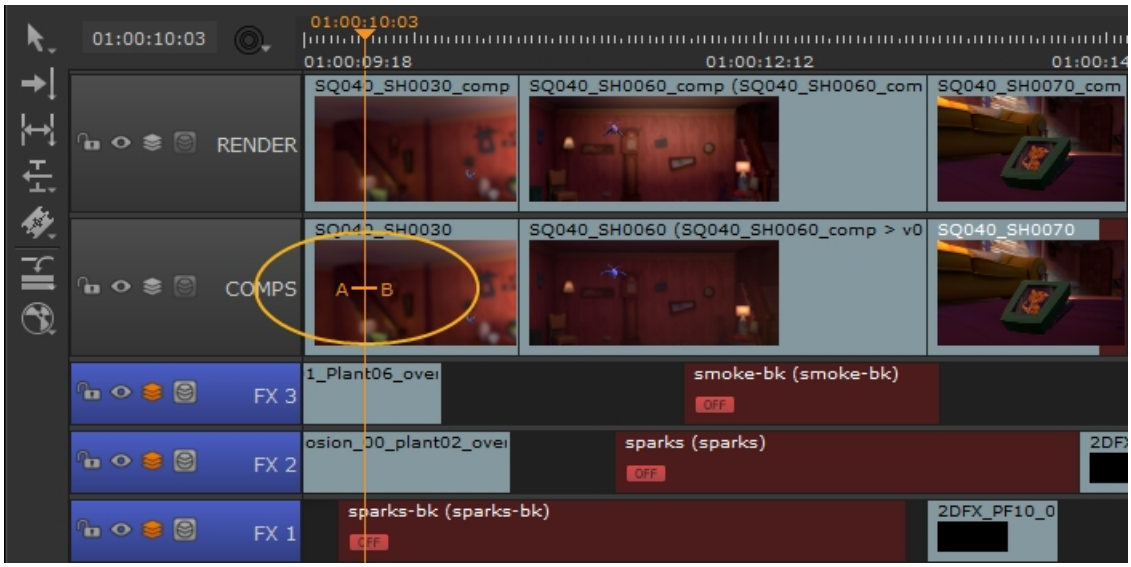
- Selecting all tracks in the current Viewer buffer draws a single line at the top of the playhead to indicate that you're viewing all tracks in the sequence from top to bottom.



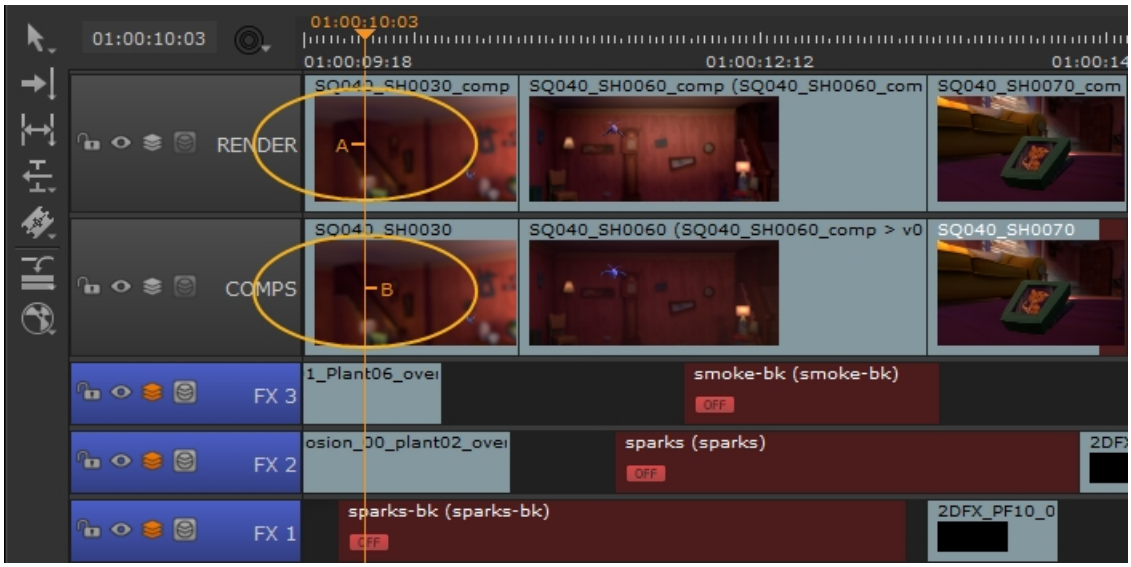
- Adding a track to the current Viewer buffer draws a single line at the playhead in the selected track.



- Adding the same track to both the A and B Viewer buffer draws a single line at the playhead in the selected track and shows A and B markers.



- Adding tracks to the Viewer A/B buffers draws A and B markers at the playhead in the selected tracks.



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.

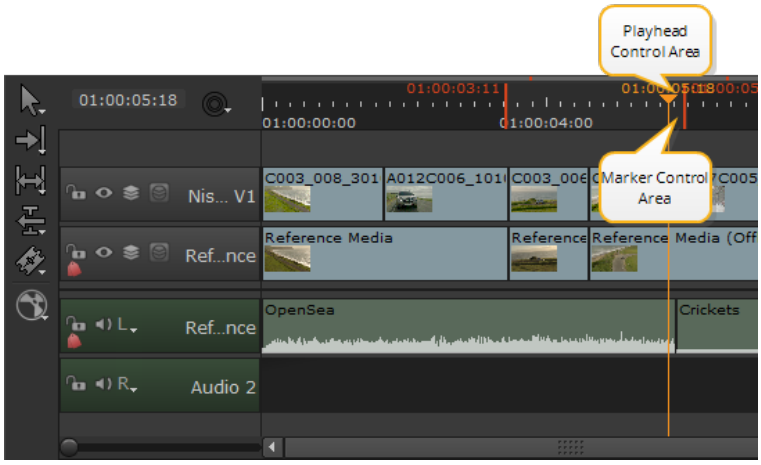
Note: You can also set markers by navigating to **View** > **Mark In** or **Mark Out**, by using the **Timeline** menu to **Mark Selection** or **Mark Clip** dependent on clip selections on the timeline, or by right-clicking a shots and selecting **Open In** > **Viewer**.

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.

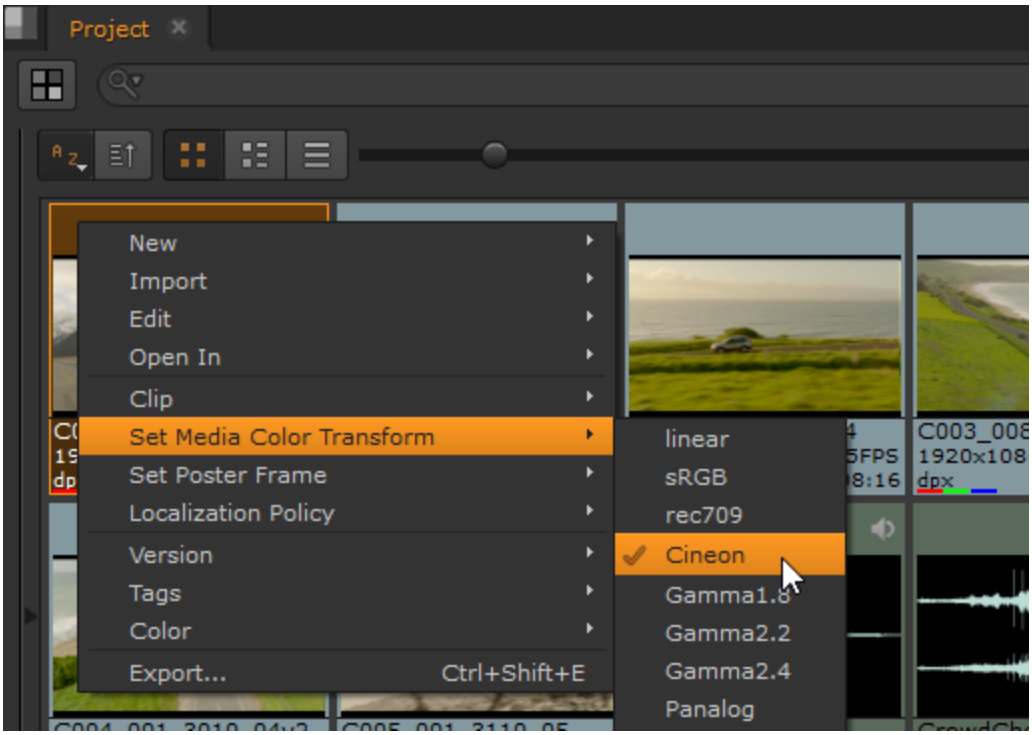


Working with Colorspaces

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:

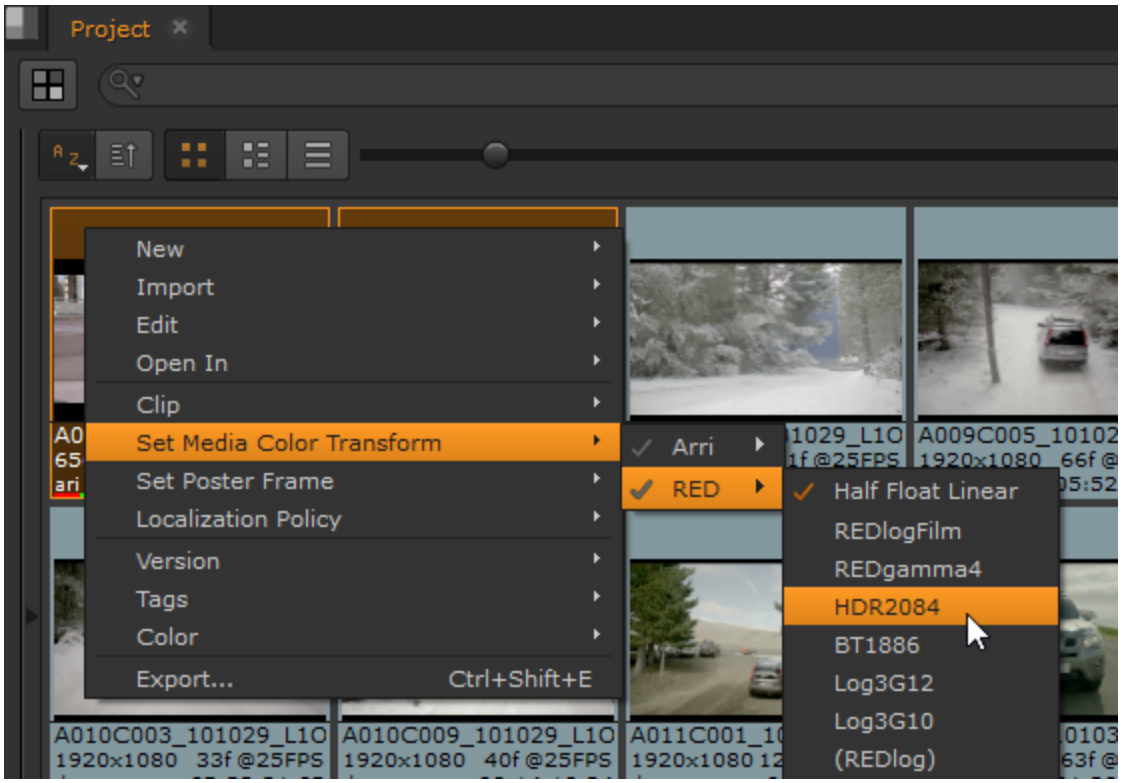
1. Select the clip or clips in the bin view.
2. Right-click a selected clip and navigate to **Set Media Color Transform**.



   The current colorspace is highlighted with a tick mark.
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:



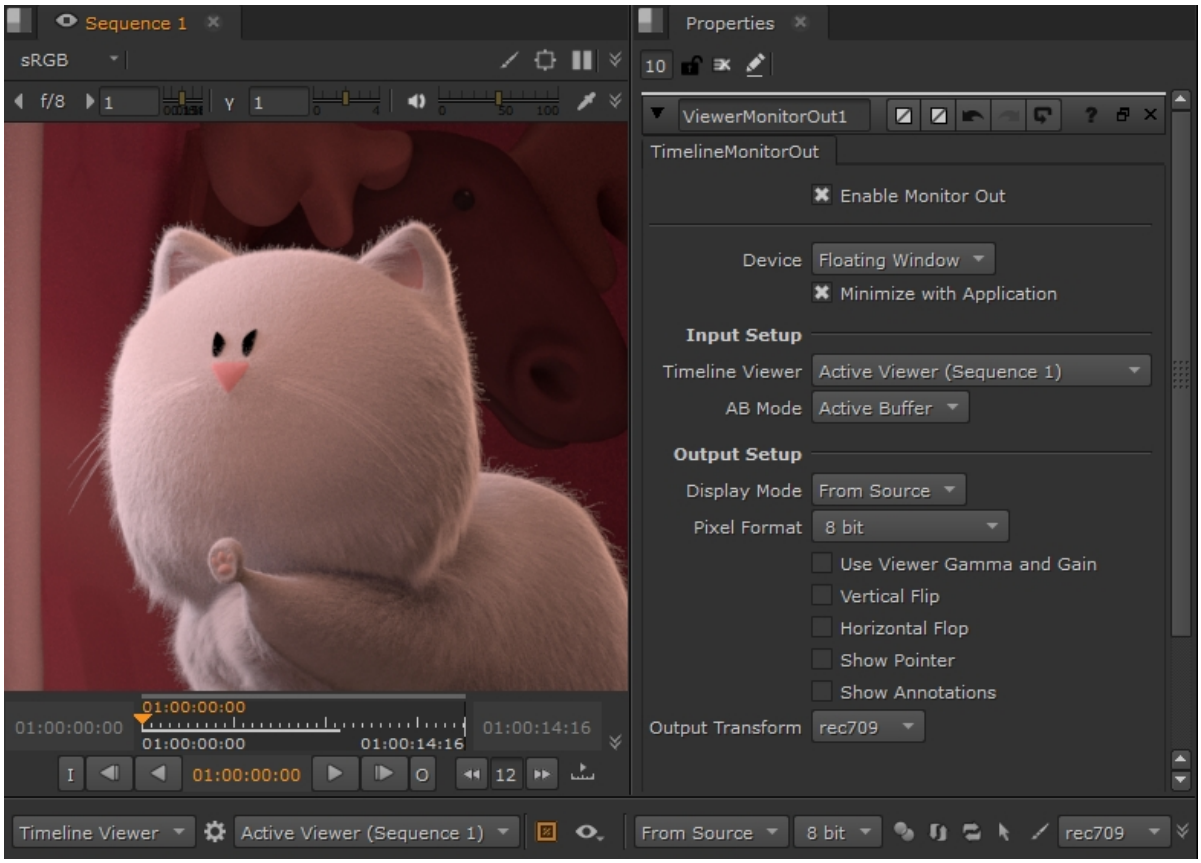
To apply colorspace changes to shots:

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.

Monitor Output in Hiero (SDI/HDMI)

The Monitor Out feature allows you to preview a timeline Viewer image in a floating window, NDI session, or on an external projector/video monitor to check the final result, including the correct output transform and aspect ratio.

A dedicated Viewer Monitor Out workspace provides easy access to the monitor out strip and its controls, but you can also access the controls through the ViewerMonitorOut in the node Properties panel.



Hiero supports internal monitor out using a Floating Window, NDI sessions, and dedicated AJA or Blackmagic cards, selected from the Device dropdown in the ViewerMonitorOut's Properties:

- Floating Window - Hiero's native monitor out uses a floating monitor and does not require additional hardware. Mac users can also take advantage of additional HDR technology including an extended sRGBf LUT. See Working with HDR Images for more information.

- NDI - A network protocol that enables video and metadata signals to be sent over standard IP networks in real-time. This option allows you to easily share a stream of the Viewer output to anyone else on the same network.

Note: To view the NDI stream, you need the NDI Studio Monitor or NDI Video Monitor application installed. See Streaming Viewer Output Over the Internet with NDI® for more information.

- Dedicated AJA or Blackmagic Design cards - Hiero supports various AJA and Blackmagic devices that can be selected from the Device dropdown after installation. See AJA and Blackmagic Design Monitor Out Devices for more information on supported hardware and drivers.

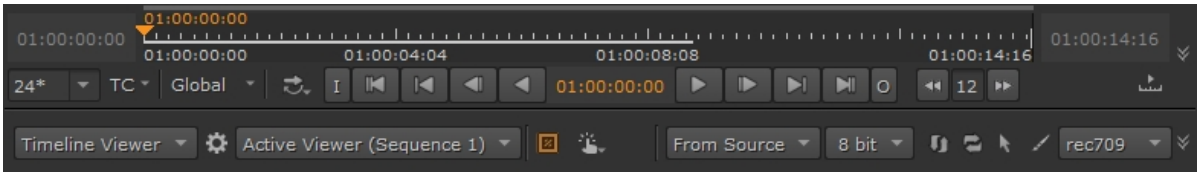
Note: Audio scrubbing is not currently available through monitor out cards. Audio scrubbing is only supported through internal audio output devices.

Enabling Hiero Monitor Out

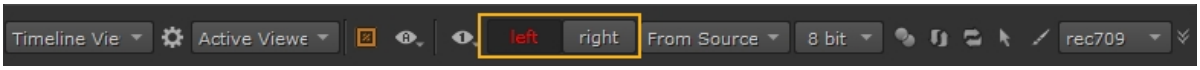
Hiero's monitor out uses a floating monitor, NDI session, or additional hardware from AJA or Blackmagic to display monitor out. The floating monitor allows Mac users to take advantage of additional HDR technology including an extended sRGBf LUT. See Working with HDR Images for more information.

1. Navigate to **Workspace** > **Viewer Monitor Out**.

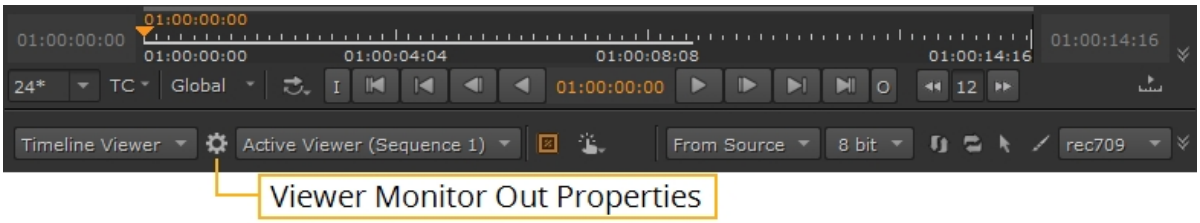
   The **Viewer Monitor Out** workspace is displayed with the Monitor Out toolbar under the Viewer by default.



   If you're working with multi-view footage in Interactive buffer mode, the monitor shows the output from the selected Viewer including the A/B player mode currently in use, such as wipe or difference. In Active, Buffer A, or Buffer B mode, additional controls display to determine the stereo mode and view to output. See Stereoscopic and Multi-View Projects for more information.



2. Click the cog icon in the Viewer Monitor Out panel to display the Properties panel.



3. Select the device you want to use from the output Device dropdown. All available devices are automatically detected and listed in this menu, along with the default options:
   - **Floating Window** - opens a floating monitor window, without the need for a monitor device and card. This is intended for full-screen use without displaying any of the interface.

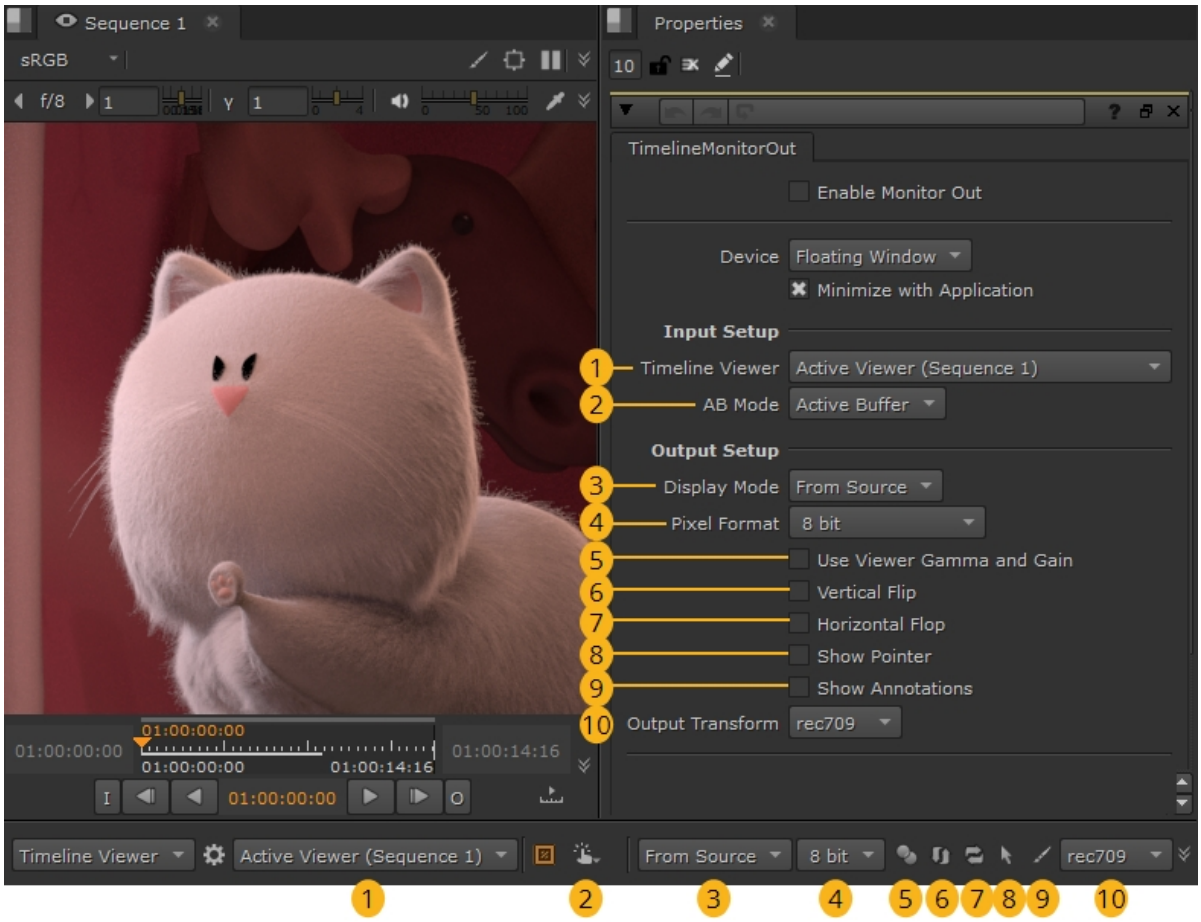
Tip: Disable Minimize with Application to allow the floating window to remain visible when Hiero is minimized.

   - NDI - A network protocol that enables video and metadata signals to be sent over standard IP networks in real-time. This option allows you to easily share a stream of the Viewer output to anyone else on the network.

Note: To view the NDI stream, you need the NDI Studio Monitor or NDI Video Monitor application installed. See Streaming Viewer Output Over the Internet with NDI® for more information.

   - Installed Cards - transfers the Viewer contents to the selected card and displays SDK and driver information associated with the card.

   The Viewer Monitor Out and Properties panel duplicate some controls for convenience.



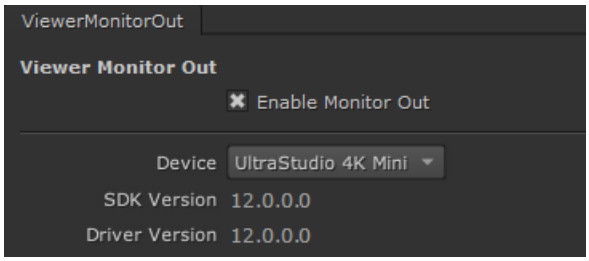
1	Viewer	Select the Viewer to feed to the output monitor using the source viewer dropdown. Selecting **Active Viewer** always displays the Viewer that is currently selected in the workspace.
2		Select the view mode using the AB Mode dropdown: • Active Buffer - the monitor always shows the output from the buffer currently in use by the selected Viewer. • Buffer A - the monitor always shows the output from the selected Viewer buffer A. • Buffer B - the monitor always shows the output from the selected Viewer buffer B. • Interactive - the monitor shows the output from the selected Viewer including the A/B player mode currently in use, such as wipe or difference.

3	Display Mode	Select the resolution output from the monitor card currently in use. The resolutions available depend on the card manufacturer. **Note:** Floating monitor out always uses the resolution of the Viewer in use.
4	Pixel Format	Select the pixel format output from the monitor card currently in use. The formats available depend on the card manufacturer.
5		Click the button or enable Use Viewer Gamma and Gain to apply the active Viewer's filtering, gamma, and gain to the monitor out.
6		Click the button or enable Vertical Flip to flip the output vertically. This control is handy if you're using a ceiling mounted projector.
7		Click the button or enable Horizontal Flip to flop the output horizontally.
8		Click to enable the mouse pointer position on the monitor. You can control the size of the cursor on the monitor in the Preferences under Panels > Viewer (Monitor Out) > Mouse Pointer Scale.
9		Click to display annotations active in the selected timeline Viewer on the monitor for review purposes. See Annotations for more information.
10	Output Transform	Select the color transform to apply to the image. If you've specified an OCIO configuration file in the preferences, these custom LUTs are also applicable.

AJA and Blackmagic Design Monitor Out Devices

Our monitor out architecture interfaces directly with the AJA and BlackMagic Design 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.

You can select installed cards from the MonitorOut's Properties panel using the Device control. The SDK and driver versions for the selected card are displayed for convenience.



We've tested the following AJA and Blackmagic Design hardware:

AJA Card:	KONA 4	KONA 5	KONA iO4k+	T-Tap Pro
See https://www.aja.com/family/desktop-io and https://www.aja.com/products/t-tap-pro#techspecs for more details on supported formats.				
Connection				
BNC	●	●	●	●
HDMI	●	●	●	●
Stereoscopic Support				
	Yes	Yes	No	No
Platforms				
	Win, Mac, Linux	Win, Mac, Linux	Win, Mac, Linux	Win, Mac
Drivers				
	Driver 16.2.5 or later, available here: • Windows: https://www.aja.com/assets/support/files/9149/en/AJA-Software-Installer_Windows_v16.2.5_Release.zip • macOS: https://www.aja.com/assets/support/files/9146/en/AJA-Software-Installer_macOS_v16.2.5_Release.zip • Linux: https://www.aja.com/assets/support/files/9140/en/AJA-Software-Installer_Linux_RedHat_CentOS_v16.2.5_Release.zip			

Blackmagic Card:	UltraStudio 4K Mini	4K Extreme	4K Extreme 12G	8K Pro
See https://www.blackmagicdesign.com/uk/products/ for more details on supported formats per card.				
Connection				
BNC	●	●	●	
HDMI			●	●
Stereoscopic Support				
	No	No	Yes	No
Platforms				
	Win, Mac, Linux	Win, Mac, Linux	Win, Mac, Linux	Win, Mac, Linux
Drivers				
	Driver 12.4.2.0 or later, available here: • **Windows**: https://www.blackmagicdesign.com/support/download/2438c76b9f734f69b4a914505e50a5ab/Windows • **macOS**: https://www.blackmagicdesign.com/support/download/2438c76b9f734f69b4a914505e50a5ab/Mac%20OS%20X • **Linux**: https://www.blackmagicdesign.com/support/download/2438c76b9f734f69b4a914505e50a5ab/Linux			

Blackmagic Card:	DeckLink Studio 4K	DeckLink 4K Extreme	DeckLink 4K Extreme 12G
See https://www.blackmagicdesign.com/uk/products/ for more details on supported formats per card.			
Interface			
BNC	●	●	●

Blackmagic Card:	DeckLink Studio 4K	DeckLink 4K Extreme	DeckLink 4K Extreme 12G
HDMI	●	●	●
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			
	Driver 12.4.2.0 or later, available here: • **Windows**: https://www.blackmagicdesign.com/support/download/2438c76b9f734f69b4a914505e50a5ab/Windows • **macOS**: https://www.blackmagicdesign.com/support/download/2438c76b9f734f69b4a914505e50a5ab/Mac%20OS%20X • **Linux**: https://www.blackmagicdesign.com/support/download/2438c76b9f734f69b4a914505e50a5ab/Linux		

Some monitor out cards allow you to extend or mirror your desktop so that the user interface is visible on the monitor. Please refer to your card's documentation for more information.

Working with HDR Images

Hiero handles HDR (high dynamic range) images in the same way as any other image, and so manipulating HDR footage in Hiero does not compromise the range of data available. The only difference is that you can't view HDR output in Hiero's native Viewer because most regular monitors don't support HDR.

Nuke, Nuke Studio, and Hiero support viewing HDR output through approved monitors and Monitor Output devices such as AJA, Blackmagic Design, and Bluefish SDI/HDMI cards. See AJA and Blackmagic Design Monitor Out Devices for more details on supported cards. The following table contains details of the output devices we've tested and approved.

Device	Operating System	Connection
Dell UP3221Q	Linux and Windows	HDMI
	macOS (Catalina 10.15.x, or later)	Thunderbolt
LG EP950	Linux and Windows	HDMI
	macOS (Catalina 10.15.x, or later)	Thunderbolt
Apple XDR	macOS (Catalina 10.15.x, or later)	Thunderbolt

EDR Displays - Mac Only (Beta)

If you're running Hiero on macOS Catalina or Big Sur and have a compatible screen capable of displaying values above 1, you can view your projects with HDR luminance ranges using sRGBf in OCIO Color Management. You can also display images in the P3 gamut, giving more accurate color on wide gamut displays.

Note: This feature requires a 2019 Mac Pro running macOS 10.15, or later, and a suitable display or a current generation iMac Pro, iMac or MacBook Pro with an HDR screen.

1. On an Apple XDR, set the profile of your display to one of the HDR options in the OS System Display Preferences:

- Pro Display XDR (P3-1600 nits)
- Apple Display (P3-500 nits)
- HDR Video (P3-ST2084) - 1000 nits

On other monitors, enable the High Dynamic Range checkbox.

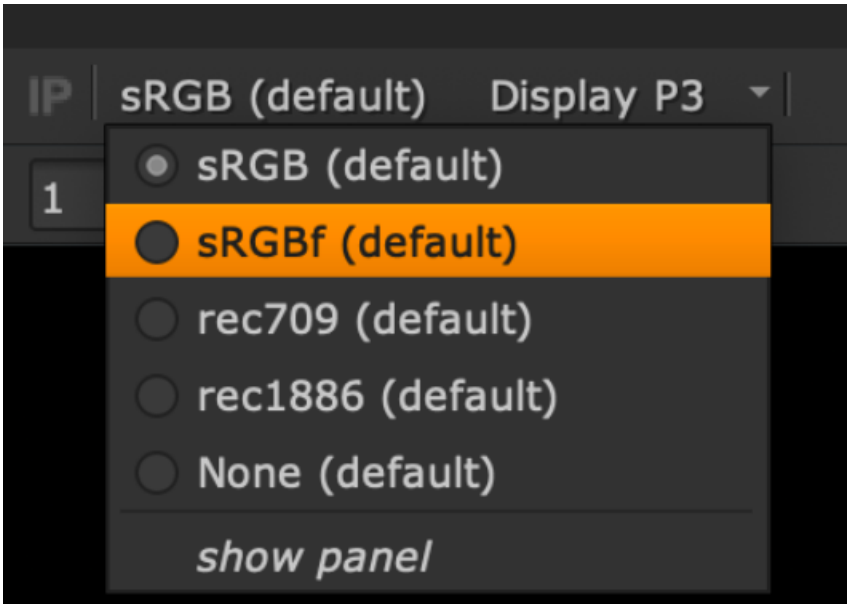
2. In the Preferences, open the Color Management > HDR section and check Enable macOS HDR Color Profile (Display P3) (Beta).

Note: Enabling this control requires you to restart the application.

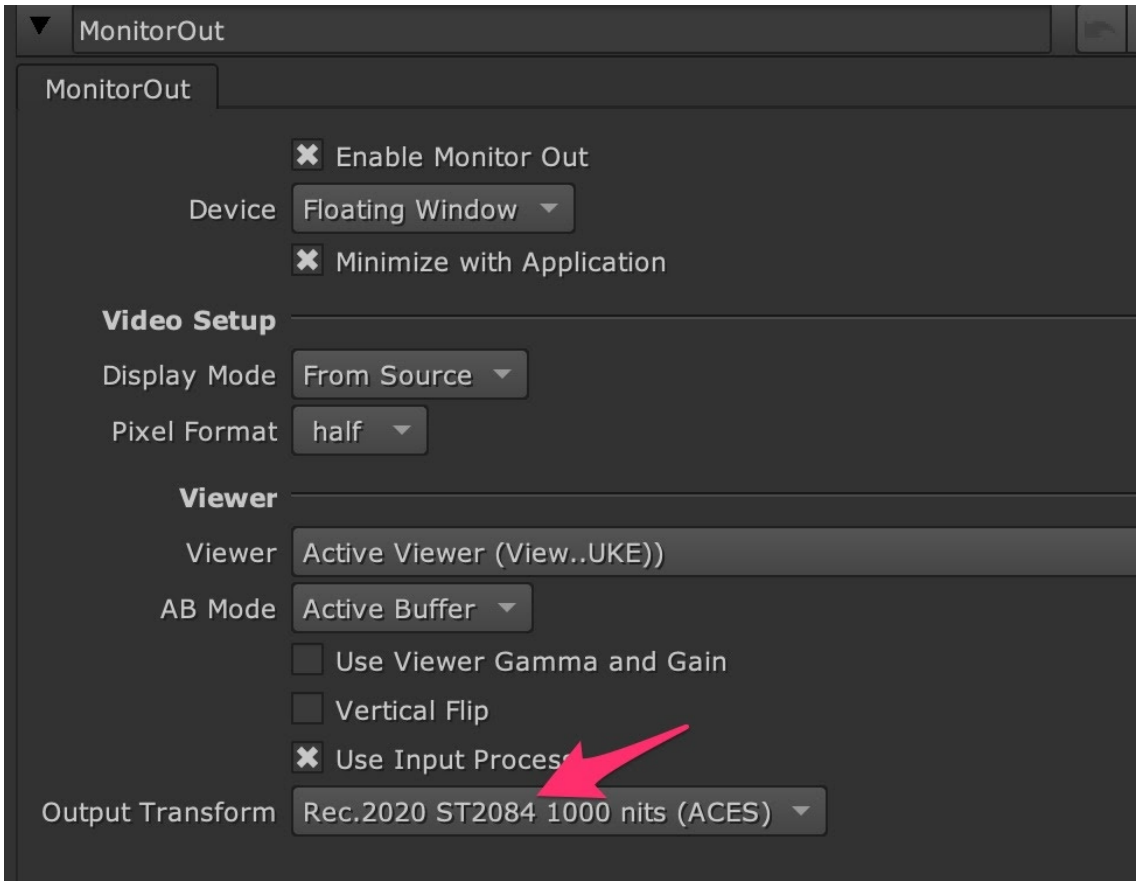
This control sets the Viewer colorspace to Display P3, so that the screen can display values above 1. This setting also applies the P3 colorspace to the following;

- All Viewers
- Node Graph
- Dope Sheet
- Curve editors
- Scopes

3. In the Viewer Properties, set the gl buffer depth to half-float or float to allow HDR values to display.
4. Set the Viewer Process to sRGBf using OCIO Color Management or provide your own extended color space. This ensures that the frame buffer is not clamped at 1.



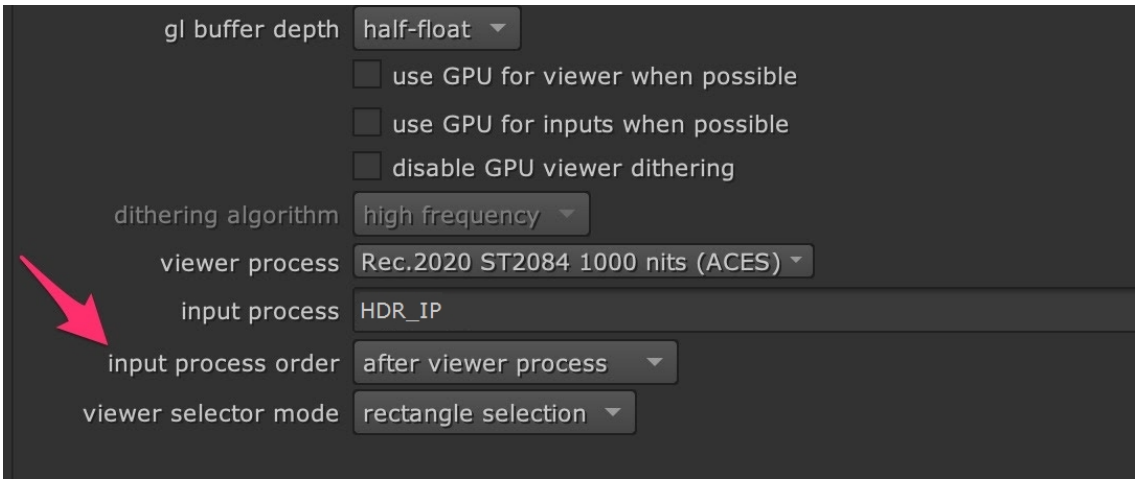
5. To use the HDR Video (P3-ST2084) setting on the monitor, select an appropriate ST2084 Output Transform in the MonitorOut Properties or in the Viewer.



A further color space conversion is required to allow HDR images to display correctly. An example of how this can be achieved in Nuke can be seen using a gizmo created by a customer, Nick Shaw - the gizmo is available for download here: PQ-EDR_v101.gizmo

For more information on gizmos and how to use them, see Accessing Gizmos in Nuke.

Using this gizmo as an Input Process and setting the input process order to after viewer process renders the correct image in the Viewer.



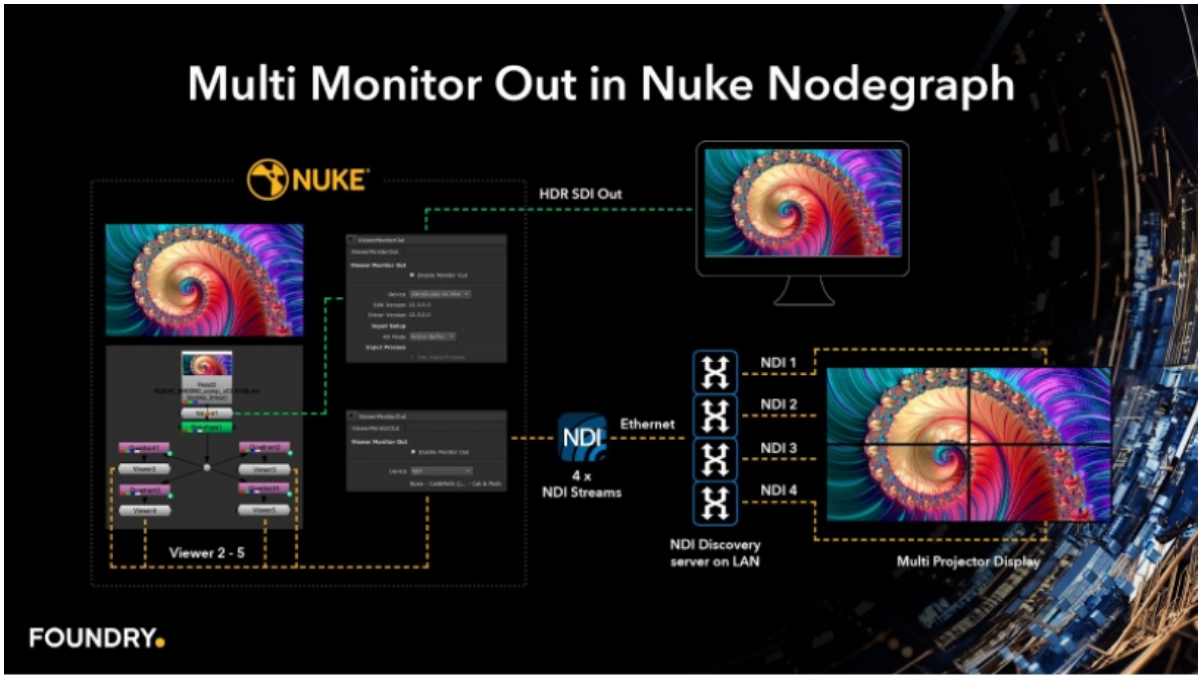
The gizmo also contains a slider called Normalization (nits). This allows you to set the PQ nit value which is normalized to 1.0. This is useful, for example, to prevent clipping of a 1000 nit PQ signal on the 16" MacBook Pro's 500 nit display.

Streaming Viewer Output Over the Internet with NDI®

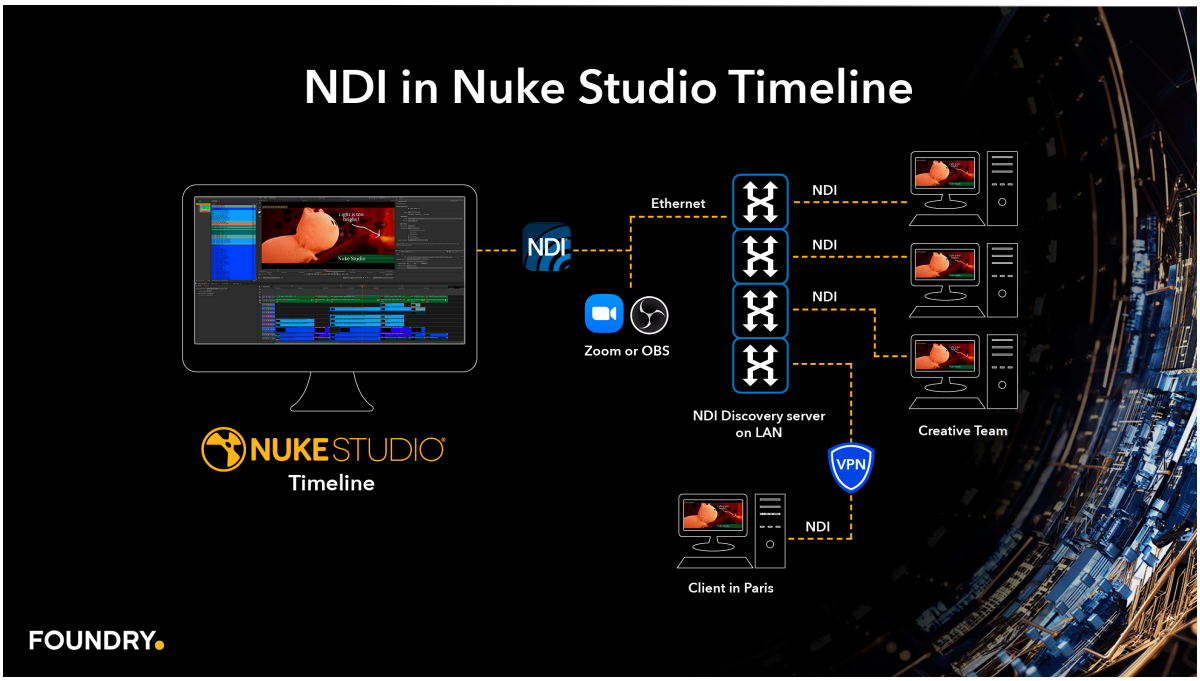
Network Device Interface, or NDI, is a network protocol from NewTek that enables you to send video, audio, and metadata signals over standard IP networks in real-time. NDI allows you to easily share a Nuke Viewer's output with anyone else on the same network.

Nuke's monitor out functionality is controlled by Viewer nodes, so you can compare or combine multiple streams from as many Viewers as you like. Each Viewer can be enabled separately to output your shots to multiple output devices, whether they're dedicated monitor out devices, NDI streams, or Nuke's native Floating Window. The example shows five Nuke Viewers in total. The first Viewer at the top of the image is feeding an HDR SDI monitor using a dedicated AJA or Blackmagic Design card and the other four are combined over NDI connections to a remote multi-projector display.

See Comparing and Combining Multi-Viewer Output with NDI for more details.



Nuke Studio's monitor out is controlled by the ViewerMonitorOut node, which only allows you to stream one timeline Viewer to a dedicated monitor out device, NDI stream, or Nuke's native Floating Window. Sharing your Viewer over an NDI connection is a great way to collaborate in a review session with your creative team on the same local network or over VPN to remote customers.



Installing the NDI Tools

To view an NDI stream, you need the NDI Studio Monitor (Windows) or NDI Video Monitor (macOS) application installed on your machine. The monitor searches your network for NDI signals from Nuke or Nuke Studio and allows you to select the stream you'd like to view.

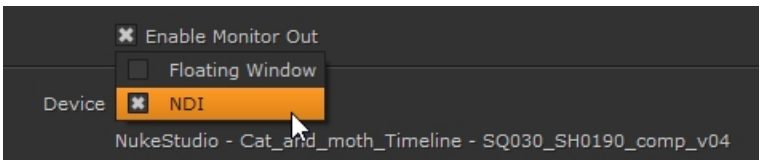
NDI Studio Monitor and NDI Video Monitor are available for download as part of the NDI Tools on Windows and macOS here:

- NDI Tools for Windows
- NDI Tools for macOS

Note: NDI streams require you to bypass the operating system's firewall software on the remote machine to work correctly. Installing the NDI Tools on the remote machine should set exceptions in the firewall automatically, but you can add exceptions manually as described under Setting Up an NDI Stream from Nuke and Nuke Studio and Adding Firewall Exceptions on macOS.

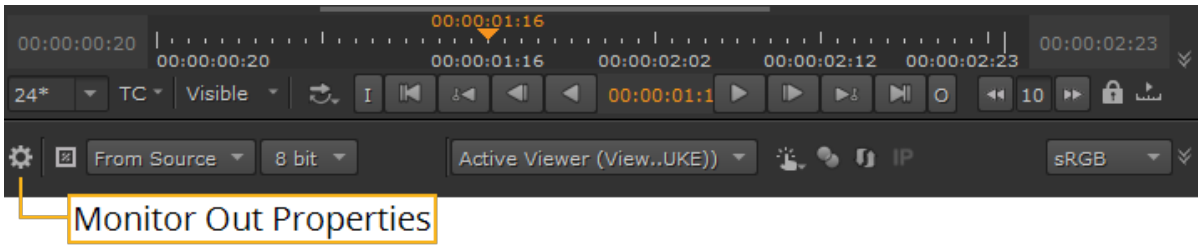
Setting Up an NDI Stream from Nuke and Nuke Studio

1. On the local machine in Nuke or Nuke Studio, go to Edit > Preferences, or press Shift+S, and navigate to Panels > Viewer Monitor Out.
2. Ensure that Enable NDI is checked.
3. In the monitor out Properties, set the monitor out Device to NDI to start broadcasting the stream over the network.

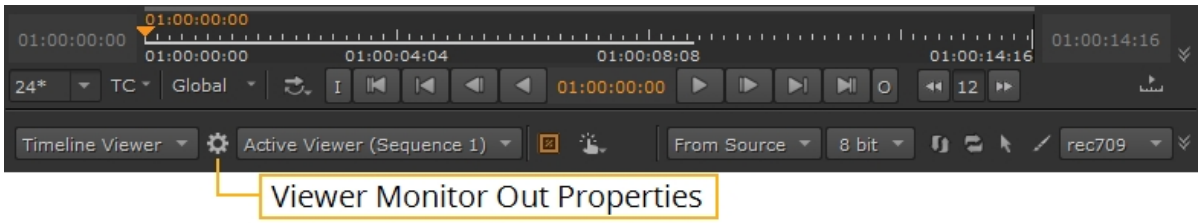


To display the Properties panel:

- Nuke - click the cog icon in the bottom-left of the Viewer Monitor Out panel or double-click the Viewer node.



- Nuke Studio - click the cog icon in the bottom-left of the Viewer Monitor Out panel.



4. On the remote machine, download and install the NDI Tools for your operating system from the links under Installing the NDI Tools.
5. Launch the NDI Studio Monitor (Windows) or NDI Video Monitor (macOS), depending on your operating system:

- Windows - In the NDI Studio Monitor, click the options icon at the top-left and then select the required stream.

- macOS - In the NDI Video Monitor, click the NDI menu at the top of the screen and then select the required stream.

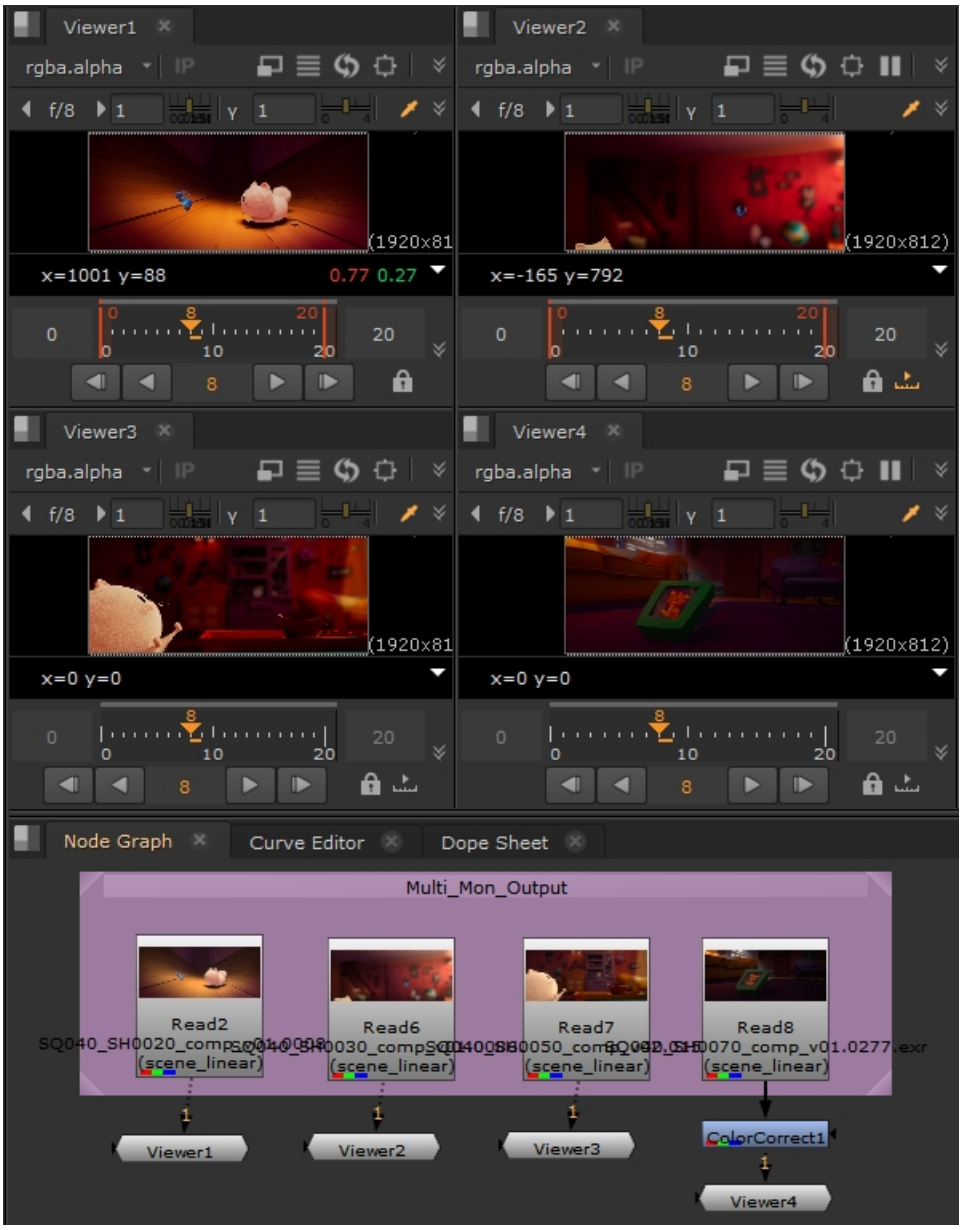
The stream name is shown in the Viewer Properties and on the NDI stream for identification and is constructed as follows: [Application] - [Project or Script] - [Viewer]

For example: Nuke - cat_and_moth - Viewer2

The Viewer is output through the NDI Studio Monitor or NDI Video Monitor.

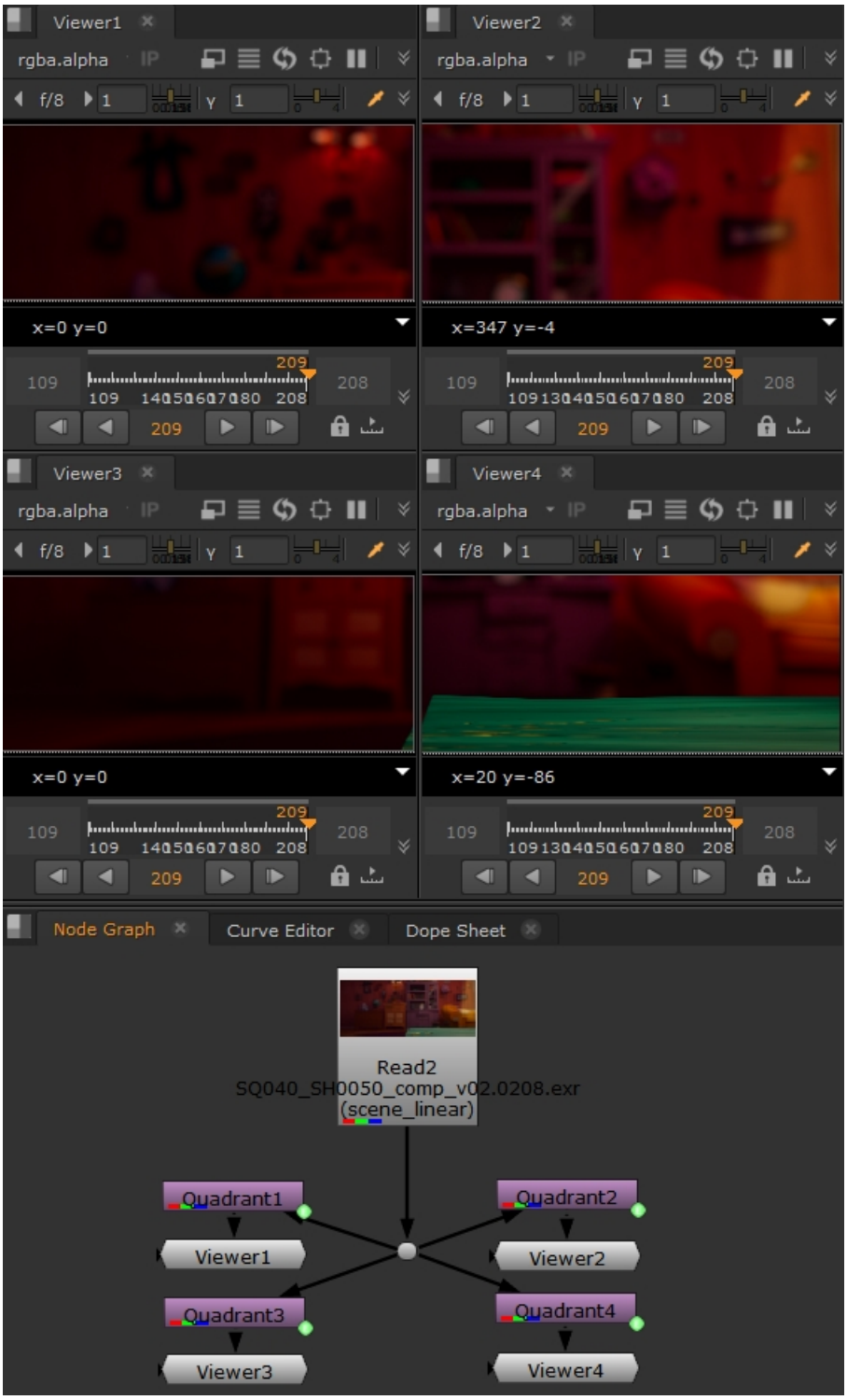
Comparing and Combining Multi-Viewer Output with NDI

Nuke's monitor out functionality is controlled by Viewer nodes, so you can compare or combine multiple streams from as many Viewers as you like. Each Viewer can be enabled separately to output your shots to multiple NDI streams. For example, using four Viewers and four NDI streams, you could compare the format of multiple shots.

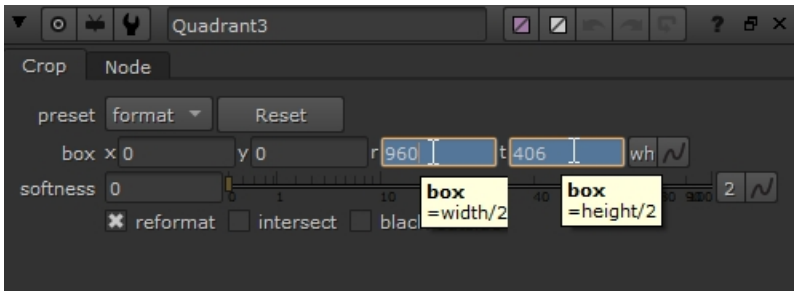


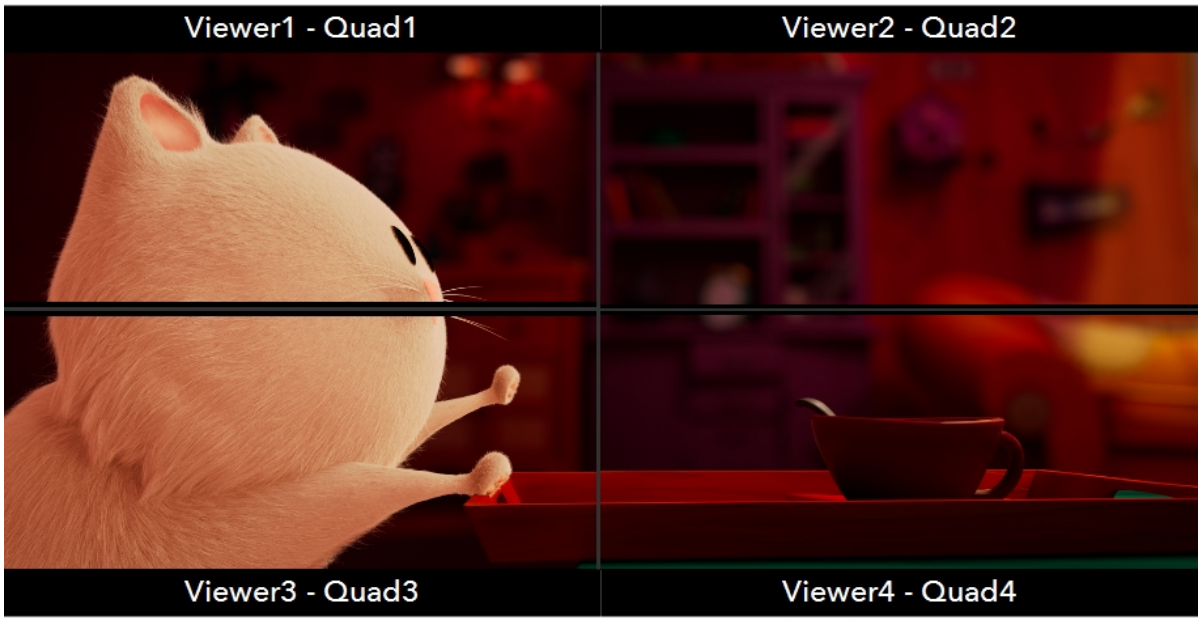
You can switch the Output Transform on each Viewer independently to check color consistency between the shots.

Using a similar four-Viewer setup, you could send four quadrants over NDI to four projectors to produce a larger image somewhere else on the same network.



Each quadrant is a Crop node that defines which part of the overall image is handled by each Viewer. For example, Quadrant3 is set up as shown here, with simple expressions to handle different formats.

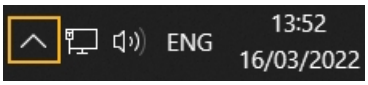




Adding Firewall Exceptions on Windows

Installing the NDI Tools on the remote machine should set exceptions in the firewall automatically, but you can add exceptions manually as follows:

1. Click the show hidden icons button on the right-hand side of the taskbar and then click Windows Security,

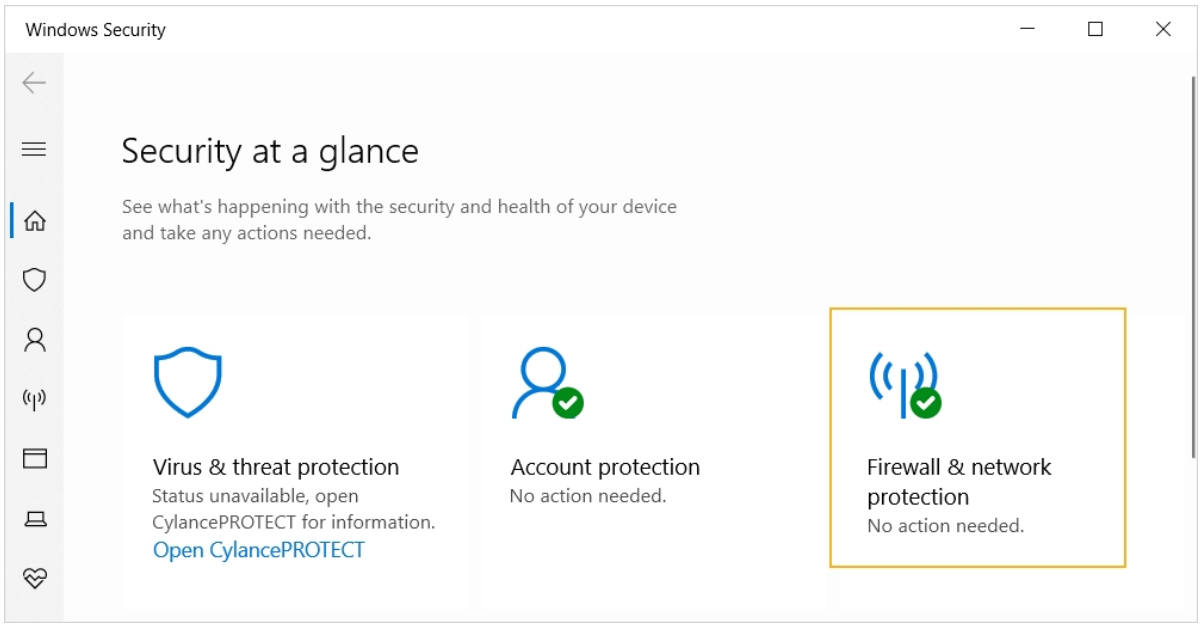
   

   OR

   Click the Windows Start button and type Windows Security and then click Windows Security.

   The Windows Security dialog is displayed.

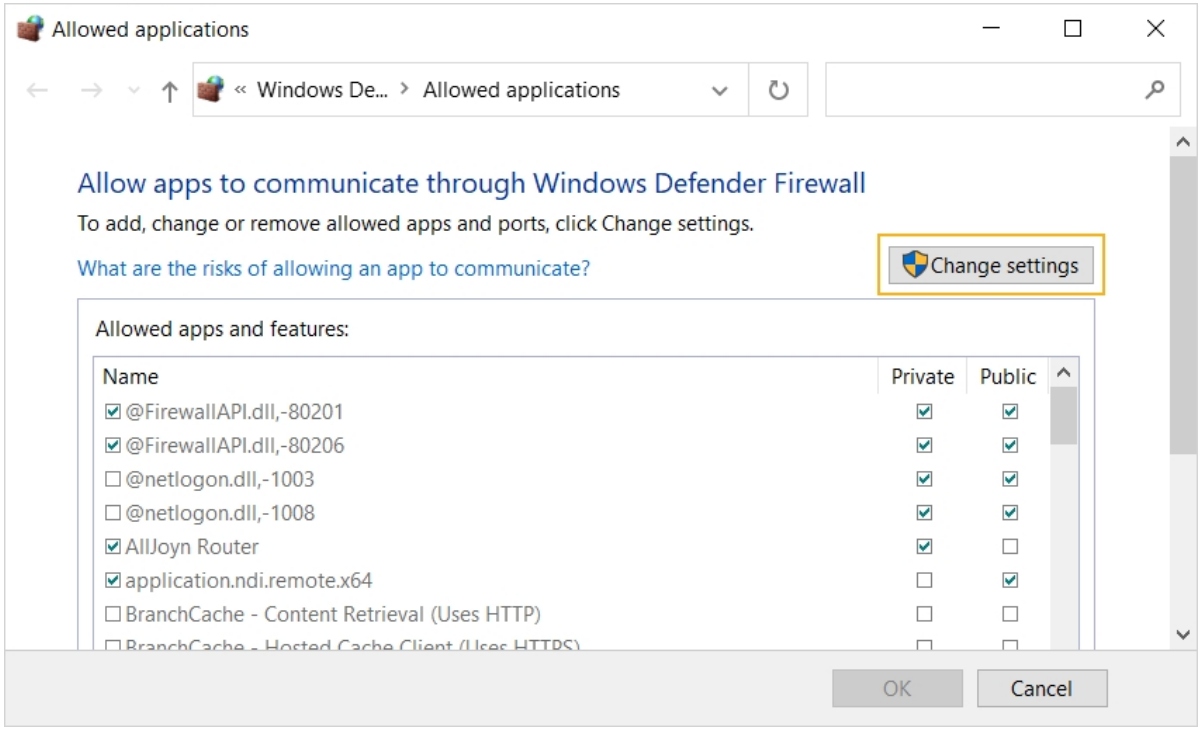
2. Click Firewall & network protection.

3. 

4. Click Allow an app through the firewall.

   The Allowed applications dialog is displayed.

5. Click Change settings to enable editing.

6. 

7. Scroll to the bottom of the window and click Allow another app.

8. Enter the file path to the NDI Studio Monitor executable or click Browse to locate the .exe using the explorer. NDI Studio Monitor is installed here by default:
   ```
   C:\Program Files\NDI\NDI 5 Tools\Studio Monitor\Application.Network.StudioMonitor.x64.exe
   ```
9. Click Add and then click OK to complete the firewall exception.

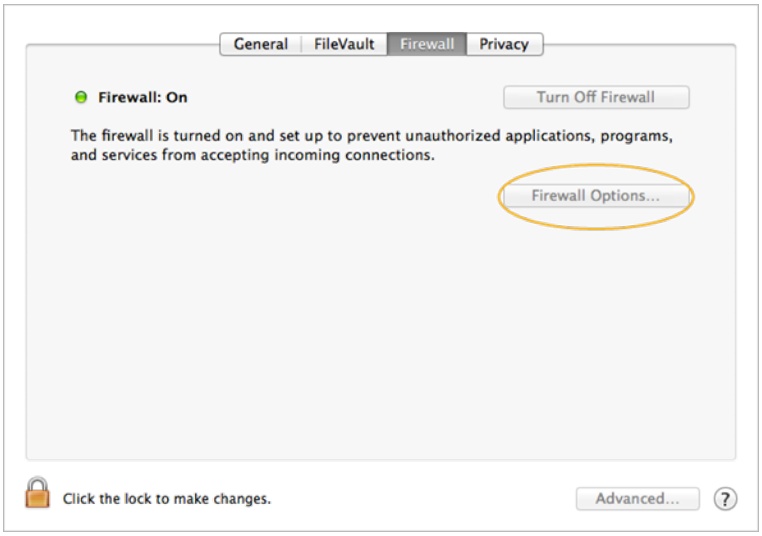
Refer to your operating system's documentation or speak to your network administrator for more details. For more information on the NDI Tools and network configuration best practice, see: https://ndi.tv/tools/education/networking/best-practices/networking-best-practice/

Adding Firewall Exceptions on macOS

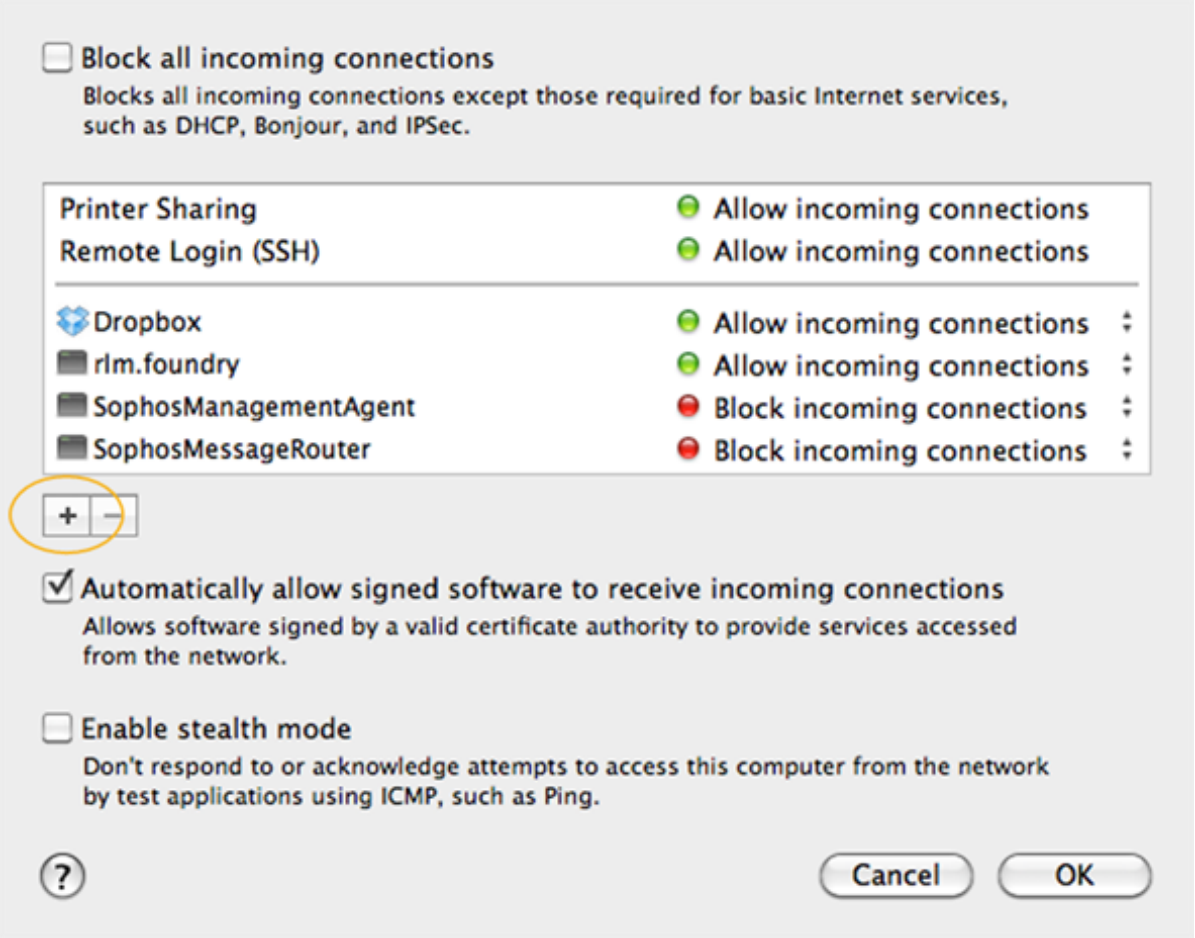
Installing the NDI Tools on the remote machine should set exceptions in the firewall automatically, but you can add exceptions manually as follows:

1. Launch System Preferences.
2. Click Security & Privacy and go to the Firewall tab.
3. Click the Firewall Options button.

> **Note:** You may need to click the lock icon at the bottom-left to enable changes.



4. Under the list of incoming connections, click on the **+** button.



5. Enter the file path to the NDI Video Monitor application or click Browse to locate the .app using the Finder. NDI Video Monitor is installed here by default:
   `/Applications/NDI Video Monitor.app`

6. Select **NDI Video Monitor.app** and click Add and then OK to complete the firewall exception.

Refer to your operating system's documentation or speak to your network administrator for more details. For more information on the NDI Tools and network configuration best practice, see: https://ndi.tv/tools/education/networking/best-practices/networking-best-practice/

Using Scopes

Hiero 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:

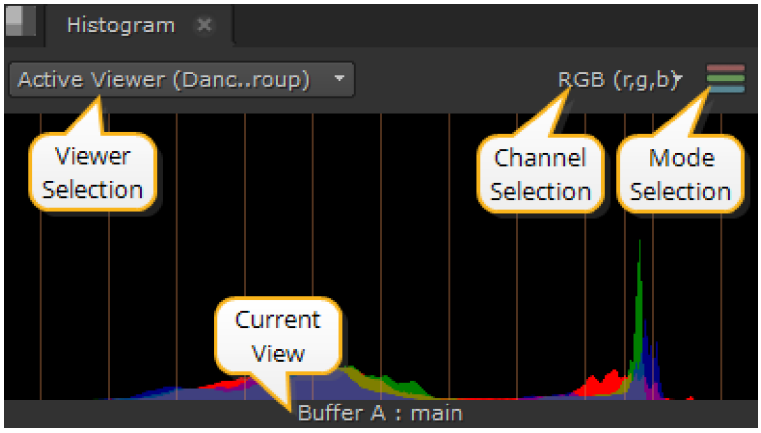
- **black point** - sets the black out of range warning level.
- **white point** - sets the white out of range warning level.
- **luma/chroma encoding** - sets the video standard to use when converting RGB to luma or chroma values in the scope displays, either **REC601** or **REC709**.
- **Include viewer 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.

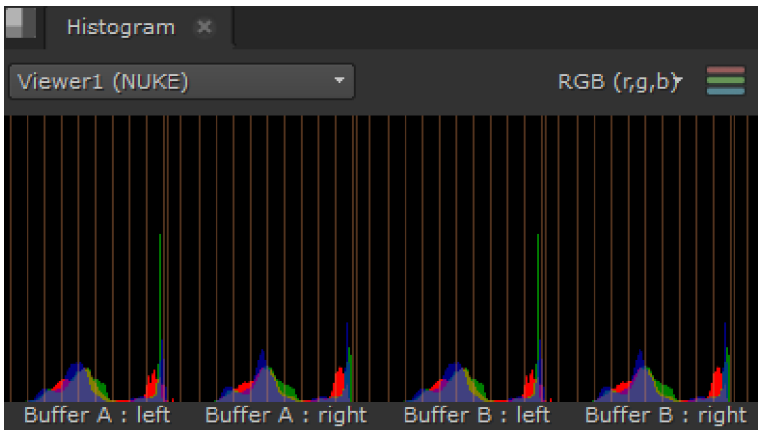
There are also **Viewer** and **Channel** selection controls on the **Histogram** tab:

- **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.

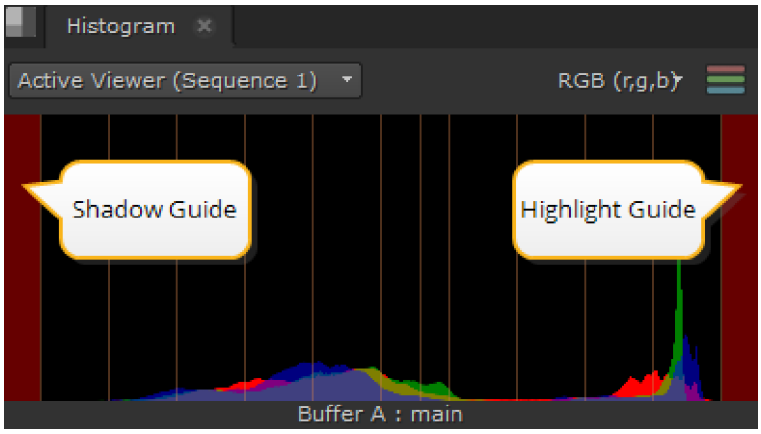
  Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

  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:



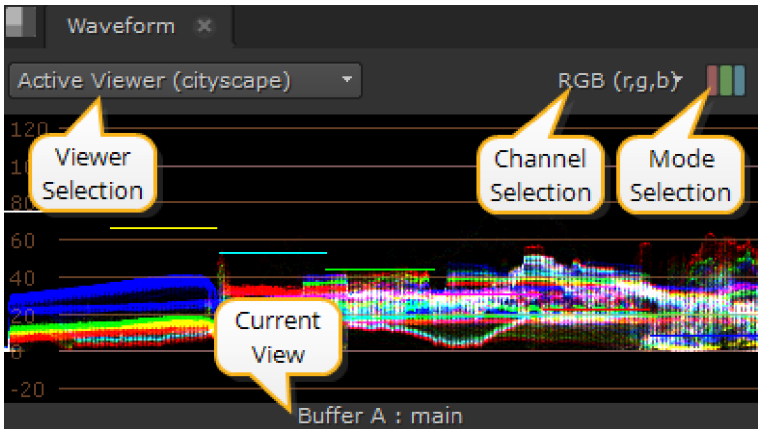
The scopes feature global customizable guides to help you view your clips. Navigate to **Preferences** > **Panels** > **Scopes** and enter values between **0** and **1** for the **Black** and **White** points. Note that this also sets the values for the Waveform display.

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:

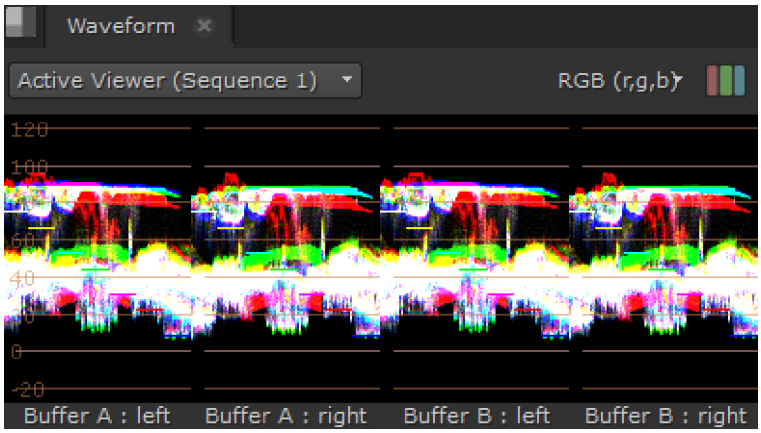
- **Viewer selection** - if you have multiple Viewers open, use the dropdown menu to associate Waveform output to the required clip.

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.

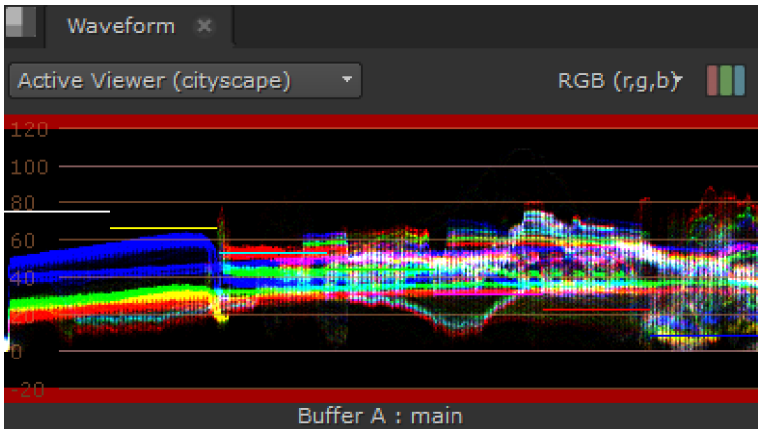
Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

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:



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.

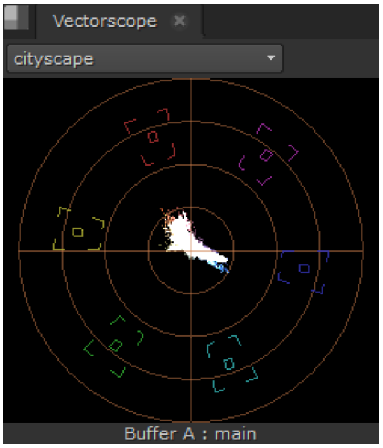
The guides at the top and bottom of the Waveform turn red to warn you when the distribution is out of range:



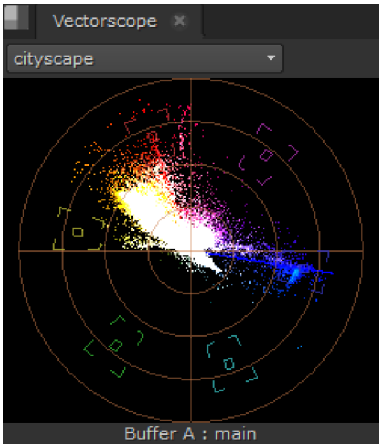
Vector

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).



Normal saturation.



High Saturation.

Tip: You can pan the view area by holding **Alt**, or the middle mouse button, and dragging in the panel.

