V__matrix

User Manual

Version: 1.11/2

Edition: Tuesday, November 30, 2021

To obtain the latest documentation and software downloads, please visit:

www.lawo.com/lawo-downloads

Copyright

All rights reserved. Permission to reprint or electronically reproduce any document or graphic in whole or in part for any reason is expressly prohibited, unless prior written consent is obtained from the Lawo AG.

All trademarks and registered trademarks belong to their respective owners. It cannot be guaranteed that all product names, products, trademarks, requisitions, regulations, guidelines, specifications and norms are free from trade mark rights of third parties.

All entries in this document have been thoroughly checked; however no guarantee for correctness can be given. Lawo AG cannot be held responsible for any misleading or incorrect information provided throughout this manual.

Lawo AG reserves the right to change specifications at any time without notice.

© Lawo AG, 2021

Table of Contents

1. Introduction

Welcome to **V__matrix**.

About this Manual

This document describes how to connect and configure the system.

Look out for the following which indicate:

Notes - points of clarification.

Tips - useful tips and short cuts.

Attention - alert you when an action should *always* be observed.

Utility Software

Each C100 core processing module is configured via a web-based user interface and/or Lawo's V__script application.

V__script is free to download from the Lawo **Downloads** area (after Login).

Further Information

Mechanical drawings and data sheets (including weights and dimensions) are available from the **Downloads** area (after login).

We also recommend that you carefully observe the release notes.

Lawo User Registration

For access to the **Downloads** area and to receive regular product updates, please register at:

www.lawo.com/registration.

2. Important Safety Instructions

Please observe all of the instructions provided in the "General Safety Information for Lawo Equipment" booklet delivered with your devices. Double-click here to open the same information (as a pdf).

3. Product Overview

The V__matrix is a software-defined IP core routing, processing and multi-viewing platform.

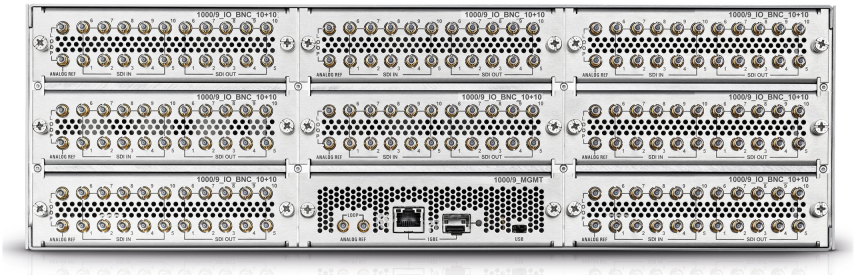
The system is divided into two parts: the physical and the virtual. The physical consists of the C100 processing blades and associated hardware. The virtual is centered around the software packages, called Virtual Modules (VM), which define the functionality.

A range of frame options support different numbers of C100 blades (fitted to the front slots) and I/O modules (fitted to the rear). The rear I/O modules can be used to connect legacy equipment such as baseband video and audio components.

Every frame includes a dual power supply (fitted to the front). This provides redundant power for the full frame and all connected blades. Optionally, a central management module (fitted to the rear) provides a centralized management port and video reference input. A removable grill (fitted to the front) provides secure protection for all processing blades.

V__matrix 8 (front view, grill removed)

V__matrix 8 (rear view)



3.1 Frame Options

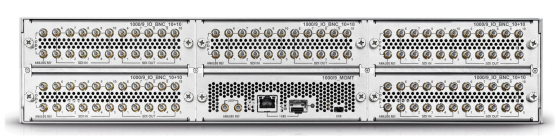
In the current release there are four types of frame available. The first three options support different numbers of slots for processing blades and I/O modules. The 2RU Silent Frame is designed for noise-sensitive installations.

V__matrix 2: 1RU (2 slots)

Front View (grill removed)

Rear View (with I/O modules)

V__matrix 5: 2RU (5 slots)



V__matrix 8: 3RU (8 slots)

V__matrix Silent Frame: 2RU (2 slots)

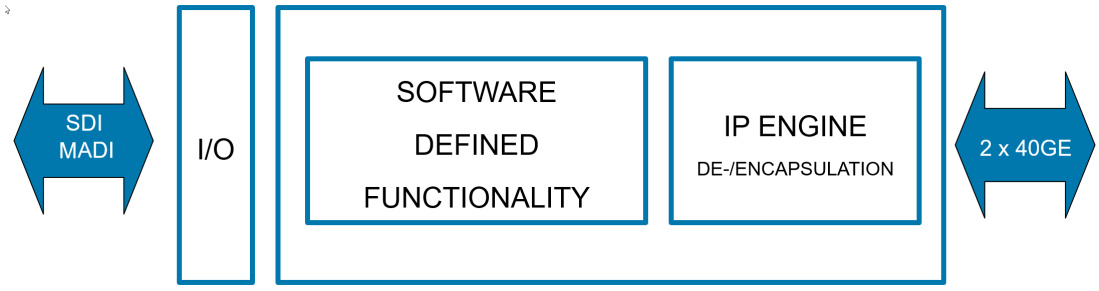
3.2 C100 Processing Blade

The C100 plug-in cards slot into the front of the V__matrix frame.

Each one has three network ports: one for management (MGMT) and two for media (2x 40GbE). Note that, optionally, the C100 can be managed via the central management port on the rear MGMT module.

The two media ports connect the red and blue networks to support redundant streaming. They can be configured for either a single 40GbE connection, or four 10GbE lanes (via QSFPs).

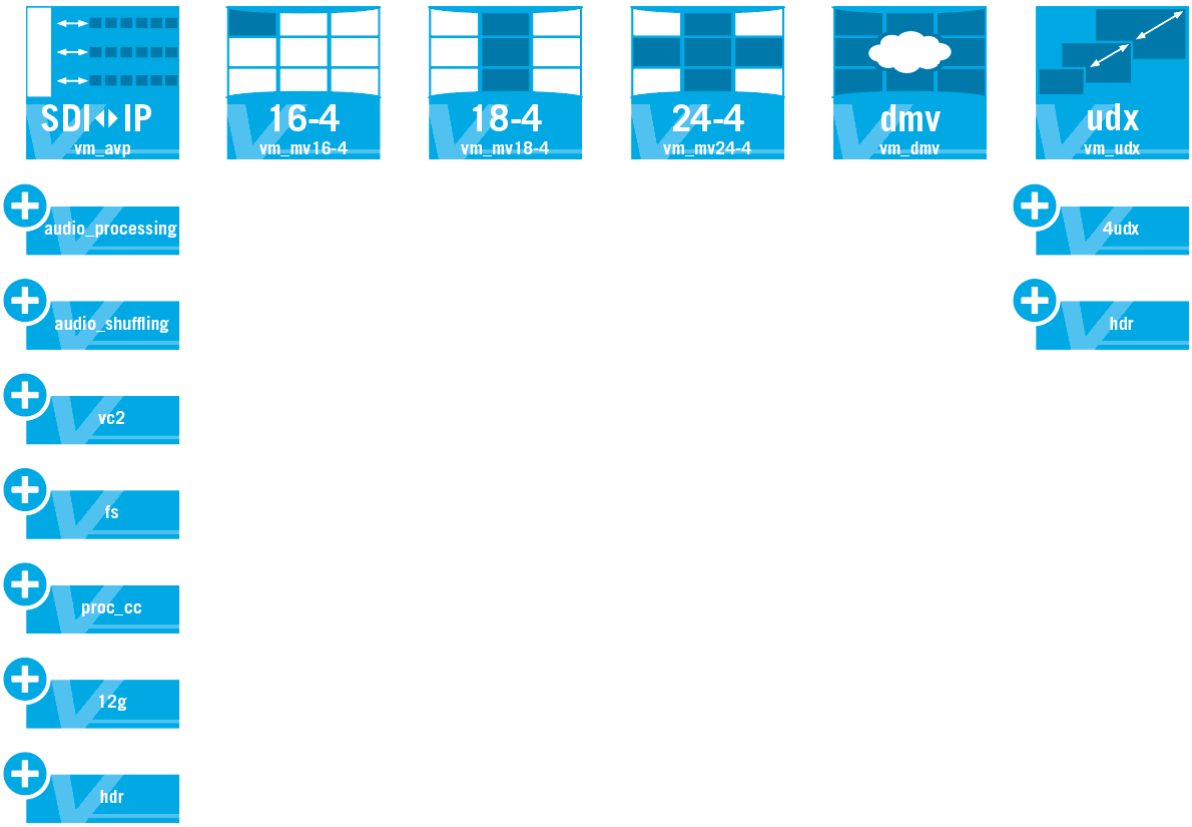
C100 Block Diagram



Each C100 card has an optional SDI backplane to connect SDI / MADI.

The IP Engine takes care of the connection to IP. The software part, called the Virtual Module (VM), defines the functionality.

3.3 Virtual Modules (VMs)



In the current release, there are four types of Virtual Module:

- **vm_avp** - an audio video processor for SDI and IP.
- **vm_mv** - a multiviewer for SDI and IP.
- **vm_dmv** - a distributed multiviewer for SDI and IP.
- **vm_udx** - an up/down/cross converter for SDI and IP sources.

The **vm_mv** app comes in three sizes.

The **vm_avp** and **vm_udx** apps can be expanded via licensed add-ons.

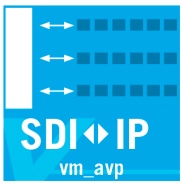
Each C100 can be configured differently, allowing a single V__matrix frame to cover a range of applications. The current app is defined by the FPGA mode. This can be changed from the advanced configuration page.

The next few pages describe each app in more detail.

Please consult the data sheets for the latest specification and more detail on each virtual module. The data sheets are available from the Lawo Download-Center (after login).

3.3.1 vm_avp

The **vm_avp** app operates as a SDI<->IP and IP<->IP gateway to ST2022 and ST2110 for UHD/3G/HD/SD.



As standard, it includes:

- 20TX and 20RX IP instances of audio and video.
- Support for seamless protection switching ST2022-7.
- Destination timed vertically accurate switching through MBB or BBM.

The functionality can be expanded by the following add-ons.

- **+audio** adds additional audio TX/RX instances for a total of 40 TX and 40 RX as well as SRC and a larger audio matrix for embedding/de-embedding and shuffling.

- **+audio_shuffling** adds additional audio TX/RX instances for a total of 128 TX and 88 RX as well as SRC and a 5312x5312 mono audio matrix for embedding/de-embedding and shuffling.

- **+vc2** adds 20 instances of Dirac/VC2 low latency codecs providing compression between 2.5:1 and 8.0:1 with less than 20 lines of delay.

- **+fs** adds 20 instances of framesync and audio/video delay including SRC.

- **+proc_cc** adds 8 instances of color correction as well as test pattern generation.

- **+12g** adds 12G single-link capabilities as well as UHD single-link to quad-link conversion (2SI or SQD).

3.3.2 vm_mv

The **vm_mv** app operates as a multiviewer for SDI and IP (ST2022-6 & ST2110) signals in various formats (UHD, 3G, HD and SD).

As standard, it includes:

- Support for seamless protection switching ST2022-7.
- 16, 18 or 24 sources from either IP and/or SDI per C100.
- 4 output heads @3G over either IP and/or SDI per C100 (1 head in UHD).
- Any source on any output in any format (same source can be on multiple outputs in different resolutions).
- A/V monitoring (black, freeze, audio missing, audio low, etc).
- Support for tally.

3.3.3 vm_dmv

The **vm_dmv** app operates as a distributed multiviewer for SDI and IP (ST2022-6 & ST2110) signals in various formats (UHD, 3G, HD and SD).

As standard, it includes:

- Support for seamless protection switching ST2022-7.
- 24 sources @3G or 12 sources @12G (or a mix) from either IP and/or SDI per C100.
- Different combinations of output heads over either IP and/or SDI per C100 (please consult datasheet for details).
- Cluster capability to expand linearly to a cluster size of up to 32x C100 modules.

3.3.4 vm_udx

The **vm_udx** app operates as a Up/Down/Cross converter for IP and SDI sources.

As standard, it includes:

- 4 instances of up/down/cross conversion for signals up to 3G. A conversion to/from UHD requires 4 instances.
- Frame sync, audio embedding / de-embedding and color correction.

The functionality can be expanded by the following add-ons.

- **+4udx** adds an additional 4 paths of Up/Down/Cross conversion.

- **+hdr** adds 4 instances (@3G) of HDR<->SDR dynamic range and color space conversion using 3D LUTs. A UHD HDR conversion requires 2 instances.

3.4 Rear I/O Modules

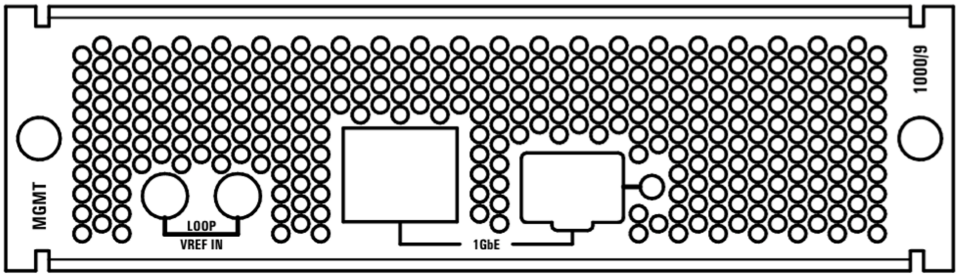
The rear I/O modules provide additional interfaces on the back of the processing blade.

All I/O modules are optional and can be specified as required. Please consider the bandwidth restrictions when designing the system.

All modules support a video reference input and loop-thru to connect Tri-level sync or Video Black Burst. The sync signal is distributed via the backplane to all I/O modules in the frame.

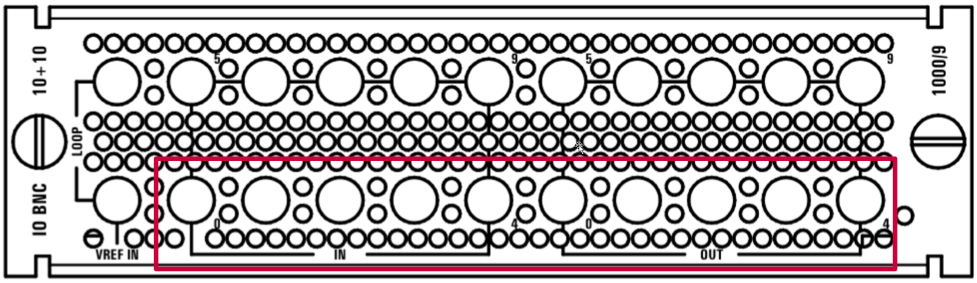
In each case, the maximum number of IN and OUT depends on the signal format: 12G, 3G, etc. Please refer to the V__matrix C100 data sheet for full specifications and supported signal options.

3.4.1 MGMT



Central management module with Ref IN + 1GbE (on RJ45 or SFP).

3.4.2 SDI 10+10

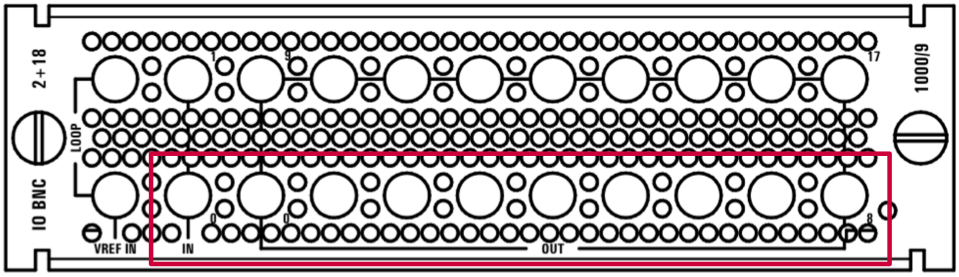


12G/3G/HD/SD (please keep in mind the max bandwidth on the 40G)

SDI 10+10: with Ref IN + 10 SDI IN + 10 SDI OUT

- **Top row**: 5x 3G/HD/SD
- **Bottom row**: 5x 12G/3G/HD/SD

3.4.3 SDI 2+18

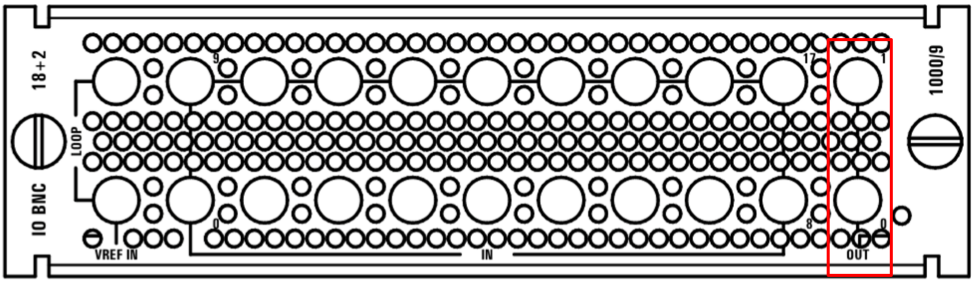


12G/3G/HD/SD (please keep in mind the max bandwidth on the 40G)

SDI 2+18: with Ref IN + 2 SDI IN + 18 SDI OUT

- **Top row**: 10x 3G/HD/SD
- **Bottom row**: 10x 12G/3G/HD/SD

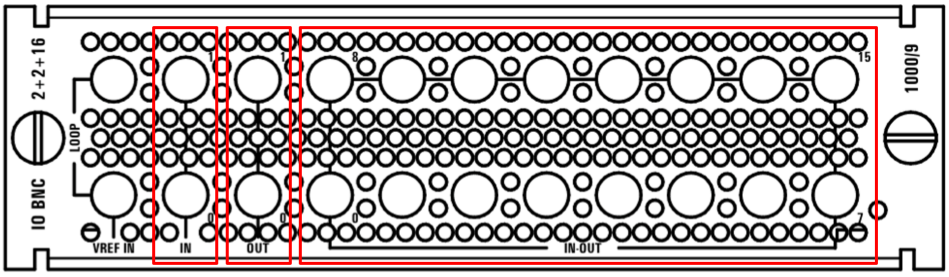
3.4.4 SDI 18+2



SDI 18+2: with Ref IN + 18 SDI IN + 2 SDI OUT (NO 4K)

- **Top row**: 10x 3G/HD/SD
- **Bottom row**: 10x 3G/HD/SD

3.4.5 SDI 2+2+16



SDI 2+2+16: with Ref IN + 2 SDI IN + 2 SDI OUT + 16 Reversible I/O (IN or OUT)

- **Top row**: 10x 3G/HD/SD
- **Bottom row**: 10x 3G/HD/SD

3.4.6 Bandwidth Restrictions

When specifying the I/O modules for SDI to IP or IP to SDI conversions, you must keep in mind the maximum amount of bandwidth available on the 40Gb connections:

- For incoming (RX) traffic, this is **36 Gbps** in SPS mode or **54 Gbps** in discrete mode.
- For outgoing (TX) traffic, this is **40 Gbps** in SPS mode or **80 Gbps** in discrete mode.

The tables below show how the maximum bandwidth can support different combinations of 12G and 3G signals. Please note that not all of the combinations given are supported by V__matrix I/O modules.

Incoming (RX) Traffic

SPS: Max 36 Gbps	Non-SPS: Max 54 Gbps
3x 12G or	4x 12G or
1x 12G + 8x 3G or	1x 12G + 14x 3G or
2x 12G + 4x 3G or	2x 12G + 10x 3G or
12x 3G	18x 3G

Outgoing (TX) Traffic

SPS: Max 40 Gbps	Non-SPS: Max 80 Gbps
3x 12G or	6x 12G or
1x 12G + 9x 3G or	1x 12G + 22x 3G or
2x 12G + 5x 3G or	2x 12G + 18x 3G or
13x 3G	26x 3G

1.5G Signals

When using 1.5G signals, there are more possibilities. "1.5G" can mean many different things, depending on the frame rate and stream format. For example:

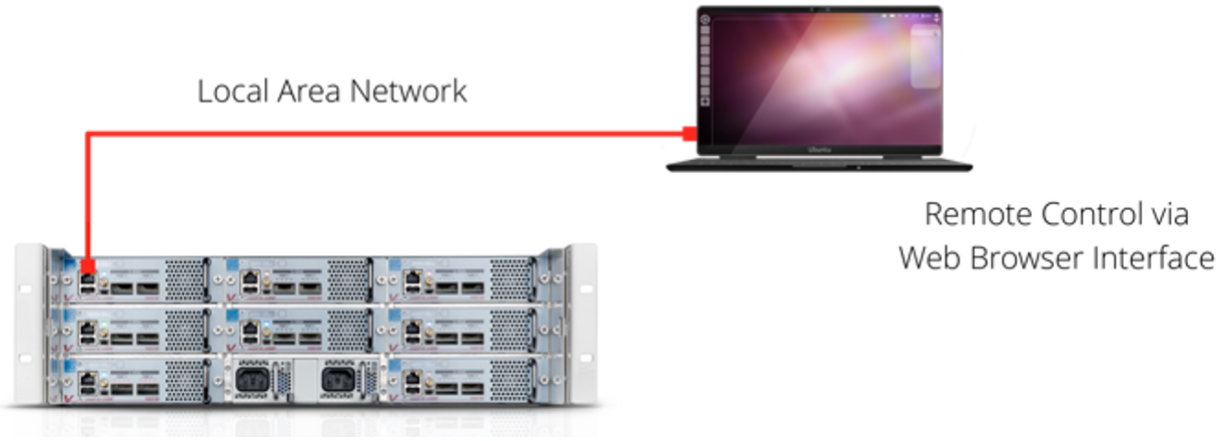
HD1080i @ 30fps (60 half-fields/s) in SMPTE ST 2110-20/-21 is 1.393 Gbps, in 25 fps it is 1.161 Gbps, but in SMPTE ST 2022-6 it is 1.540 Gbps.

4. Getting Started

Each C100 has the same connectivity options and runs as a stand-alone module. However, depending on the selected FPGA mode, the functionality will differ (software defined processing).

When starting a C100 for the first time, no configuration is available and the system needs to be setup completely. In this state, the blade has a set of default IP addresses only.

The configuration is stored locally and can be edited by opening a web browser connection from a remote computer. This can be achieved by connecting the computer's network interface directly to the C100's front MGMT port.



4.1 Possible Control Schemes

Two possible control schemes are supported: out-of-band control (via the management network) or in-band control (via the media network).