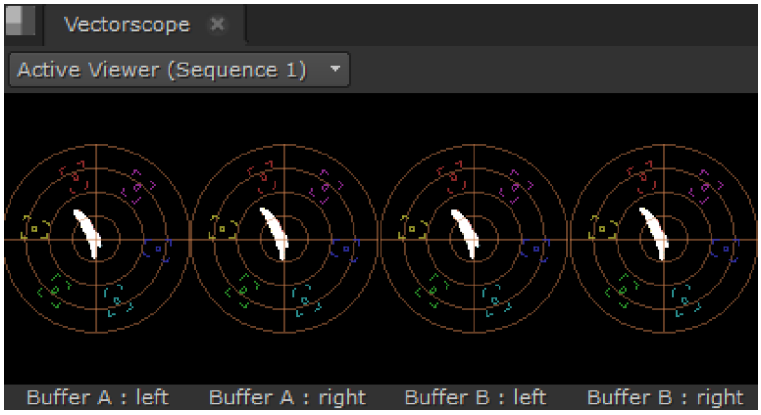
There is also a **Viewer** selection control and Current View label on the **Vectorscope** tab:

- **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.

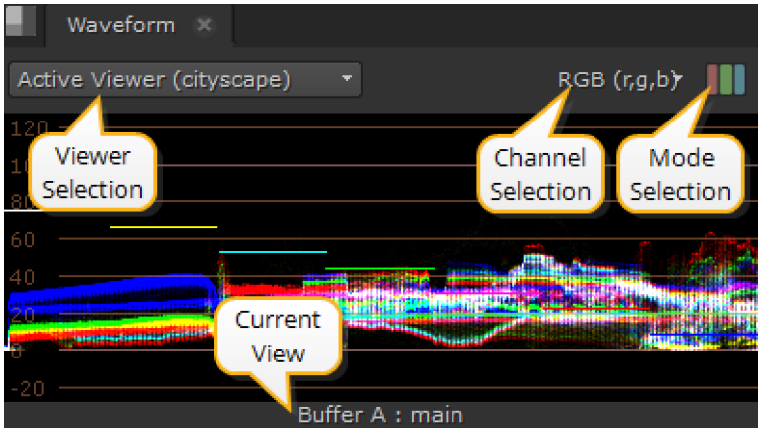
  Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

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:



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.

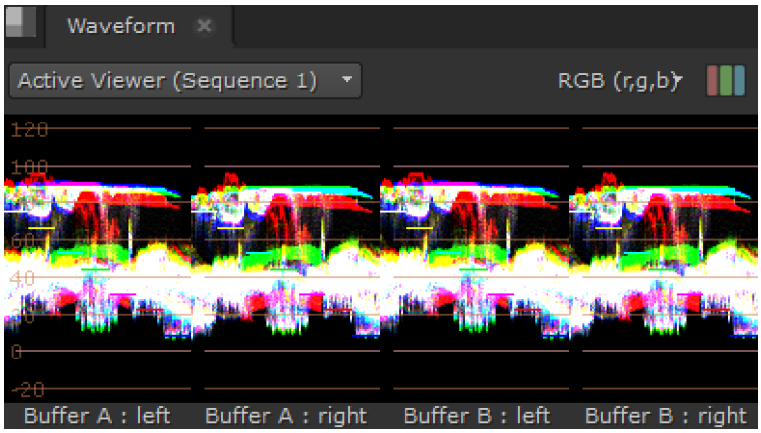
There are also **Viewer** and **Mode** selection controls on the **Waveform** tab:

- **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.

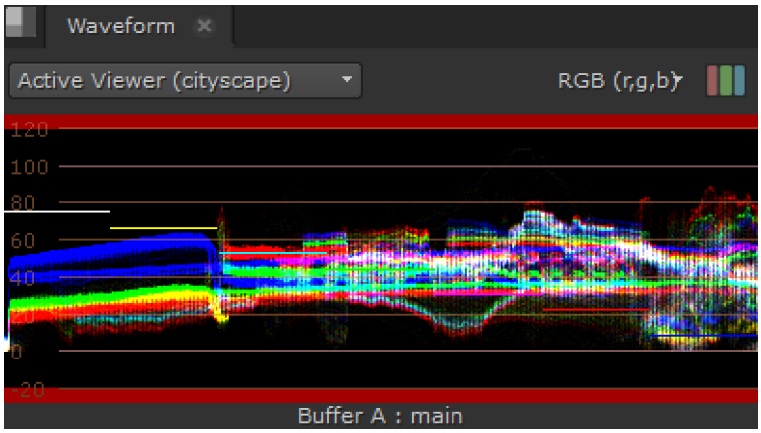
  Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

  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:



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.

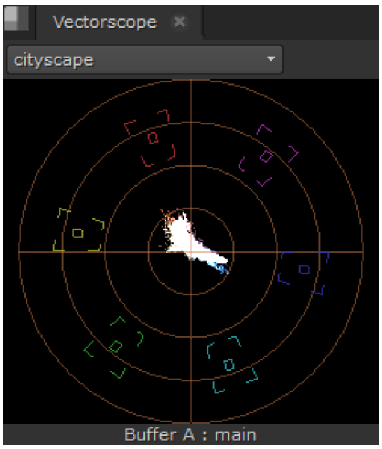
The guides at the top and bottom of the Waveform turn red to warn you when the distribution is out of range:



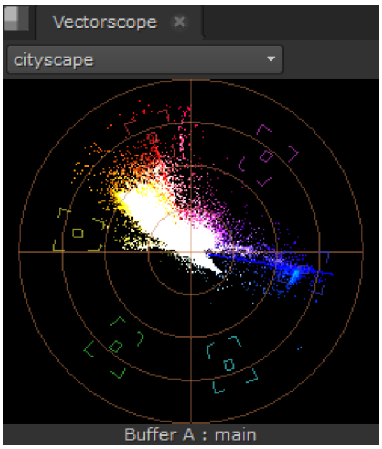
Vector

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).



Normal saturation.



High Saturation.

Tip: You can pan the view area by holding **Alt**, or the middle mouse button, and dragging in the panel.

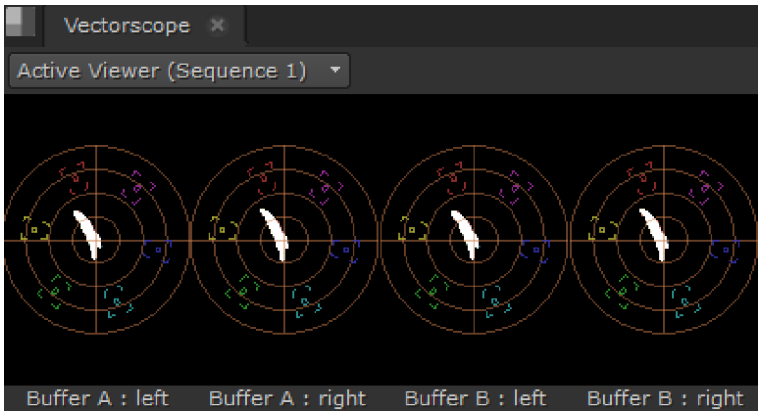
There is also a **Viewer** selection control and Current View label on the **Vectorscope** tab:

- **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.

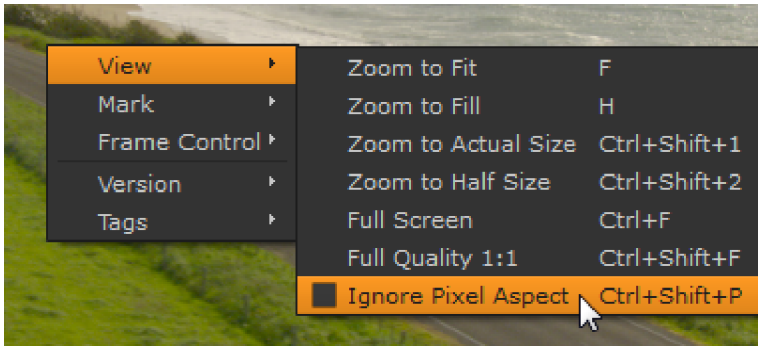
  For example, with two stereo Read nodes, one in each input buffer, and **wipe** and **Side by Side** active, the scopes display something like this:



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.

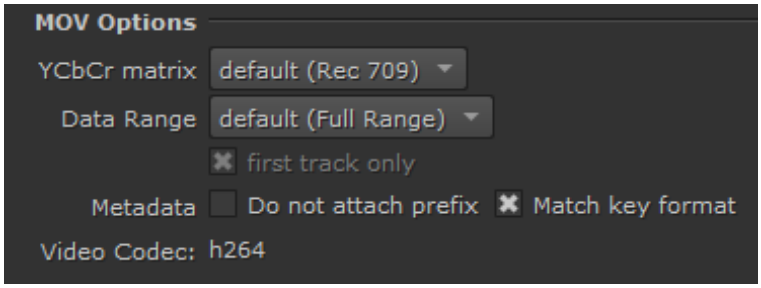


About QuickTime Media

Working with **.mov** files can be unpredictable when compared to other formats, so Hiero gives you a few QuickTime options when reading and writing **.mov** files.

Hiero 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 Hiero 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 Hiero.
- **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 Y'CbCr when dealing with a non-RGB codec.

In addition to the **nclc**, **gama**, and **ProRes** data Hiero, 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 Hiero 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.

Synching to VBlank on Linux

Hiero automatically accounts for tearing that can occur due to NVIDIA and OpenGL issues on Linux, but with dual screen setups, you need to specify which screen to synchronize. The __GL_SYNC_DISPLAY_DEVICE environment variable enables you to specify the sync device by appending the screen device name to the variable.

To determine your screen names:

1. From a command line, enter:
   `nvidia-settings`
   The **NVIDIA X Server Settings** dialog box displays.

2. Open up the **GPU** dropdown to display the screen device names.

3. Make a note of the device names, for example, **DFP-0** or **CRT-1**.
   Set the __GL_SYNC_DISPLAY_DEVICE environment variable value to the device name as described in Setting Environment Variables.

Using Tags

Tags are used by Hiero 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 Sequences as EDL, OTIO, and XML for more information.

Using Quick Tags

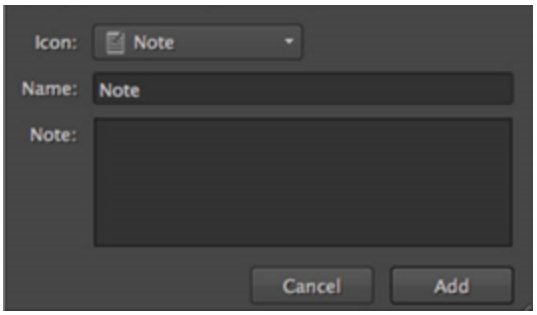
Quick tags allow you to add tags, depending on context, by right-clicking a selection and then choosing the type of tag to apply. If you're tagging a large amount of media, you might find it more convenient to use the drag-and-drop methods described later on.

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.



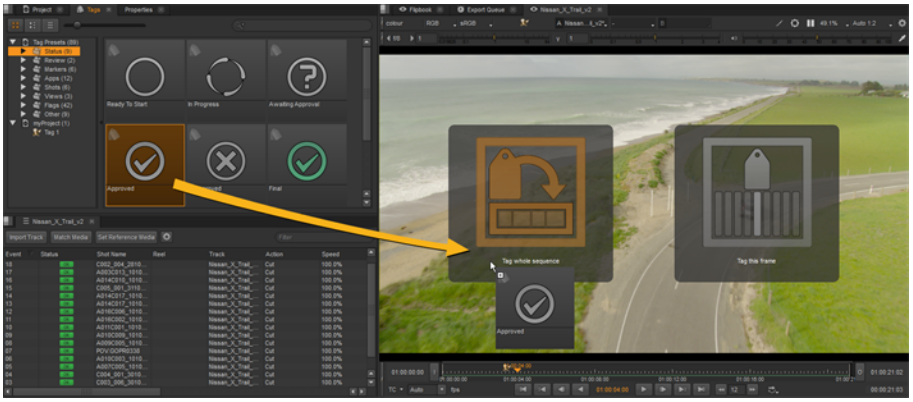
3. Select the icon to represent the tag using the **Icon** dropdown.
4. Enter a **Name** and **Note** as required.
5. Click **Add** to mark your selections with the chosen tag.

See Creating Custom Tags and Removing Tags for more information.

Tagging Using the Viewer

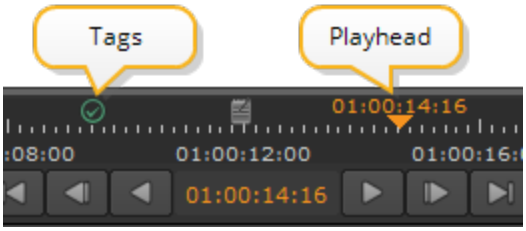
To apply a tag using the Viewer:

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.



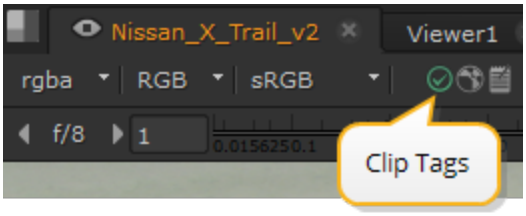
Depending on whether you're looking at a clip or sequence, drop the tag on **Tag this frame**, **Tag whole clip**, or **Tag whole sequence** as required.

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

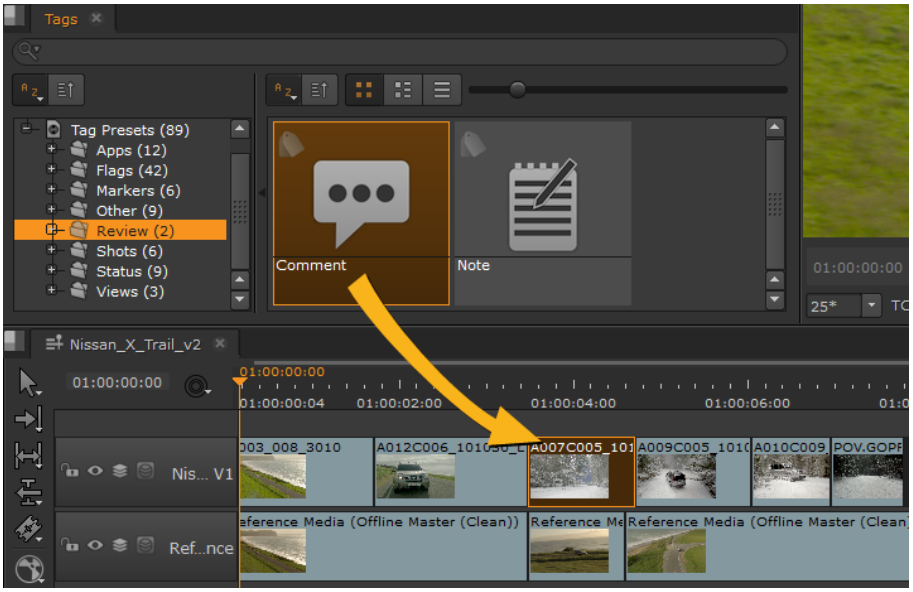
To apply a tag to a shot on the timeline:

1. Click the **Tags** tab, or navigate to **Window** > **Tags**.

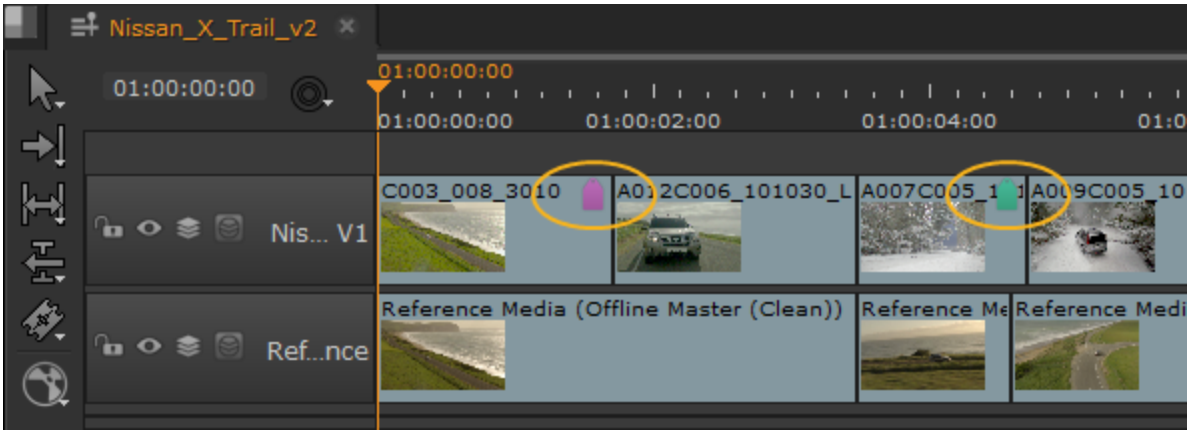
   The **Tags** panel displays.

2. Drag-and-drop the required tag from the **Tags** panel to the timeline.

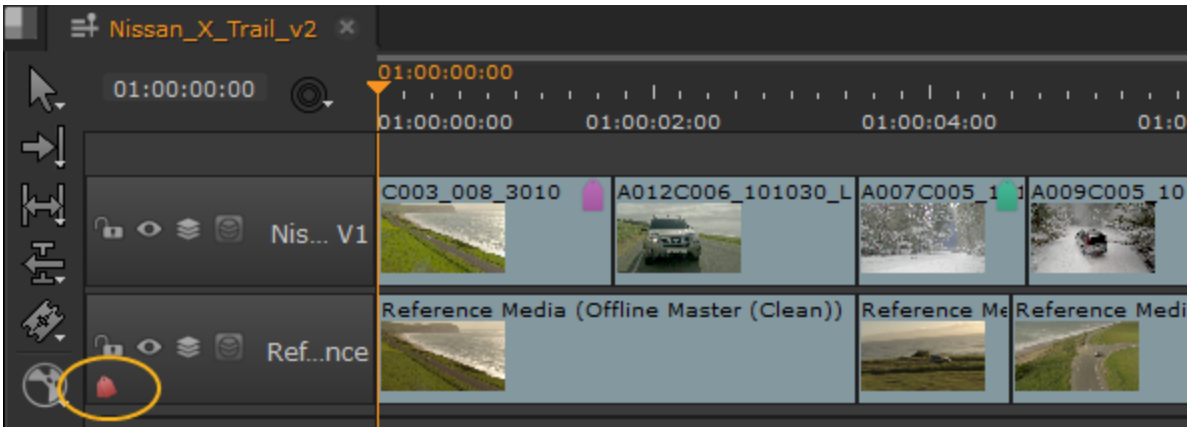
   Depending on where the tag is dropped, you'll mark a shot (or items if you make multiple selections) or a track.



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.


Nissan_X_Trail_v2
01:00:00:00
01:00:00:00
01:00:00:00
01:00:02:00
01:00:04:00
C003_008_3010
A012C006_101030_L
A007C005_1
A009C005_10
Nis... V1
Reference Media (Offline Master (Clean))
Reference Me
Reference Medi
Ref...nce

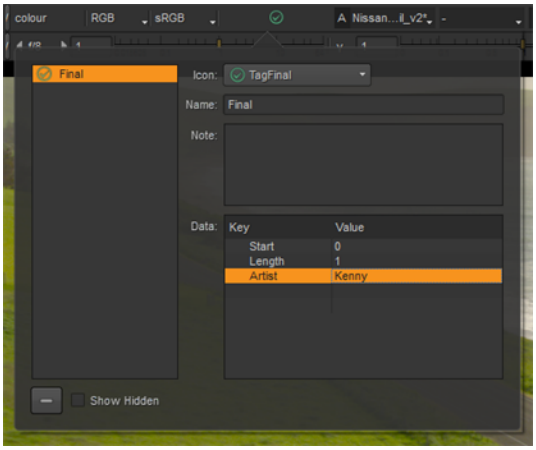
Adding Notes to Tags

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.

Hiero 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.

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**.

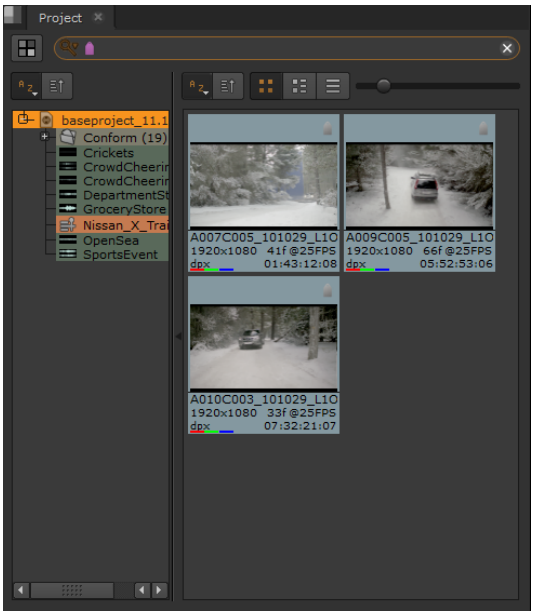
There are two types of tag search you can perform: **Filter** and **Flag**. Select the desired search type by clicking the magnifier icon in the **Project** tab.

- **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.

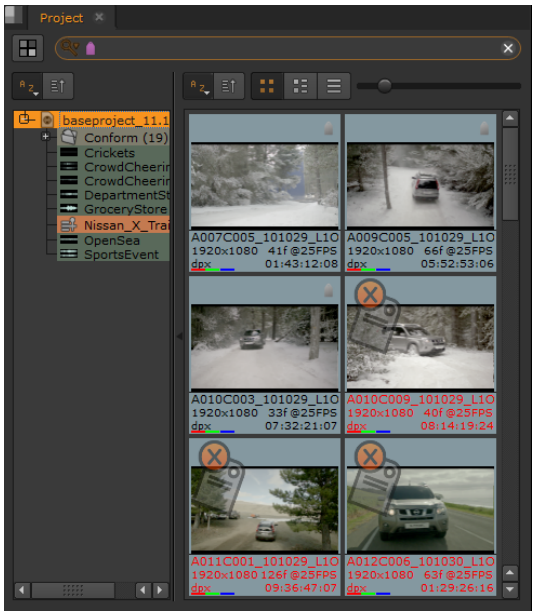
Drag the required tag from the **Tags** panel into the search box and select the bin or bins you want to Filter or Flag.

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.



Filtering ...



... Flagging.

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:

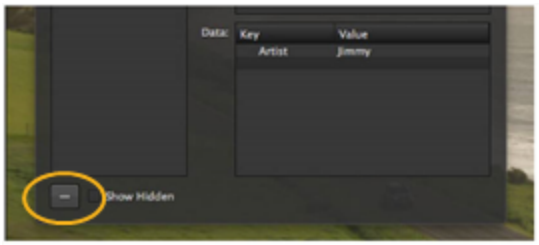
1. Click the **Tags** tab, or navigate to **Window** > **Tags**.
2. Select your project and navigate to **Project** > **New Tag**, or press **Ctrl**/**Cmd**+**Y**.
   The new tag is placed in the selected project.
3. Double-click the tag to open the **Edit Tag** dialog box.
4. Click the **Icon** dropdown 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.

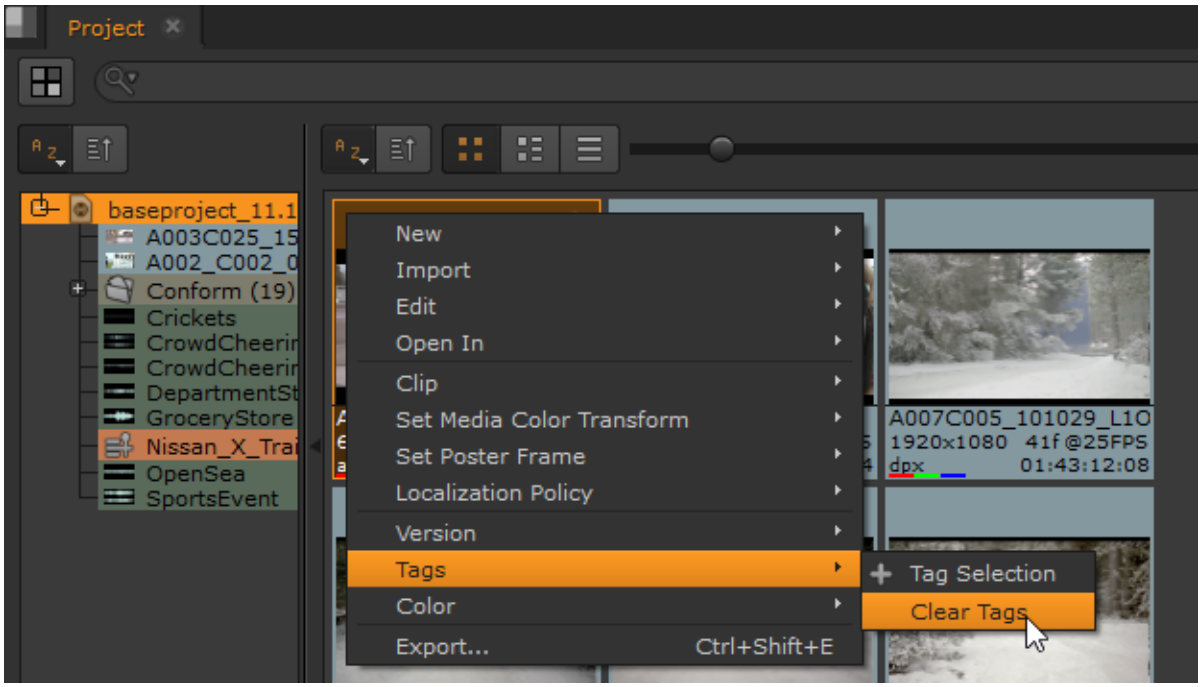
5. Enter a description for the tag in the **Name** field.
6. Click **OK** to save your changes.

Removing Tags

To remove a tag from a frame or shot, click the tag and then click  .



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  to remove your selection.

Viewing Metadata

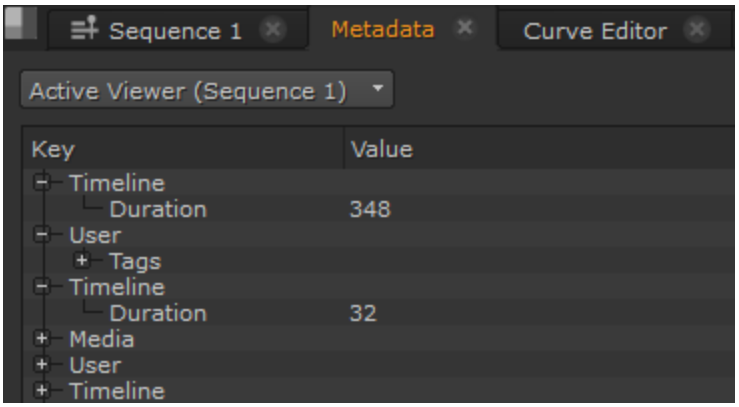
Metadata is information that describes media content, separate from the clip itself, in the form of a table on the **Metadata** tab. Types of metadata include **Duration**, **File Size**, and the **Path** to the location of the source media.

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**.



Conforming Using Hiero

Conforming describes the process of matching up the footage from a shoot with the required edit decisions to create a meaningful timeline. Hiero accepts sequences either from EDLs (edit decision lists), OTIO (OpenTimelineIO), AAFs (advanced authoring format), or Final Cut Pro XML files from a specified directory structure containing the source media files. Hiero attempts to conform the media, warning you if there are missing media.

Hiero conforms EDLs into single tracks, and OTIOs, 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.

See Project Settings, Importing Sequences, and Conforming Sequences for more information.

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 colorspaces 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 Hiero.
   - **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.

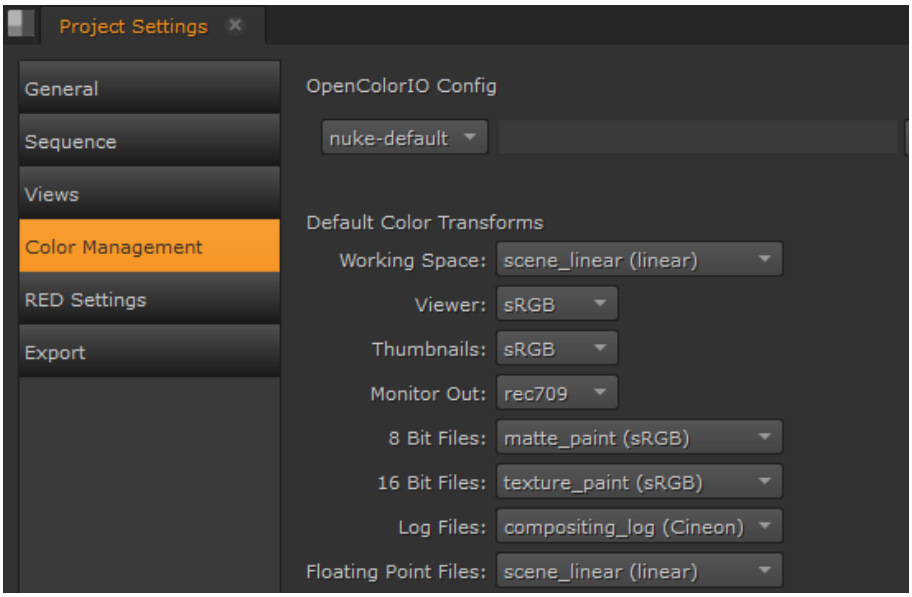
     If you want this setting to apply to all new projects, use the **Preferences** > **Project Defaults** > **General** panel settings.
   - **Custom Export Directory** - when **Export Directory** is set to custom, enter the required custom export directory.
   - **Shot Preset** - sets the default preset to use when exporting.

Color Management Settings

Hiero uses OpenColorIO for color management. All of the colorspaces in Hiero, whether those shipped with the application or custom colorspaces 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 colorspaces) that you can set in Hiero. 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 Hiero 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.



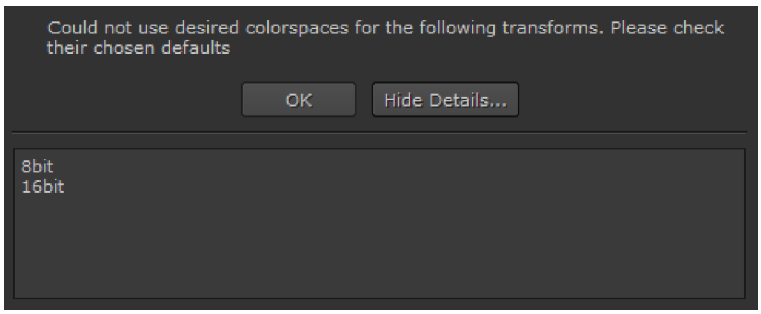
1. Set the OpenColorIO Config you want to use for this project.

   Hiero 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.



   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 Hiero under the hood.

> **Note:** In earlier releases of Hiero, 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.

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 Hiero 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 Hiero, with no integrated OCIO transforms. When **OCIO** is selected:

- Reads and Writes use OCIO transforms, with no Hiero 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 Hiero project with the same config. These defaults can be overridden using Python callbacks. See the following path for the default implementation that ships with Hiero:

  `<install_dir>/plugins/nuke/colorspaces.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.
- Hiero-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 Hiero 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 **.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 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\

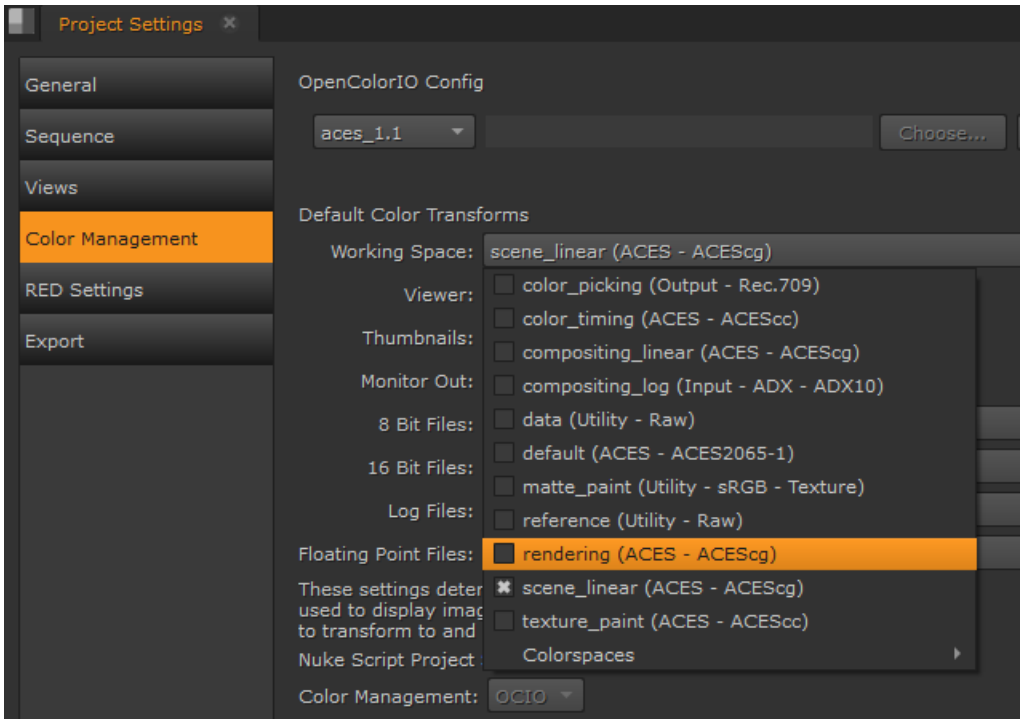
To find out if the HOME and USERPROFILE environment variables are set and where they are pointing at, enter **%HOME%** or **%USERPROFILE%** into the address bar in Windows Explorer. If the environment variable is set, the folder it's pointing at is opened.

2. Recreate the following structure within your **.nuke** 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 Hiero 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.

Adding OCIO Roles

OCIO roles allow you to set custom role names for different colorspaces to make it easier for artists to instinctively know which LUT to use for any shot. For instance, if an element is coming from your matte painting department and should always be brought into Nuke as sRGB, you can create a **matte painting** role, which is associated with the sRGB colorspace for your artist to select.

OCIO roles are the primary method for selecting colorspaces. All of the colorspaces in the OCIO config file are still accessible, but they have been grouped together into a **Colorspaces** menu under the roles.



OCIO roles are stored in **config** files, some of which ship with Nuke in the following directory:

```
<install_dir>/plugins/OCIOConfigs/configs/
```

For example, the aces_1.1 **config** file includes the following roles:

```
roles:
  color_picking: Output - Rec.709
  matte_paint: Utility - sRGB - Texture
  scene_linear: ACES - ACEScg
  texture_paint: ACES - ACEScc
```

The first part of the role defines the name of the **role** displayed in Nuke and second part describes colorspace **family** and **name**. The family and name define which colorspace is associated with the role. For example:

```
- !<ColorSpace>
  name: ACES - ACEScg
  family: ACES
  equalitygroup: ""
  bitdepth: 32f
  description: |
    The ACEScg color space

    ACES Transform ID : ACEScsc.ACEScg_to_ACES
  isdata: false
  allocation: lg2
  allocationvars: [-8, 5, 0.00390625]
  to_reference: !<MatrixTransform> {matrix: [0.695452, 0.140679, 0.163869, 0,
```

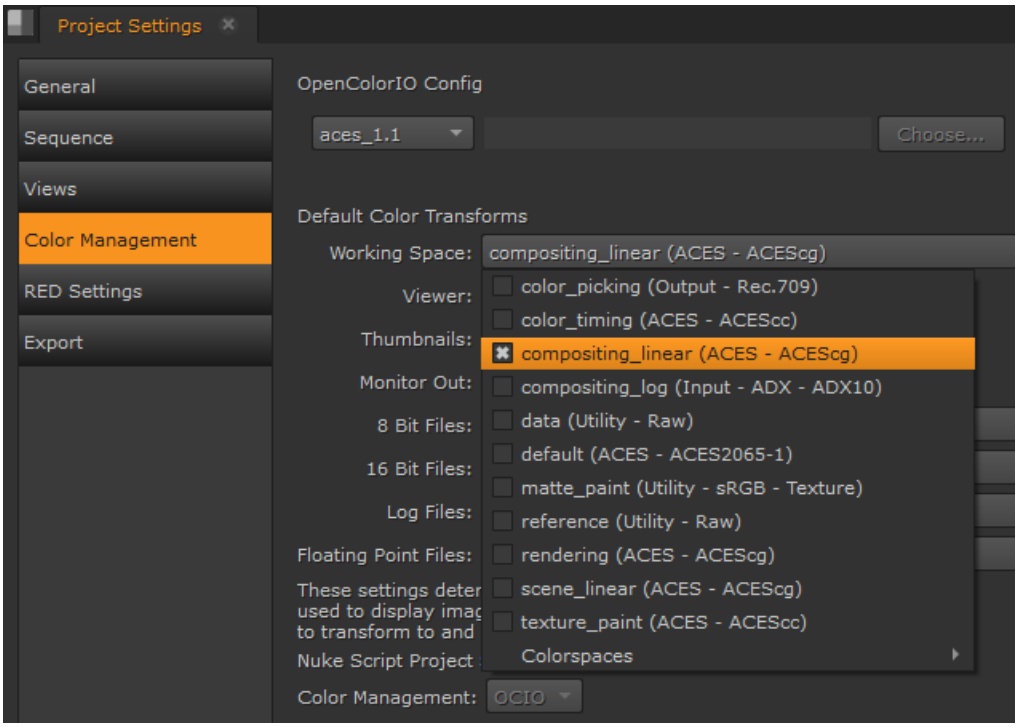
```
0.0447946, 0.859671, 0.0955343, 0,
  -0.00552588, 0.00402521, 1.0015, 0, 0, 0, 0, 1]}
```

Note: The LUT specified must exist in the **luts** directory in the same location as **config** file for the role to pick up the required colorspace.

You can edit these files to add roles or create your own custom config and then point Nuke to the file using the **Project Settings** > **Color Management** > **OpenColorIO Config** field.

To add a role to a config file:

1. Open the required config file or create a custom config.
2. Enter the **role** , **family**, and **name** of the role under the **roles:** line. For example:
   `compositing_linear: ACES - ACEScg`
3. Save the file and open Nuke.
4. Open the **Project Settings** and click the **Color Management** tab.
5. You can now pick your role from the default LUT settings. For example, you can set the working space to **compositing_linear** if you want to work in the ACEScg colorspace.



Define Color Space Mappings Using OCIO Aliases

Hiero supports the use of aliases in an OCIO config and checks these when attempting to set the input transform for a given image or sequence. The aliases feature in OCIO allows config authors to define a set of alternate names for a color space, which can then be used in place of the primary color space name. For example, it may be useful to define a more suitable name for file paths, or a color space name might have changed between versions of a config, so it may be necessary to add an alias for the older version of the name for backwards compatibility. Additionally if an application tries to set defaults based on a specific name then aliases can be used to ensure a suitable color space is found.

Below is part of a ColorSpace definition from an ACES 1.3 OCIO config.

```
- !<ColorSpace>

 name: ARRI LogC3 (EI800)
 aliases: [arri_logc3_ei800, Input - ARRI - V3 LogC (EI800) - Wide Gamut,
 logc3ei800_alexawide]
 family: Input/ARRI
 equalitygroup: ""
 bitdepth: 32f
 description: |
 Convert ARRI LogC3 (EI800) to ACES2065-1
```

Here you can see the **name:** attribute is defined as **ARRI LogC3 (EI800)**. This is the primary name for the color space.

Directly below is the **aliases:** attribute. This is where the alternate names are defined, in this case:

- `arri_logc3_ei800`
- `Input - ARRI - V3 LogC (EI800) - Wide Gamut`
- `Logc3ei800_alexawide`

The first and third aliases **(arri_logc3_ei800** and **logc3ei800_alexawide)** could be an example of a suitable name for including in a file path.

The second alias **(Input - ARRI - V3 LogC (EI800) - Wide Gamut)** is the name that was used for this color space in ACES 1.2. This is useful when switching from an ACES 1.2 config to ACES 1.3. The Hiero script will likely reference the older name but the alias means the input transform should be set correctly without any user intervention required.

Adding an alias to a color space in an OCIO config can be as simple as appending it to the end of the comma separated list. For example:

```
- !<ColorSpace>
```

```
name: ARRI LogC3 (EI800)
aliases: [arri_logc3_ei800, Input - ARRI - V3 LogC (EI800) - Wide Gamut,
logc3ei800_alexawide, AlexaV3LogC]
...
```

Here we've added the alias **AlexaV3LogC**. This is the color space name that Hiero looks for in the first instance when it detects an ARRI V3 LogC file.

Note: In versions of Hiero prior to 14.1 the example above would typically result in an error, because there isn't a matching color space name in the ACES config.

From Hiero 14.1, the following aliases are included in our ACES 1.3 configs:

```
- Aliases (Nuke > ACES):

 AlexaV3LogC > ARRI LogC3 (EI800)
 ARRILogC4 > ARRI LogC4
 Blackmagic Film Generation 5 > BMDFilm WideGamut Gen5
 Log3G10 > Log3G10 REDWideGamutRGB
 SLog3 > S-Log3 S-Gamut3
 Gamma1.8 > Gamma 1.8 Rec.709 - Texture
 Gamma2.2 > Gamma 2.2 Rec.709 - Texture
 Gamma2.4 > Gamma 2.4 Rec.709 - Texture
```

The color space names that Hiero currently looks for can be seen in the **nuke-default** OCIO config.

For more information on aliases and authoring configs please see the official OCIO documentation.

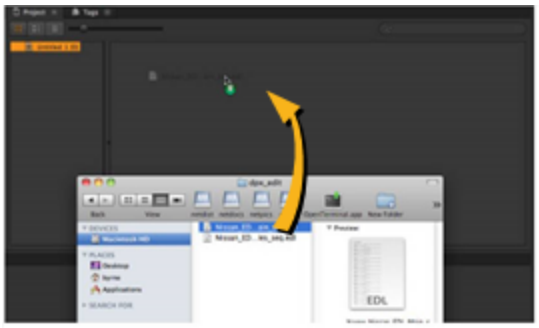
Importing Sequences

Hiero allows you to import your EDL, OTIO, XML, or AAF sequences in one of two ways, depending on your preferences. Either:

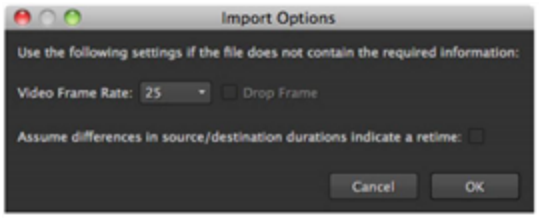
- Navigate to File > Import EDL/XML/AAF or Import OTIO (Beta) and then use the browser to locate the EDL, OTIO, XML, or AAF, and then select the file and click Open to import the sequence,

  OR

- Drag-and-drop the EDL, OTIO, 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. SeeTimeline 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.

   OTIO, XML and AAF files imported into Hiero 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 .otio, .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 Hiero. See Notes on AAF Sequences for more information.

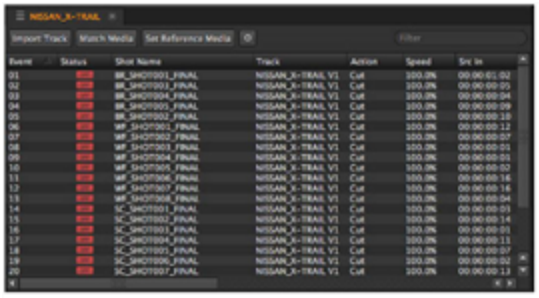
Note: Non-linear animation curves from .otio or **.xml** may not appear as expected when imported. As a result, you may need to adjust the handles on curves to match footage between keyframes in the Curve Editor or Dope Sheet.

See the *Nuke User Guide, Getting Started, Using the Compositing Environment, Animating Parameter* section or the Nuke Online Help for more information.

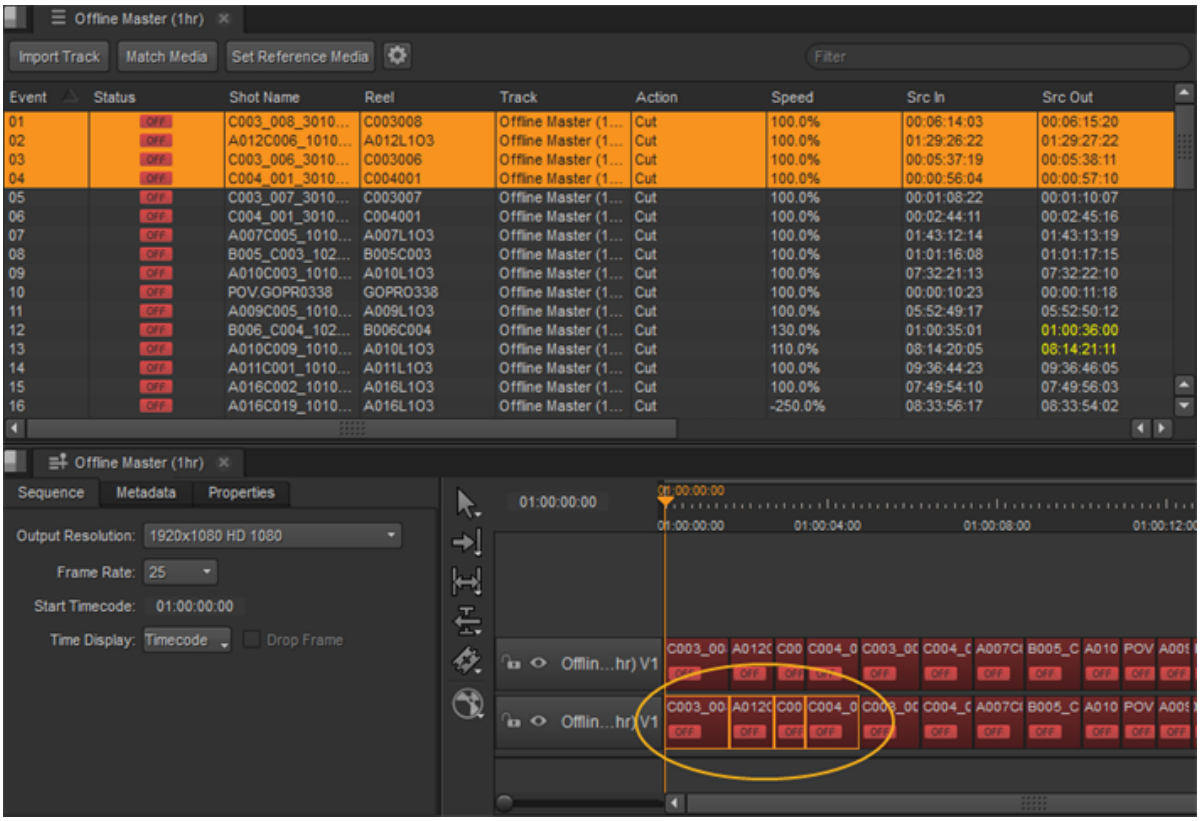
Additionally, Premiere Pro **.xml** exports only support constant, linear retimes. As a result, retimed shots on the Nuke Studio timeline may not match those on the Premier Pro timeline, because certain non-linear retime data is not written into the exported **.xml** file.

After importing the EDL, OTIO, 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.



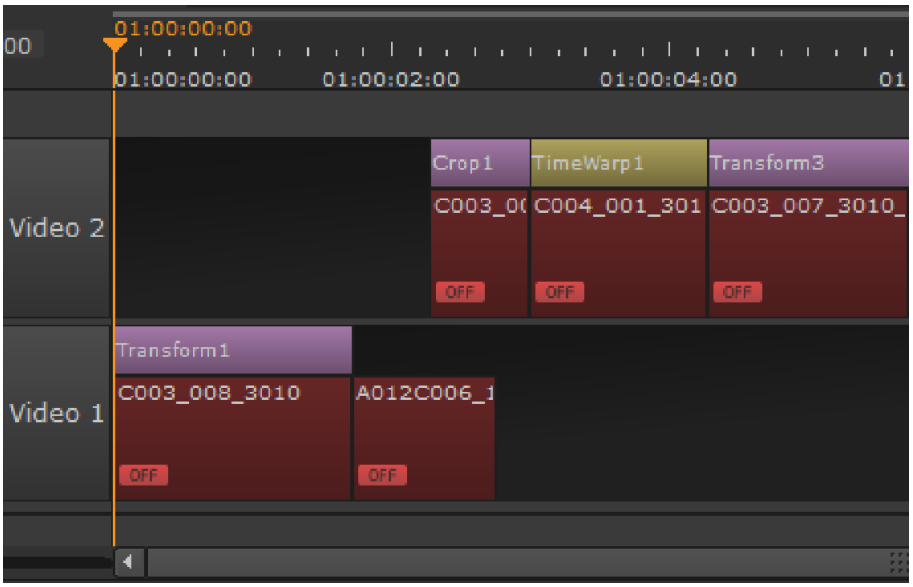
Notice that clicking entries in the spreadsheet highlights the corresponding shots on the timeline?



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 Hiero has automatically matched media for you.

Any transform, crop, or retime edit decisions from third-party software .otio, .xml or .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**. Hiero 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.

Hiero'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 Hiero don't match those in Avid, but the resulting curve should match the original very closely.

Note: Hiero currently only supports Fixed Keyframes from Avid Media Composer.

Tip: If you need to adjust the handles on curves, see https://learn.foundry.com/nuke/content/getting_started/using_interface/animating_parameters.html for more information.

Hiero's TimeWarp effect supports the following Spline types when importing **.aaf** files:

- Shelf
- Linear

- Spline
- Bezier

Conforming Sequences

Once your EDL, OTIO, 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 Hiero.

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
```

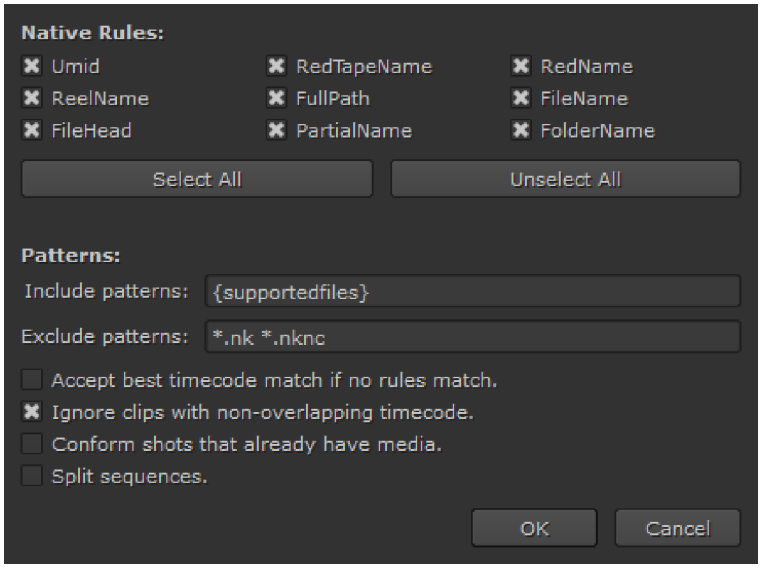
Conforming Using a Browser

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.



Hiero 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.

Rule	Description
Umid	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.
RedTapeName	Match a RED-style camera reel name from the event to the candidate media

Rule	Description
	name.
RedName	Look for a RED-style camera file name in the event that matches the candidate media name.
ReelName	Look for the event's reel name in the candidate's media name.
FullPath	Match the event's entire filepath to the candidate media's entire filepath. 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 **Hiero** is testing the conform rules against.
FileName	Match only the event's file name (no path) to the candidate media's file name.
FileHead	Match the event's file name head (no path, file extension, or padding) to the candidate media's file name.
PartialName	Look for the event's name in the candidate media's name and vice versa.
FolderName	Look for the event's name in the filepath of the candidate media.

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.

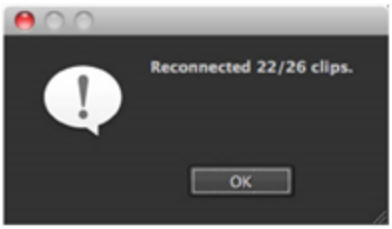
3. Enable **Accept best timecode match...** to use the nearest source timecode match to conform the event, if no rules are matched.

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.

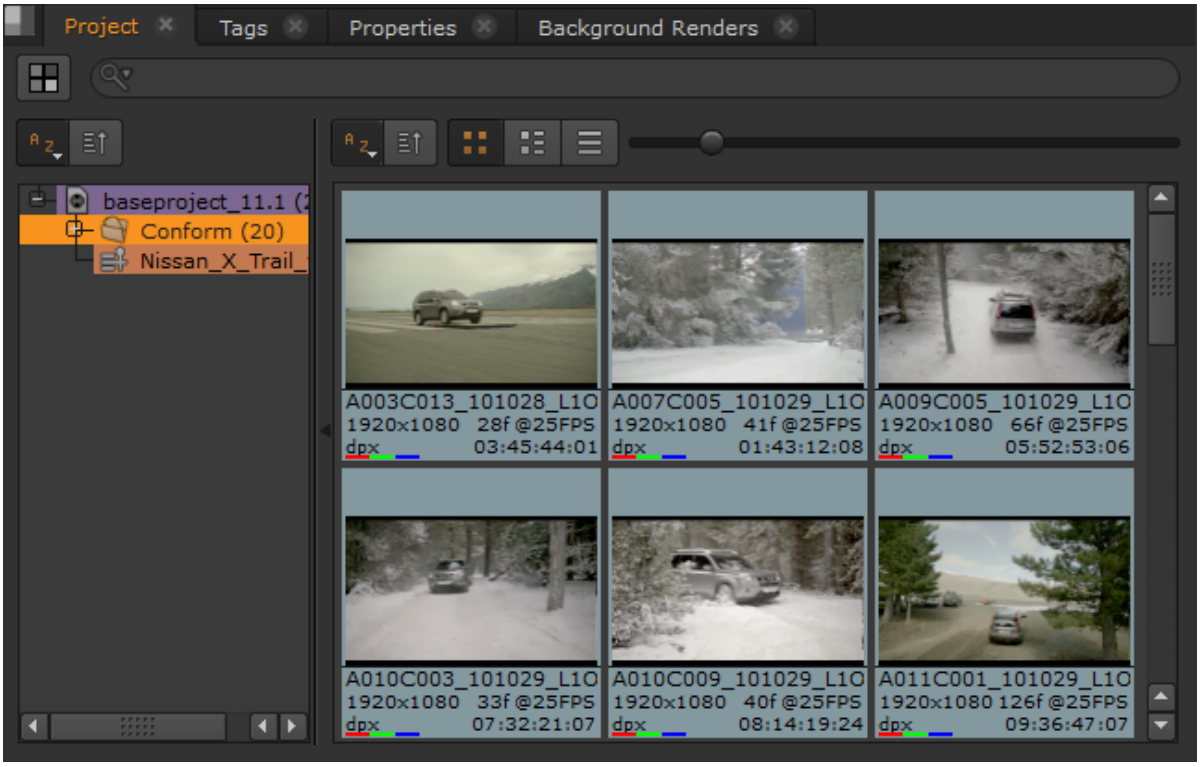
   Disabling **Ignore clips with non-overlapping timecodes** causes Hiero to fall back to the other selected conform rules, even if the timecodes don't overlap.
5. Check **Conform shots that already have media** if you want to update all timeline shots. By default, the application doesn’t try to conform 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.

   Hiero attempts to conform the edits with the selected media.

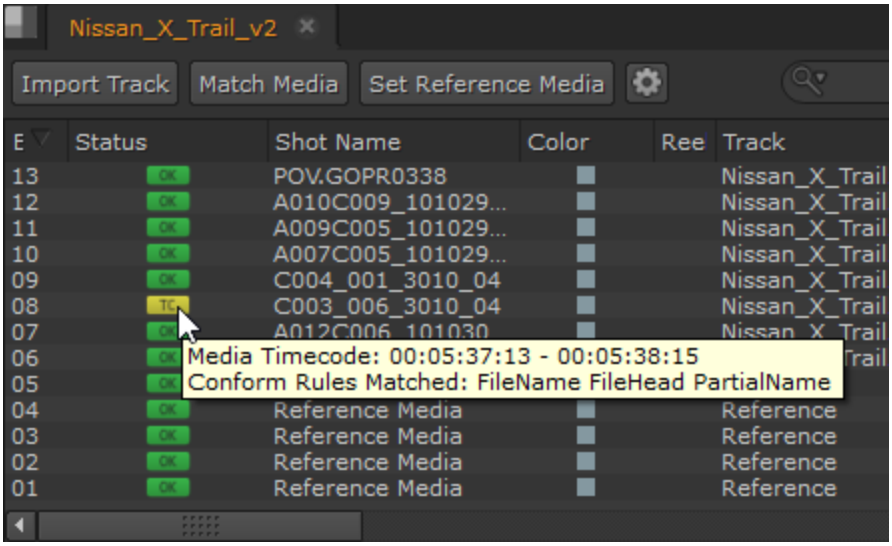
   A dialog box informs you of the success rate once the conform is complete.



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.

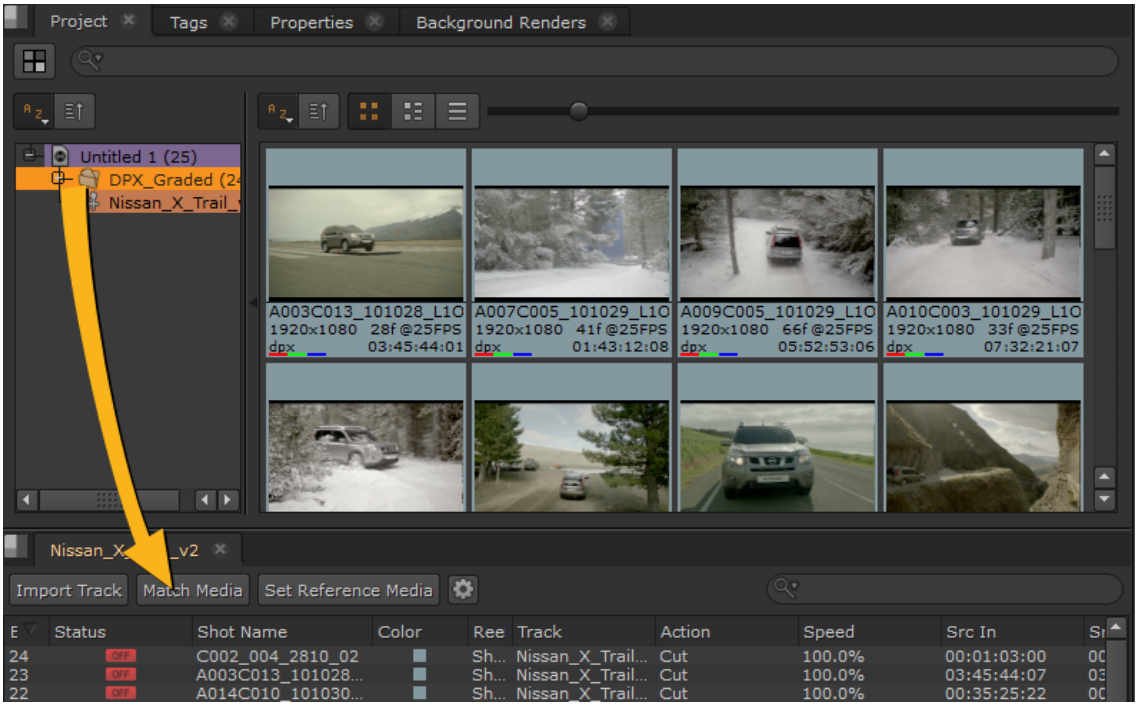


Conforming with Pre-ingested Media

To conform with pre-ingested media:

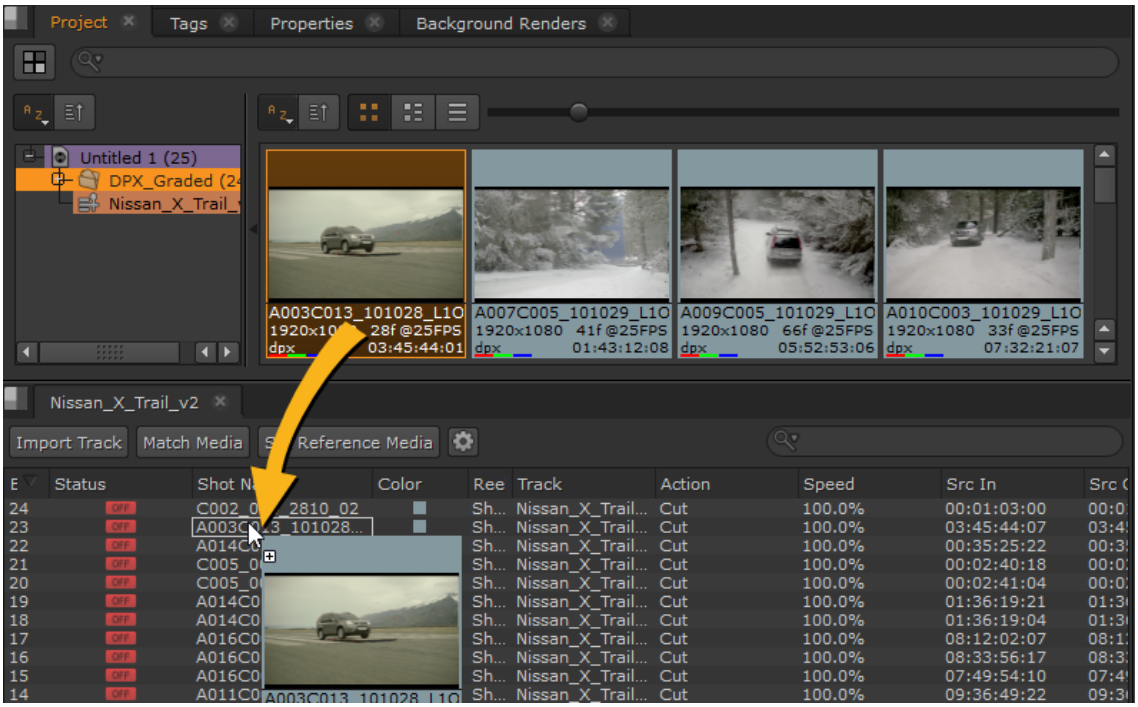
1. If your source media has been ingested, you can drag-and-drop media from the bin view onto the **Match Media** button.

   See Ingesting Media for information on getting media into Hiero.



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 Hiero 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

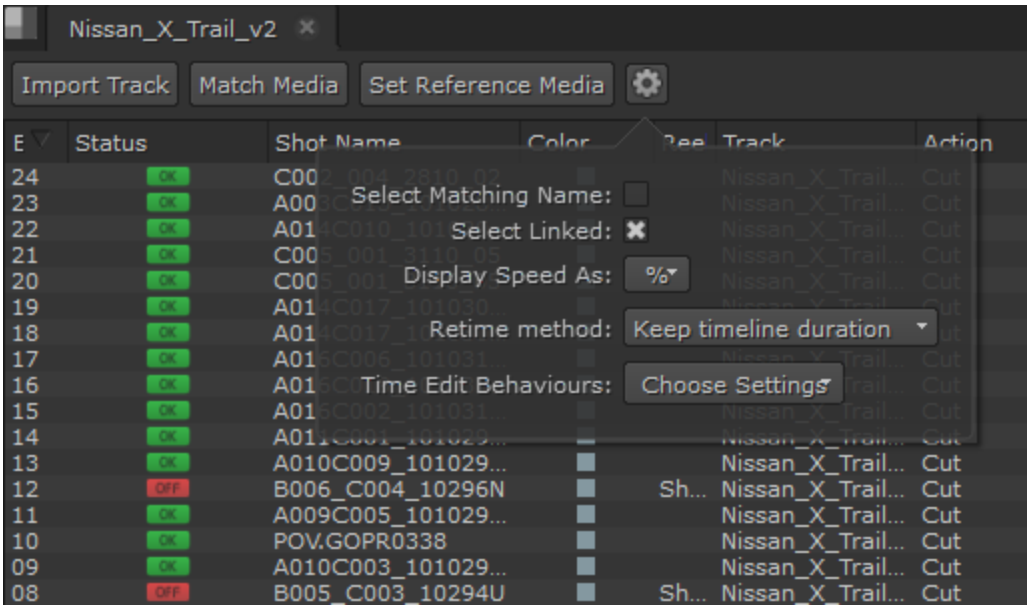
The spreadsheet can be organized in much the same way as accounting spreadsheets:

- 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: Hiero'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. Hiero 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.

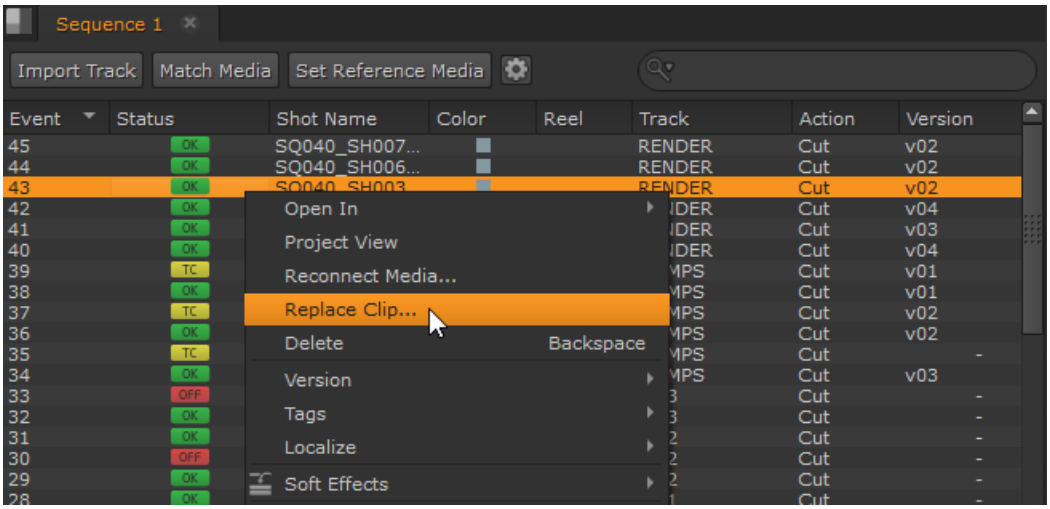


- **Select Matching Name** - when enabled, selecting an item in the spreadsheet highlights all items with the same name.
- **Select Linked** - when enabled, selecting an item in the spreadsheet highlights other items linked to it, such as audio tracks ingested with video tracks.
- **Display Speed As** - 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.
- Version - access the powerful versioning system to control which version of the selected shot is visible in the sequence. See Using Versions for more information.
- Tags - add tags to your selection directly from the spreadsheet view. See Using Quick Tags for more information.
- Localize - control the localization of clips, tracks, and sequences from the spreadsheet. See Localizing Media for more information.
- Soft Effects - add soft effects to shots directly from the spreadsheet. See Adding Effects on the Timeline 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.

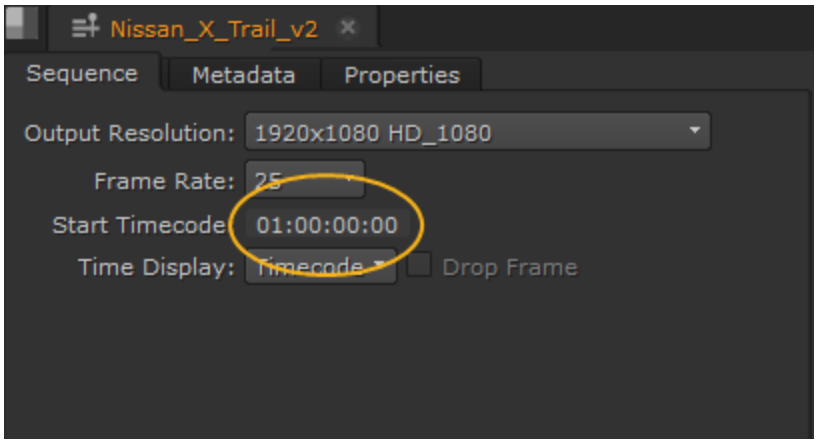
Example	Result
01:05:43:21	Sets the timecode at 1 hour, 05 minutes, 43 seconds, and 21 frames
01054321	

   - **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.

Current Position	Example	Result
01:05:43:21	+1h	02:05:43:21
	-110	01:05:42:11
	+10000	01:06:43:21
	-6m	00:59:43:21

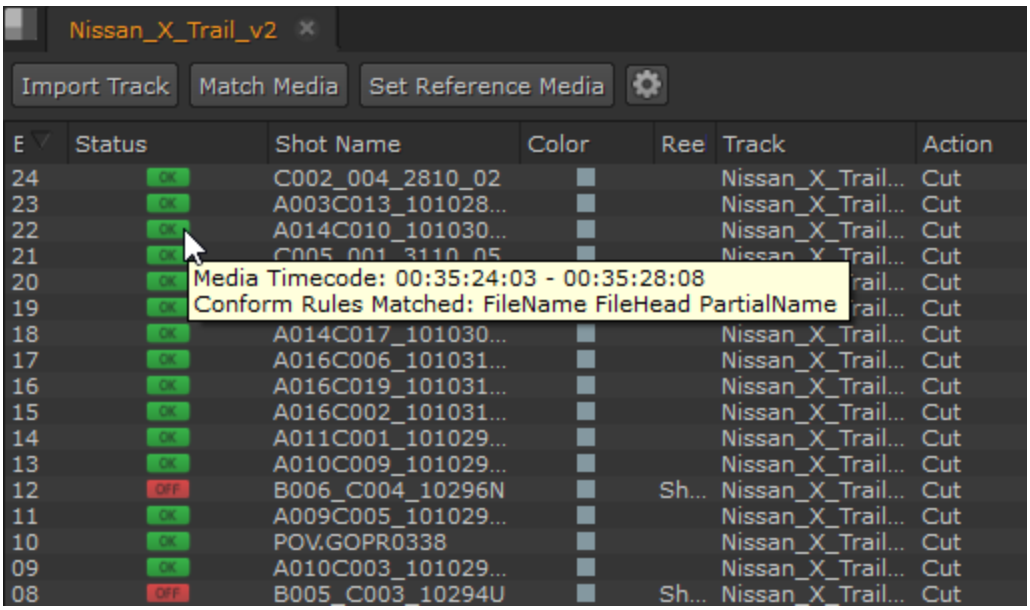
The media changes state to OK.

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:

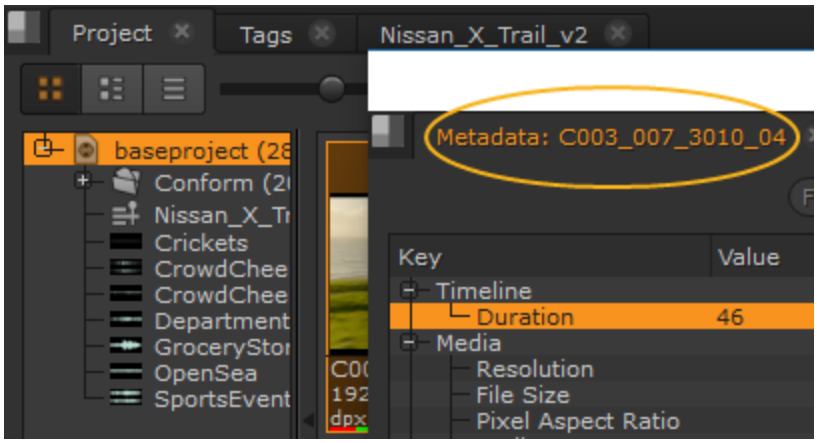
• Hover the mouse over the target entry in the spreadsheet to view a timecode tooltip.



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.


Project
Tags
Nissan_X_Trail_v2
Metadata: C003_007_3010_04
baseproject (28
Conform (2
Nissan_X_Tr
Crickets
CrowdChee
CrowdChee
Department
GroceryStor
OpenSea
SportsEvent
C00
192
dpx
Key
Value
Timeline
Duration
46
Media
Resolution
File Size
Pixel Aspect Ratio

Renaming Shots on the Timeline

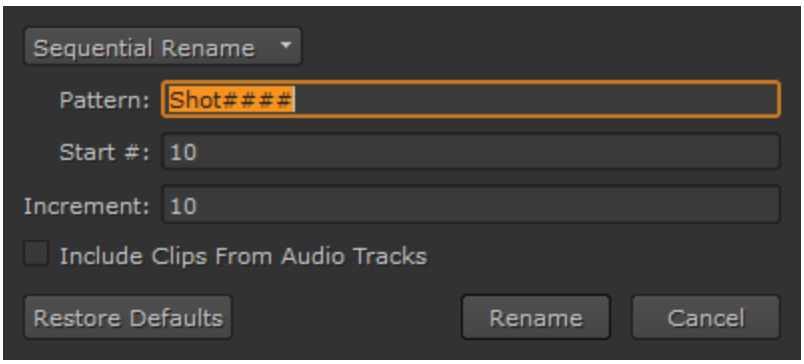
Once you've conformed your edit, you may want to rename shots on the timeline sequentially for clarity.

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:

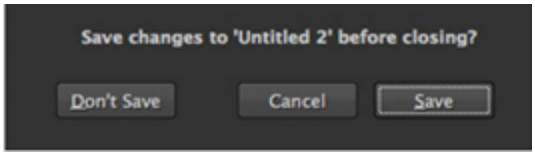
Token	Resolves to
{clip}	The name of the source clip referenced by the target shot.
{event}	The EDL event number associated with the target shot.
{filename}	The file name of the shot's source.
{fps}	The frame rate of the sequence containing the rename target.
{sequence}	The name of the sequence containing the rename target.
{shot}	The name of the shot.
{track}	The name of the track containing the rename target.

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 Clips** during round-tripping. See Building VFX Tracks and Comp Clips for more information.

If you quit the application without saving, you'll be prompted to save or discard your changes:



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.

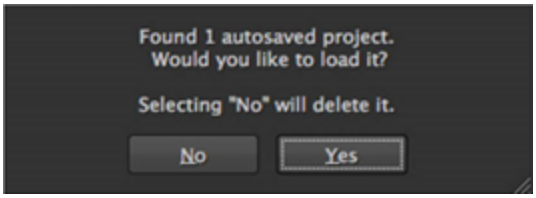
   The **Open Project** dialog box displays.
2. Browse to the file location and click **Open**.

Your project opens and populates the necessary panel automatically.

Autosaved Projects

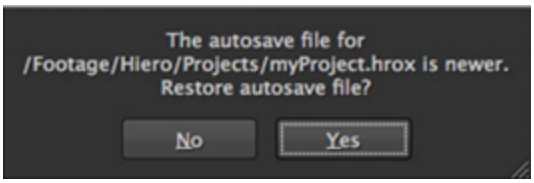
The autosave function creates a temporary project save file at 5 minute intervals, but you can adjust **force project autosave after** in the **Preferences** > **General** dialog. See Appendix A: Preferences for more information.

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.

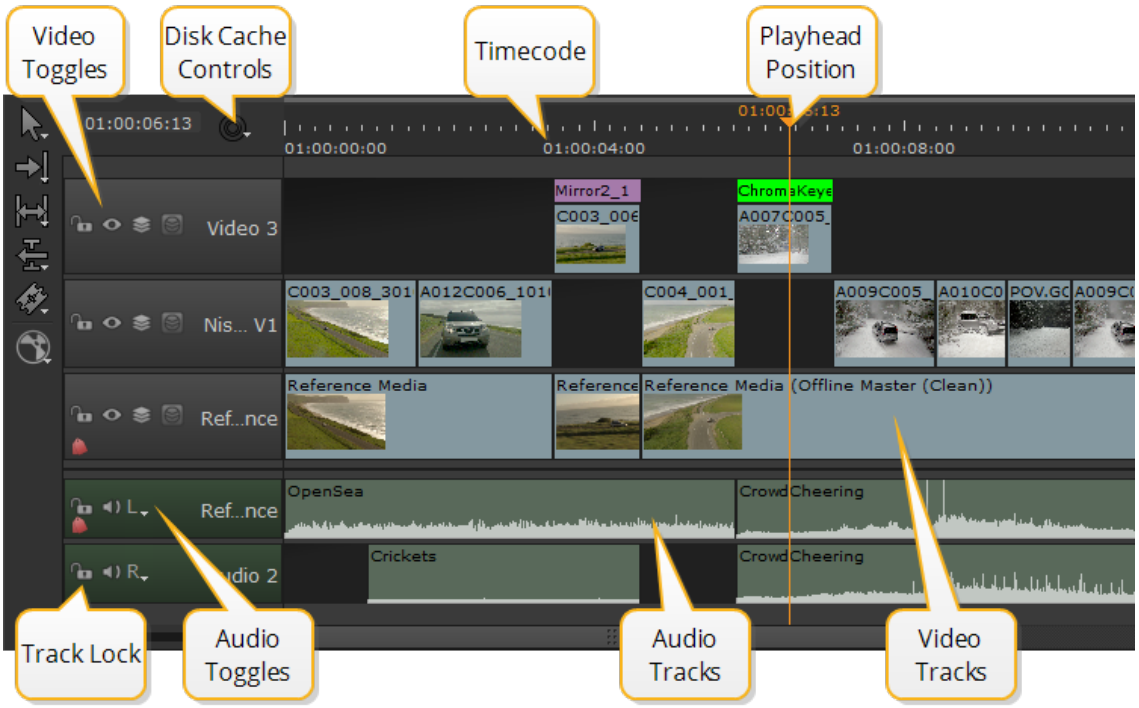
Note: Clicking **No** does not delete the autosaved project in this case.

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.

Hiero also features real-time soft effects on the timeline and the ability to add shots containing **.nk** scripts. See Adding Effects on the Timeline and Building VFX Tracks and Comp Clips 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.



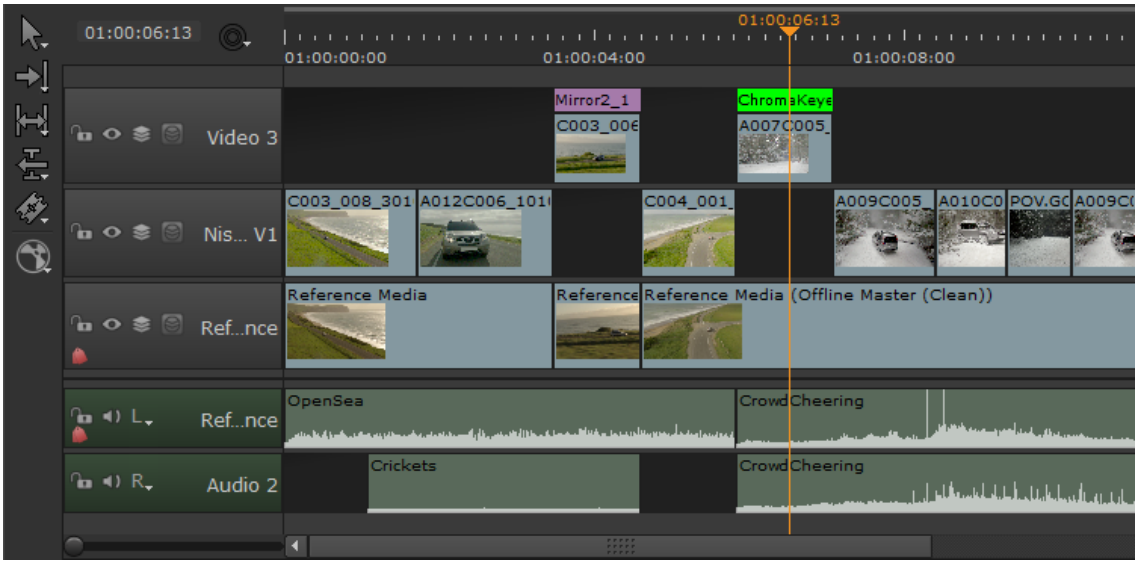
- **Video Toggles** - quickly turn off and on video tracks during playback. Hold **Alt** and click to solo the selected track. You can also enable and disable track blending and masking. See Blending Tracks on the Timeline for more information.

- **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

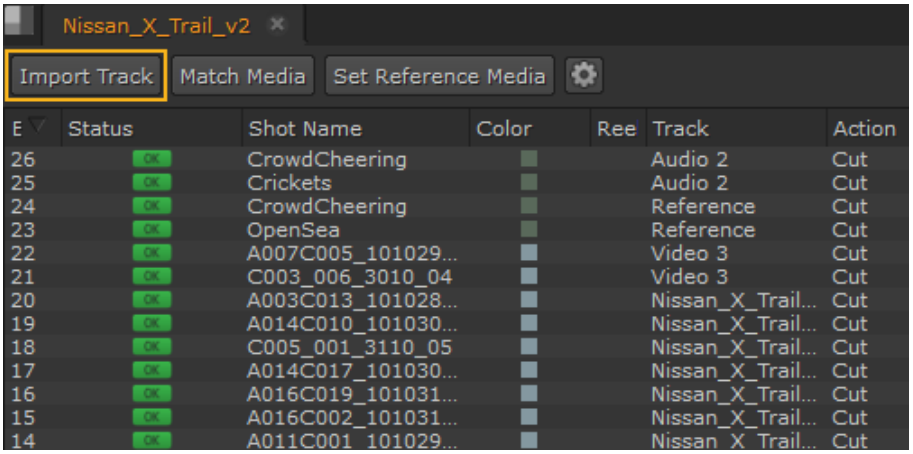
You can add empty tracks to existing timelines or import other EDL, OTIO, AAF, or XML edits - effectively another sequence within the timeline.

To import EDL, OTIO, 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 New Track(s) from OTIO (Beta),

   OR

   Click **Import Track** in the spreadsheet tab.



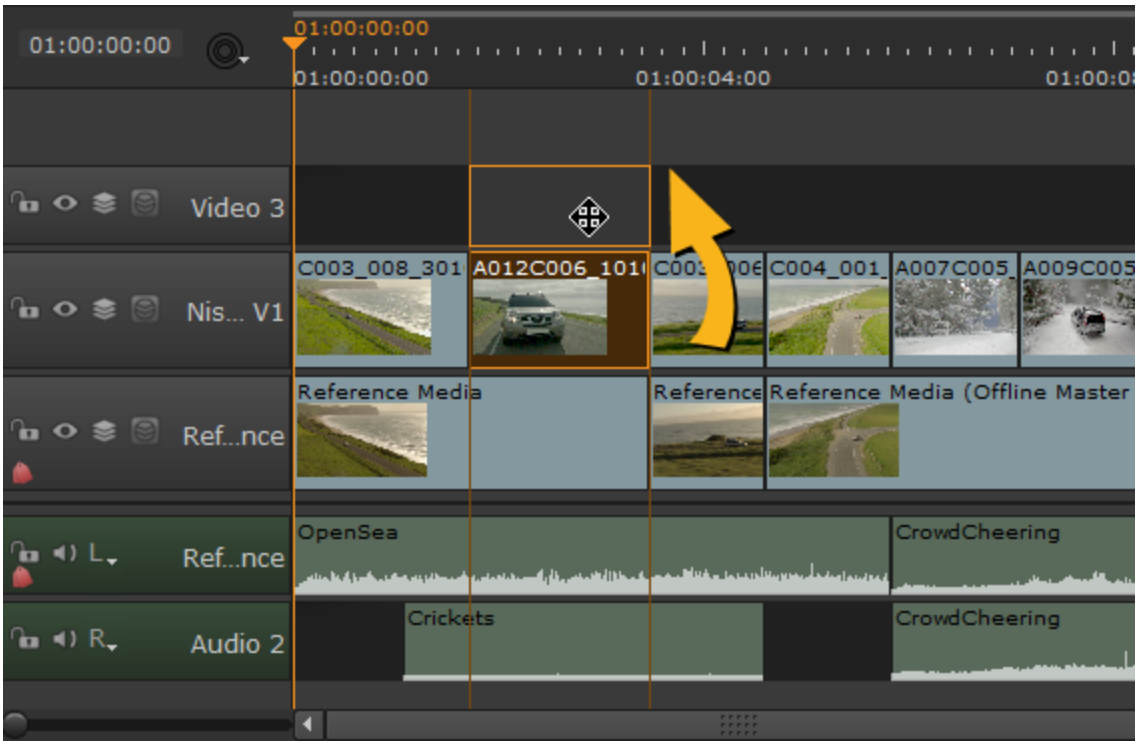
2. Use the browser to locate the EDL, OTIO, AAF, or XML files, select the file(s) and click **Open** to import the sequence.

> **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.

3. Conform the new track as described in Conforming Sequences.

To add new tracks:

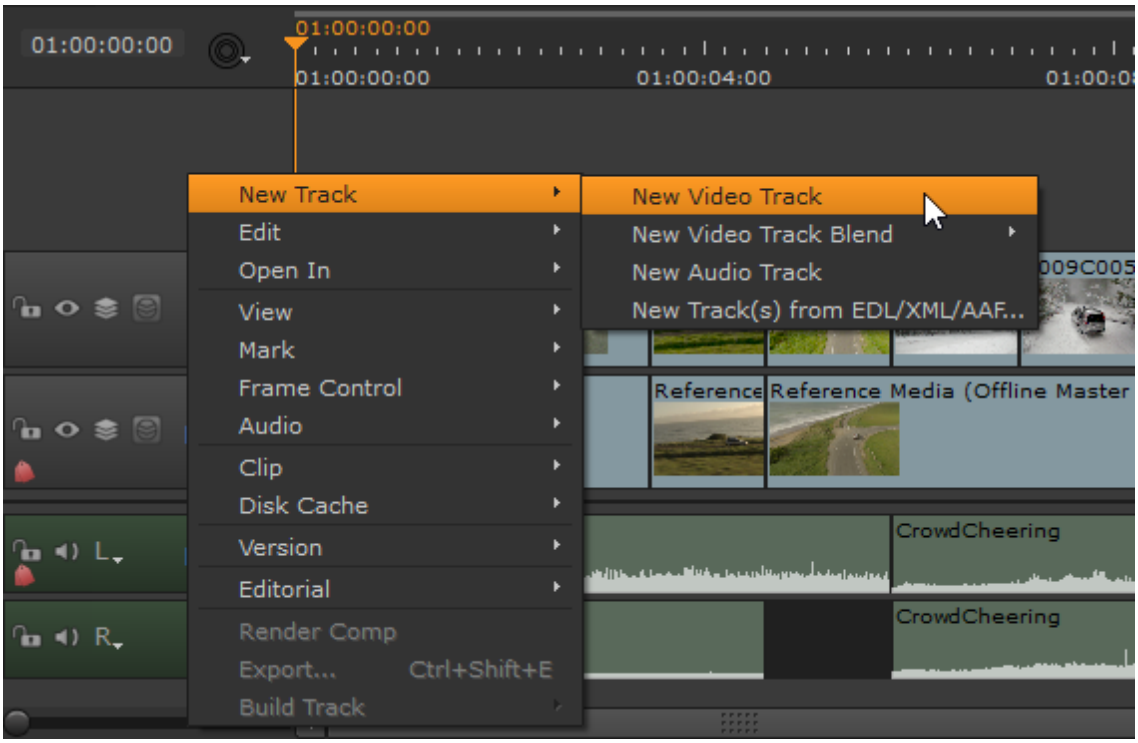
• Hold Shift and drag a shot to insert a new track above or between existing tracks,



Tip: This is the default behavior. If you disable Preferences > Panels > Timeline > Hotkey required to insert new track, the new track is created automatically.

OR

• Right-click in the timeline and select **New Track** > **New Video Track** or **New Audio Track**.



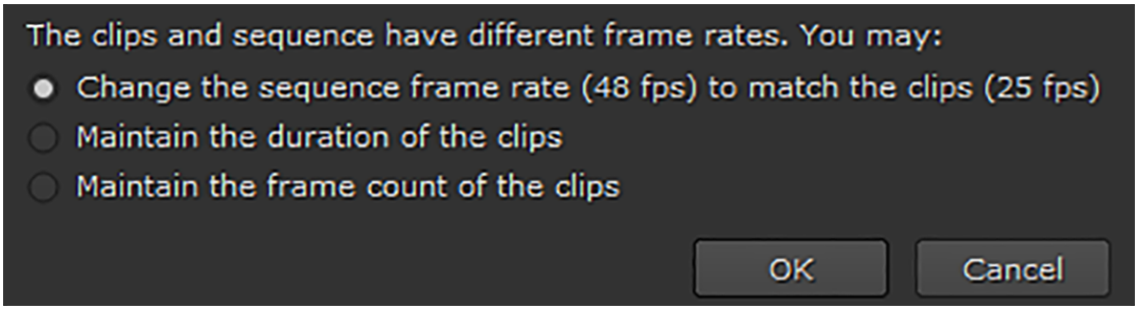
Note: You can also collapse and expand existing tracks using the right-click **Editorial** menu, and resize the track header to accommodate longer track names.

Adding Clips to the Timeline

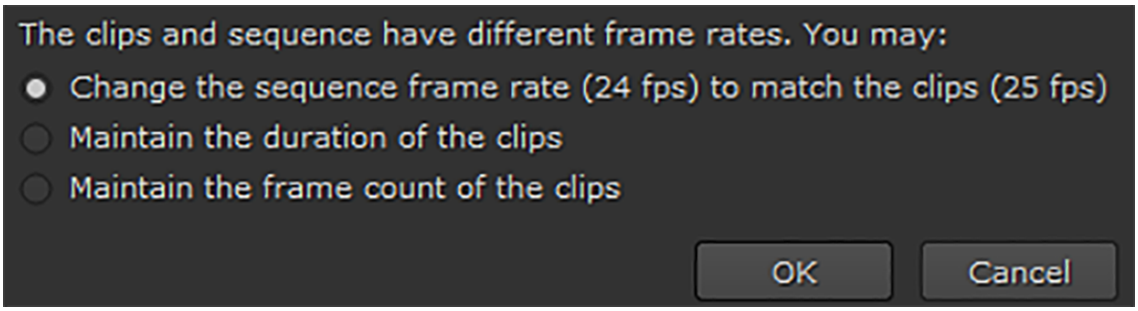
The timeline allows you to add clips by simple drag-and-drop from either the Viewer or bins. Using the Viewer restricts you to a single clip, the current clip, but you can drag as many clips as you like from bins.

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.



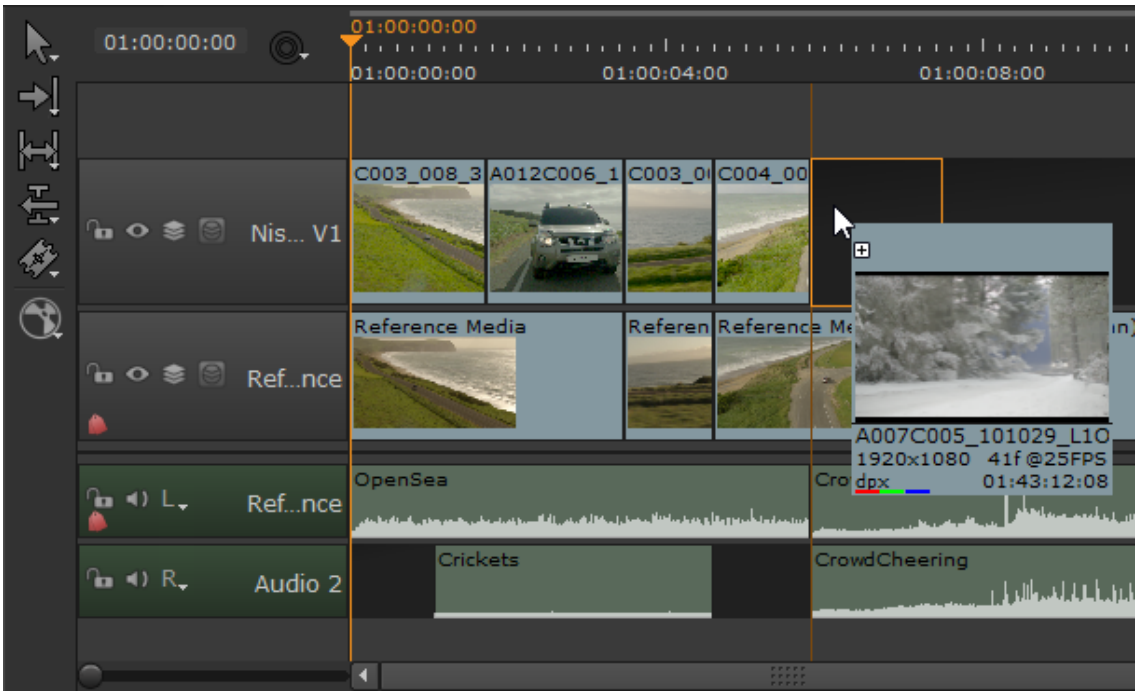
Take care not to overwrite existing shots - the most recent clip overlays any existing shot. To avoid this, do one of the following:

- 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.

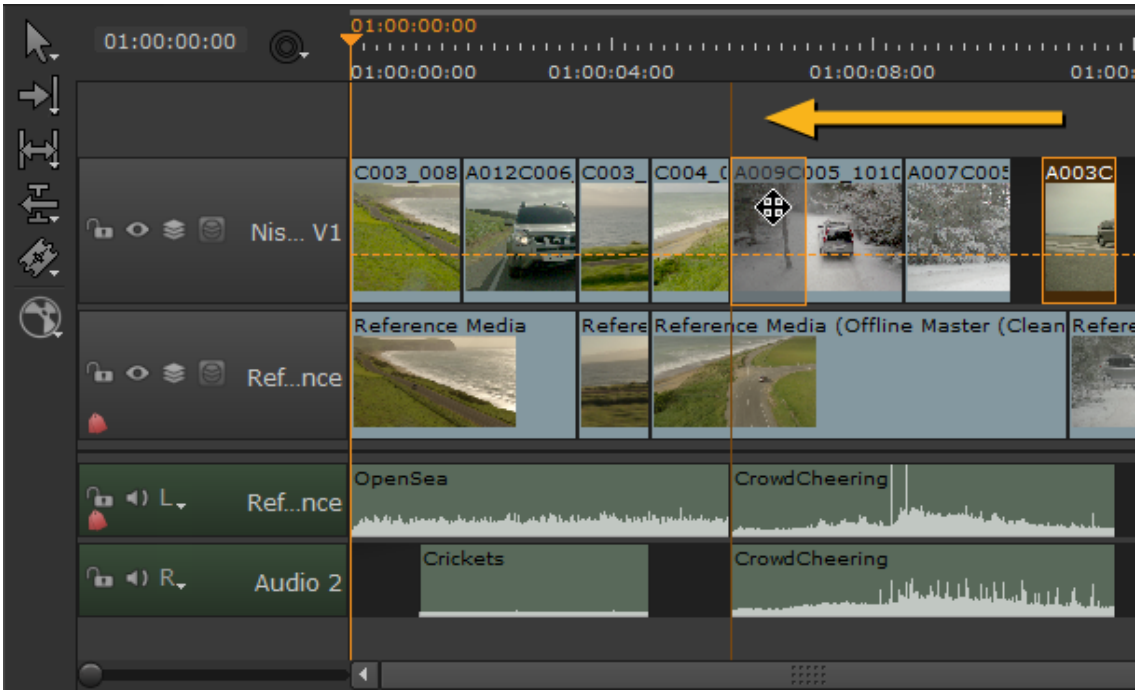
Note: You can only **Insert** or **Overwrite** using clips from the current project.

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.



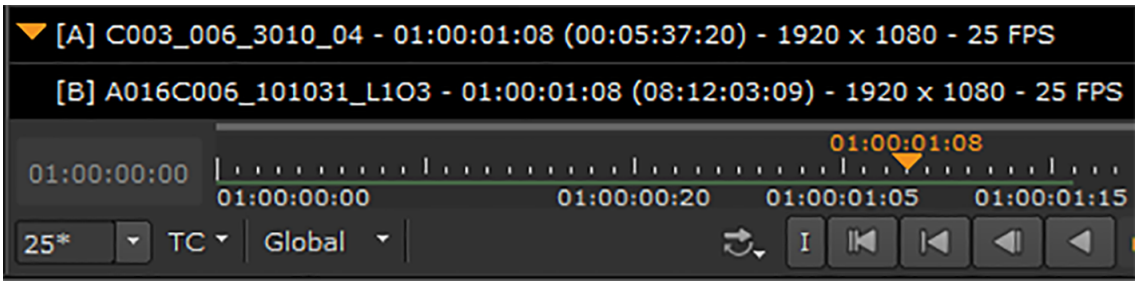
Tip: Enabling **Preferences** > **Panels** > **Timeline** > **show frame end marker** draws an extra line on the timeline to the right of the playhead, indicating the end of the current frame.

Audio and the Timeline

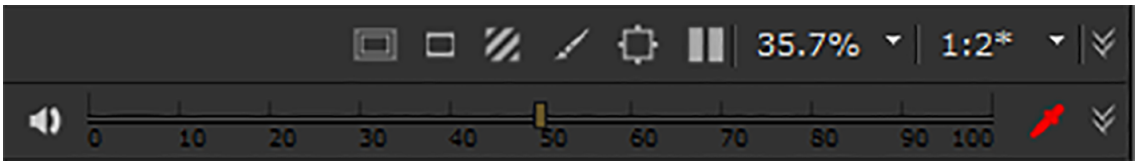
Audio tracks on the timeline are handled in much the same way as video tracks. By default, linked audio and video tracks are edited at the same time, but you can lock either track and move them independently or hold **Alt** to select a single track, if required.

Hiero supports integrated audio from container files, such as **.mov**, and audio specific files, such as **.wav**, on the timeline. Hiero assigns the channels in the audio file to separate tracks and matches the channels in the file to the available channels on the timeline.

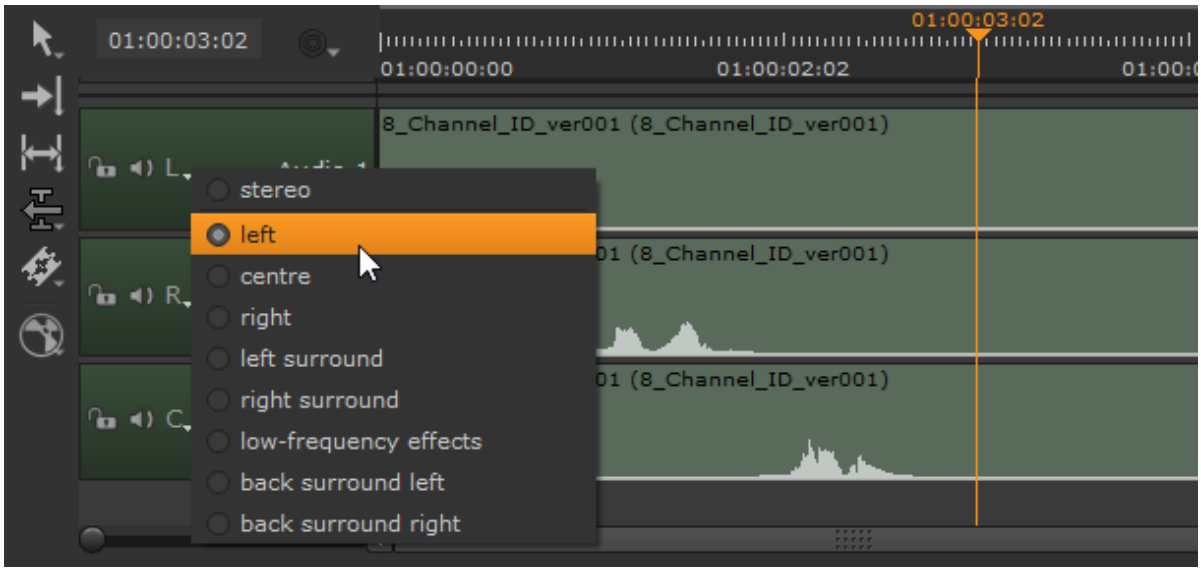
When both Viewer inputs contain clips, the audio output is set by the orange marker in the color picker information bar, displayed by clicking  . In the following example, input A is providing the audio output:



The volume slider in the upper-right corner of the Viewer controls the output level for that Viewer only.

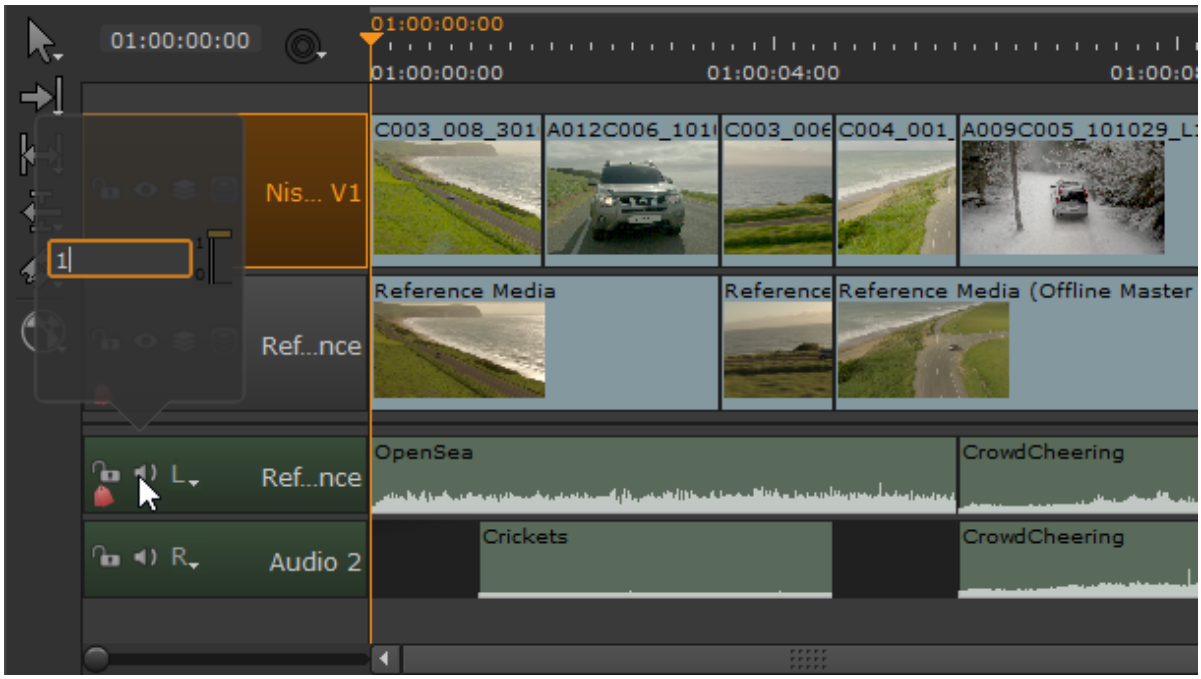


Audio output for shots can be toggled between eight channels, depending on the requirements of the clip or shot: left, center, right, left surround, right surround, low-frequency effects, back surround left, and back surround right. Click on the dropdown to cycle between outputs:



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.

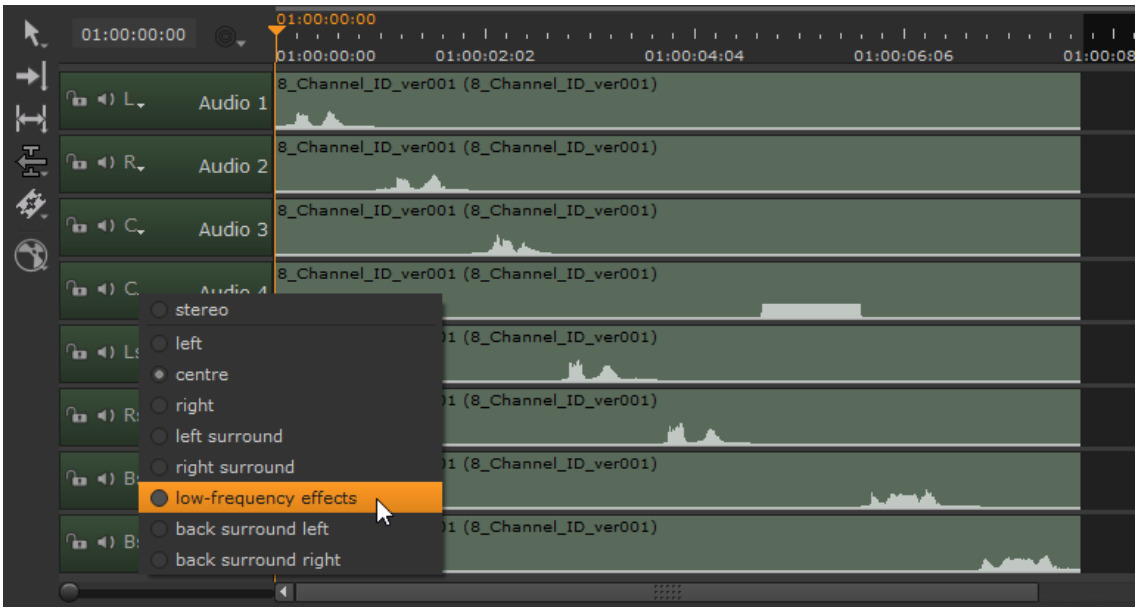
WAV Shots

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.

Hiero assigns the channels in the audio file to separate tracks and matches the channels in the file to the available channels on the timeline. If a channel is not matched correctly, you can select the channel manually using the dropdown in the track header. For example, **LFE** (low frequency effects) may be assigned to **C** (center) incorrectly.



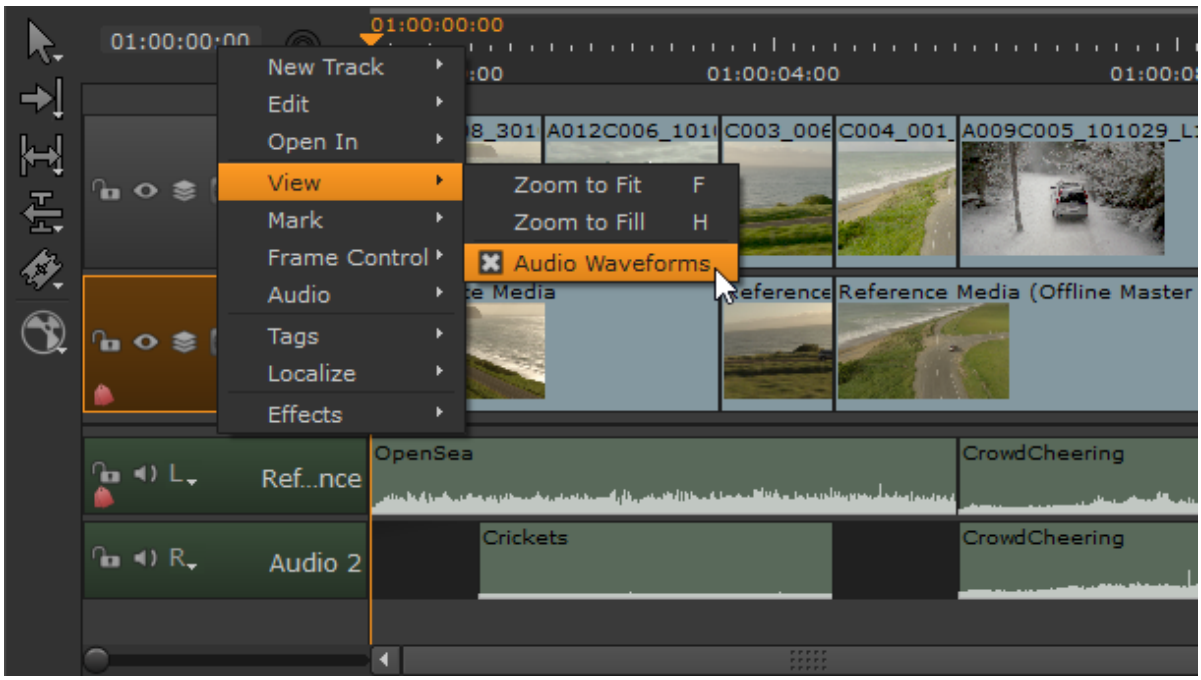
Use the Timeline Editing Tools to move the clip into place and set its output.

Displaying Audio Waveforms

Visualizing an audio waveform helps synchronization with video events, and Hiero 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 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 Between Shots and Audio Clips for more information.

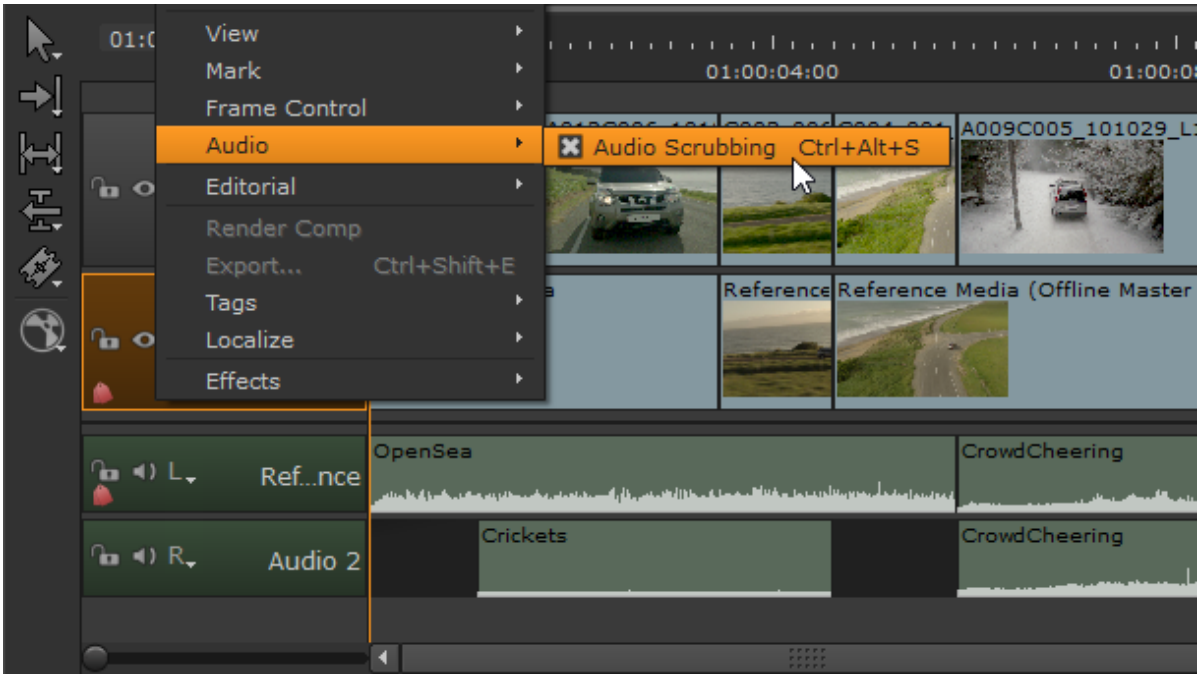
You can toggle the waveform display on and off by right-clicking in the timeline and selecting **View** > **Audio Waveforms**. You can also control how the waveform appears, when enabled. Open the **Preferences** and navigate to **Panels** > **Timeline** to toggle between full and **half waveforms**.



Displaying waveforms in audio-heavy projects can cause significant slow down on the timeline, so Hiero 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

Hiero's timeline supports audio scrubbing, allowing you synchronize audio and video more easily while scrubbing the playhead. Audio scrubbing is disabled by default, but you can enable 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** > **RAM Cache** > **Clear Audio Cache**.

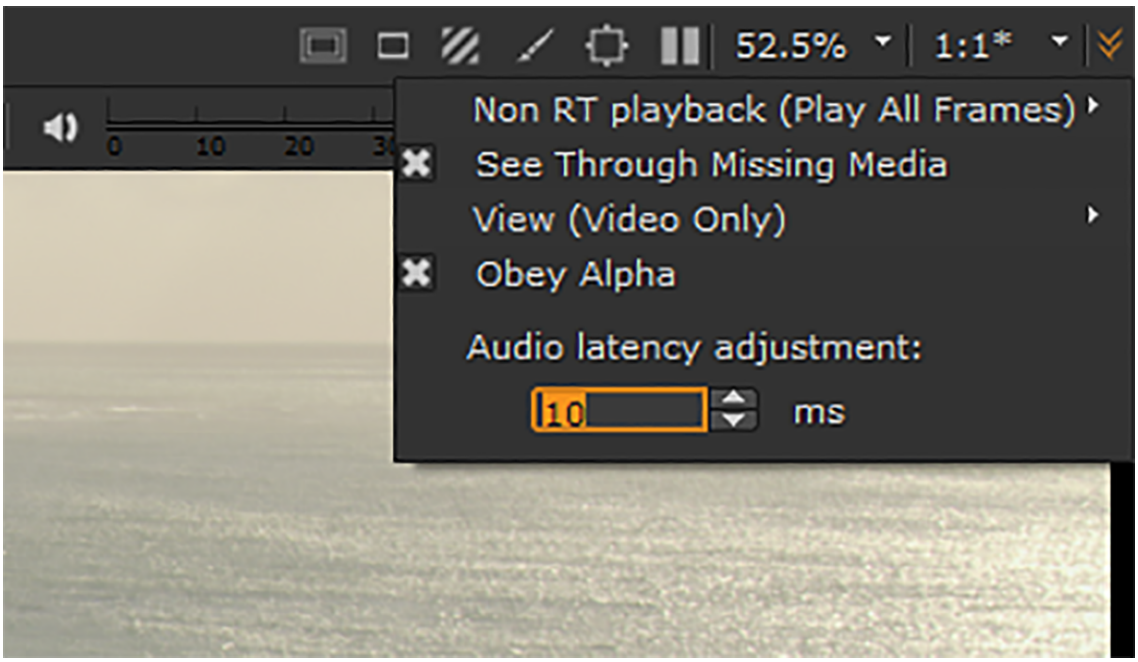
Note: Audio scrubbing is not currently available through monitor output cards. Audio scrubbing is only supported through internal audio output devices.

Synchronizing Audio and Video

Hiero 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 Hiero 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:

Note: PulseAudio restarts automatically when you restart you machine, but you can prevent this by navigating to **System** > **Preferences** > **Startup Applications** and disabling the PulseAudio Sound System.

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: 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

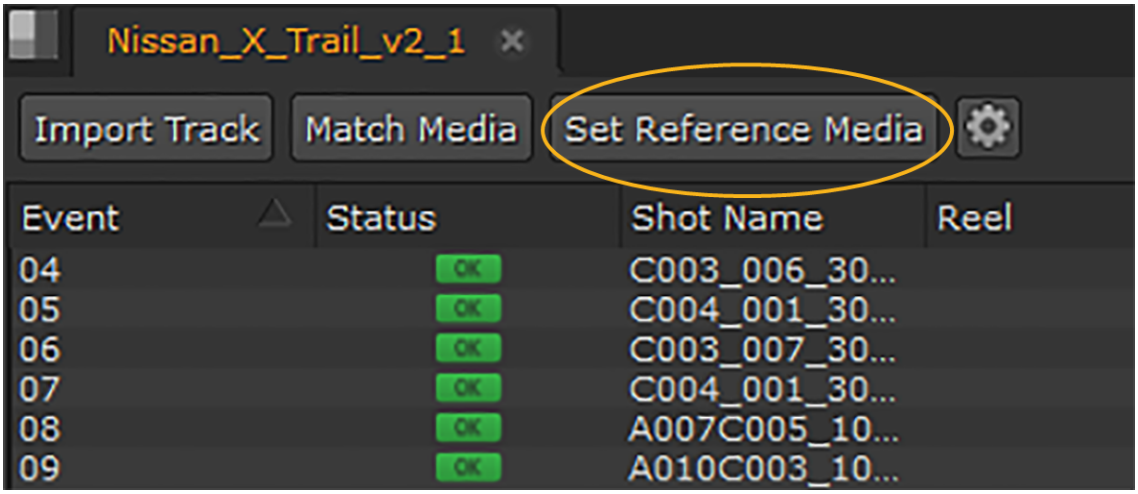
To start the PulseAudio daemon, do the following:

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:
   `autospawn = 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

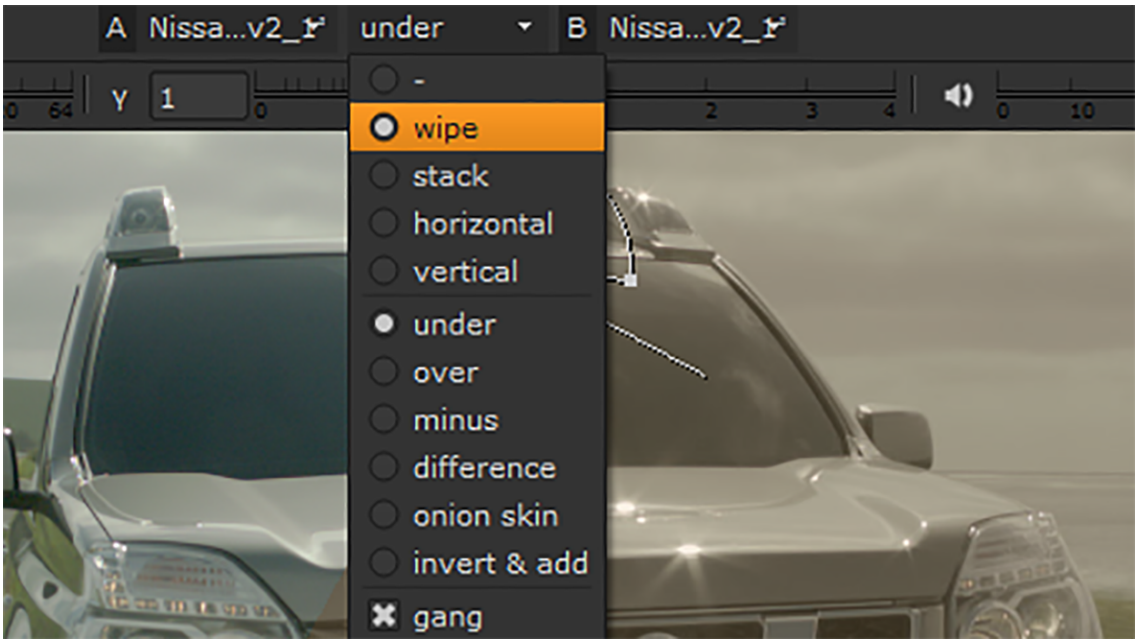
Importing a reference version of your timeline enables you to compare your current timeline against the reference media to avoid issues with continuity, missed frames, and so on.

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. SeeComparing Media for more information.

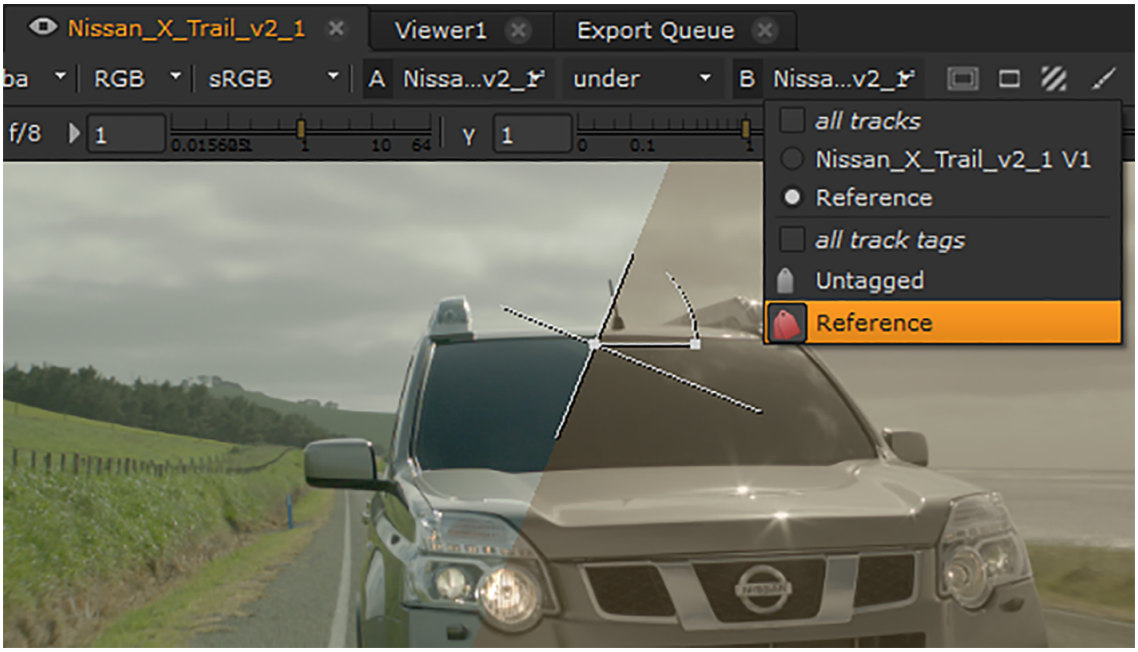


Comparing Media

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 **A** or **B** 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.

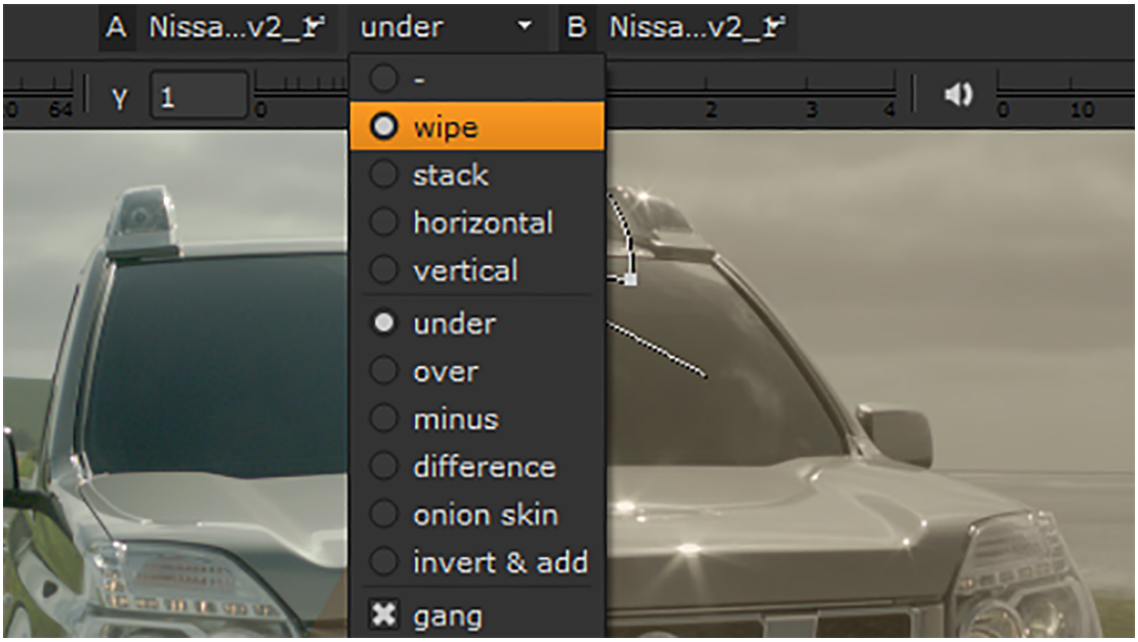
When the Viewer input buffers contain sequences, the A and B dropdowns control what is displayed in the Viewer using track names and tags. Selecting a track or tag from the dropdown displays the selected media in the Viewer.



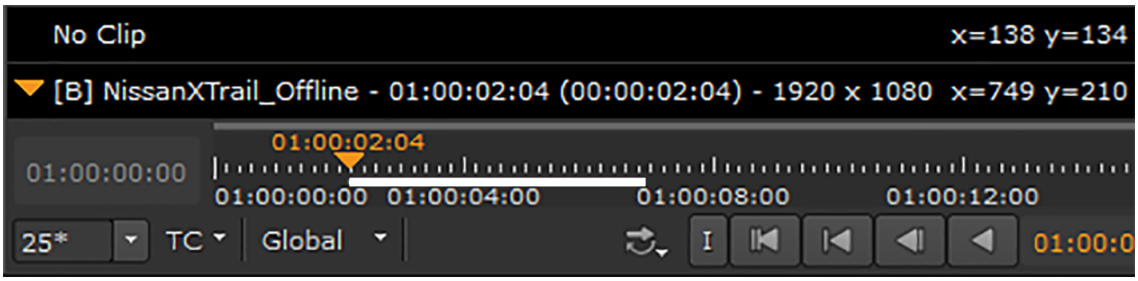
Use the **wipe**, **stack**, **horizontal**, and **vertical** modes to control how the buffers are displayed in the Viewer.

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.



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.

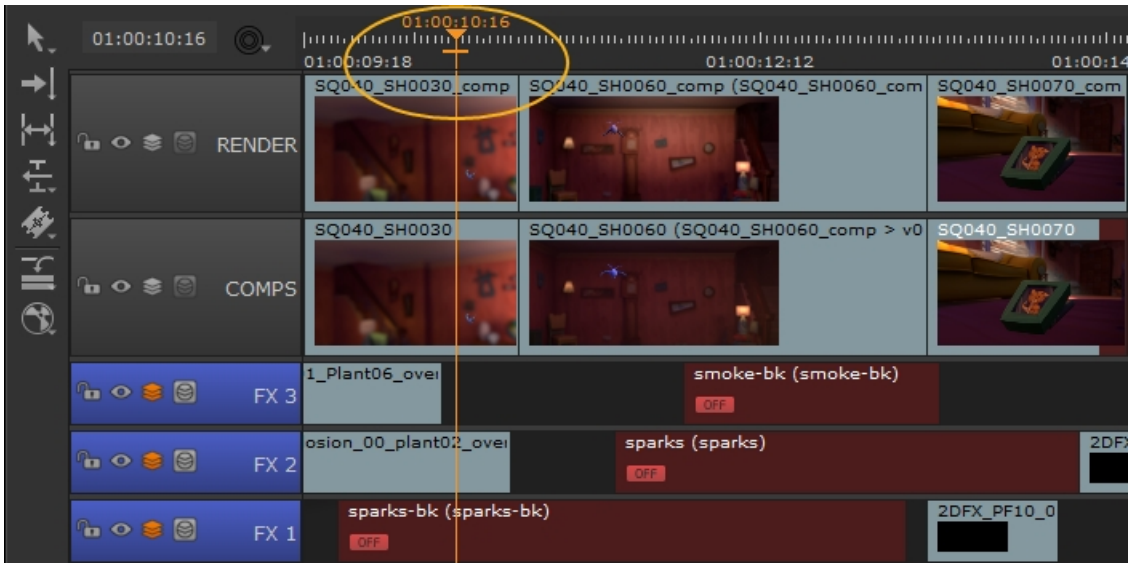


The orange triangle in the overlay denotes the clip currently supplying audio and timecode information in the Viewer.

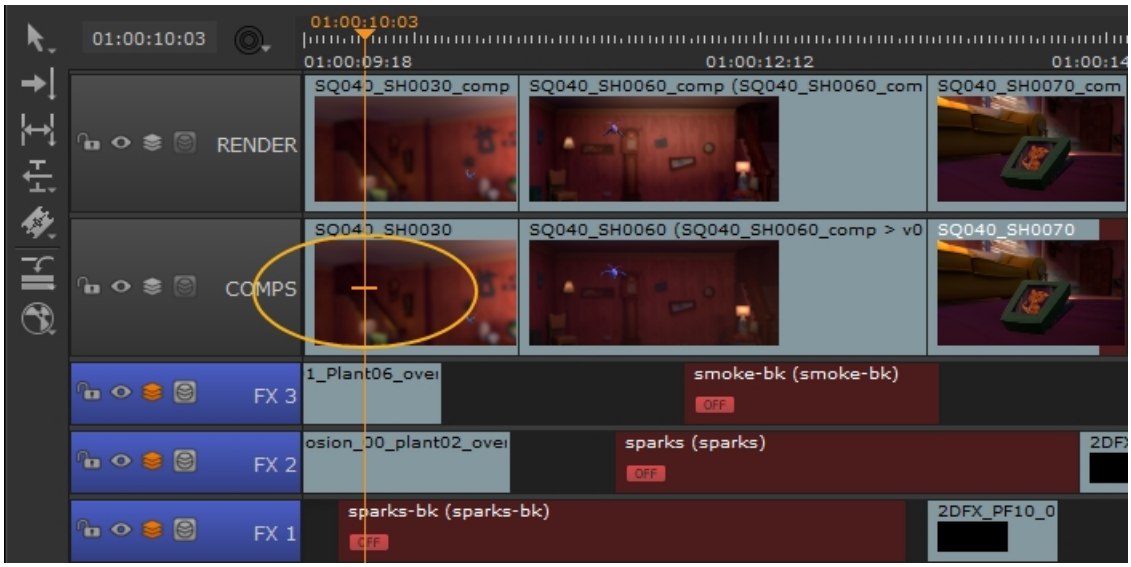
Playhead A/B Indicators

The timeline playhead indicates what is currently displayed in the Viewer using a number of different display markers. Swapping tracks between buffers automatically updates the playhead so that you always know which tracks you are looking at in the Viewer.

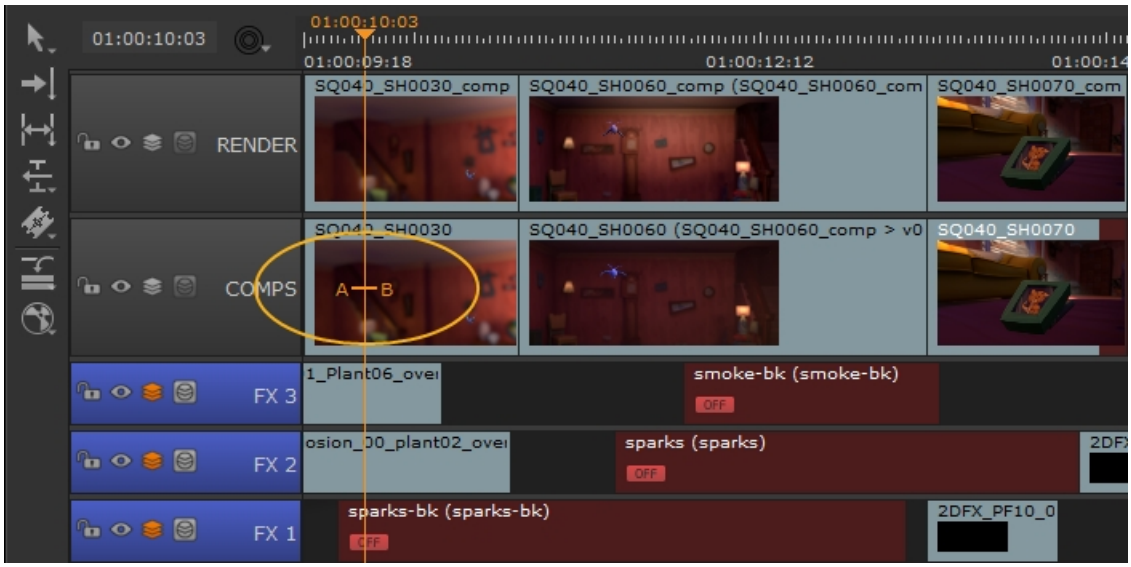
- Selecting all tracks in the current Viewer buffer draws a single line at the top of the playhead to indicate that you're viewing all tracks in the sequence from top to bottom.



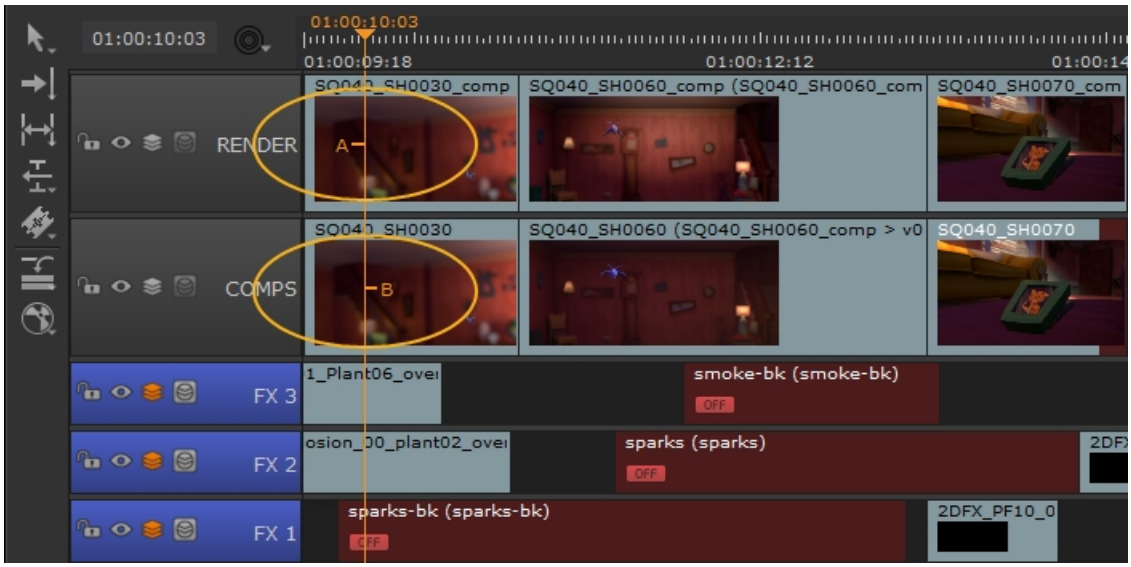
- Adding a track to the current Viewer buffer draws a single line at the playhead in the selected track.



- Adding the same track to both the A and B Viewer buffer draws a single line at the playhead in the selected track and shows A and B markers.



- Adding tracks to the Viewer A/B buffers draws A and B markers at the playhead in the selected tracks.



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 orLocalizing 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 **Hiero** > **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 Hiero, 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 Hiero, 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 Hiero's general cache directory from within Hiero by hitting **X** on your keyboard, when the focus is on the Node Graph, and then running the following Tcl command:

```
value preferences.TimelineDiskCachePath
```

Cached frames are represented in the timeline by the state of the timeline cache icon:

- Automatic caching is active. See Update Changes to Cached Frames Automatically for more details.
- 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:

• Click the timeline cache icon and select **Cache Settings**, or

• Open the **Preferences** and navigating to **Performance** > **Caching**.

Note: You can override the caching mode in the Preferences using the Sequence panel's Disk Caching Mode dropdown.

Update Changes to Cached Frames Automatically

Timeline disk caching has two modes, **Automatic** and **Manual**, providing greater flexibility when used in conjunction with soft effects to ensure the best possible performance when playing back on the timeline.

- Manual - the default mode. When frames are cached, any changes made to soft effects on the cached frames are not automatically re-cached.
- Automatic - when frames are cached, any changes to soft effects made after the initial cache operation automatically triggers a re-cache of the affected region of your cached timeline. This means that your timeline is always cached and up to date in the selected regions, regardless of changes made.

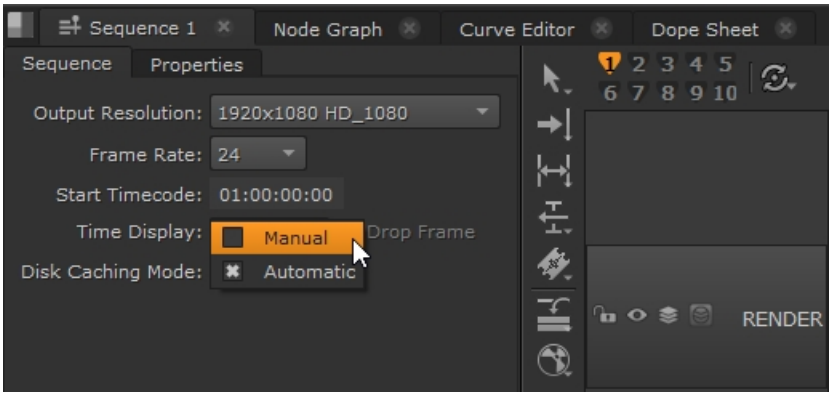
To set the caching mode for all new sequences:

1. Navigate to Nuke > Preferences (macOS) or Edit > Preferences (Linux and Windows),

   OR

   Use the Preferences keyboard shortcut Shift+S.
2. Select Performance > Caching.
3. Click the default disk caching mode dropdown and select Manual or Automatic.

   Any new sequences you create default to the option selected in the Preferences.
4. You can override this preference in the Sequence panel using the Disk Caching Mode dropdown.



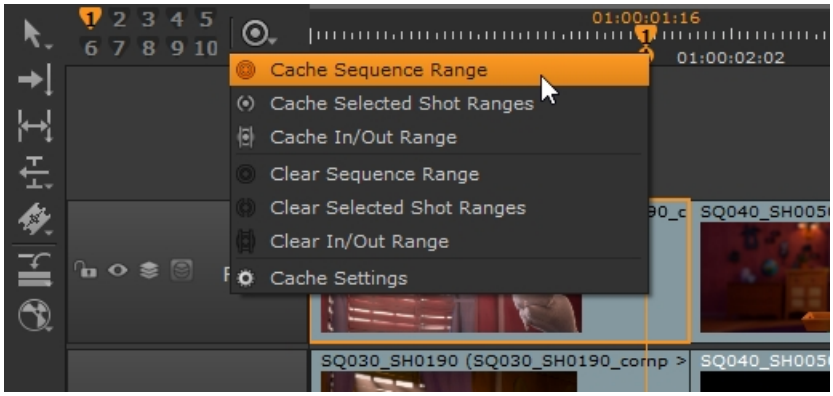
Caching Sequence Ranges

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 you cache to disk.

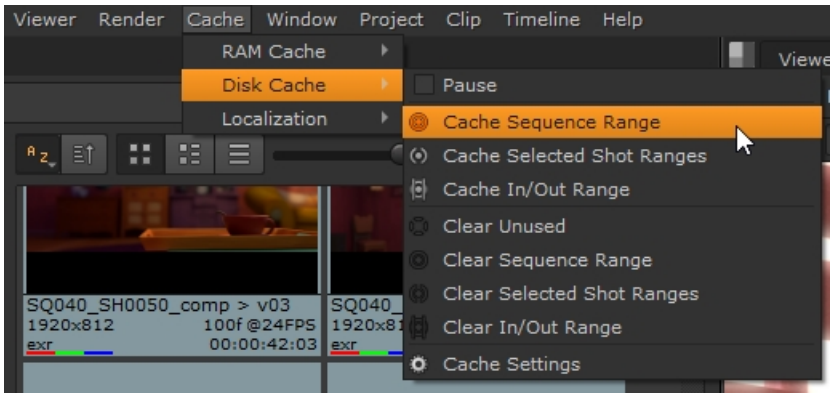
You can only cache one sequence at a time, either from the timeline itself, the **Cache** menu, or the **Project** bin:

Tip: You can pause caching at any time by navigating to **Cache** > **Disk Cache** > **Pause** in the menu bar.

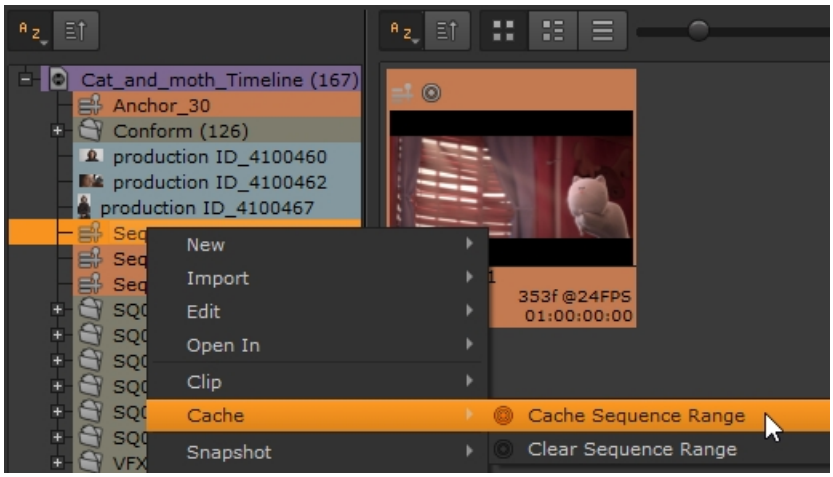
- Select the required sequence in the Sequence tab, click the icon, and then select **Cache Sequence Range**.



- Select the required sequence in the Sequence 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**.



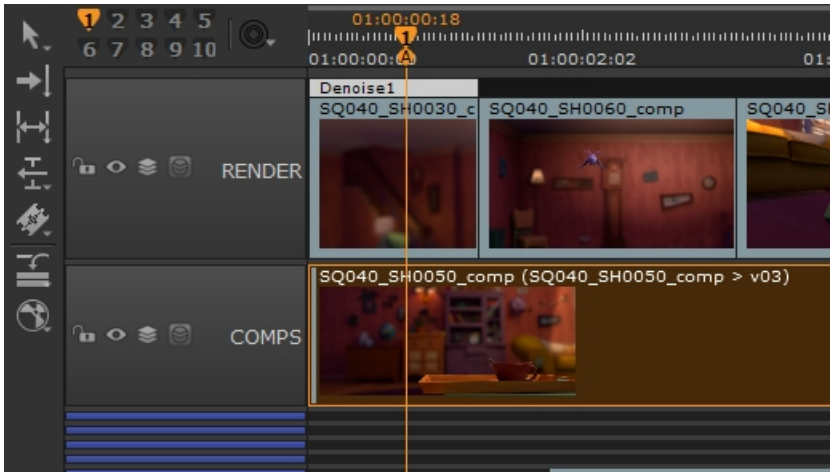
See Clearing Cached Frames for information on how to clear frames from the disk cache.

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.

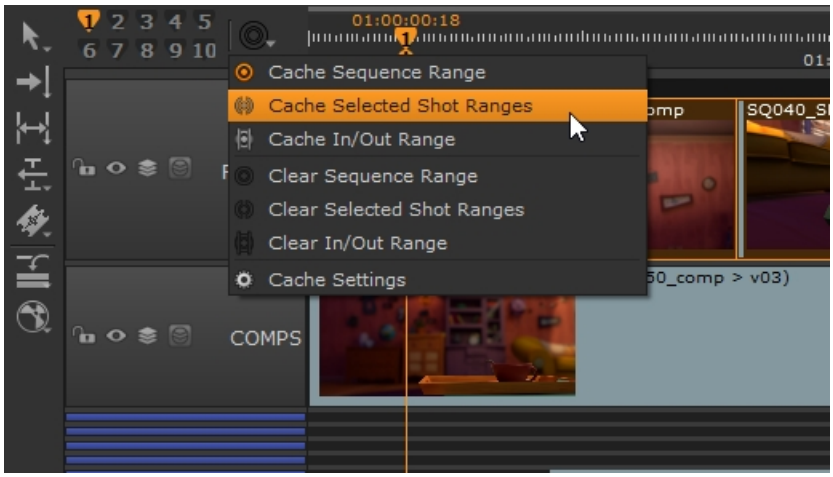
Tip: You can pause caching at any time by navigating to **Cache** > **Disk Cache** > **Pause** in the menu bar.

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 shot on the COMP track.

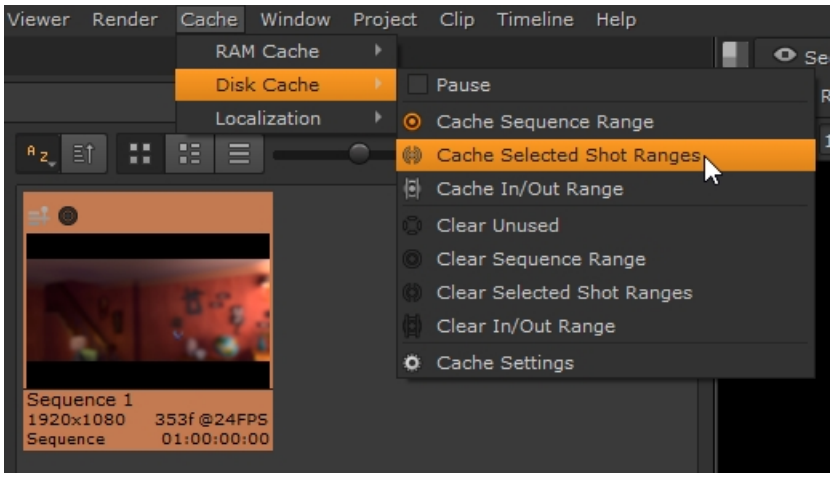


You can cache clip ranges from the timeline itself or from the **Cache** menu:

- Select the required clip range in the Sequence tab, click the icon, and then select **Cache Selected Shot Ranges**.



- Select the required clip range in the Sequence 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.

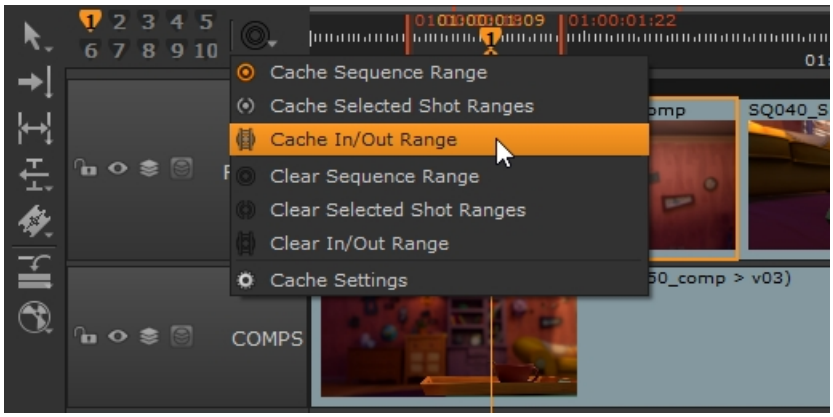
Caching In/Out Ranges

Caching ranges uses In and Out points set on the timeline to determine what you cache 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 you cache to disk. See Using In and Out Markers for more information.

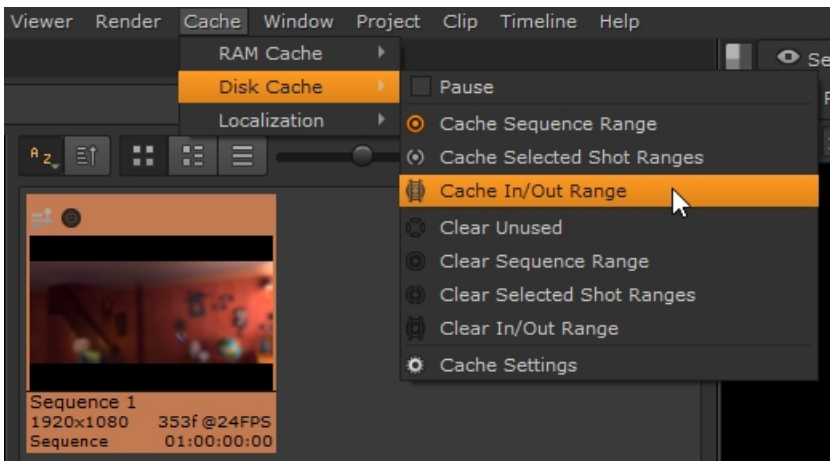
Tip: You can pause caching at any time by navigating to **Cache** > **Disk Cache** > **Pause** in the menu bar.

After setting In and Out points, you can cache that range from the timeline itself or from the **Cache** menu:

• In the Sequence tab, click the icon, and then select **Cache In/Out Range**.



• Select **Cache** > **Disk Cache** > **Cache In/Out Range** from the menu bar.



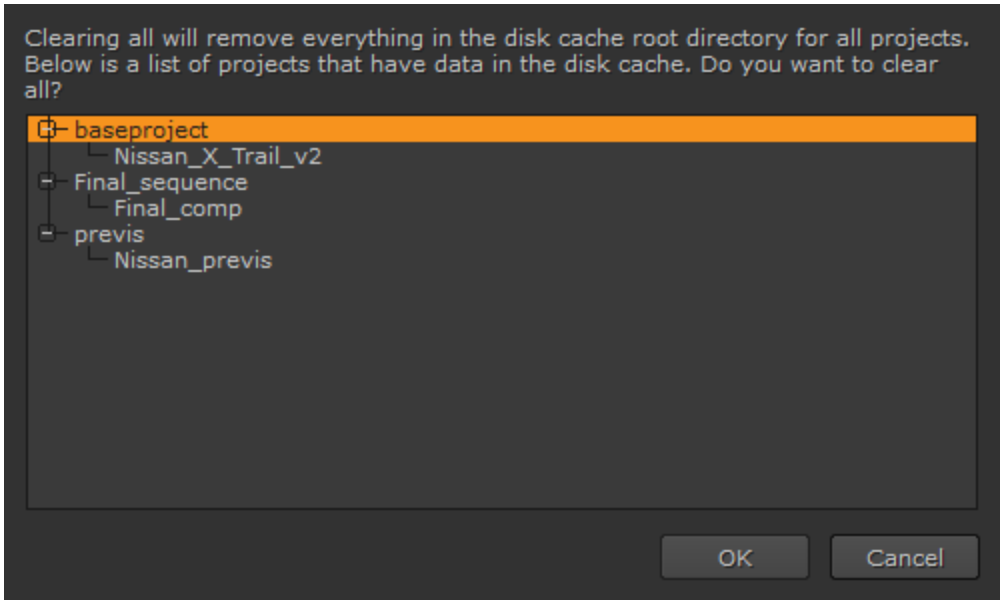
See Clearing Cached Frames for information on how to clear frames from the disk cache.

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.

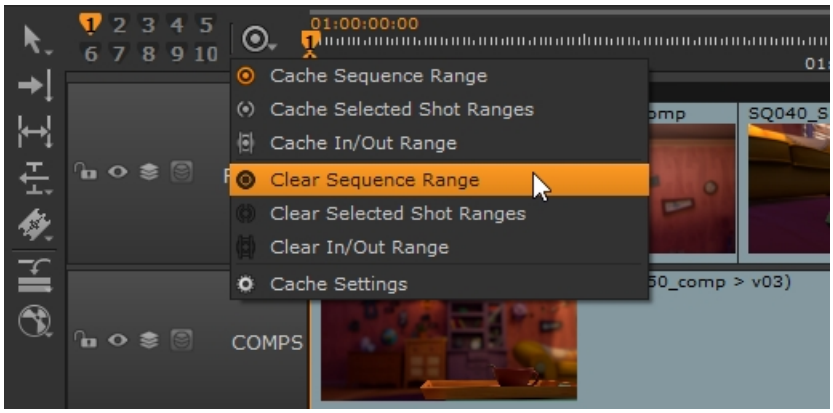


Click **OK** to proceed or **Cancel** to retain all the frames in the cache.

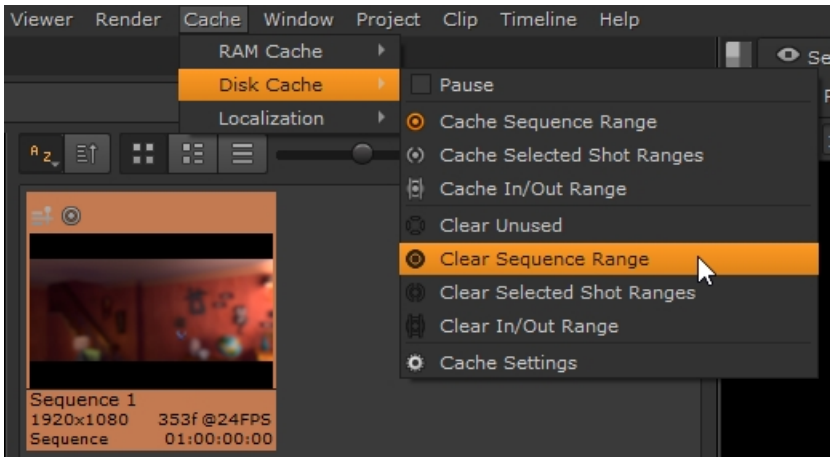
Clearing Sequences

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:

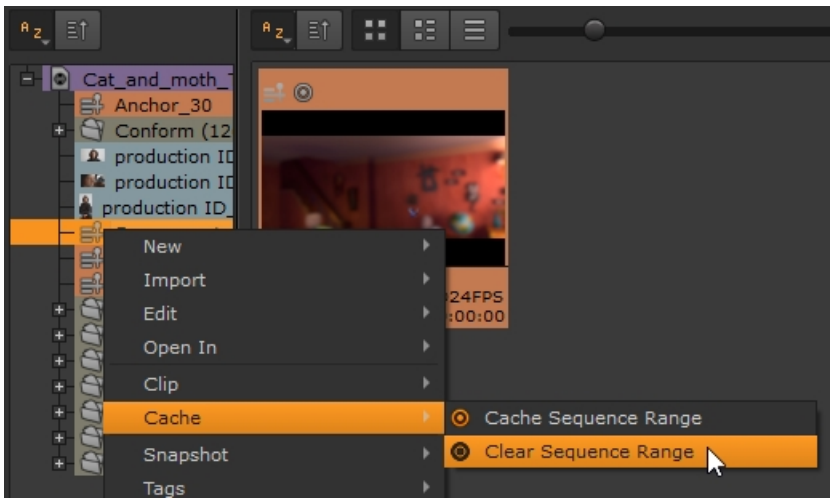
- Select the required sequence in the Sequence tab, click the  icon, and then select **Clear Sequence Range**.



- Select the required sequence in the Sequence tab and then select **Cache** > **Disk Cache** > **Clear Sequence Range** from the menu bar.



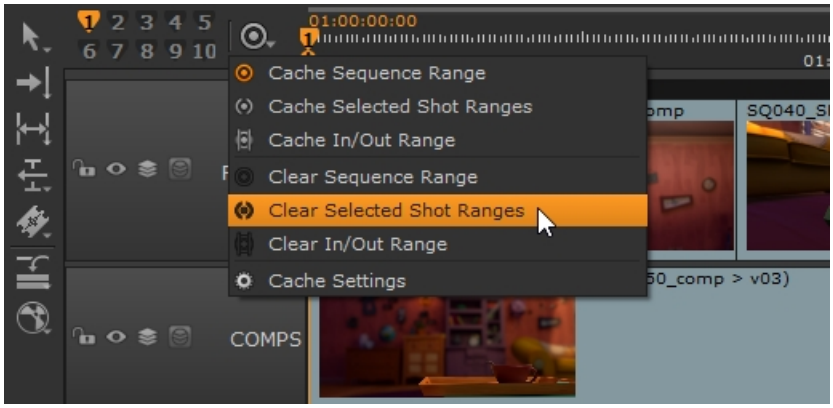
- In the **Project** bin, right-click the sequence you want to clear, and then select **Cache** > **Clear Sequence Range**.



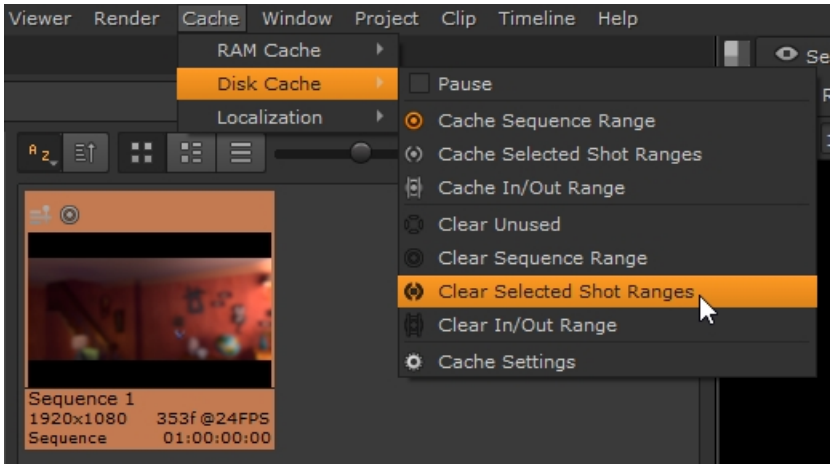
Clearing Selected Shot Ranges

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 Sequence tab, click the icon, and then select **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.

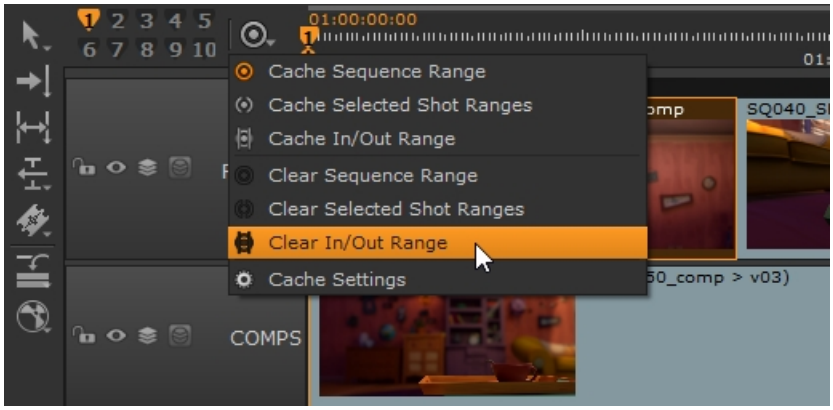


Clearing In/Out Ranges

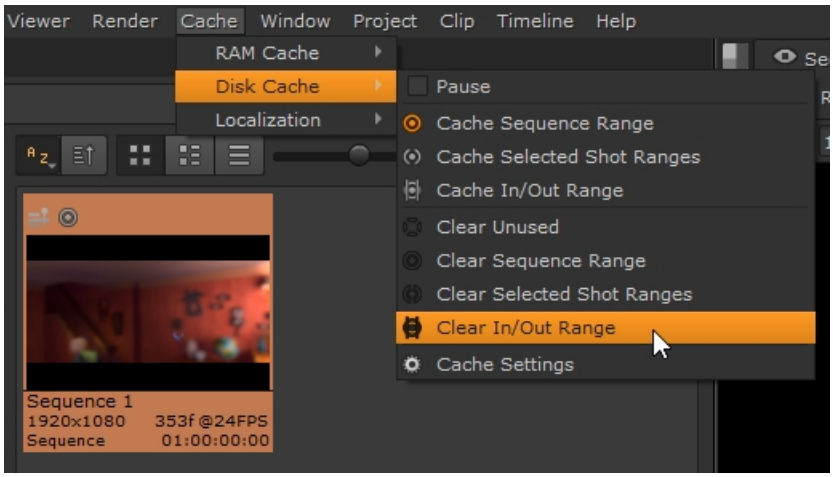
Clearing frames using In and Out markers is the most fine-grained method for removing frames from the timeline cache. Only the frames bracketed by the markers are removed, regardless of how the frames were cached originally. See Using In and Out Markers for more information.

After setting In and Out points, you can clear that range from the timeline itself or from the **Cache** menu:

• In the Sequence tab, click the icon, and then select **Clear In/Out Range**.

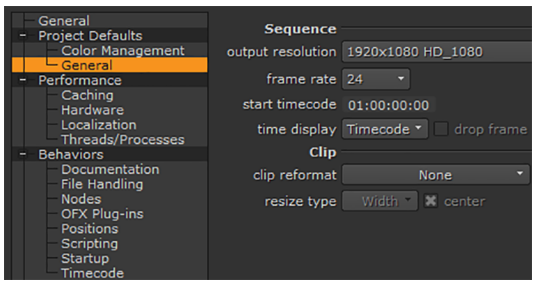


• Select **Cache** > **Disk Cache** > **Clear In/Out Range** from the menu bar.

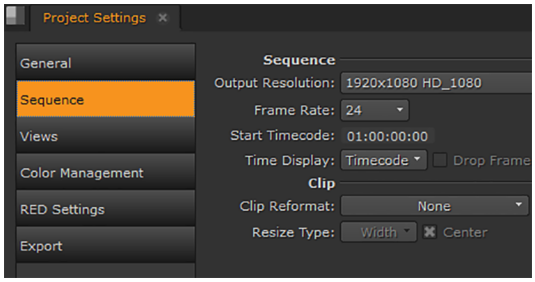


Viewing Multi-Format Timelines

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.

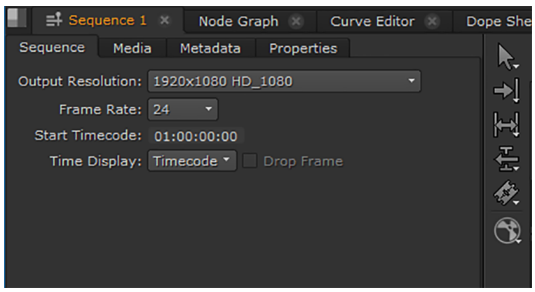


The **Preferences** dialog controls the default settings for new projects. Existing projects are unaffected by changes to the preferences.



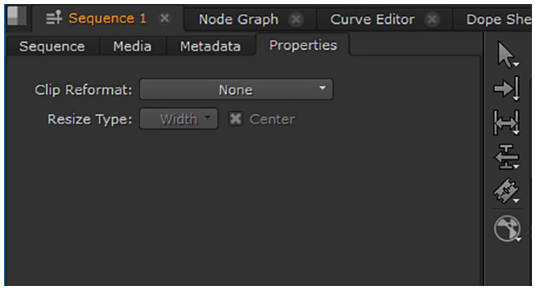
The **Project Settings** control the default settings for new sequences and shots in the current project. Existing sequences and shots are unaffected.

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 CompClips**. The reformat options in the **Export** dialog are applied after the transforms applied here. See Exporting from Hiero 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.

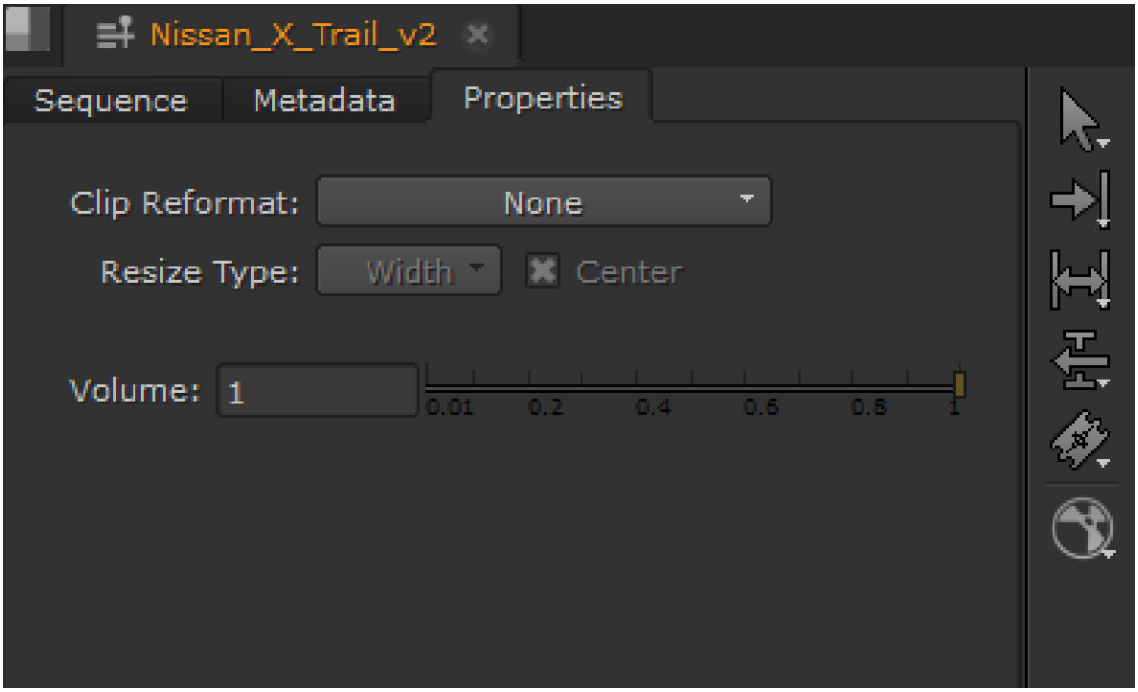
The same shot with **Output Resolution** set to 720x486.

Shots with a higher resolution than the sequence format enable you to apply soft effects, such as Transform, more easily. See Adding Effects on the Timeline 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.



Note: The **Volume** control is only available when you have an audio shot selected.

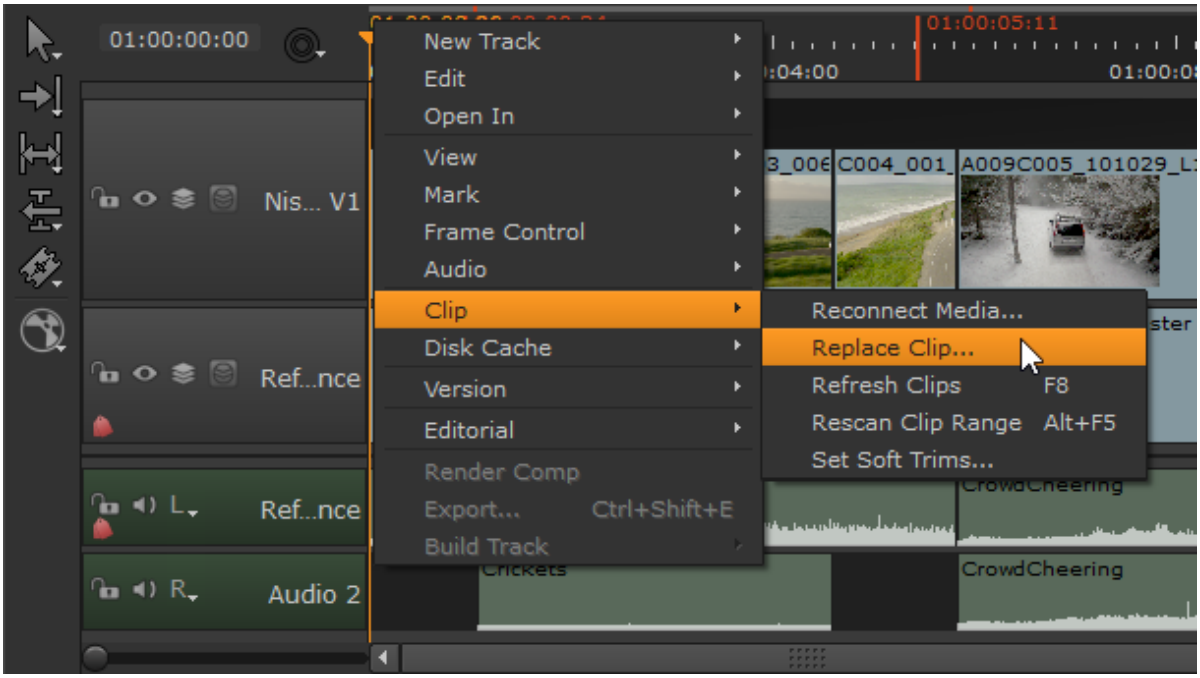
3. Click the **Clip Reformat** dropdown and selected **To Sequence Resolution**.
4. The shot is reformatted to the **Output Resolution** specified on the **Sequence** tab.
5. Depending on the source clip, you may need to change the **Resize Type**:
   - **Width** - scales the original until its width 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.

- **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. Hiero 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. Hiero 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). SeeSetting Soft Trims for more information.

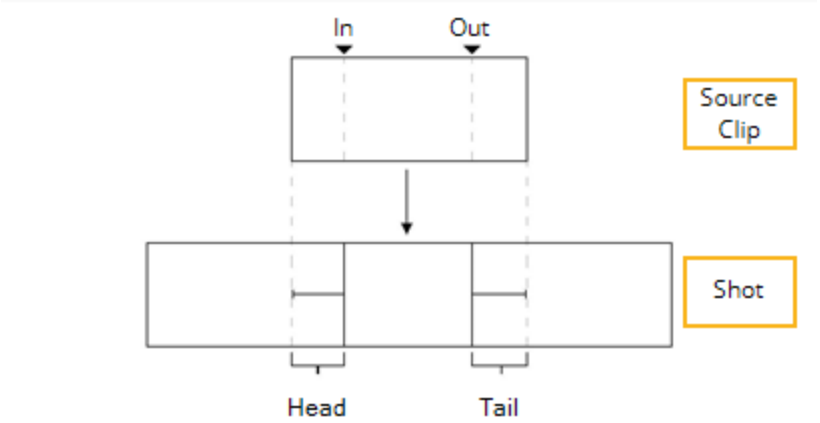
Setting Soft Trims

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**.

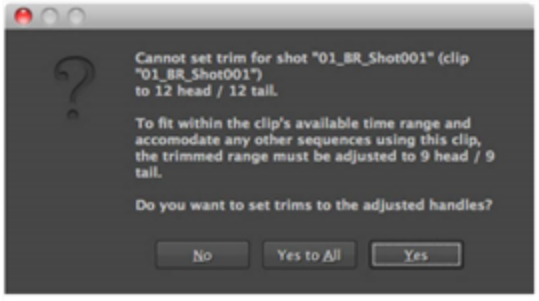
   The **Set Soft Trims** dialog displays.
3. Set the number of frames to add to the head and tail of each shot:



   • **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.



   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

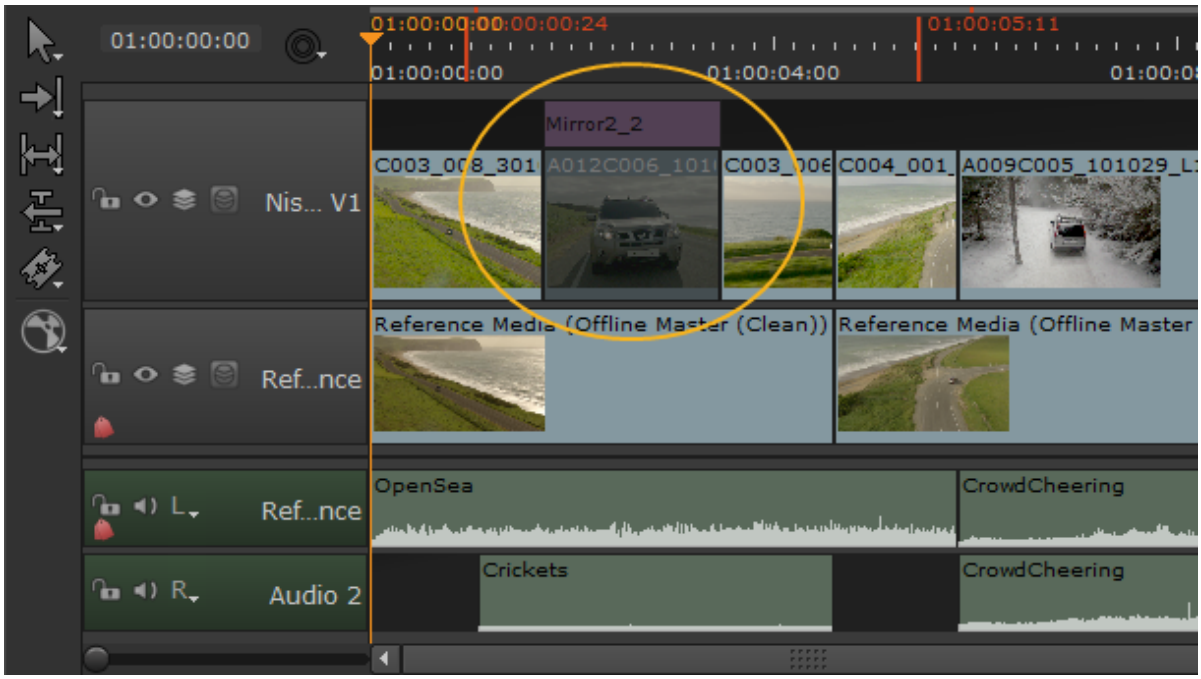
You can temporarily enable or disable tracks and shots on the timeline to selectively view your media without removing shot(s), for example if you wanted to view to lower level video tracks within a timeline.

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 Between Shots and Audio Clips

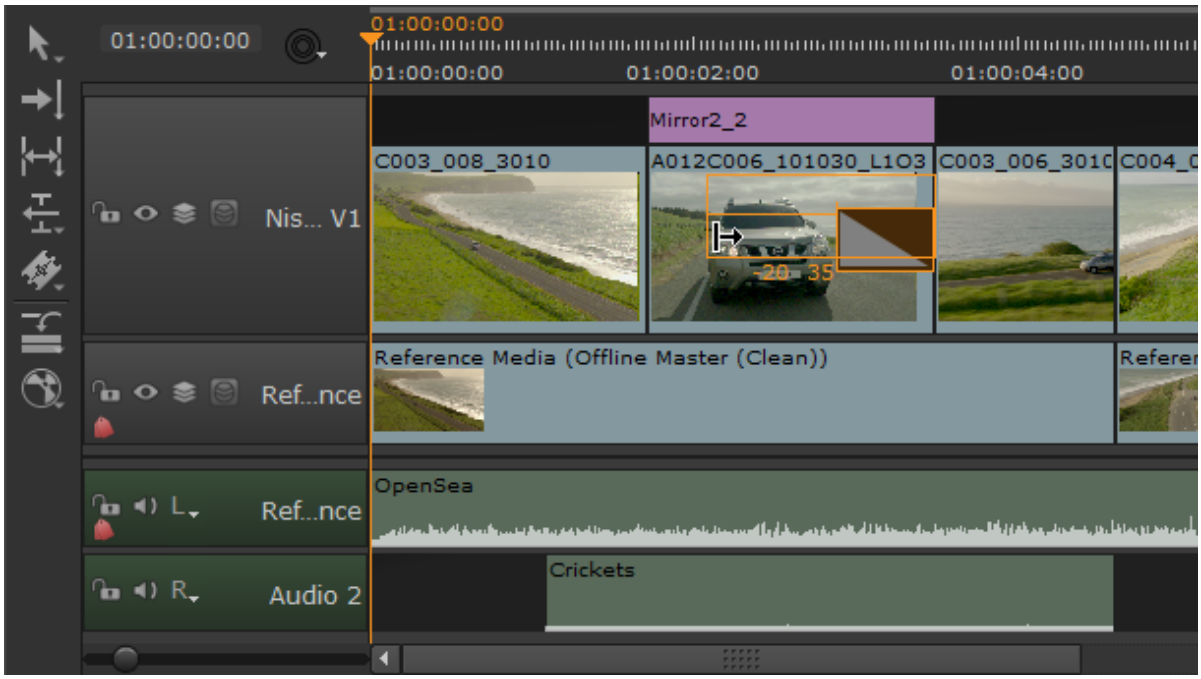
Hiero supports video and audio fade transitions as well as dissolves between shots on the same track. Transitions come in three flavors:

- **Fade in** - fades in from black on a single shot or fades in audio.
- **Fade out** - fades out to black on a single shot or fades out audio.
- **Dissolve** / Audio Crossfade - fades out from one shot or audio clip and into the next, by merging frames or audio.

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:

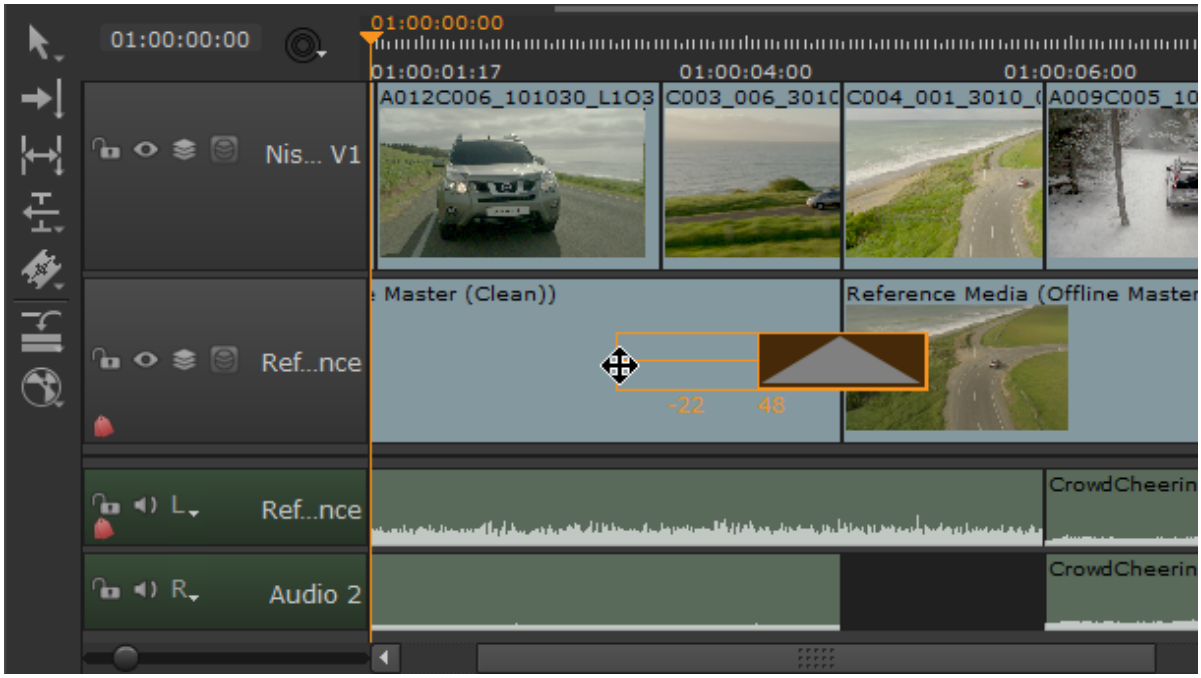
1. Right-click the target shot or audio clip and select **Editorial** > **Add Transition** > **Fade In**, **Fade Out**, **Audio Fade In** or **Audio Fade Out** to add the fade icon.
2. Adjust the fade by dragging the fade icon using the **Multi Tool** or **Move/Trim** tool.



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.

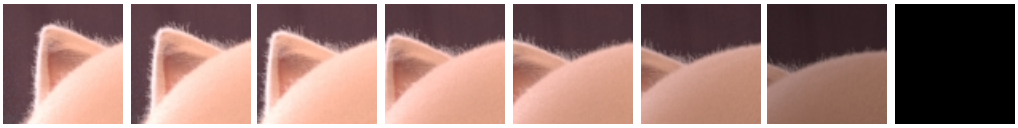
1. Select the **Multi Tool** or **Roll Edit** tool and hover the mouse pointer over an edit between two shots or audio clips.
2. > **Tip:** Clicking and holding the edit point displays available handles as a red overlay.
3. Right-click and select **Editorial** > **Add Transition** > **Dissolve** or **Audio Crossfade**, or use the **Ctrl/Cmd+T** keyboard shortcuts, to add the dissolve icon to the edit.
4. Adjust either side of the dissolve by dragging the icon, in a similar way to using the **Multi Tool** or **Move/Trim** tool.



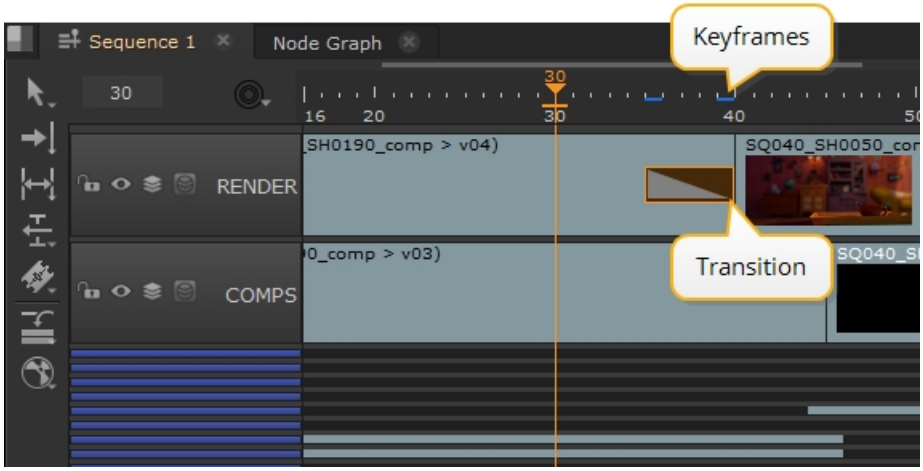
Non-Linear Transitions

Transitions in Hiero are linear by default, meaning that the fade or dissolve happens at the same rate throughout the transition. For example, a linear fade out on the last five frames of a shot would appear as follows:

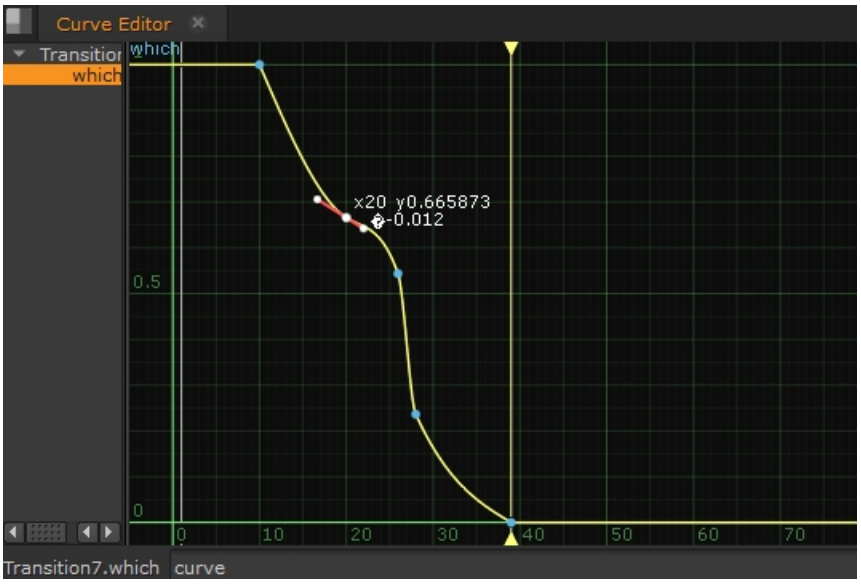
Non-linear transitions allow you to vary the speed at which the transition happens, giving you more creative control. For example, you might want a fade out to be slow and then speed up towards the end of the shot.



When you add a transition, keyframes are added to the timeline as blue chips, so you can see where the transition begins and ends.

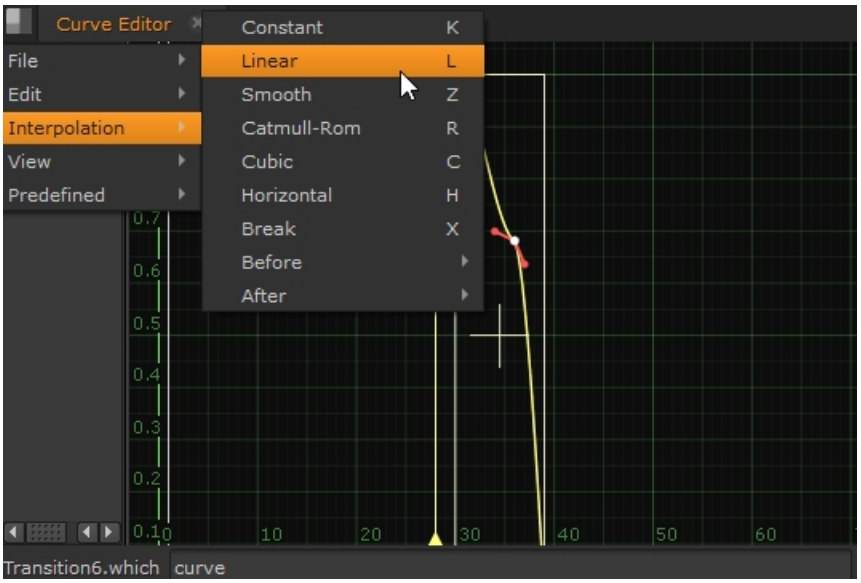


You can adjust the keyframes in the Curve Editor to achieve the required transition speed. You can adjust the keyframes manually by dragging them to different frames and adjusting the curve handles to change the speed of transition. You can also add keyframes to edit the curve with even more precision by pressing Ctrl/Cmd+Alt and clicking on the curve.



The Curve Editor also has some preset curves that you can apply to keyframes:

1. Select all the keyframes by pressing Ctrl/Cmd+A.
2. Right-click in the Curve Editor and select Interpolation to display the available presets.



See Using the Curve Editor for more information on the Curve Editor and keyframes.

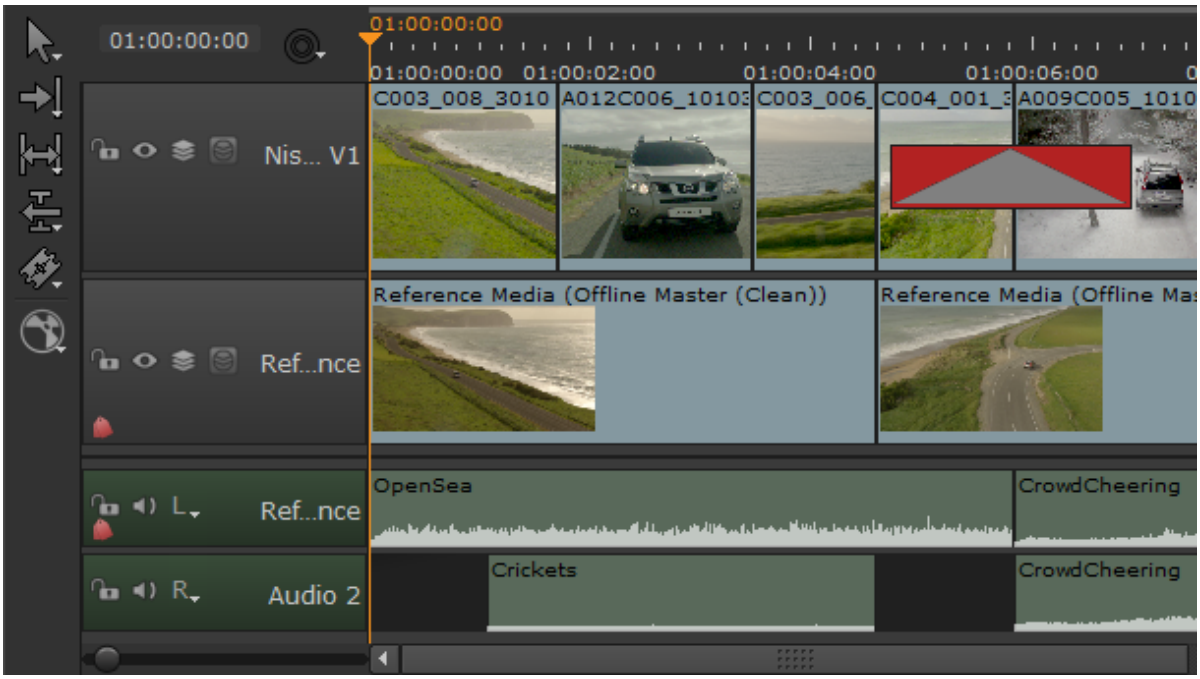
Invalid Transitions

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, Hiero 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.

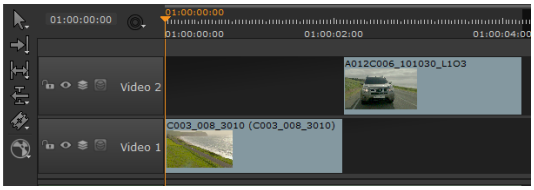
     Retiming a shot to 200% renders frames 1, 3, 5, 7, and so on in the Viewer, but the item's length is halved on the timeline.
   - **Keep timeline duration** - the shot length on the timeline is maintained regardless of the retime applied.

     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.

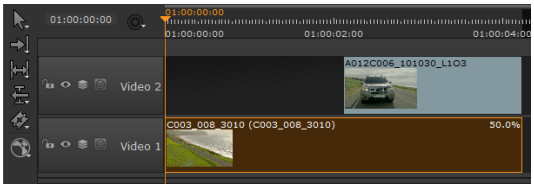
     Retiming a shot to 200% renders frames 1, 3, 5, 7, and so on in the Viewer, but the item's length on the timeline remains the same. If no extra frames are available from the source, the item is filled with black frames.
3. Double-click the **Speed** column and enter the retime value.

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.

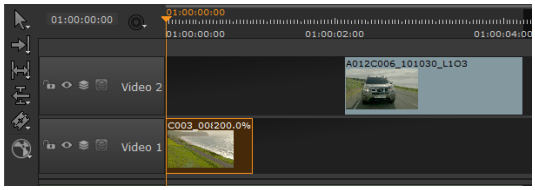
Notice that the **Keep timeline duration** method doesn't change the length of the shot on the timeline and inserts blank filler frames on the 200% retime?



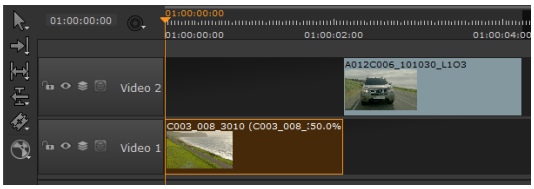
Original clip



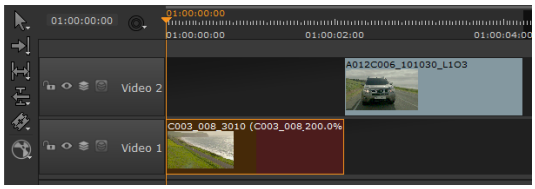
50% **Source** retime



200% **Source** retime



50% **Destination** retime

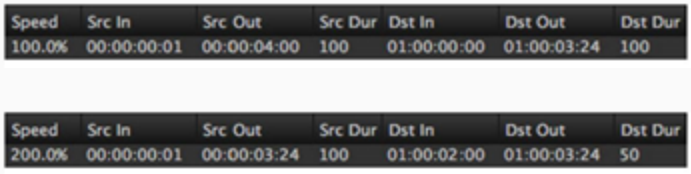
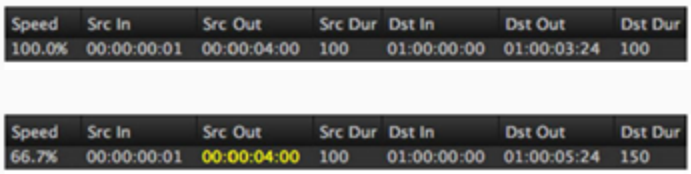


200% **Destination** retime

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**:

Modify	Using	Result
Src In	Retime	Adjusts the event's **Src In** and retimes the remaining frames to maintain **Dst Duration**. Before and after a 2 second **Src In** increase: Speed 100.0% | Src In 00:00:00:01 | Src Out 00:00:04:00 | Src Dur 100 | Dst In 01:00:00:00 | Dst Out 01:00:03:24 | Dst Dur 100 Speed 50.0% | Src In 00:00:02:01 | Src Out 00:00:04:00 | Src Dur 50 | Dst In 01:00:00:00 | Dst Out 01:00:03:24 | Dst Dur 100
Src Out	Retime	Adjusts the event's **Src Out** and applies a retime to maintain **Dst Duration**. Before and after a 2 second **Src Out** increase: Speed 100.0% | Src In 00:00:02:01 | Src Out 00:00:06:00 | Src Dur 100 | Dst In 01:00:00:00 | Dst Out 01:00:03:24 | Dst Dur 100 Speed 151.0% | Src In 00:00:00:01 | Src Out 00:00:06:00 | Src Dur 151 | Dst In 01:00:00:00 | Dst Out 01:00:03:24 | Dst Dur 100
Src Dur	Retime	Adjusts the event's **Src Dur** and **Src Out**, and applies a retime to

Modify	Using	Result
		maintain **Dst Duration**. Before and after a 50 frame **Src Dur** increase: 
Dst In	Retime	Adjusts the event's **Dst In** and retimes the remaining frames to maintain the relationship between **Dst In** and **Out**. Before and after a 2 second **Dst In** increase: 
Dst Out	Retime	Adjusts the event's **Dst Out** and retimes the remaining frames to maintain the relationship between **Dst Out** and **In**. Before and after a 2 second **Dst Out** increase: 
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: 

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:

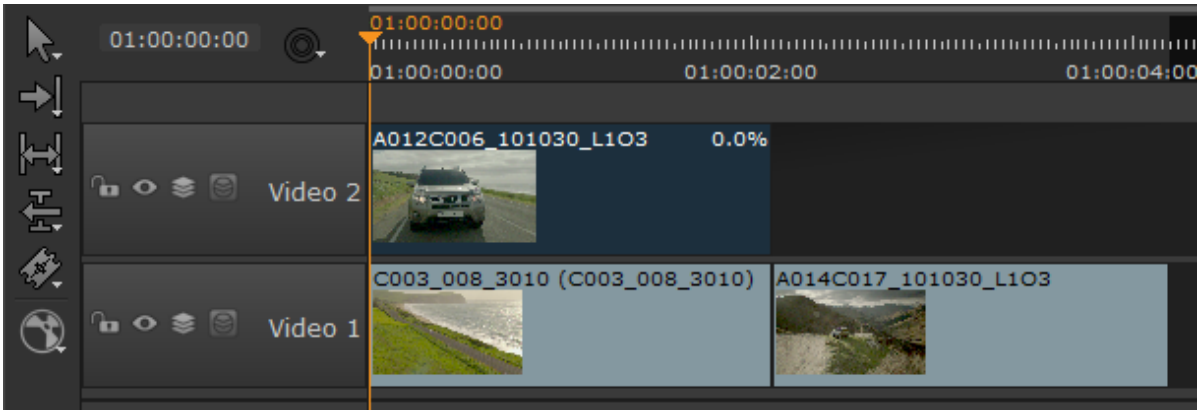
1. Select the target item(s) on the timeline.
2. Right-click the item and select **Editorial** > **Make Freeze Frame**,

   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

Hiero allows you to perform merges between tracks in the timeline, for example overlaying a logo on a shot without heading into Nuke. 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.

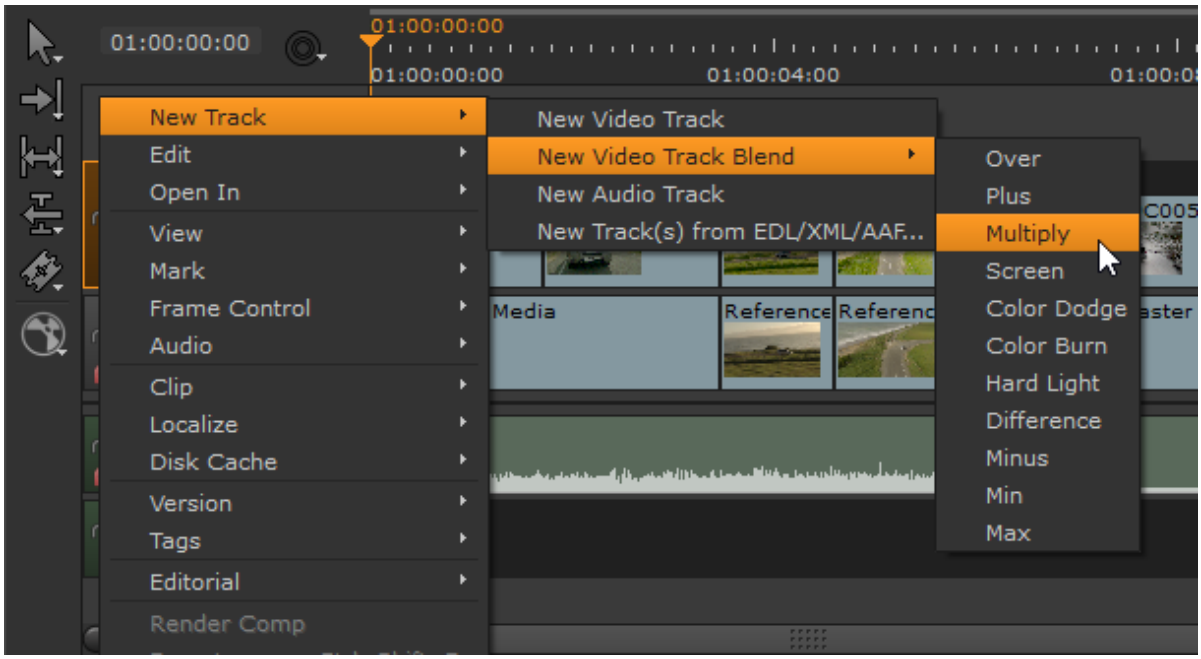
See the Nuke Online Help for a full description of the available blend modes.

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 Adding Effects on the Timeline for more information) and blended tracks are included along with the shot in Export operations. See Exporting from Hiero and using **Create Comp Clips** during round-tripping Building VFX Tracks and Comp Clips for more information.

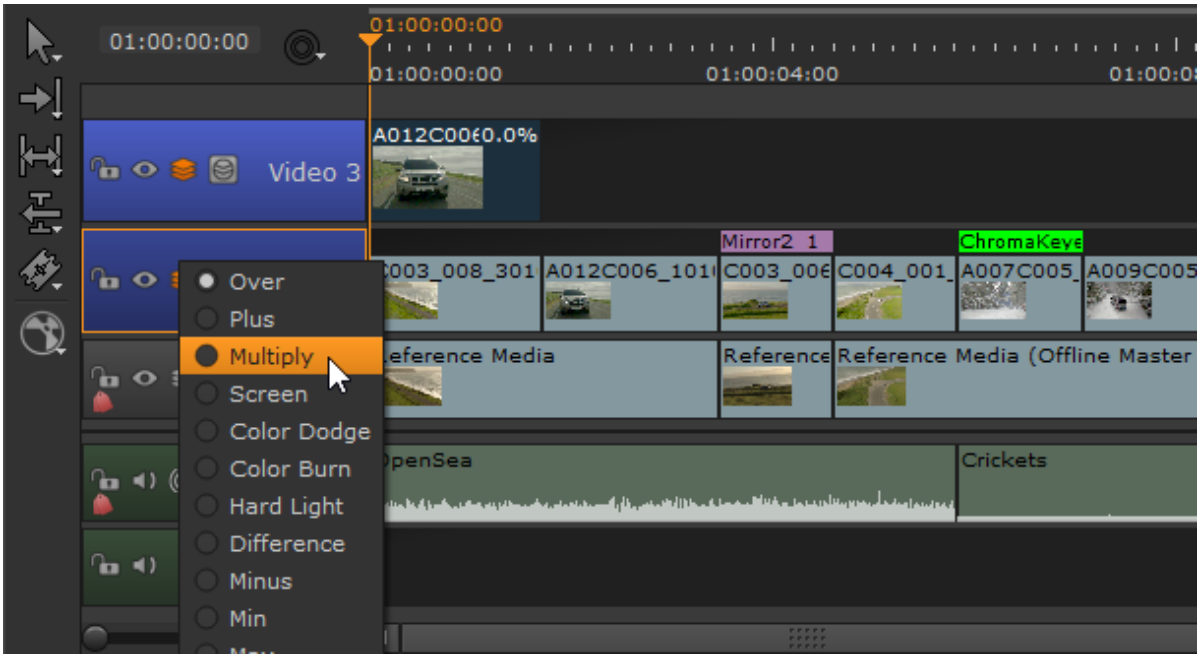
Adding New Blend Tracks

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.



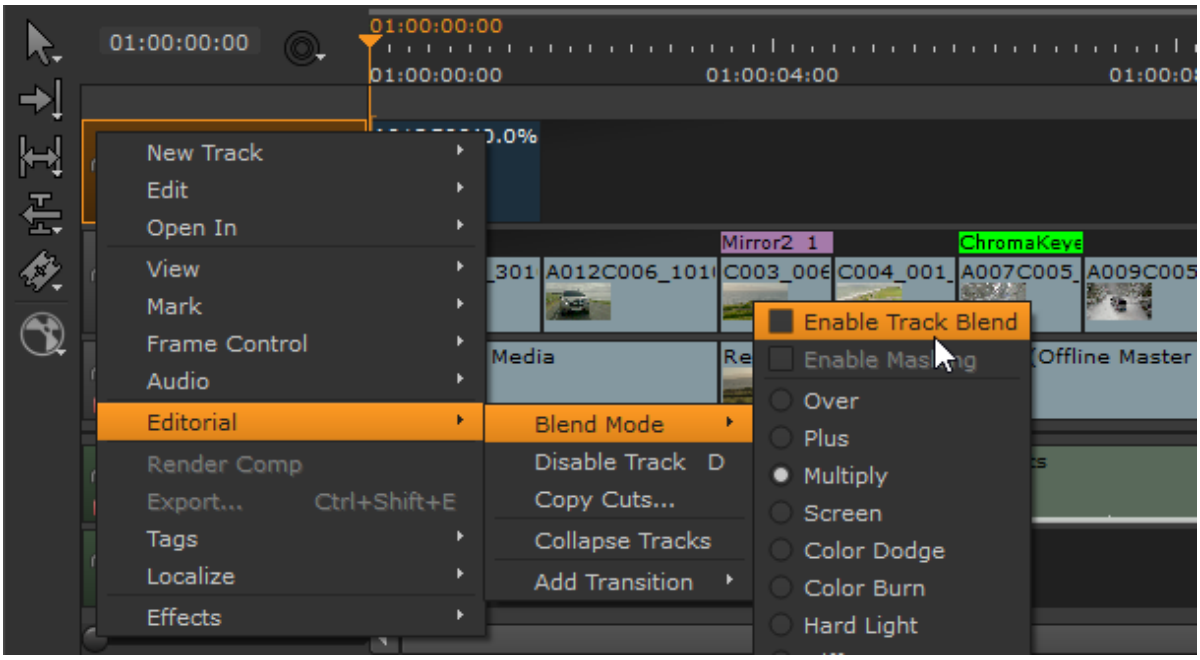
4. Click the **Blend** icon to toggle blending on and off.

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.

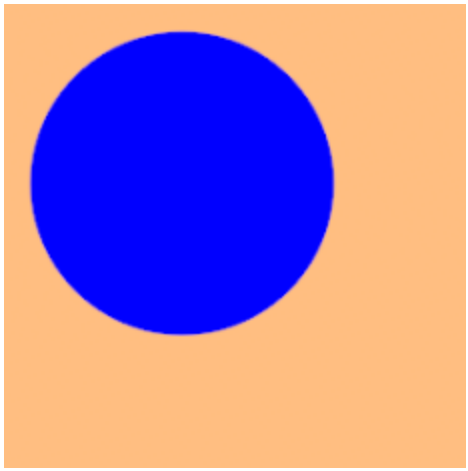
2. You can add shots to the blend track as you would any other shot. See Adding Clips to the Timeline for more information.

   The Viewer displays the higher track blended with the track below.

3. Click and hold the **Blend** icon to select the blend mode.

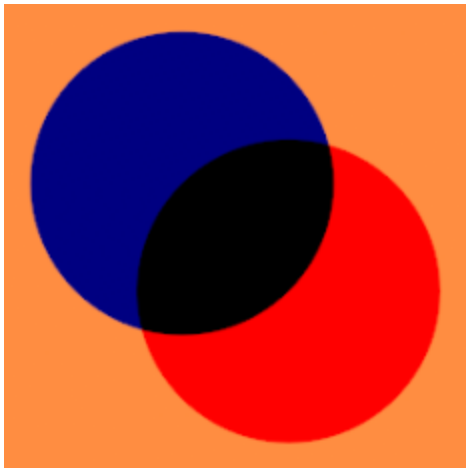
Masking Blended Tracks

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.

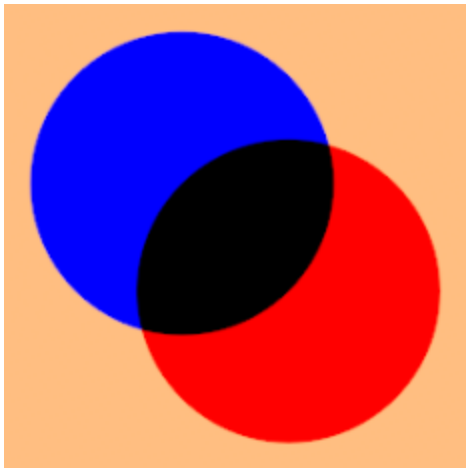


Background track **B**.

Blend track **A**.



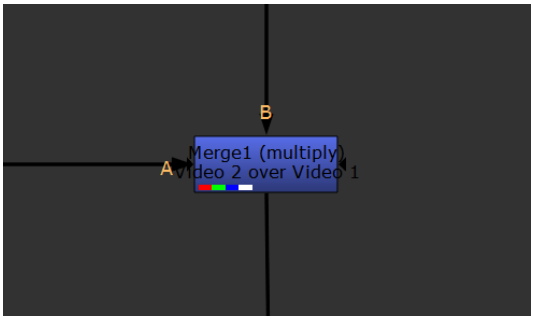
A multiplied by **B** with Masking disabled.



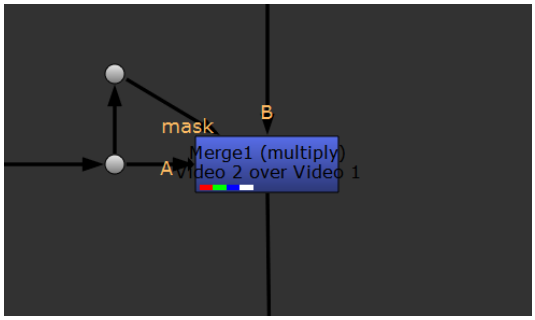
A multiplied by **B** with Masking enabled.

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.

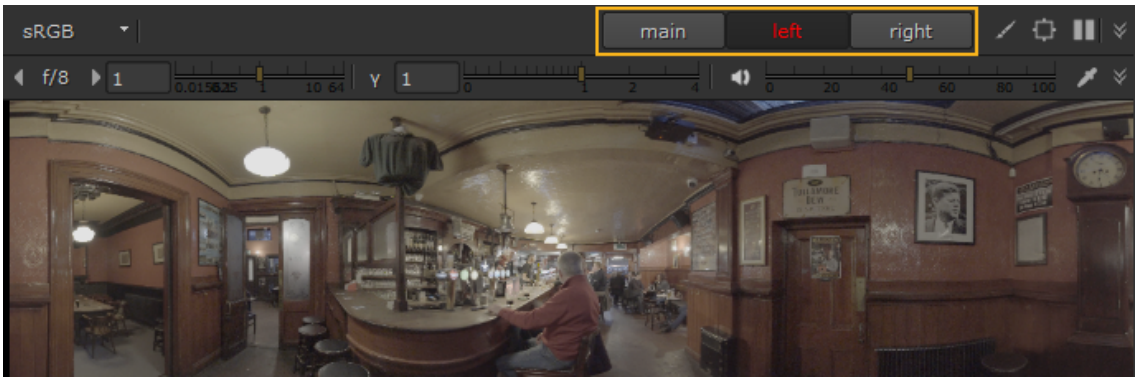


The same comp using the **multiply** merge operation, but with masking enabled.

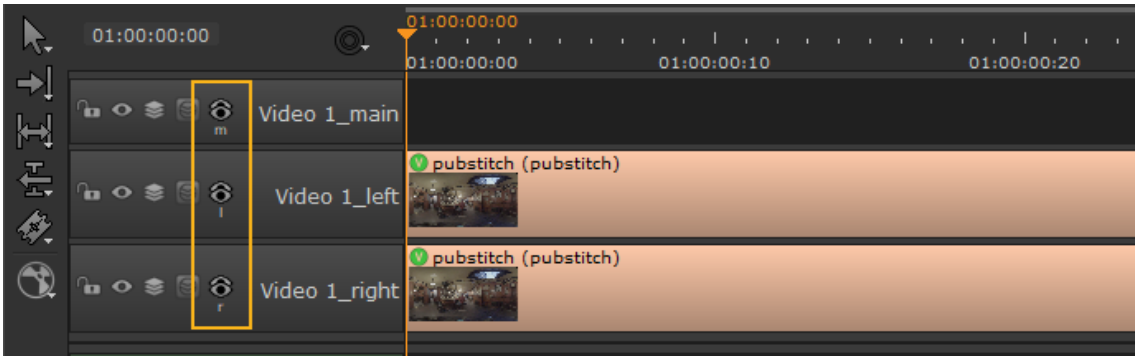
Stereoscopic and Multi-View Projects

Hiero 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.

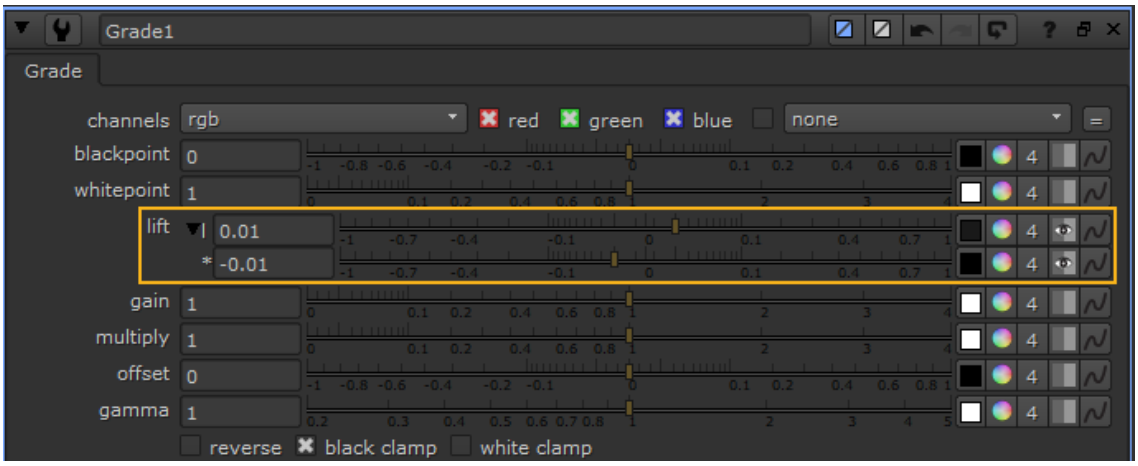
Existing views inside a project are managed in the Viewer, timeline, and in the **Properties** panel of most soft effects. In the Viewer, all views in the current project are represented by buttons that allow you to switch between views with a single click. See Displaying Views in the Viewer for more 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 export multi-view shots and effects in a similar way to regular shots. See Exporting Multi-View Source Clips for more information.

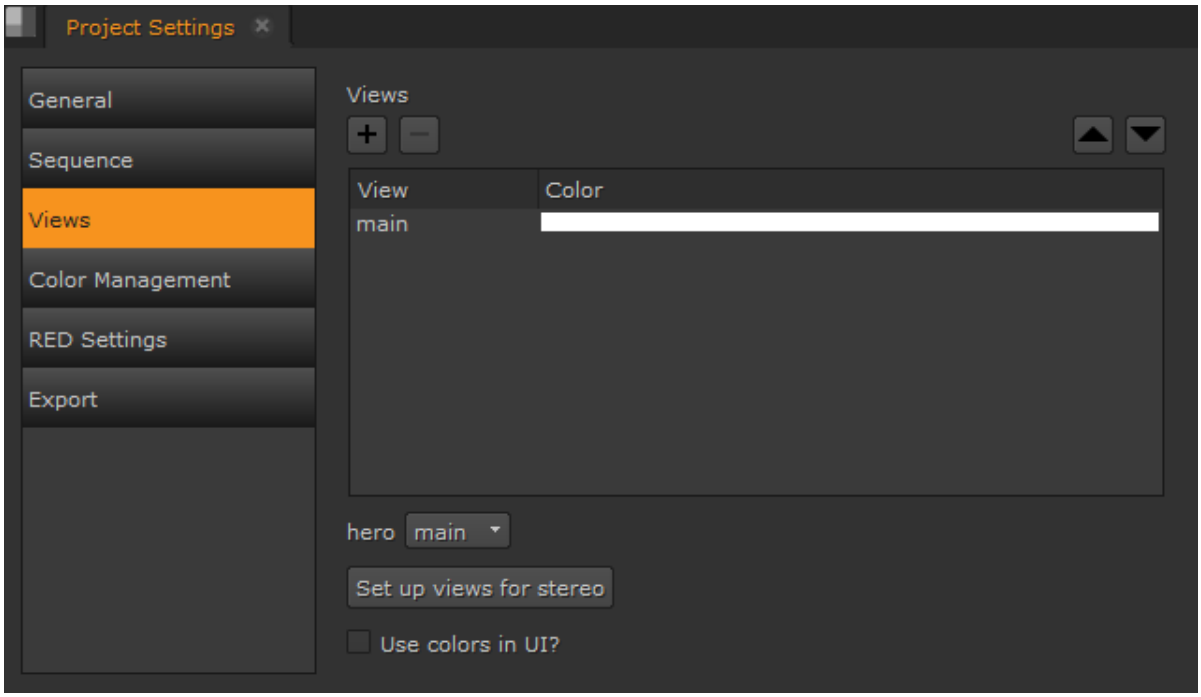
Creating Views in a Project

Views in Hiero 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**, Hiero 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.

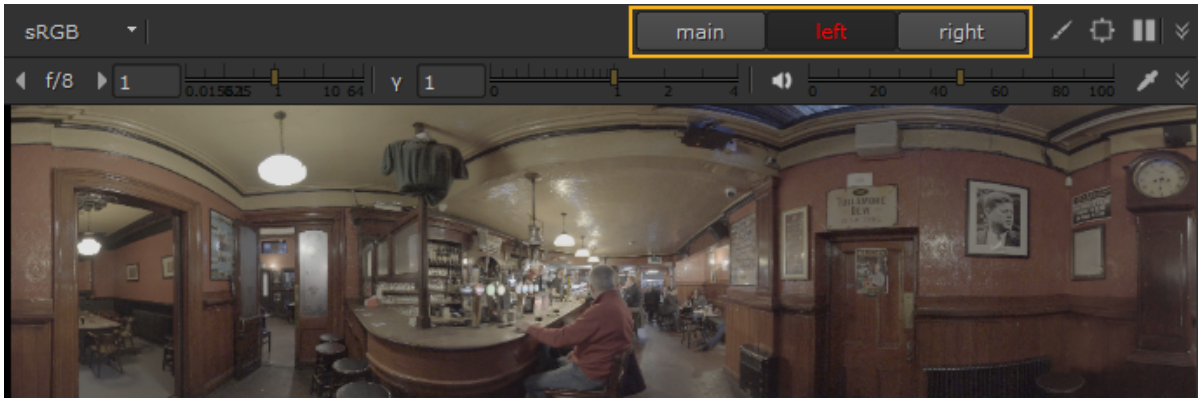
1. Select **Project** > **Edit Settings**.
2. Go to the **Views** sub-menu. The available views are listed in the **Views** panel.



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 Hiero.

Importing Source Clips

Hiero supports multi-view clips in two formats:

- **single-view** - formats such as **.dpx** and **.jpg**, with file names that Hiero 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**, **.sxr**, and **.mov**, that support multiple views per file. For example:

  `myStereo_comp1.####.exr` containing two layers called **left** and **right**.

> **Note:** Hiero 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**, Hiero 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.

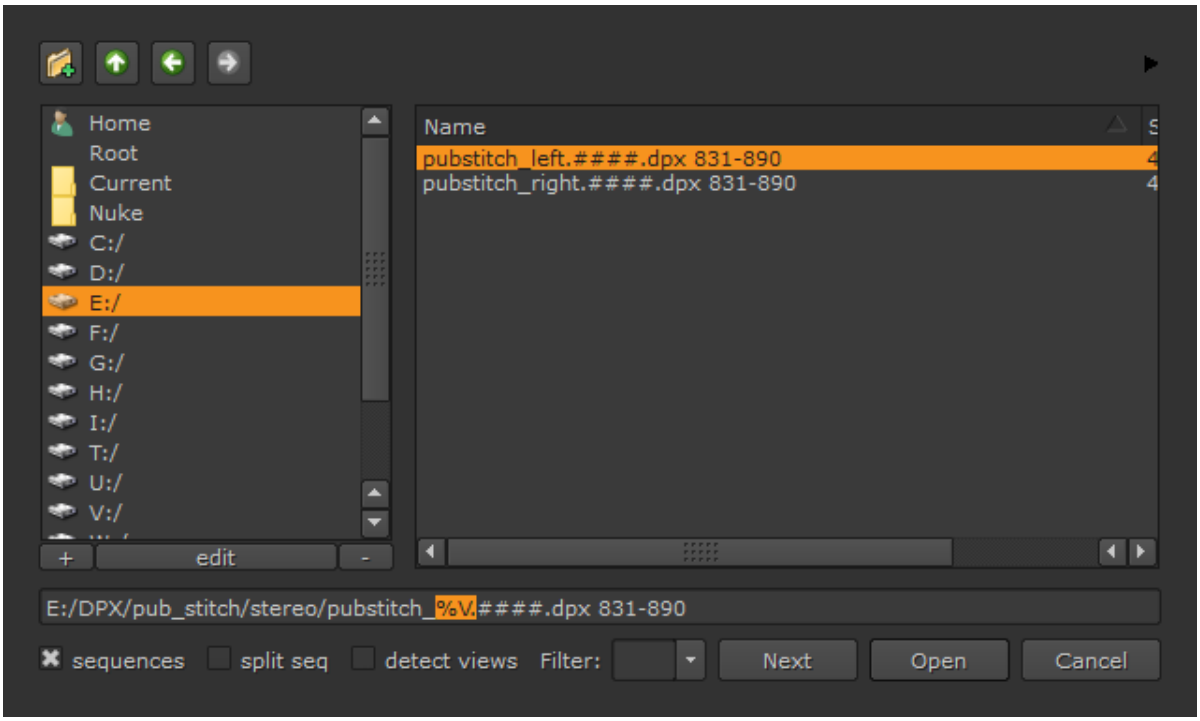
File Name Variables

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, Hiero 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**

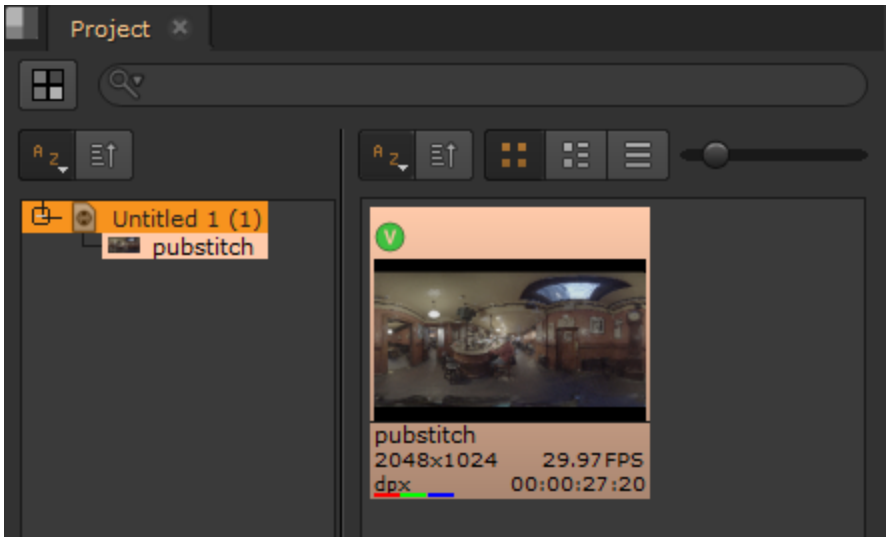


4. Click **Open**.

   Hiero 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 V icon, which can display any of the combined views.



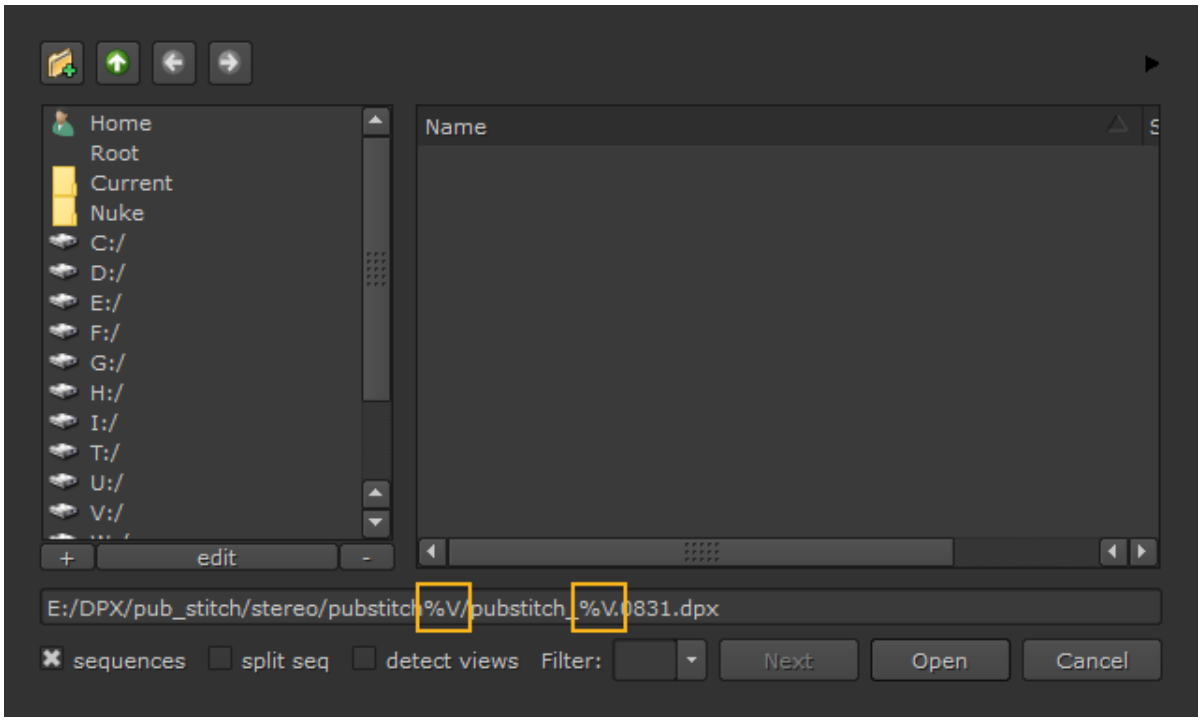
See Displaying Views in the Viewer and Displaying Views in the Timeline for more information.

Directory Name Variables

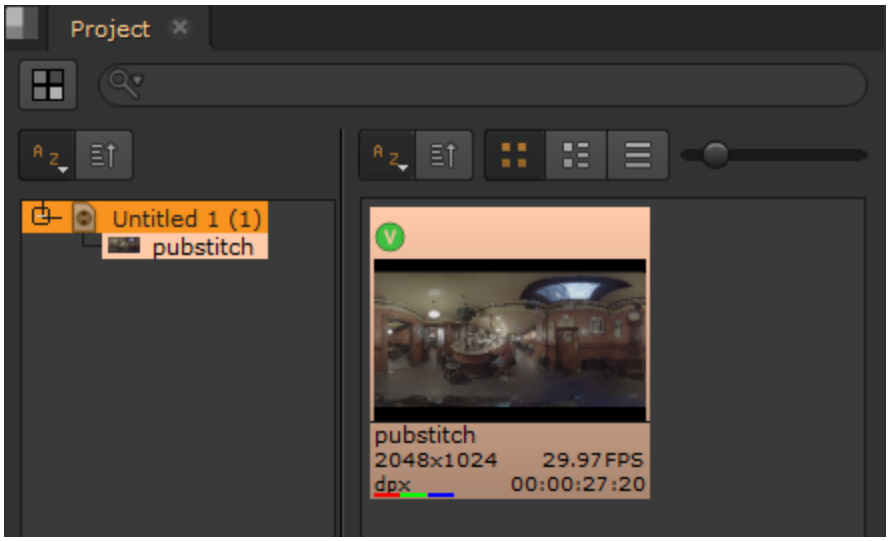
You can also use the **%V** and **%v** variables at a directory level. For example, if you have set up views called **cam3**, **left**, and **right**, and you have the following directories and files:

```
stereo/pubstitchcam3/pubstitch_cam3.####.dpx
stereo/pubstitchleft/pubstitch_left.####.dpx
stereo/pubstitchright/pubstitch_right.####.dpx
```

If you read in **pubstitch_cam3.####.dpx** and change the file path to **stereo/pubstitch%V/pubstitch_%V.####.dpx**, 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 V icon, which can display any of the combined views.



See Displaying Views in the Viewer and Displaying Views in the Timeline for more information.

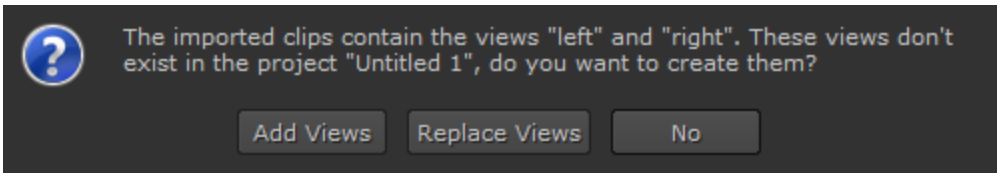
Importing Multi-View Clips

Nuke offers to automatically create views when you import single files containing multiple views, such as **.exr** and **.sxr**, 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 V 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.

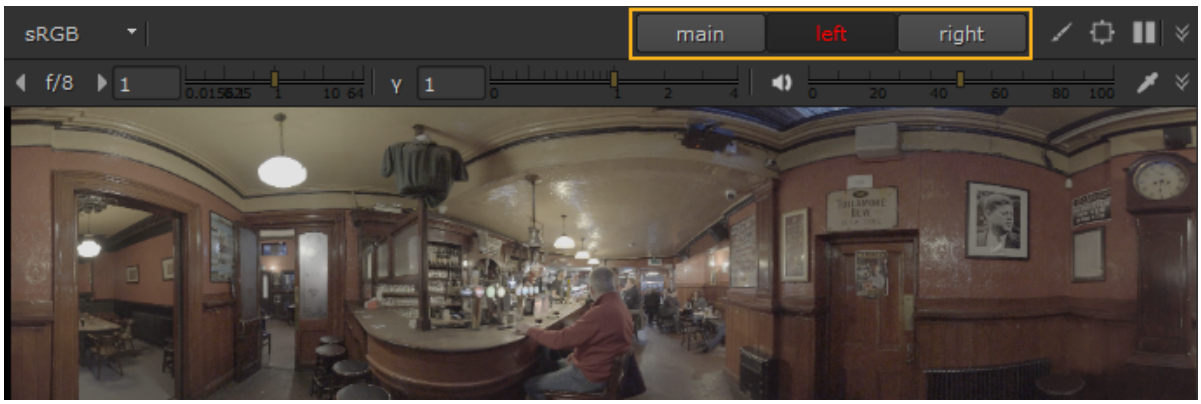
Displaying Views in the Viewer

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: Hiero 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:

- **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.

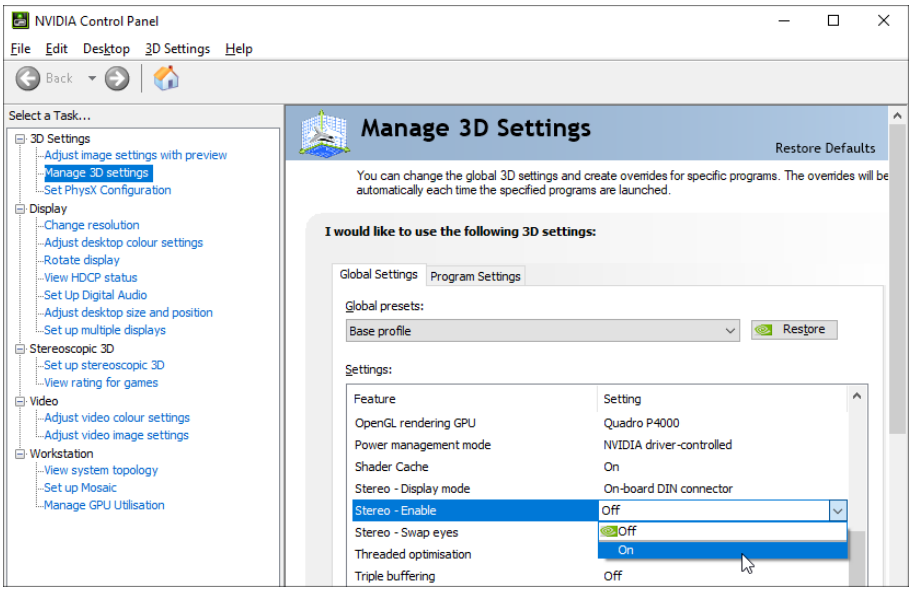
Note: OpenGL Stereo mode is not supported on Mac due to limitations in macOS and not supported with AMD GPUs on Linux.

Enabling OpenGL Stereo Output

Windows

To enable NVIDIA GPU stereo output:

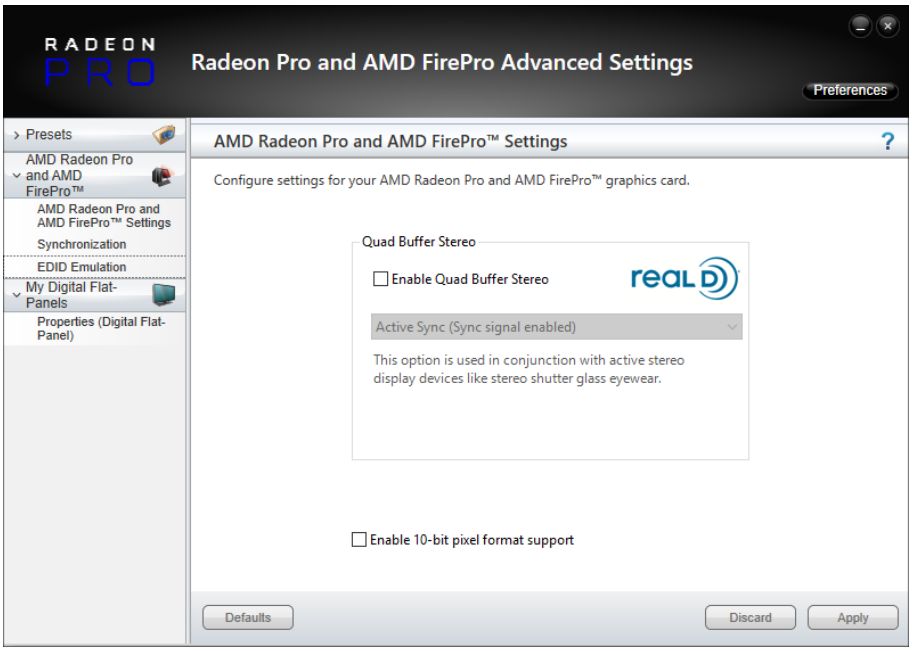
1. Right-click on the desktop and select NVIDIA Control Panel.
2. Navigate to **3D Settings** > **Manage 3D Settings** > **Stereo - Enable**.



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`

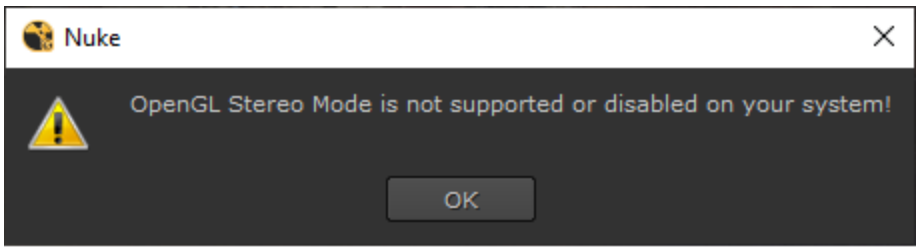
Tip: For more information on the **nvidia-xconfig** utility, please see the **man** page: `man nvidia-xconfig`

2. Proceed to Switching to OpenGL Stereo Output.

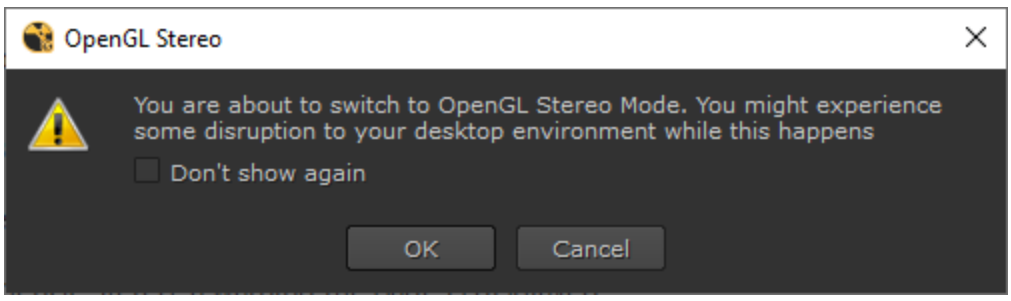
Note: **OpenGL Stereo** mode is not supported with AMD GPUs on Linux.

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**.

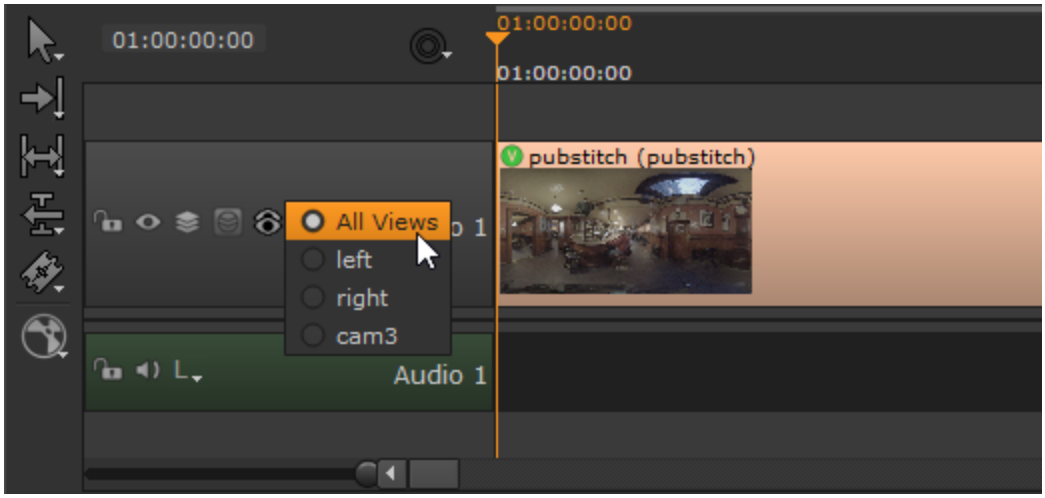
OpenGL Stereo is displayed in the Timeline Viewer.

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.



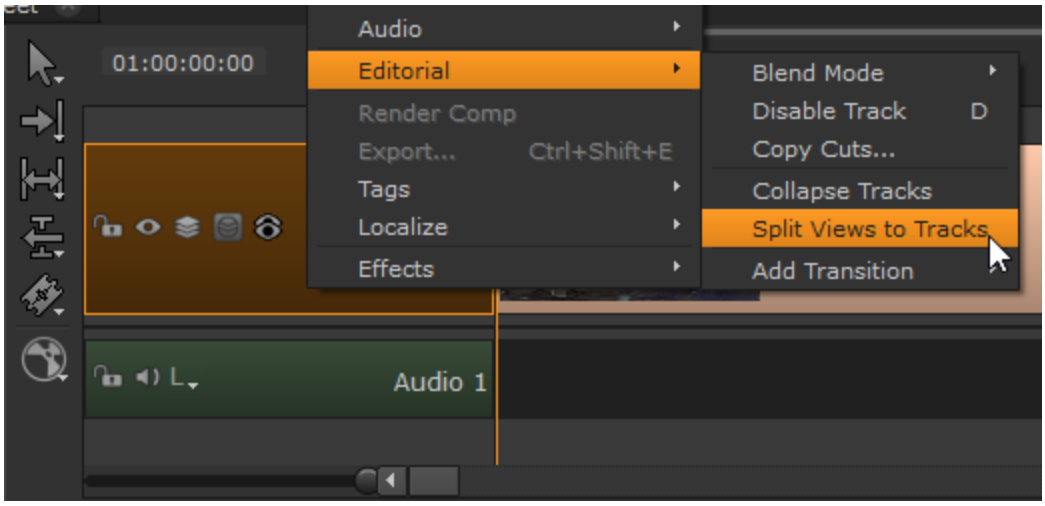
Selecting a particular view for a track in the timeline, such as **left**, means that only the left view is visible in the Viewer for that track.

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**.

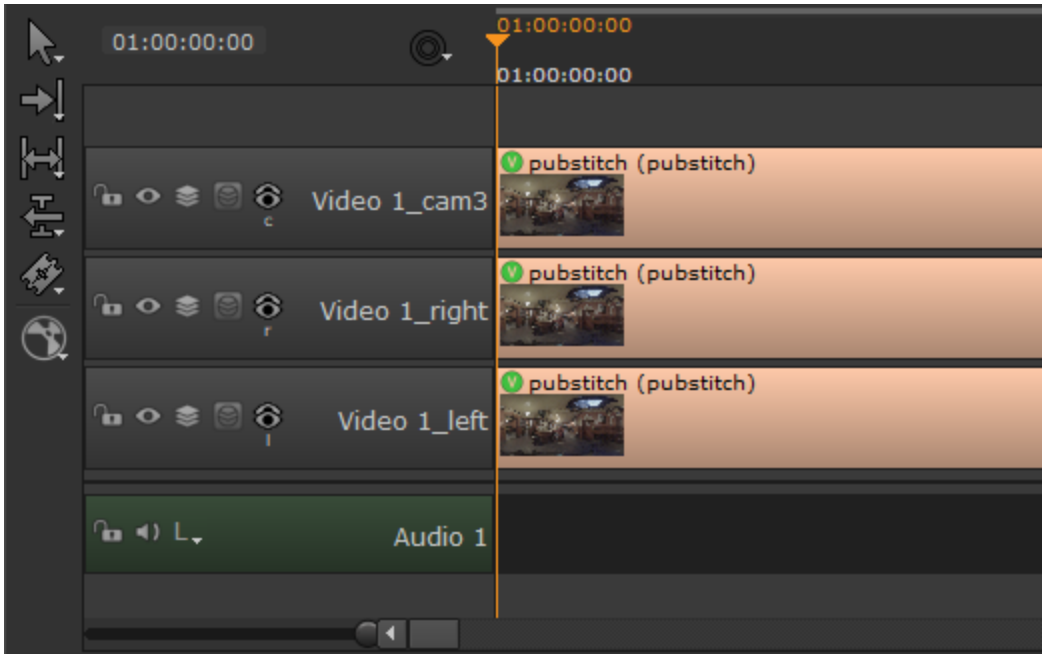
Splitting Views to Separate Tracks

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. Hiero 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**.



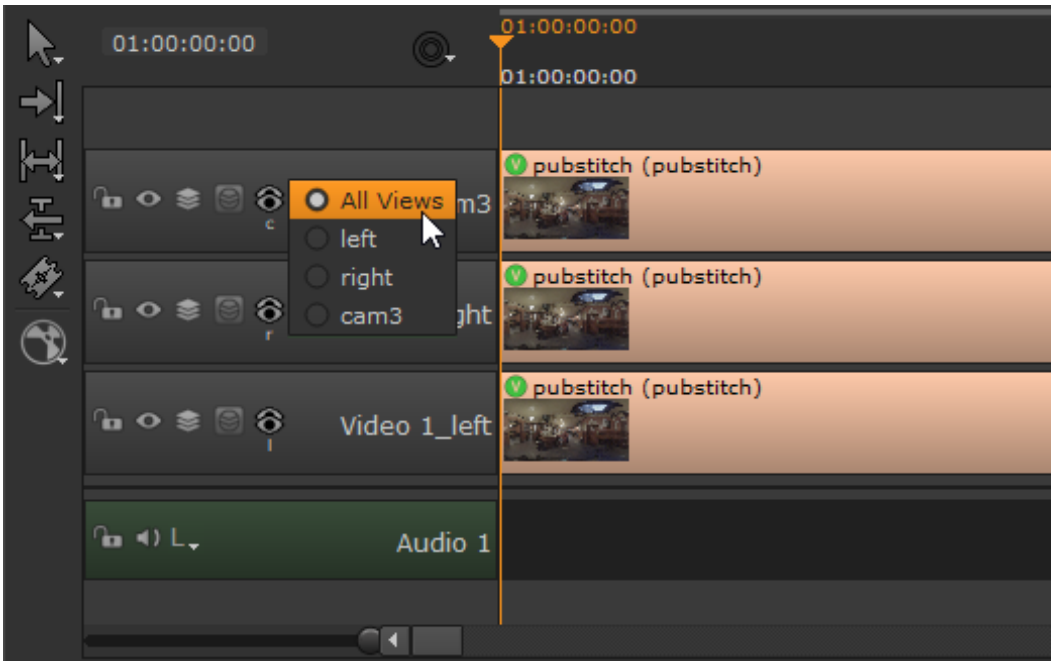
A separate track is created for each view in the group or file.



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.



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 on 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 on 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.

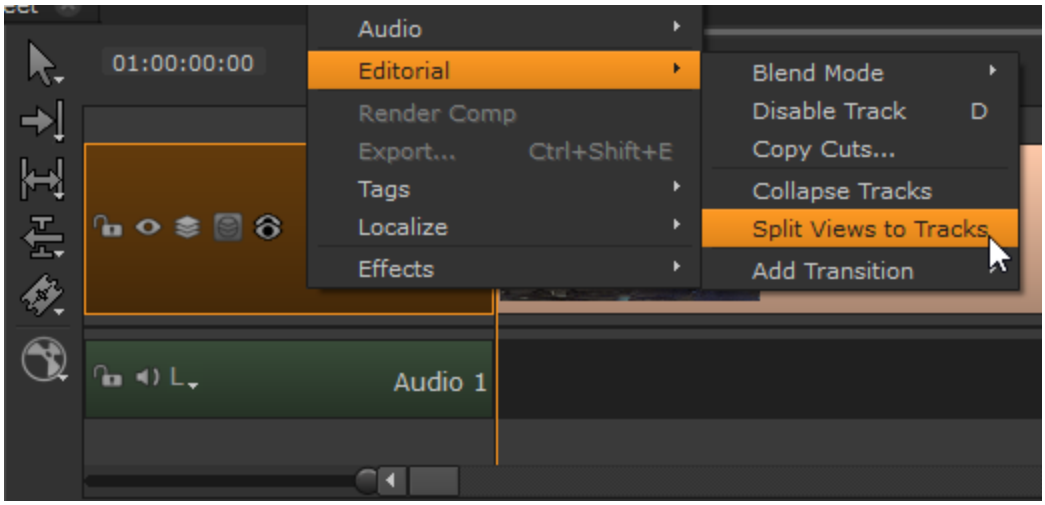
Applying Changes to Selected Views

By default, Hiero 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:

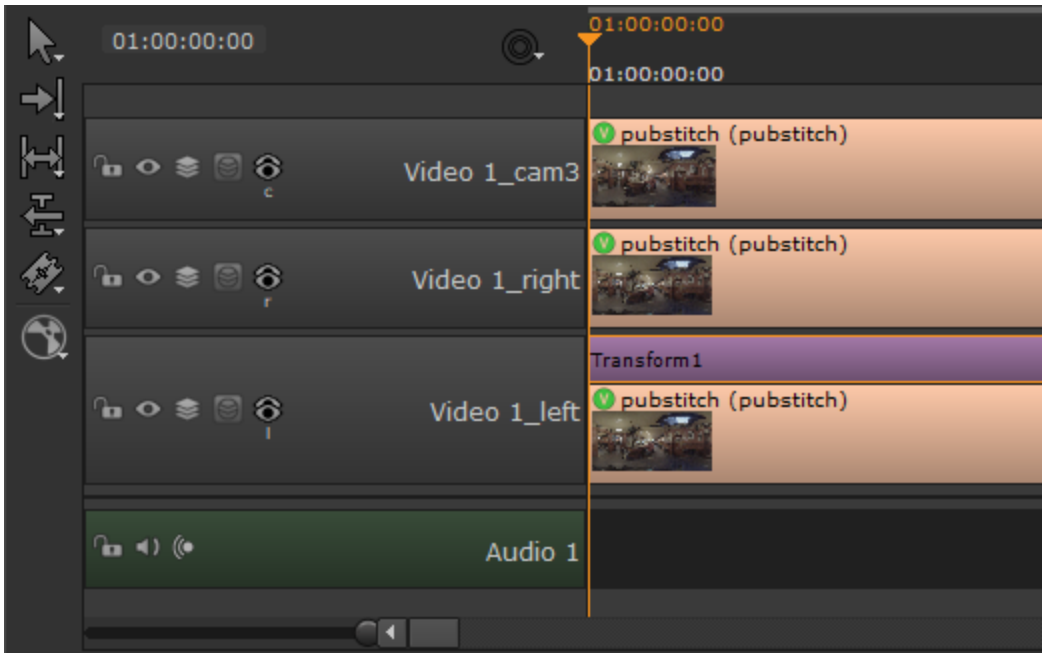
- Split the views into separate tracks. Separating views allows you to apply effects, such as a Transform or Timewarp, to individual views without affecting the other views. See Splitting Views to Tracks for more information.
- Split the view off in the soft effect's **Properties**. Splitting properties 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 Adding Effects on the Timeline for more details on adding effects to shots.

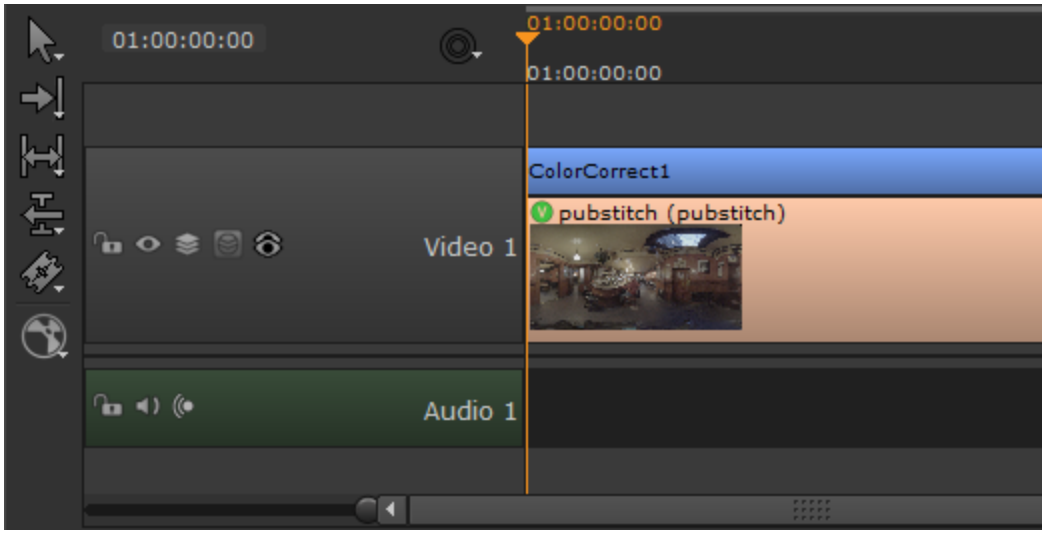


You can export multi-view shots and effects in a similar way to regular shots. See Exporting Multi-View Source Clips for more information.

Splitting Views in the Properties Panel

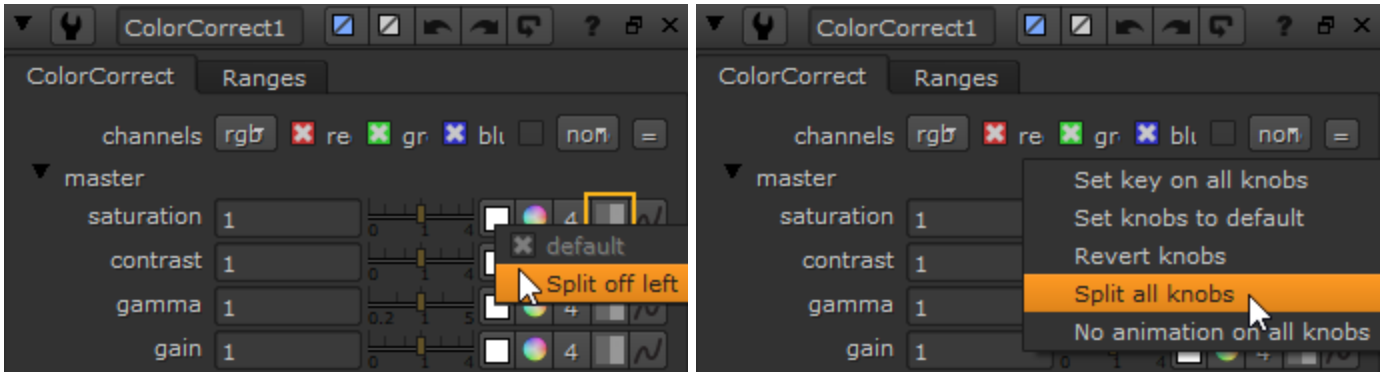
1. Right-click the track to which you want to add the soft effect, select **Effects** and then the required soft effect. For example, adding a ColorCorrect a multi-view shot.

   See Adding Effects on the Timeline 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.

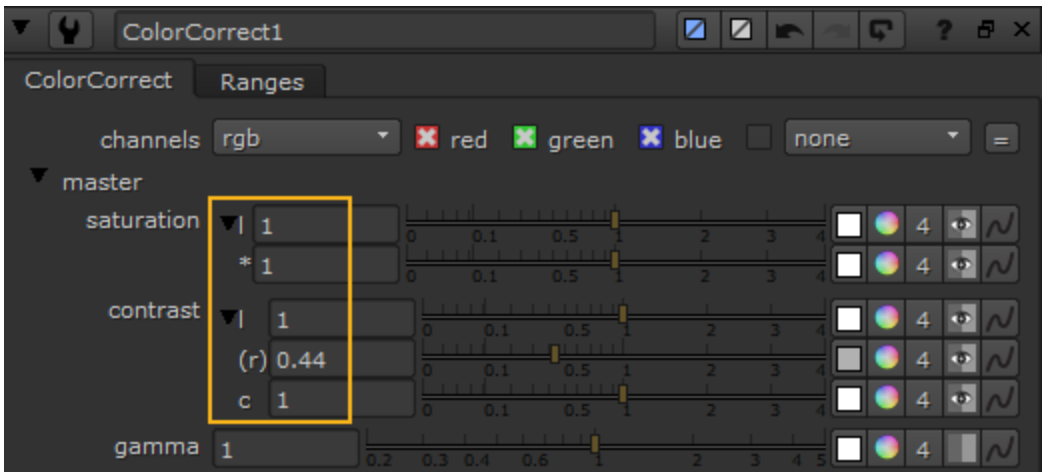


An eye appears on the split view button. Any changes you make using the control in question are only applied to the view you chose to split off. Changes to controls that have not been split off are still applied to all views.

You can 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.



In the example, the **left** view is split for the **saturation** control and **left** and **cam3** views are split for the **contrast** control.

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.

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.

Adding Effects on the Timeline

A soft effect is a real-time effect, such as Transform or TimeWarp, processed on GPU instead of CPU. You can add soft effects to your timeline in any of the workspaces.

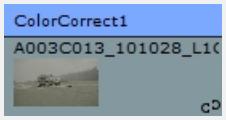
You can add custom plug-in or gizmo soft effects to the **Add Effect** menu using Python. Valid custom soft effects must have a GPUEngine implementation using **DD::Image::Iop::gpuEngine** type functions. For more information see Hiero's NDK Reference Guide.

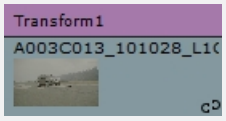
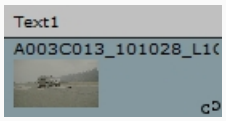
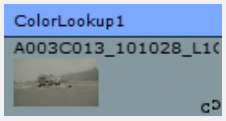
Soft Effects must also be registered after creation. An example of how to register a plug-in or gizmo as a custom soft effect is located in:

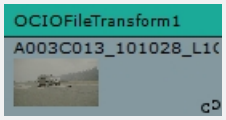
`<install_directory>/pythonextensions/site-packages/hiero/examples/custom_soft_effect.py`

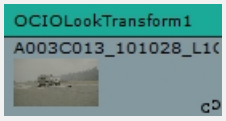
Available Soft Effects

Below is a brief summary of the available soft effects. These are similar to the tools in Hiero's Node Graph. See the Hiero Online Help for more information about them.

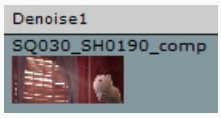
Soft Effect	Color Code	Summary
ColorCorrect		Make quick adjustments to saturation, contrast, gamma, gain, and offset. You can apply these to a clip's entire tonal range, shadows, midtones, or highlights. You can control the range of the image that is considered to be in the shadows, midtones, and highlights using the lookup curves on the **Ranges** tab. However, do not adjust the midtone curve - midtones are always equal to 1 minus the other two curves. Alt+C inserts a ColorCorrect soft effect on any shot selections or at the playhead

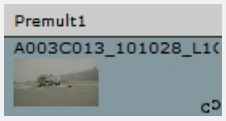
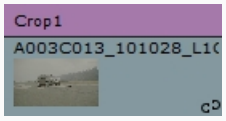
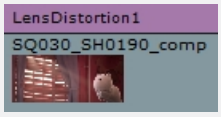
Soft Effect	Color Code	Summary
Grade		Define white and black points by sampling pixels from the Viewer. For example, you can use this for matching foreground plates to background plates. Alt+G inserts a Grade soft effect on any shot selections or at the playhead
Transform		Translate, rotate, scale, and skew shots from a single control panel. Alt+T inserts a Transform soft effect on any shot selections or at the playhead
Text		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.
TimeWarp		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. **Note:** TimeWarp effects are only allowed on tracks with clips as they are linked to the clips.
Color Submenu (OCIO soft effects are also available in HieroPlayer)		
ColorLookup		Make color adjustments using lookup tables (LUTs). LUTs refer to line graphs of a given color channel's brightness. The horizontal axis represents the original, or input, values and the vertical axis represents the new, or output values.

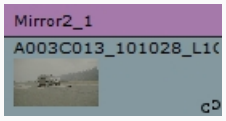
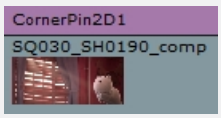
Soft Effect	Color Code	Summary
OCIO CDL Transform		Applies an ASC CDL (American Society of Cinematographers Color Decision List) grade based on the OpenColorIO Library. See OCIOCDLTransform in the Nuke Online Help for more information on the Properties panel for this effect.
OCIO ColorSpace		Converts images from the input colorspace of the shot on the timeline to an output colorspace more appropriate to a given process or intended display device.
OCIO Display		Applies a colorspace conversion to an image sequence so that it can be accurately represented on a specific display device. See OCIODisplay in the Nuke Online Help for more information on the Properties panel for this effect.
OCIO FileTransform (LUT)		Loads a colorspace conversion from a file (usually a 1D or 3D LUT) and applies it. You can also load other file-based transformations, for example an ASC ColorCorrection XML. File transforms are applied without any input or output colorspace handling. If the file expects log-encoded pixels, but you apply the node to a linear image, you will get incorrect results. See OCIOFileTransform in the Nuke Online Help for more information on the Properties panel for this effect.
OCIO LogConvert		Converts from compositing_log to scene_linear, or the reverse. Sometimes log data is required for certain processing operations, like plate resizing or pulling keys. Usually a colorspace is provided specifically for these operations.

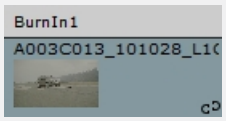
Soft Effect	Color Code	Summary
OCIO LookTransform		Applies a creative colorspace transform using a predefined look in the OCIO config file. A look is a named color transform intended to modify the 'look' of an image in a creative way, rather than a technically or mathematically defined colorspace conversion. See OCIOLookTransform in the Nuke Online Help for more information on the Properties panel for this effect.
OCIO NamedTransform		OCIONamedTransform uses NamedTransforms in the OCIO configs shipped with Nuke to emulate older methods of color management that ignored the RGB primaries and simply applied one-dimensional transformations. These NamedTransforms have been added to the config as sometimes it is helpful to include one or more transforms in a config that are essentially stand-alone transforms that do not have a fixed relationship to a reference space or a process space. An example would be a utility curve transform where the intent is to simply apply a LUT1D without any conversion to a reference space. See OCIONamedTransform in the Nuke Online Help for more information on the Properties panel for this effect.

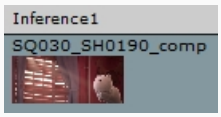
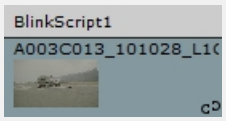
Tip: For more information on the OpenColorIO (OCIO) library, see https://opencolorio.org/

Denoise		The Denoise effect 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. See Removing Noise with Denoise for more information.

Soft Effect	Color Code	Summary
ChromaKeyer		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.
Merge Submenu		
Premult		Just like the Premult and Unpremult node in Nuke, these effects multiply or divide the shot's rgb channels by its alpha channel. If the alpha channel doesn't exist or covers the whole frame equally, premultiply and unpremultiply have no effect. See Premult and Unpremult in the Nuke Online Help for more information on the Properties panel for this effect.
Unpremult		
Transformations Submenu		
Crop		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.
LensDistortion		The LensDistortion effect estimates the lens distortion in a given image, either through Grid Detection or manual Line Detection. The warp can then be used to add or remove distortion or produce an STMap in the motion channel for use elsewhere. See Correct Shots Using LensDistortion for more information.

Soft Effect	Color Code	Summary
Mirror		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.
CornerPin		The CornerPin2D effect is designed to map the four corners of an image sequence to or from positions derived from tracking data. In practice, this effect lets you replace any four-cornered feature with another image sequence. You can use it to place an image in an on-screen television, for example. See Using the CornerPin2D Node for more information.
ModifyMetaData		Inject metadata key pairs into the shot during export in a similar way to the ModifyMetaData node in Nuke.

Burn-In		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.
		
		You can also reference custom metadata from shots. For example:
		`hiero/shot/tags/Approved`
		
		Extracts the **Approved** tag from the shot. You can also append **note** to include any notes associated with the tag:
		`hiero/shot/tags/Approved/note`
		
		Note: You must precede spaces and slashes in the tag name with \\ (backslashes) to enable Hiero to process the tag name correctly. For example:

Soft Effect	Color Code	Summary
		`hiero/shot/tags/Blue\\ Screen/note` You can also add burn-in through the **Export** dialog, see Adding Burn-in Text to Exports for more information.
Inference (Machine Learning)		The Inference effect applies a pre-trained .cat file to create the effect modeled by a neural network across the input image. You can use pre-trained .cat files from Nuke's CopyCat node, models from the Cattery, or third-party converted files from CatFileCreator can be used in the Inference effect. See Apply Pre-Trained Models to Shots Using Inference for more information.
BlinkScript		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 complied 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 Hiero, the BlinkScript soft effect only contains one input source and produces only one output. • The following functions are not supported by the BlinkScript effect if all the images have eAccessPoint data access:

Soft Effect	Color Code	Summary
		• log10 • round • rsqrt • abs for integer types • modf(a, *b) • sign • rcp • max, min, and clamp for integer types • median, atomicInc and atomicAdd. • The only data types supported by the BlinkScript effect are **int**, **float**, and **bool**. **Warning:** Soft effects using RIP will not work if there is a mismatch between your OpenCL GPU (blink device) and the OpenGL GPU. **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/15.0/BlinkKernels/

Adding Sequence-Level Soft Effects

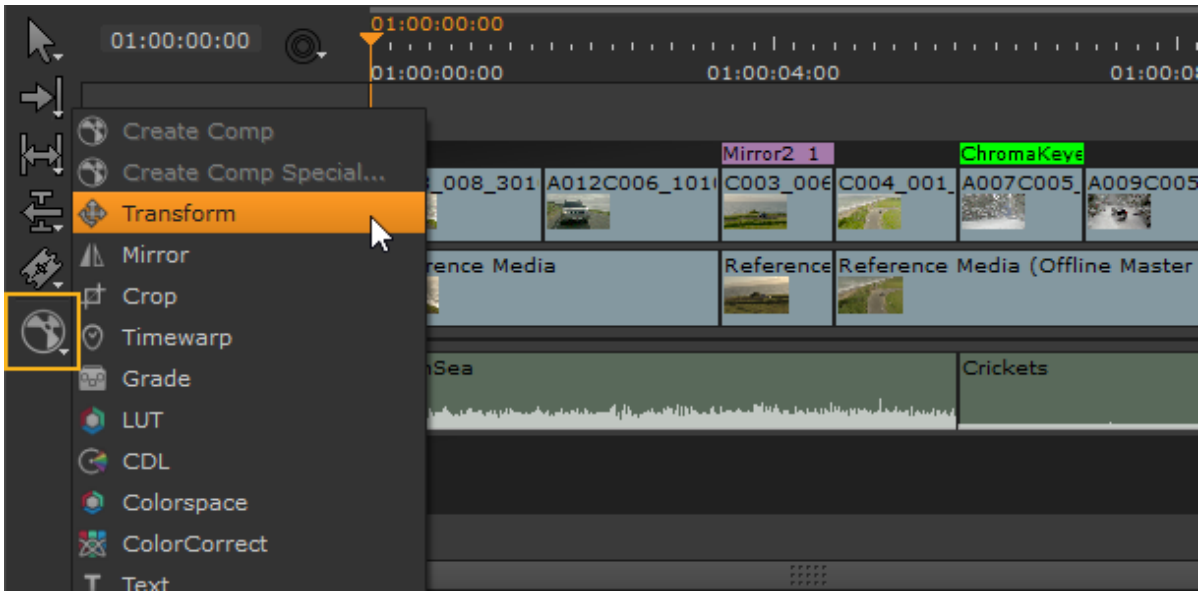
Warning: Please keep in mind the following:

• 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 **Soft 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.



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

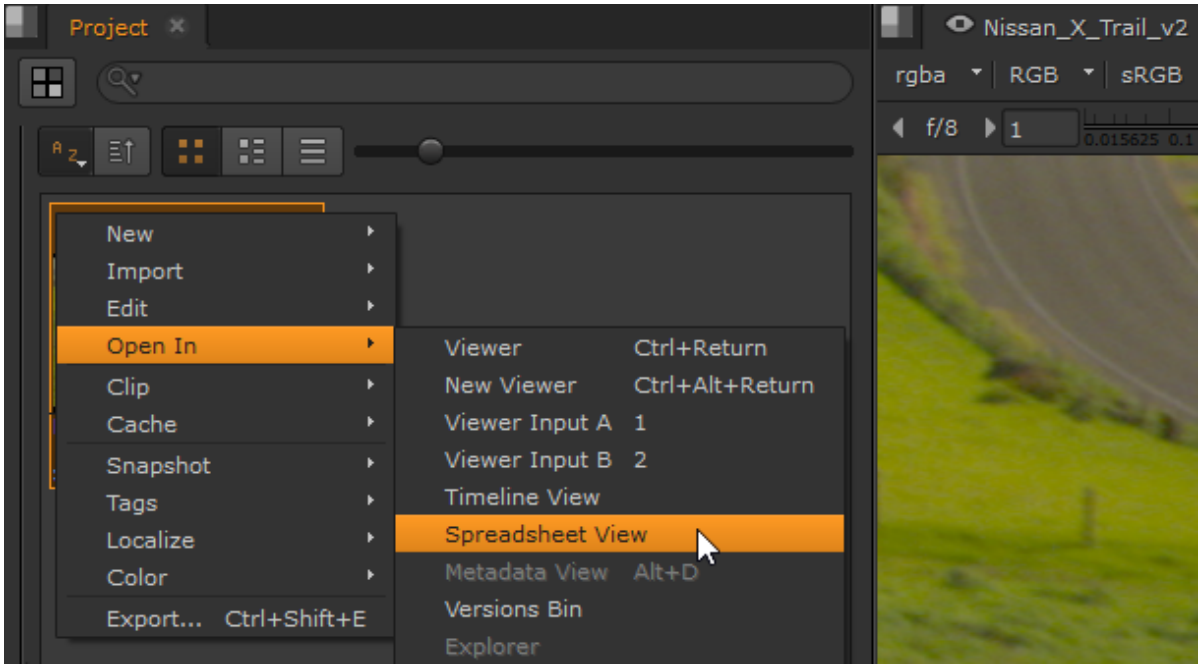
You can add soft effects using the spreadsheet view in any workspace by doing the following:

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**.

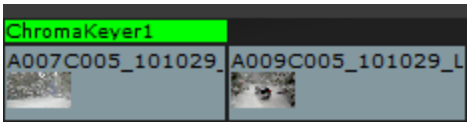


   This loads the sequence in the spreadsheet view that you previously opened.

4. Right-click an event from the list in the spreadsheet view and select **Soft Effects** to open a list of all available soft effects.

5. Select the required soft effect from the **Soft 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.



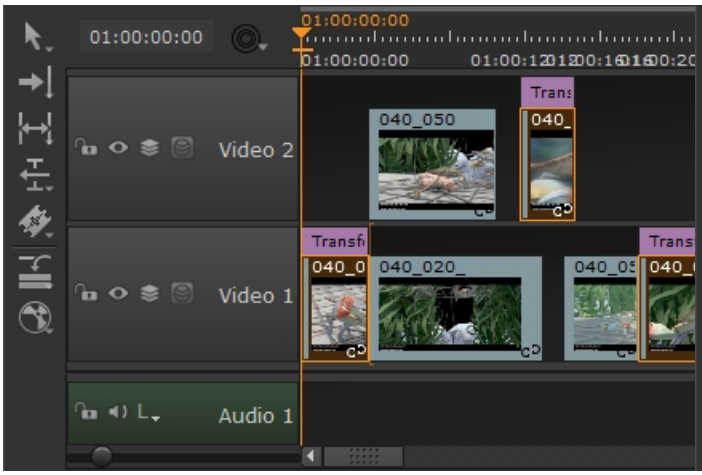
   When you insert a soft effect, its Properties open automatically. If you have it open, the effect properties panel displays in the **Properties** pane. If the **Properties** pane is not open, the effect's properties panel appears as a floating dialog.

Soft Effect Keyboard Shortcuts

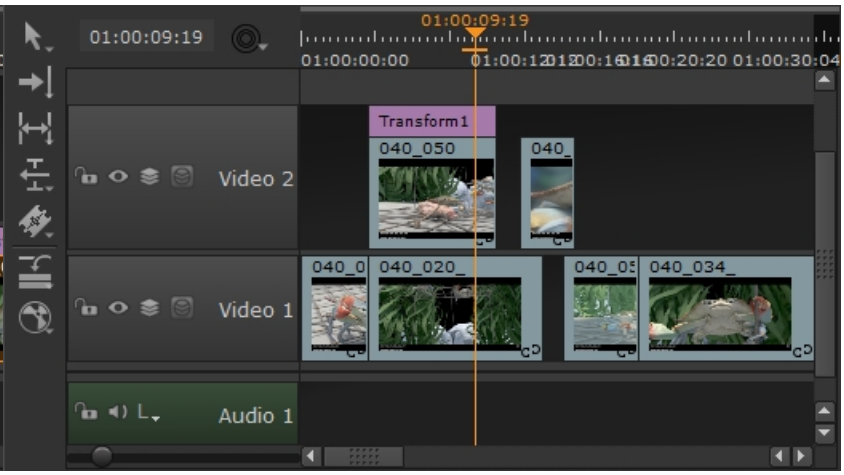
You can add Transform, Grade, and ColorCorrect effects using the keyboard shortcuts Alt+T, Alt+G, and Alt+C, just like the Node Graph equivalents in the composting environment. You can add effects

to multiple shot selections or at the playhead if no shots are selected.

Note: If no shots are selected, the effect is added to the top-most shot under the playhead.



Adding effects to multiple selections.



Adding effects at the playhead.

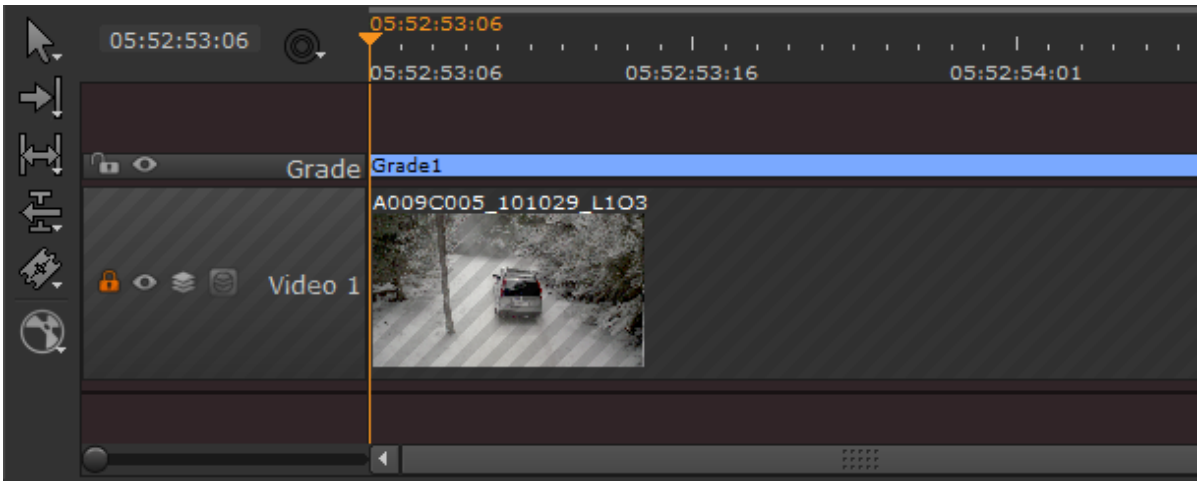
Adding Shot-Level Soft Effects

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.

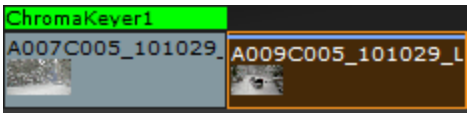
You can add shot-level soft effects on the timeline, by doing the following:

1. Right-click the shot that you want to add a soft effect to.
2. Select **Open In** > **Timeline View**.
3. Click the **Soft 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.

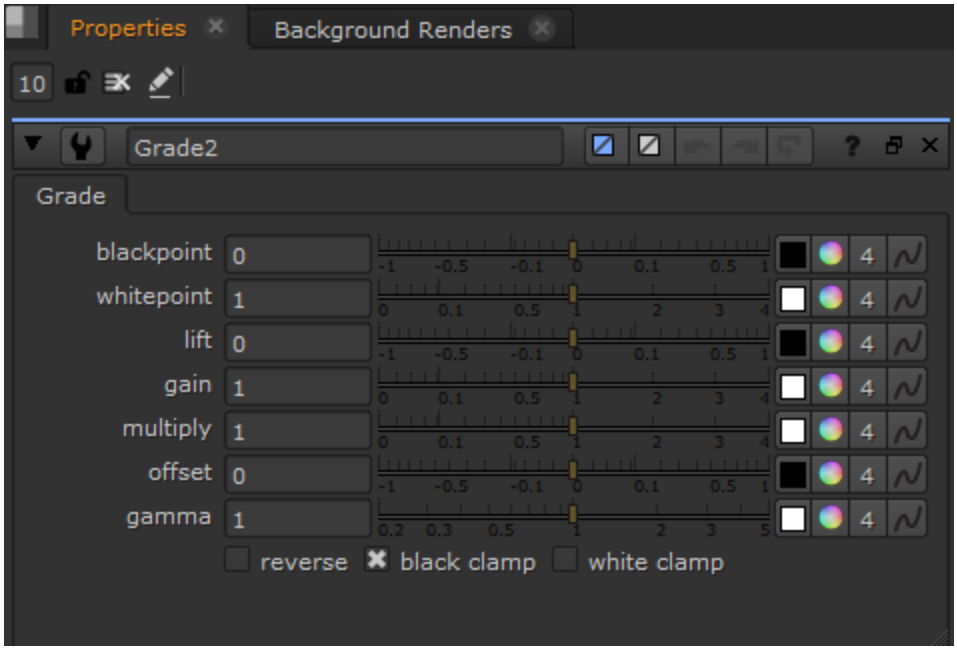


When you close the timeline view of the shot – as it is a shot-level soft effect – the soft effect is displayed as a colored line within the top of the shot. The color of the line displayed reflects the highest soft effect added to the shot.



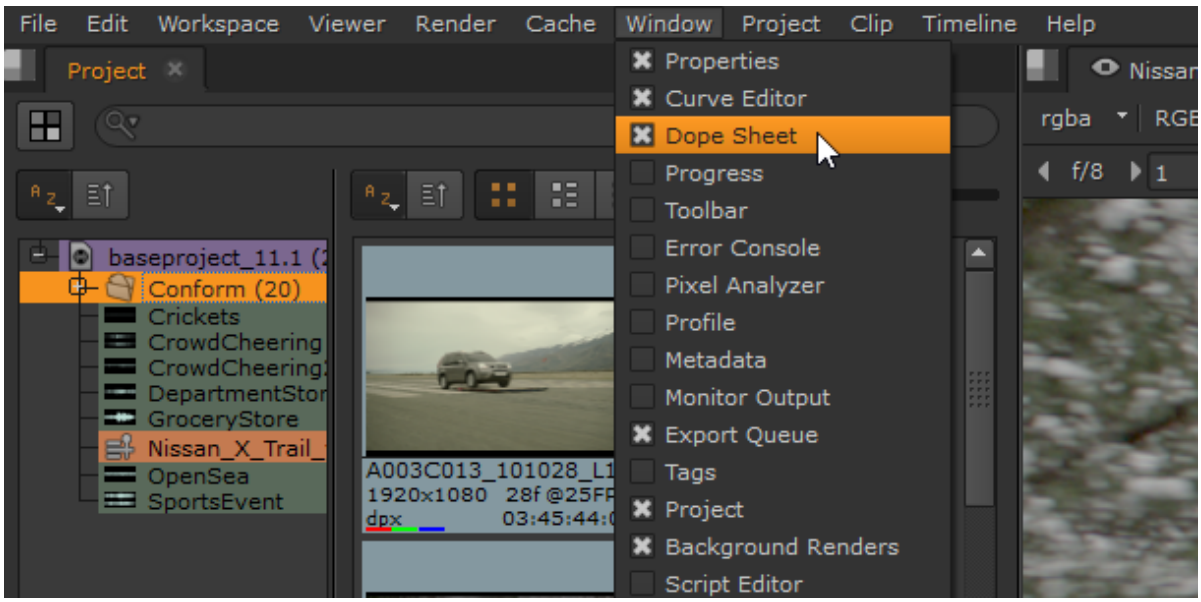
Soft Effect Controls

Adding an effect displays the associated controls in the **Properties** panel, similar to Hiero 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 the *Hiero User Guide* > *Using the Compositing Environment* > *Properties Panels* section, the *Hiero Reference Guide,* or the Online Help 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 the *Hiero User Guide Getting Started* > *Using the Compositing Environment* > *Animating Parameters* section or the Online Helpfor 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, different sequences, and 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

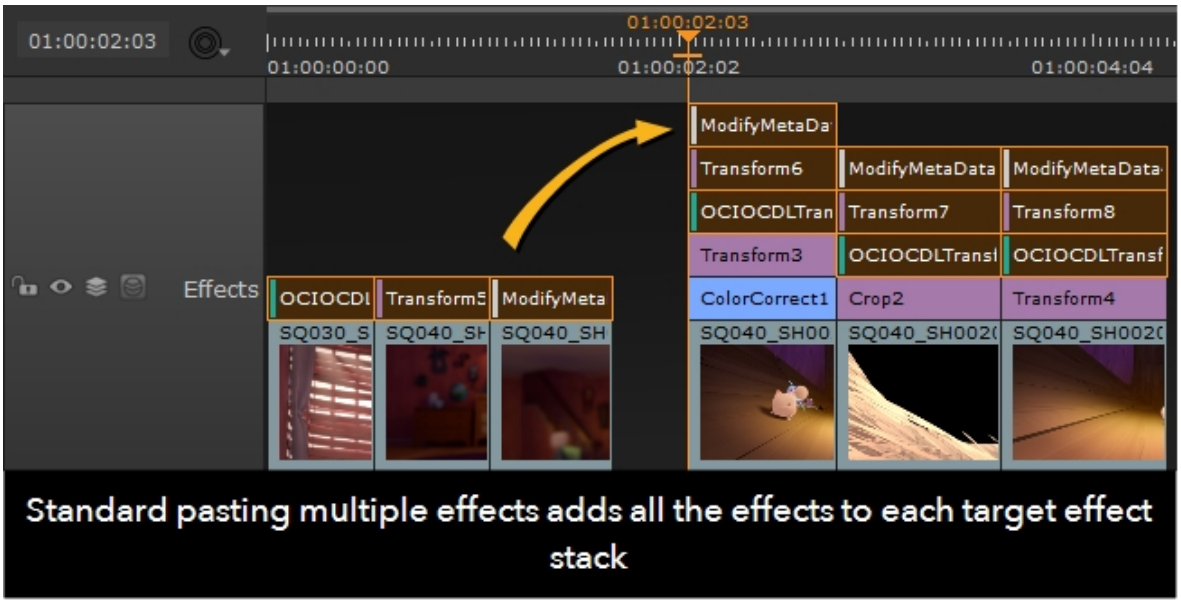
Nuke Studio and Hiero allow you to copy a sequence-level soft effect above the original to create a stack, to a different track, or to a different sequence. You can also copy a sequence-level soft effect to

a shot open in the timeline view, therefore pasting it as a shot-level soft effect. There are several copy operations available for soft effects, standard copy/paste, replace paste, sequential paste, and sequential replace paste.

Standard Copy and Paste

Standard copy and paste adds the copied effect(s) to the stack on the target shot(s). Effects are read from the top down, so pay attention to the order of the stack.

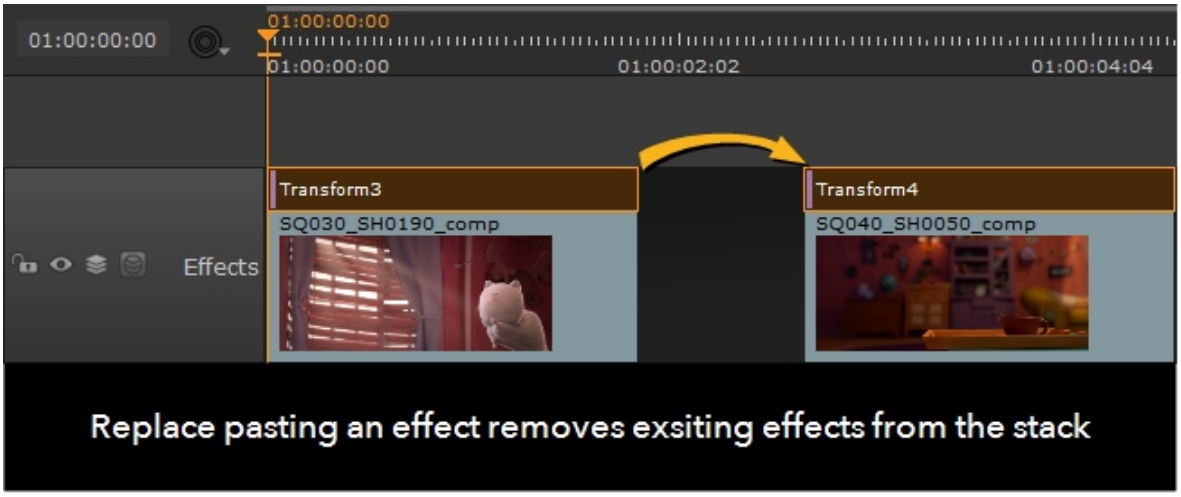
1. Select the soft effect you want to copy by clicking it.
2. Press **Ctrl**/**Cmd**+**C** or right-click and select **Edit** > **Copy**.
3. Select the target shot or move the playhead to where you want to paste the copy.
4. Press **Ctrl**/**Cmd**+**V** or right-click and select **Edit** > **Paste**.



Replace Paste

Replace paste copies the source effect(s) to the target shot and removes any existing effects on the shot.

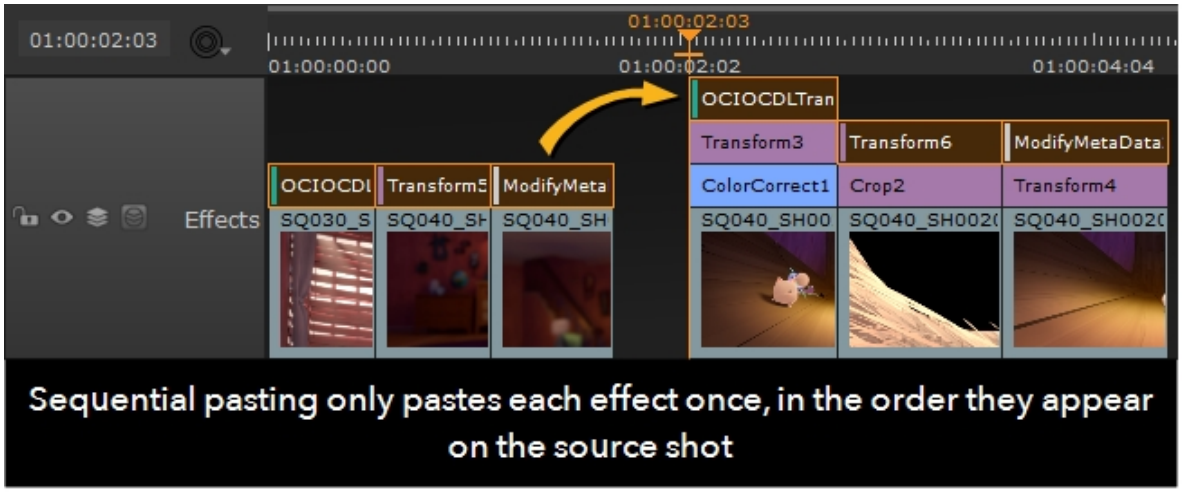
Press Shift+V or right-click and select Edit > Replace Paste.

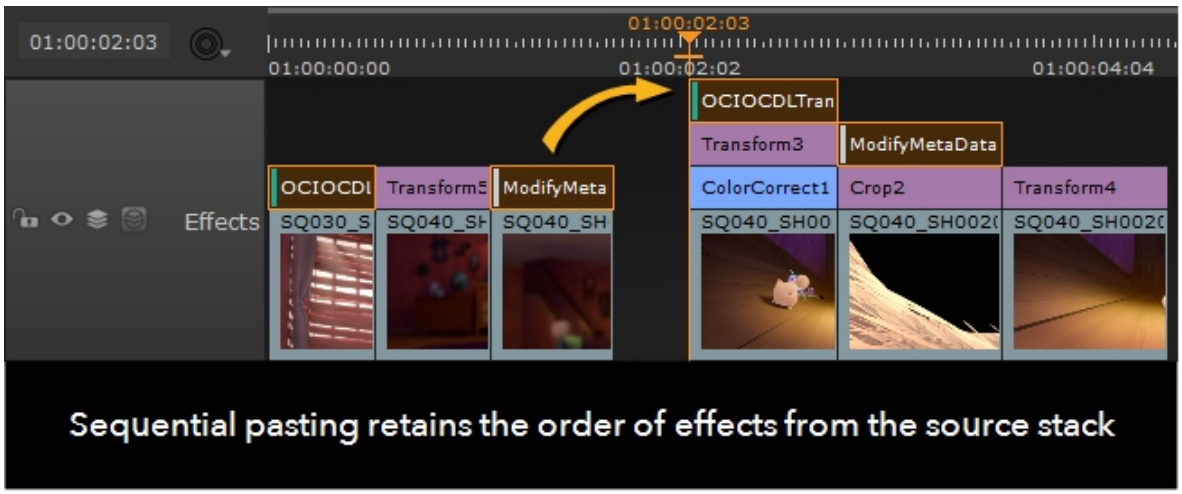


Sequential Paste

Unlike standard paste, sequential paste only pastes each effect once, in the order that they appear on the source shots.

Press Ctrl/Cmd+Alt+V or right-click and select Edit > Sequential Paste.





Sequential Replace Paste

Sequential replace paste only pastes each effect once, in the order that they appear on the source shots, removing any existing effects from the target shots.

Press Alt+Shift+V or right-click and select Edit > Sequential Replace Paste.

Cloning

You can 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. Hiero 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**.

Correct Shots Using LensDistortion

Nuke Studio and Hiero's soft effect collection also includes a LensDistortion effect in the timeline. Using LensDistortion in the timeline enables new workflows for shot prep and distribution and you don’t have to go back and forth between the timeline and comp environments. The results can be applied to multiple shots that use the same lens metadata or at sequence-level across an entire show.

Just like Nuke's LensDistortion node, the timeline effect estimates the lens distortion in a given shot, either through Grid Detection or manual Line Detection. The warp can then be used to add or remove distortion or produce an STMap in the motion channel for use elsewhere.

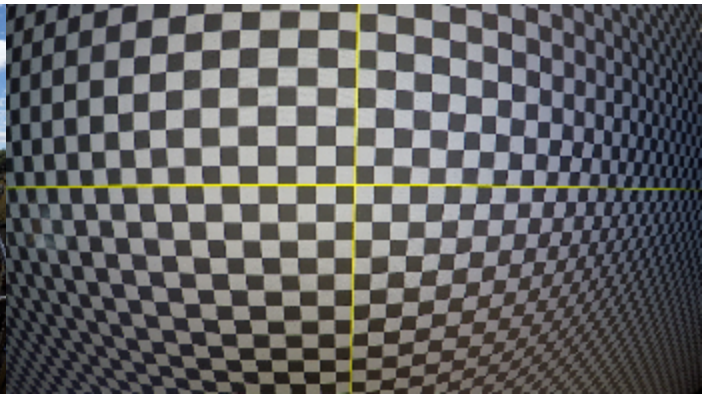
See Working with LensDistortion for more information.

Remove Lens Distortion from a Shot

Here's how to remove distortion from a shot using the LensDistortion effect on the timeline.

1. Either select the shot on the timeline manually or use the handy auto-select feature to choose a shot for you depending on the current playhead's position.
2. Click the soft effect menu icon  on the left-hand side of the timeline.
3. Select Transformations > LensDistortion to add the effect to the shot.
4. If you have a lens distortion grid shot with the same camera used for filming, you can use that to estimate the lens distortion applied to your shots,

The original distorted shot

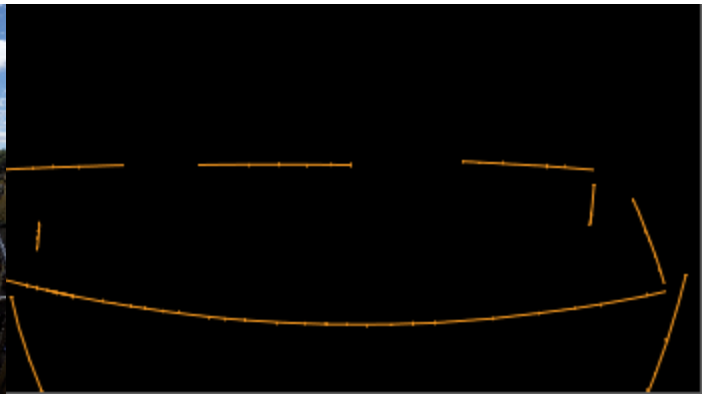


Associated grid shot

OR

If no estimation grid is available, you can use the LensDistortion effect's manual line drawing workflow to estimate lens distortion.

The original distorted shot

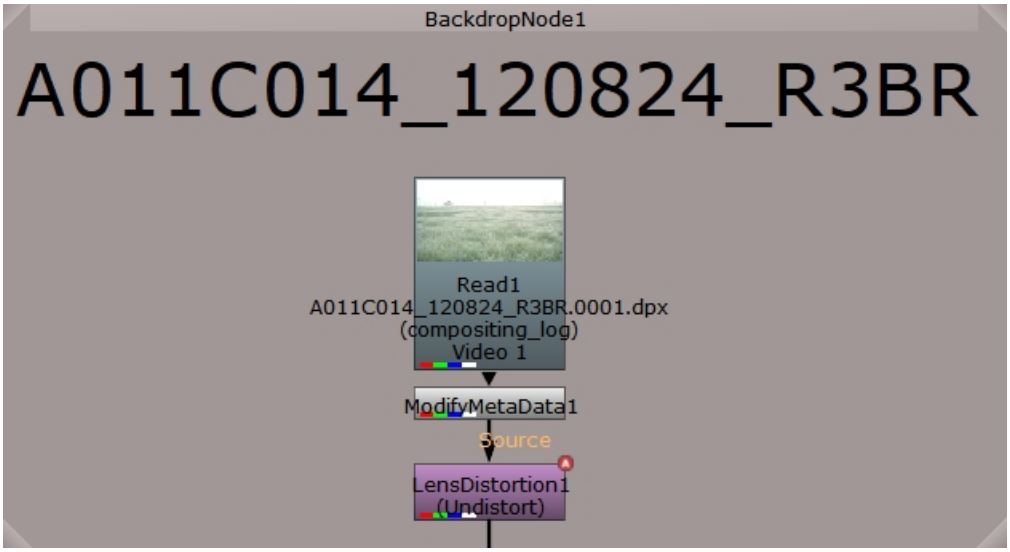


Example lines in the alpha channel

See Estimating Lens Distortion Using a Grid or Estimating and Removing Lens Distortion Using Lines for more information.

5. When you've calculated the required lens distortion, make sure that the LensDistortion effect's Mode is set to Undistort in the Properties panel.

   This step removes the calculated distortion from the shot so that you can send the undistorted plate to your comp department using Create Comp. Nuke Studio or Hiero automatically adds a LensDistortion node to the comp, as you can see in the example.

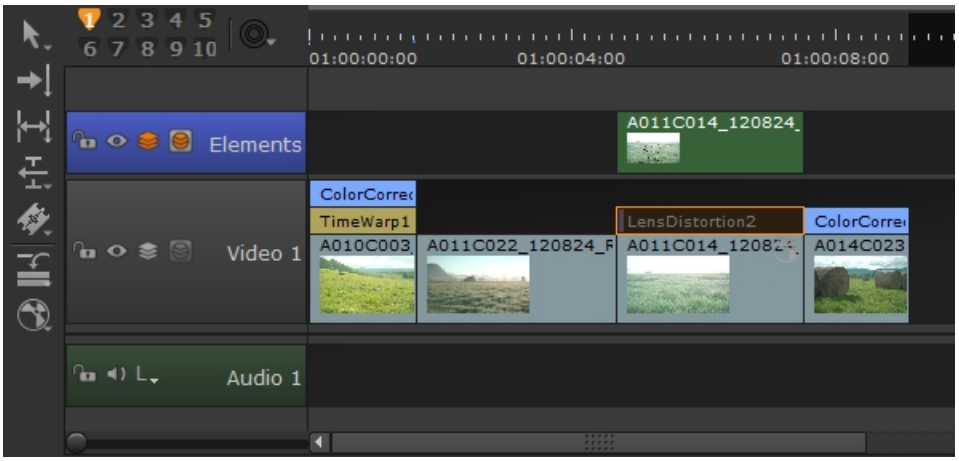


Your comp department can now add their work to the undistorted plate.

Add Lens Distortion to a Shot

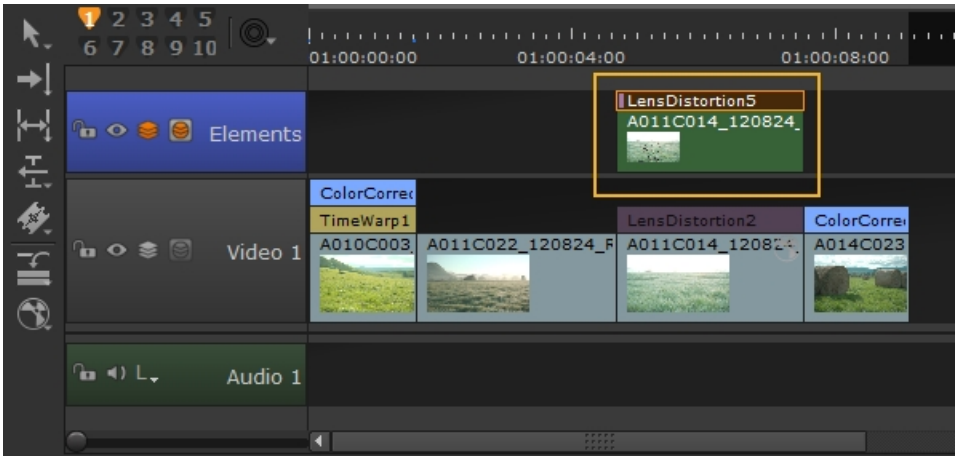
You can quickly add lens distortion to shots or whole sequences on the timeline by copying a LensDistortion effect set to Undistort and changing the Mode to Redistort. You might need to do this if your comp department didn't redistort the comp for you before sending it back from Nuke. In this example, we'll assume that our comp has come back from Nuke, but the CG elements are undistorted.

The shot on the Elements track is merged down onto the Video 1 track, but because the LensDistortion effect is disabled, the original plate is no longer undistorted.



To make the CG elements match the original plate's lens distortion:

1. Copy the LensDistortion effect from the Video 1 track and paste it on to the CG shot on the Elements track.



Note: The copy of the LensDistortion effect is not a clone, so the effect name is incremented as normal. In this case, to LensDistortion5.

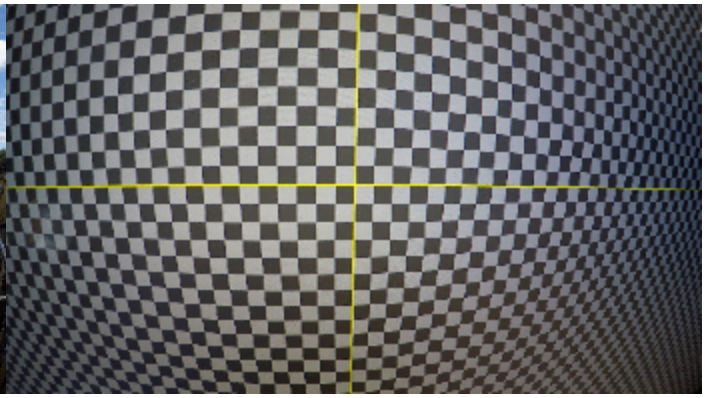
2. Double-click the pasted LensDistortion effect to display its Properties panel.
3. Click the Mode dropdown and select Redistort to invert the distortion applied by the original effect.

Remove Lens Distortion from Multiple Shots

If you intend to undistort multiple shots, it's a good idea to apply LensDistortion to one shot first and then clone the effect to the other shots in case you update the lens solve later on.

1. Either select the shot on the timeline manually or use the handy auto-select feature to choose a shot for you depending on the current playhead's position.
2. Click the soft effect menu icon  on the left-hand side of the timeline.
3. Select Transformations > LensDistortion to add the effect to the shot.
4. If you have a lens distortion grid shot with the same camera used for filming, you can use that to estimate the lens distortion applied to your shots,

The original distorted shot

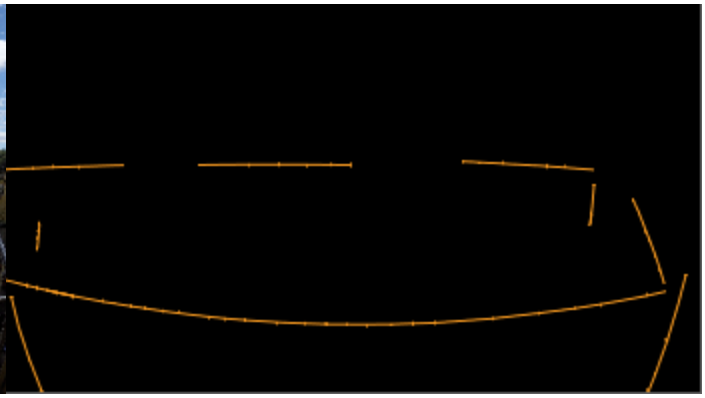


Associated grid shot

OR

If no estimation grid is available, you can use the LensDistortion effect's manual line drawing workflow to estimate lens distortion.

The original distorted shot

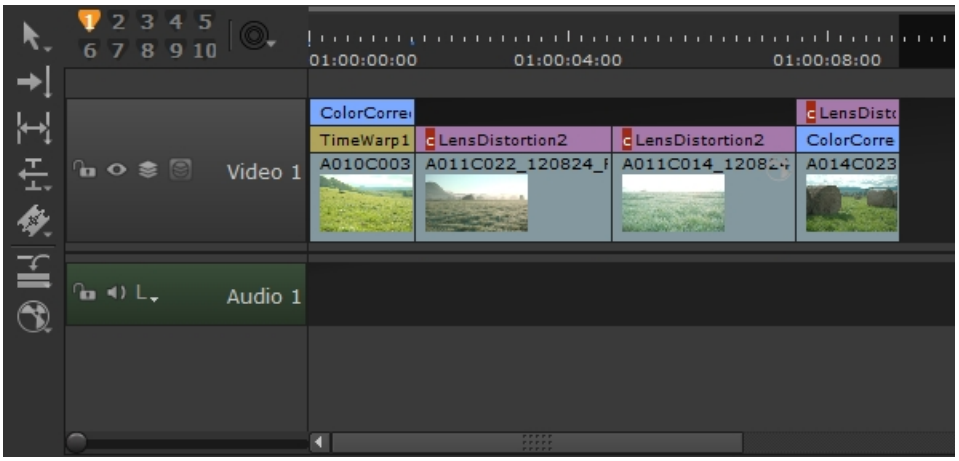


Example lines in the alpha channel

See Estimating Lens Distortion Using a Grid or Estimating and Removing Lens Distortion Using Lines for more information.

5. When you've calculated the required lens distortion, make sure that the LensDistortion effect's Mode is set to Undistort in the Properties panel.
6. Select the LensDistortion effect, right-click and choose Edit > Copy As Clones or press the Ctrl/Cmd+K keyboard shortcut.
7. Select the other shots in the timeline to which you want to add the cloned effect.
8. Right-click the selection and choose Edit > Paste or press the Ctrl/Cmd+V keyboard shortcut.

   The effect is cloned to the selected shots and the LensDistortion effect is applied. Cloned effects are clearly marked with a **C** and retain the same effect name, in this case LensDistortion2, so you can quickly locate them.



If you update the lens solve further to improve the results, simply update the clone source LensDistortion effect to apply it to all the clones on the timeline.

9. You can now use Create Comp to send all the affected shots to your comp department.

Apply Pre-Trained Models to Shots Using Inference

Nuke Studio and Hiero's soft effect collection also includes an Inference effect in the timeline. The Inference effect applies a pre-trained .cat file to create the effect modeled by a neural network across shots on the timeline. The Inference effect is designed to maximize artist efficiency by eliminating the need for round-tripping between the timeline and comp environments in order to see the results of CopyCat training on multiple shots. You can also convert your own models to .cat files using the CatFileCreator, or download open-source models from Cattery for use on the timeline

See Train Neural Networks to Replicate Effects Using Machine Learning and Import Pre-Trained PyTorch Models for more information on creating and sourcing .cat files for use in the Inference effect.

Warning: The timeline Inference effect can apply models trained on RGB input images only. .cat files requiring additional input channels (motion, position etc) are not supported.

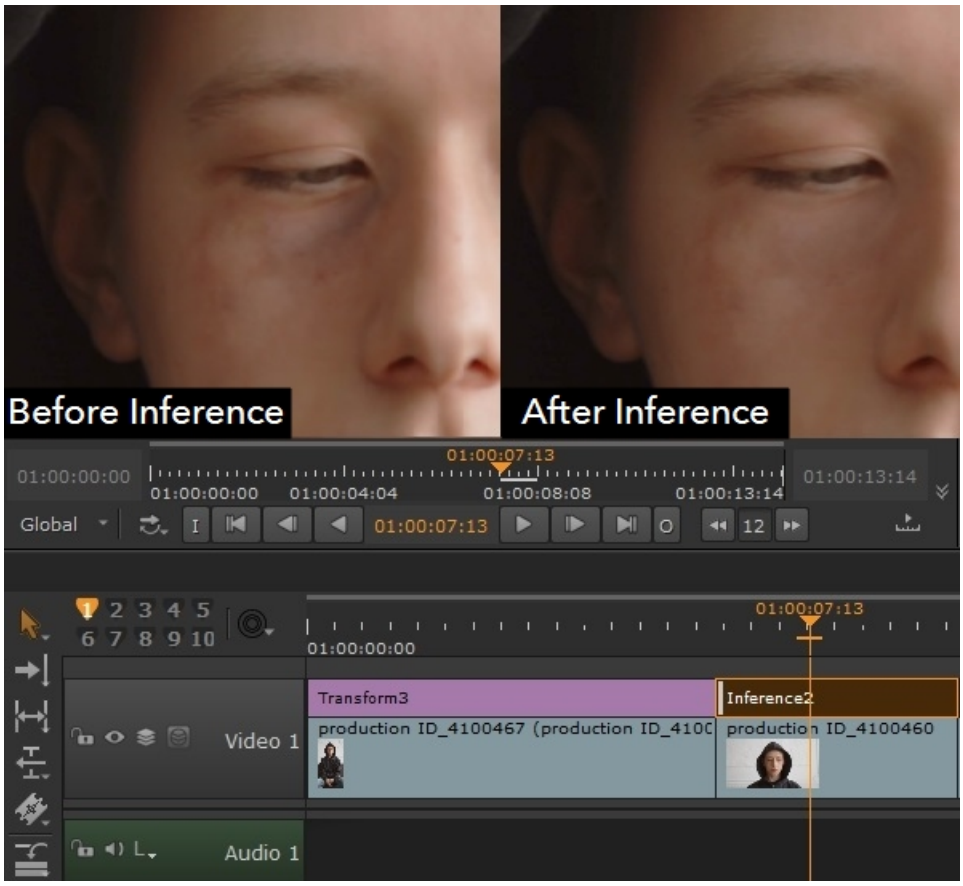
Add an Inference Effect to a Shot

So you've trained your model and it's ready to go. In this example, we've trained a model to remove a bruise from an actor's face. Here's how to apply the model to a shot on the timeline.

1. Either select the shot on the timeline manually or use the handy auto-select feature to choose a shot for you depending on the current playhead's position.
2. Click the soft effect menu icon  on the left-hand side of the timeline.
3. Select Inference to add the effect to the shot.

   An error is displayed in the Viewer because no .cat file is selected by default.
4. In the Properties panel, enter a Model File path to the .cat file you want to apply to the shot.

   The model is applied to the shot and you'll see the result in the Viewer.



Note: Depending on the model size and complexity, there may be a delay while the effect is applied to the shot.

5. You can enable Optimize for Speed and Memory if you want a faster preview of the results. The Properties panel also displays the Channels In and Channels Out for reference.

Tip: You can also apply Inference soft effects to an entire sequence by creating a new track and adding the effect to the track, instead of individual shots. See Adding Sequence-Level Soft Effects for more information.

Apply an Inference Effect to Multiple Shots

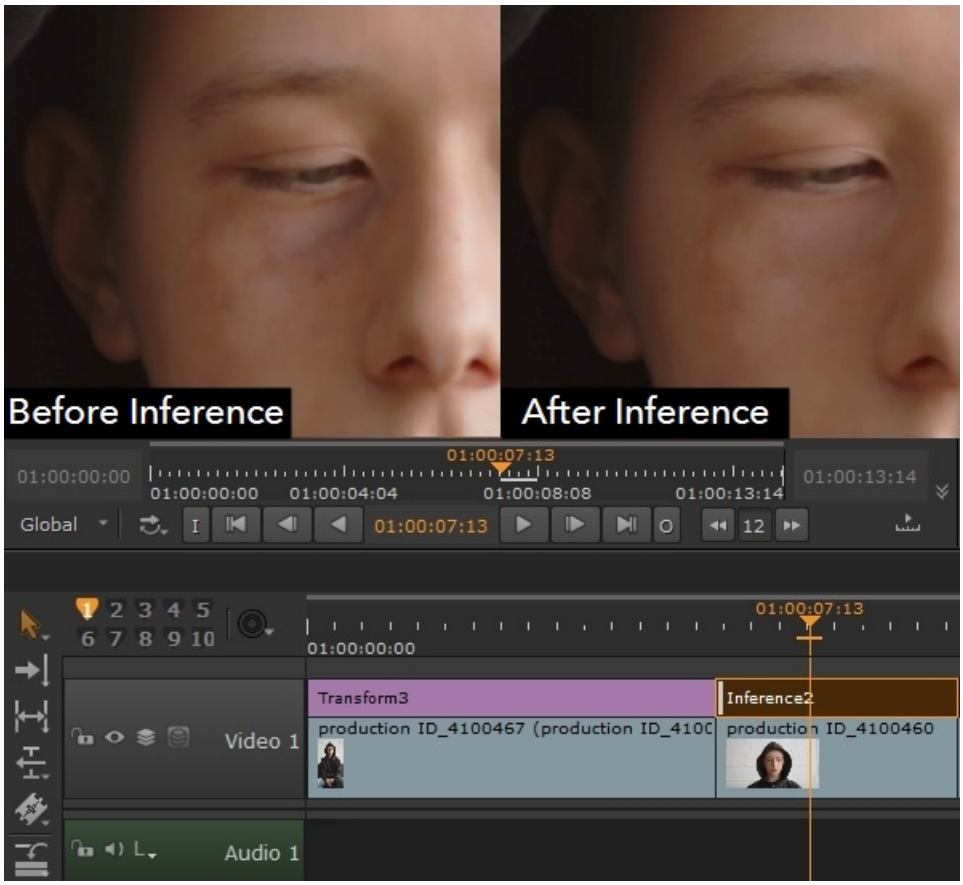
You can easily test how well the Inference effect works on other shots by simply copying and pasting the effect onto those shots. However, once you intend to use an Inference effect on multiple shots, it's a good idea to apply it to one shot first and then clone the effect to the other shots in case you update

the .cat file later on with further training. Another useful method is to apply the soft effect to a track above the clip, then extend it to cover as many clips on the tracks below as needed. You can also cut the soft effect into pieces for more control over which clips it applies to.

1. Select the shot you want to use as the source of the cloned Inference effect.
2. Click the soft effect menu icon on the left-hand side of the timeline.
3. Select Inference to add the effect to the shot.

   An error is displayed in the Viewer because no .cat file is selected by default.
4. In the Properties panel, enter a Model File path to the .cat file you want to apply to the shot.

   The model is applied to the shot and you'll see the result in the Viewer.

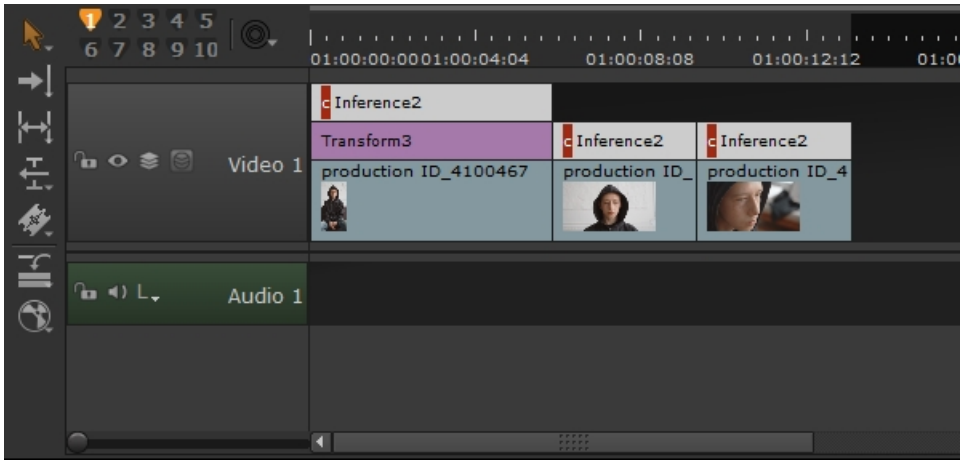


Note: Depending on the model size and complexity, there may be a delay while the effect is applied to the shot.

5. Select the Inference effect, right-click and choose Edit > Copy As Clones or press the Ctrl/Cmd+K keyboard shortcut.
6. Select the other shots in the timeline to which you want to add the cloned effect.

7. Right-click the selection and choose Edit > Paste or press the Ctrl/Cmd+V keyboard shortcut.

   The effect is cloned to the selected shots and the Inference effect is applied. Cloned effects are clearly marked with a **C** and retain the same effect name, in this case Inference2, so you can quickly locate them.



   If you train the model further to improve the results, simply update the clone source Model File control with the new .cat file to apply it to all the clones on the timeline.

Annotations

Annotations can be used as quick instructions for a compositor to implement in Nuke'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 Hiero's 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 in an external Nuke application.

   See Viewing Annotations in Nuke for more information.
6. After the suggested changes are made in the in the external Hiero, 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.

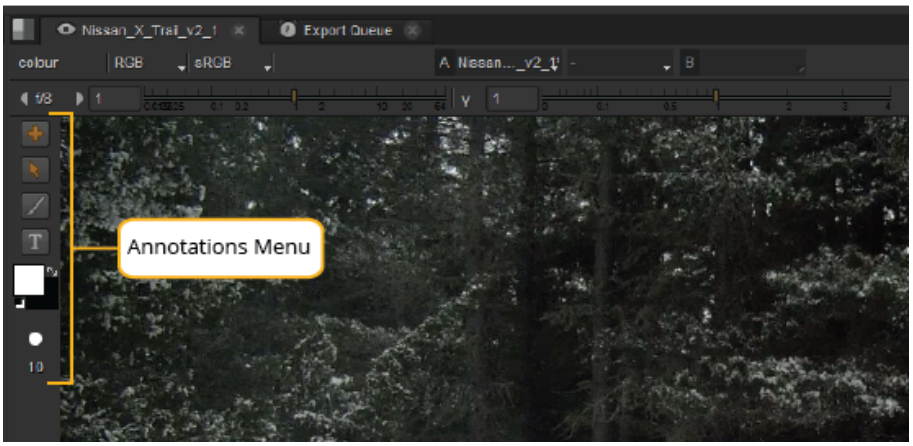
9. After adding the new annotations, right-click the rendered Nuke Comp and select **Export** > **Re-Export Annotations**.
10. Open the rendered Nuke Comp in an external Nuke application.
11. Double-click the Precomp node to open its properties and version it up. For example, if the file path has **v01.nk** 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.



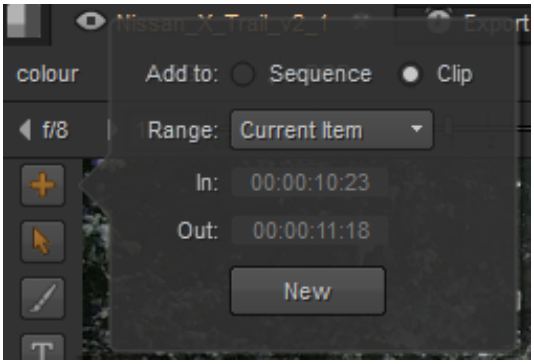
Note: Annotations on sequences and shots are only visible when you have the Annotations menu open.

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.

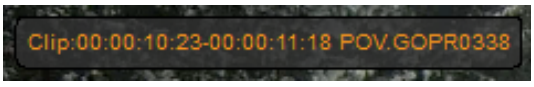
   When you add an annotation to the clip level, a turquoise line is displayed within the top of the shot. When you apply an annotation to the sequence level, a turquoise box appears above the selected section on a separate **Annotations** track.

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.

Note: The timecode displayed in the In and Out fields is derived from the clip's metadata, not its position in the sequence.

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  .

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.
3. Select **Edit** > **Paste** (or press **Ctrl**/**Cmd**+**V**).

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.

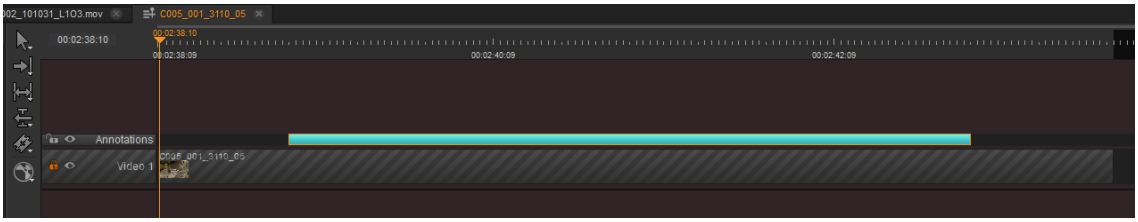
Editing Shot-Level Annotations

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 Nuke

If you want to view annotations in an external Hiero application, 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 using **Create Comp Clips** during round-tripping under Building VFX Tracks and Comp Clips for more information.

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 Export.

   The **Export** 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 **Export**.

   A message warns that you've changed the export templates and asks whether you want to keep them.
4. Click **Yes** or **No**.

   See Building VFX Tracks and Comp Clipsfor more information on bringing Nuke Comps back into Hiero.

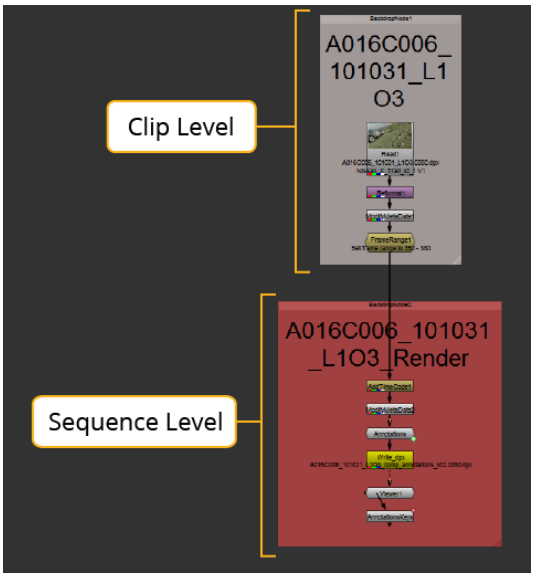
Viewing the Annotation Node Group

If you wish, you can view the annotations in an external Hiero 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 Hiero.
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

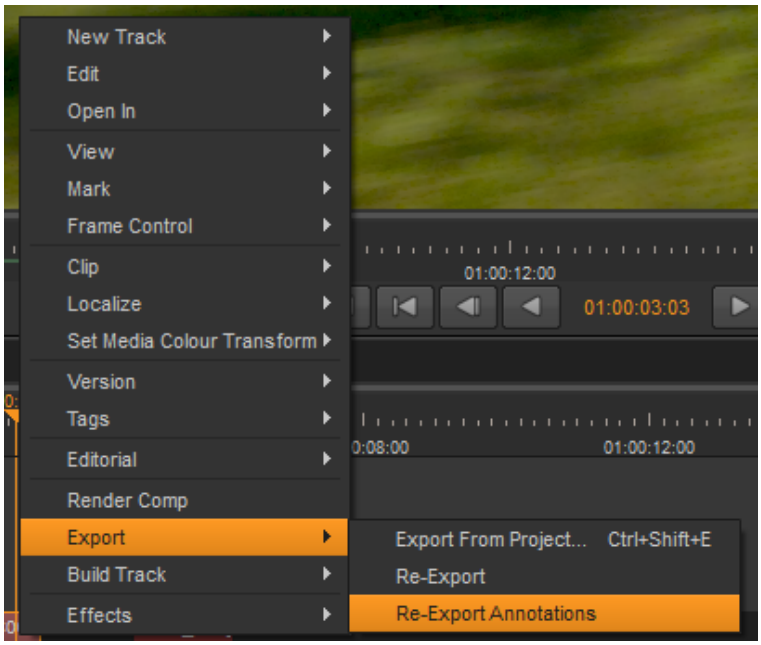
You may want to add new annotations after the Nuke Comp has been edited and rendered. In this case, you can re-export the annotations only.

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 Hiero 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.

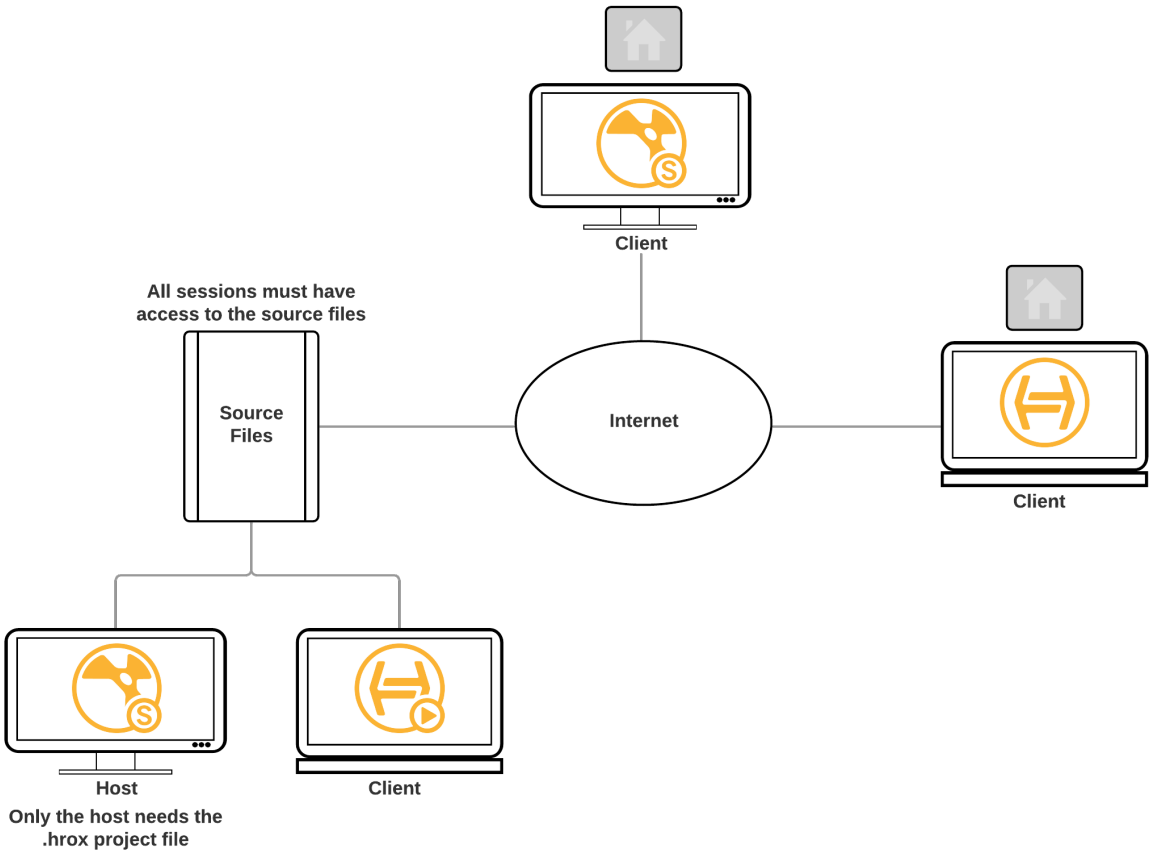