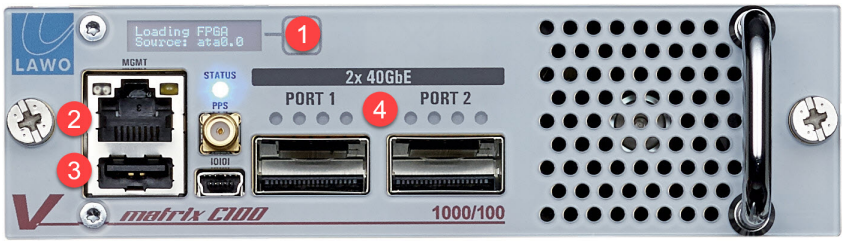
Out-of-band control can be achieved by connecting to management port 1 (on the front of the C100) or to management port 2 (via the central management module on the rear panel).

In-band control of a C100 is possible via any or all of the 2x 40GE or 2x 4x 10GE interfaces.

Depending on the configuration, up to 10 IP addresses (IPv4 or IPv6) are supported. As each interface belongs to its own name space and behaves like a separate device, each IP address must be unique.

4.2 C100 Connectivity

C100 (front view)



1 Push-button

The push-button cycles through the status display pages. You can use this to view the current IP address of the management port.

2 MGMT 100/1000 Port

The MGMT port provides a network connection to the C100 control system. It can be used to access the configuration via either the web-based user interface or Lawo's V__script.

To set up the C100, you should connect your computer directly to the MGMT port using a UTP cable.

Once the initial setup is complete, you can choose to use either the front or rear management port depending on the hardware and system design preference.

3 USB Port

The USB port provides access to the watchdog.

Optionally, you can connect two C100s to have advanced watchdog features. In this instance, use a female to male USB cable to connect the second C100 device.

4 2x 40GbE Ports

These ports connect the red and blue media networks. You can use either a 40GbE or 4x 10GbE QSFPs. Only optical QSFP transceivers and/or QSFP active optical cables are supported.

4.3 C100 Power-up

V__matrix 2 (front view)

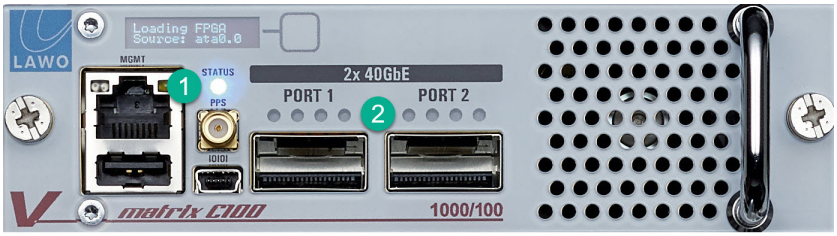
The V_matrix has no on/off switch but starts automatically when either power supply is connected to the mains.

You will hear the fans begin to operate, and the C100 status displays show the information "Loading FPGA". The system takes approximately 40 seconds to boot from power on, and loads the latest settings.

The LEDs on the PSU module indicate the status of the two power supplies:

- **Green** = power supply is active.
- **Off** = power supply is inactive (no mains input or internal PSU is faulty).

C100 LEDs (front view)



1 STATUS LED

This LED indicates the operating status of the C100.

2 QSFP LEDs

The first three LEDs (on Port 1) indicate whether the RAM calibration has succeeded or failed:

- **Green** = RAM calibration succeeded.
- **Red** = RAM calibration failed.

4.4 Connecting a Control Computer

After making the physical connections, you should configure the network settings for your computer's LAN port. The exact steps vary depending on your OS version.

The IP address must be unique, and set within the same range as that of the port you are connecting to. The subnet masks should be identical.

Default IP Settings

Each C100 can have up to 10 IP addresses depending on the configuration of the media network ports:

- When loading a FPGA mode with 4x 10GbE connectivity, there are up to 10 IPs.
- When loading a FPGA mode with 40GbE connectivity, there are up to 4 IPs.

For management, you can use either the front or rear ports depending on the hardware and system design preference.

The default settings are listed below.

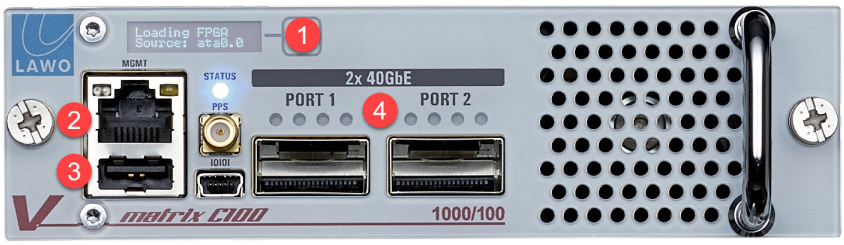
Port	Default IP	Port	Default IP
10GE port 1.0 or 40GE port 1	172.16.1.0	10GE port 2.0 or 40GE port 2	172.16.1.5
10GE port 1.1	172.16.1.1	10GE port 2.1	172.16.1.6
10GE port 1.2	172.16.1.2	10GE port 2.2	172.16.1.7
10GE port 1.3	172.16.1.3	10GE port 2.3	172.16.1.8
MGMT port 1 (front port)	172.16.1.4	MGMT port 2 (rear port)	172.16.1.9

Using MGMT port 1

If your computer is connected to the front MGMT port then, by default, this is accessed via IP: **172.16.1.4**.

If the C100 has been configured before and the IP is unknown, then press the push-button (1) to view the current IP settings in the status display.

C100 (front view)



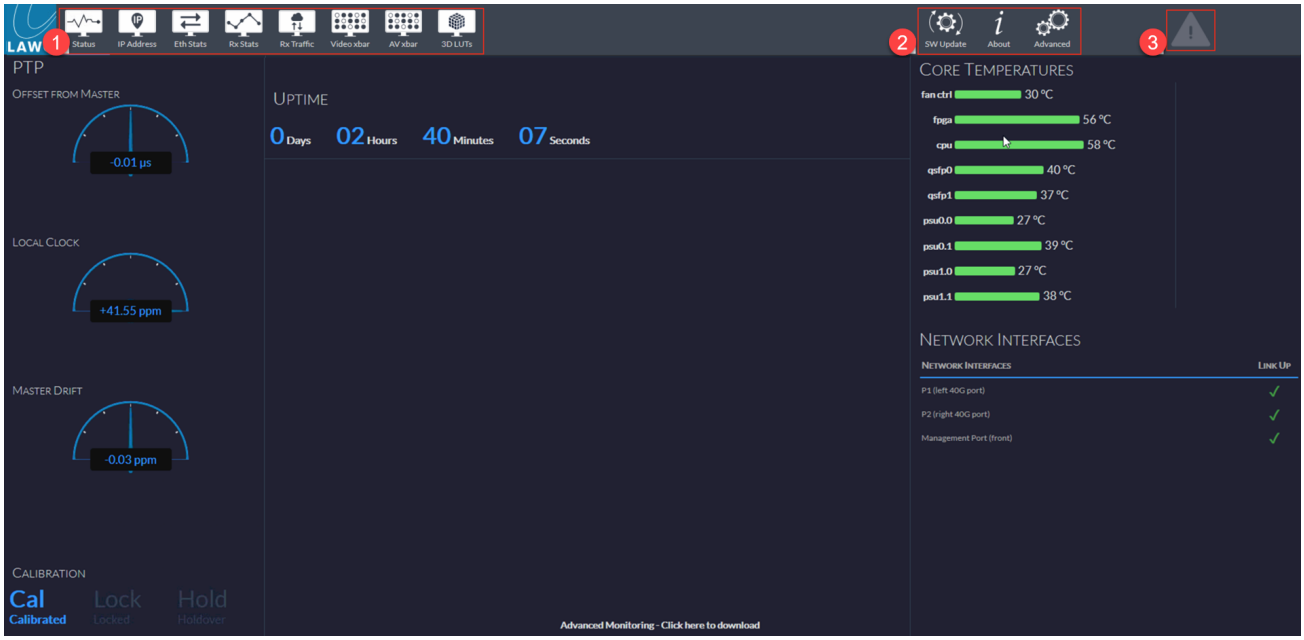
4.5 Browser Access & Configuration

To access the C100 configuration:

1. Open a web browser application. We recommend using Chrome.
2. Enter the management IP address (e.g. 172.16.1.4) into the URL field and press Enter.

If the connection is successful, the C100 "Status" page appears.

The headline at the top of the page is used for navigation.



It is divided into three areas:

1 Main Menu Bar

You can open any main page by clicking on a menu option - the current selection is underlined in blue (e.g. **Status**).

2 Advanced Menu Bar

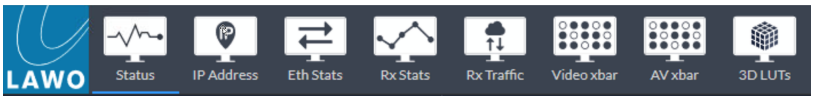
As above, but for more advanced functions.

Note that the **Advanced** menu opens the advanced part of the web interface. The operating principles for this page are described later.

3 System Alarm

If a new error is detected, then the warning triangle (3) lights in red. Click on the triangle to view the details.

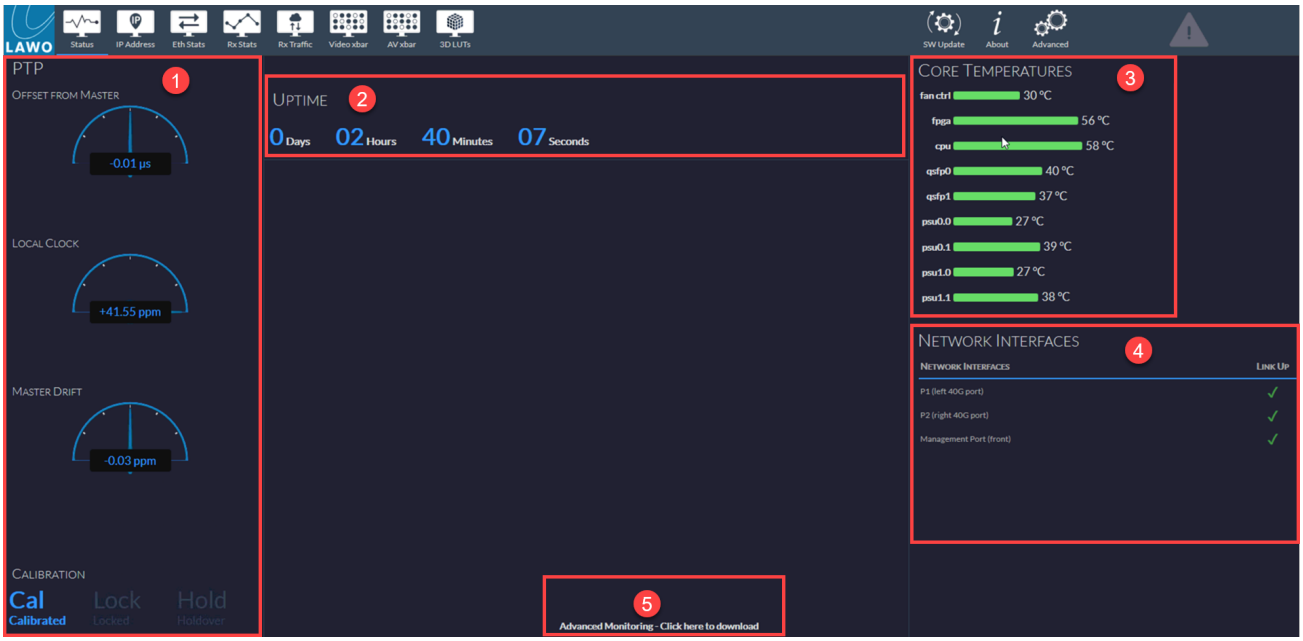
4.6 Main Menus



There are eight main pages which are used mostly for informational purposes.

- **Status** - shows device information and statuses.
- **IP Address** - shows network settings (including IP addresses and LLDP neighbour information).
- **Eth Stats** - shows network statistics (useful for checking CRC errors and the current payload).
- **Rx Stats** - shows detailed network statistics per RX receiver.
- **Rx Traffic** - shows detailed network traffic information per RX receiver.
- **Video xbar** - controls the internal video crossbars (created via the advanced page).
- **AV xbar** - controls the internal AV crossbars (created via the advanced page).
- **3D LUTs** - imports and configures the 3D LUTs (only available in vm__UDX).

4.6.1 Status



The Status page shows information about the C100.

1 **PTP** - the status of the PTP clock including calibration.

2 **Uptime** - the running time since start-up.

3 **Core Temperatures** - a snapshot of the current temperatures.

4 **Network Interfaces** - the current status of the network interfaces: link up or link down.

5 **Advanced Monitoring** - click here for more information about telemetry.

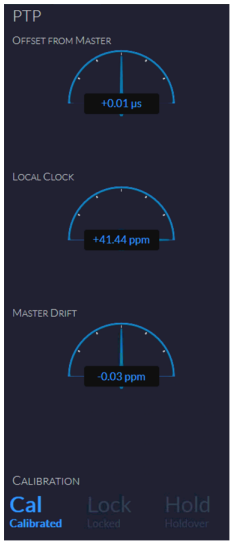
PTP Status

Of particular note is the **PTP** area which shows the current status of the PTP clock.

The three “speedometers” show the current offset once a PTP clock source has been configured. The CALIBRATION area shows the calibration status.

Example 1 (uncalibrated)

Example 2 (calibrated)



The C100's PTP clock source can be set to one of the following:

- PTP - arriving from the network.
- Analog Reference (Tri-level/Black Burst) - connected to a rear I/O module.
- SDI input - connected to the rear I/O module.
- Internal Clock - generated by the C100.

The PTP configuration is handled via the advanced page and is described later.

4.6.2 IP Addresses

The "IP Address" page shows information about the QSFPs and LLDP neighbours.

The **QSFP** tab allows you to monitor all of the network connections at the same time:

- Port 1 = red media network
- Port 2 = blue media network
- MGMT 1 = front management port
- MGMT 2 = rear management port

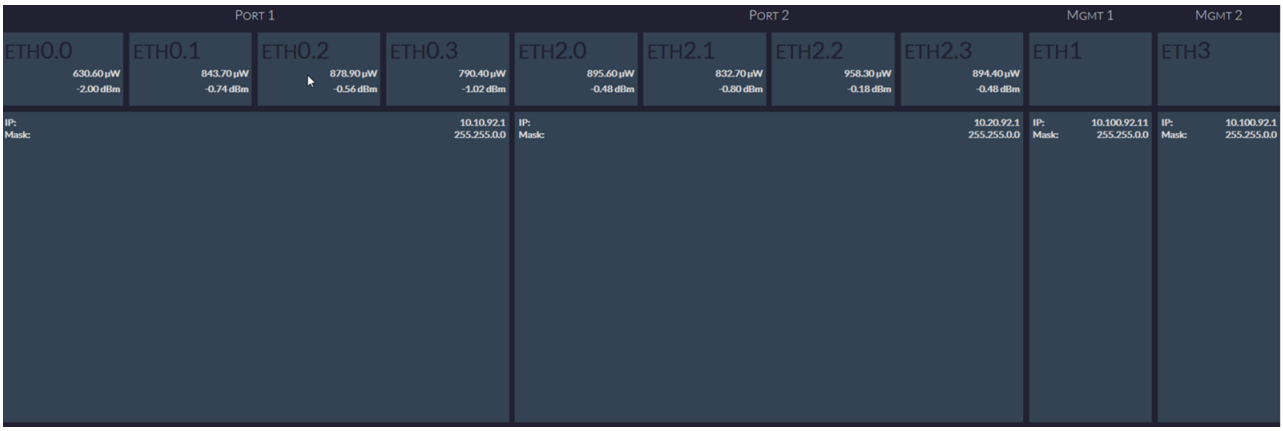
Port 1 and Port 2 are subdivided by four 10GbE lanes. In the example below we have a single cable (1x 40GbE) connected to eth0.0. When using a split cable to connect 4x 10GbE, you will have 4 IPs to monitor.

The configuration of the IP addresses is handled via the advanced page and is described later.

The link status for each of the physical connections can be checked by navigating back to the "Status" page (under NETWORK INTERFACES).

1x 40GbE Connection

"IP Address" page



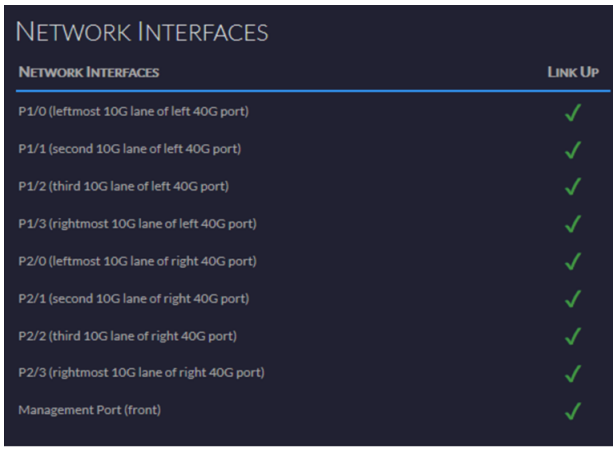
"Status" page

4x 10GbE Connection

The next example shows the use of a 40GbE split cable to 4x 10GbE. Port 1 is cabled for the red network and has 4 individual IPs. Port 2 is cabled for the blue network and has 4 individual IPs.

When using a split cable, you need to be sure to distribute the TX and RX between the different “ports” to be sure you don’t overload the 10GbE connections. Mgmt1 is setup to be used for the control network via the front and Mgmt 2 is used for the rear panel management connection.

"Status" page



LLDP Information

The area at the bottom of the "IP Address" page shows the details of LLDP neighbors. LLDP is used to get link layer discovery protocol information. It allows you to see which port each link is connected to on the network switch.

In the example below, we are connected to an Arista switch for the red and blue media networks. The management ports are not connected to the Arista switch and, therefore, do not show up in the LLDP overview.

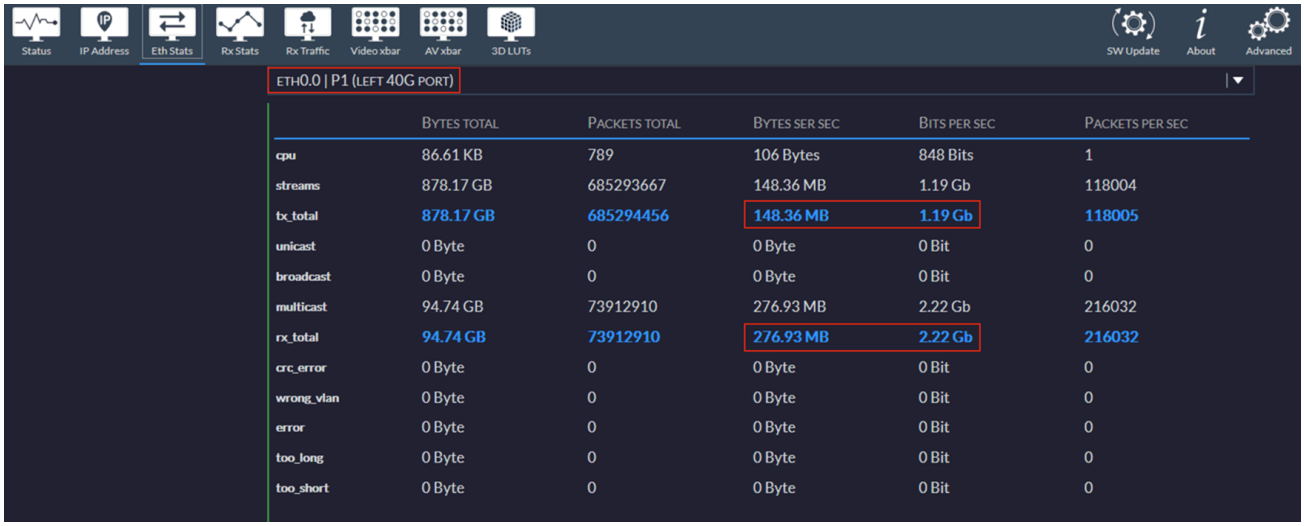
"IP Address" page (LLDP info)

PORT	LLDP NEIGHBOR	SYSTEM NAME	CHASSIS ID	INTERFACE	MGMT ADDRESS	SYSTEM DESCRIPTION
port0 (ns_eth0.0)	LLDP neighbor #0	MediaSwitch9	44:4c:a8:47:4d:6f	Ethernet7/1	10.101.101.9	Arista Networks EOS version 4.21.1F running on an Arista Networks DCS-7050QX-32S
port1 (ns_eth2.0)	LLDP neighbor #0	MediaSwitch9	44:4c:a8:47:4d:6f	Ethernet21/1	10.101.101.9	Arista Networks EOS version 4.21.1F running on an Arista Networks DCS-7050QX-32S

4.6.3 Eth Stats

The "Eth Stats" page shows the network statistics for a port. It is possible to select the port you wish to monitor.

In the example below, we have 1x 2110-20 1080i TX and 1x 2110-20 1080p RX. Notice the difference in the data rate.

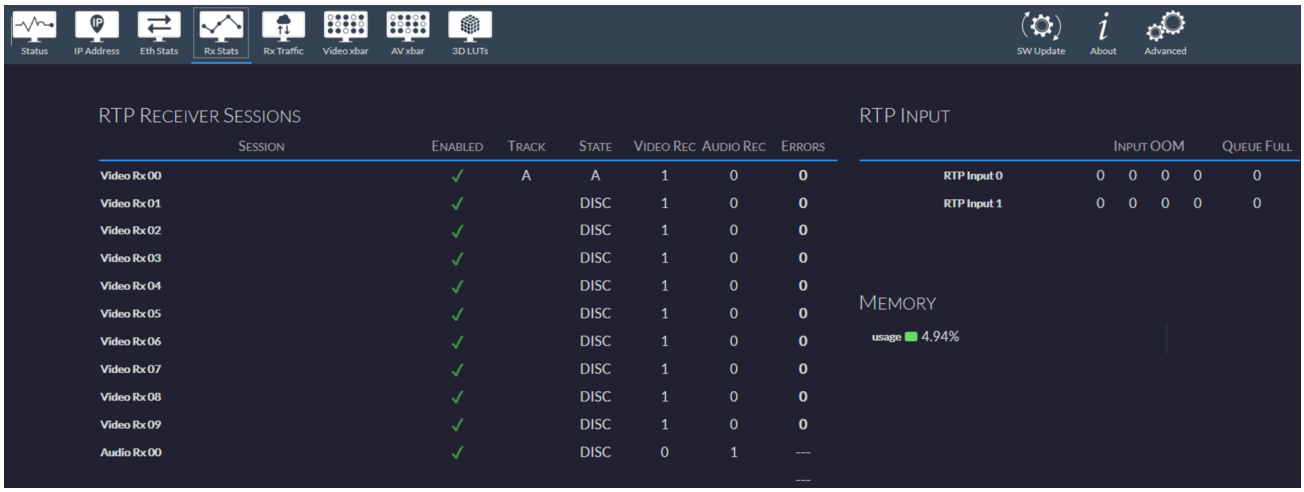


	Bytes total	Packets total	Bytes ser sec	Bits per sec	Packets per sec
cpu	86.61 KB	789	106 Bytes	848 Bits	1
streams	878.17 GB	685293667	148.36 MB	1.19 Gb	118004
tx_total	878.17 GB	685294456	148.36 MB	1.19 Gb	118005
unicast	0 Byte	0	0 Byte	0 Bit	0
broadcast	0 Byte	0	0 Byte	0 Bit	0
multicast	94.74 GB	73912910	276.93 MB	2.22 Gb	216032
rx_total	94.74 GB	73912910	276.93 MB	2.22 Gb	216032
crc_error	0 Byte	0	0 Byte	0 Bit	0
wrong_vlan	0 Byte	0	0 Byte	0 Bit	0
error	0 Byte	0	0 Byte	0 Bit	0
too_long	0 Byte	0	0 Byte	0 Bit	0
too_short	0 Byte	0	0 Byte	0 Bit	0

4.6.4 RX Stats & RX Traffic

The "Rx Stats" page shows details for the rtp receivers. A rtp receiver must be set up before you can monitor the details. In our example below, RTP receiver session VideoRx00 is active on track A; all other receivers are disconnected. On the right, you can monitor the Input Out of Order Messages (INPUT OOM) and current memory use.

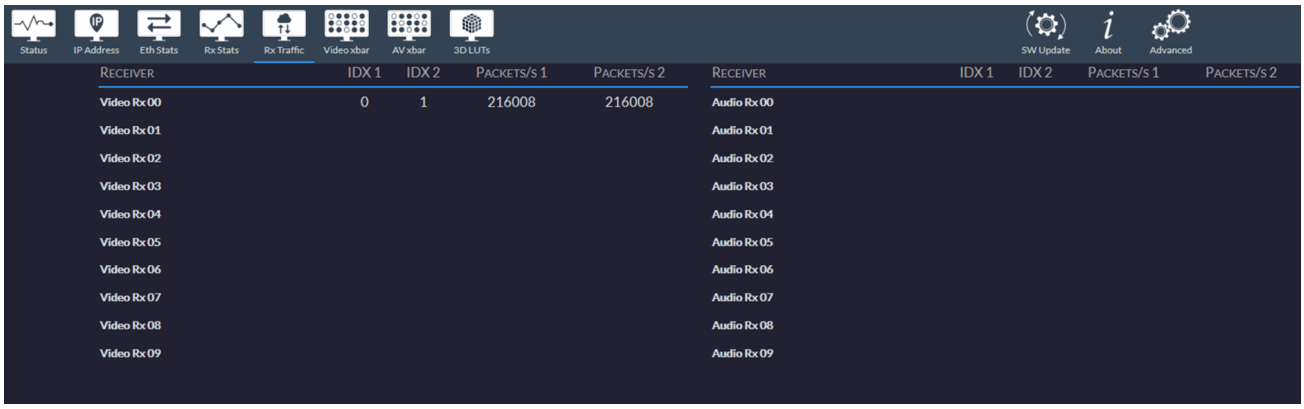
"Rx Stats" page



The "Rx Traffic" page shows more details for the receivers.

IDX1 is your left Rx, and IDX2 is your right Rx when using SMPTE ST2022-7 redundancy. In this case, the packet value should be identical on both legs.

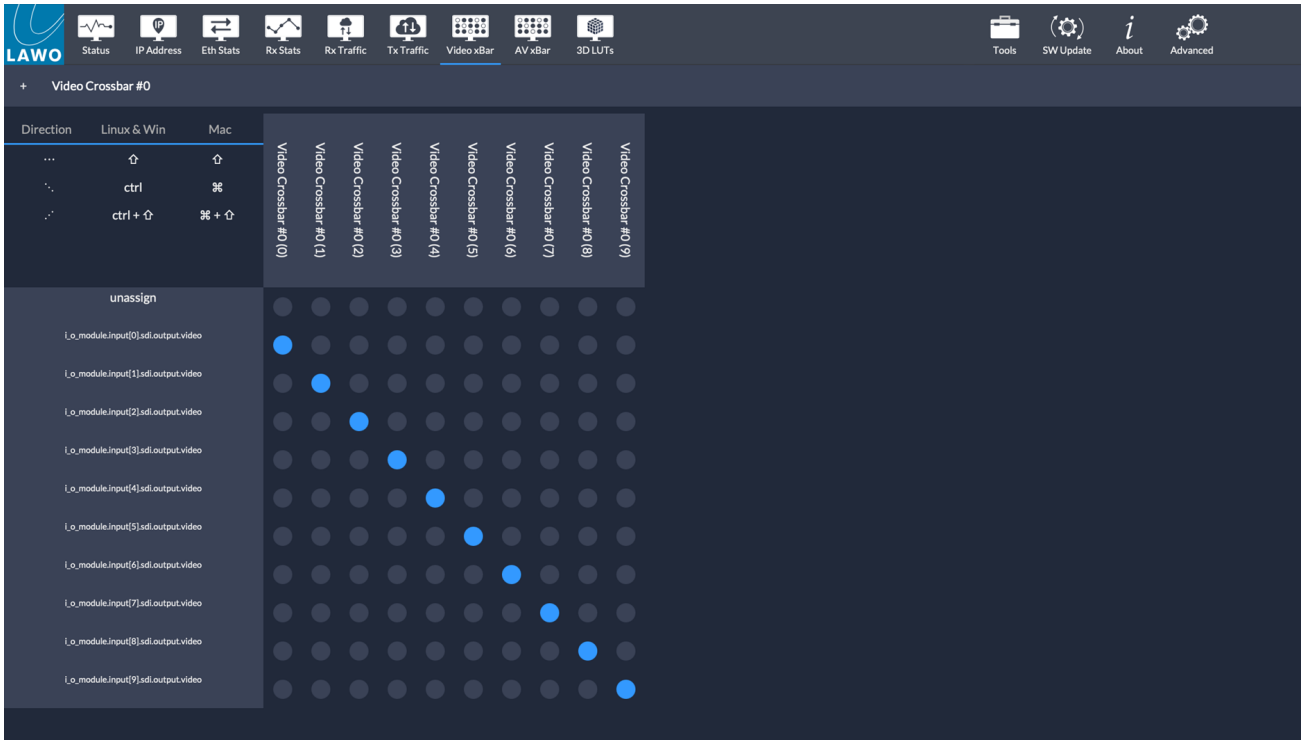
"Rx Traffic" page



4.6.5 Video xbar

The "Video xbar" page provides control of the internal video crossbars (created via the advanced page).

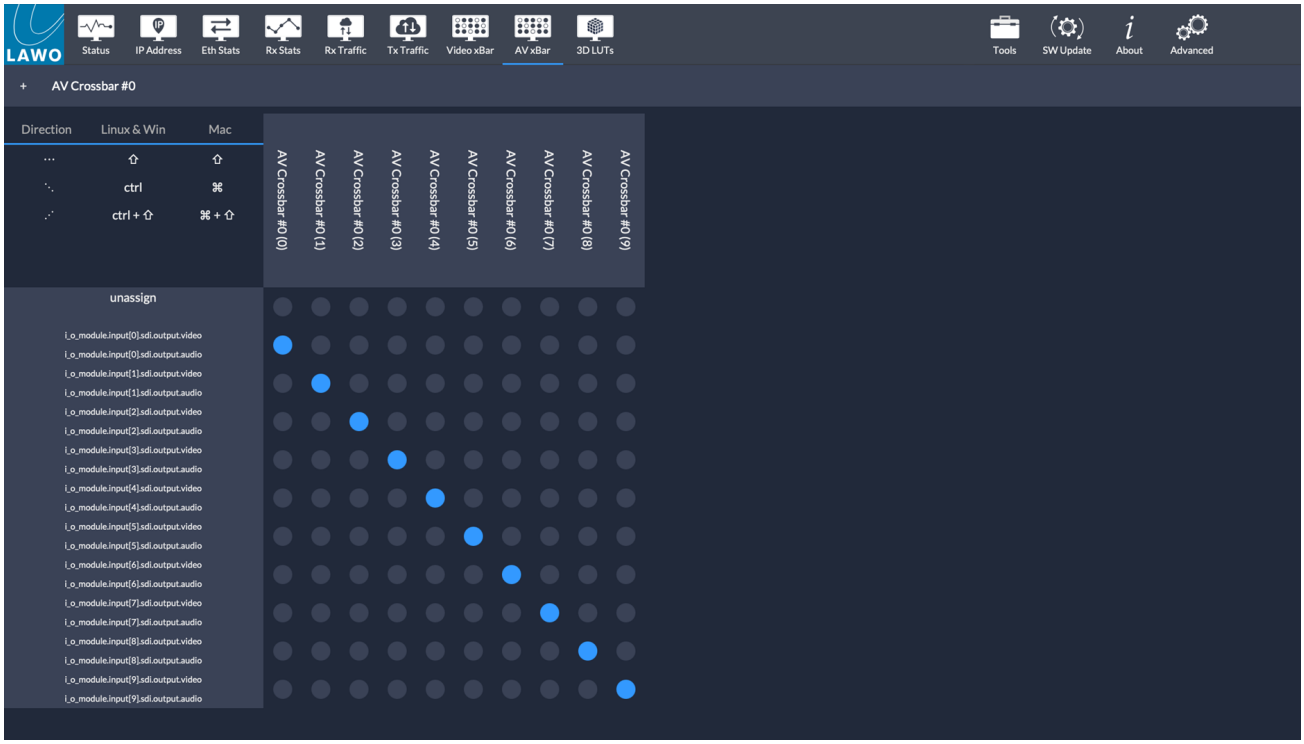
By default, the configuration has no video crossbars. Once configured, you can select them from the drop down menu.



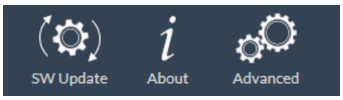
4.6.6 AV xbar

The "AV xbar" page provides control of the internal Audio Video crossbars (created via the advanced page).

By default, the configuration has no AV crossbars. Once configured, you can select them from the drop down menu.



4.7 Advanced Menus



The advanced menu bar provides access to the more advanced functions.

- **SW Update** - updates the system software (via the V__matrix.zip installation package).
- **About** - shows information about the system.
- **Advanced** - opens the advanced page.

4.7.1 Software Update

The "SW Update" page can be used to update the system.

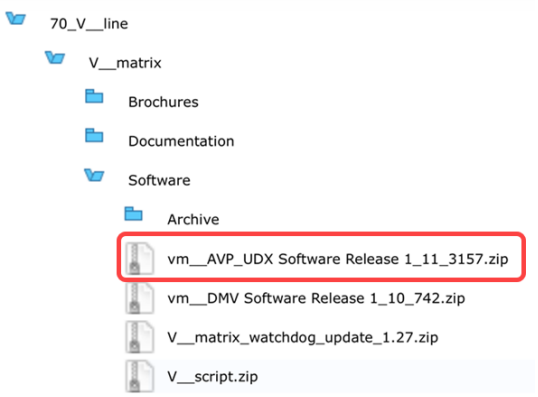
Preparation

Please check the compatibility of the V__matrix release with the control system and upgrade your other system components as required. Information on C100 compatibility with other Lawo products can be found here:

https://confluence.lawo.com/display/PKD/VSM+Release+Information+and+Compatible+Software+Bundles

The software installation package can be downloaded from the Lawo website (after login).

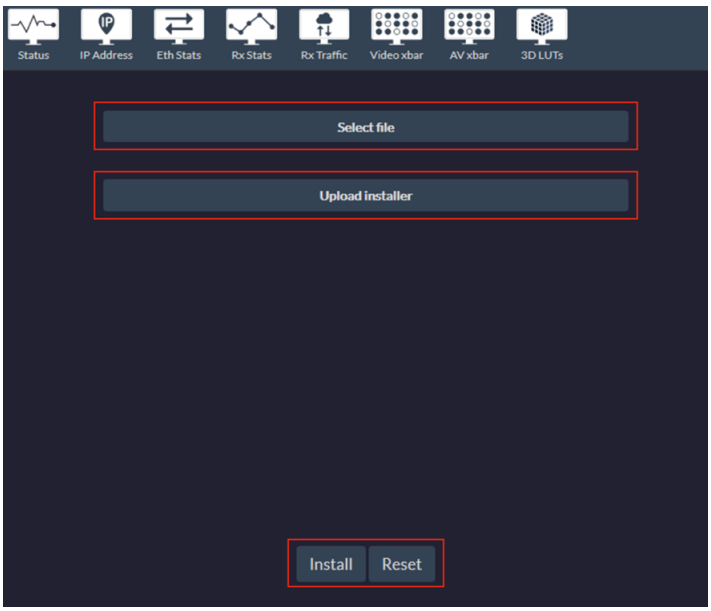
Lawo Website (Downloads area)



Each V__matrix release comes with two installation files. A normal file for partitions one and two of the C100, and a recovery partition update (marked with filename ending "RPU"). It is highly advised to run the normal installer twice to update partitions one and two, then run the RPU installer to update the recovery partition.

Update Procedure

1. Select **SW Update**, from the Advanced Menu Bar, to open the update page.



2. Use **Select file** to select the update file (unzip after download from the Lawo website).

The **Reset** button can be used to cancel the selected package if you need to change the file selection.

3. Once the correct file is selected, click on **Upload Installer**.
4. Once the package is uploaded, you can select **Install**. It is recommended to select **Install** twice to make sure that the two system partitions are updated.

An overview of the upload/install process is shown in the space below **Upload Installer**.

4.7.2 About

1. Select **About**, from the Advanced Menu Bar, to show information about the system.

The boxes at the top represent the V__matrix frame slots as viewed from the front. Here you can see the status of the two power supplies and C100 processing blades.

2. Select a blade to interrogate its information - the current selection is highlighted in blue and the "Device C100" area updates.

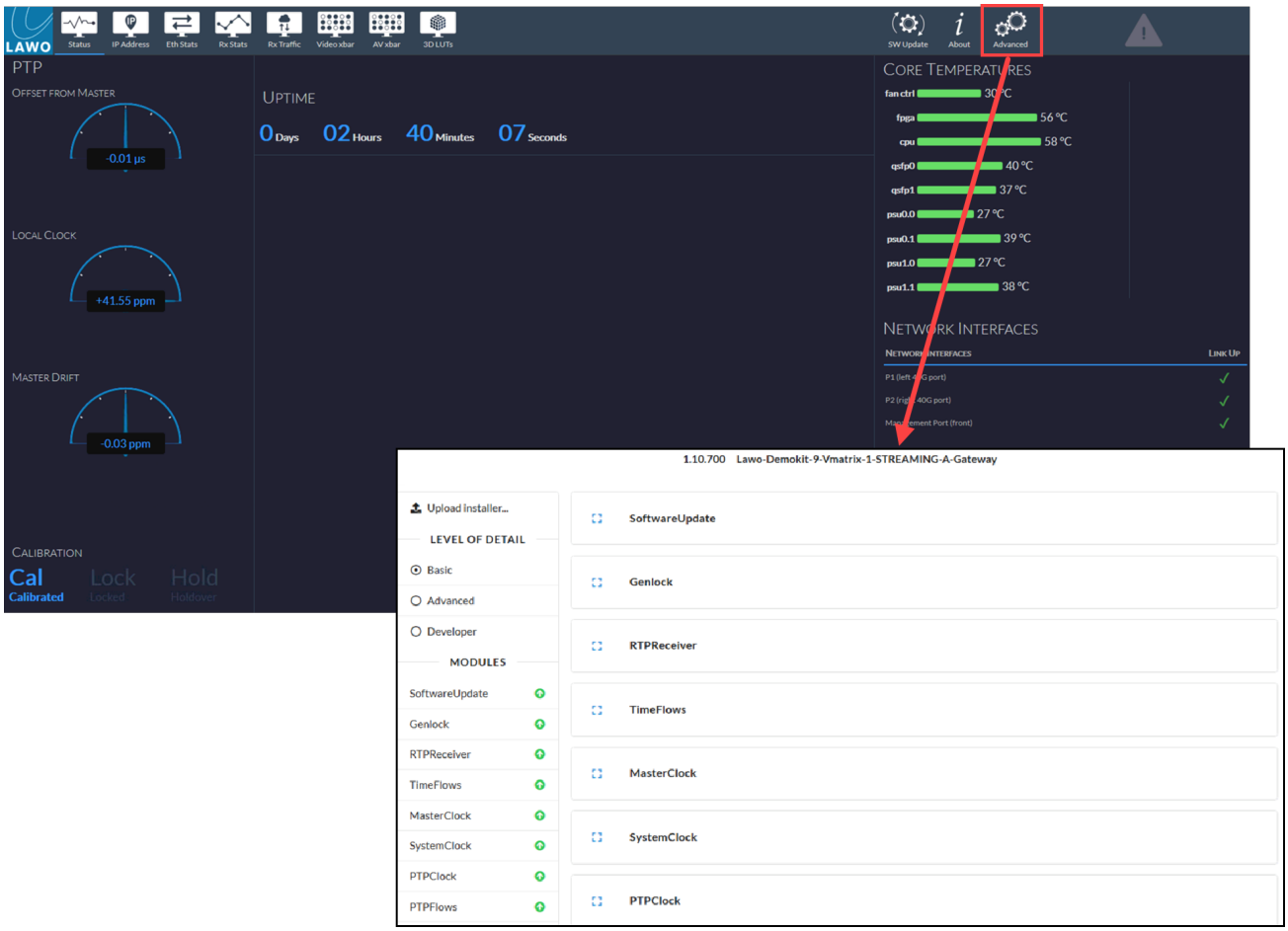
This area shows the current installed software version, Barebox, FPGA and Watchdog version. It also detects and shows the rear I/O module type and version.

3. Hover over a C100 blade to see its IP list.

4.7.3 Advanced

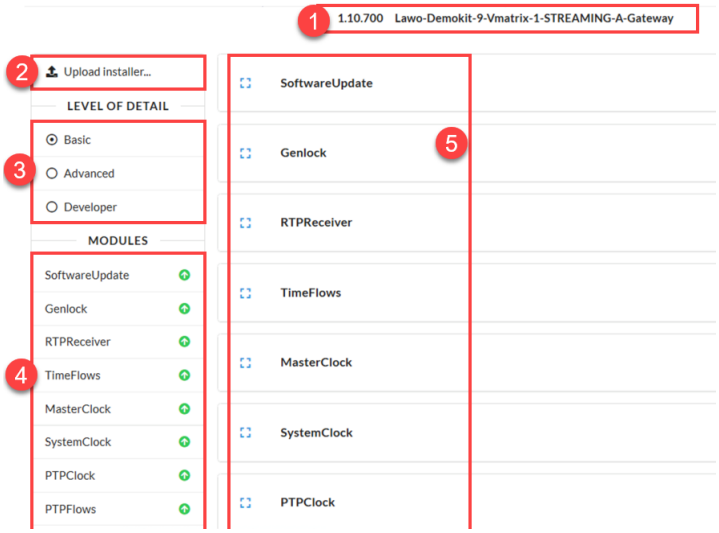
Most of the configuration is handled via the advanced part of the web interface.

1. Select **Advanced**, from the Advanced Menu Bar, to open the page.



The advanced page is auto-generated from the C100's internal data structure and mainly useful for debug purposes. The main configuration of the C100 is done via it's websocket and JSON based API using scripts (Javascripts or Typescripts). The websocket interface is also used by external control systems such as Lawo's VSM and theWall. The rest of this chapter will introduce you to the advanced page and it's configuration options.

Advanced Page (top)



1 Software Version & C100 Name

Displayed at the top are the software version number and the configured name for the C100.

2 Upload Installer

Via **Upload Installer** you can quickly upload the unzipped installer file.

3 Level of Detail

The level of detail determines which sub menus and options are available for each module.

It is recommended to stay in **Basic** mode at the beginning. We will advise when it is necessary to use **Advanced** mode.

4 Module Shortcuts

This area provides a list of shortcuts to all the different configuration modules. If a module is not available, then it is grayed out.

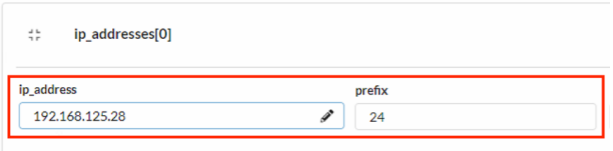
Whenever you click on a shortcut, the page jumps to a placeholder. Note that this does not open the module, but merely navigates to the correct position on the page. If you wish to open the module, click on its blue icon.

5 Modules

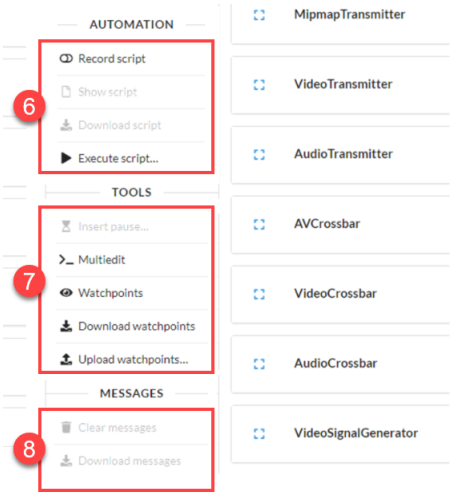
On the right you will see all of the different configuration modules. In each case, click on the blue icon to open each module. The visible settings and sub menus are revealed. These are controlled by the Level of Detail (3).

Settings are adjusted either by making a selection from a drop-down menu or typing in a value. When entering a value, you must press Enter to apply. If a field is still in edit mode, then a pencil icon is shown.