   Your new annotation is now visible in the Viewer.

Sync Review

Sync Review allows multiple participants to collaborate on a single project in Nuke Studio, Hiero, or HieroPlayer. The host allows multiple clients to join the review session and all participants have access to the playback controls, annotations, and versioning system.

Warning: All participants must use the same version of the application to run a sync session. For example, if the host is using 15.0v3, clients can only connect to the session if they are using 15.0v3 as well.

Sync Review works by connecting clients to the host using the host's IP address or hostname and an open port number. The project **.hrox** file only needs to exist on the host machine, but the source clips referenced by the shots on the timeline must be available for all participants either locally or from a central server.



Hosting a Sync Review Session

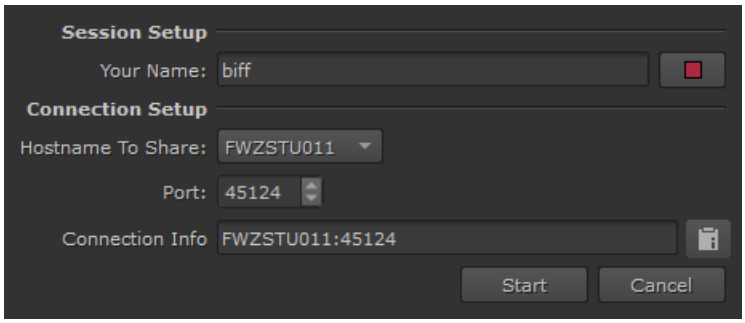
Nuke Studio, Hiero, and HieroPlayer can all act as host in Sync Review sessions, though you only need one host to connect multiple clients.

To start a host session:

1. Go to **File** > **Open Project** to load the project you want to review.

> **Note:** Only the host loads the project, client sessions don't require the **.hrox** file.

2. Go to **Workspace** > **Sync Session** to load a dedicated review workspace,
   OR
   Go to **Window** > **Sync Session** to open the **Sync Session** panel.
3. Click **Host**.
   The **Session Setup** dialog is displayed.



4. Enter the name you want to display on participant sessions and pick a cursor color by clicking on the color swatch.

> **Tip:** You can set your name and cursor color for new sessions in the **Preferences** under **Project Defaults** > **Sync Review**.

5. Select the host name and port to which you want the client sessions to connect.
   Your selection is displayed in the **Connection Info** field.

> **Tip:** Click the copy to clipboard button to automatically copy the connection details so that you can pass them on to participants.

6. Click **Start** to begin hosting.
   The **Sync Session** panel is populated with your details automatically.
7. Send the connection details to the participants that intend to join the review session.

Tip: If you didn't copy them to the clipboard earlier, click **Sync Panel** > **Copy Info**.

Connecting to a Sync Review Session

Nuke Studio, Hiero, and HieroPlayer can all connect as clients in Sync Review sessions, all you need is the connection information from the host.

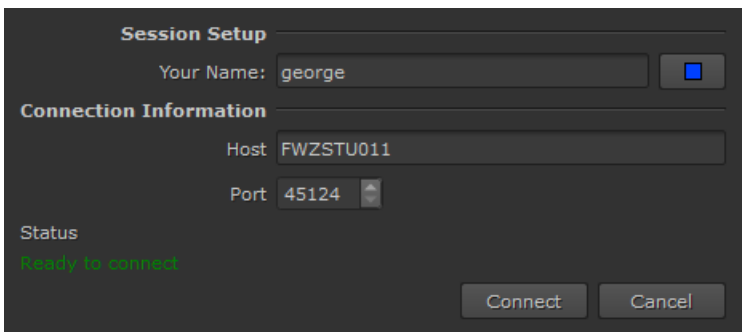
Warning: All participants must use the same version of the application to run a sync session. For example, if the host is using 15.0v3, clients can only connect to the session if they are using 15.0v3 as well.

To connect a client session to a host:

1. Ensure that you have access to the source files used in the host project.
   The files can be on a server on the same network as the host machine or accessed over the internet using suitable security, such as VPN.

Note: Only the host loads the project, client sessions don't require the **.hrox** file.

2. Go to **Workspace** > **Sync Session** to load a dedicated review workspace,
   OR
   Go to **Window** > **Sync Session** to open the **Sync Session** panel.
3. Click **Connect**.
   The **Session Setup** dialog is displayed.

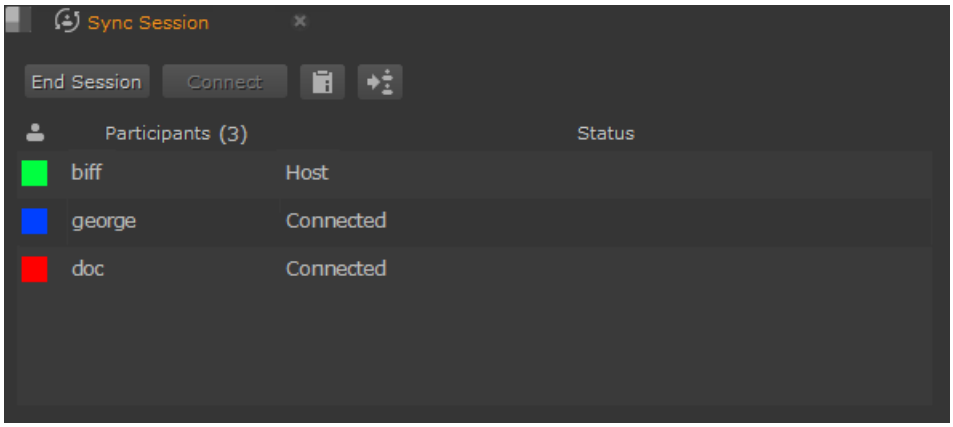


4. Enter the name you want to display on client sessions and pick a cursor color by clicking on the color swatch.

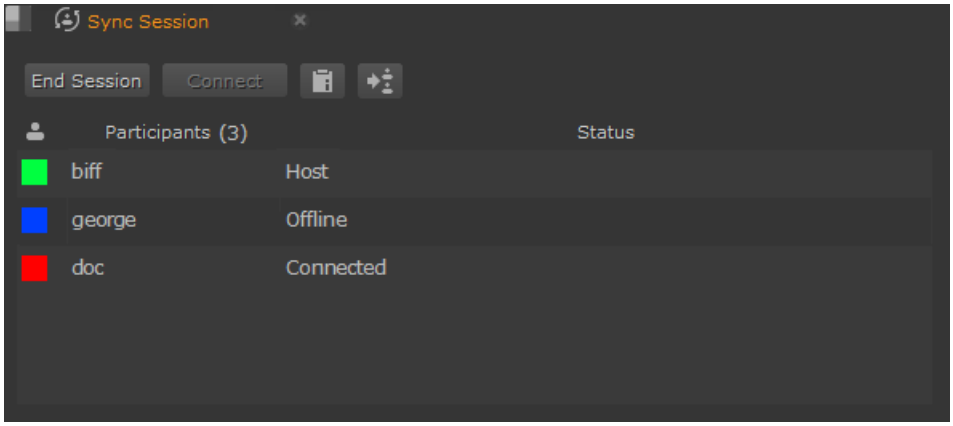
Tip: You can set your name and cursor color for new sessions in the **Preferences** under **Project Defaults** > **Sync Review**.

5. Enter the **Host** name and **Port** supplied by the Sync Review session host.
6. Click **Connect** to join the review.

   A list of participants is displayed in the **Sync Session** panel.



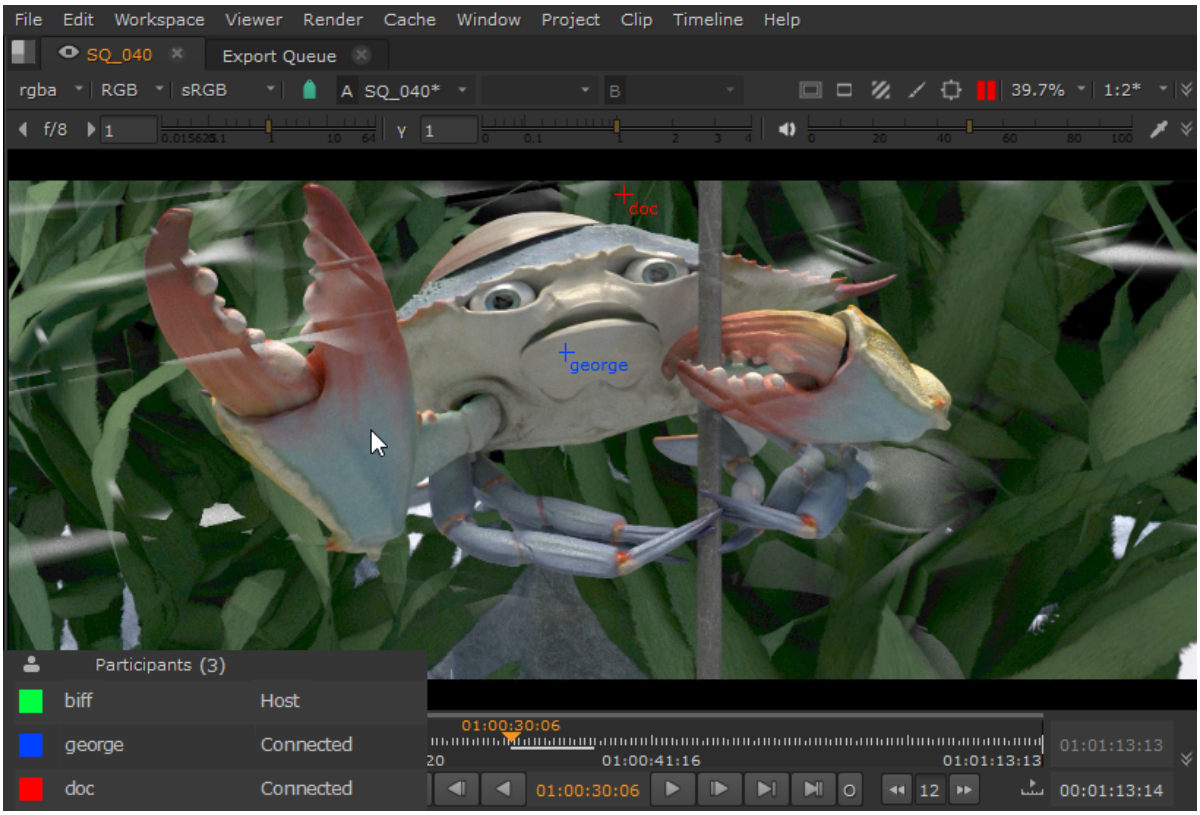
Client sessions can connect or disconnect at any time and the **Status** updates as required.



If the Host ends the session, all client sessions are also ended.

Collaborative Review

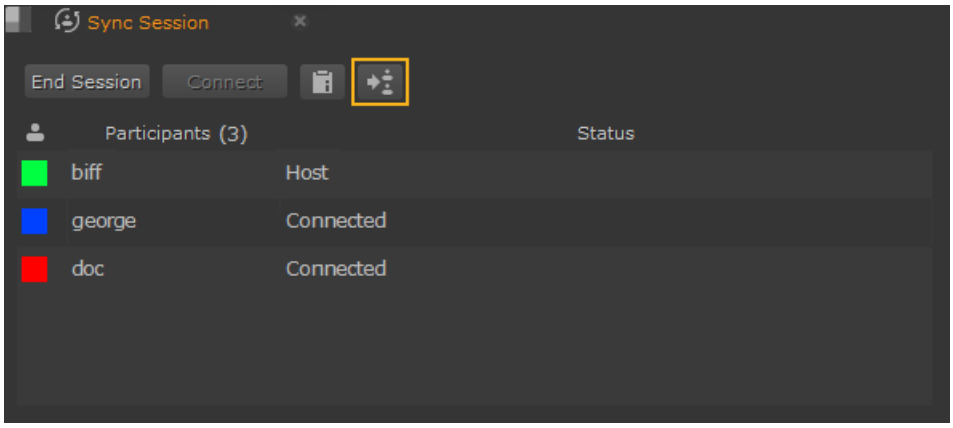
Participants in a Sync Review session can all contribute equally to the review. Each participant has a named and color-coded cursor that other participants can see in the Viewer. In the example, biff's cursor is shown as normal because it's their session.



Most participant actions available in the Viewer, Project panel, and timeline are synced automatically during the session, including:

- Viewer - Layer and channel selection, TC/Frames option, guides and Viewer selection options, Zoom in/out.
- Project panel - Importing files, creating new sequences, tags, and bins and creating, renaming, moving or deleting project items.
- Timeline - Creating new tracks, renaming shots, adding or deleting soft effects, changes in soft effects parameters, retimes and transitions, Sync Lock Status of tracks, trimming and moving clips in the timeline.

If participant sessions are out of sync, you can push actions to other participants manually. Click the update button in the **Sync Session** panel or use the **Ctrl**/**Cmd**+**P** keyboard shortcut to force an update across all participant sessions.


Sync Session
End Session
Connect
Participants (3)
Status
biff
Host
george
Connected
doc
Connected

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.

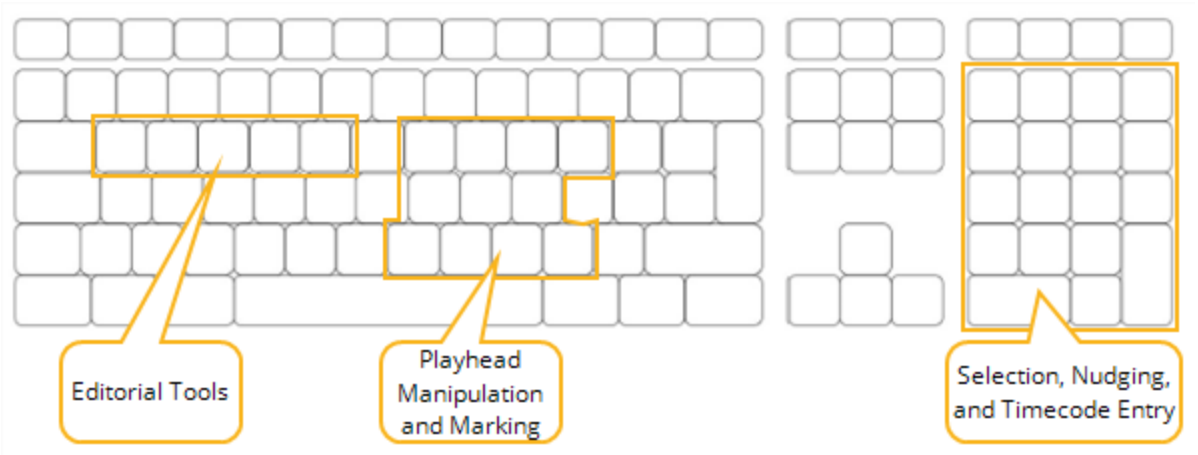
Icon	Tools	Description
	Multi Tool	The **Multi Tool**'s functionality is equivalent to most of the other tools combined, but doesn't require modal tool selection.
	Move/Trim	The **Move/Trim** tool allows you to manipulate the position of a shot or its output by adding or removing handles.
	Select	The 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.
	Selection by Track	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.
	Slip Clip	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.
	Slide Clip	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.
	Roll Edit	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.
	Ripple Edit	The **Ripple Edit** tool operates similarly to the trim function of the

Icon	Tools	Description
		Move/Trim tool, except that downstream shots are rippled to automatically close any resulting gaps in the timeline.
	Retime Clip	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.
	Razor	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.
	Join	The **Join** tool can only be used on edit points between two razored shots, denoted by the yellow arrows at the edit.

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.



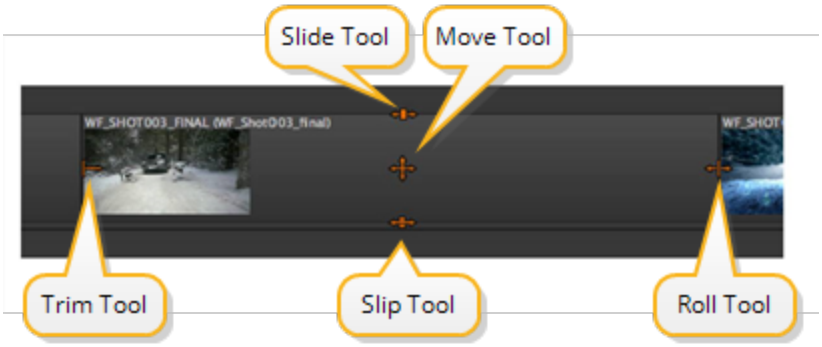
Pressing a keyboard shortcut multiple times selects the tools within each mode. For example, pressing **E** twice, rapidly in succession activates **Slide Clip**. Pressing them slowly in succession does not achieve the same result, but instead, remains on the first item in the menu. This allows you to 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.

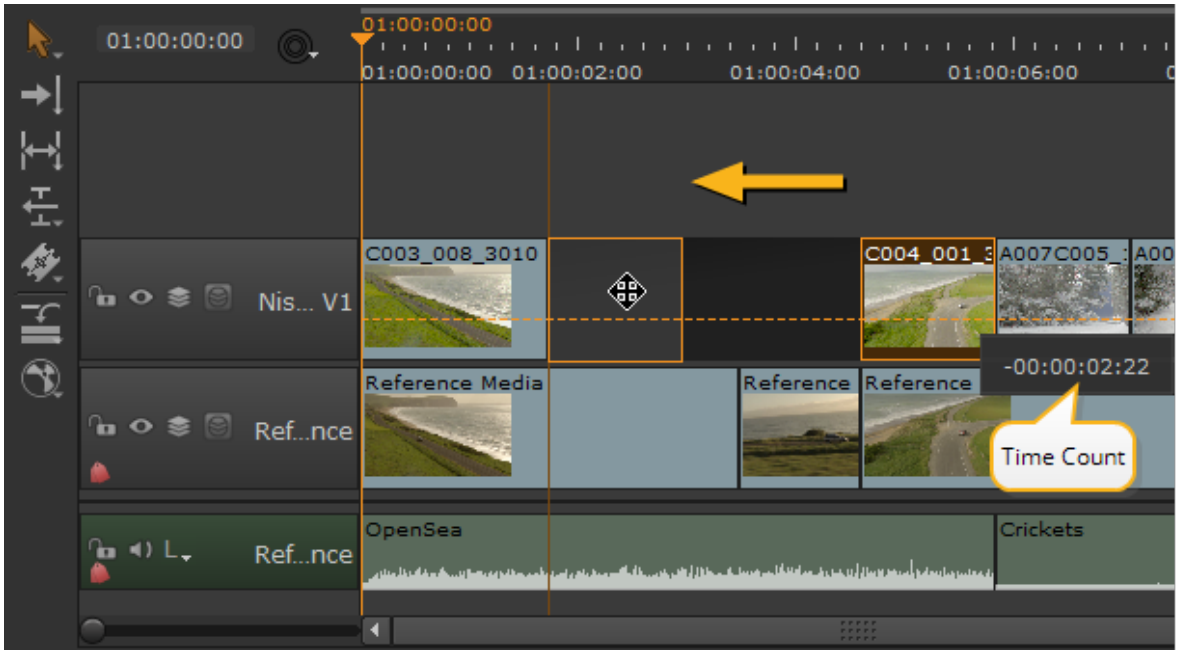
See Using the Razor and Join Tools for more information.

Using the Move/Trim Tool

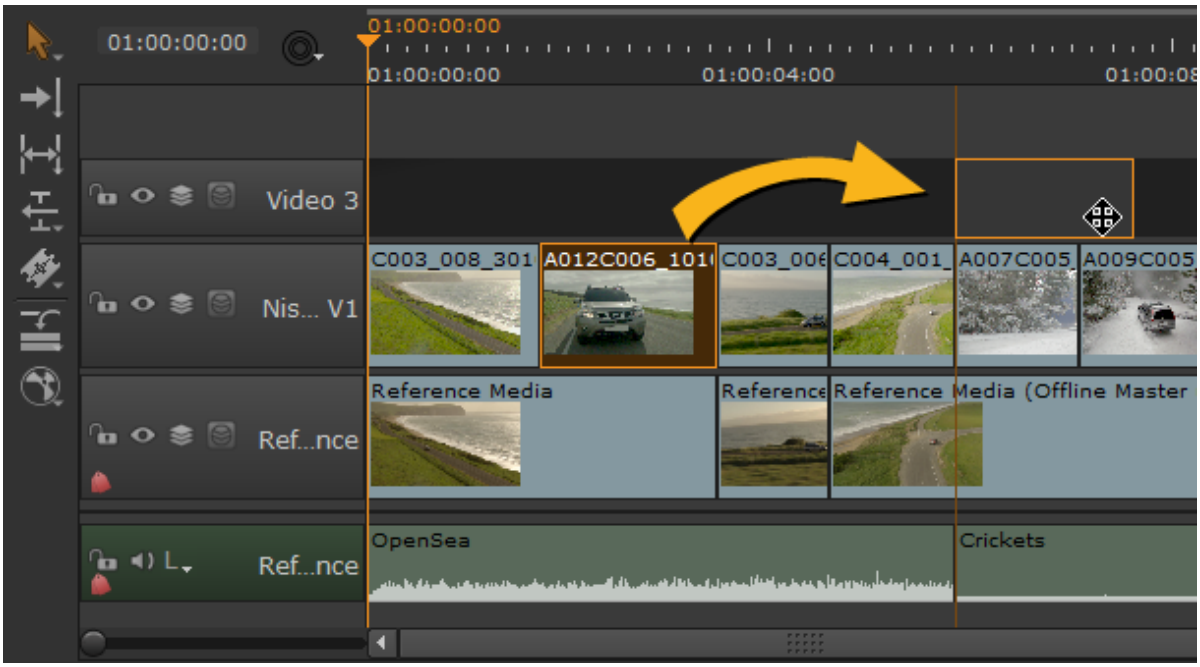
The **Move/Trim** tool allows you to manipulate the position of a shot or its output by adding or removing handles. Activate the **Move/Trim** tool by clicking the tool or pressing **Q** twice.

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:

Mode	Method	Description
Overwrite	drag-and-drop	The default move mode. The dragged shot overwrites any items that are present in the move location.
Ripple	drag then hold **Alt** and drop	Drag-and-drop shots on top of other items without overwriting content - items are pushed down the timeline to accommodate the move.
Duplicate	hold **Alt** and drag then release **Alt** and drop	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.
Ripple and Duplicate	hold **Alt** then drag and drop while holding **Alt**	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.

Note: On Linux, hold **Ctrl**+**Alt** for **Duplicate** and **RippleDuplicate** modifiers.

Action	Keyboard Shortcut	Description
Delete	**Backspace**	Delete the selected shot(s) or gap(s).
Ripple Delete	**Shift** + **Backspace**	Remove the selected shot(s) and ripple items down stream to close gaps in the timeline. **Note:** The ripple effect may not close gaps entirely, because Hiero 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**:

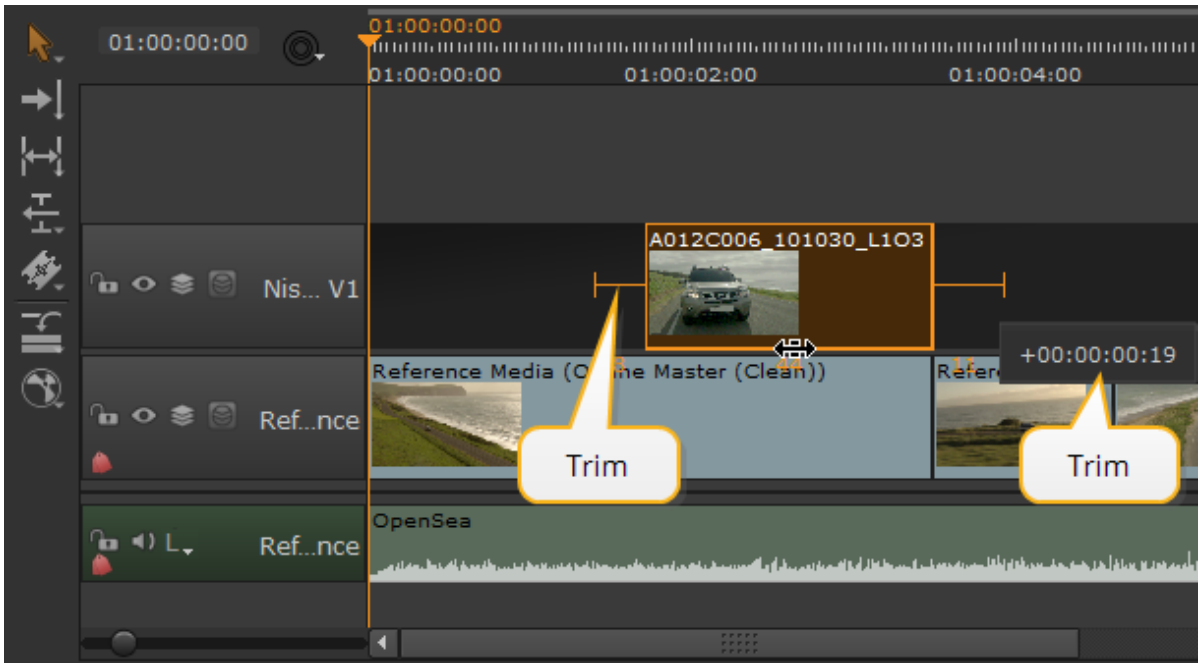
Modify	Using	Result
Dst In	Move Destination	Adjusts the event's **Dst In** and **Out** by the same amount, moving the shot's position on the timeline by the specified amount, while maintaining Speed. Before and after a 2 second **Dst In** increase: Speed | Src In | Src Out | Src Dur | Dst In | Dst Out | Dst Dur 100.0% | 00:00:00:01 | 00:00:04:00 | 100 | 01:00:00:00 | 01:00:03:24 | 100 Speed | Src In | Src Out | Src Dur | Dst In | Dst Out | Dst Dur 100.0% | 00:00:00:01 | 00:00:04:00 | 100 | 01:00:02:00 | 01:00:05:24 | 100
Dst Out	Move	Adjusts the event's **Dst Out** and **In** by the same amount, moving the

Modify	Using	Result
	Destination	shot's position on the timeline by the specified amount, while maintaining Speed. Before and after a 2 second **Dst Out** increase: Speed | Src In | Src Out | Src Dur | Dst In | Dst Out | Dst Dur 100.0% | 00:00:00:01 | 00:00:04:00 | 100 | 01:00:00:00 | 01:00:03:24 | 100 Speed | Src In | Src Out | Src Dur | Dst In | Dst Out | Dst Dur 100.0% | 00:00:00:01 | 00:00:04:00 | 100 | 01:00:02:00 | 01:00:05:24 | 100

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.

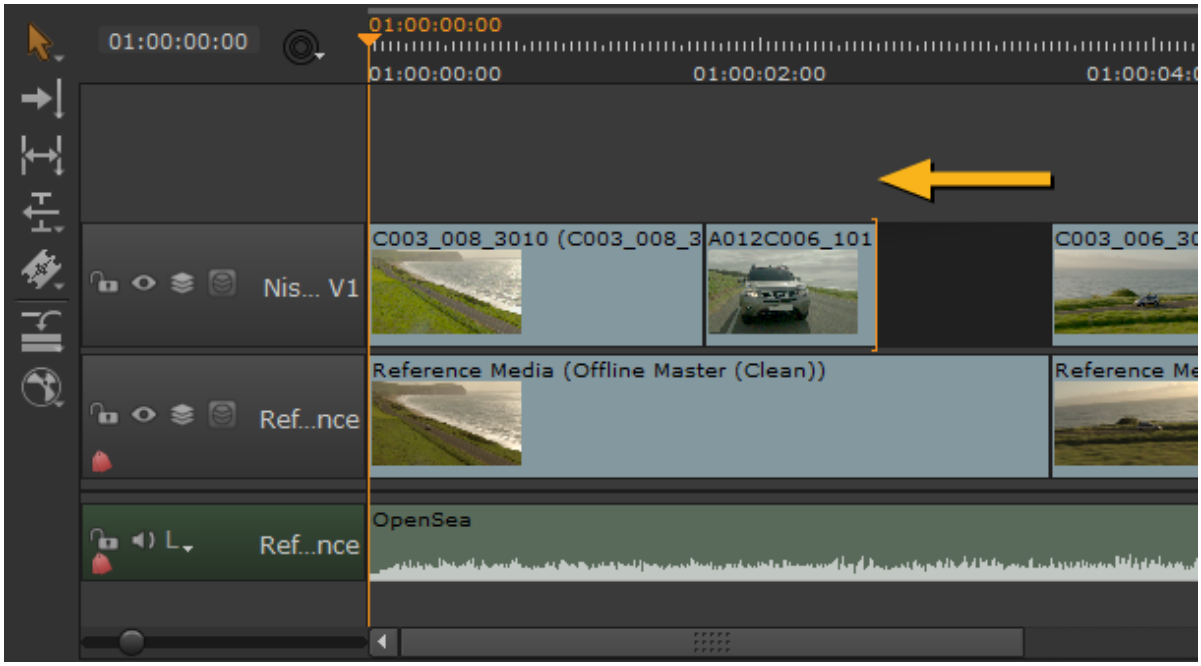


Tip: Use the **Ripple Edit** tool, activated by pressing **R** twice, to ripple downstream shots automatically.

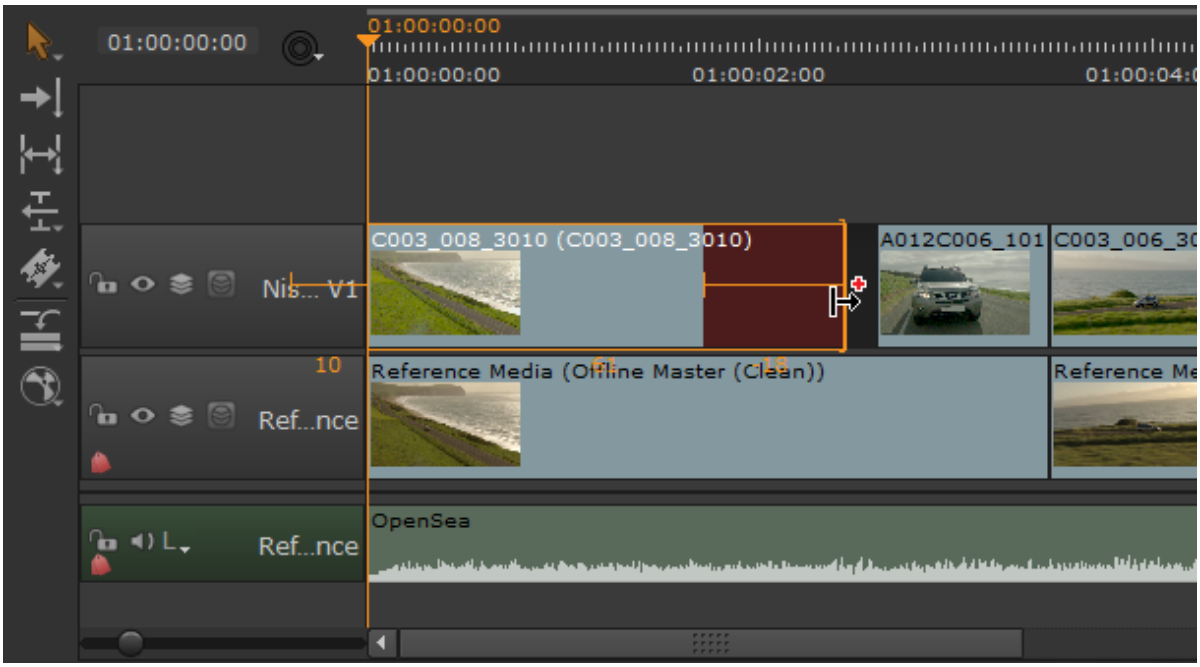
The Viewer displays the new In or Out point (depending on whether you're adjusting the beginning or end of the shot), allowing you to accurately gauge the new output.

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**+←→ (numeric pad) keys or hold **Shift** to nudge by the **Frame Increment** set under the Viewer.



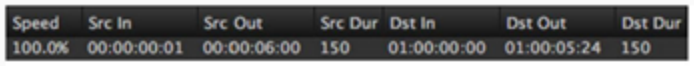
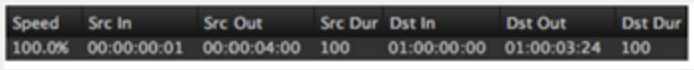
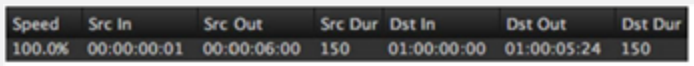
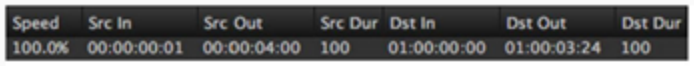
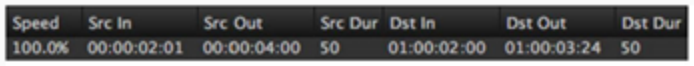
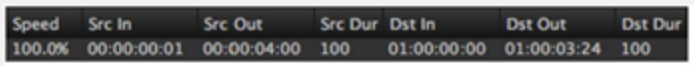
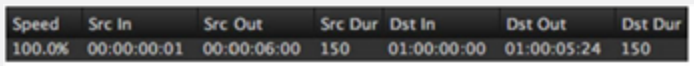
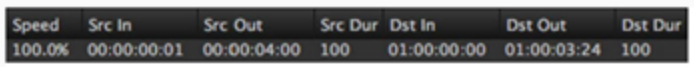
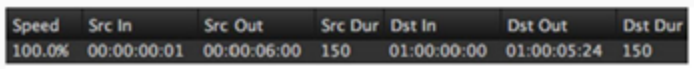
By holding **Ctrl/Cmd** and dragging an edit, you can add blank frames past the end of the shot's handles. Blank frames are colored red on the timeline for clarity:



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:

Modify	Using	Result
Src In	Trim In	Trims the event's **Src In**, **Dst In**, and **durations** while maintaining speed. Before and after a 2 second **Src In** increase: Speed | Src In | Src Out | Src Dur | Dst In | Dst Out | Dst Dur 100.0% | 00:00:00:01 | 00:00:04:00 | 100 | 01:00:00:00 | 01:00:03:24 | 100 Speed | Src In | Src Out | Src Dur | Dst In | Dst Out | Dst Dur 100.0% | 00:00:02:01 | 00:00:04:00 | 50 | 01:00:02:00 | 01:00:03:24 | 50
Src Out	Trim Out	Trims the event's **Src Out**, **Dst Out**, and **durations** while maintaining speed. Before and after a 2 second **Src Out** increase: Speed | Src In | Src Out | Src Dur | Dst In | Dst Out | Dst Dur 100.0% | 00:00:00:01 | 00:00:04:00 | 100 | 01:00:00:00 | 01:00:03:24 | 100

Modify	Using	Result
		
Src Dur	Trim Out	Trims the event's **Src Dur**, **Dst Dur**, and **Src/Dst Out** while maintaining speed. Before and after a 50 frame **Src Dur** increase:  
Dst In	Trim In	Trims the event's **Dst In**, **Src In**, and **durations** while maintaining speed. Before and after a 2 second **Dst In** increase:  
Dst Out	Trim Out	Trims the event's **Dst Out**, **Src Out**, and **durations** while maintaining speed. Before and after a 2 second **Dst Out** increase:  
Dst Dur	Trim Out	Trims the event's **Dst Dur**, **Src Dur**, and **Dst/Src Out** while maintaining speed. Before and after a 50 frame **Dst Dur** increase:  

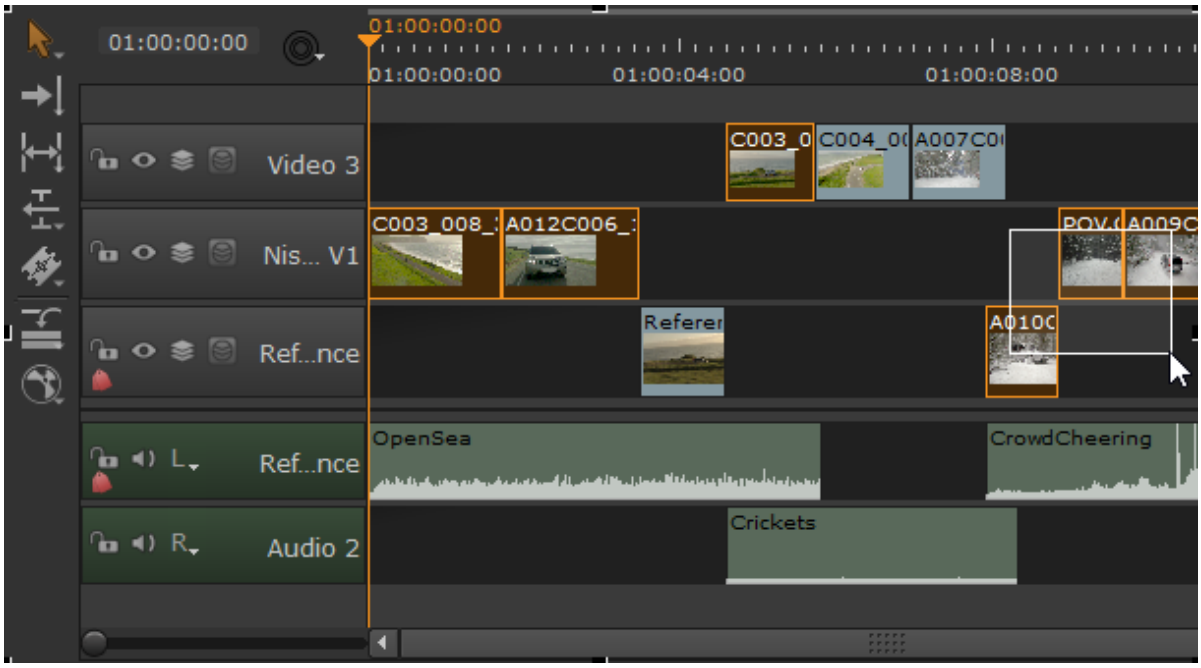
3. Adjust the values as required to trim the shot(s) by the specified amount.

Using the Selection Tools

The timeline editing tools include a marquee selection tool and several context dependent track selection tools.

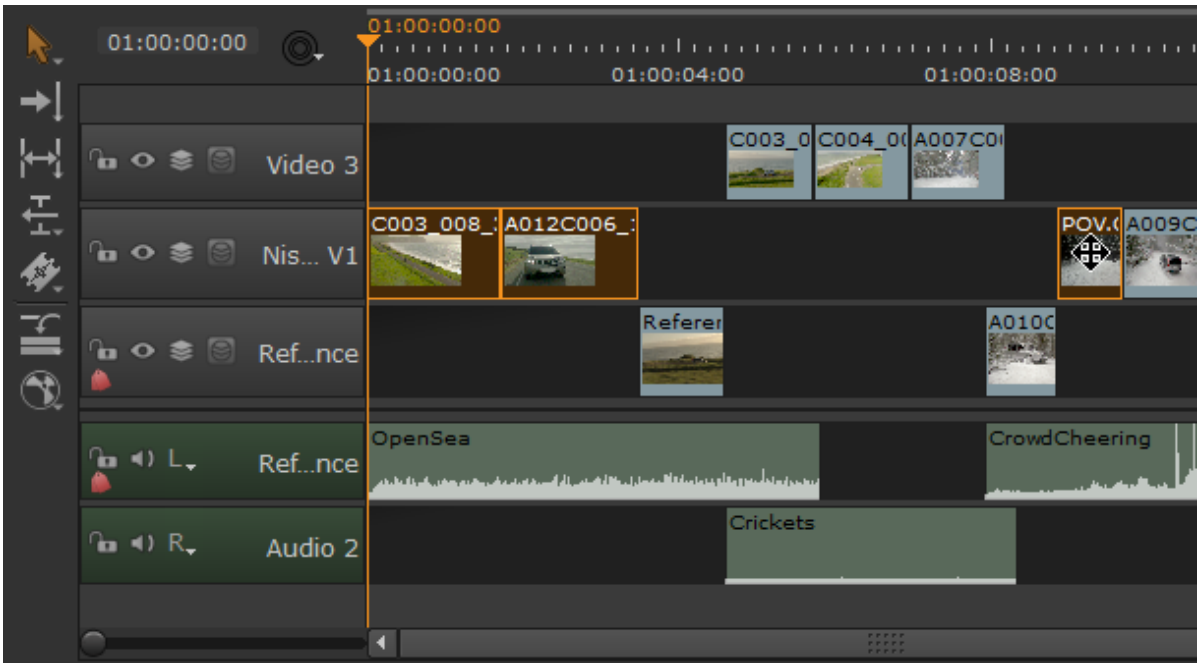
The marquee **Select** tool, activated by clicking the tool or pressing **Q** three times, allows you to make multiple selections quickly by lassoing shots.

Hold **Shift** to add to the selection:

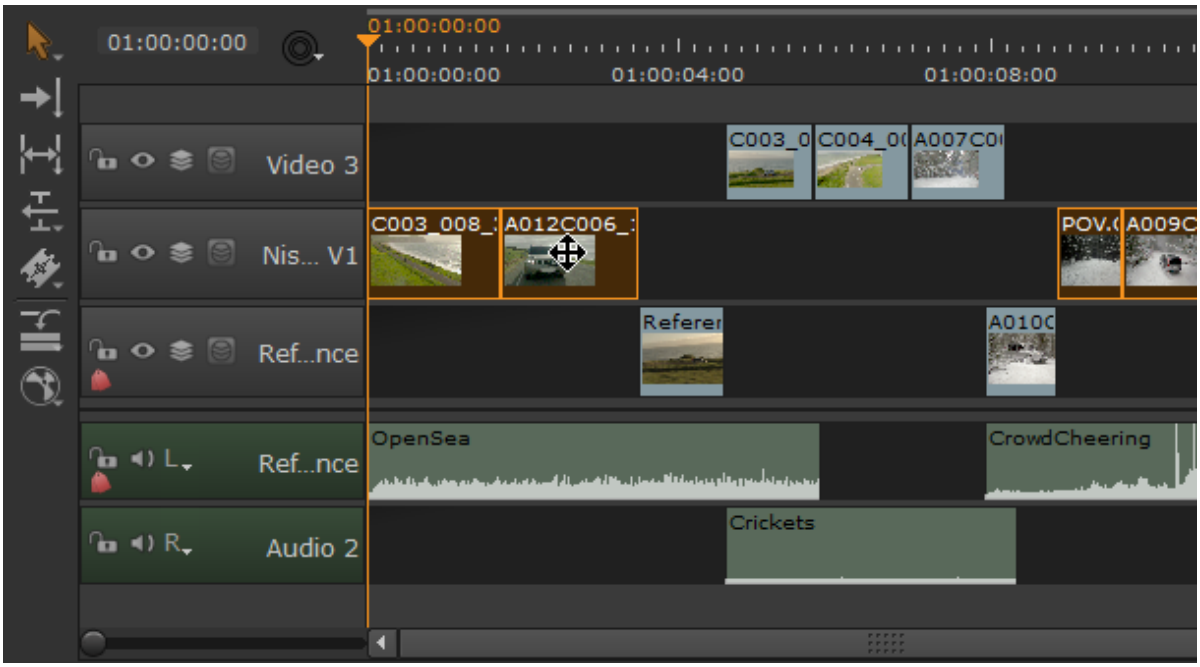


The track selection tools, activated by clicking the tool or by pressing **W**, selects multiple items depending on the initial selection:

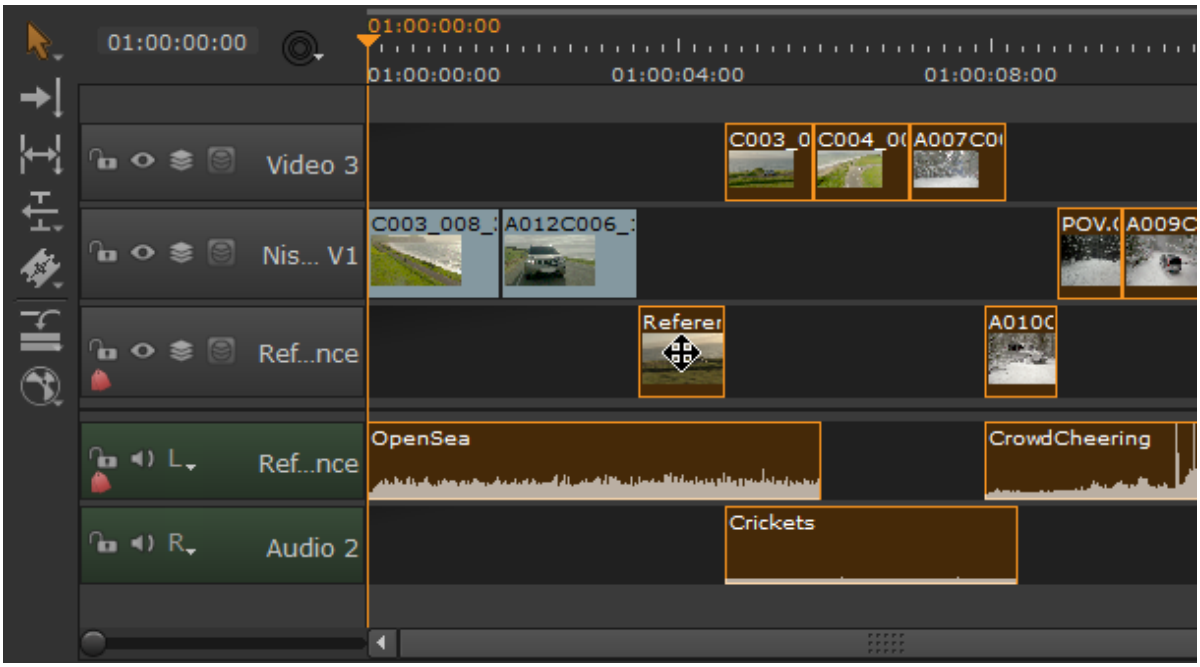
- **Select Track to Right** or **Left** - all shots right or left of the target shot are selected, within a single track.



- **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.


01:00:00:00
01:00:00:00
01:00:00:00
01:00:04:00
01:00:08:00
Video 3
Nis... V1
Ref...nce
Ref...nce
Audio 2
C003_0
C004_0(
A007C0(
C003_008_:
A012C006_:
POV.(
A009C
Referer
A010C
OpenSea
CrowdCheering
Crickets

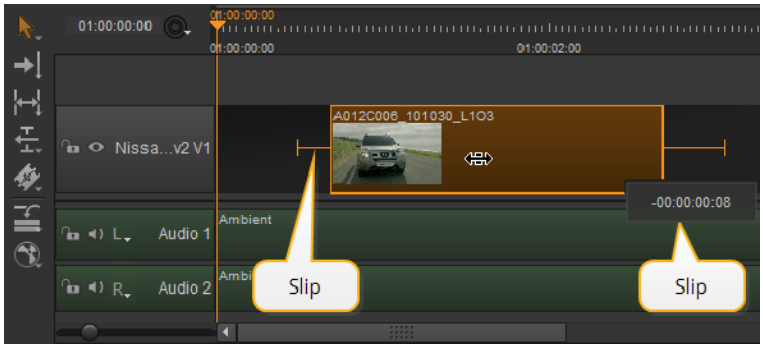
Using the Slip Clip Tool

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.



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.

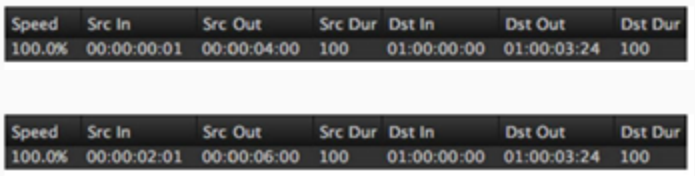
- When the playhead is on the target shot and A/B compare is active, the Viewer displays the target shot (1) and the reference shot (2), allowing you to synchronize your working track against the reference track.

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:

Modify	Using	Result
Src In	Slip Source	Adjusts the **Src In** and **Src Out** by the same amount, slipping the event while maintaining speed. Before and after a 2 second **Src In** increase: 
Src Out	Slip Source	Adjusts the **Src Out** and **Src In** by the same amount, slipping the event while maintaining speed. Before and after a 2 second **Src Out** increase: 

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.

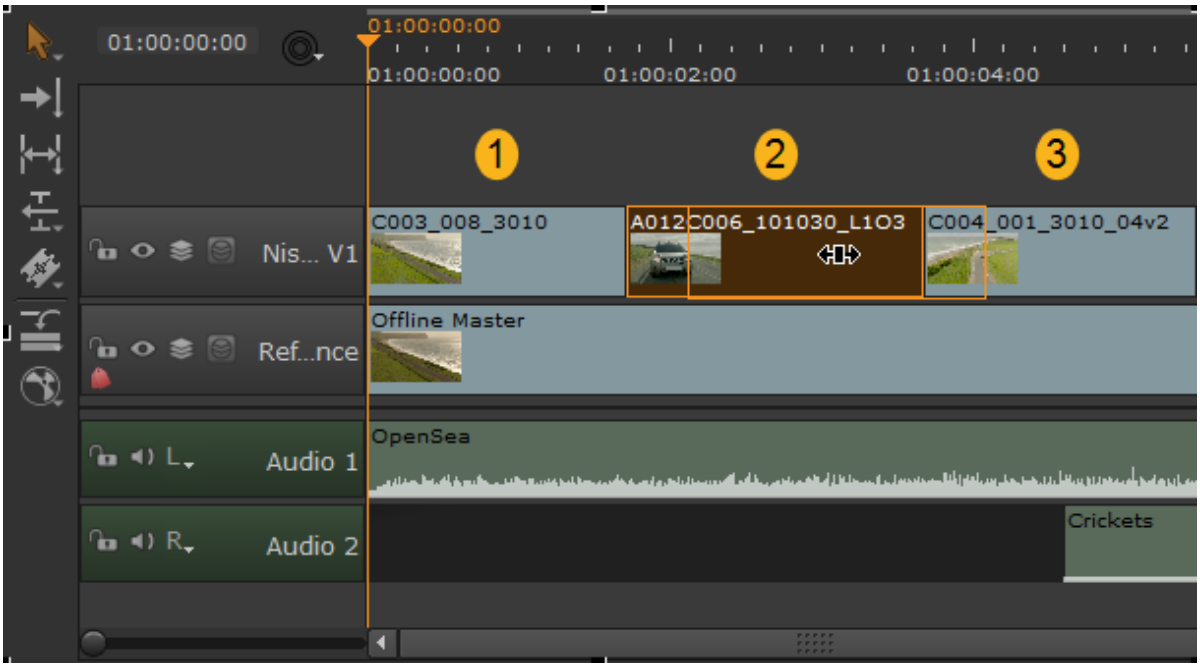
The shot either side of the target are shortened or lengthened within the limits of their handles to accommodate the slide.

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.

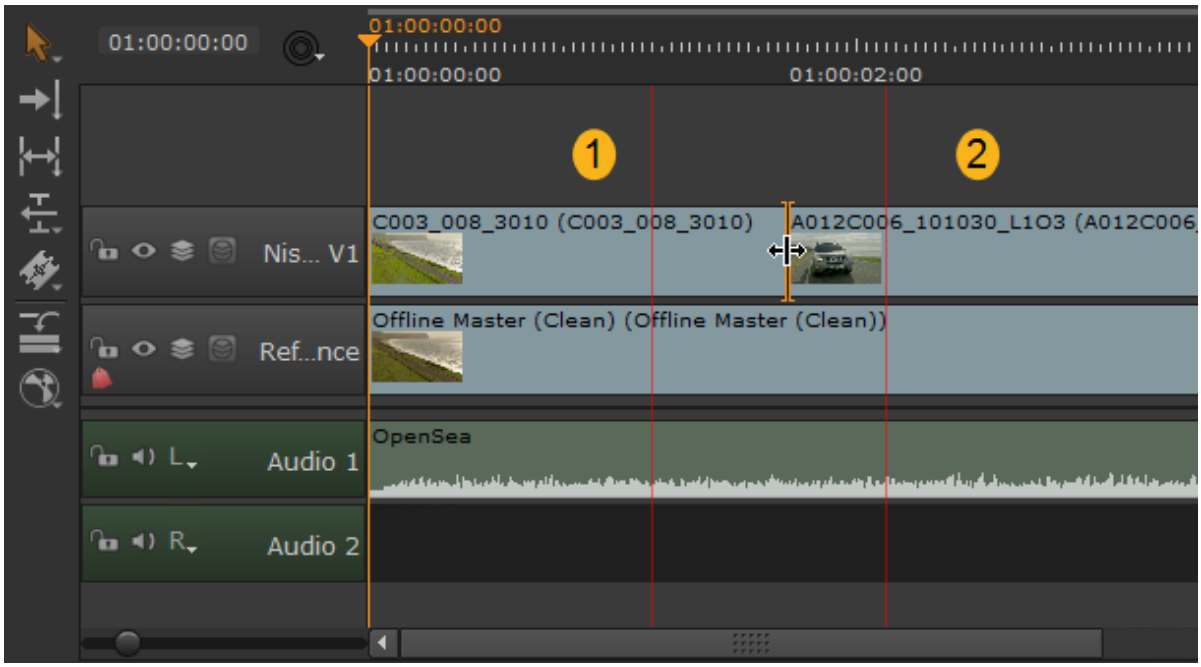
Using the Roll Edit Tool

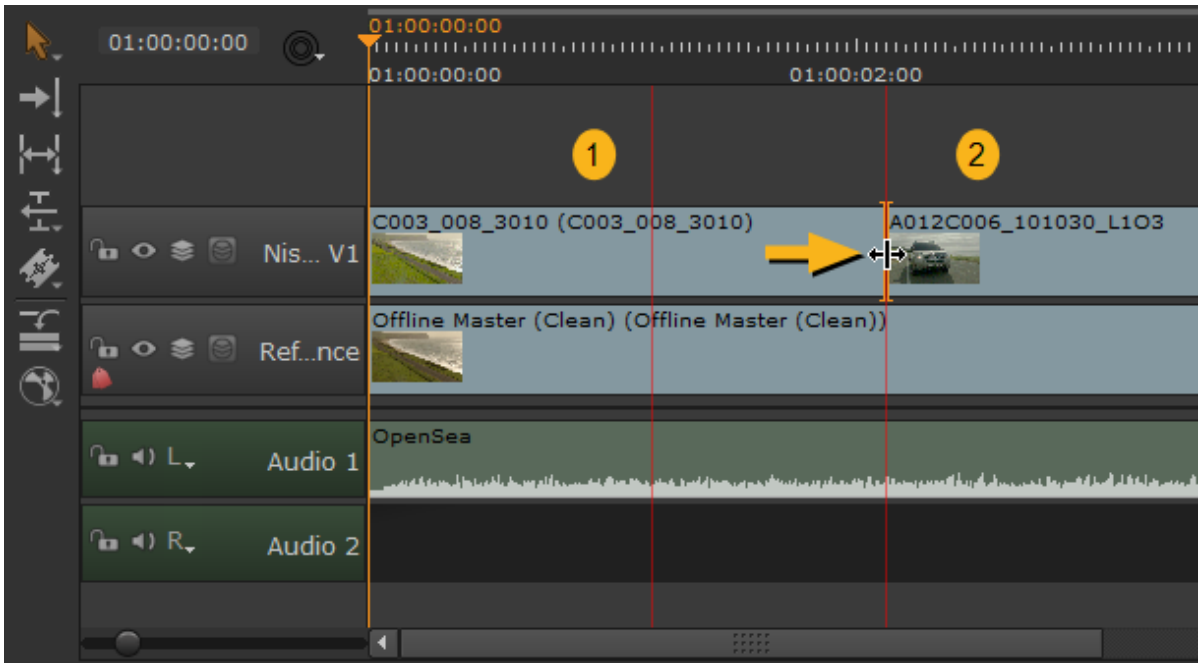
The **Roll Edit** tool enables you to roll a single edit within the available handles, shortening one shot while lengthening the other, but keeping the overall duration the same. Activate the **Roll Edit** tool by clicking the tool or pressing **R**.

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.





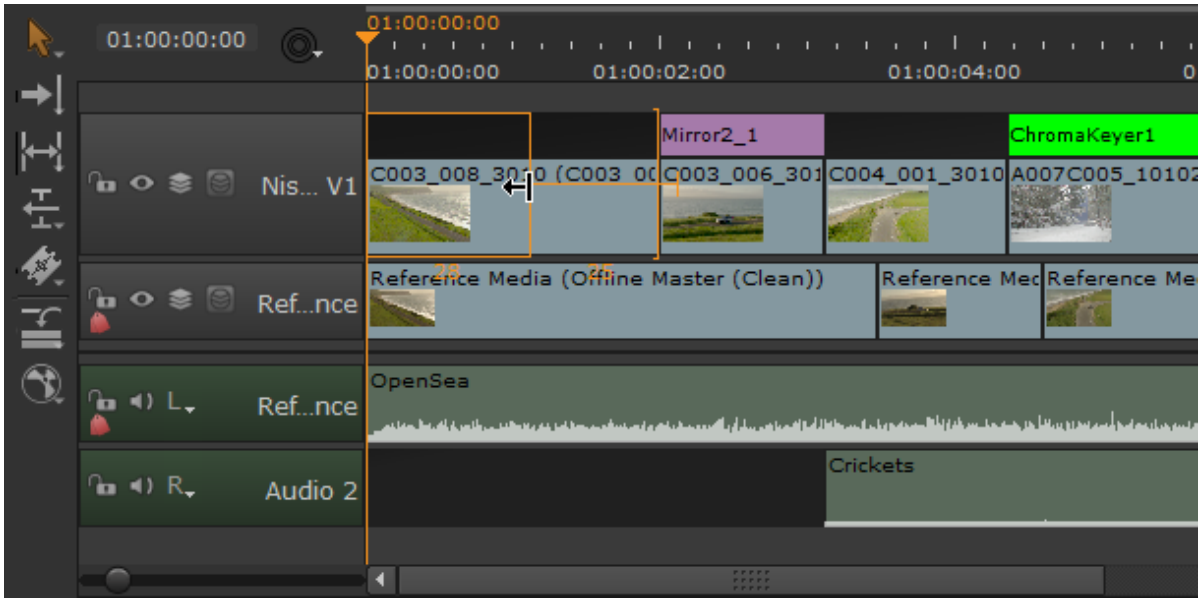
The Viewer displays the pre-edit shot on the left and the post-edit item on the right, allowing you to accurately gauge the new position of the edit.

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 **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**.

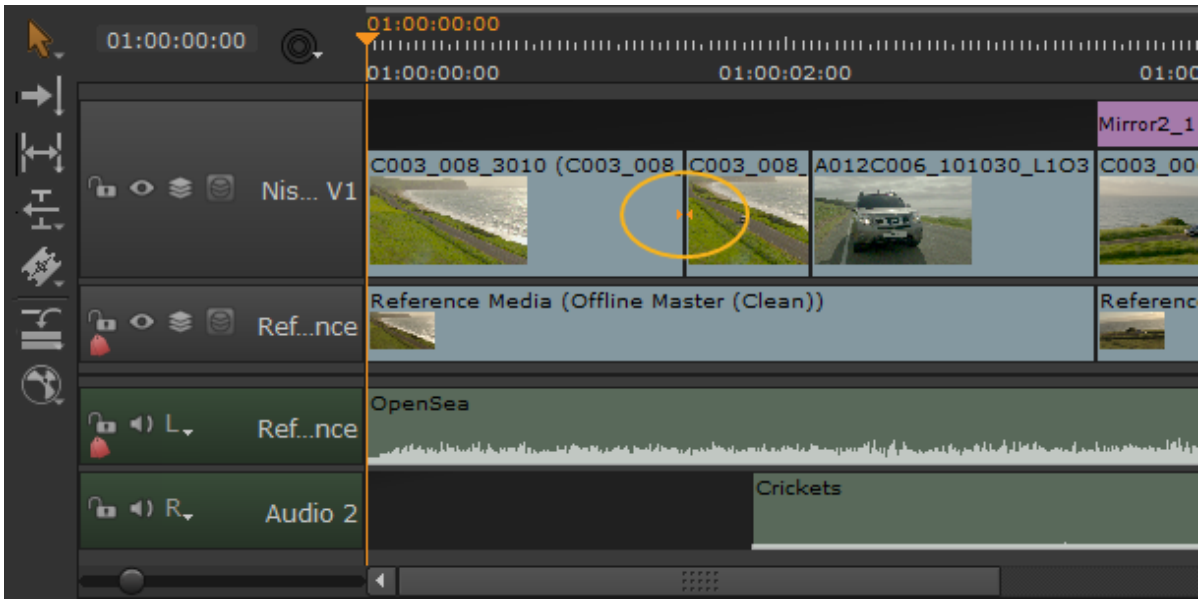
Place the cursor on the target shot, and if the cut is permissible, click to razor the shot or all shots depending on which tool you have selected.

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**.

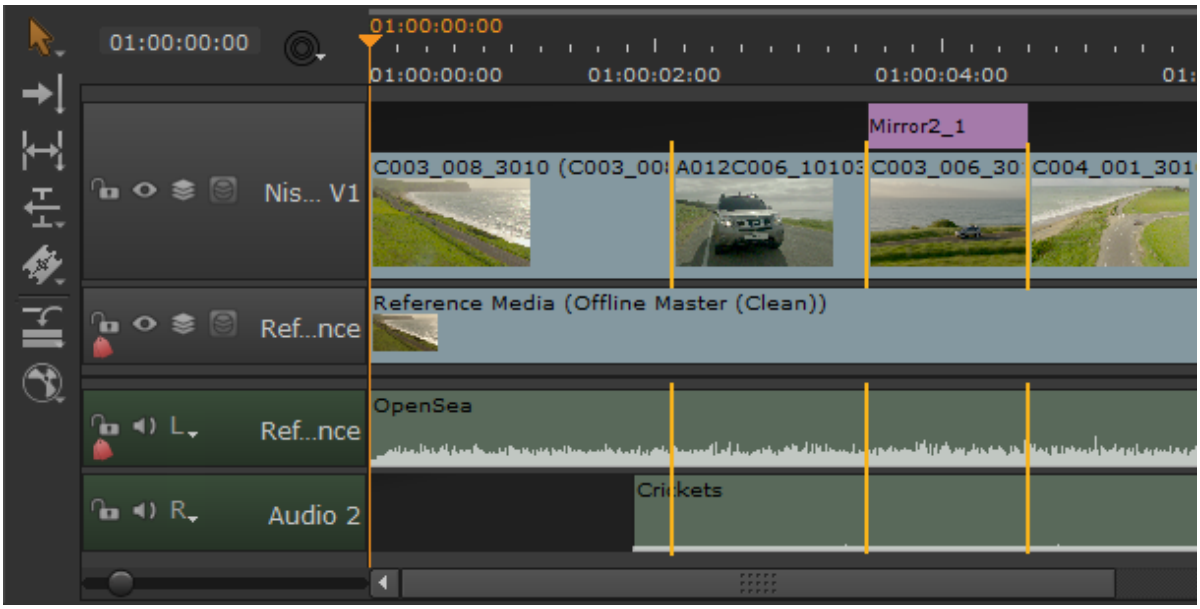
Tip: Use the **C** (with the shot under the playhead selected) and **Shift**+**C** keyboard shortcut, or the right-click context menu, to perform the respective cuts.

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

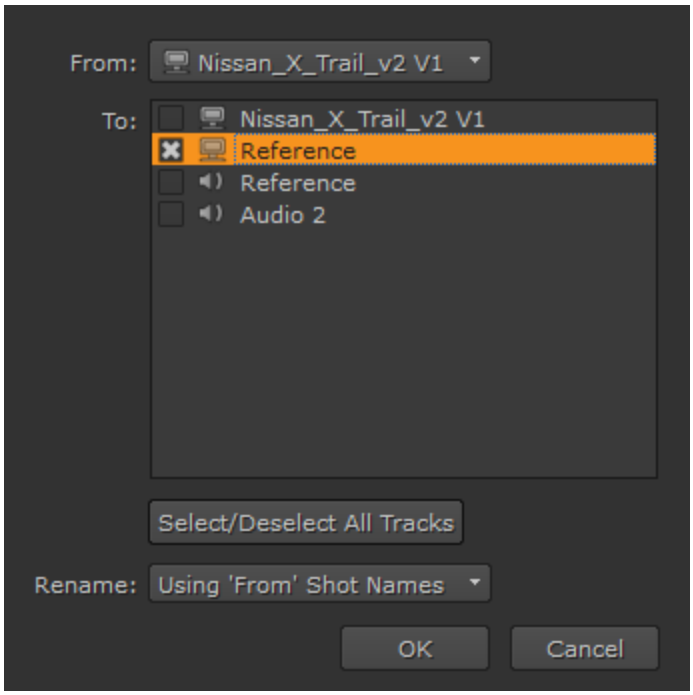
The **Copy Cuts** function allows you to quickly apply cuts from one track to other tracks on the timeline. For example, in the timeline shown, you could copy the cuts from the second video track to the Reference audio tracks.



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.

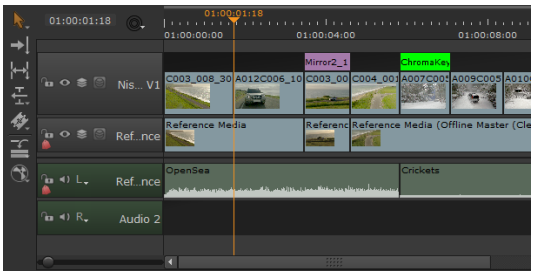
   Selecting **None** retains the destination clip name.
7. Click **OK** to copy the cuts to the destination track(s).

Insert, Overwrite, and 3-Point Editing

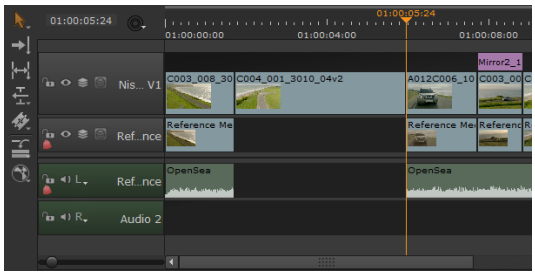
Insert and **Overwrite** edits are applied at the current playhead position by default, but the use of In and Out points in the clip Viewer and/or sequence Viewer can give you greater control over the result. 3-point editing, makes use of In and Out points in the clip Viewer and an In or Out in the sequence 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



Post-insert timeline

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.

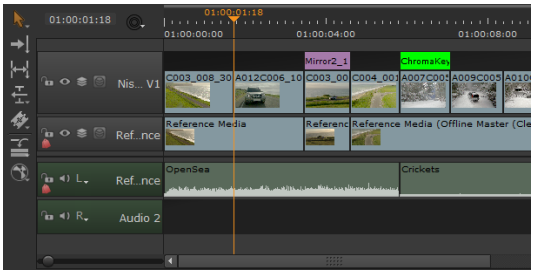
2. Double-click your sequence in the bin view to load it into the right-hand sequence Viewer .
3. Double-click the source clip to load it into the left-hand clip Viewer .
4. Place the playhead at the required edit point or timecode and select the target track, if necessary.
5. Navigate to **Clip** > **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:

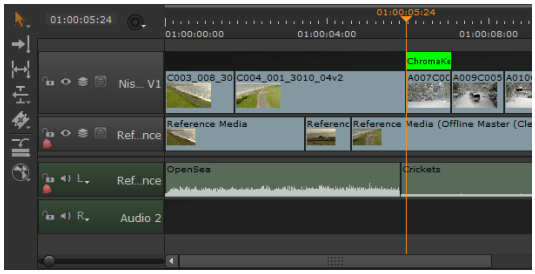
1. Navigate to **Workspace** > **Editing** to display the 2-up Viewer layout.
2. Double-click your sequence in the bin view to load it into the right-hand sequence Viewer .
3. Double-click the source clip to load it into the left-hand clip Viewer .
4. Place an In or Out point on the timeline to determine 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



Post-overwrite timeline

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**.

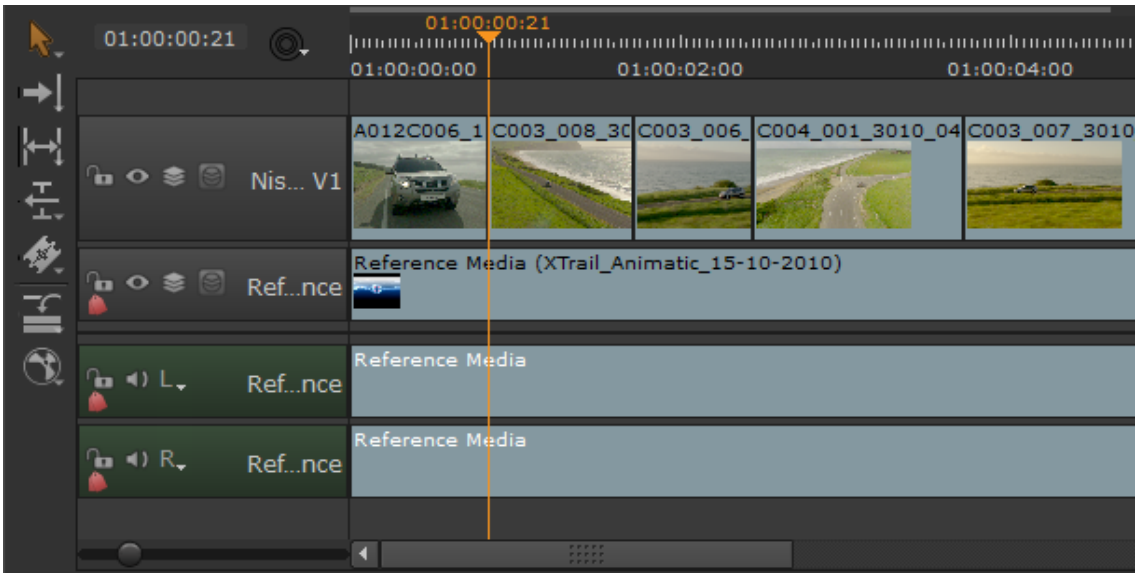
Tip: You can set both In and Out points on the timeline, but bear in mind that there may be insufficient source frames for the range specified. If this is the case, blank frames are added and highlighted in red.

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.

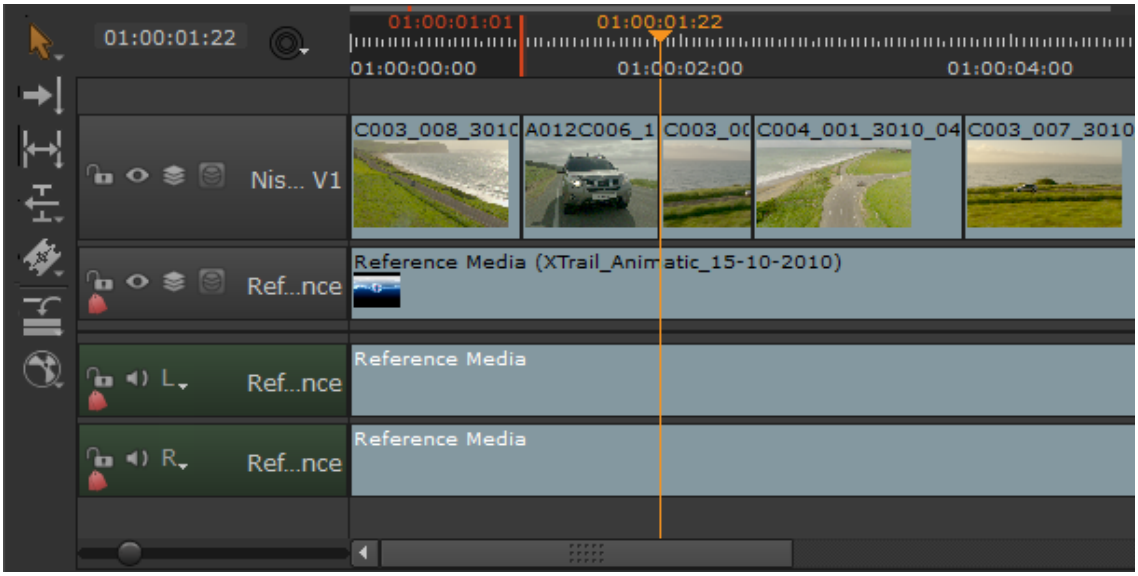
1. Navigate to **Workspace** > **Editing** to display the 2-up Viewer layout.
2. Double-click the required source clip to load it into the left-hand clip Viewer .
3. Set the required frame range using In and Out points.
4. Double-click your sequence in the bin view to load it into the right-hand sequence 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:

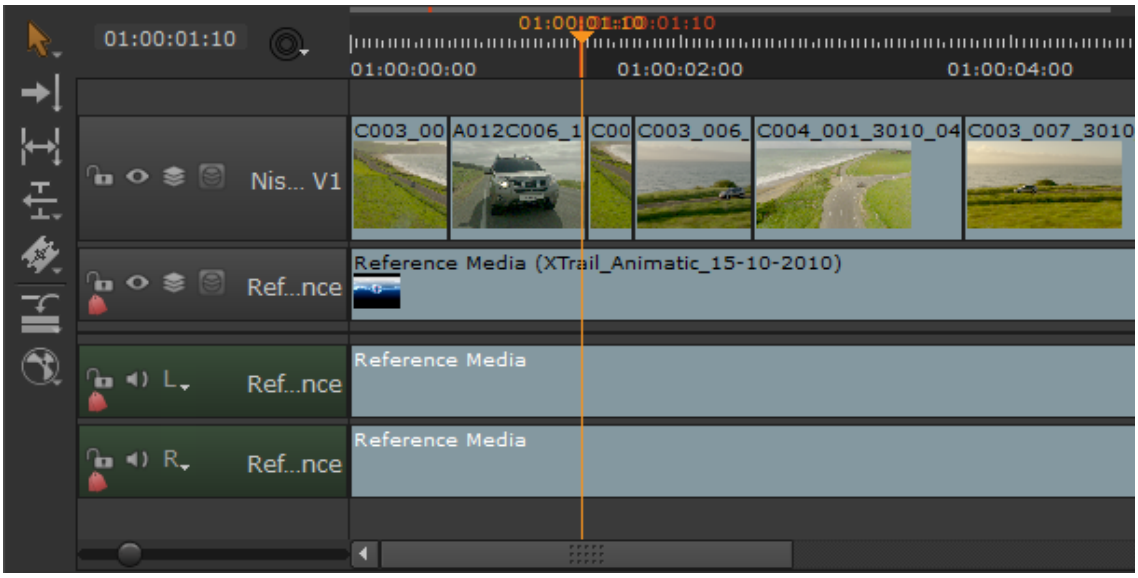
- **No In or Out points** - insert or overwrite at the current playhead position, for the range currently set in the clip Viewer.



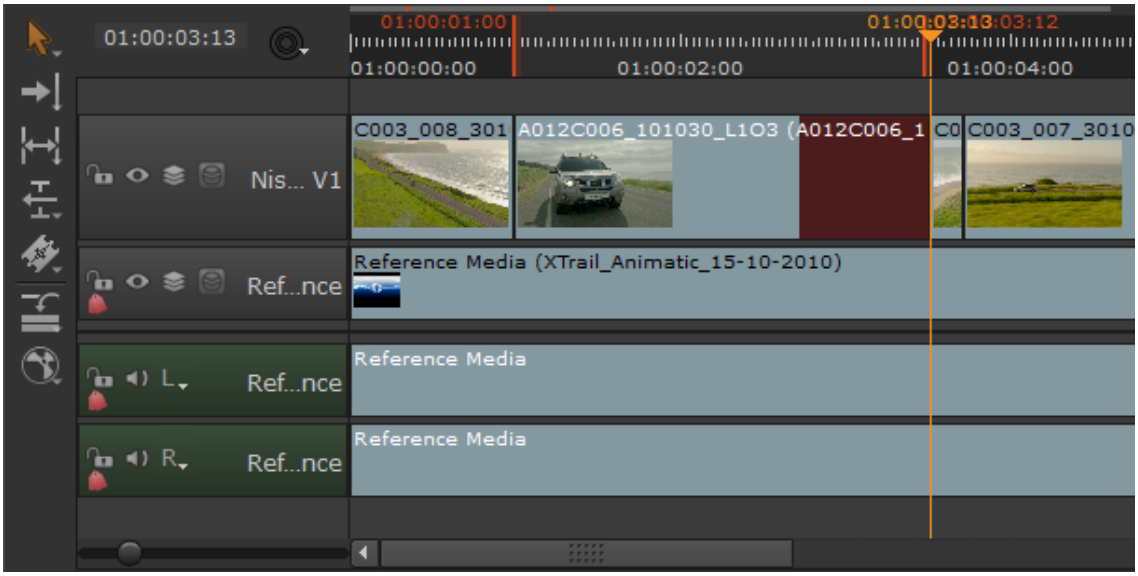
- **In point but no Out point** - insert or overwrite from the In point position downstream, for the range currently set in the clip Viewer.



- **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, Hiero 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. Source clips have three states: Linked (default), Unlinked and Mixed, and you can define if a shot is linked to its source clip or not. See Version Linking for more information.
- **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 directory structure conventions to discover versions:

Convention	Description	Example
File name constants		
Clip name	The file name base must remain the same.	**myClip**_v1.0001.dpx **myClip**_v2.0001.dpx **myClip**_v3.0001.dpx
Version prefix	The delineation between the file name and version information must be either _ (underscore) or **.** (period) and remain the same for all versions.	myClip_v1.0001.dpx myClip_v2.0001.dpx myClip_v3.0001.dpx
File name variables		
Version padding	The version number padding in the clip name can be increased or decreased.	myClip_**v1**.0001.dpx myClip_**v002**.0001.dpx myClip_**v03**.0001.dpx
Frame padding	The frame padding in the clip name can be increased or decreased.	myClip_v1.**01**.dpx myClip_v1.**1**.dpx myClip_v1.**0001**.dpx
Extension	The file format is interchangeable. See Appendix C: Supported File and Camera	myClip_v1.01.**png** myClip_v1.0001.**dpx**

Convention	Description	Example
	Formats for more information.	myClip_v1.**mov**
 **Note:** If the file extension is a movie format, such as **.r3d** or **.mov**, the **Frame padding** can be omitted.		
Directory name constants		
Root directory	The root directory name must remain the same for all directories containing versions.	**~/version/**v1/myClip_v1.0001.dpx **~/version/**v2/myClip_v2.0001.dpx **~/version/**v3/myClip_v3.0001.dpx
Directory name variables		
Version padding	The version number padding in the directory name can be increased or decreased.	09_WF_Shot004_**v1** 09_WF_Shot004_**v002** 09_WF_Shot004_**v03**

Version Linking

Control linking between source clip versions and shot versions.

Versions in Bins

Apply versioning to Project bin source clips.

Versions in Sequences

Apply versioning to shots in a sequence.

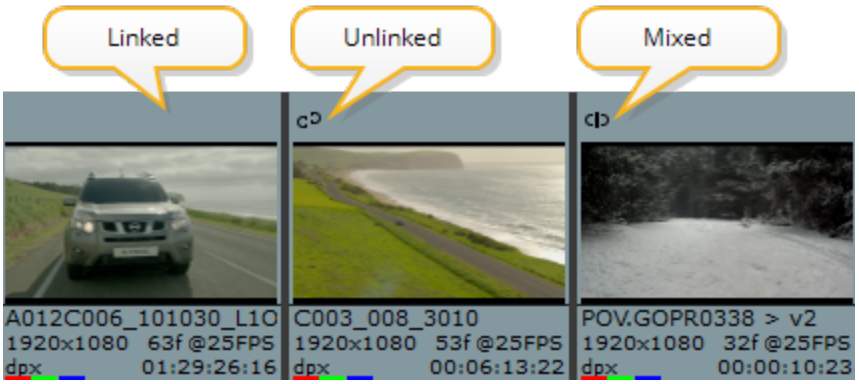
Version Linking

Version Linking allows linking and unlinking of source clip versions between source clips and shots. You can define if a shot version is linked to its source clip version or not, and there are three clip states:

- **Linked** - the source clip and all instances of shots referencing the clip are linked. Versioning up or down on any linked item affects all items.

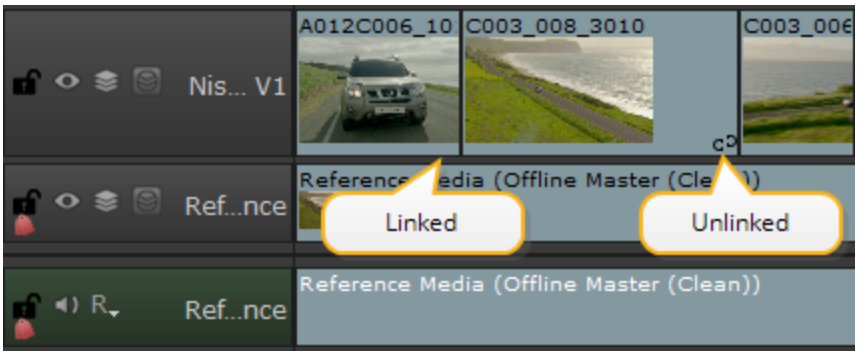
> **Note:** Linked is the default state for all clips and shots, but you can set the default in the **Preferences** under **Project Defaults** > **General** > **Link bin and track item version**.

- **Unlinked** - the source clip and all instances of shots referencing the clip are unlinked. Versioning up or down on any clip or shot only affects that item.
- **Mixed** - the source clip is linked to some instances of shots referencing the clip, but not all of them. Versioning up or down on any clip or shot only affects linked items. Unlinked items are unchanged.

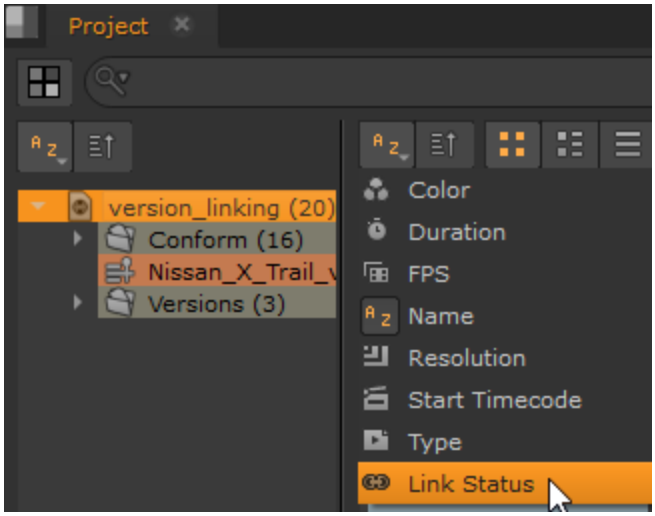


Two states govern shots in sequences:

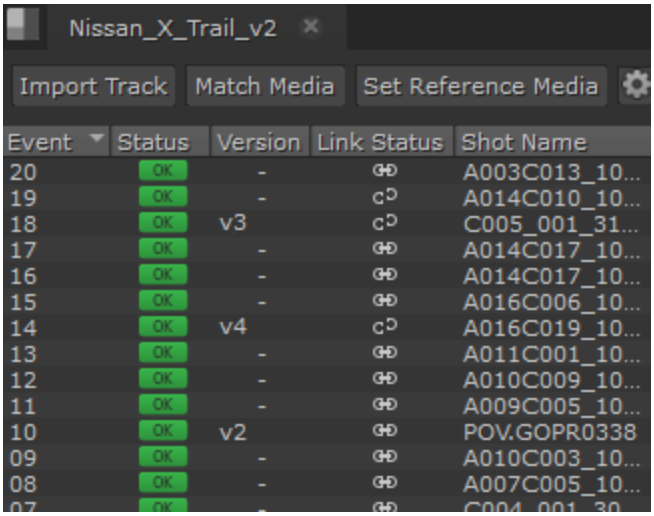
- **Linked** - The shot is linked to the version used by the source clip it references. Versioning up the shot or the source clip affects both.
- **Unlinked** - The shot is not linked to the version used by the source clip it references. Versioning up the shot or the source clip only affects the versioned item.



You can order the **Project** bin by **Link Status** to keep track of clip status and the spreadsheet view includes columns for **Version** and **Link Status** to keep track of shot status.



Source clip link status.

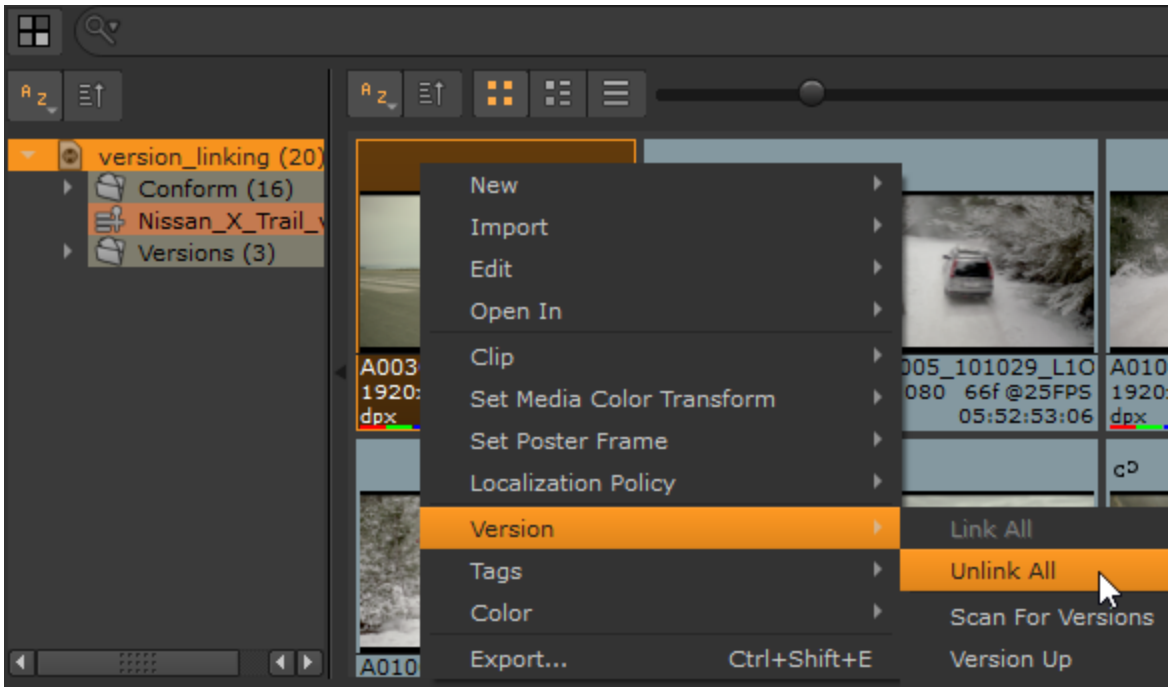


Shot version and link status.

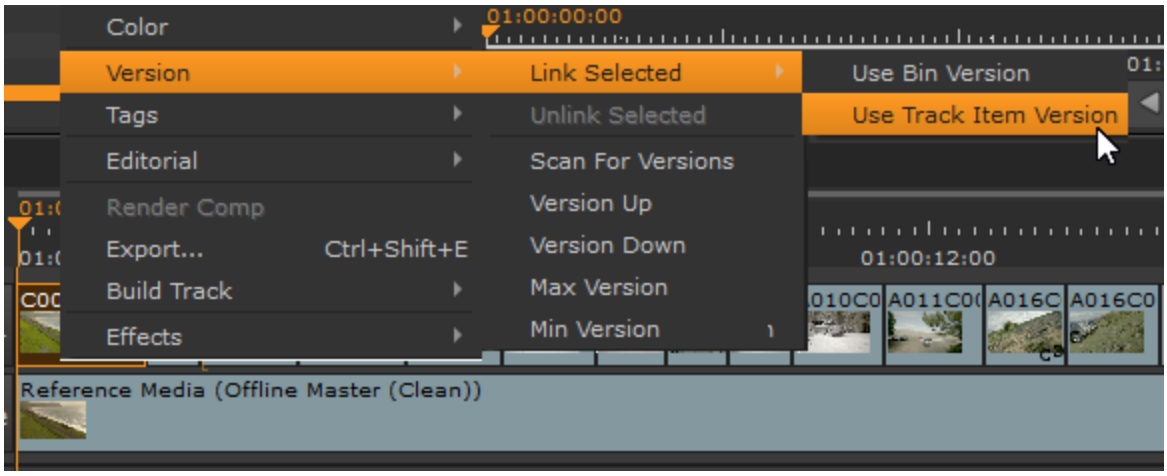
Linking and Unlinking Clips and Shots

Clip and shot versions are linked by default so that versioning up or down on any linked item affects all other items. You can unlink all clips and shots in new projects by disabling **Project Defaults** > **General** > **Link bin and track item version** in the **Preferences**. If you want more control over linking, you can right-click a source clip or shot and set the link state individually.

Source clips allow you to link or unlink all instances of shots in one action. **Link All** and **Unlink All** affect the version of all shots that reference the selected source clip.



Shots can be linked or unlinked to their source clip's version individually. If you choose to link a shot, you can choose to use the shot's current version or the clip's current version as the linked version.



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.

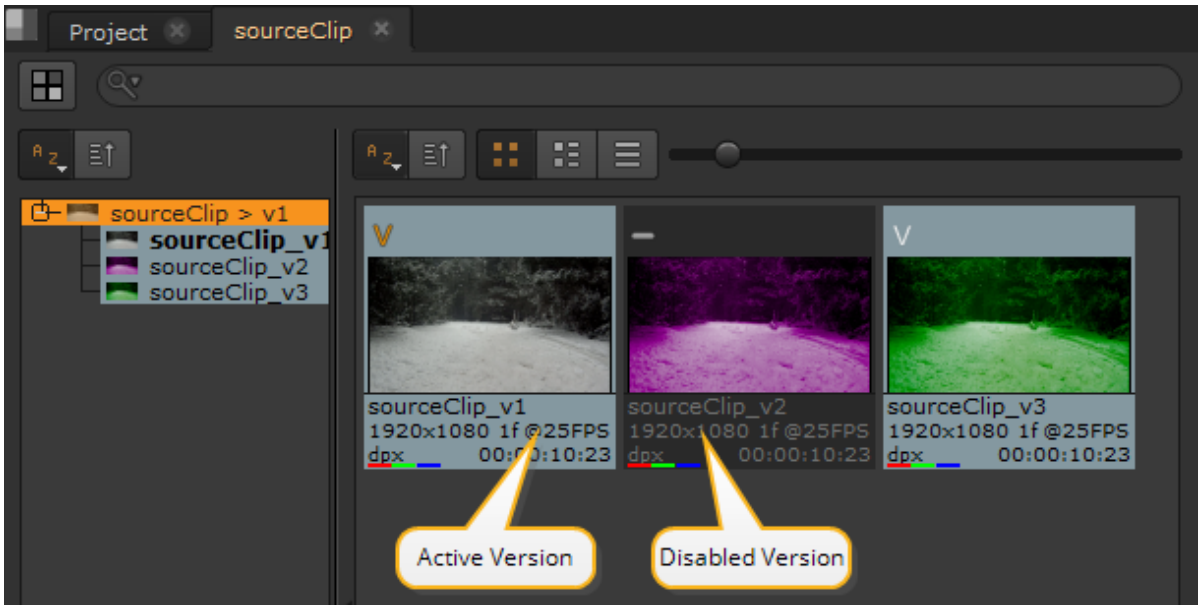
   A dialog box lets you know how many versions were discovered.
2. Use the right-click **Version** menu to:
   - Go to the next **Version Up** or **Version Down**.
   - Go to the **Minimum** or **Maximum Versions**.

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.

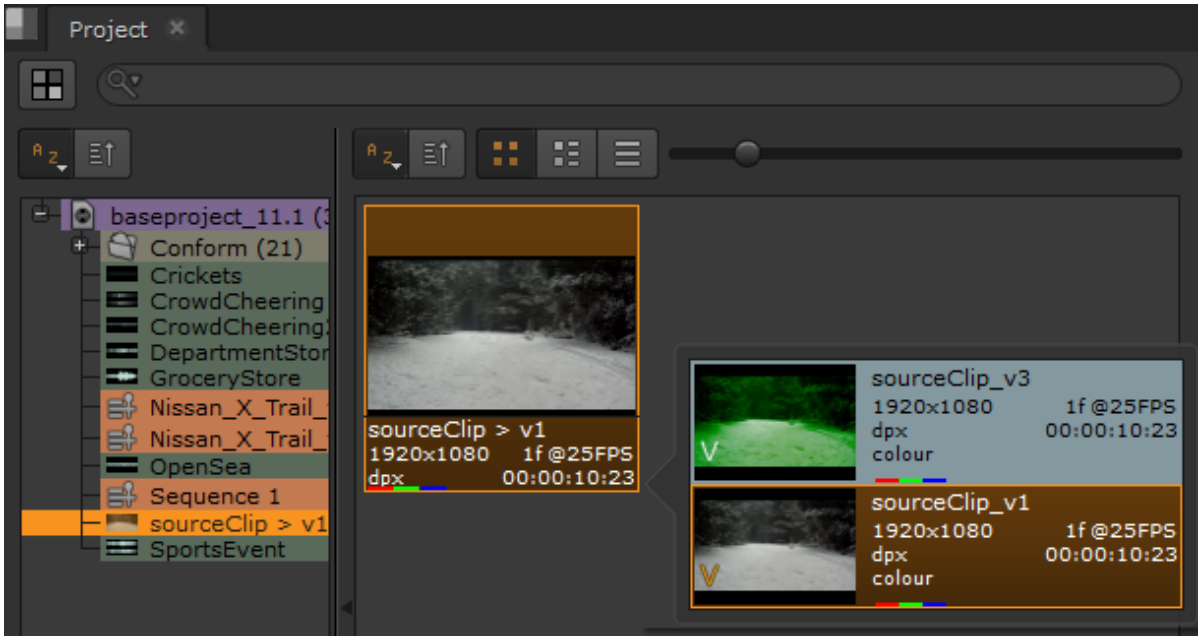
   When you reach the end of the discovered versions, incrementing the version automatically scans for new versions that may have become available.
3. For source clips only, you can right-click the clip and select **Open In** > **Versions Bin** to display all discovered versions of the target clip.

   The versioning conventions may allows 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.

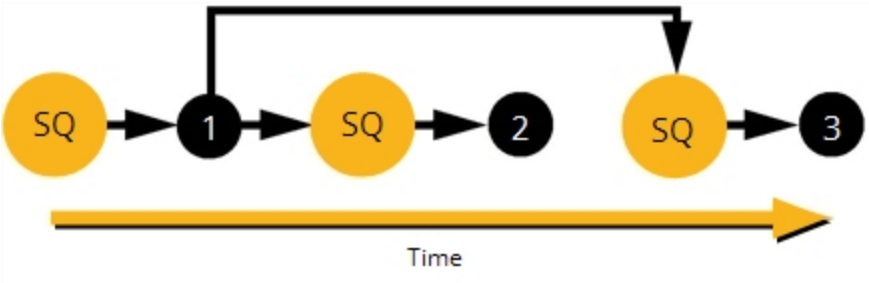
Once you've scanned for versions, select a shot on the timeline and press **V** to display all available versions for that item in a convenient window.

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:

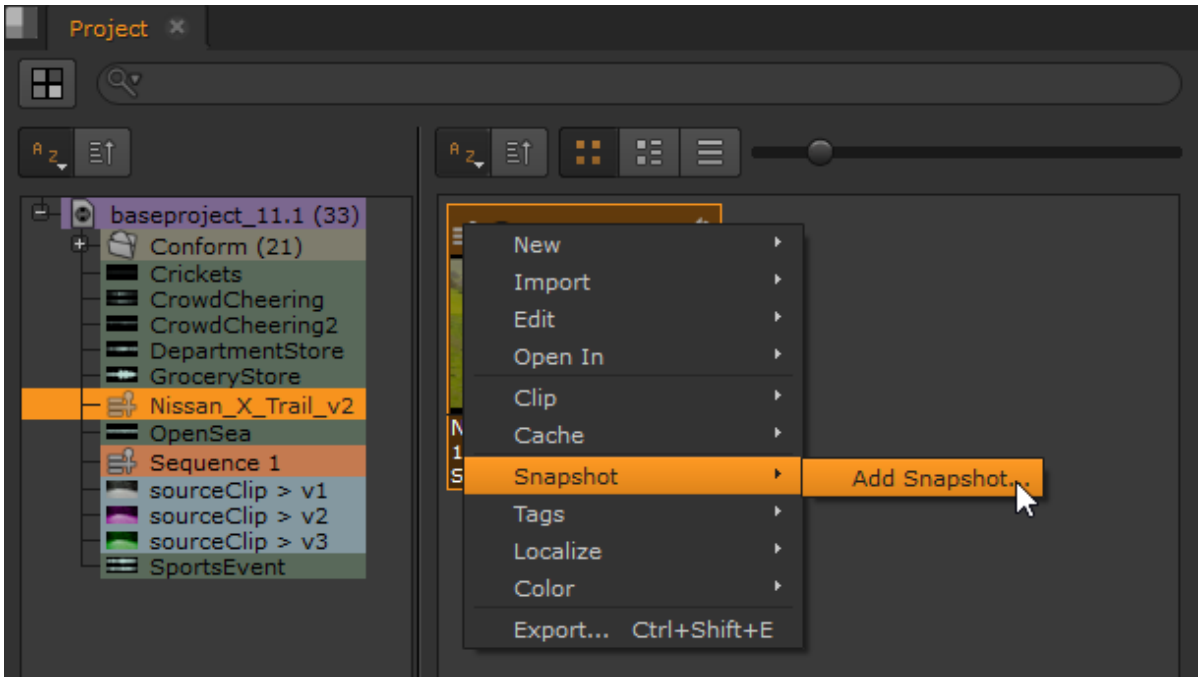


1. Two snapshots of the sequence (SQ) are recorded after edits. See Creating Snapshots.
2. Snapshot 1 is then restored. See Restoring Snapshots.
3. Further edits are made, then the sequence is recorded as snapshot 3.

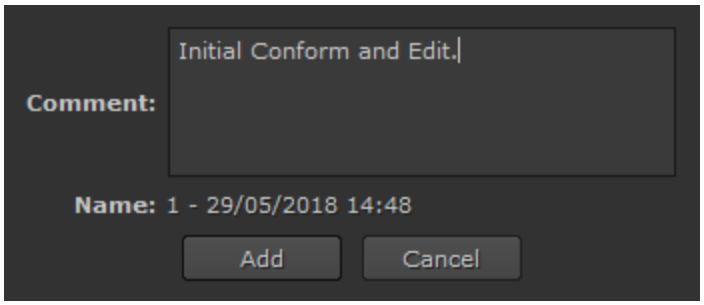
Creating Snapshots

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**.

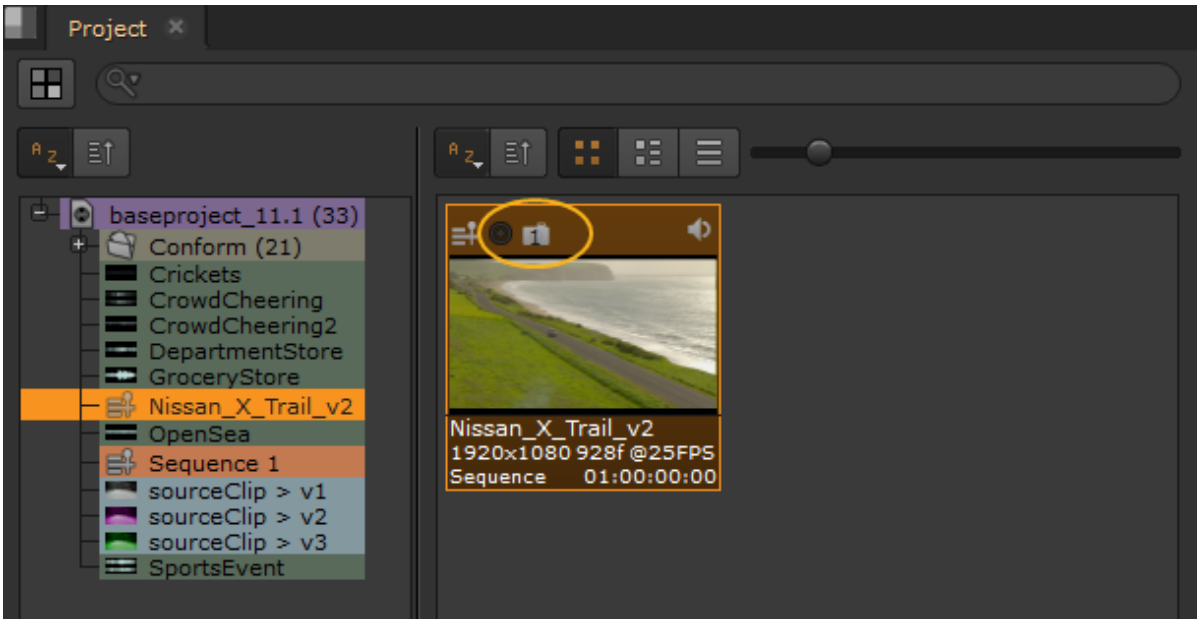


The **Add new snapshot** dialog box displays.



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.

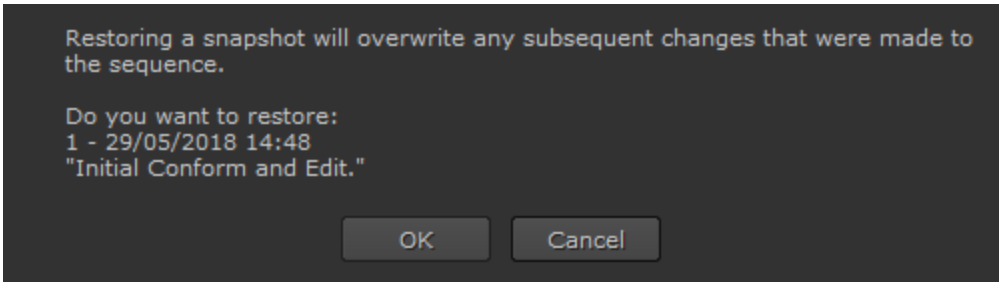

Project
baseproject_11.1 (33)
Conform (21)
Crickets
CrowdCheering
CrowdCheering2
DepartmentStore
GroceryStore
Nissan_X_Trail_v2
OpenSea
Sequence 1
sourceClip > v1
sourceClip > v2
sourceClip > v3
SportsEvent
Nissan_X_Trail_v2
1920x1080 928f @25FPS
Sequence 01:00:00:00

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 Hiero

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 Clips** passes data between Hiero and Nuke.

The export suite can transcode, export clip selections from a timeline or bin, write out EDL, OTIO, and XML files, 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 Clips** sends clips back and forth between Hiero and Nuke using **Local** and **Project Presets**.

Hiero 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 Hiero.
- 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, OTIO, or XML exports.

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: Hiero is non-destructive and can slot into your pipeline if you setup your shot template to mirror the existing file structure.

Round-Tripping Using the Export Dialog

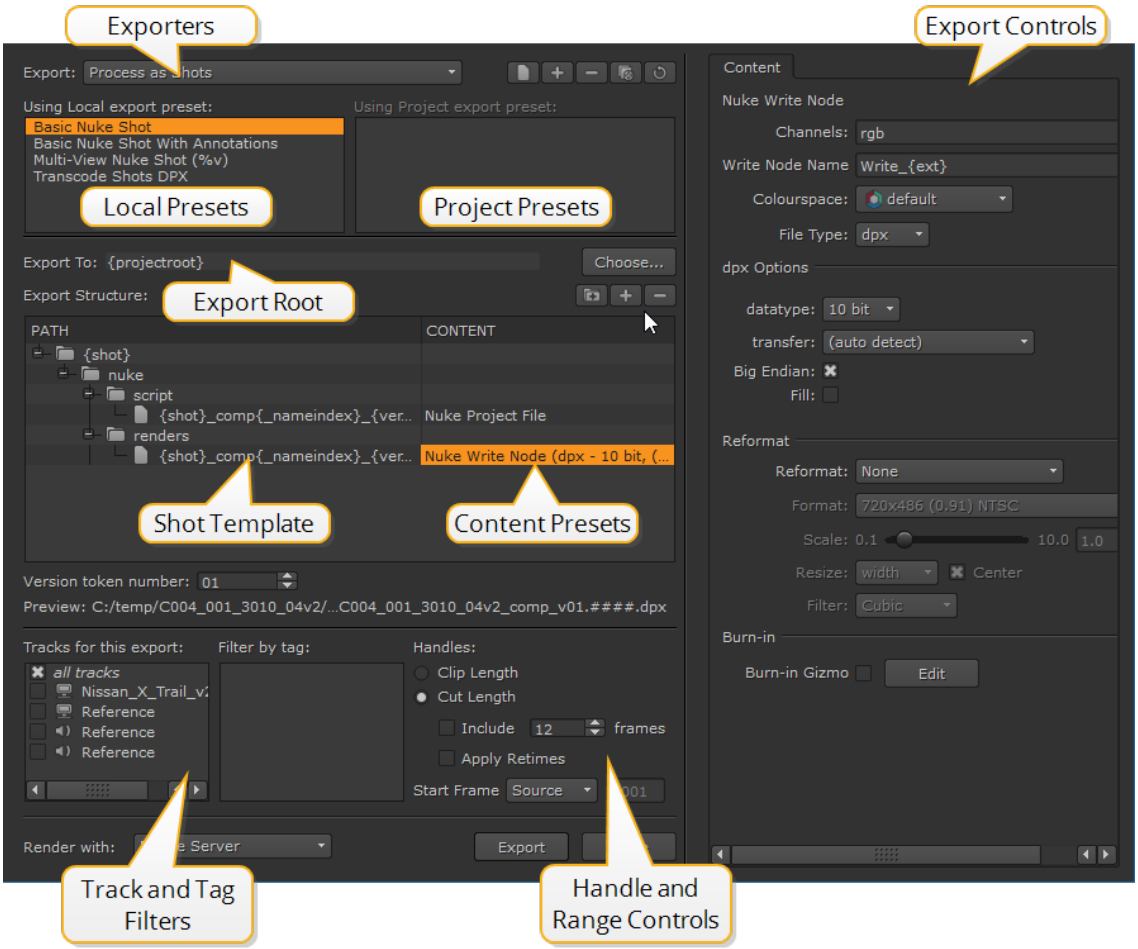
A round-trip describes the process of exporting clips to another application, in this case Nuke, and then paving the way back into Hiero to conform the new VFX clips.

Introduction to the Export Dialog

Hiero uses presets and shot templates to perform export operations, including round-tripping and EDL/OTIO/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 Clips**, passing shots from Hiero and sending rendered Write nodes from the Node Graph back to Hiero.



Hiero 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, Hiero ships with a number of ready-made Content Presets, but you can edit these as required:

- Audio Export - copies any audio tracks to .wav files in a specified location.
- EDL Exporter - used to export a sequence to the EDL format.
- External Render - inserts a placeholder in the template so that other tasks can find the location of external renders.
- Nuke Annotations File - defines the script name and paths used by Nuke Write nodes and Precomp group during a round-trip or Create Comp Clips.
- Nuke Project File - defines the script name and paths used by Nuke Read and Write nodes during a round-trip or Create Comp Clips.
- Nuke Write Node - defines the render format for Nuke Write nodes. Add multiple Nuke Write Node presets to create multiple Write nodes in the resulting Nuke script.
- OTIO Exporter - used to export a sequence to OTIO format.
- SymLink Generator - creates symlinks to the location of the source files, rather than making copies.
- Transcode Images - defines transcode parameters allowing you to save your most-used file type conversions.
- XML Exporter - used to export a sequence to XML format.

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 Nuke. 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 Clips** sends clips back and forth between Hiero and Nuke. 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.

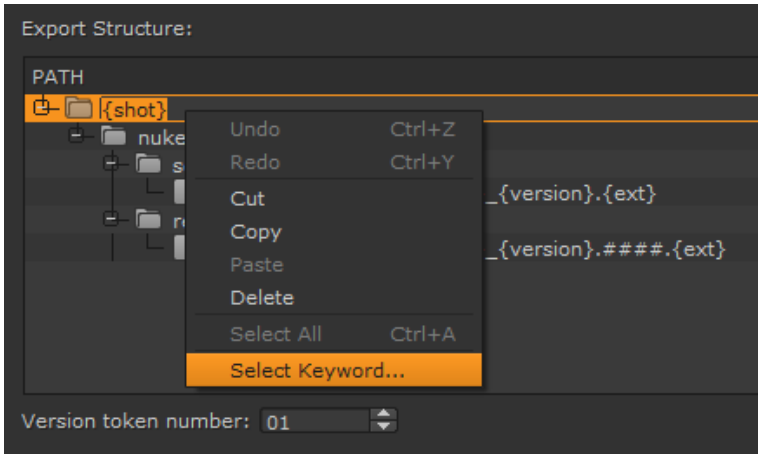
Exports can resolve the following tokens:

Token	Resolves to
_nameindex	The index of the shot name in the sequence, preceded by _ (underscore), to avoid clashes with shots of the same name.
{ampm}	The local equivalent of either AM or PM.
{binpath}	The bin structure to preserve. Including this token recreates your bin structure up to the nearest parent bin.
{clip}	The name of the clip used in the shot processed.

Token	Resolves to
{day}	The local weekday name, abbreviated to Mon, Tue, and so on.
{DD}	The day of the month as a decimal, 01, 02, and so on.
{event}	The timeline event number associated with the shot to process.
{ext}	The extension of the file to output, such as **.dpx** or **.mov**
{filebase}	The base of the clip name to process. For example, the filebase of Shot01_####.dpx is Shot01_####.
{fileext}	The format of the clip to process, such as **.dpx** or **.mov**
{filehead}	The source clip filename not including frame padding or extension. For example, the filehead of Shot01_####.dpx is Shot01.
{filename}	The source clip name of the media to process.
{filepadding}	The source filename padding, which you might use for formatting frame indices.
{filepath}	The full file path to the source media referenced in the export.
{fullbinpath}	The full bin structure to preserve. Including this token recreates the bin structure up to the project level.
{fullday}	The local full weekday name.
{fullmonth}	The local full month name.
{hierotemp}	The temp directory as specified in the Preferences.
{hour12}	The export start time hour component (12-hour clock).
{hour24}	The export start time hour component (24-hour clock).
{MM}	The month of the year as a decimal, 01, 02, and so on.
{minute}	The export start time minute component.
{month}	The local month name, abbreviated to Jan, Feb, and so on.
{project}	The name of the parent project of the export item.
{projectroot}	The root export file path as specified in the Project Settings.

Token	Resolves to
{second}	The export start time second component.
{sequence}	The sequence name to process.
{shot}	The name of the shot to process.
{timestamp}	The export start time in the 24-hour clock format (HHMM).
{track}	The name of the track to process. Exporting EDLs using this token generates a separate EDL for each track.
{user}	The current username.
{version}	The string **v#**, defined by the number (#) set in the Version section of the export dialog
{YY}	The year of the century as a decimal, 01, 02, and so on.
{YYYY}	The year, including century.

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.

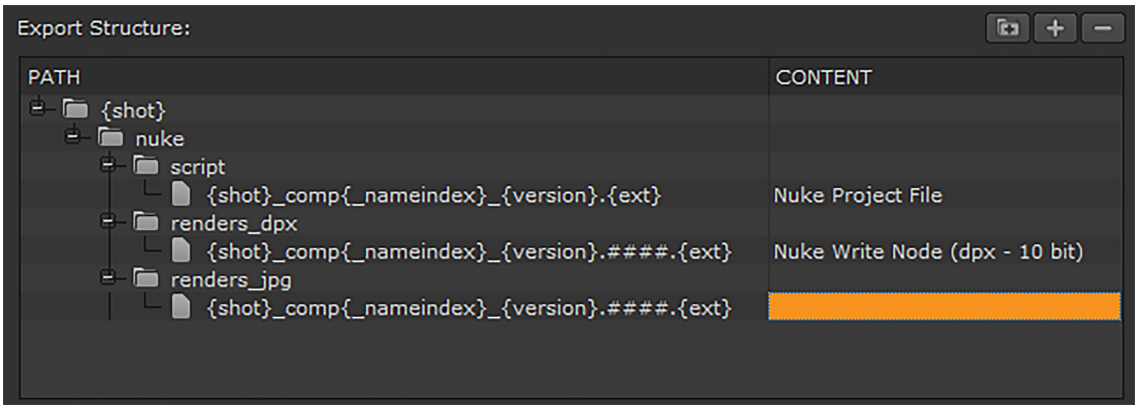
Multi-format Exports

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.

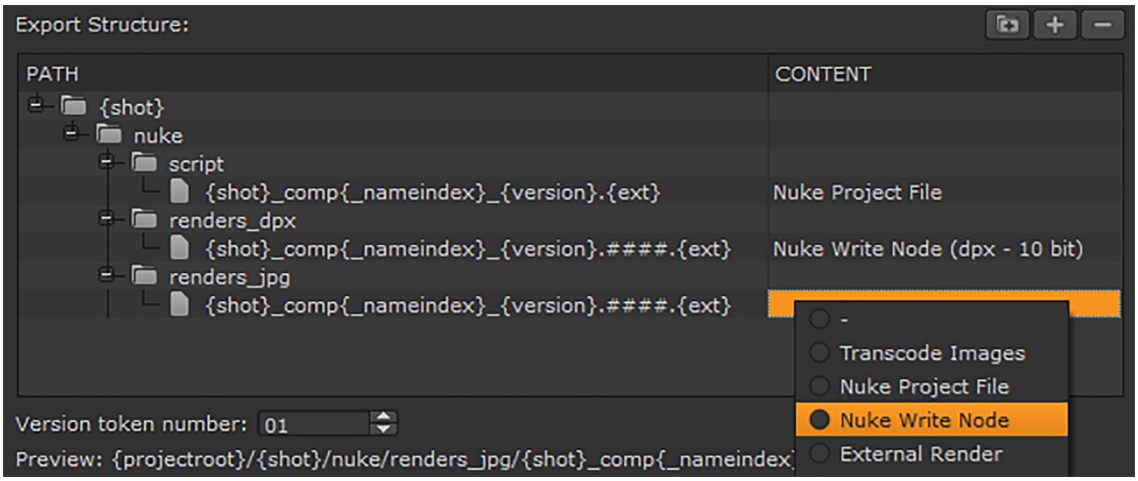
1. In the **Export** dialog, select **Basic Nuke Shot** in the **Local Presets** panel to auto-complete the shot template with the preset values.
2. Click **Duplicate selected preset** and give the new preset a name.
3. Rename the **renders** folder **renders_dpx**.
4. Select the **nuke** folder and 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:

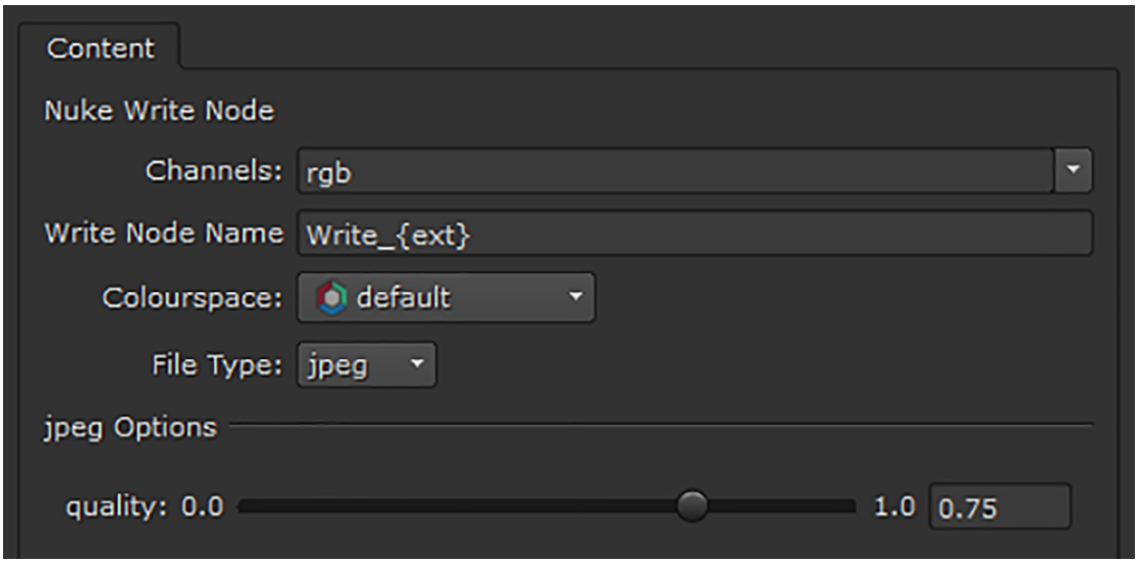


7. Click the **Content** column and select **Nuke Write Node**.



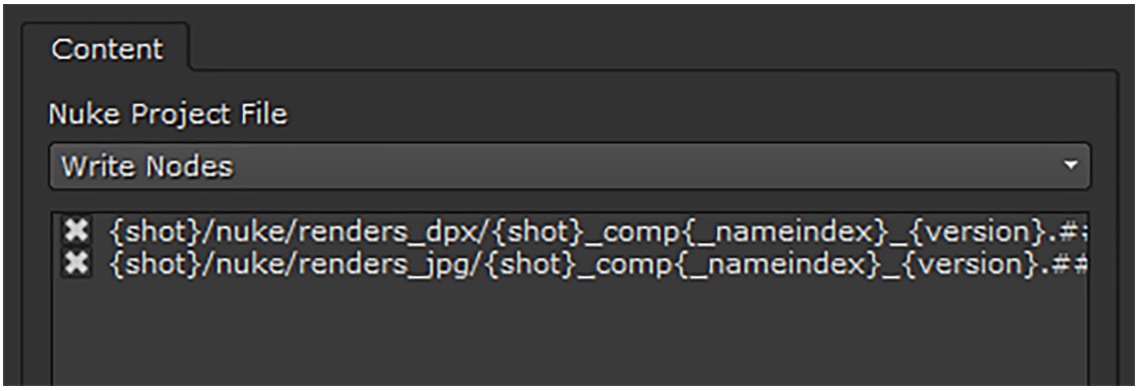
Note: When using a third party application to produce the VFX work, select **ExternalRender** instead of **Nuke Write Node**.

8. In the **Content** settings tab, use the **File Type** dropdown to select **jpeg**.



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**.

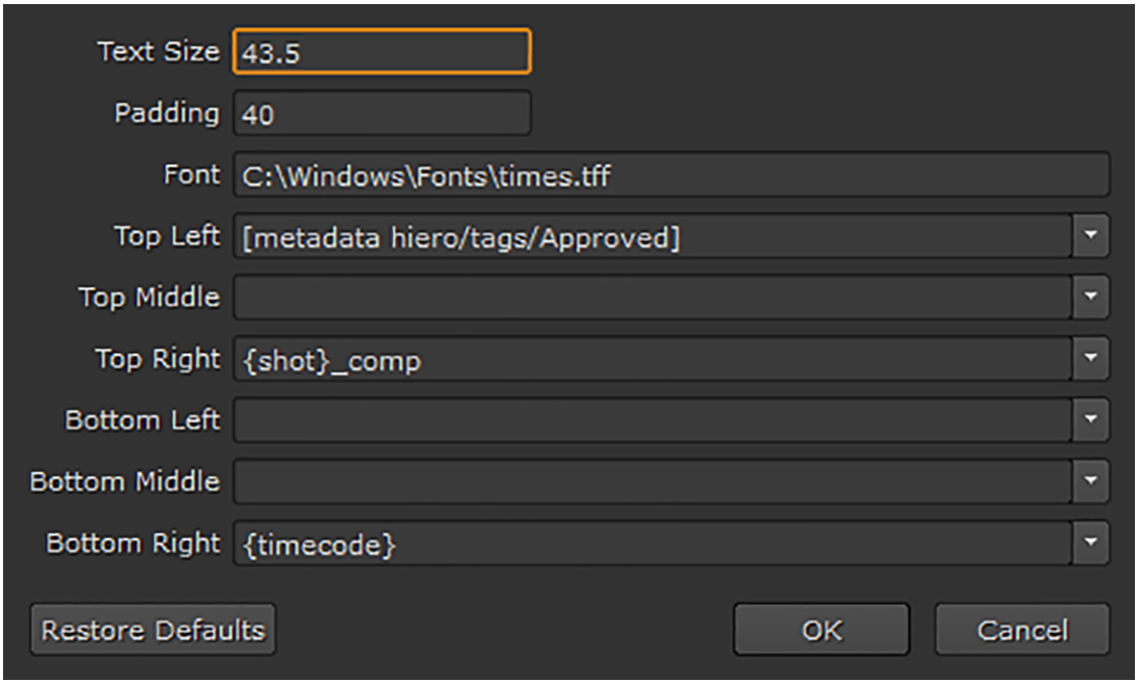
Adding Burn-in Text to Exports

Hiero 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 Adding Effects on the Timeline 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 nam 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/shot/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/shot/tags/Approved/note]
```