In the example below, the "ip_address" is in edit mode, while the "prefix" is already applied.



Advanced Page (middle)



6 Automation

Automation allows you to to record the clicks and values you apply in the web interface and save them as a Javascript file. With an advanced editor like Notepad++ (PC) or Visual Studio Code (macOS) you can edit your script. It is highly advised not to use a basic text editor as this might have an invisible influence on the script.

You can also execute pre-recorded scripts.

7 Tools

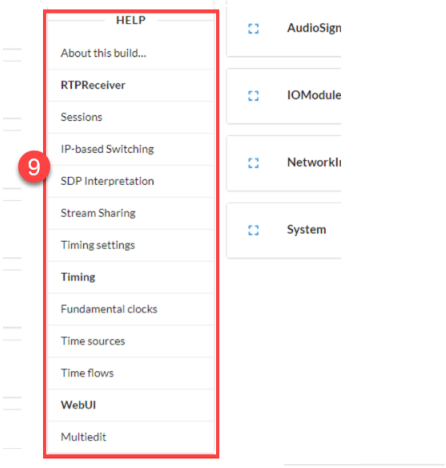
This area provides some useful tools:

- Multiedit can be used to perform a batch edit. It is also a great way to get status reports for the C100.
- Watchpoints allow you to monitor the changed values on a C100.

8 Messages

This area shows messages which are generated by the system after applying changes or when something happens. There are options to clear or download the list of messages.

Advanced Page (bottom)



9 Help

The help section includes tips and tricks, plus information on how to set up the device.

The content will evolve with each release and highlight new features or configuration methods.

5. V__script

V__script is a Linux and macOS application that allows you to:

- Monitor the current installed version.
- Backup and restore a C100 blade using snapshots.
- Upgrade or downgrade multiple C100 blades.
- Run scripts on multiple C100 blades.

No license is required to use the application.

The Linux and macOS version have the same features.

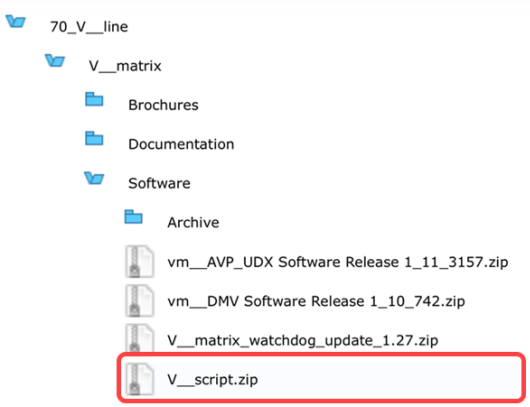
The Linux version can run on a VirtualBox VM in Windows environments.

The computer must be connected to the current management IP of the C100 (as described earlier).

5.1 Downloading the Installation Package

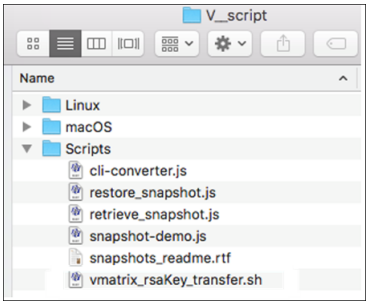
The "V_script.zip" installation package can be downloaded from the Lawo website (after login).

Lawo Website (Downloads area)



The package contains the application for macOS and Linux, plus a few basic scripts in the "Scripts" subfolder.

V__script.zip (unzipped)



The Linux version comes with a Linux AppImage or quick-run version.

The macOS version has two types of zip but they both contain the same application. The app should be moved to the "Applications" folder.

5.2 Installing V__script on macOS

On macOS, unzip the "mac.zip" file, and move the app to the "Applications" folder:

V__script can now be launched by double-clicking on the app icon.

As the application is not an official download from the Apple AppStore, you may need to confirm that you want to open it before using it the first time.

Updating the Version

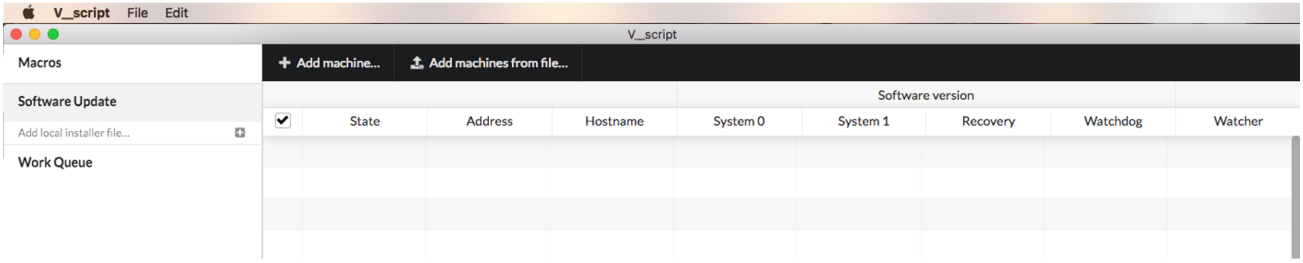
To update to a newer version of V__script, download the new version, unzip and replace the old application with the new one. You must confirm that you wish to replace the old version with the new.

If the app is already open, then it is not possible to overwrite. Close the app and try again.

5.3 First Steps

1. When you open V__script for the first time, you will see an empty window like this.

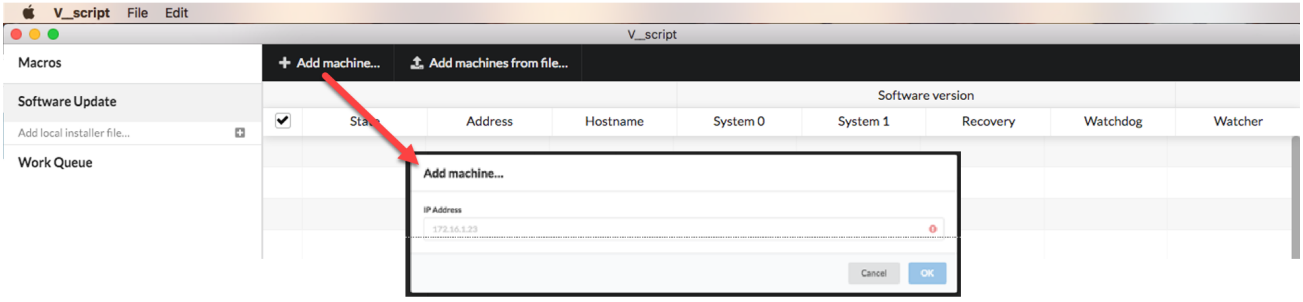
V__script (no machines configured)



2. A C100 blade can be added by selecting the **+ Add machine...** button.

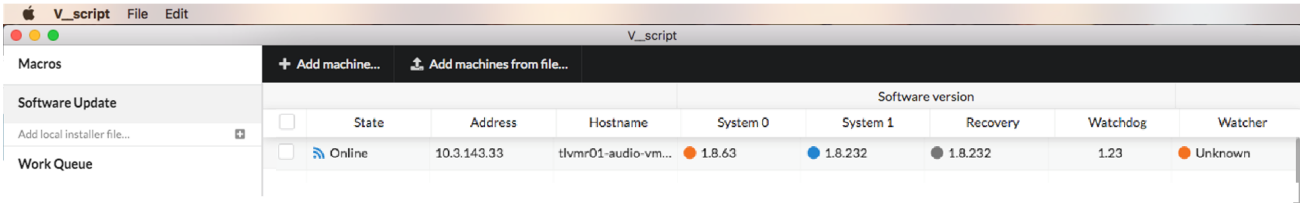
This opens a pop-up window where you can enter the management IP address of the blade.

V__script (add machine)



3. Type in the IP and select **OK** - the blade is added and V__script will connect.

V__script (machine connected)

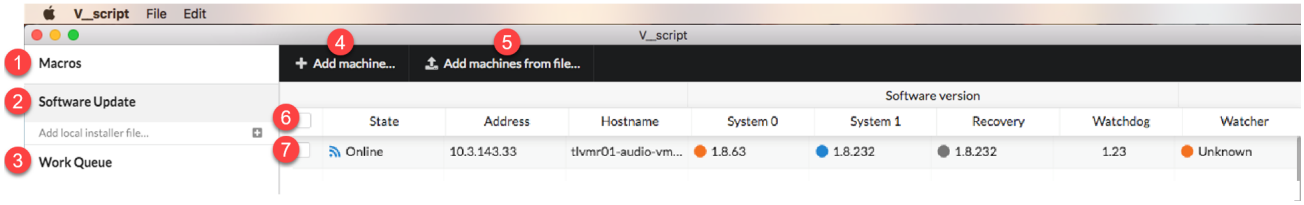


4. Repeat steps 2 and 3 to add all of the C100s you wish to monitor.

Each entry appears as a separate line in the machines list.

5.4 Using V__script

V__script (configuration window)



The V__script operating window is divided into the following areas.

1 Macros

Tab with imported JavaScript macros.

2 Software Update

Tab with imported software update installer files.

3 Work Queue

Overview of running and finished tasks.

4 Add machine...

Use this button to add a C100 manually (as described earlier).

5 Add machines from file...

Use this button to add several C100 machines from a pre-saved file.

6 Select All

Use this button to select all C100 machines in the list.

7 Select

Use this button to select the C100 machine individually. When working with several C100s, you can select multiple entries if you wish.

5.5 Software Partitions

By default, each C100 has three software partitions: two active partitions (System 0 and System 1) and one recovery partition.

When rebooting, the system toggles between System 0 and System 1. The Recovery partition is used when the system cannot boot from either 0 or 1.

The status of each partition is shown in the V__script machines list.

V__script (configuration window)

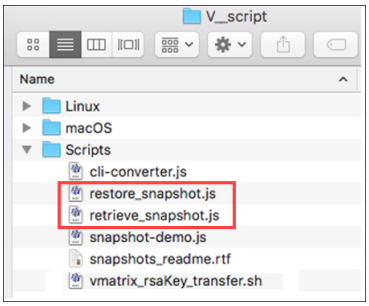
The blue dot identifies the active partition - in our example, System 0.

System 0 and System 1 should *always* carry the same version, except when a software update is in progress.

5.6 Adding Scripts

The V__script installation package includes a few default scripts inside the "Scripts" subfolder..

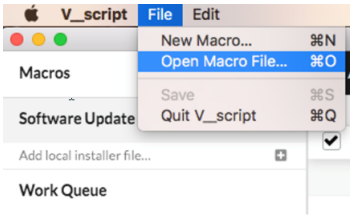
V__script.zip (unzipped)



The "retrieve_snapshot.js" and "restore_snapshot.js" scripts can be used to backup and restore a C100 blade. They can be added to V__script as follows.

1. Select **File** -> **Open Macro File...** (or press CMD+O).

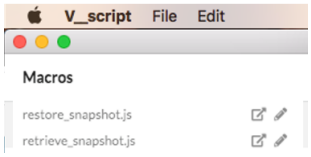
V__script (File menu)



2. Select the "retrieve_snapshot.js" file followed by **Open**.
3. Repeat the steps to import the "restore_snapshot.js" file.

Once added, the scripts appear in the **Macros** area on the left of the configuration window.

V__script (with Macros added)



In each case, click on the macro name to load it. Click on the pencil icon to edit the name.

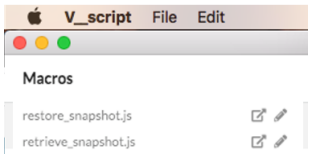
5.7 Backing up a C100

The "retrieve_snapshot.js" script can be used to backup a C100 blade to a snapshot.

Attention: The C100 will reboot once the backup is complete. To prevent this, you can adapt the script to not reboot.

To run the script:

1. In the **Macros** area, click on the script name: **retrieve_snapshot.js**.

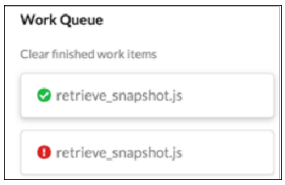


2. Select the **Run** button.

3. Enter the IP of the C100 you wish to retrieve a snapshot from and select **OK**.

4. Select the folder in the Finder window where you wish to store the snapshot

5. In the **Work Queue**, you will see either a green icon (to confirm a successful operation), or a red icon (to report an error). Click on the item for more details.

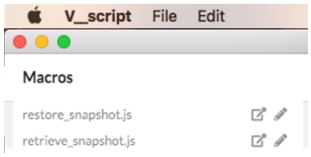


Attention: If a previous snapshot exists, then V__script will report an error. In this instance, rename the existing folder and then try the steps again.

5.8 Restoring a C100

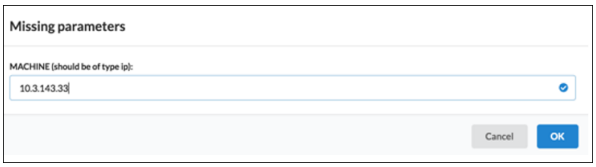
The "restore_snapshot.js" script can be used to restore a previously saved snapshot to a C100 blade. Please note that this does not work across different software versions.

1. In the **Macros** area, click on the script name: **restore_snapshot.js**.



2. Select the **Run** button.

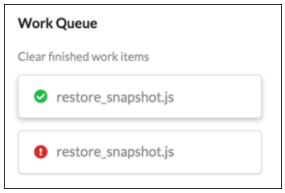
3. Enter the IP of the C100 blade you wish to restore and select **OK**.



4. Select the folder in the Finder window where the snapshot is stored.

This must be the root folder of the snapshot.

5. In the **Work Queue**, you will see either a green icon (to confirm a successful operation), or a red icon (to report an error). Click on the item for more details.



5.9 Upgrading or Downgrading a C100

V__script can be used to upgrade or downgrade the software version running on a C100 blade. This provides an alternative to using the "Software Update" page in the web browser interface.

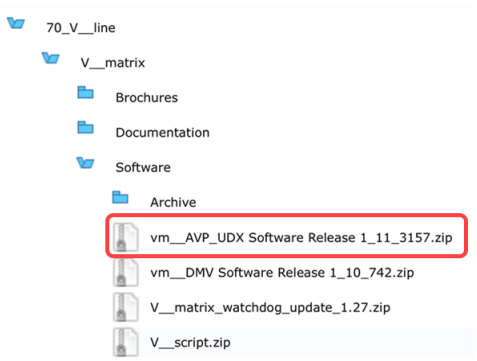
Preparation

Please check the compatibility of the V__matrix release with the control system and upgrade your other system components as required. Information on C100 compatibility with other Lawo products can be found here:

https://confluence.lawo.com/display/PKD/VSM+Release+Information+and+Compatible+Software+Bundles

The software installation package can be downloaded from the Lawo website (after login).

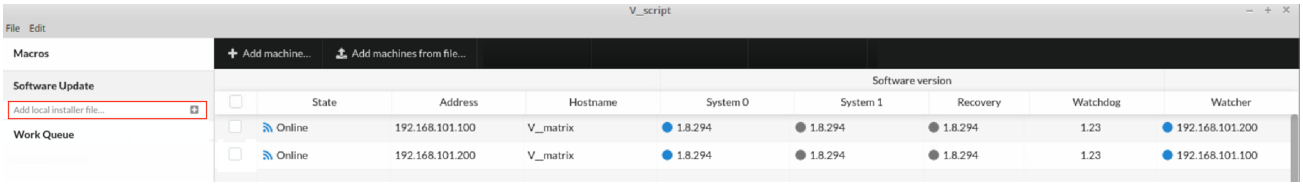
Lawo Website (Downloads area)



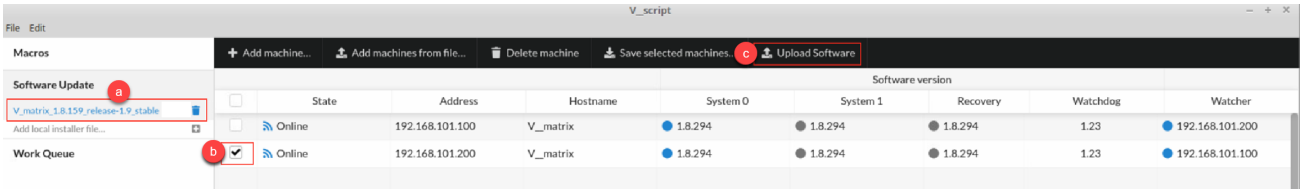
Each V__matrix release comes with two installation files. A normal file for partitions one and two of the C100, and a recovery partition update (marked with filename ending "RPU"). It is highly advised to run the normal installer twice to update partitions one and two, then run the RPU installer to update the recovery partition.

Update Procedure

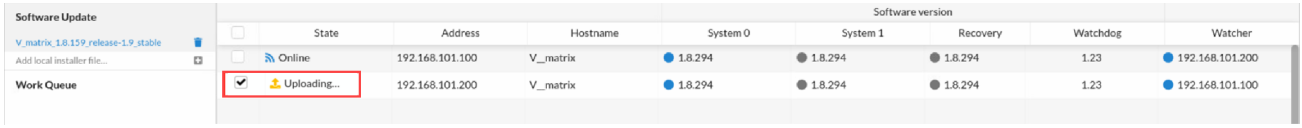
1. Add the "Software Update" package to V__script by clicking on the **+** symbol and selecting the file.



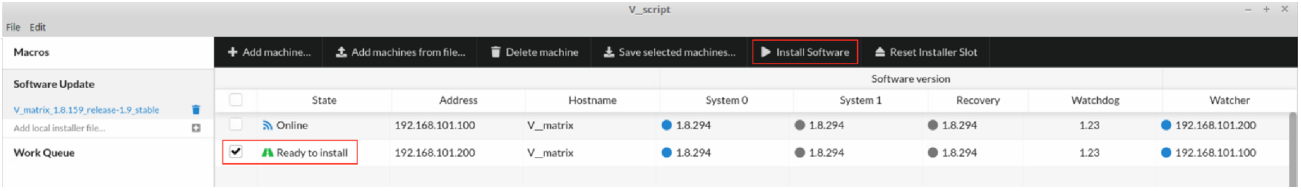
2. Select the name of the "Software Update" package, followed by the C100 you wish to upgrade. Then select the **Upload Software** button.



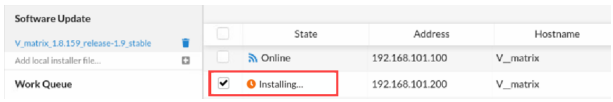
The state of the C100 changes to "Uploading".

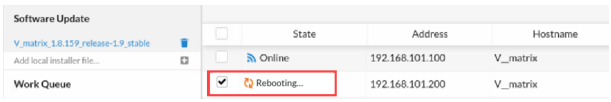


4. When the state shows "Ready to install", select the **Install Software** button to execute the upgrade.



The state of the C100 changes to "Installing" and then to "Rebooting".

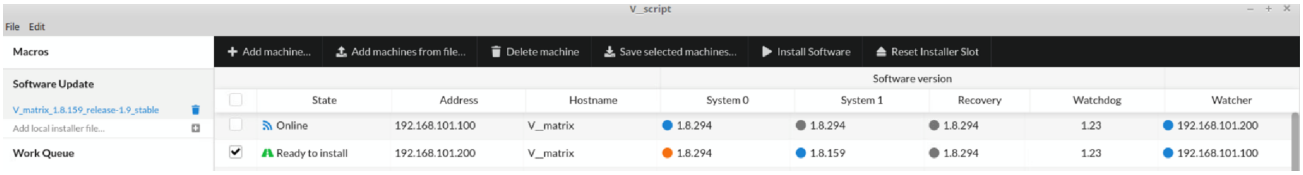




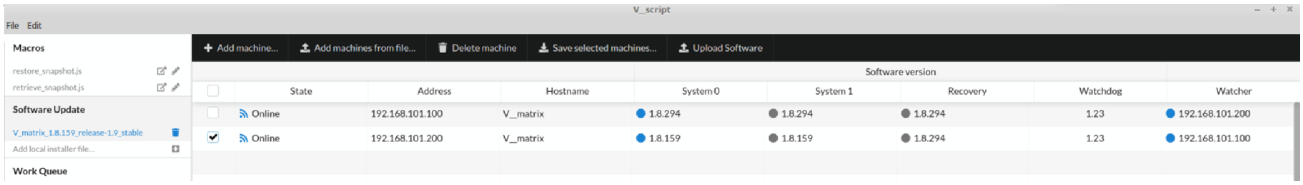
The upgrade of the first partition is now complete.

5. Reselect the package and follow the upload/install steps again to upgrade the second partition.

In the example below the upgrade of System 1 is complete and System 0 is ready to install.



6. After completing the process twice, the two active partitions (System 0 and System 1) should have the same version.



For safety reasons, the upgrade is a 2-step process to provide a roll-back option. The recovery partition is only updated if you use the RPU installer. The blue dot identifies the active partition.

After upgrading a C100 blade, all IP settings remain the same. However, other parts of the configuration may be lost. You should re-run your updated configuration scripts.

Before updating, please consult the release notes of your desired software version.

6. Network Configuration

To use a C100 in production, you will need to configure its network interfaces.

6.1 Network Port Summary

Each C100 has at least two IP addresses to configure (1 media network + 1 management), but can have up to ten depending on the configuration of the media network ports:

- When using a QSFP cable with 4x 10GbE to 40GbE, there are up to 10 different configurable ports.
- When using a QSFP cable with 40GbE connectivity, there are up to 4 different configurable ports.

The table below describes the port numbering for each mode.

	4x 10GbE mode	1x 40GbE mode
Red Media Network	Port 0 to 3	Port 0
Blue Media Network	Port 4 to 7	Port 1
Front Management port	Port 8	Port 2
Rear Management port	Port 9	Port 3

For management, you can use either the front or rear ports depending on the hardware and design preference.

6.2 Configuring a Network Interface

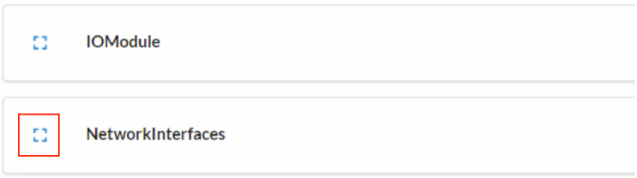
The example which follows describes how to configure port 0 (P1/0). This is the leftmost 10GbE lane of the first 40GbE port.

Once you know how to set one network interface, you can set them all.

6.2.1 Open Network Interfaces

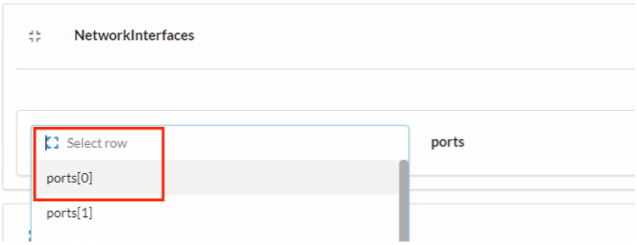
Start by opening the advanced page in the usual manner.

Open **Network Interfaces** by clicking on its shortcut in the MODULES list, and then clicking on its icon.



6.2.2 Select Port

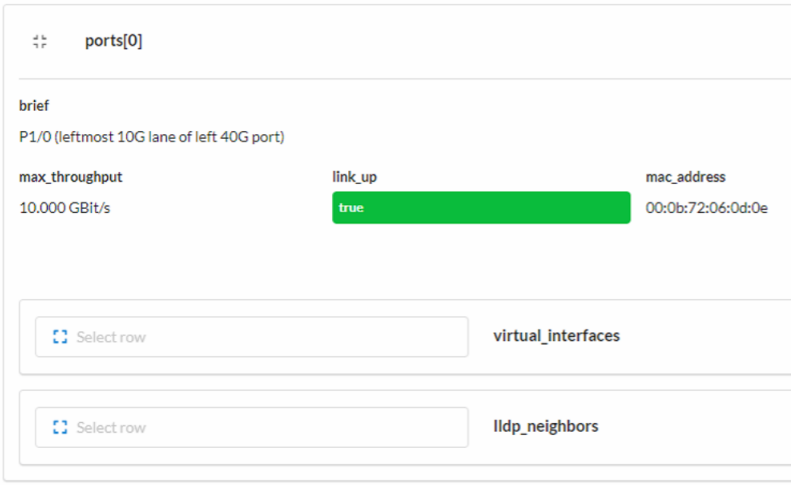
Choose a port from the drop-down menu - for our example, **ports[0]**.



6.2.3 Port Details

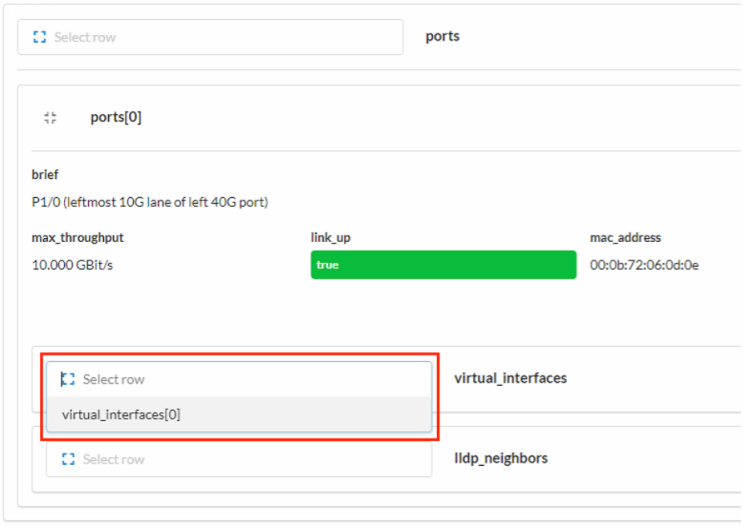
The page updates to show information about the port such as its name and location, maximum throughput, link status and MAC address.

In **Basic** mode, you will see options for the virtual interfaces and LLDP neighbors.



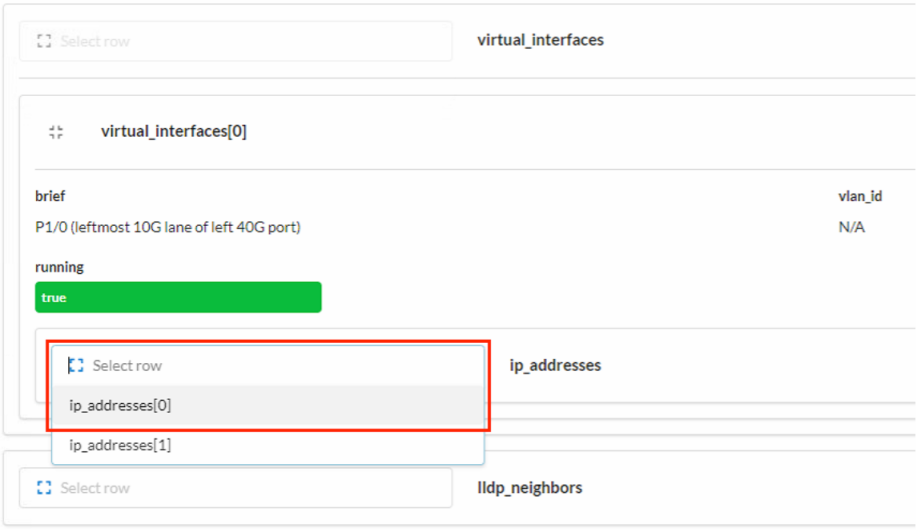
6.2.4 Select Virtual Interfaces

The IP address of the NIC is configured via the virtual interfaces, so choose **virtual interfaces[0]** from the drop-down menu.



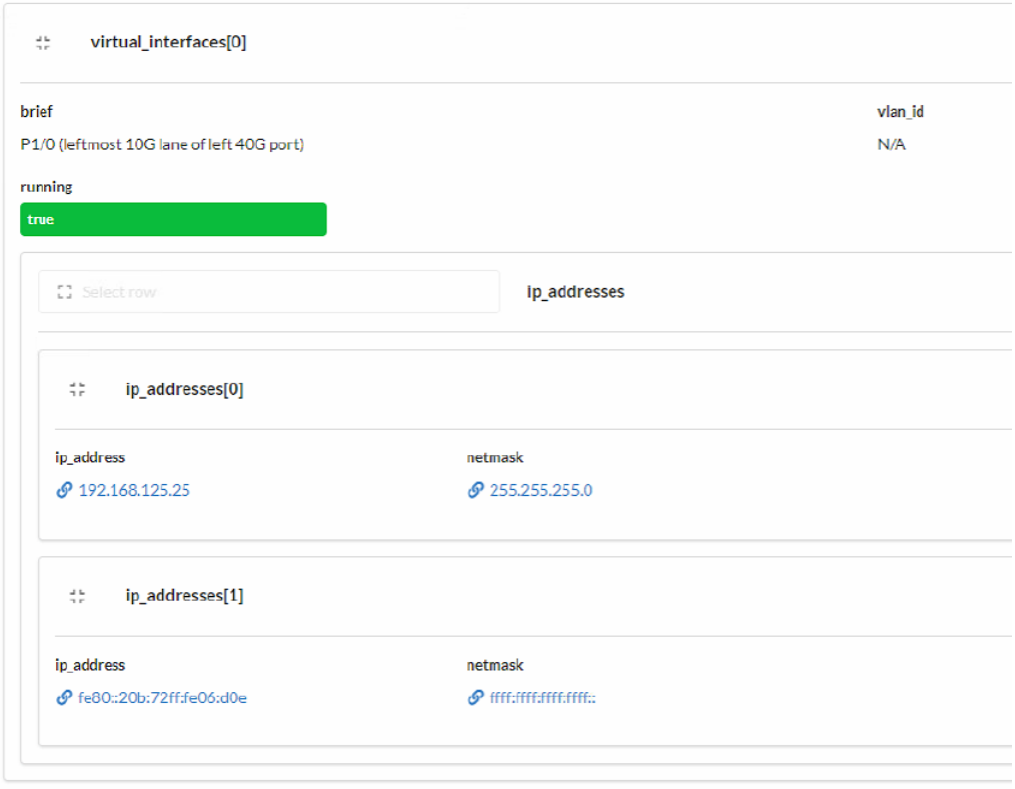
6.2.5 Select IP Address [0]

We are interested in **ip_addresses[0]** but feel free to open both.



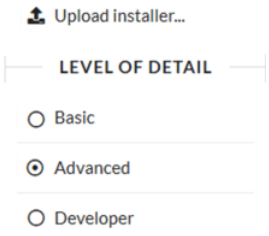
6.2.6 IP Address Details

The page updates to show the current IP address and netmask. In **Basic** mode, the settings can be viewed but not edited.



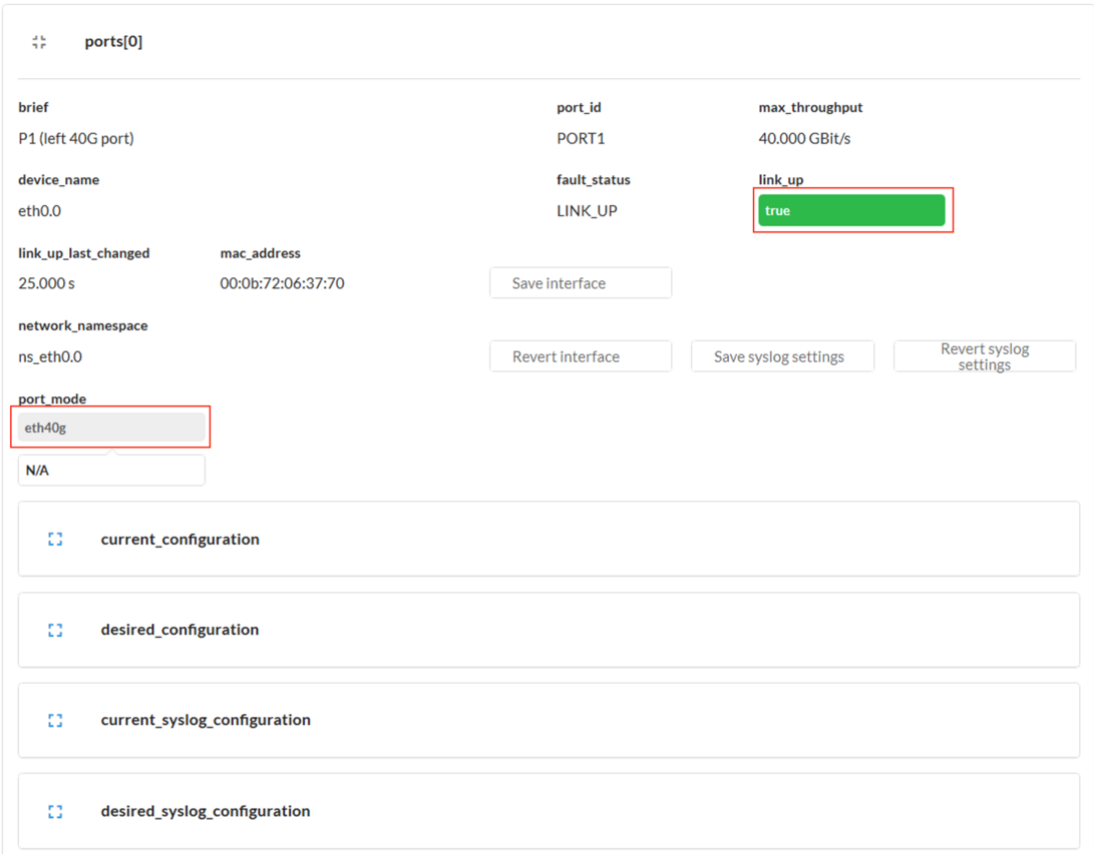
6.2.7 Change to Advanced Mode

To change the IP address, you must switch to **Advanced** mode (via the LEVEL OF DETAIL options at the top of the page).



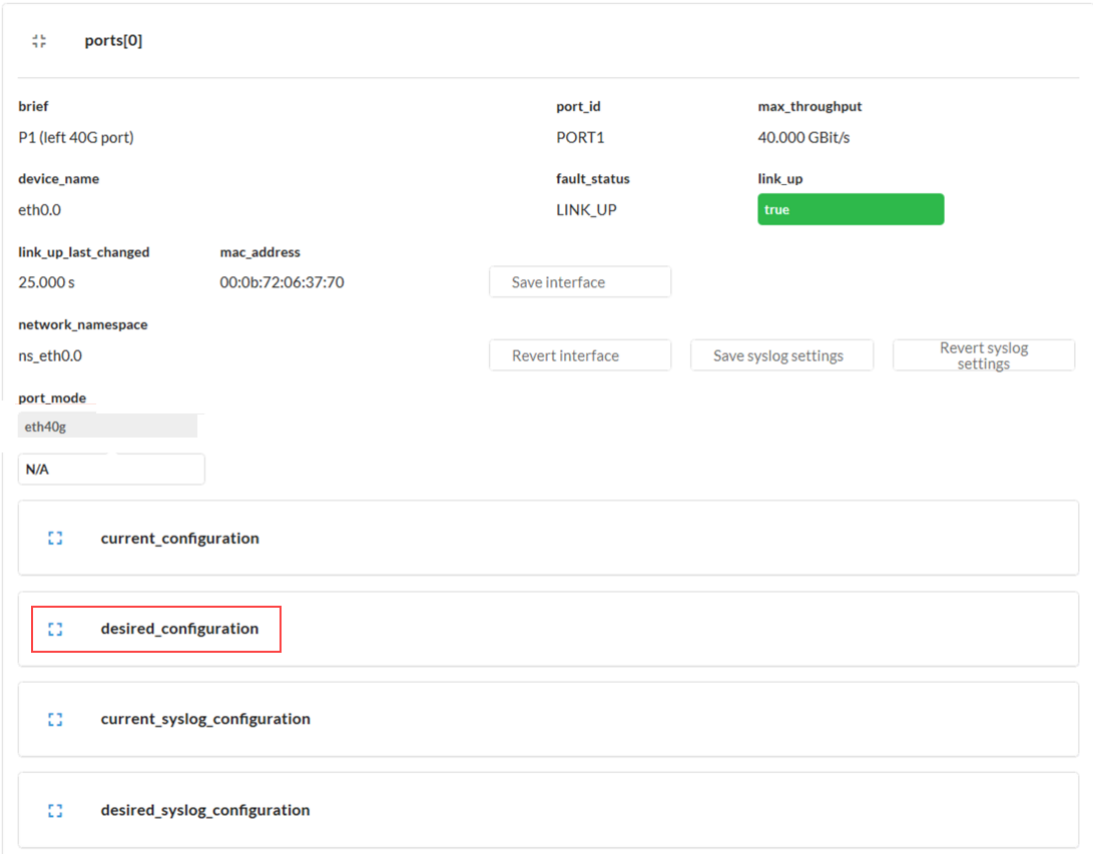
6.2.8 Port Mode

Several additional options are made visible including the port mode (which shows the connection speed) and link state (true or false).

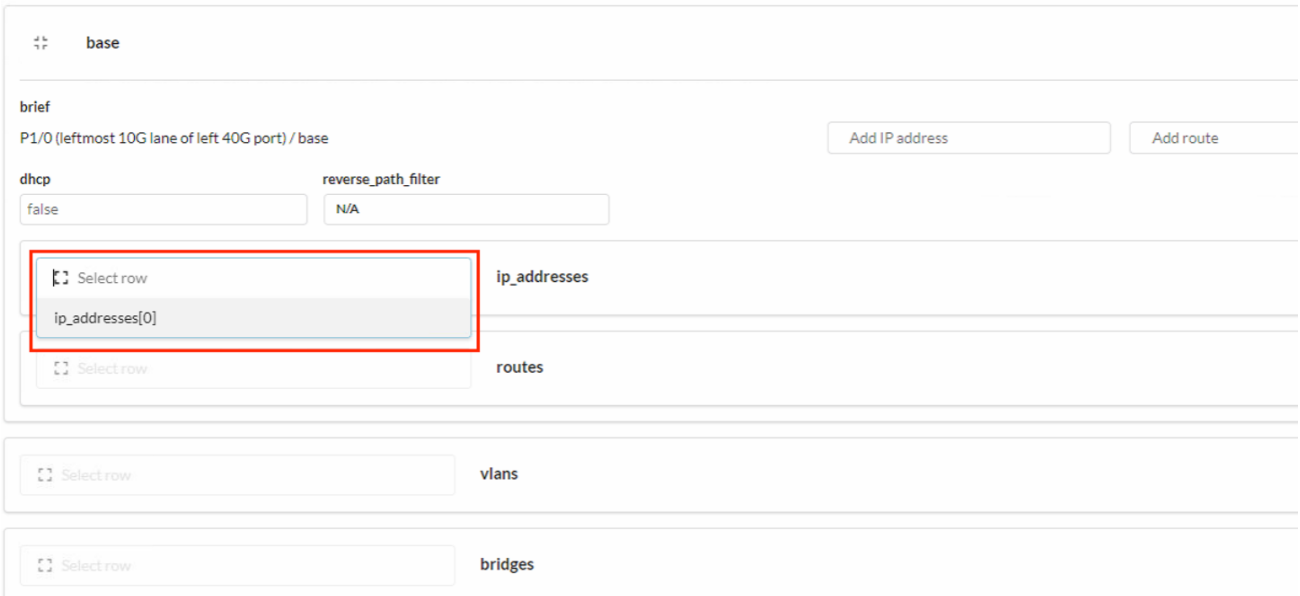


6.2.9 Desired Configuration

To edit the IP address of the port, open the **desired_configuration** menu:



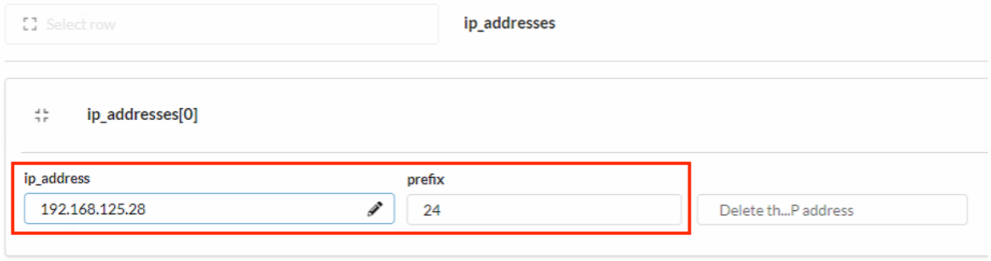
Then open the **base** menu and select **ip_addresses[0]**. If no address exists, use *Add IP address* to add one first.



6.2.10 Set IP Address

Type in the desired IP address and press Enter to apply. The pencil icon identifies the field as edited but not applied.

The **prefix** value is used to set the subnet mask. For example, a /24 network stands for 255.255.255.0



6.2.11 Routes (Gateway)

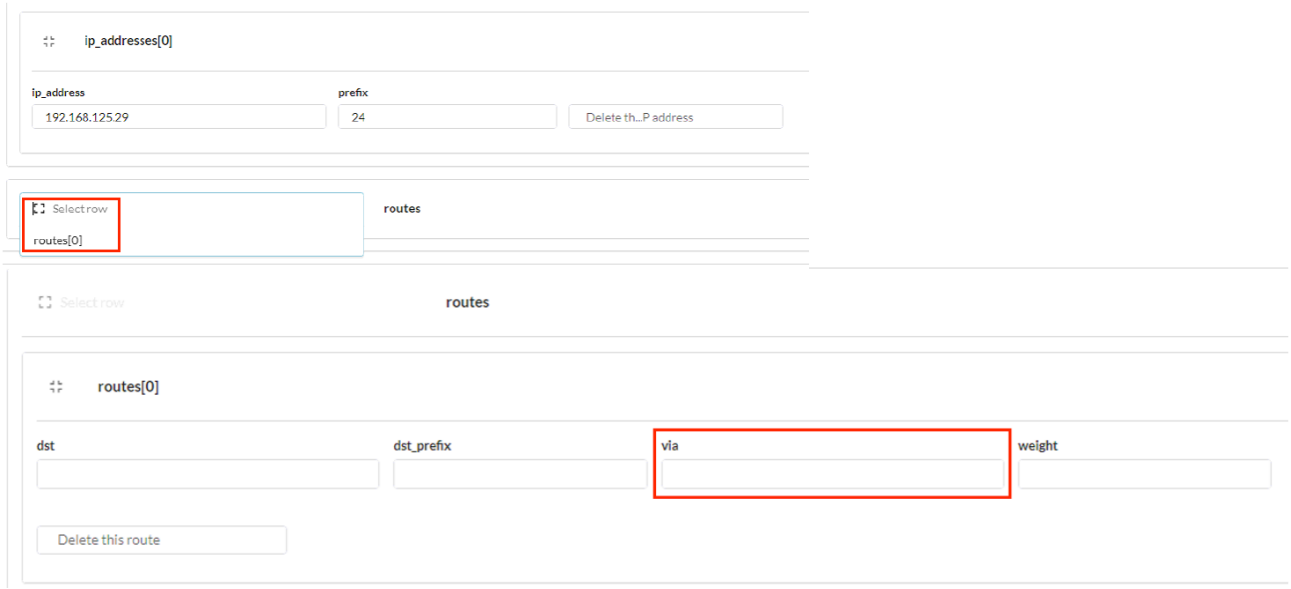
If you need a gateway, then this can be added using the **routes** function.

1. Select **routes[0]** from the drop-down menu.

If no route exists, use *Add route* to add one first.

2. Use the **via** field to specify the gateway.

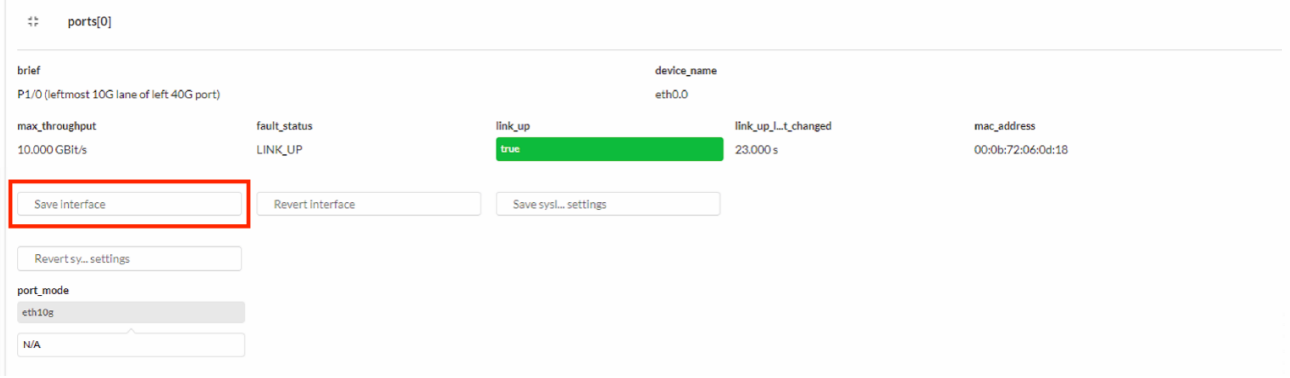
Remember to press Enter to apply the changes.