Note: You must precede spaces and slashes in the tag name with \\ (backslashes) to enable Hiero to process the tag name correctly. For example: **[metadata hiero/shot/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 Adding Effects on the Timeline for more information.

Adding Additional Nodes During Export

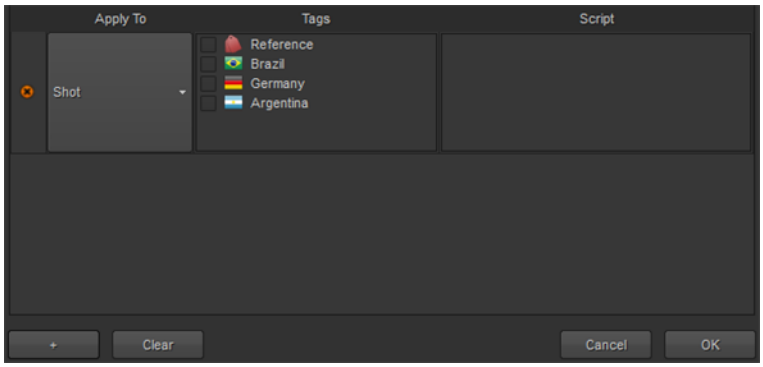
Hiero can include additional nodes, in any **Nuke Project File** or **Transcode** export in the Shot Template, by simply copying and pasting scripts from the Node Graph.

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.

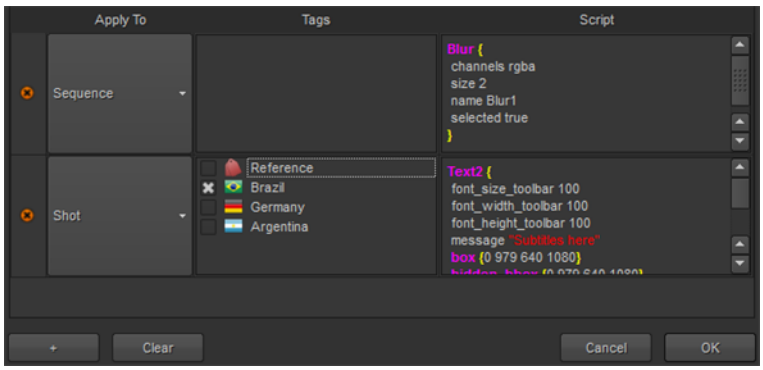
2. Click  to add an entry.



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 Render Machines

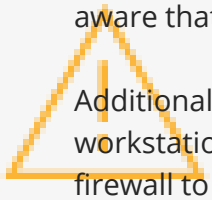
Although Hiero is capable of rendering frames internally, running the Frame Server on a render machine can accelerate the process considerably by sharing work across a network of machines.

Note: The Frame Server requires a Hiero license (hiero_i) on the main workstation, but only a Nuke render license (nuke_r) on the render machines. Local Frame Server processes use ports 5558-5662.

If you want to use an interactive license (nuke_i) on the render machines, add the **--useInteractiveLicense** argument to the **runframeserver.py** command described below.

Configuring the Frame Server on Render Machines

Hiero's Frame Server can be set up on a render machine (or a number of machines) to render from your Hiero session. To do this, you need to run the **runframeserver.py** script on the render machines, found inside the Python site-packages, with specific command line arguments.

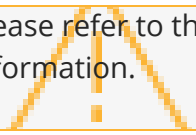


Warning: In order for everything to work smoothly, you need to ensure that both your external render machines and main Hiero session can read and write files to a shared location, such as an NFS share.

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 render 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 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 Hiero 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 application on the render machine.

Tip: On Windows, if there are spaces in the file path, remember to place the path in quotes. For example, **--nukepath="C:\Program Files\Nuke15.0v3\Nuke15.0.exe"**

On a Linux render machine, an example command prompt entry running from the application 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=./Nuke15.0
```

On a Windows render machine, an example command prompt entry running from the application 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=Nuke15.0.exe
```

On a macOS render machine, an example command prompt entry running from the application directory might look like this:

```
./python ../Resources/pythonextensions/site-
packages/foundry/frameserver/nuke/runframeserver.py --numworkers=2 --
nukeworkerthreads=4 --nukeworkermemory=8096 --workerconnecturl=tcp://bob:5560 --
nukepath=/Applications/Nuke12.1v1/Nuke12.1v1.app/Contents/MacOS/Nuke12.1
```

In the examples, we specify that the render machine uses two workers, with four threads and 8 GB RAM each, and are linked to the main Hiero workstation running on **bob**.

Tip: If your render machines run a different OS than your main Hiero 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 render machine file path. Hiero 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 render machine 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:

```
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 render machine, and **bob.local** is the name of the main Hiero 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 to 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.

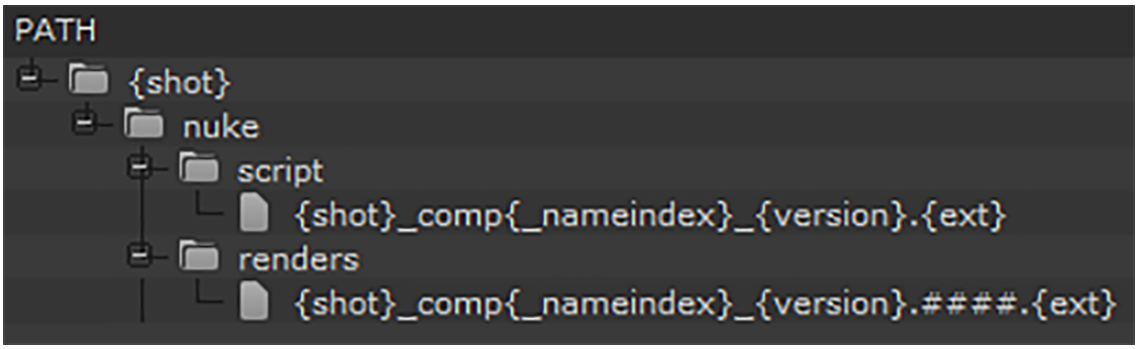
Exporting Sequences and Shots

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.



   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 Hiero expects to find the files when they're rendered.

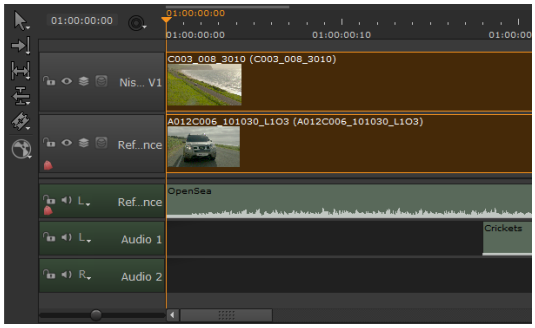
Note: If you included a **Nuke Annotations File** Content preset, enable the **Annotations** Precomp creator. See Annotations for more information.

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 Hiero script by default. If you don't need the soft effects, disable **Include Effects** to omit them from the script. See Adding Effects on the Timeline for more information.
5. Select the required Reformatting 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.

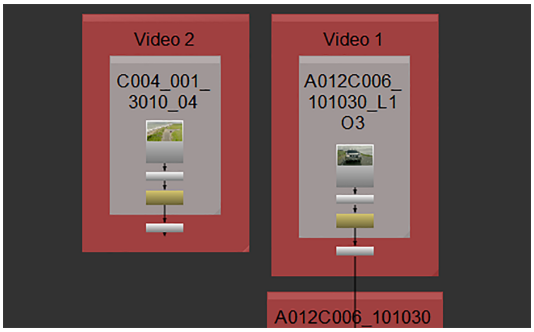
Note: If you have a Read node selected, you can't enable the **Collate** functions.

For example:

• **Collate Shot Timings** - Items on track 1 that would otherwise be hidden by track 2.



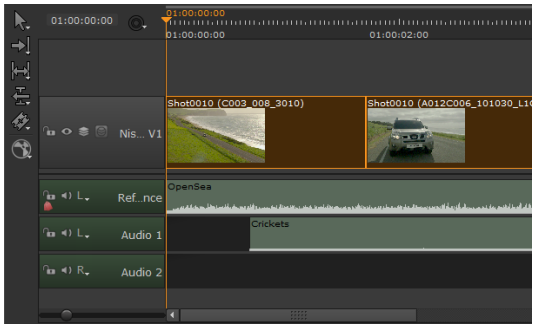
Timeline environment



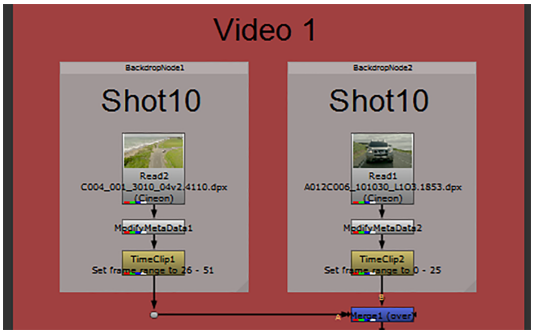
Compositing environment

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



Compositing environment

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 **Hiero Project File** and **Hiero 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 Hiero 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 Hiero 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 controls, allowing you to choose a **Codec**, **Codec Profile** or **Pixel Format**, and in some cases, **YCbCrMatrix**. The matrix control enables you to use the **Rec 601** and **Rec 709** encoding methods.

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 Hiero 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 Hiero 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 the Nuke Online Help for more information.

- **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	To Sequence Resolution	To Scale	Custom
Format - sets the format to render out in Hiero, such as **1920x1080 HD 1080**.			●
Scale - sets the proportion by which to scale the output format.		●	
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.		●	●

Reformat Control	To Sequence Resolution	To Scale	Custom
Filter - sets the filtering algorithm used to transform pixels. See the Nuke Online Help for more information.	●	●	●

6. You can apply text burn-in to the media using a Hiero 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 Hiero.

2. Select **Tracks for this export** by enabling or disabling the tracks in the list. Hiero 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 Clips**, any TimeWarp soft effects are not included in the resulting Nuke script. When the new shot is

created through **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 Hiero should render your export using the **Render with** dropdown. Hiero provides the following options:
   - **Frame Server** - uses multiple Nuke processes to speed up render times, improving resource management.

     See Using the Frame Server on Render 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. Hiero 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 the Nuke Online Help. 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 Hiero, 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 Hiero to build tracks from previous exports. See Building Tracks From Export Structure for more information.

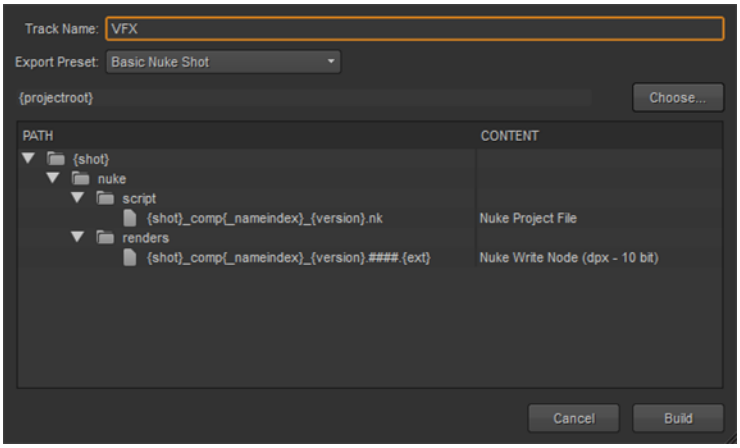
Alternatively, if you have a history of rendered VFX clips, different versions and so on, you can also build tracks from export tags to select from a list of available clips. This method allows you to add **Comp Clips** to the timeline, which act as containers for Hiero **.nk** scripts, or placeholders for Hiero 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.

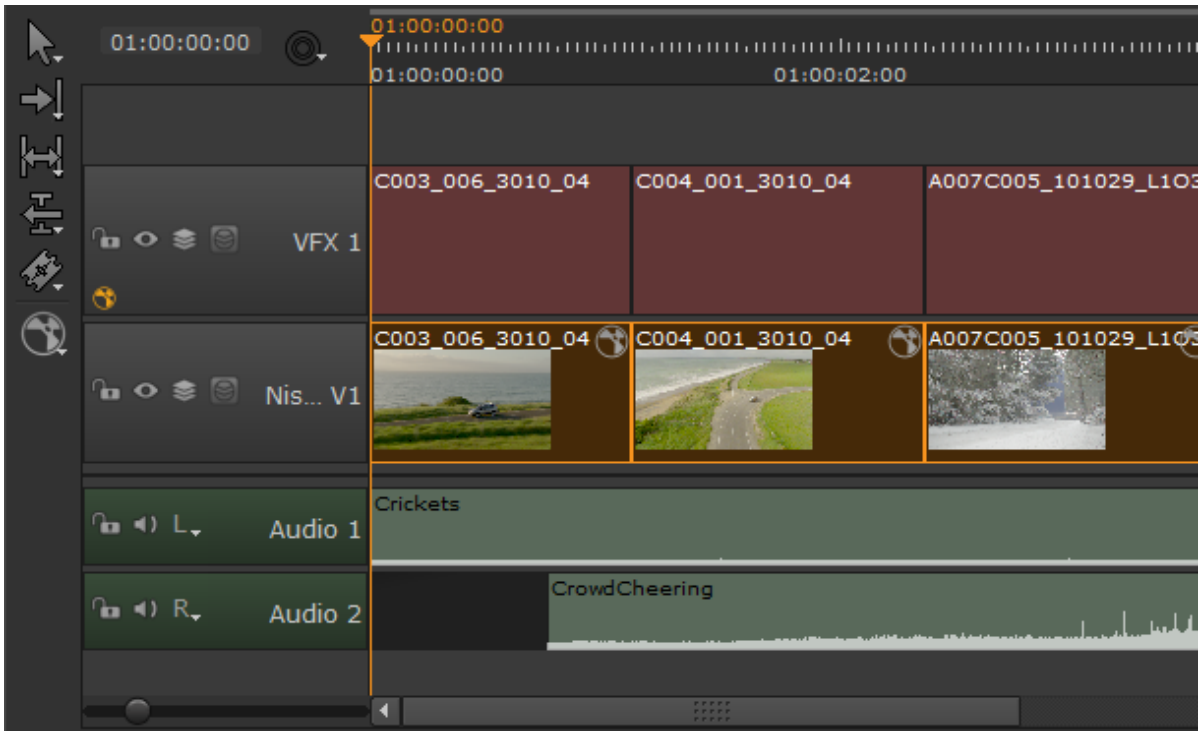
Note: The root directory is the location entered in **Export To** when exporting the project.

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: Hiero warns you if no selection is made in the **Content** column.

Hiero 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, Hiero imports Comp Clips containing the Hiero script, by default. Comp Clips are shots that reference Hiero scripts, rather than being placeholders for offline clips as in the Building Render Placeholders From Export Tags workflow.

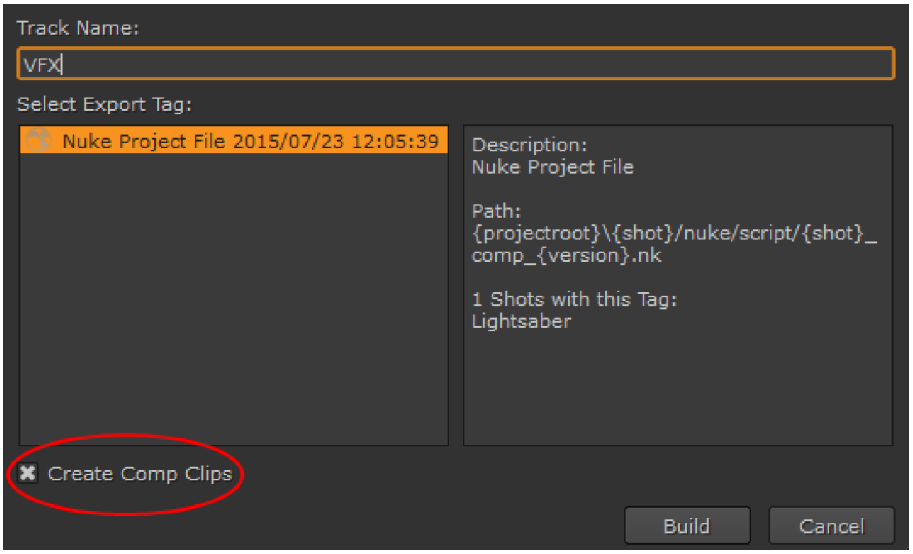
1. Select the required clips on the timeline and right-click to display the context sensitive menu.

> **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**.

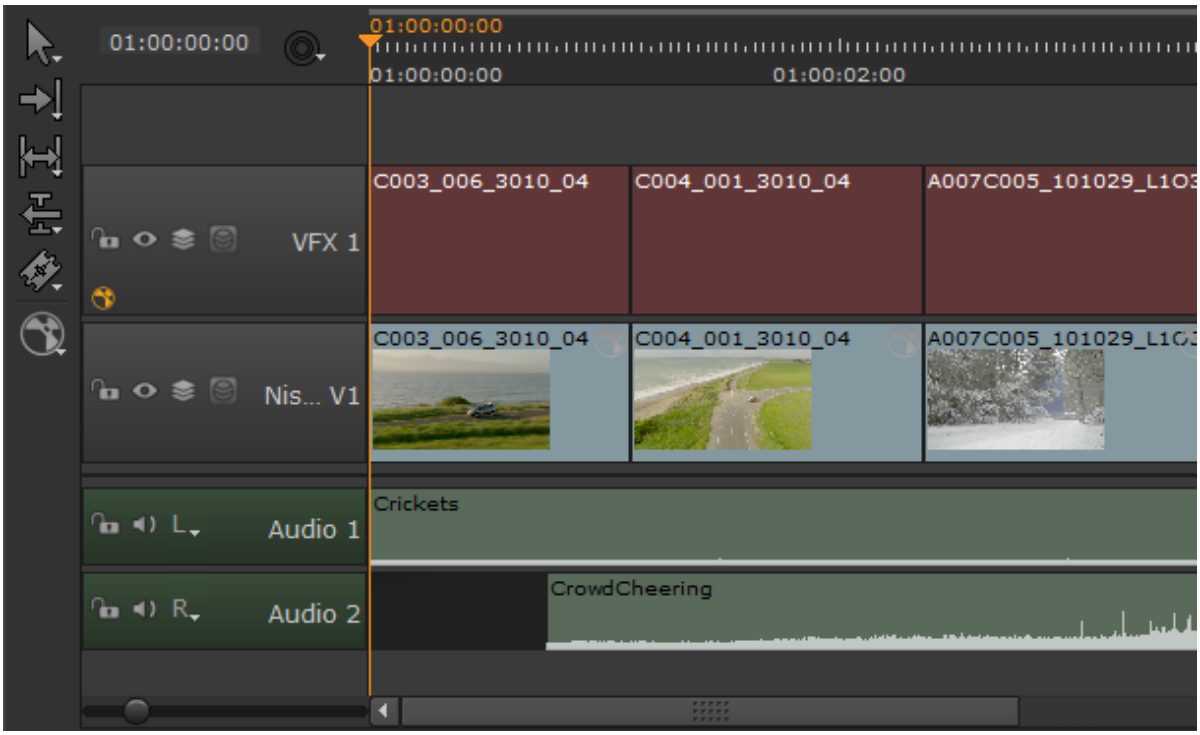
   The **Build Track From Export Tag** dialog displays.
3. Enter a **Track Name** or use the default **VFX**.
4. Select the required export tag in the left-hand panel to display tag information in the right-hand panel.

   Hiero imports the **.nk** Comp Clip by default.



   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.

   Hiero 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 **Ctrl**/**Cmd**+double-click Comp Clips to open them an external Hiero to make edits as required.

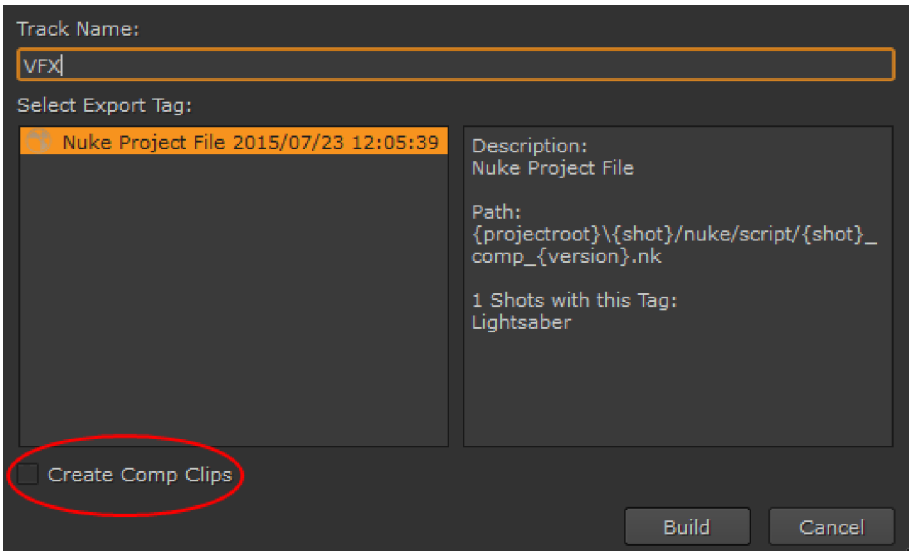
Building Render Placeholders From Export Tags

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 Hiero 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.

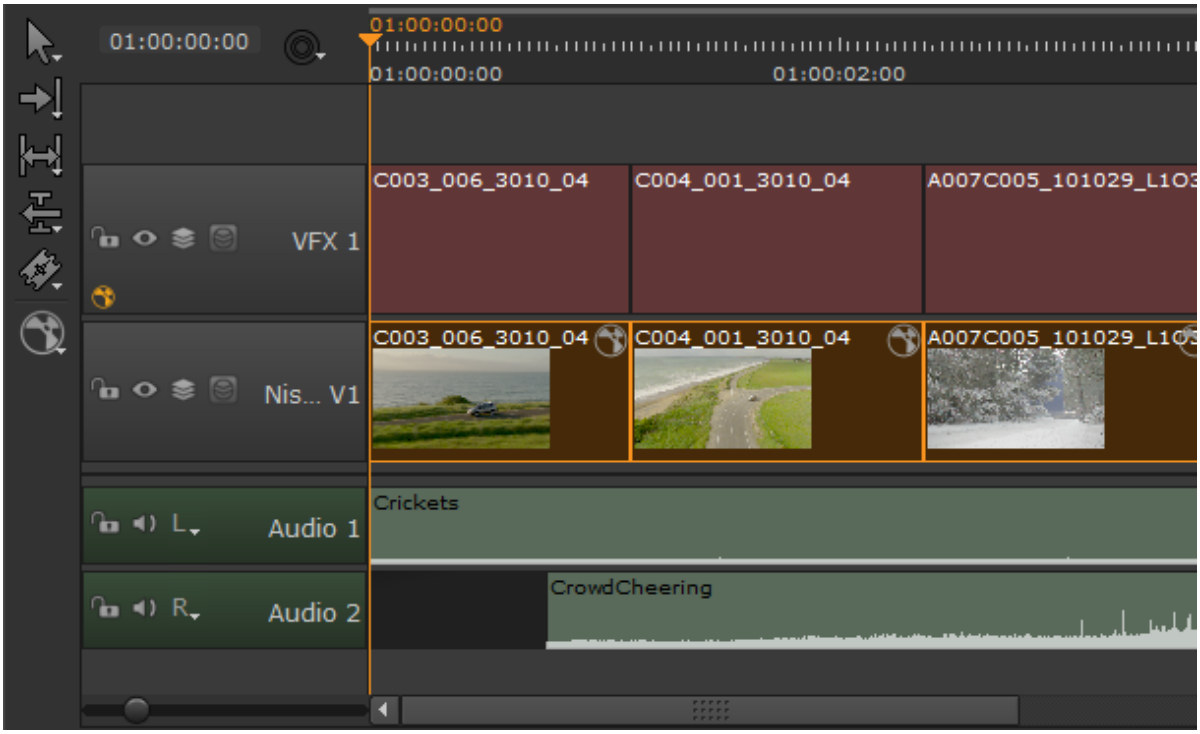


If you want to import the **.nk** Comp Clips, enable the **Create Comp Clips** checkbox. See Building Comp Clips From Export Tags for more information.

6. Click **Build** to create the VFX track.

   Hiero 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.



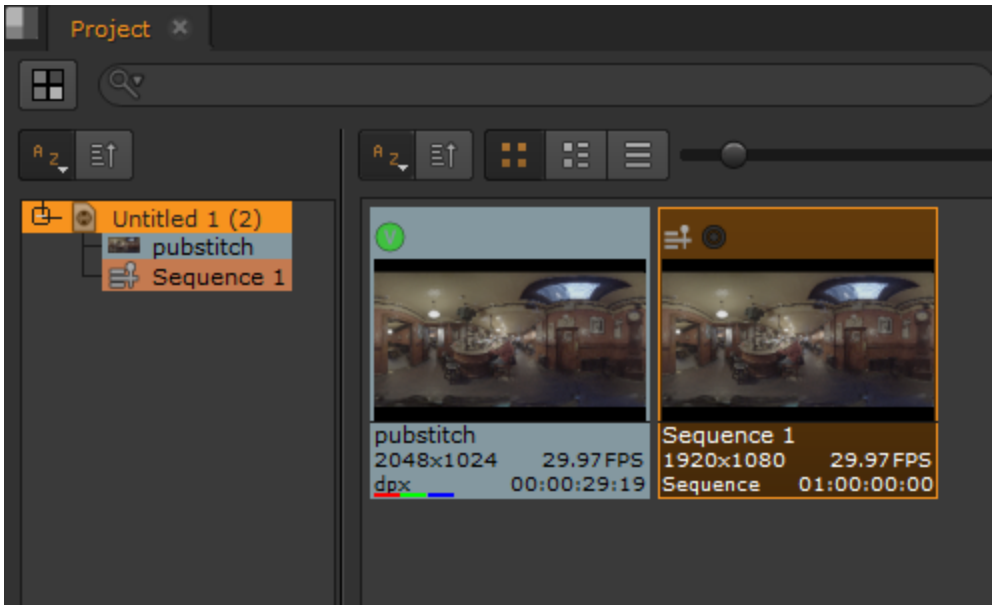
Exporting Multi-View Source Clips

Multi-view exports are similar to regular exports, but the **DPX Multi-View** or **Multi-View Nuke Shot** templates are used to created 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.

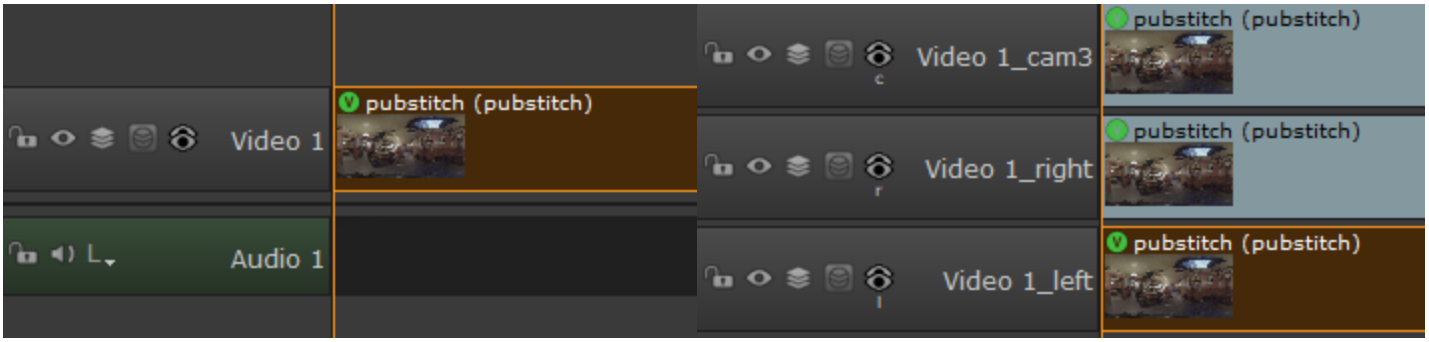
See Stereoscopic and Multi-View Projects for more information on working with multi-view footage.

1. Right-click on the sequence or shot that you want to export.

   Sequences are exported from the bin view.



   Shots can be a multi-view or single views split into separate tracks. See Displaying Views in the Timeline for more information.



   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**.

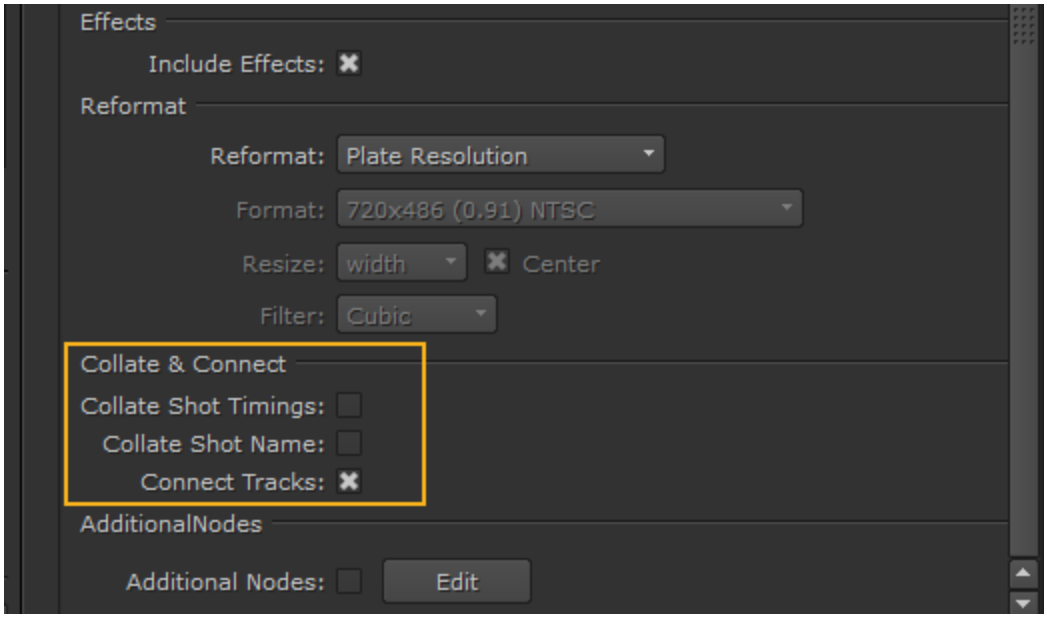
The **Export** dialog is displayed.

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 Hiero, 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 Hiero 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.

   For a full description of the file type options available, see the Write node topic in the Nuke Reference Guide.

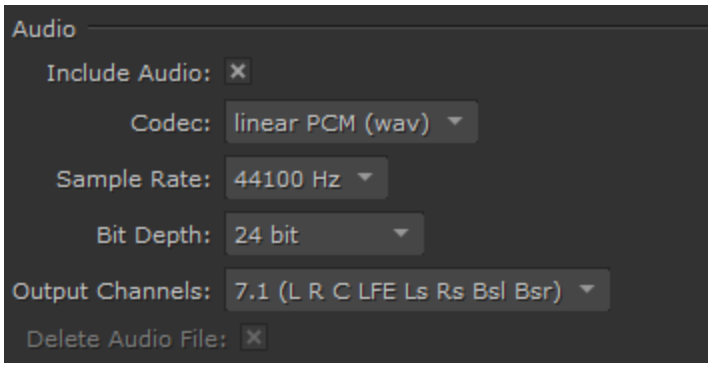
Note: Selecting **mov** from the dropdown provides additional controls, allowing you to choose a **Codec**, **Codec Profile** or **Pixel Format**, and in some cases, **YCbCrMatrix**. The matrix control enables you to use the **Rec 601** and **Rec 709** encoding methods.

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 Hiero 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	To Scale	Custom
Format - sets the format to render out in Hiero, such as **1920x1080 HD 1080**.		●
Scale - sets the proportion by which to scale the output format.	●	
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 the Nuke Online Help for more information on filters.	●	●

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 Hiero script by default. If you don't need the soft effects, disable **Include Effects** to omit them from the script. See Adding Effects on the Timeline for more information.
   - **Include Audio** - when enabled, any audio tracks are exported alongside the video.



- **Delete Audio 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 **.dpx** 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 Hiero.

2. Select **Tracks for this export** by enabling or disabling the tracks in the list. Hiero 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 Hiero renders your export using the **Render with** dropdown. The following options are available:
   - **Frame Server** - uses multiple Hiero processes to speed up render times, improving resource management.
   - **Single Render Process** - uses a single Hiero 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. Hiero 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 the Nuke Online Help. 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.

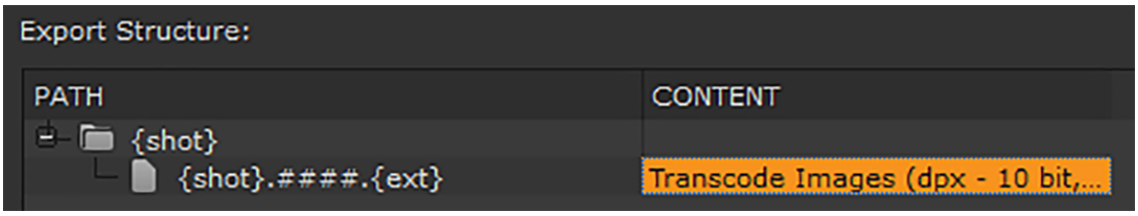
Transcoding a Sequence as Shots

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:



   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: SeeUsing Versions for more information on how versioning works in Hiero.

11. Set how Hiero should render your export using the **Render with** dropdown. Hiero provides the following options:
    - **Frame Server** - uses multiple Nuke processes to speed up render times, improving resource management.

      See Using the Frame Server on Render 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. Hiero 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 the Nuke Online Help. 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.

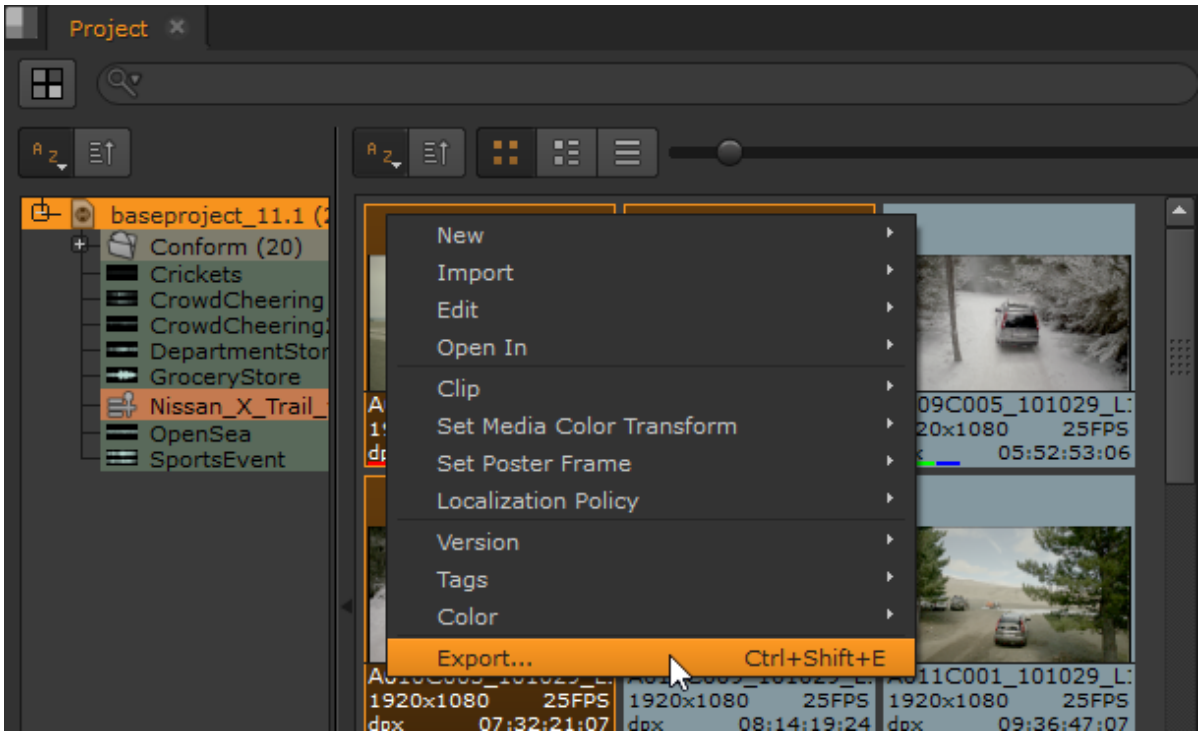
    Once the export is complete, the file structure specified in the shot template is created. You can then import the Nuke clips on a separate track when they're ready.

Tip: Click the magnifying glass icon to reveal the file structure in a browser window.

Transcoding 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.

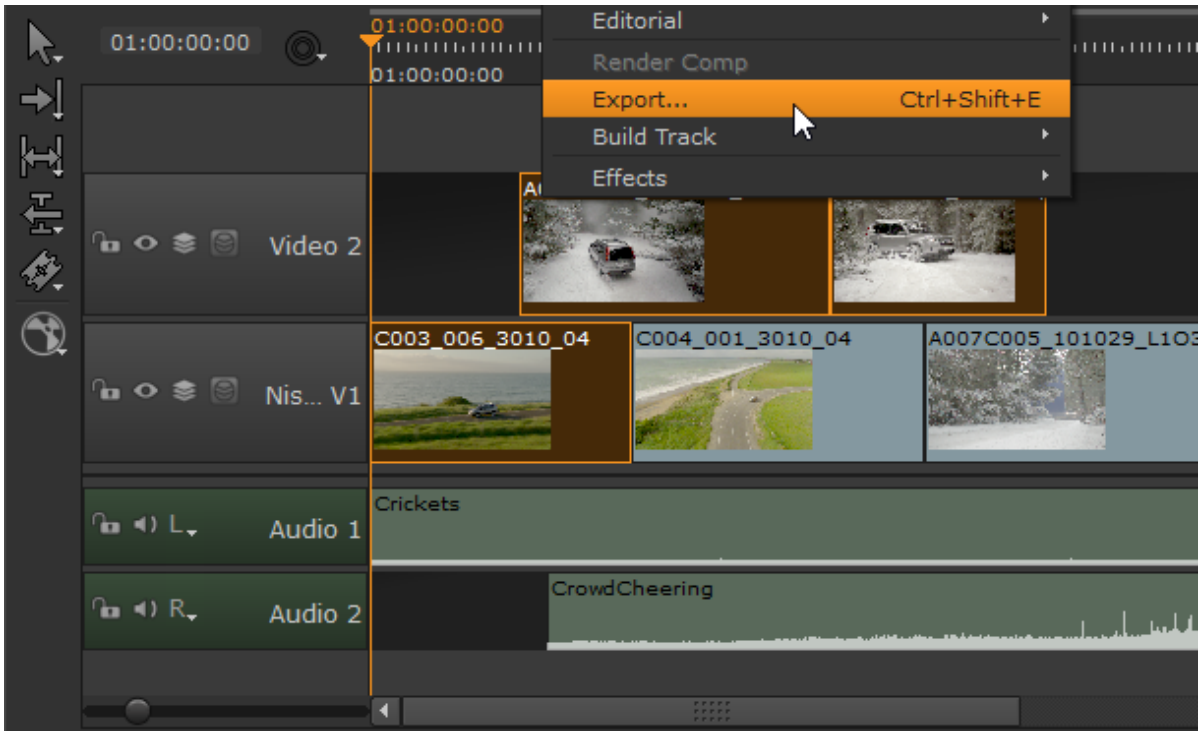
3. Select **Process as Clips** and modify the shot template, if required.
4. Follow the steps under Transcoding a Sequence to complete the export.

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.

Ad Hoc Exports

This section covers exports that you might not perform on a per project basis, such the EDL, OTIO, or XML Exporters and Copy Exporter. Exporters are available for sequences, shots, and clips as described in the following table.

Exporter	Source		
	Sequences	Shots	Source Clips
EDL Exporter	●		
OTIO Exporter	●		
XML Exporter	●		
Audio Exporter	●	●	●
Copy Exporter		●	●
SymLink Exporter		●	●

Exporting Sequences as EDL, OTIO, and XML

Hiero supports export to EDL, OTIO, and XML using very similar methods, the main difference being that EDL doesn't support multiple video tracks in a single file whereas OTIO and XML does.

Note: Hiero can read AAF files, but not write them out.

To export to EDL, OTIO, 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, OTIO (Beta), 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.

Note: OTIO does not currently support any custom export options.

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 Hiero.

7. Click the **Tracks and Range** tab and select the **Tracks For This Export** by enabling or disabling the tracks in the list. Hiero exports **all tracks** by default.
8. If you set in and out point on the sequence, enable **In/Out Points** to export only the selected frames.
9. 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. Set how Hiero should render your export using the **Render with** dropdown. Hiero provides the following options:
    - **Frame Server** - uses multiple Nuke processes to speed up render times, 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. Hiero 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 the *Nuke User Guide*, *Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts* section. 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.

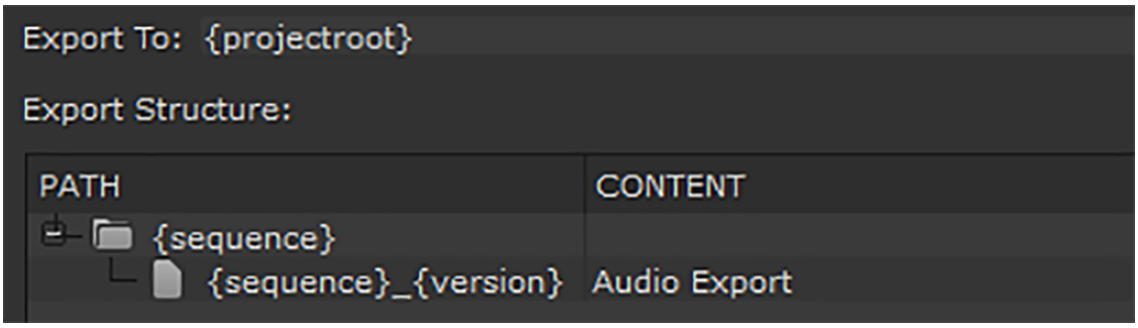
Exporting Audio from Sequences

1. Select a sequence in the bin view and navigate to **File** > **Export...**

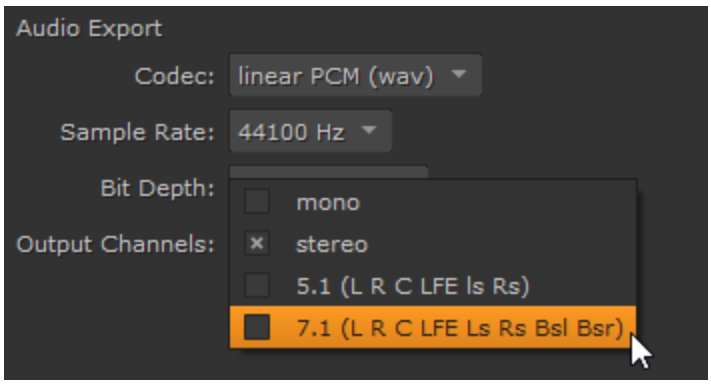
   The **Export** dialog displays.
2. Select **Process as Sequence** from the **Export** dropdown and select a preset, duplicate it, and give it a name.
3. Enter the **Export To** directory or click **Choose...** and browse to the location.

   The **Export To** directory is the starting point from which the shot template builds your shot hierarchy.
4. Build a custom shot template using **Path** tokens and the folder and +/- buttons and then click the **Content** field and select **Audio Export** from the list.

   An example shot template is shown below:



5. Click the **Audio Export** field and select the required export options, including the **Output Channels** for multi-channel audio.



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 Hiero.

7. Select **Tracks for this export** by enabling or disabling the tracks in the list. Hiero 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 Hiero should render your export using the **Render with** dropdown. Hiero provides the following options:
   - **Frame Server** - uses multiple Nuke processes to speed up render times, improving resource management.

     See Using the Frame Server on Render 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. Hiero 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 the Nuke Online Help. 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

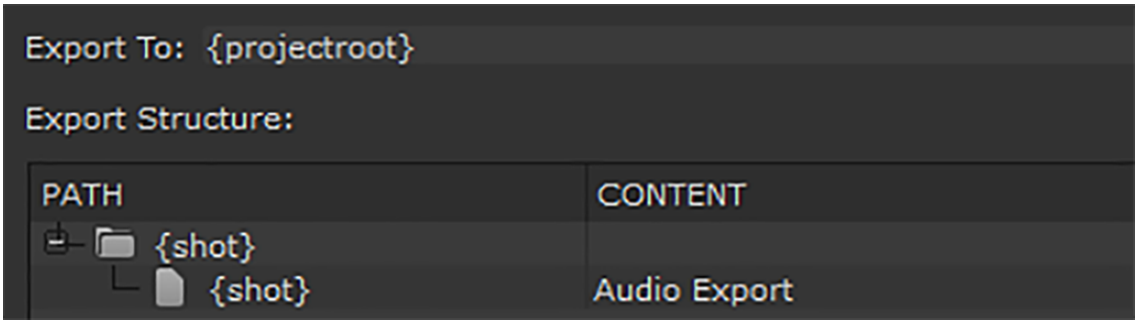
1. Select the required shots in the timeline and navigate to **File** > **Export...**

The **Export** dialog displays.

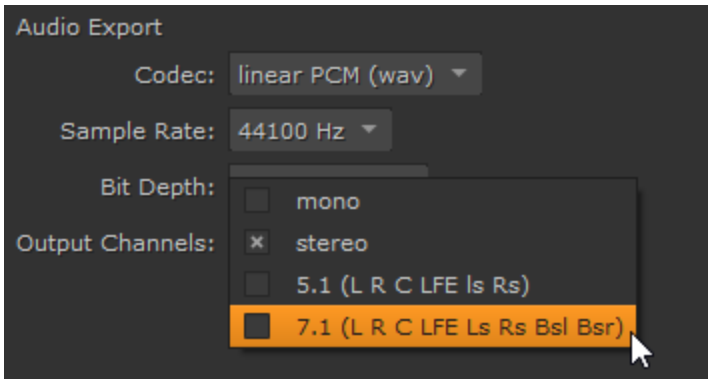
2. Select **Process as Shots** from the **Export** dropdown and select a preset, duplicate it, and give it a name.
3. Enter the **Export To** directory or click **Choose...** and browse to the location.

   The **Export To** directory is the starting point from which the shot template builds your shot hierarchy.
4. Build a custom shot template using **Path** tokens, the **Contents** field, and the folder and +/- buttons.

   An example shot template is shown below:



5. Click the **Audio Export** field and select the required export options, including the **Output Channels** for multi-channel audio.



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: SeeUsing Versions for more information on how versioning works in Hiero.

7. Select **Tracks for this export** by enabling or disabling the tracks in the list. Hiero 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 Hiero should render your export using the **Render with** dropdown. Hiero provides the following options:
    - **Frame Server** - uses multiple Nuke processes to speed up render times, improving resource management.

      See Using the Frame Server on Render 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. Hiero 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 the Nuke Online Help. 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.

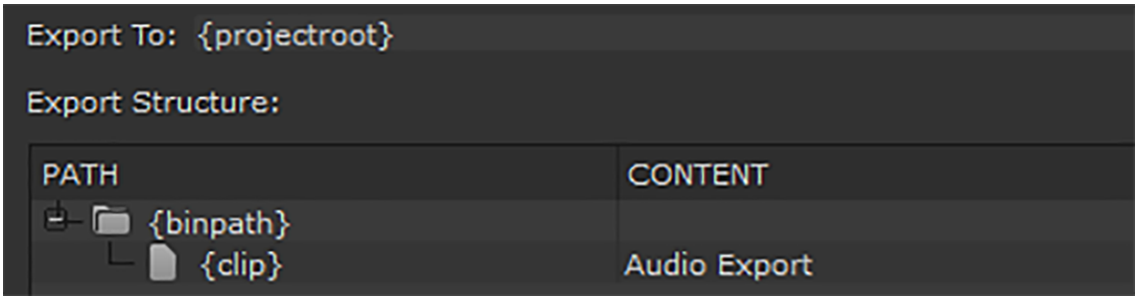
Exporting Audio from Source Clips

1. Select the required source clips and navigate to **File** > **Export...**

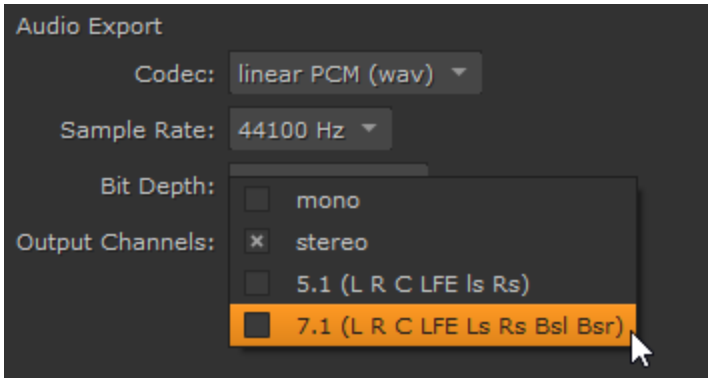
   The **Export** dialog displays.
2. **Process as Clips** is selected automatically from the Export list, so select a preset, duplicate it, and give it a name.
3. Enter the **Export To** directory or click **Choose...** and browse to the location.

   The **Export To** directory is the starting point from which the shot template builds your shot hierarchy.
4. Build a custom shot template using **Path** tokens, the **Contents** field, and the folder and +/- buttons.

   An example shot template is shown below:



5. Click the **Audio Export** field and select the required export options, including the **Output Channels** for multi-channel audio.



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 Hiero.

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 Hiero should render your export using the **Render with** dropdown. Hiero provides the following options:
   - **Frame Server** - uses multiple Nuke processes to speed up render times, improving resource management.

     See Using the Frame Server on Render 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. Hiero 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 the Nuke Online Help. 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.

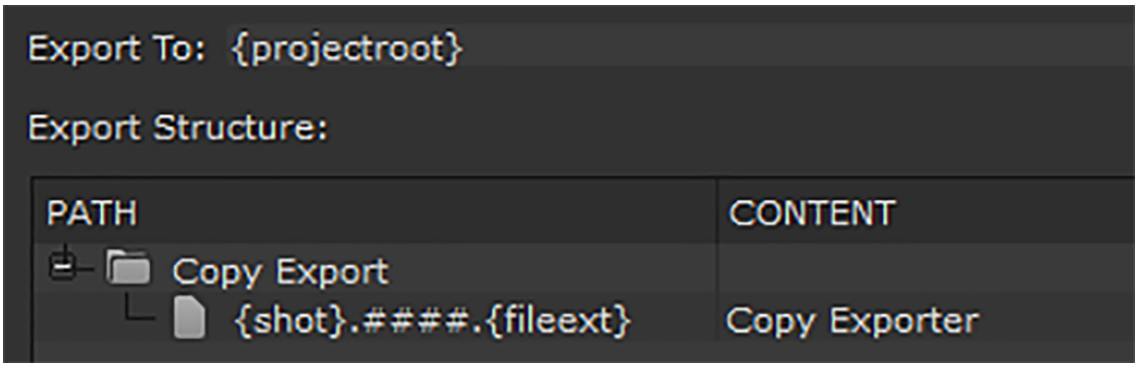
To copy media to a named location:

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:



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 Hiero.

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 Hiero should render your export using the **Render with** dropdown. Hiero provides the following options:

- **Frame Server** - uses multiple Nuke processes to speed up render times, improving resource management.

  See Using the Frame Server on Render 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. Hiero 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 the Nuke Online Help. 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.

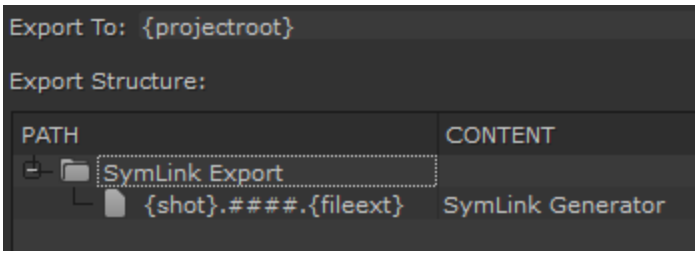
To create symbolic links to a named location:

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:



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 Hiero.

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 Hiero should render your export using the **Render with** dropdown. Hiero provides the following options:
   - **Frame Server** - uses multiple Nuke processes to speed up render times, improving resource management.

     See Using the Frame Server on Render 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. Hiero 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 the Nuke Online Help. 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.

Customizing Hiero

These pages provide a basic overview and some simple examples to get you started, but more information is available in the dedicated Python API document included in the Hiero package. See **Help** > **Python Dev Guide** for more information.

Environment Variables

Environment variables are used to store values that influence the behavior of Hiero and HieroPlayer, such as adding additional file paths to check at startup for plug-in scripts.

Setting Environment Variables

The process of setting environment variables differs depending on the operating system you're using:

On Mac and Linux

1. The first step is to determine the type of the shell you're using. Launch a terminal and enter:
   `echo $SHELL`
   Your shell type is printed to the screen. For example, `/bin/bash` indicates that your shell is "bash".
2. Depending on the output of the $SHELL command, do one of the following:
   - **csh** or **tcsh** - 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 HIERO_PLUGIN_PATH /SharedDisk/Hiero
     setenv TIMELINE_DISABLE_PBO_UPLOADS 1
     ```
   - **bash** or **ksh** - 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 HIERO_PLUGIN_PATH=/SharedDisk/Hiero
     export TIMELINE_DISABLE_PBO_UPLOADS=1
     ```

Note: These steps set the environment variable until you manually remove it from the file in your home directory. If you want to set a variable for a single session, you can simply type the variable in the terminal then run Hiero from the same session.

On Windows

1. Right-click on **My Computer** and select **Properties**.
2. Click 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.

Note: You must have administrator privileges to set system-wide variables.

5. In the **Variable** name field, enter the name of the environment variable you want to set.
6. In the **Value** field, enter the value for the variable. The value can be a directory path, for example. You can also disable and enable variables by adding the value 0 or 1, respectively.
7. Click **OK** to save the variable.

Environment Variables

Environment variables are used to store values that influence the behavior of Hiero and HieroPlayer, such as adding additional file paths to check at startup for plug-in scripts.

Setting Environment Variables

The process of setting environment variables differs depending on the operating system you're using:

On Mac and Linux

1. The first step is to determine the type of the shell you're using. Launch a terminal and enter:
   `echo $SHELL`
   Your shell type is printed to the screen. For example, `/bin/bash` indicates that your shell is "bash".
2. Depending on the output of the $SHELL command, do one of the following:

- **csh** or **tcsh** - 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 HIERO_PLUGIN_PATH /SharedDisk/Hiero
  setenv TIMELINE_DISABLE_PBO_UPLOADS 1
  ```