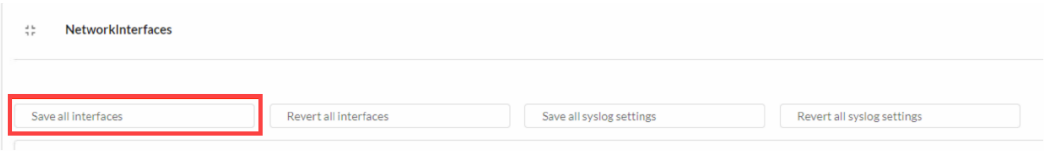
6.2.12 Save all Interfaces

Any changes must be saved before you reboot the device.

This can be done for the current interface using the *Save interface* button.



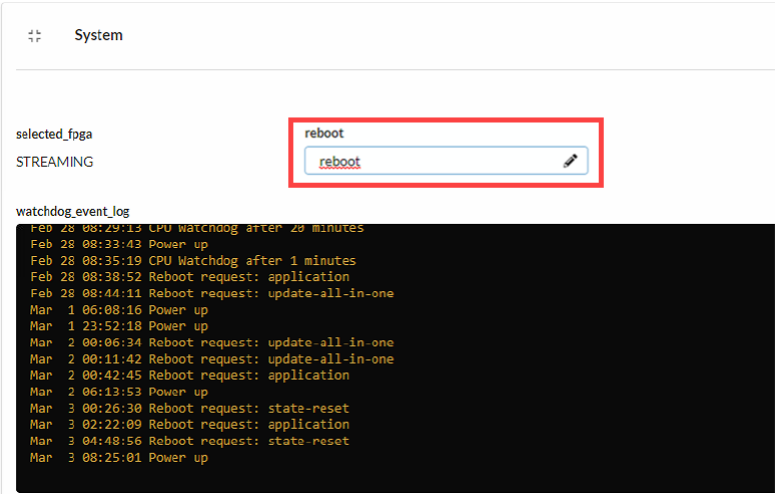
Or, for all interfaces using the *Save all interfaces* button (in the main **NetworkInterfaces**).



6.2.13 Reboot

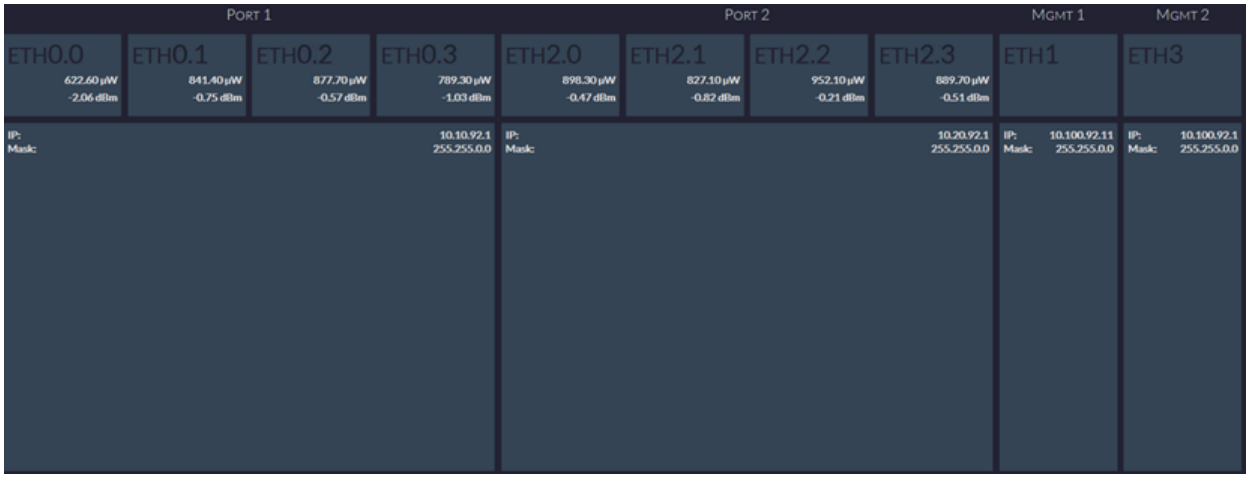
All saved changes are applied after the next reboot.

Open the System module, type **reboot** into the reboot field and press Enter.



6.2.14 Monitor the IP Configuration

Once boot-up is complete, open the main web UI "IP Address" page to check that the changes have been applied.



Then repeat the steps to configure each of the other network interfaces as required.

6.3 Fault Finding Tools

The advanced page provides two useful tools for fault-finding: multiedit and watchpoints.

6.3.1 Multiedit

The multiedit feature can be used to change or get status information for multiple parameters at once. For example, to quickly turn off all TX or RX of a particular type.

The tool works by entering a text-based command. For example:

- **video_transmitter.pool[:].active=true** - will turn on all the video transmitters.
- **video_transmitter.pool[:].active=false** - will turn off all the video transmitters.

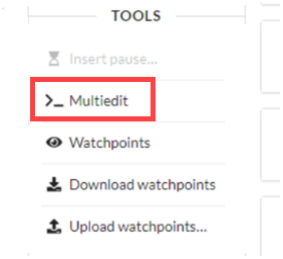
Notice that the command is split into sections which describe the parameter type, its options and desired value.

Parameter Type	Options	Value
video_transmitter	pool[:]	active=true
video_transmitter	pool[:]	active=false

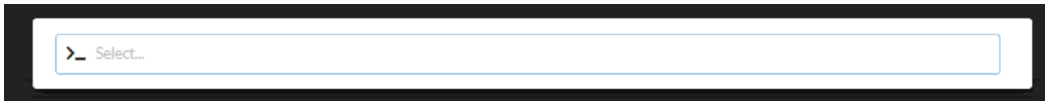
The instructions which follow describe how to use the system's auto-complete feature to find the correct syntax for our example.

Once you understand the principles, you can apply these to other parameters. A few of the most common parameter types are **r_t_p_receiver**, **video_transmitter** and **audio_transmitter**.

1. To launch the tool, click on **>_Mutliedit** from the TOOLS menu, or press CTRL + ENTER on your computer keyboard.

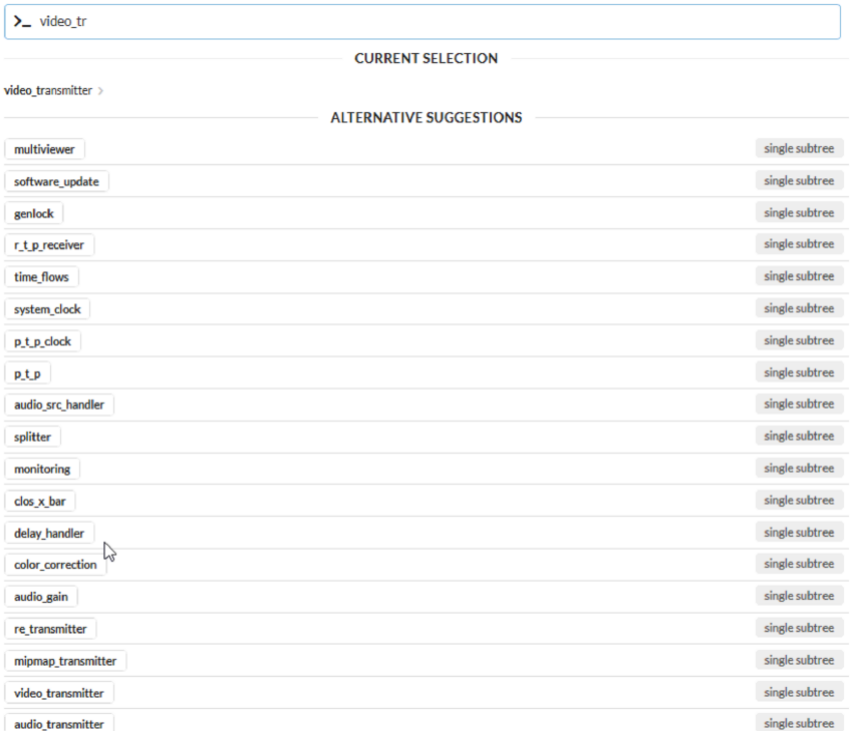


A bar appears where you can enter the text-based command.

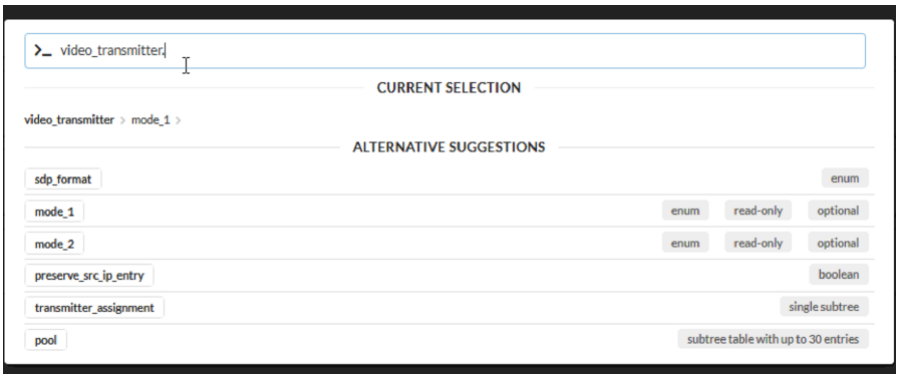


2. Type in the first part of the parameter type - for example, **video_tr**.

As soon as you start typing, the system offers some suggestions, so press TAB to auto-complete the selection. For example, type in **video_tr** and press TAB to enter **video_transmitter**.

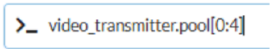


3. Press TAB again to view the available options.



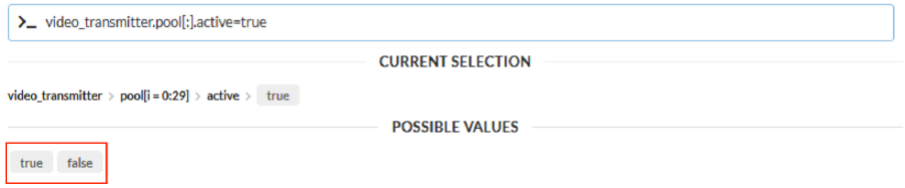
4. Type in **po** and press TAB to access the video transmitter's "pool" list.

By default, all pools are pre-selected but you can set a range by adding a suffix. For example, if you enter **video_transmitter.pool[0:4]**, then only the first five pools are affected.

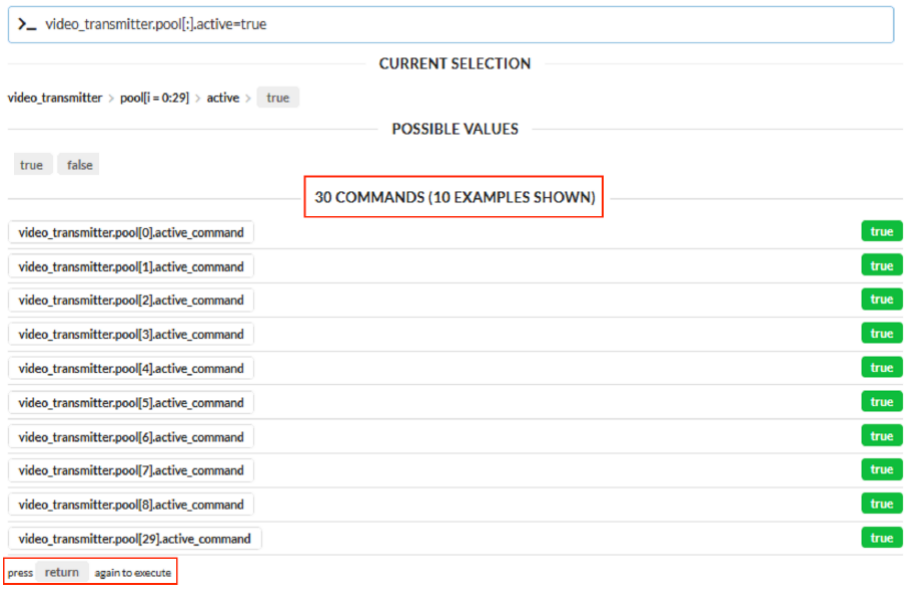


5. Press TAB again to access the next level of options. We are interested in the **active** option.

If you press TAB again at this stage, then no further options are displayed as the system is expecting a value. The correct syntax is displayed in the POSSIBLE VALUES area - in our example, either **true** or **false**.



6. To activate all of the selected video transmitters, type in **active=true** and press Enter or TAB - the current status of the requested values is shown.



Note that the system identifies that 30 commands will be executed (for pools 0 to 29), even though only the first 10 examples are shown.

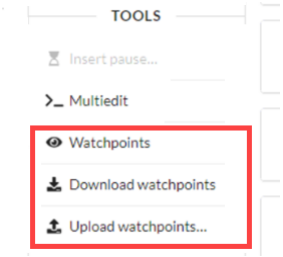
7. Press Enter again to execute all of the commands - the window closes and the changes are applied.

8. To toggle to the alternate state (all video TX off), repeat the steps but this time set the final option to **active=false**.

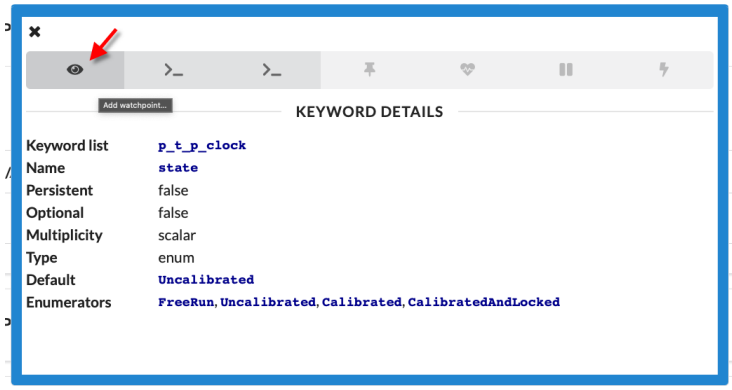
Tip: You can use the UP / DOWN arrows on your computer keyboard to recall the most recently used commands. This makes it very quick to toggle between the video TX on and off states in our example.

6.3.2 Watchpoints

Watchpoints allow you to monitor parameter changes in the C100 advanced WebUI. They can be accessed from the TOOLS section in the sidebar:



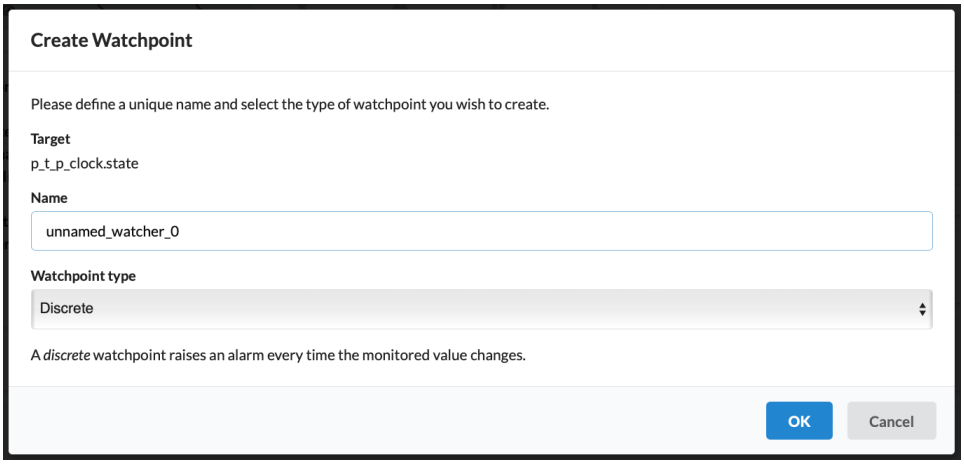
To create a watchpoint, navigate to the desired parameter in the advanced WebUI, right-click on the parameter and select the eye symbol (in the top left corner of the pop-up window):



A watchpoint setup window opens. Here you can see the chosen target parameter, set a unique name for your watchpoint and define the watchpoint type. You can choose between two watchpoint types: discrete and custom.

Discrete Watchpoint

A discrete watchpoint raises an alarm every time the monitored value changes.

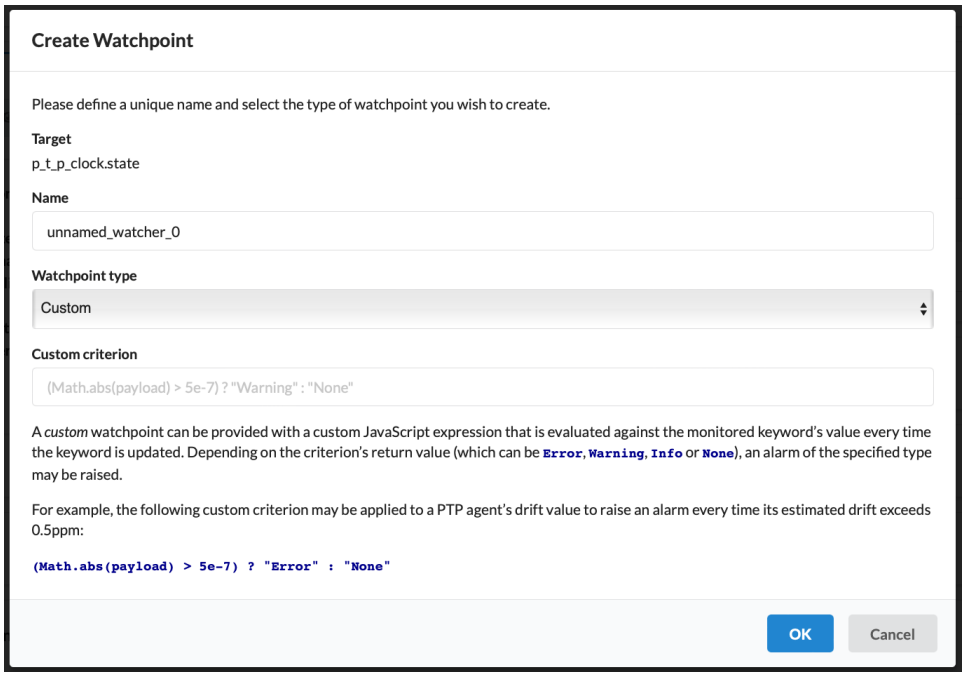


Custom Watchpoint

A custom watchpoint can be provided with a custom JavaScript expression that is evaluated against the monitored keyword's value every time the keyword is updated. Depending on the criterion's return value (which can be "Error", "Warning", "Info" or "None"), an alarm of the specified type may be raised.

For example, the following custom criterion may be applied to a PTP agent's drift value to raise an alarm every time its estimated drift exceeds 0.5ppm:

```
(Math.abs(payload) > 5e-7) ? "Error" : "None"
```



Managing Watchpoints

It is possible to set up several watchpoints. They can be downloaded and saved to your computer and re-uploaded at a later point in time or to a different C100 module via the watchpoints sidebar section.

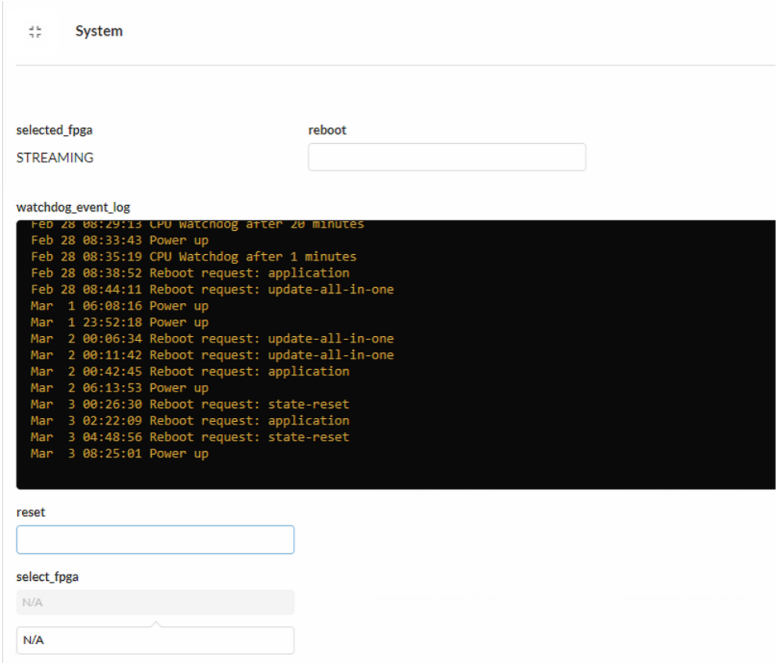
7. System Tools

The following useful tools can be accessed via the advanced page.

- **System** module:
  - **FPGA Mode** - selects the FPGA mode.
  - **Reboot** - reboots the device.
  - **Reset** - resets all of the C100 settings, except the IP addresses.
  - **Power Cycle** - power cycles the device (available in Advanced mode only).
- **Services** module:
  - **Avahi** / **SNMP** - activates or deactivates the avahi and SNMP services.

7.1 System Module Tools

1. Start by opening the advanced page in the usual manner.
2. Open **System** by clicking on its shortcut in the MODULES list, and then clicking on its icon.



In **Basic** mode, you can view and change the FPGA mode, reboot the device or perform a reset.

In **Advanced** mode, you can also power cycle the device.

7.1.1 Changing the FPGA Mode

The **selected_fpga** field shows the current FPGA mode.

To change the mode:

1. Click on the **select_fpga** field and choose an option from the drop-down menu.

The FPGA modes correspond to the Virtual Modules described earlier.

For AVP (audio video processor), you must take care to select the correct connection speed:

- **AVP** - for 4x 10GbE.
- **AVP_40GbE** - for 1x 40GbE.

For the multiviewer and UDX options, the connection type is fixed (at 1x 40GbE).

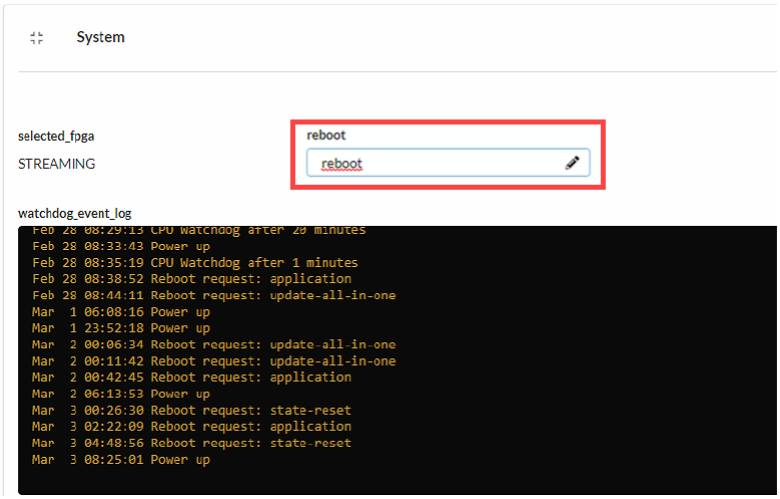
The **N/A** option should be selected if you have been playing with this menu and wish to leave the current FPGA mode unchanged.

2. The selected FPGA will be loaded following a reboot of the device.

7.1.2 Reboot

A reboot will restart the C100 blade. If a new FPGA mode is selected, then this will load after the reboot.

To perform a reboot, type **reboot** into the reboot field and press Enter.

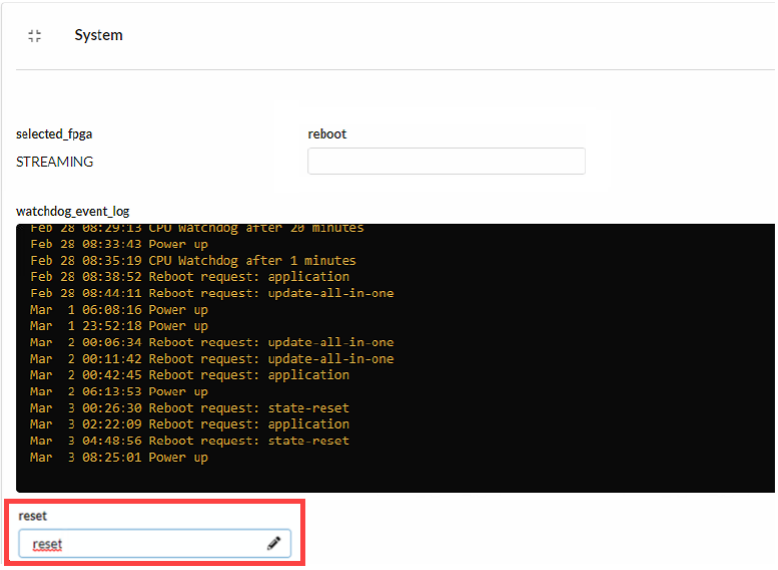


The progress can be monitored in the **watchdog_event_log**.

7.1.3 Reset

A reset will clear all of the C100 settings and then reboot the device. Note that all settings are cleared except the IP configuration. If a new FPGA mode is selected, then this will load after the reset.

To perform a reset, type **reset** into the reset field and press Enter.



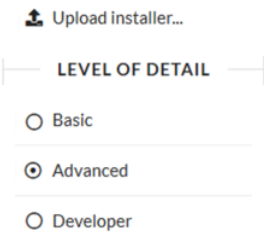
The progress can be monitored in the **watchdog_event_log**.

Attention: The **reset** function should be used with caution, as this resets all of the C100 settings except the IP configuration.

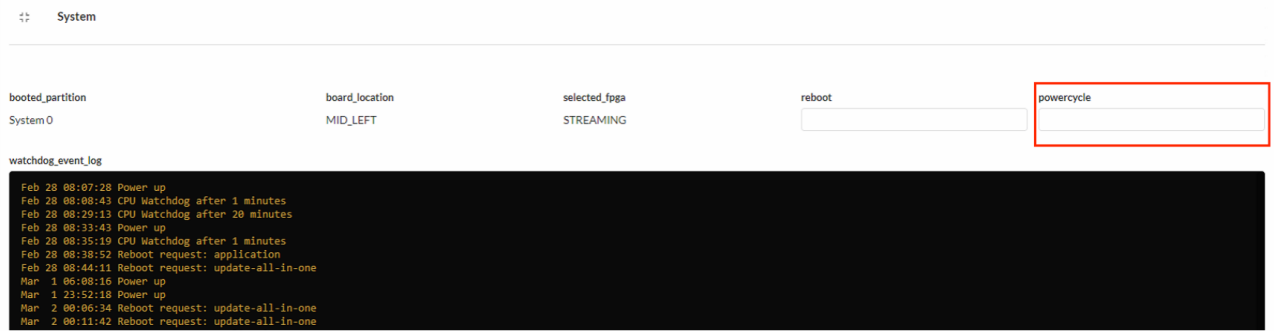
7.1.4 Power Cycle

A power cycle disconnects the power from the C100, and then boots up after a few seconds. If a new FPGA mode is selected, then this will load after the power cycle.

To access this function you must switch to **Advanced** mode (via the LEVEL OF DETAIL options at the top of the page).



Then type **powercycle** into the powercycle field and press Enter to execute.



The progress can be monitored in the **watchdog_event_log**.

7.2 Network Service Tools

From the **Services** module you can activate or deactivate the avahi and SNMP services.

1. Open **services** by clicking on its shortcut in the MODULES list, and then clicking on its icon.

The current status is shown immediately below the two headers. This can be either true (if the service is active) or false (if the service is disabled).

2. Click on the entry field to reveal a drop down menu, then select either **true** or **false** to turn the service on or off.

> Avahi is a data network service (similar to Bonjour) that allows devices to publish and discover nodes running on a Local Area Network. Avahi is an example of a zeroconf networking implementation. Other zeroconf systems include Bonjour (licensed by Apple).

8. PTP Setup

Precision Time Protocol (PTP) is a way of synchronizing clocks within a computer network. Correctly implemented it can achieve a clock accuracy in the sub-microsecond range and is suitable to synchronize media streams. Version 2 is applicable today and has been standardized as IEEE1588-2008. More information about PTP can be found in the Lawo IP Networking Guide.

The C100 can lock to PTP arriving from the network.

A suitable PTP Grandmaster must be installed and connected to the media network. When using redundant networks, it is highly recommended to install two separate PTP GMs.

8.1 Configuring C100 as a PTP Slave

To lock to PTP arriving from the network, the C100 should be configured as a PTP slave.

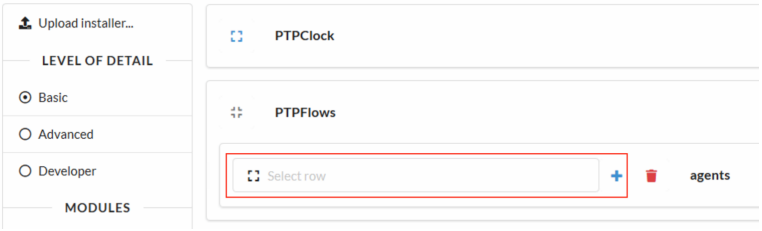
There are two parts to configure. Firstly, a PTP agent must be defined to listen for the incoming PTP signal. Secondly, the PTP clock source must be set to the correct input.

8.1.1 Create PTP Agent

1. Start by opening the advanced page in the usual manner.
2. Open **PTP Flows** by clicking on its shortcut in the MODULES list, and then clicking on its icon.

Take care not to confuse this with the **PTP Clock** module which will be used later to set the clock source.

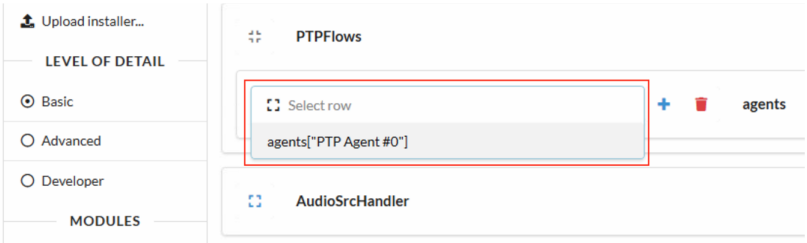
3. By default, there are no PTP agents and so one must be created by clicking on the **+** button.



The PTP agent is used to connect to the PTP Grandmaster and subscribe to its time information.

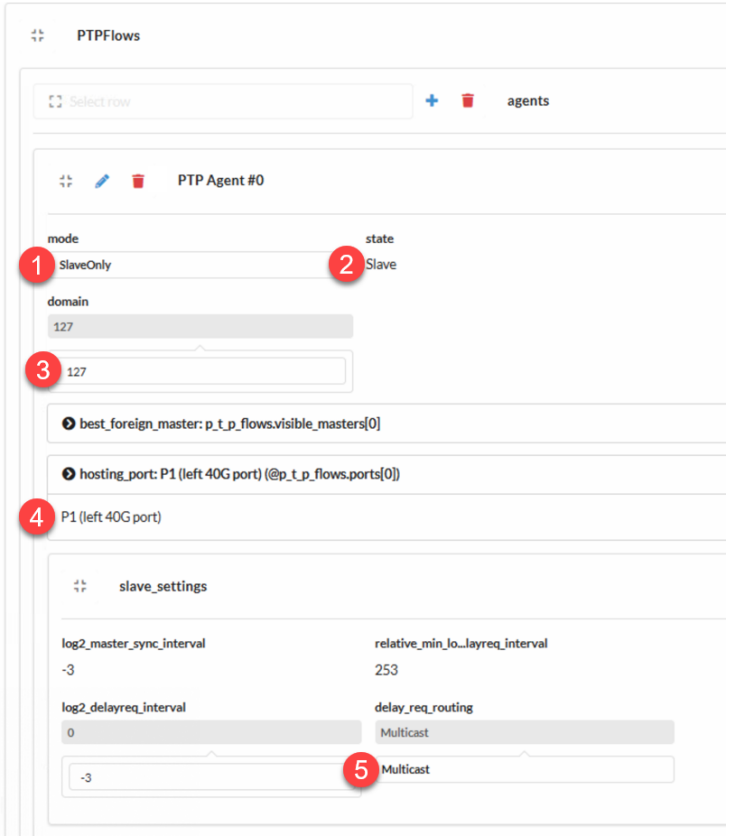
When using redundant networks, you should create at least two agents, one for the red network and one for the blue network.

4. Once an agent exists, it can be selected from the drop-down menu.



8.1.2 Configure PTP Agent

Configure the selected PTP agent as follows.



1 mode

To listen for PTP arriving from the network, set the mode to **SlaveOnly**. This ensures that the C100 will not run as a PTP master.

2 state

This field shows the current PTP state. It should read **Slave** if the C100 is operating in Slave Only mode.

3 domain

This field defines the PTP domain. The value will vary depending on the PTP profile in use. Please refer to the Lawo IP Networking Guide for more details.

4 hosting_port

This field should be assigned to the first port connected to the media network.

When using redundant networks, you should assign the first port connected to the red network for PTP Agent #0, and the first port connected to the blue network for PTP Agent #1.

5 delay_req_routing

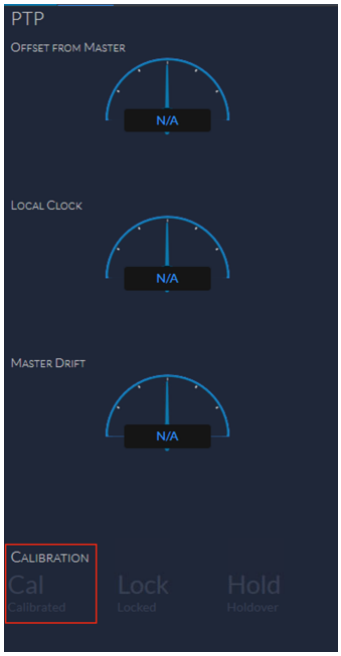
This field can be set to either **Unicast** or **Multicast**. It is recommended to use **Unicast** providing this works for your setup.

Whether unicast will work or not, depends on the number of slaves you are trying to accommodate. Some networks are impermeable to unicast delay responses, and so in this case you must use **Multicast**.

8.1.3 Monitor PTP Status

Open the main web UI "Status" page so that you can check the PTP calibration during the next stage.

Notice that the PTP is not yet calibrated as, so far, only the listening agents have been configured and not the PTP clock source.

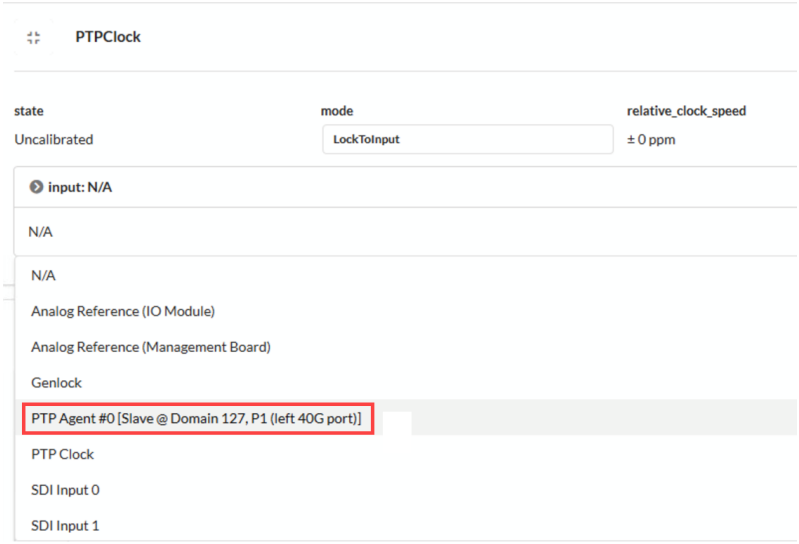


8.1.4 Define PTP Clock

The next step is to define the PTP clock source.

1. Open **PTP Clock** by clicking on its shortcut in the MODULES list, and then clicking on its icon.
2. Use the **> input:** menu to choose a sync input.

For our example, choose **PTP Agent #0** to select the listening agent defined earlier.



The **> input:** menu shows all of the possible sync sources available. In our example, these are:

- **Analog Reference (I/O Module)** - Tri-level or Video Black Burst connected to the C100's rear I/O module
- **Analog Reference (Management Board)** - Tri-level or Video Black Burst connected to the central MGMT module.
- **Genlock** - internal clock generated by the C100 (see note below).
- **PTP Agents** - the PTP listening agents defined earlier.
- **PTP Clock** - internal clock generated by the C100 (see note below).
- **SDI Input 0**, **1** - SDI connected to the C100's rear I/O module.

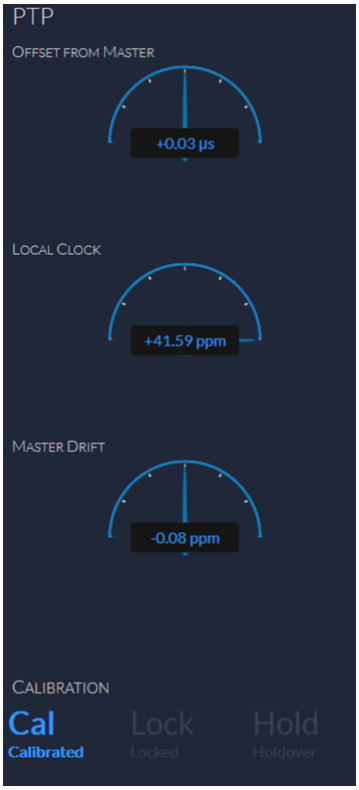
If a TimeFlows Combinator has been created to provide clock redundancy, then this can also be selected.

> It is not recommended to use the C100 internal clock as the sync input, and so both the **Genlock** and **PTPClock** options should be avoided.
> If there is no valid PTP on the network, then it is best to use an **Analog Reference**. This can be configured as a backup source (as described later).

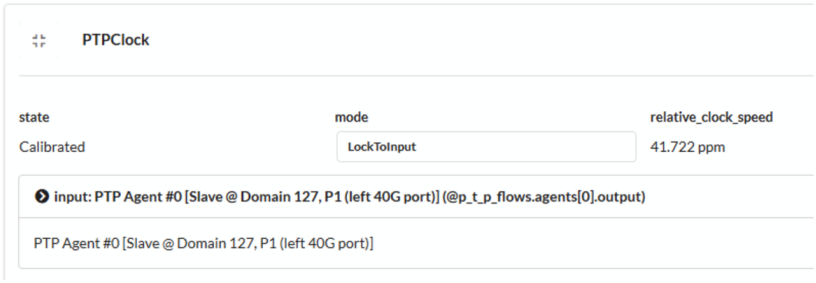
8.1.5 Calibration

Once the PTP clock source is selected, the calibration process begins.

On the main "Status" page, the three gauges jump around until the clock stabilizes. Once calibration is complete, the status changes from **Uncalibrated** to **Calibrated** and the gauges should only move slightly.



The **state** field in the Advanced page also updates.



For a non-redundant setup, the C100 configuration is now complete.

8.2 Configuring Redundant PTP

If you are using red and blue media networks or wish to add a backup sync source, then you will need to use a **TimeFlows Combinator**.

A TimeFlows Combinator selects the best PTP source available from several inputs. For example, to switch from the red network PTP Grandmaster to the blue network PTP Grandmaster should the red network fail. Or, to switch to a backup source, such as Video Black Burst, if there is no valid PTP on the network.

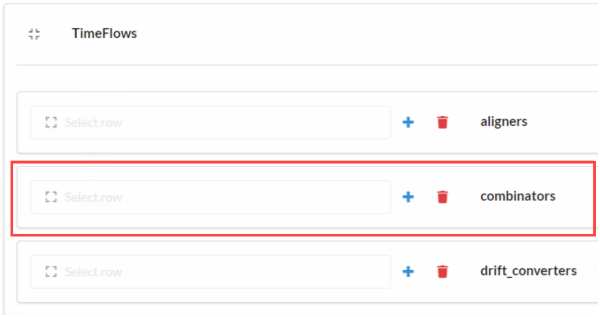
To use this feature, you will need to define a combinator with the required number of inputs in order of priority. For example:

- Set **Input 0** to **PTP Agent #0** to listen for PTP on the red media network.
- Set **Input 1** to **PTP Agent #1** to listen for PTP on the blue media network.
- Set **Input 2** to an **Analog Reference** to lock to Tri-level or Video Black Burst in the event that there is no valid PTP on either network.

Once the combinator is configured, it can then be selected as the clock source in the PTP Clock module.

8.2.1 Create TimeFlows Combinator

1. Start by opening the advanced page in the usual manner.
2. Open **TimeFlows** by clicking on its shortcut in the MODULES list, and then clicking on its icon.
3. Create a new combinator by clicking on the **+** button.



4. Once a combinator exists, it can be selected from the **combinators** menu.

8.2.2 Configure TimeFlows Combinator

1. Configure the combinator settings as follows.

1 required_type

By default, this is set to **Absolute**.

2 required_source_type

The preferred option is **UseGPSOrBetter**, but this will only work with a GPS-based Grandmaster clock true to the SPG.

Our example uses a V__Remote which is not GPS clocked. You should keep this in mind when selecting your clock class.

3 drift_reference_frame

Set to **PTP**.

4 min_drift_tolerance

Set to **4e-7**.

5 min_offset_tolerance

Set to **1.0 µs**.

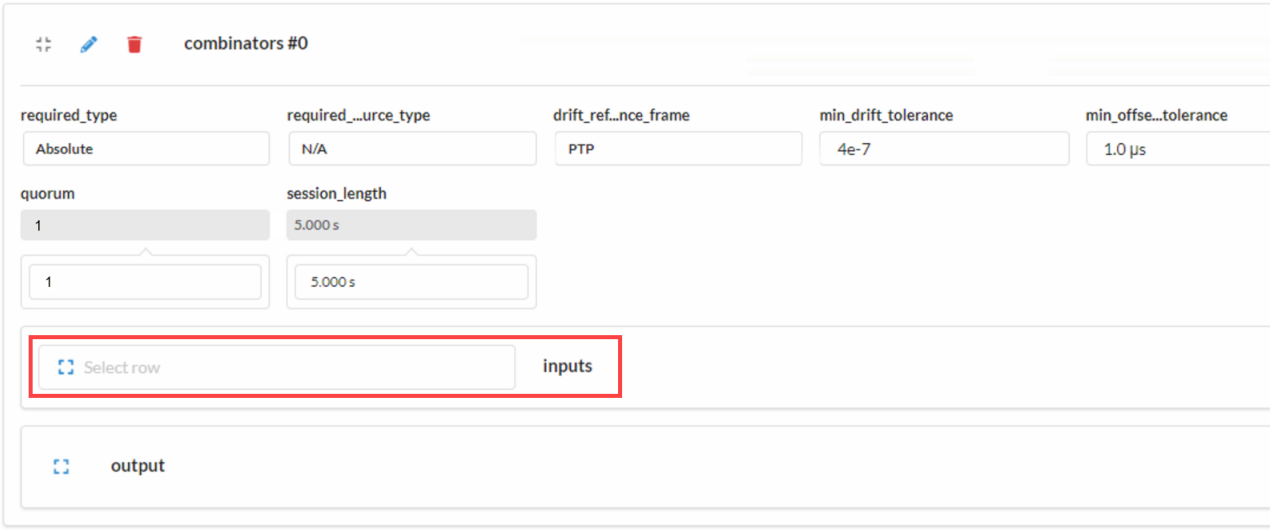
6 quorum

Set to **1**.

7 session_length

Set to **5.000s**.