- **bash** or **ksh** - 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 HIERO_PLUGIN_PATH=/SharedDisk/Hiero
  export TIMELINE_DISABLE_PBO_UPLOADS=1
  ```

Note: These steps set the environment variable until you manually remove it from the file in your home directory. If you want to set a variable for a single session, you can simply type the variable in the terminal then run Hiero from the same session.

On Windows

1. Right-click on **My Computer** and select **Properties**.
2. Click 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.

Note: You must have administrator privileges to set system-wide variables.

5. In the **Variable** name field, enter the name of the environment variable you want to set.
6. In the **Value** field, enter the value for the variable. The value can be a directory path, for example. You can also disable and enable variables by adding the value 0 or 1, respectively.
7. Click **OK** to save the variable.

Hiero Environment Variables

The following table lists the environment variables Hiero recognizes.

Environment Variable	Description
__GL_SYNC_DISPLAY_DEVICE	On Linux, set this variable to the name of screen device to synchronize in dual screen setups. See Synching to VBlank on Linux for more information.
FN_CRASH_DUMP_PATH	Allows you to specify where Issue Reporter dumps are saved by default.
FN_DISABLE_LICENSE_DIALOG or FN_NUKE_DISABLE_TMPLIC_NOTIFY_DIALOG	By default, if you have installed a temporary license, Hiero 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. **Note:** You still get a warning if no license is found, for example if you only have a HieroPlayer license but you try to run Hiero.
FN_LICENSE_DIALOG_DAYS_LEFT_BEFORE_PROMPT	By default, if you have installed a temporary license, Hiero 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. **Note:** You still get a warning if no license is found, for example if you only have a HieroPlayer license but you try to run Hiero.
FN_SUBSCRIPTION_LICENSE_DIR	On Windows, user names containing non-ASCII characters can cause subscription licensing to fail. If a licensing error similar to the following displays: **Unable to create subscription license directory: C:\Users\Zoë Hernández\FoundryLicensing**

Environment Variable	Description
	Try changing the license directory to an alternate location using this environment variable.
foundry_LICENSE	The location of the Hiero RLM license file, if the following recommended location is not used: **On Mac and Linux**: `/usr/local/foundry/RLM` **On Windows:** *`drive letter`*`:\Program Files\The Foundry\RLM` **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.
FOUNDRY_LICENSE_ DEBUG	This variable prints additional licensing information to the command line or Terminal.
FOUNDRY_LICENSE_FILE	The location of the Hiero FLEXlm license file, if the following recommended location is not used: **On Mac and Linux**: `/usr/local/foundry/FLEXlm` **On Windows:** *`drive letter`*`:\Program Files\The Foundry\FLEXlm` **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.
FOUNDRY_LOG_FILE	This variable specifies the location of Hiero's logfile. If you don't specify a logfile, all output is to screen.
FOUNDRY_LOG_LEVEL	This variable sets the level of logging Hiero produces during operation. There are four levels of detail, on a sliding scale from minimal to verbose:

Environment Variable	Description
	• **error** • **warning** • **message** • **verbose** **Note:** Setting the logging level to verbose can produce large log files when FOUNDRY_LOG_FILE is specified.
FRAMESERVER_LOG_DIR	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. See Using the Frame Server on Render Machines for more information.
HIERO_DISABLE_THUMBNAILS	Set this variable to stop Hiero loading thumbnails.
HIERO_DISABLE_THUMBNAILS_CACHE	Set this variable to stop Hiero caching thumbnails for improved access once loaded. **Note:** This variable does not clear the cache, you must remove cached files manually.
HIERO_PLUGIN_PATH	Set this variable to the location of additional scripts to run at startup. You can use **.<folder>** to point to hidden folders, for example **.myPlugins**. For example, you could create a facility-wide location to share presets and use this variable to point multiple Hiero's to it. See Sharing Presets for more information. Use the following method to display set paths: `print hiero.core.pluginPath()`
HIERO_SINGLE_THREADED_PLAYBACK	This variable launches Hiero in single-thread mode, which can solve playback issues on various Linux Fedora distributions.

Environment Variable	Description
NUKE_AJA_CHANNEL	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: • **NUKE_AJA_CHANNEL** - set this variable to 2, 3, or 4 to output a single stream through SDI2, SDI3, or SDI4. • **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. Combining these two environment variables can force the stream to split and output through alternate SDI outputs. For example: • DUALOUTPUT + CHANNEL=1 OR CHANNEL=2 results in two 1.5G streams coming from SDI1 and SDI2. • DUALOUTPUT + CHANNEL=3 OR CHANNEL=4 results in two 1.5G streams coming from SDI3 and SDI4. **Note:** Certain modes, such as 12-bit 444, require a 3G stream. Otherwise, the card uses the single stream on the channel number specified.
NUKE_AJA_DUALOUTPUT	
NUKE_ALLOW_GIZMO_SAVING	Nuke does not allow you to **Overwrite** and **Save as** 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.
NUKE_CRASH_HANDLING	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. When NUKE_CRASH_HANDLING is set to 1, crash reporting is enabled in both GUI and terminal mode. When NUKE_CRASH_HANDLING is set to 0, crash reporting is disabled in both GUI and terminal mode.
NUKE_DISK_CACHE	The location where Hiero saves all recent images displayed in the

Environment Variable	Description
	Viewer. Ideally, this should be a local disk with the fastest access time available.
NUKE_DISK_CACHE_GB	The maximum size the disk cache can reach (in gigabytes).
NUKE_EXR_TEMP_DIR	On Linux, this is the location Hiero uses for temporary files while reading PIZ-compressed **.exr** files. This environment variable is only relevant on Linux. If this variable is not set, the location is determined by NUKE_TEMP_DIR.
NUKE_EXR_TEMP_NAME	Changes the naming convention of **.exr** temporary files during rendering. Setting the variable to **1** writes temporary **.exr** files as **<filename>.exr.tmp**, rather than **<filehash>.tmp** as in previous releases.
NUKE_FONT_PATH	The location that Hiero checks for available font files when the Text node properties panel is opened.
NUKE_LEGACY_CHANNEL_SORTING	This variable disables the new channel sorting behavior, where the RGBA layer is sorted first. Enabling this variable causes Hiero to sort channels alphabetically.
NUKE_LOCALIZATION_NUMWATCHERS	Controls the number of threads available for localization tasks. Increasing the number of threads can improve localization performance.
NUKE_MOV64READER_ENABLE	Set this variable to 0 to disable Hiero's 64-bit mov decoding and fall back to 32-bit decoding.
NUKE_NO_CRASH_PROMPT	When crash handling is enabled in GUI mode, this allows you to control whether reports are automatically submitted or not: When NUKE_NO_CRASH_PROMPT is set to 1, crash reports are submitted automatically without displaying a crash reporter dialog. When NUKE_NO_CRASH_PROMPT is set to 0, Hiero always displays a crash reporter dialog before submitting a crash report.
NUKE_TEMP_DIR	The location where Hiero saves any temporary files that do not have a

Environment Variable	Description
	particular place defined for them. 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`
NUKE_ WINDOWMANAGER_ DEBUG	When enabled, data from Hiero's window manager is printed to the command line or Terminal.
OCIO	Set this variable to the location of your OCIO configuration file for color conversion.  **Note:** If you plan to use the OCIO config file specified in the Preferences, ensure that the **Preferences** > **Project Defaults** > **Color Management** > **Export** > **use OCIO nodes when exportingto a Comp** checkbox is enabled.
QT_COMPRESS_TABLET_ EVENTS	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.
QT_PLUGIN_PATH	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.
TIMELINE_DISABLE_PBO_ UPLOADS	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.

Sharing Presets

Hiero and HieroPlayer support facility-wide preset sharing by pointing multiple seats to a single network location using the HIERO_PLUGIN_PATH. As long as the folder hierarchy mirrors what the application expects, you can place this repository anywhere on your network.

To setup sharing:

1. Create the shared repository directory, for example:
   `/Volumes/myFacility/Shared/`
2. Follow the instructions under Setting Environment Variables to point individual machines to the shared location. For example, if you're using a **bash** shell, you could use:
   `export HIERO_PLUGIN_PATH=/Volumes/myFacility/Shared/`
   Any file that can be read locally can be placed in the shared path.

Startup Projects

Hiero and HieroPlayer search **<path>/StartupProjects/Hiero**, **<path>/StartupProjects/HieroPlayer** and **<path>/Templates** for preset projects saved as **.hrox** files. See Mac, Linux, and Windows for a description of the **<path>** variable by platform.

Projects placed in these folders behave differently, depending on which folder they are read from:

- **StartupProjects/Hiero** or **HieroPlayer** - projects in these folders are displayed in the bin view at startup and are read only. An example project, **Tag Presets**, is included in the **Tags** tab.
- **Templates** - projects in this folder appear in a dropdown menu when you open a new project. Select the template to use for the new project.

Note: You may need to create the **StartupProjects/Hiero**, **StartupProjects/HieroPlayer**, and **Templates** folders manually in the chosen path.

To remove a startup project, either:

- Remove it from **<path>/StartupProjects/Hiero**, **<path>/StartupProjects/HieroPlayer** or **<path>/Templates**, or
- Rename the file extension so the application doesn't recognize it on startup. For example, **myStartup.hrox_hidden**.

To modify a startup project:

1. Load the project using **File** > **Open**.
2. Make any required edits and save over the original in the same location.
   The edited project loads at startup.

Using the Python API

Hiero and HieroPlayer ship with a comprehensive Python application programming interface (API) enabling you to perform most user interface actions using Python scripting.

This section provides a basic overview and some simple examples to get you started, but more information is available in the dedicated Python API document included in the Hiero package. See **Help** > **Python Dev Guide** for more information.

Running Python During Startup

On startup, Hiero and HieroPlayer scan various directories for Python scripts that customize the behavior of the application. These scripts can be downloaded from online resources and placed in specific locations on disk, depending on the platform you're using.

If you're familiar with Python scripting, you can place custom startup scripts in the same locations, though bear in mind that you should take care as mistakes can produce performance issues.

Some example scripts are included in the application package in the following directories:

On Mac:

• /Applications/Hiero15.0v3/Hiero15.0v3.app/Contents/Plugins/site-packages/hiero/examples/

On Linux:

• /usr/local/Hiero15.0v3/Plugins/site-packages/hiero/examples/

On Windows:

• drive letter:\Program Files\Hiero15.0v3\plugins\site-packages\hiero\examples

At startup, Hiero and HieroPlayer search **<path>/Python/Startup** and **<path>/Python/StartupUI** for any Python **.py** modules or packages containing **__init__.py**. Scanning is done in two passes - all the Startup folders are searched and then all the StartupUI folders.

You can also add your own Startup folders, which are scanned after the standard paths, by creating a Python file pointing to the custom location. For example:

```
import hiero.core
hiero.core.addPluginPath("/myCustomPlugins")
```

Searches **/myCustomPlugins/Python/Startup** after the standard paths.

Each package or module discovered is imported and added to the built-in package **hiero.plugins**.

Note: Python startup scripts are imported in alphabetical order so you can force ordering if necessary.

The **<path>** varies by platform as follows:

Mac

- /Applications/Hiero15.0v3/Hiero15.0v3.app/Contents/Plugins
- /Library/Application Support/TheFoundry/Hiero
- ~/Library/Application Support/TheFoundry/Hiero
- ~/.nuke

Linux

- /usr/local/Hiero15.0v3/Plugins
- ~/.nuke

Windows

- drive letter:\Program Files\Hiero\plugins\hieroscripts
- drive letter:\Program Files (x86)\Hiero\plugins\hieroscripts
- ~\.nuke

Note: 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 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 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.

You can specify any number of user-defined paths using the environment variable HIERO_PLUGIN_PATH, separating them with **:** just like the standard unix PATH environment variable and Nuke's NUKE_PATH. See Setting Environment Variables for more information.

Using the Script Editor

Hiero and HieroPlayer include a Python **Script Editor**, accessible from the user interface, allowing you to enter Python statements directly.

To access the Script Editor:

1. Navigate to **Window** > **Script Editor**.

   The **Script Editor** displays.
2. Enter scripts in the lower half of the editor and press **Ctrl/Cmd** + **Enter** to execute the script.

   The results are displayed at the top of the editor. A simple example might be:

   ```
   from hiero.core import *
   bin = projects()[-1].clipsBin()
   bin.addItem(Bin("Plates"))
   bin["Plates"].importFolder("/Footage/Hiero/finalshots")
   ```

   Which imports **hiero.core**, defines **bin** in the last project opened, creates a bin called **Plates** at root level, and imports the specified folder into **Plates**.

Script Editor buttons and commands are described below:

Icon	Hotkeys	Description
	Ctrl/Cmd+[	Go to the previous script.
	Ctrl/Cmd+]	Go to the next script.
	n/a	Clear the script history.
	n/a	Load and execute a script.
	n/a	Load an existing script.
	n/a	Save a script as a **.py** file.
	Ctrl/Cmd+ enter	Run the current script.

Icon	Hotkeys	Description
	n/a	Show the input pane only.
	n/a	Show the output pane only.
	n/a	Show both the input and output panes.
	Ctrl/Cmd+ Backspace	Clear the output pane.

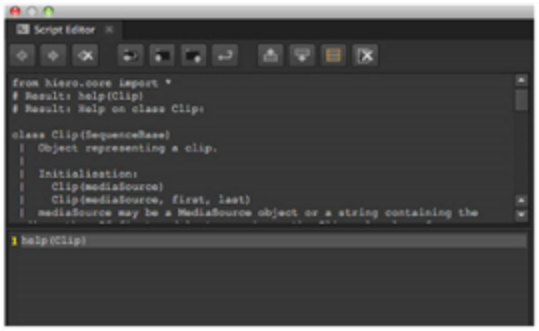
The Script Editor also features auto-completion and help for commands. Classes, attributes and so on are discoverable by:

• Entering the beginning of an object name and pressing the **Tab** key.

For example, typing **Bin** and pressing **Tab** displays a list of objects starting with Bin:

• Using the **help** command for a known object.

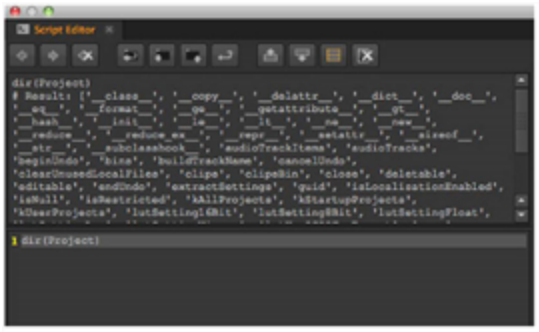
For example, **help(Clip)** returns a list of useful information on Clip:



Scroll down in the top pane to see more information relating to **Clip**.

• Using the **dir** command for a known object.

For example, **dir(Project)** returns a concise list of useful information on Project:



Note: The last entry in the Script Editor is restored at startup by default, but you can disable this behavior by navigating to **Preferences** > **Script Editor** and deselecting **Save and restore script editor history**.

For more information on Python scripting, see **Help** > **Python Dev Guide**.

Appendix A: Preferences

The Available Preference Settings

The **Preferences** dialog is divided into the following sections:

General	Settings for auto-saving and path substitutions.
Project Defaults	General project settings, and settings for color management.
Performance	Settings for caching, hardware, localization, and threads/processes.
Behaviors	Settings for start up, file handling, export options, scripting, node behaviors, and more.
Panels	Settings for the interface appearance, file browser, control panels, nodes, Viewers, script editors, and scopes.

General

General	
Autosave	
force project autosave after <300> seconds	Set the number of seconds after which to automatically save your project. Disable this by setting it to zero.
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 **Mac/Linux** column is replaced with the text in the **Windows** column, or vice versa. For example, if you enter **/Volumes/networkmount** in the **Mac/Linux** column and **Z:** in the **Windows** column: • On Mac and Linux, any file paths that start with **Z:** are converted to start

General	
	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. **Note:** Path substitution is invaluable for Sync Review sessions if you're using local source files rather than files from a central server. See Sync Review for more information.
+	Adds a row under **path substitutions**.
-	Deletes the selected row(s) under **path substitutions**.
Nuke	
Open In > New Hiero Session launches NukeX	When enabled, new Hiero sessions are launched as NukeX.

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. Hiero only supports 1023 uniquely named channels per script. The **Channel Count** is displayed in the bottom-right of the interface, next

Channel Management	
	to the Localization Mode indicator.  **Note:** Hiero 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	
Default Color Management	
Default Color Management	Sets the default color management system to use in new projects.
OpenColorIO config	
OpenColorIO config file	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.  **Note:** Hiero also includes an environment variable method for setting a config file. See Nuke Environment Variables for more information.
Default Color Transforms	
working space	Sets the colorspace files should be converted to, on import, and from, during render - it's the colorspace used under the hood.
viewer	Sets the default LUT applied to Viewers.
thumbnails	Sets the default LUT applied to thumbnails when ever they are generated.
Monitor Out	Sets the default LUT applied to external monitors.
8 bit files	Sets the default LUT applied to the specified ingested file type.

Color Management	
16 bit files log files floating point files	
OCIO Roles	
Allow OCIO Roles in Colorspace Knobs	When enabled, OCIO roles are available in all colorspace controls.
Prioritize OCIO Roles	When enabled, OCIO roles are created in the main menu of cascading dropdowns, with Colorspaces in a sub-menu. When disabled, roles are demoted to a sub-menu called Roles. You can also control this preference using an environment variable. See Nuke Environment Variables for more information.
HDR (macOS only)	
Enable macOS HDR Color Profile (Display P3) (Beta)	When enabled, running under macOS with a screen capable of displaying values greater than 1 allows you to display HDR images in the Display P3 colorspace correctly.
Hiero Script Project Settings	
color management	Sets whether Hiero 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

General - these preferences only apply to new scripts and projects. To affect the current project, use the **Project Settings**.	
	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.
frame rate	Select the frame rate for new projects in the Timeline environment.
start timecode	Use this to define the start timecode for new projects. For shots, this overrides the timecode defined in the media.
time display	You can use this to select the display format for times. You can select either **Timecode** or **Frames**.
drop frame	Use this to choose whether timecodes from this sequence are displayed in drop frame times or not. 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).

General - these preferences only apply to new scripts and projects. To affect the current project, use the **Project Settings**.	
	Note: Enabling Drop Frame is a Timecode display feature only - the source media remains a continuous stream of frames.
Clip	
clip reformat	Sets how new clips added to a sequence are formatted. The default, **None**, displays shots at the source clip resolution, whereas **To Sequence Resolution** reformats the shot to the **output resolution** using the **resize type** control.
resize type	When **clip reformat** is **To Sequence Resolution**, determines how the shot is resized to fit the output resolution. This control is disabled when **clip reformat** is set to **None**.
center	When **clip reformat** is **To Sequence Resolution**, disabling **center** places the clip at the bottom-left of the **output resolution**, rather than the center.
Link bin and track item versions	When enabled, selecting a new version for a clip in a bin or a shot on the timeline updates all instances of that media. When disabled, you can version clips and shots independently, even if they reference the same media.
Poster Frame	
poster frame	Sets a preset poster frame for all new source clips or select **Custom** to select a relative frame offset as the poster frame: • **First** - new source clips display the first frame of the file on disk as the poster frame. • **Middle** - new source clips display the middle frame of the file(s) on disk as the poster frame. • **Last** - new source clips display the last frame of the file(s) on disk as the poster frame. • **Custom** - 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.

General - these preferences only apply to new scripts and projects. To affect the current project, use the **Project Settings**.	
	For example, a **.dpx** sequence **myClip.####.dpx 1001-1500** can have a custom poster frame between **0** and **499**.
frame offset	When poster frame is set to **Custom**, enter the relative **frame offset** from the first available frame in the file(s) on disk.

Sync Session	
Default Appearance	
name	Sets the name and color you want to display on participant sessions. Click the color swatch to open the color picker.
Default Behavior	
open sync panel on sync session startup	When enabled, automatically open the Sync Session panel when a session is started.
Project Saving	
save directory	Sets the default directory where all Sync Session projects are saved.

Views - these preferences only apply to new scripts and projects. To affect the current project, use the **Project Settings**.	
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.

Views - these preferences only apply to new scripts and projects. To affect the current project, use the **Project Settings**.	
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	
Timeline Disk Caching	
directory path	The directory where all **.exr** files from timeline disk caching are stored by default. The caching directory should be on a local disk, preferably with the fastest access time available. It's also important to leave enough space for the maximum disk cache size (defined below).
limit to (GB)	This allows you to set the maximum amount of space (in GB) to use for the timeline disk cache. Set to zero for unlimited size. Values lower than zero, leave that amount of space free. The **currently in use** and **currently free** fields display how much free cache remains from the total specified. If this limit is reached during caching, a dialog displays options to free up disk space.
EXR compression	Sets the type of compression applied when writing files to the disk cache: • DWAB • Zip (1 scanline) **Note:** **DWAB** compression produces smaller cache files more quickly, but can be lossy compared to **Zip** compression.

Caching	
clear cache	**Clear All** removes all cached files from the root directory specified in the **directory path** control. A dialog displays a list of files for confirmation before the files are deleted.
default disk caching mode	Sets the default dick caching mode for new sequences: • Manual - the default mode, when a region is cached any changes made after the initial cache operation are not automatically re-cached. • Automatic - when a region is cached, any changes to soft effects made after the initial cache operation automatically triggers a re-cache of the affected region of your cached timeline. This means that your timeline is always cached and up to date in the selected regions, regardless of changes made.  **Note:** You can override this preference in the Sequence panel using the Disk Caching Mode dropdown.
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 Hiero 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).
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

Caching	
	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 Hiero. **Note:** When you switch back to the timeline, the cached files are re-cached, which can be time consuming.
Audio Waveforms	
waveform memory (MB)	Sets the amount of memory available for storing timeline audio waveforms.
Application in Background	
pause timeline Viewer when the application goes to the background	When enabled, pause timeline Viewer caching when the application is in the background.
clear timeline Viewer cache when the application goes to the background	When enabled, the timeline Viewer cache is cleared when the application goes into the background. **Note:** This preference is only available when **pause timeline Viewer when the application goes to the background** is enabled.
Undo Caching	
undo history size	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.
minimum undo events	Use this to set the amount of undo events. This setting always applies, even if it breaches the **undo history size** limit.

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**.
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. **Note:** You must restart Hiero for this option to take effect.
enable vsync	When enabled, synchronize new timeline Viewer's playback frame rate with the refresh rate of the monitor. Enabling vsync limits the playback fps to the monitor refresh rate and may reduce performance. When disabled, timeline Viewer performance is unaffected but you may see tearing in timeline Viewers. **Note:** This preference is only shown in Nuke Studio and Hiero sessions and only affects new timeline Viewers. Close and reopen any existing Viewers after enabling this option.
Localization	
System	

Localization	
mode	Sets the overall localization mode: • **on** - 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. • **manual** - checks for updates to source clips and Read nodes with **localization policy** set to **On Demand** and prompts you to update them manually. • **off** - no source clips or Read nodes are localized, regardless of the their localization policy.  **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:
	• 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

Localization	
	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 remote working folder. If you leave this field blank, automatic local file caching doesn't take place.
localize to	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). 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**
Storage	
limit to (GB)	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 **currently in use** and **currently free** fields display how much free storage remains from the total specified.

Localization	
	If this limit is reached during localization, a dialog displays options to free up storage space.
Network	
check for updated files every	When files are localized, specifies the time interval (in minutes) before Hiero checks for updated versions of the files.
Appearance	
progress bar	Sets the colors used to represent the localization states of source clips and Read nodes.

Threads/Processes	
Playback	
default number of threads per reader	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.
override number of threads per reader	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.
OpenEXR helper threads to use	Sets the number of helper threads to use for OpenEXR only. The default, zero, automatically sets the number of helper threads used.
Rendering	
frame server render timeout	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

Threads/Processes	
	value.
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	Set the number of worker Hiero processes to run for the frame server. **Note:** You must restart Hiero for this setting to take effect.
export renders	You can select from several render options: • **limit renderer (more responsive ui)** – Select this option to make the user interface more responsive during transcodes. It tells Hiero to use 2 threads to transcode and to use 25% of RAM to cache. Using this option is likely to result in slower transcodes. • **no renderer limits (fastest transcoding)** – Select this option to ensure that transcodes happen as quickly as possible. This option may result in a less responsive user interface during transcodes. • **customize render limits** – Select this option to manually configure the **number of threads** used and **cache memory** available when transcoding files. **Note:** You must restart Hiero for this setting to take effect.
number of threads	Sets the number of threads that Hiero 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 Hiero using the **-m** option.
cache memory (GB)	Use this to set the number of gigabytes of RAM that Hiero uses for its cache 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 Hiero with the **-c** option.
Downsize Filtering	
8-bit images	Customizes the downsize filtering behavior by bit-depth. The default (1x)

Threads/Processes	
10-, 12- and 16-bit integer images 16-bit float images 32-bit images	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 Hiero's built-in HTML help server. Hiero'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.
auto port	When enabled, assign a free port automatically.
local port	Specify a local documentation server port manually. This is usually >= 1024. You can also set this to 0 to automatically assign the port.
range	Specify a range of ports to attempt with the **local** documentation server.

File Handling	
scan for file sequence range on drop into Bin view	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 **.mov** and **.r3d**.)
automatically rescan	When enabled, incrementing a source clip or shot's version past the end of

File Handling	
versions when moving off end of the version list	the previously discovered versions list, forces a rescan to update the versions list. See Using Versions for more information.
frame number style	Sets the sequence display mode to be used in the file browser.
assume fixed width frame numbers in file sequences	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:

Files	With fixed frame width assumed	With fixed frame width NOT assumed
sequence 1.18.exr, sequence 1.19.exr, and sequence 1.20.exr	sequence 1.##.exr / sequence 1.%02d.exr	sequence 1.#.exr / sequence 1.%d.exr

Positions	
show menus with previous item under the cursor	When enabled, opening contextual menus positions them with the most recently used item under the pointer.

Startup	
startup workspace	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.
show splash screen at	When enabled, display the splash screen on startup.

Startup	
startup	
show startup dialog	When enabled, display the dialog on startup.
restore workspace when opening projects	When enabled, restore the selected saved workspace when opening projects.
Analytics	
submit usage statistics	When enabled, certain usage statistics are collected from the machine on which you license Hiero, NukeX, Nuke Studio, Hiero, and HieroPlayer. When disabled, we don't collect any usage data from your machine.  **Note:** The port number used to communicate with Foundry is 443, the same one used for uploading crash reports.

Timecode	
R3D file timecode	Sets the source timecode for RED files. You can choose from **Default From File**, **Absolute Timecode**, or **Edge Timecode**.
other media timecode	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.
max valid timebase (fps)	Sets the maximum image header timebase 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.
EDL style spreadsheet	When disabled, the srcOut and dstOut values in the spreadsheet use the

Timecode	
timecodes	film convention, representing the last frame of the cut. When enabled, the srcOut and dstOut values in the spreadsheet use the video convention, representing the frame directly after the cut.

UI Scaling	
 **Note:** You must restart the application to apply changes to **UI Scaling** preferences.	
UI Scaling Mode	Sets the scaling mode for the UI on Windows and Linux. macOS handles UI scaling automatically. • **off** - no scaling is applied to the UI, regardless of screen resolution. • **auto** - the UI attempts to scale appropriately dependent on screen resolution. • **all** - enables the **Scale All** dropdown, which allows you to scale all screens by the selected factor. • **per display** - enables the **Per Display** table, which allows you to set the scaling applied per screen.
Scale All	When **UI Scaling Mode** is set to **all**, allows you to scale all screens by the selected factor.
Per Display	When **UI Scaling Mode** is set to **per display**, allows you to set the scaling applied per screen.

Panels

Appearance	
Font	Change the type, weight, angle, and size of the font used on Hiero's user interface.
UI Colors - right-click on any color button and select **Set color to default** to revert changes.	

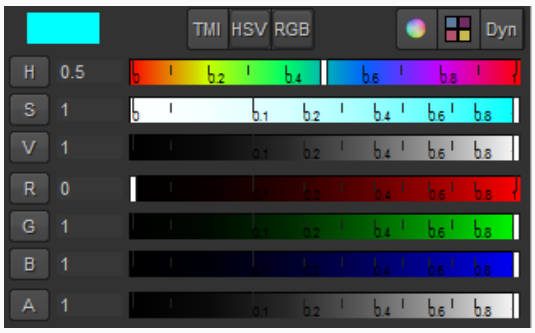
Appearance	
Background	Change the background color of most user interface elements (menus, toolbars, panes, properties panels, Viewers, and pop-up dialogs).
Base	Change the color of input fields, the input pane of the Script Editor, and the left side of the Curve Editor.
Highlight	Change the color of 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.
Highlighted Text	Change the color of any highlighted text (for example, text you select in node properties).
Label	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.
Button	Change the color of buttons and dropdown menus.
Animated	Change the color that indicates a control has been animated.
Keyframe	Change the color that indicates a keyframe has been set.
Disk cached frames	Change the color of the disk cached frames on the Viewer timeline.
RAM cached frames	Change the color of the RAM cached frames on the Viewer timeline.
Playhead	Change the color of the frame marker on the Viewer timeline.
In/Out Markers	Change the color of the in and out frame markers on the Viewer timeline.
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.
background	Change the background color of the Dope Sheet tab.
unselected key	Change the color used for an unselected key on the Dope Sheet.
part-selected key	Change the color used for a part-selected key on the Dope Sheet.
selected key	Change the color used for a selected key on the Dope Sheet.

Appearance	
timeline	Change the color used for the timeline on the Dope Sheet.
control text	Change the color used for the control text on the Dope Sheet. These indicate the frame number of a key when you select one.
control text shadow	Change the color used for the shadow of the control text on the Dope Sheet.
time label	Change the color used for the time labels on the Dope Sheet. These indicate frame numbers.
current frame	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.
project frame range	Change the color used for the project frame range on the Dope Sheet. These are two vertical lines indicate your frame range.

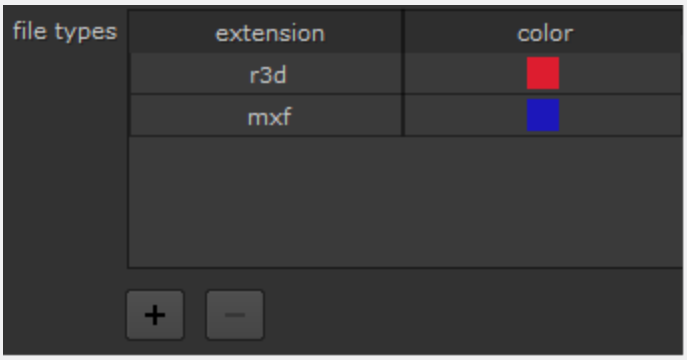
Control Panels	
max nodes in properties bin	Use this to set the maximum number of panels that can be open in the **Properties** pane.
reopen acts like new panel	When this is enabled, double-clicking a node that has been open before, places the panel in the same place as a new panel. If this is disabled, the panel appears in its previous position.
expand / collapse panels in Properties bin to match selection	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.
Color Panel	
color picker button opens	Sets the type of color picker displayed when you click a color picker button in the properties panel: • **in-panel color picker** - opens a color wheel and sliders in the properties panel. • **floating color picker** - opens a color wheel and sliders in a floating panel.

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

File Browser	
start file browser from most recently used directory	When enabled, new file browsers open at the last location used. When disabled, new file browsers open at the current working directory.

Node Colors	
autocolor	Deselect this checkbox to ignore individual soft effect color settings and, instead always use the **All Others** color.
<node name or type>	The soft effects 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. 
All Others	Use this to select the color to use as default for all soft effects not otherwise specified above, or all soft effects if **autocolor** is disabled.

Project Items	
shade project items	When enabled, additional shading is applied to source clips and shots in the Project bin and timeline.
Item Labels	
project bin	Click to change the color of labels in the **Project** and timeline panels. A color wheel displays allowing you to select the required color.
timeline	
auto-adjust contrast	When enabled, label colors are automatically adjusted if a potential color-clash is detected.
Item States	
offline	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.
error	
freeze	
comp not rendered	
comp out of date	
comp rendered	
Item Colors	
display in project panel	When enabled, the specified item colors, or defaults, are displayed in the **Project** panel.
display in sequence panel	When enabled, the specified item colors, or defaults, are displayed in the timeline panel.
spreadsheet color rows	When enabled, the specified item colors applied to rows in the spreadsheet.

Project Items	
project	Click the buttons to change the color of items in the **Project** and timeline panels. A color wheel displays allowing you to select the required color.
bin	
sequence	
source	
audio	
comp	
file types	Allows you to add custom color-coding by file extension. Click the + button and then select a file type from the **extension** 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. 

Scopes	
black point	Use the slider or the entry box to select the black point value.
white point	Use the slider or the entry box to select the white point value.
luma/chroma encoding	Use this to select the video standard to use when converting from RGB to luma and chroma for scope display.
Include viewer color	Select this checkbox to include applied Viewer color transforms (gain,

Scopes	
transforms	gamma, and LUT) in scope data. If this checkbox is disabled, all Viewer transforms are ignored. **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	
font	Use this to select the font to use in the Script Editor. **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 Hiero.
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	
Highlight clones of selected clone(s)	When enabled, selecting a clone in the timeline highlights all clones associated with the selected clone.

Timeline	
Playhead	
Show frame end marker	When enabled, an extra line is drawn on the timeline to the right of the playhead, indicating the end of the current frame.
Show buffer labels	When enabled, A/B labels are displayed on the buffer indicators at the playhead if the buffers are enabled and have shots assigned.
Visible range follows playhead	When enabled, the timeline scrolls with the playhead, constantly updating the view. When disabled, the playhead is allowed to move off screen.
Audio Tracks	
half waveforms	When enabled, audio tracks on the timeline display only the rectified waveform. When disabled, the full waveform is displayed.

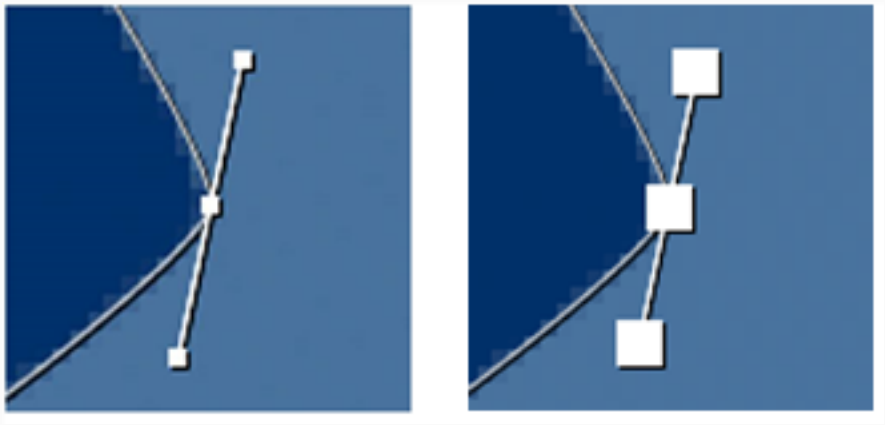
Viewer (Monitor Out)	
use video legal range for monitor out	When enabled, automatically limit monitor out to the legal range when swapping between the Timeline environment and Compositing environment.
always keep the floating window on top	When enabled, Hiero's internal floating monitor out window always remains on top of other open application windows.
Enable support for HMD devices	When enabled, VR headsets such as the Oculus Rift and HTC Vive are added to the Monitor Out > Device dropdown. Choose the required headset from the dropdown to pass the monitor out to the selected device.
Mouse Pointer Scale	Sets the size of the pointer displayed on the monitor when the Show Pointer control is enabled for Nuke Studio monitor out.
Background	
[background]	Sets the background color used to fill Hiero's floating window when the output is not full-frame: • Black • Gray - use the slider to the right to set the brightness of the plain

Viewer (Monitor Out)	
	background. • Checkerboard - use the slider to the right to set the brightness of the checkerboard background.
NDI	
Enable NDI	When enabled, you can stream Viewer output over the local network from the Monitor Out controls using NDI from NewTek.

Viewer	
default flipbook	Sets the default flipbook mode, Hiero's **Flipbook Viewer** or **HeiroPlayer**. • **Flipbook Viewer** opens a new Viewer inside Hiero containing the flipbooked content. • **HieroPlayer** launches an instance of the application containing the flipbooked content, but requires either a Nuke Studio or HieroPlayer license.
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 maintain the frame rate. • **Play All Frames, Buffering** - plays all frames by buffering and playing frames back as they become available.
guides	You can use this to choose to show overlays in the image area. You can choose from: • **Title Safe** – Indicates where text should be entered to be visible. • **Action Safe** – Indicates the area in which to place actions so that they are visible. • **Format** – Displays the size of the format over the Viewer.
fullscreen display	Use this to select which display to use for fullscreen mode. This setting takes effect the next time fullscreen mode is entered.

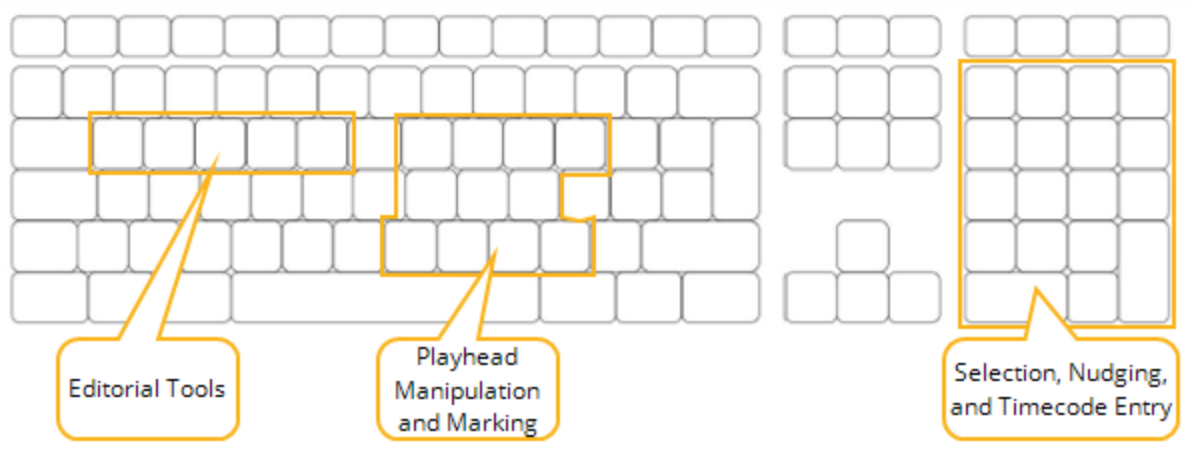
Viewer	
see through missing media	Select this checkbox to see through missing media in the timeline, displaying the first displayable media in the underlying tracks.
background	Use this to select the Viewer background. You can select black, or gray (using the slider to determine the grayscale), or checkerboard (using the slider to determine the brightness).
frame increment	Use this to set the default number of frames skipped by the Viewer skip controls, and the timeline **Nudge More** commands.
filtering mode	Use this to determine the filtering used during rendering in the Timeline environment. You can select **Auto**, **Nearest neighbour**, or **Linear**. **Auto** uses ths same automatic selection as in the Compositing environment. This does not affect exports or rendering in the Compositing environment.
Audio	
default latency adjustment (ms)	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: • at 25fps, a video frame is 1000/25 = 40ms, or • a 1.5 video frame delay = 1.5 * 40ms = 60ms.
default volume	Use the slider or numeric field to set the default volume.

Viewer Handles	
Controls	
2D handle size	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.

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.

Appendix B: Keyboard Shortcuts

This page lists the keyboard shortcuts used by Hiero and HieroPlayer, along with their function. In general, contiguous areas of the keyboard are assigned to related functionality. The left-hand keys on a QWERTY keyboard are used for tool selection and actions, the right-hand side for playhead manipulation including source/record and insert/overlay, the arrow keys for selection and nudging, and the numeric pad for timecode entry.



Hotkeys	Button	Menubar	Function
Timeline and Editing Controls			
Q		n/a	Cycles between the available move tools: • **Multi Tool** • **Move/Trim** • **Select**
W		n/a	Cycles between the available selection tools: • **Select Track to Right/Left** • **Select All in Track** • **Select All Tracks Right/Left** **Note:** Holding **Alt** toggles between track and all tracks behavior.

Hotkeys	Button	Menubar	Function
E		n/a	Cycles between the **Slip Clip** and **Slide Clip** tools.
R		n/a	Cycles between the available edit tools: • **Roll Edit** • **Ripple Edit** • **Retime Clip**
T		n/a	Cycles between the available razor tools: • **Razor** • **Razor All** • **Join**
left-click	n/a	n/a	Select a clip including any linked tracks.
Alt + left-click	n/a	n/a	Select a clip, ignoring linked tracks (for example, audio only).
Shift + drag shot	n/a	n/a	Insert a new track
left-click then Shift + left-click	n/a	n/a	Select all clips between the left-clicks (use **Shift** + **Alt** + **left-click** to ignore linked tracks).
drag then Alt	n/a	n/a	Activate Ripple mode while dragging the shot.
Alt then drag	n/a	n/a	Duplicate the dragged shot.
Alt and drag	n/a	n/a	Ripple and duplicate the dragged shot.
Alt and left-click	n/a	n/a	Select a clip, ignoring linked tracks.
Ctrl/Cmd + A	n/a	n/a	Select all shots.
Ctrl/Cmd + Shift + A	n/a	n/a	Deselect all shots.
File Menu			
Ctrl/Cmd +	n/a	File > New Project	Open a new project.

Hotkeys	Button	Menubar	Function
Shift + N			
Ctrl/Cmd + O	n/a	File > Open	Open an existing project.
Alt + Shift + <num>	n/a	File > Open Recent > Project Name	Open a previously saved project from the list.
Ctrl/Cmd + W	n/a	File > Close	Close the current project.
Ctrl/Cmd + S	n/a	File > Save	Save the current project.
Ctrl/Cmd + Shift + S	n/a	File > Save As...	Save the current project with a different name.
Ctrl/Cmd + I	n/a	File > Import Clips	Import a clip or clips.
Ctrl/Cmd + Shift + I	n/a	File > Import Folder	Import a folder or folders.
Ctrl/Cmd + Alt + Shift + I	n/a	File > Import EDL/XML	Import an EDL or XML sequence.
Ctrl/Cmd + Shift + E	n/a	File > Export	Export the selected sequence.
Ctrl/Cmd + Q	n/a	File > Quit	Close Hiero and display a prompt to save any unsaved work.
Edit Menu			
Ctrl/Cmd + Z	n/a	Edit > Undo	Undo the last action.
Ctrl/Cmd + Shift + Z	n/a	Edit > Redo	Redo the last undo action.
Ctrl/Cmd + X	n/a	Edit > Cut	Cut the selected item(s) and place in the buffer.
Ctrl/Cmd + C	n/a	Edit > Copy	Copy the selected item(s) to the buffer.
Ctrl/Cmd + V	n/a	Edit > Paste	Paste item(s) from the buffer.
Backspace /	n/a	Edit > Delete	Delete selected item(s).

Hotkeys	Button	Menubar	Function
Delete			
Ctrl/Cmd + A	n/a	Edit > Select All	Select all files in the current bin.
Ctrl/Cmd + Shift + A	n/a	Edit > Clear Selection	Deselect all files in the current bin.
Ctrl/Cmd + D	n/a	Edit > Duplicate	Duplicate the selected object(s).
Shift + S	n/a	Edit > Preferences	Open the Preferences dialog.
Viewer Menu			
Ctrl/Cmd + Shift + 1	n/a	View > Zoom to Actual Size	Zoom the contents of the Viewer to actual size.
Ctrl/Cmd + Shift + 2	n/a	View > Zoom to Half Size	Zoom the contents of the Viewer to half size.
H	n/a	View > Zoom to Fill	Fit the current clip to the height or width of the Viewer, depending on the clip's format.
F	n/a	View > Zoom to Fit	Zoom to fit the clip to the Viewer.
+	n/a	View > Zoom In	Zoom in to the Viewer.
- (minus)	n/a	View > Zoom Out	Zoom out of the Viewer.
Ctrl/Cmd + F	n/a	View > Full Screen	Toggle Full Screen mode.
Ctrl/Cmd + Shift + F	n/a	View > Full Quality 1:1	Toggle Full Screen mode at Full Quality 1:1 zoom.
Esc	n/a	n/a	Exit full screen mode.
E	n/a	View > Clipping Warning	Toggle the Level Warning tool on and off.
I		View > Mark In	Set the In Point at the current frame.
O		View > Mark Out	Set the Out Point as the current frame.

Hotkeys	Button	Menubar	Function
Alt + I	n/a	View > Clear In Point	Clear the In point for the current clip or timeline.
Alt + O	n/a	View > Clear Out Point	Clear the Out point for the current clip or timeline.
Alt + U	n/a	View > Clear In and Out Points	Clear the In and Out points for the current clip or timeline.
Home		View > Go to Start	Go to the start of all clips.
End		View > Go to End	Go to the end of all clips.
Shift + I	n/a	View > Go to In Point	Go to the In Point in the current clip.
Shift + O	n/a	View > Go to Out Point	Go to the Out Point in the current clip.
← (K + J)		View > Frame Backwards	Skip one frame backwards.
→ (K + L)		View > Frame Forwards	Skip one frame forwards.
Shift + ←		View > Skip Backwards	Skip backwards by the frame increment value specified in the Viewer tools.
Shift + →		View > Skip Forwards	Skip forwards by the frame increment value specified in the Viewer tools.
Alt + ←		View > Previous Edit	Go to the previous clip in the Viewer.
Alt + →		View > Next Edit	Go to the next clip in the Viewer.
Alt + Shift +←	n/a	View > Previous Tag	Go to previous frame tag.
Alt + Shift + →	n/a	View > Next Tag	Go to next frame tag.

Hotkeys	Button	Menubar	Function
Space		View > Play	Play the clip or sequence currently in the Viewer.
J		View > Play Backward	Play clip or sequence currently in the Viewer backward (click multiple times to increase playback speed).
K		View > Pause	Pause playback.
L		View > Play Forward	Play clip or sequence currently in the Viewer forward (click multiple times to increase playback speed).
Enter (numeric pad)	n/a	View > Edit Playhead Time	Shift focus to the timecode field under the Viewer, allowing you to manually edit the playhead position.
Ctrl/Cmd + Shift + P	n/a	View > Ignore Pixel Aspect	Display the clip in the Viewer with a 1:1 aspect ratio.
Ctrl/Cmd + /	n/a	Show Timeline Editor	Display the timeline pane, if it is not already visible.
Return (main keyboard)	n/a	View > Swap Inputs	Controls the AB inputs: • **Stack** - swaps which input is visible. • **Horizontal** - swaps the inputs horizontally. • **Vertical** - swaps the inputs vertically.
R, G, B, A, Y		View > Channels	Set the Viewer output channel.
PgUp	n/a	n/a	Scroll up and down through available layers in the Viewer.
PgDn	n/a	PgDn	n/a
Q	n/a	View > Show Overlays	Toggle the HUD overlays.
W		View > Toggle SplitWipe	Toggle the SplitWipe compare mode on and off.

Hotkeys	Button	Menubar	Function
Cache Menu			
F12	n/a	Cache > Clear Playback Cache	Click to clear the frames stored in the playback cash, represented by the green bar under the Viewer playhead.
Window Menu			
Alt + S	n/a	Window > Toggle Fullscreen	Expand the interface to fill the available screen space.
Ctrl/Cmd + Alt + Shift + ‘ (backtick)	n/a	n/a	Cycle backwards through all current panes.
Ctrl/Cmd + Alt + ‘ (backtick)	n/a	n/a	Cycle forwards through all current panes.
Ctrl/Cmd + Shift + [	n/a	n/a	Jump between tabs in the current pane.
Ctrl/Cmd + Shift +]	n/a	n/a	Ctrl/Cmd + Shift +]
Shift + F1	n/a	Window > Workspace > Conforming/Editing	Switch to Hiero’s Conforming workspace or HieroPlayer’s Editing workspace.
Shift + F2	n/a	Window > Workspace > Editing/Flipbook	Switch to Hiero’s Editing workspace or HieroPlayer’s Flipbook workspace.
Shift + F3	n/a	Window > Workspace > Reviewing	Switch to the Reviewing workspace.
Shift + F4	n/a	Window > Workspace > Timeline	Switch to the Timeline workspace.
Shift + F<num>	n/a	Window > Workspace	Switch to the custom workspace associated

Hotkeys	Button	Menubar	Function
		> Custom Workspace Name	with the number selected.
Project Menu			
Ctrl/Cmd + B	n/a	Project > New Bin	Create a new bin in the current bin.
Ctrl/Cmd + Y	n/a	Project > New Tag	Create a new tag in the current bin (tags can only be created in the **Tags** tab).
Ctrl/Cmd + N	n/a	Project > New Sequence	Create a new sequence in the current bin.
Clip Menu			
Ctrl/Cmd + Return (main keyboard)	n/a	Clip > Open In > Viewer	Open the selected clip in the Viewer.
Ctrl/Cmd + Alt + Return (main keyboard)	n/a	Clip > Open In > New Viewer	Open the selected clip a new Viewer.
A	n/a	Clip > Open In > Viewer Input A	Open the selected clip in Viewer input A.
B	n/a	Clip > Open In > Viewer Input B	Open the selected clip in Viewer input B.
Alt + D	n/a	Clip > Open In > Metadata View	Open the metadata view for the selected clip.
D	n/a	Clip > Hide Version	Hide the currently selected version(s) of a clip in the Versions Bin view.
Alt + ↑	n/a	Clip > Version Up	Cycle up through the available clip versions.
Alt + ↓	n/a	Clip > Version Down	Cycle down through the available clip versions.

Hotkeys	Button	Menubar	Function
Alt + Shift + ↑	n/a	Clip > Max Version	Jump to the maximum known version or scan for new versions if the max version is already reached.
Alt + Shift + ↓	n/a	Clip > Min Version	Jump to the minimum known version or scan for new versions if the min version is already reached.
V	n/a	Clip > Select Version	Display the available versions of a source clip or sequence.
F8	n/a	Clip > Refresh Clips	Reload the source clip or 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 source clip's current frame range.
Alt + F5	n/a	Clip > Rescan Clip Range	Similar to **Refresh Clips**, above, but rescan also checks for additional frames that may have been added to the source clip and adds them to the shot's frame range.
N	n/a	Clip > Insert	Insert the contents of a source Viewer into the timeline at the current playhead position and ripple existing shots downstream to accommodate the change.
M	n/a	Clip > Overwrite	Insert the contents of a source Viewer into the timeline at the current playhead position overwriting existing shots.
Timeline Menu			
Shift + U	n/a	Timeline > Mark Selection	Place the In and Out markers around the selected shot or shots on the timeline.
U	n/a	Timeline > Mark Clip	Place the In and Out markers around the upper-most timeline clip under the current playhead position.
C	n/a	Timeline > Razor	Apply the Razor tool to the currently selected

Hotkeys	Button	Menubar	Function
		Selected	track(s) within the selected clip(s).
Shift + C	n/a	Timeline > Razor All	Apply the Razor tool to all tracks within the selected clip(s).
Shift + Backspace	n/a	Timeline > Ripple Delete	Remove the selected clip(s) and ripple clips down stream to close gaps in the timeline.
Ctrl/Cmd + T	n/a	Timeline > Transition > Dissolve	Add a dissolve between two selected shots.
D	n/a	n/a	Enable or disable the selected clip(s) or track (s).
Ctrl/Cmd + Alt + A	n/a	Timeline > Select All in Track	Select all the shots on the current track.
, (comma)	n/a	Timeline > Nudge > Nudge Left	Trim, roll, or nudge a selected transition or shot to the left (within the available handles for trim and roll).
. (period)	n/a	Timeline > Nudge > Nudge Right	Trim, roll, or nudge a selected transition or shot to the right (within the available handles for trim and roll).
Shift + , (comma)	n/a	Timeline > Nudge > Nudge Left More	Trim, roll, or nudge a selected transition or shot to the left by the frame increment value specified in the Viewer tools (within the available handles for trim and roll).
Shift + . (period)	n/a	Timeline > Nudge > Nudge Right More	Trim, roll, or nudge a selected transition or shot to the right by the frame increment value specified in the Viewer tools (within the available handles for trim and roll).
Alt + , (comma)	n/a	Timeline > Nudge > Nudge Up	Nudge the selected shot(s) up one track.
Alt + . (period)	n/a	Timeline > Nudge > Nudge Down	Nudge the selected shot(s) down one track, where possible.

Hotkeys	Button	Menubar	Function
Ctrl/Cmd + Alt + S	n/a	Timeline > Audio Scrubbing	Enable or disable audio scrubbing.
Alt + Shift + /	n/a	Timeline > Rename Shots	Display the **Rename Shots** dialog for the selected shots.
Viewer Controls			
Alt + left-click + drag (middle-click and drag on Linux)	n/a	n/a	Pan the contents of the Viewer.
Scroll wheel (over Viewer or Viewer timeline)	n/a	n/a	Zoom the Viewer or Viewer timeline.
Scroll wheel middle-click + drag (over Viewer timeline or timeline)	n/a	n/a	Zoom the Viewer timeline to the indicated frame range.
Scroll wheel Alt + middle-click + drag (over Viewer or timeline)	n/a	n/a	Zoom the contents of the Viewer or timeline.
middle-click (over Viewer timeline or	n/a	n/a	Zoom to fit the Viewer timeline to Viewer contents.

Hotkeys	Button	Menubar	Function
timeline)			**Note:** If your mouse preferences are not set to **Button 3** for the middle mouse button, you may have to use **Ctrl/Cmd** + **middle-click**.
Alt + close Viewer		n/a	Close only the selected tab. Clicking close without holding **Alt** closes any linked tabs as well.
Spreadsheet Controls			
Alt + left-click	n/a	n/a	Move the playhead to the selected clip's **In** point on the timeline.
Alt + double left-click	n/a	n/a	Move the playhead to the selected clip's **In** point on the timeline and zoom to fit the clip in the timeline view.
Alt + close Spreadsheet		n/a	Close only the selected tab. Clicking close without holding **Alt** closes any linked tabs as well.
Bin View Searchbox Controls			
left-click	n/a	n/a	Select a tag search criteria.
Ctrl/Cmd + left-click	n/a	n/a	Toggle select individual tag search criteria.
Shift + left-click	n/a	n/a	Select a range of tag search criteria.
Script Editor Controls			
Ctrl/Cmd + [		n/a	Go to the previous script.
Ctrl/Cmd +]		n/a	Go to the next script.
Ctrl/Cmd + Enter		n/a	Execute current script.

Hotkeys	Button	Menubar	Function
Ctrl/Cmd + Backspace		n/a	Clear the output pane.

Appendix C: Supported File and Camera Formats

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.

Format	Bit Depths	Read/Write	Extension	Notes
Apple ProRes	10, 12	read and write	mov	Support for the following: • Apple ProRes 4444 XQ 12-bit • Apple ProRes 4444 12-bit • Apple ProRes 422 HQ 10-bit • Apple ProRes 422 10-bit • Apple ProRes 422 LT 10-bit • Apple ProRes 422 Proxy 10-bit
ARRIRAW	12	read only	ari	Alexa Mini LF
Codex			arx	Codex HDE

Format	Bit Depths	Read/Write	Extension	Notes

Camera/Sensor	Nuke 13.2		Nuke 14.0		Nuke 14.1		Nuke 15.0
ARRIRAW SDK (.ari, .arx, and .mxf)	v6.2.3.0	v7.0.0	v6.2.3.0	v7.0.0	v6.2.3.0	v8.0.0	v8.0.0
CodexHDE SDK	4.0.2	-	4.0.2	-	4.0.3	4.0.3	4.0.3
CUDA GPU Support	●	●	●	●	●	●	●
ALEXA 35	-	●	-	●	-	●	●
ALEXA Mini LF	●	-	●	-	●	-	●
ALEXA LF	●	-	●	-	●	-	●
ALEXA SXT(W)	●	-	●	-	●	-	●
ALEXA Mini	●	-	●	-	●	-	●
ALEXA 65	●	-	●	-	●	-	●
ALEXA XT	●	-	●	-	●	-	●
ALEXA Studio	●	-	●	-	●	-	●
ALEXA Classic	●	-	●	-	●	-	●
AMIRA	●	-	●	-	●	-	●

Note: Nuke, Nuke Studio, and Hiero do not support **.mxf** from the AMIRA camera.

AVI	n/a	read and write	avi	On Windows, the AVI reader uses DirectShow multimedia

Format	Bit Depths	Read/Write	Extension	Notes
				architecture. When decoding .avi files, DirectShow tries to find the appropriate codec on the system. If the codec is not available, .avi files cannot be opened.
BRAW	12	read	braw	
CIN	10 (log)	read and write	cin	
DNG	8, 12	read	dng	Includes RAW 2.5K CinemaDNG
DPX	8, 10, 12, and 16	read and write	dpx	YCbCr encoded DPX files are not supported on the timeline.
GIF	8	read only	gif	
Radiance	16	read and write	hdr, hdri	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.
JPEG	8	read and write	jpg, jpeg	Adjust

Format	Bit Depths	Read/Write	Extension	Notes
				compression levels using the quality slider in the Write node's properties panel.
MOV	n/a	read and write	mov	The writer supports the following codecs: • Animation • Apple ProRes 4444 XQ 12-bit • Apple ProRes 4444 12-bit • Apple ProRes 422 HQ 10-bit • Apple ProRes 422 10-bit • Apple ProRes 422 LT 10-bit • Apple ProRes 422 Proxy 10-bit • Avid DNxHD 444 10-bit 440Mbit • Avid DNxHD 422 10-bit 220Mbit • Avid DNxHD 422 8-bit 220Mbit • Avid DNxHD 422 8-bit 145Mbit • Avid DNxHD 422 8-bit 36Mbit

Format	Bit Depths	Read/Write	Extension	Notes

				Note: Interlaced writing is not supported. See Avid DNxHD Notes for more information.
				• Avid DNxHR:
				  • 8-bit 4:2:2 DNxHR LB, DNxHR SQ, and DNxHR HQ
				  • 12-bit DNxHR HQX 422 and DNxHR 444
				• H.264 Main 4:2:0 8-bit
				• H.264 High 4:2:0 8-bit
				• Motion JPEG A
				• Motion JPEG B
				• MPEG-4
				• Photo - JPEG
				• PNG
				• Uncompressed

Format	Bit Depths	Read/Write	Extension	Notes
		write only	mov	DNxHR Pattern: OP-1A, OP-Atom Profiles: • 4:4:4 12-bit • HQX 4:2:2 12-bit • HQ 4:2:2 8-bit • SQ 4:2:2 8-bit • LB 4:2:2 8-bit
MXF	8, 10, 12	read only	mxf	Supported codecs include: • Uncompressed 4:2:2 YCbCr 8-/10-bit • Uncompressed 4:4:4:4 RGBA 8-/10-bit • Uncompressed Avid 4:2:2 YCbCr 8-/10-bit • Uncompressed Avid 4:4:4:4 RGBA 8-/10-bit • JPEG2000 • Avid DNxHD (1080p and 720p 1920x1080 and 1280x720, 4:4:4:4 and 4:2:2) 36, 115, 120, 145, 175, 185, 220, 220x See Avid DNxHD Notes for more information.

Format	Bit Depths	Read/Write	Extension	Notes

				• Avid DNxHR:
				  • 8-bit 4:2:2
				  DNxHR LB, DNxHR SQ, and DNxHR HQ
				  • 12-bit
				  DNxHR HQX 422 and DNxHR 444
				• Sony Raw from the F65, F55, F5 and FS700 cameras. All formats that these cameras provide: 4K, 2K, 1K, 0.5K and 0.25K
				• Sony X-OCN from the VENICE, F55, and F5 cameras.
				• ARRIRAW from the Alexa Mini.

		write only	mxf	DNxHR Pattern: OP-1A, OP-Atom Profiles: • 4:4:4 12-bit • HQX 4:2:2 12-bit • HQ 4:2:2 8-bit • SQ 4:2:2 8-bit • LB 4:2:2 8-bit

Format	Bit Depths	Read/Write	Extension	Notes

Camera/Sensor	Nuke 13.2	Nuke 14.0	Nuke 14.1	Nuke 15.0
SONY SDK (**.mxf**)	v3.3	v4.0.0	v4.0.0	v4.0.0
SONY SMDK	v4.19.0	v4.21.0	v4.21.0	v4.21.0
GPU Support	●	●	●	●
VENICE 2	-	●	●	●
VENICE	●	●	●	●
F55	●	●	●	●
F5	●	●	●	●
F65	●	●	●	●
FS700	●	●	●	●

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 fps (which is actually 45 Mbps @ 29.97fps). You should look at the codec format (422/444, 8/10-bit).

Note: Hiero 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

Format	Bit Depths	Read/Write	Extension	Notes
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.)				
OpenEXR and OpenEXR 3.1.6	16, 32	read and write	exr	OpenEXR handles 16- and 32-bit float. This 16 is also called "half float" and is different from the 16-bit integer that all the other formats that support 16-bit use. Hiero supports multi-part OpenEXR files. For more information, see the *Hiero User Guide*.
EXR Compression				

Format	Bit Depths	Read/Write	Extension	Notes
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				
PIC SoftImage®	8	read and write	pic	
PNG	8, 16	read and write	png (8-bit) png16 (16-bit)	
PSD	8, 16	read only	psd	While Hiero reads standard Photoshop® blend modes, it doesn't read Photoshop layer comps or recognize group blend modes. Photoshop layers

Format	Bit Depths	Read/Write	Extension	Notes
				are read into separate Hiero layers and anything that doesn't map into that is ignored.
R3D	16	read only	r3d	

Camera/Sensor	Nuke 13.2	Nuke 14.0	Nuke 14.1	Nuke 15.0
R3D SDK (.r3d)	v8.0.4	v8.3.0	v8.3.0	v8.3.0
CUDA GPU Support	●	●	●	●
V_RAPTOR XL	-	●	●	●
V-RAPTOR	-	●	●	●
KOMODO	●	●	●	●
MONSTRO 8K VV	●	●	●	●
HELIUM 8K S35	●	●	●	●
SCARLET-W 5K	●	●	●	●
RED RAVEN 4.5K	●	●	●	●
WEAPON DRAGON 6K	●	●	●	●
EPIC/SCARLET DRAGON	●	●	●	●
RED ONE MYSTERIUM 4K	●	●	●	●
MYSTERIUM-X	●	●	●	●
MYSTERIUM-X MONOCHROME	●	●	●	●

Format	Bit Depths	Read/Write	Extension	Notes
RAW	n/a	read only	n/a	DSLR raw data files, such as Canon .CR2 files. These are only supported via the dcraw command line program, which you can download from the dcraw website. Bit depth and other specifications depend on the device. Some devices may not be supported.
RLA Wavefront®	8	read only	rla	
SGI	8, 16	read and write	sgi, rgb, rgba (8-bit sequences) sgi16 (for 16-bit sequences)	
TIFF	8, 16, and 32	read and write	tif, tiff (8-bit sequences) tif16, tiff16 (16-bit sequences) ftif, ftiff (32-bit sequences)	If utilized, the compression schema on imported TIFF sequences must be LZW®.
TARGA Truevision®	8	read and write	tga, targa	

Format	Bit Depths	Read/Write	Extension	Notes
XPM	8	read and write	xpm	This is the text-based format in which Hiero's interface elements are stored.
YUV	8	read and write	yuv	This format does not specify resolution, so Hiero assumes a width of 720 pixels.

Reader Fallback Paths

If the application can't find a suitable reader when reading source clips from disk, the following paths are searched for fallback **.dylib**, **.so**, and **.dll** readers compiled against the comparable Hiero version:

Mac:

- /$HOME/.nuke
- /Library/Application Support/Nuke/x.x/plugins/

Linux:

- /$HOME/.nuke
- /usr/local/Nuke/x.x/plugins/

Windows:

- \$HOME\.nuke
- C:\Program Files\Common Files\Nuke\x.x\plugins\
- C:\Program Files (x86)\Common Files\Nuke\x.x\plugins\

Note: You can also add your own search paths using the NUKE_PATH environment variable. See Setting Environment Variables for more information.

Supported Audio Formats

The following table lists supported audio formats.

Format Name	Extension
Audio only formats	
All platforms	
Wave	wav
Audio interchange format	aiff
Audio Codec 3	ac3
MPEG 1 Audio Stream	mp2
Mac and Windows	
MPEG 4 Audio	m4a
Integrated audio formats	
RED Audio	r3d
QuickTime Audio	mov

Appendix D: Third-Party Software

Third-Party Contributions

The following table lists third-party contributions included in Hiero, along with their licenses.

Contributor	Description	License
Pixar	DTEX file reader	Copyright © Pixar. All rights reserved. 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. 1. Definitions The terms "reproduce," "reproduction," "derivative works," and "distribution" have the same meaning here as under U.S. copyright law. A "contribution" is the original software, or any additions or changes to the software. A "contributor" is any person or entity that distributes its contribution under this license. "Licensed patents" are a contributor's patent claims that read directly on its contribution. 2. Grant of Rights (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. (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,

Contributor	Description	License
		import, and/or otherwise dispose of its contribution in the software or derivative works of the contribution in the software. 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.

Third-Party Library Versions

The following table lists third-party libraries included in the application and their current version. This version of Nuke is compliant with VFX Reference Platform 2022.

Note: We use the ARRI RAW 6.2 SDK for all ARRI RAW files except those from the new Alexa 35 camera. For the Alexa 35 we use ARRI IMAGE 8.0.

	Version		
Library	**Linux**	**Mac**	**Windows**
AAF	1.2.0	1.2.0	1.2.0
ACES	1.3	1.3	1.3
AJA	16.2.2	16.2.2	16.2.2
Alembic	1.8.3	1.8.3	1.8.3
ARRI RAW SDK	6.2.3.0	6.2.3.0	6.2.3.0
ArriMXF	3.0.0.10	3.0.0.10	3.0.0.10
ARRI Image SDK	8.0.0	8.0.0	8.0.0
Blackmagic RAW SDK	2.2	2.2	2.2
Blosc	1.14.3	1.14.3	1.14.3
BMD Decklink SDK	12.4.2	12.4.2	12.4.2
Boost	1.76.0	1.76.0	1.76.0

Library	Version		
	Linux	Mac	Windows
Breakpad	r1338	r1338	r1338
bzip	1.0.6	1.0.6	1.0.6
clFFT_AMD	2.14.0	2.14.0	2.14.0
Clang	N/A	10.2.1	N/A
CodexHDE SDK	4.0.3	4.0.3	4.0.3
cppunit	1.12.1	1.12.1	1.12.1
Ctypes	1.0.2	1.0.2	1.0.2
CUDA	11.8.0	N/A	11.8.0
cuDNN	8.4.1	N/A	8.4.1
Curl	7.72.0	7.72.0	7.72.0
DirectX	N/A	N/A	Jun10
DNG SDK	1.6	1.6	1.6
DNxHD and DNxHR	2.7.3	2.7.3	2.7.3
EuCon	N/A	2.5.5	N/A
Expat	2.2.0	2.2.0	2.2.0
FBX	2020.3.1	2020.3.1	2020.3.1
FFmpeg	4.2	4.2	4.2
FFTW	3.1.3	3.1.3	3.1.3
Flatbuffers	23.1.4	23.1.4	23.1.4
fmt	8.1.1	8.1.1	8.1.1
FreeType	2.10.4	2.10.4	2.10.4
FTGL	2.1.3	2.1.3	2.1.3

Library	Version		
	Linux	Mac	Windows
GCC	9.3.1	N/A	N/A
GLEW	1.13.0	1.13.0	1.13.0
googletest	1.12.1	1.12.1	1.12.1
gperftools	2.10	2.10	2.10
HDF	5.1.8.7	5.1.8.7	5.1.8.7
HIDAPI	0.7.0_fn.2	0.7.0_fn.2	0.7.0_fn.2
imath	3.1.6	3.1.6	3.1.6
Intel Libraries	12.1.0258	12.1.0258	12.1.0258
Intel MKL	2020.4	2020.4	2020.4
Intel TBB	2020 Update 3	2020 Update 3	2020 Update 3
JPEG	9e	9e	9e
JWT	0.4.0	0.4.0	0.4.0
Libexif	0.6.24	0.6.24	0.6.24
Libpng	1.4.8	1.4.8	1.4.8
libsndfile	1.0.31	1.0.31	1.0.31
Libtiff	4.3.0	4.3.0	4.3.0
Libusb	1.0.22	N/A	N/A
libuv	1.45.0	1.45.0	1.45.0
LLVM	13.0.0	13.0.0	13.0.0
Murmur3	20150501	20150501	20150501
Nablet	2.50.4	2.50.4	2.50.4
NDI SDK	5.5.2.0	5.5.2.0	5.5.2.0

Library	Version		
	Linux	Mac	Windows
nlohman	3.9.1	3.9.1	3.9.1
Oculus Rift SDK	N/A	N/A	1.15.0
OFX	1.2	1.2	1.2
OfxHostSupport	1.2	1.2	1.2
OpenAssetIO	1.0.0-alpha.14	1.0.0-alpha.14	1.0.0-alpha.14
openassetio-mediacreation	1.0.0-alpha.7	1.0.0-alpha.7	1.0.0-alpha.7
OpenCL	2021.7.1	N/A	2021.7.1
OpenColorIO	2.1.3	2.1.3	2.1.3
OpenColorIOConfigs	1.0.3	1.0.3	1.0.3
OpenEXR	3.1.6	3.1.6	3.1.6
OpenHMD	0.3.0	0.3.0	0.3.0
OpenImageIO	2.3.13.0	2.3.13.0	2.3.13.0
OpenJPEG	1.5.0	1.5.0	1.5.0
OpenSSL	1.5.0	1.5.0	1.5.0
OpenSubDiv	3.4.3	3.4.3	3.4.3
OpenTimelineIO	0.15.0	0.15.0	0.15.0
OpenVDB	9.0.0	9.0.0	9.0.0
OpenVR	1.2.10	1.2.10	1.2.10
picojson	1.3.0	1.3.0	1.3.0
PNG	1.6.37	1.6.37	1.6.37
PoissonRecon	V2	V2	V2
PortAudio	v19_20111221	19.7.0	19.7.0

Library	Version		
	Linux	Mac	Windows
Primatte	5.0.2023	5.0.2023	5.0.2023
ProResCodec	2021-01-29	N/A	2021-01-29
Protobuf	v23.3	v23.3	v23.3
pybind11	2.9.2	2.9.2	2.9.2
psutil	5.8.0	5.8.0	5.8.0
Ptex	2.4.x	2.4.x	2.4.x
PyString	1.1.0	1.1.0	1.1.0
Python	3.9.1	3.9.1	3.9.1
PyTorch	1.12.1	1.12.1	1.12.1
PyZMQ	13.0.2	13.0.2	13.0.2
Qt	5.15.2	5.15.2	5.15.2
Qt for Python (PySide2)	5.15.2.1	5.15.2.1	5.15.2.1
QtSingleApplication	2.6.0	2.6.0	2.6.0
R3D SDK	8.3.0	8.3.0	8.3.0
Revulytics	6.2.0	6.2.0	6.2.0
RLM	15.0	15.0	15.0
sentry	0.6.5	0.6.5	0.6.5
SCons	2.1.0	2.1.0	2.1.0
Skein	1.1	1.1	1.1
SonyRAW	4.0.0	4.0.0	4.0.0
SonyRenderman	15.1	N/A	15.1
sony-smdk	4.21.0	4.21.0	4.21.0

Library	Version		
	Linux	Mac	Windows
SQLCipher	3.1.0	3.1.0	3.1.0
SQLite	3.32.3	3.32.3	3.32.3
sse2neon	1.6.0	1.6.0	1.6.0
symbolicator	v0.7.0	v0.7.0	v0.7.0
Tcl	8.4.16	8.4.16	8.4.16
TinyXML	2.6.0	2.6.0	2.6.0
TorchVision	0.13.1	0.13.1	0.13.1
Ultimatte	1.0	1.0	1.0
URIParser	0.8.0	0.8.0	0.8.0
USD	23.05	23.05	23.05
uuid	1.0	N/A	N/A
VideoToolbox	N/A	HW/SW	N/A
Visual Studio	N/A	N/A	2019
VXL	1.18.0	1.18.0	1.18.0
xmlrpcpp	N/A	N/A	0.7
ZeroMQ	3.2.5	3.2.5	3.2.5
Zipstream	1.0	1.0	1.0
zlib	1.2.11	1.2.11	1.2.11

For a full list of all third-party licenses and fonts used in the application, please see the Nuke Online Help, *Third-Party Libraries and Fonts*, or click here.

Nuke's Machine Learning tools were trained using the image libraries referenced and licensed in this AIR Image Licences PDF, available for download by clicking the link.