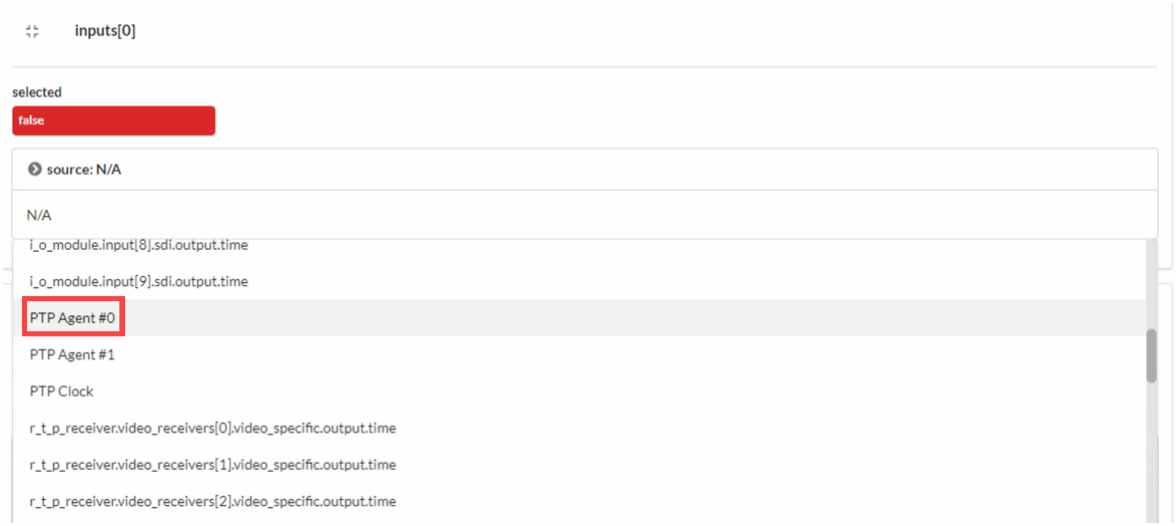
2. Open the **inputs** menu to assign each of the inputs.



Note that the **output** menu is for monitoring purposes only.

3. Start with the first input - **inputs[0]** - and make a selection from the **source:** menu.

For our example, choose **PTP Agent #0** to assign the PTP listening agent for the red network.



4. Repeat for each of the other inputs you wish to configure.

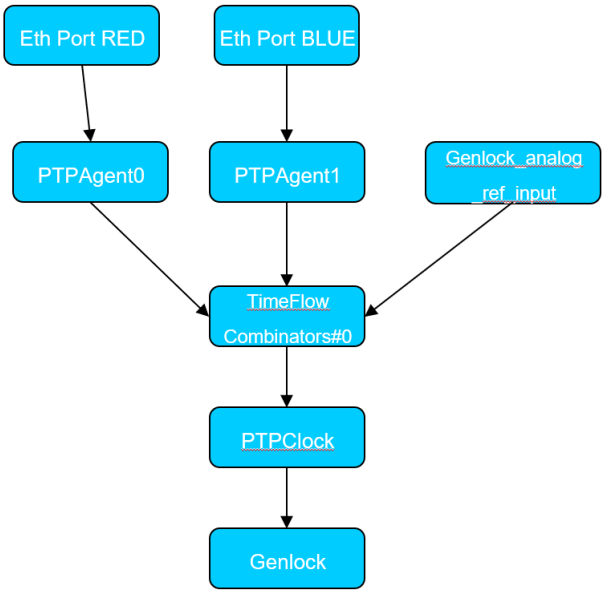
For example, select **inputs[1]** and assign **PTP Agent #1** to assign the PTP listening agent for the blue network.

Then select **inputs[2]** and assign one of the **Analog Ref** options to assign a local source such as Tri-level or Video Black Burst.

Once all of the required inputs are assigned, the first part of the configuration is complete. You can now assign the combinator as the clock source in the PTP Clock module..

8.3 PTP Flow

The diagram below illustrates the flow of the PTP clocking signal.



Whenever you wish to use the C100's PTP clock for transmitters, receivers or any other function that has a sync option, it is advised to choose **Genlock**, instead of **PTPClock**, in the drop-down menus.

This Genlock is the internal genlock and not the "genlock_analog_ref" input from the backplane. It has the same value as the PTPClock option but should be selected instead of PTPClock wherever possible.

Whenever you wish to change the sync on the C100, change the PTP clock source (using the PTP Clock module), and all Genlock users will follow the new applied setting.

8.4 Locking Policy

When setting up the PTP via a script, you might see some extra options like the Locking Policy. This can be set to either dynamic or locking as follows.

Await write("p_t_p_clock.parameters", "locking_policy", "Locking");

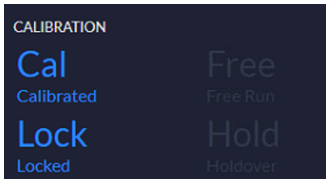
locking policy = dynamic for setting up / rehearsal events or similar

locking policy = locking for ... during event or similar... would run in holdover (lost ptp GM) for a couple of hours without any problem

As most setups have a mixture of devices, and not just C100s, it is better to use locking policy = dynamic. This means that if the Grandmaster is lost, the C100 will fail over to the next elected master.

If locking policy = locking is used, then the C100s will run in holdover and continue on the timing of the lost grandmaster for a couple of hours. Please note that this can create problems if other decices in your network synchronize to a different clock after grandmaster failure. This should be avoided by a proper PTP redundancy setup.

Open the main web UI "Status" page and look at the CALIBRATION area to check the C100 status: **Lock** (lit) = PTP is locked; **Lock** (unlit) = PTP is dynamic.



8.5 Best Practice PTP

Below is an example of the best practice PTP configuration for the most commonly used cases.

Keep in mind that the **UseGPSOrBetter** option will only work when the Grandmaster is locked to a GPS signal. For example, it would not work if the Grandmaster is a V__Remote.

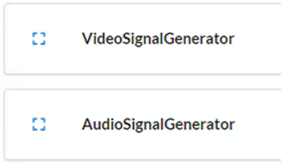
```
await create_table_row("p_t_p_flows.agents");
await create_table_row("p_t_p_flows.agents");
await write("p_t_p_flows.agents[0]", "hosting_port_command", "p_t_p_flows.ports[0]");
await write("p_t_p_flows.agents[1]", "hosting_port_command", "p_t_p_flows.ports[1]");
await write("p_t_p_flows.agents[0]", "domain_command", 127);
await write("p_t_p_flows.agents[1]", "domain_command", 127);
await write("p_t_p_flows.agents[0].slave_settings", "delay_req_routing_command", "Multicast");
await write("p_t_p_flows.agents[1].slave_settings", "delay_req_routing_command", "Multicast");
await create_table_row("time_flows.combinators");
await write("time_flows.combinators[0]", "required_ptp_source_type", "UseGPSOrBetter");
await write("time_flows.combinators[0].inputs[0]", "source_command", "p_t_p_flows.agents[0].output");
await write("time_flows.combinators[0].inputs[1]", "source_command", "p_t_p_flows.agents[1].output");
await write("time_flows.combinators[0]", "quorum_command", 1);
await write("p_t_p_clock", "input_command", "time_flows.combinators[0].output");
await write("system_clock", "input", "time_flows.combinators[0].output");
await write("system_clock", "time_standard_command", "UTC")
```

9. Signal Generator

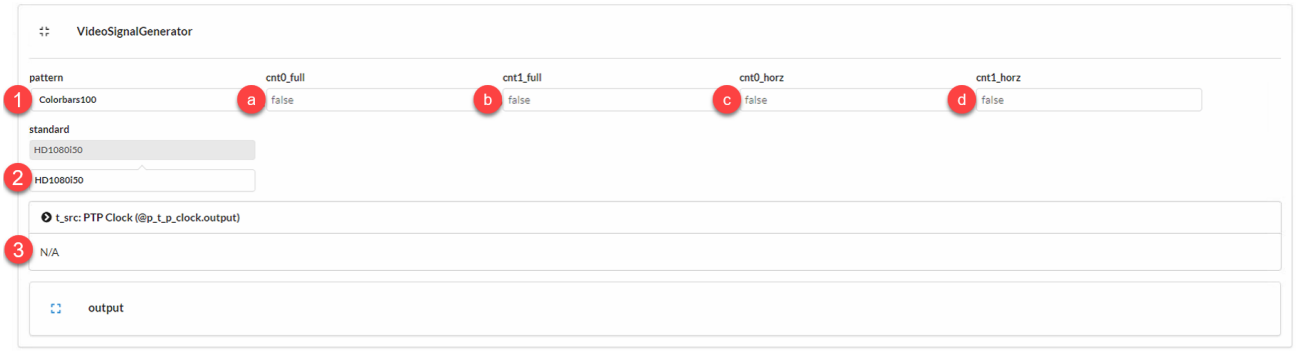
Each C100 includes a built-in signal generator with settings for both the video and audio signals. It can be used for signal path testing. For example, to send video and audio signal to an SDI output, a transmitter or a multiviewer PiP.

9.1 Configuring the Video Signal Generator

1. Start by opening the advanced page in the usual manner.
2. Open **Video Signal Generator** by clicking on its shortcut in the MODULES list, and then clicking on its icon.



3. Edit the settings according to your needs.



1 pattern

This field sets the test pattern. You can choose between **Colorbars**, a **RP198** signal or **Counters**. If the **Counters** option is selected, then the **cnt0_full**, **cnt1_full**, **cnt0_horz** and **cnt1_horz** fields apply.

2 standard

This field defines the video standard. A wide range of standards are supported including PAL, NTSC, HD720, HD1080, HD2160, etc. All possible options are shown in the drop-down menu.

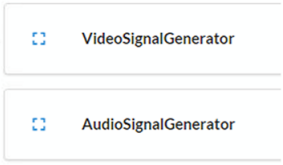
3 t_src

This field defines the timing source. By default, the PTP Clock is selected, but this can be changed if needed.

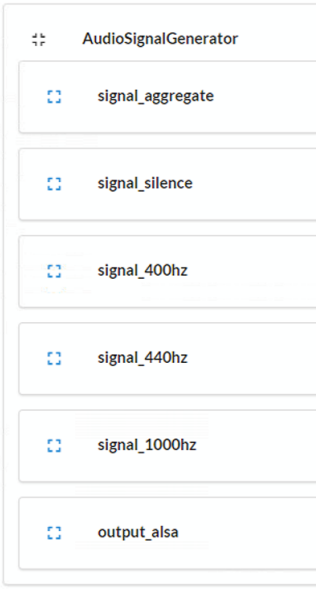
9.2 Configuring the Audio Signal Generator

The audio signal generator does not really require any configuration as, by default, it provides several different audio signals. The volume can be adjusted if you wish.

1. Open **Audio Signal Generator** by clicking on its shortcut in the MODULES list, and then clicking on its icon.



2. Open a sub menu to access the relevant settings.



The available audio signals are:

- Silence.
- 400Hz signal.
- 440Hz signal.
- 1KHz signal.
- ALSA (Advanced Linux Sound Architecture: the internal Linux soundcard audio).

Note that the ALSA option is currently not operational.

10. I/O Modules

Each V__matrix frame can be fitted with a variety of rear panel I/O modules.

Most do not need any configuration as the amount of inputs and outputs are pre-defined. Some, however, are configurable.

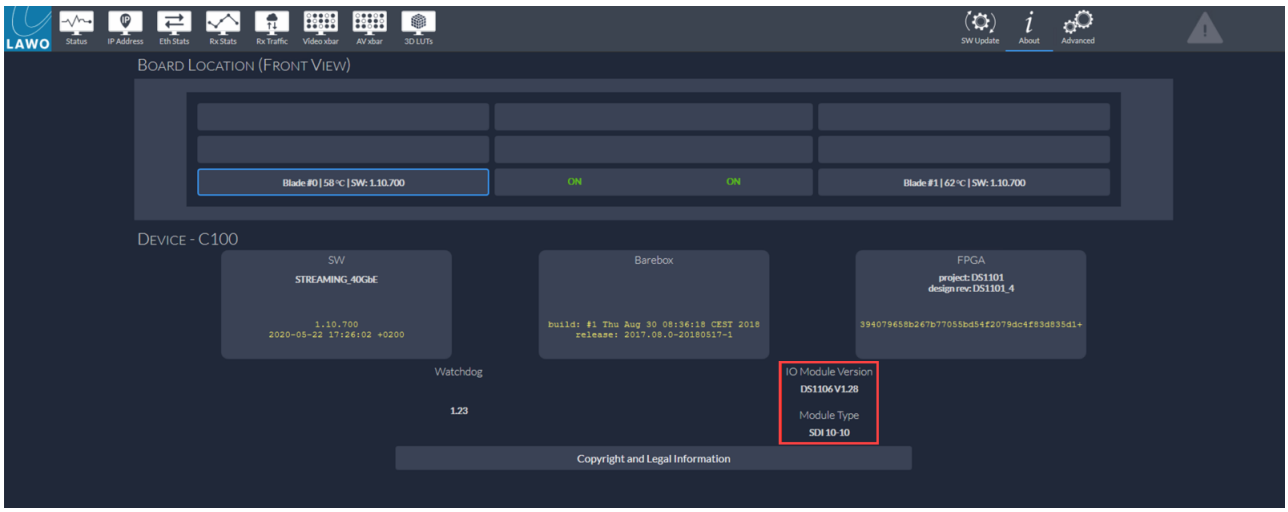
In all cases, you can use the main web UI "About" page and the advanced configuration "I/O Module" to provide status information.

10.1 Checking the Hardware Status

1. Open the "About" page, in the main web UI, to check the status of the C100's rear I/O module.

If the I/O module is correctly installed and working properly, then you see its type and version (highlighted in red below).

2. To check the status of all I/O modules in the frame, repeat this operation for each slot/C100.



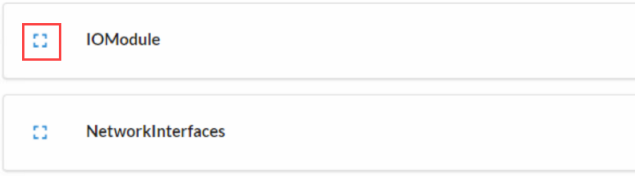
10.2 Configuring an I/O Module

Once you have confirmed that the I/O module is working properly, the Advanced page can be used to access its other settings.

The example which follows describes how to configure the 16 bidirectional I/Os on the SDI 2+2+16 I/O module. The same principles can be applied to other modules.

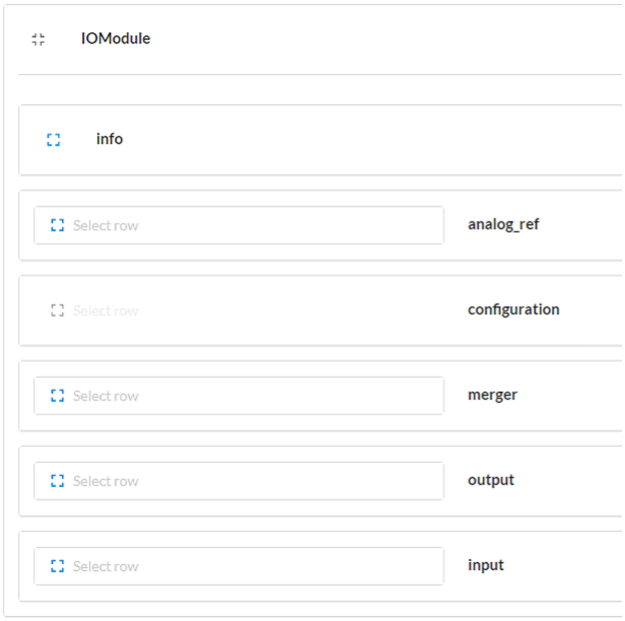
10.2.1 Open I/O Module

1. Start by opening the advanced page in the usual manner.
2. Open **I/O Module** by clicking on its shortcut in the MODULES list, and then clicking on its icon.



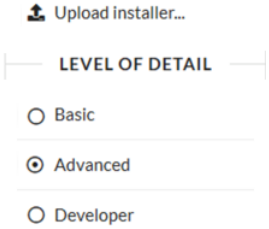
You will see six sub menus which are used as follows.

- **info** - shows information about the module.
- **analog_ref** - shows information about the video reference input.
- **configuration** - sets the direction to either IN or OUT, if the I/O is configurable.
- **merger** - can be used to convert 4K to 12G.
- **output** - shows information about the outputs and provides access to output options such as the connector type (BNC or MADI).
- **input** - shows information about the inputs.



10.2.2 Change to Advanced Mode

To open the **configuration** menu, you must switch to **Advanced** mode (via the LEVEL OF DETAIL options at the top of the page).

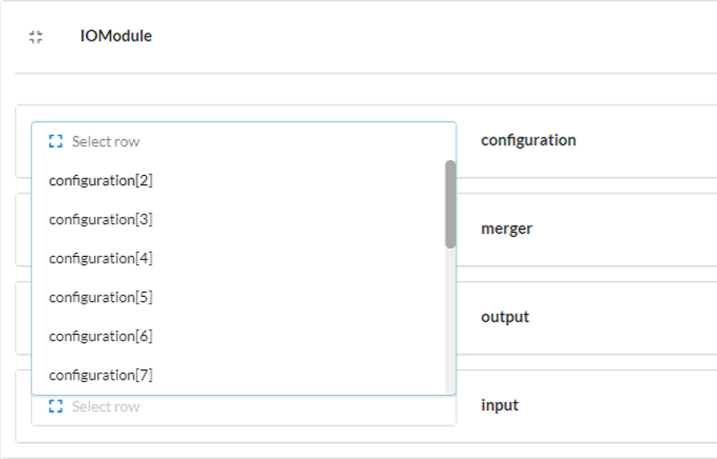


10.2.3 Define I/O Direction

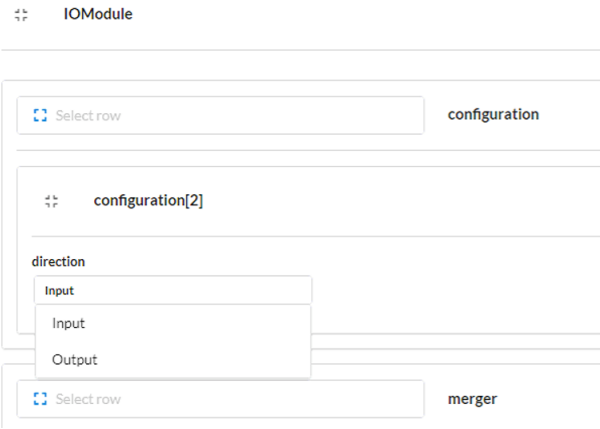
1. You can now select a configurable I/O from the drop-down menu - e.g. **configuration[2]**.

Notice that the available numbers start only from 2, as 0 and 1 are used for the first fixed connectors.

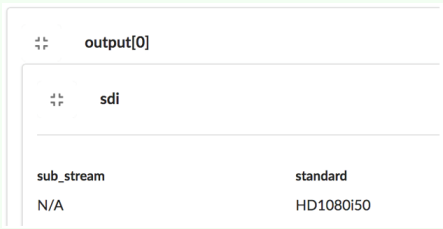
This means that, in our SDI 2+2+16 example, the first configurable I/O port is **configuration[2]** and the last configurable I/O port is **configuration[17]**.



2. Use the drop-down menu to set the direction to either **Input** or **Output**.



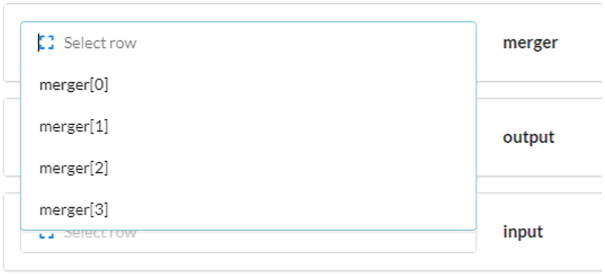
The detected video standard can be checked by selecting the corresponding input or output (from the **input** or **output** menu).



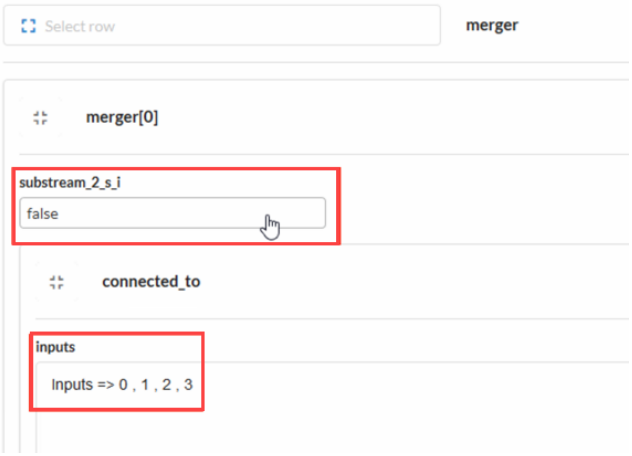
10.2.4 Converting Quad-link 2SI to 12G

A merger can be used to convert a quad-link 2SI signal to 12G single-link. It is not possible to use the 1080p quadrant signals in parallel as sources within the C100.

1. Start by selecting a merger from the drop-down menu - e.g. **merger[0]**.



2. Set the **substream 2_s_i** value to **true** (to activate the conversion).
3. Then connect your 2SI signals to the I/O module inputs that are listed in the "connected_to" section.

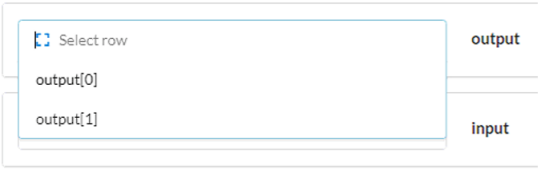


10.2.5 Set Output Mode

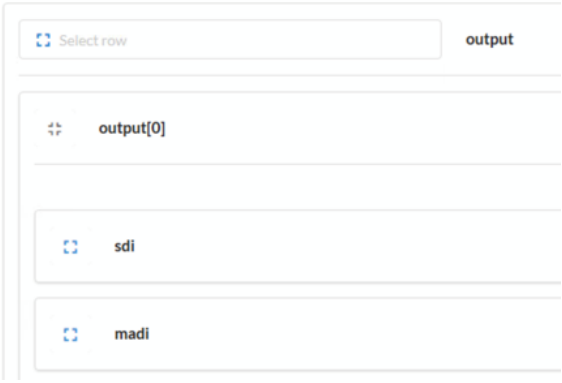
From the **output** menu, you can set each output to either SDI or MADI depending on the source.

1. Start by selecting an output - e.g. **output[0]**.

Note that the number of outputs shown depends on the I/O module and its configuration. In our example, **output[0]** and **output[1]** are the first two fixed outputs. If more outputs are defined using the configurable I/O, then the list will be longer.



2. Use the **sdi** or **madi** menus to set the connector type.



Under **sdi**, there are four additional sub menus which can be used as follows.

- **constraint** - apply a constraint to the SDI output. For example, to prevent the selection of an incompatible video signal.
- **vid_src** - assign a video source to the SDI output.
- **vanc_control** - bypass the vertical ancillary (VANC) data received from the source signal.
- **audio_control** - define the audio part of the SDI output.

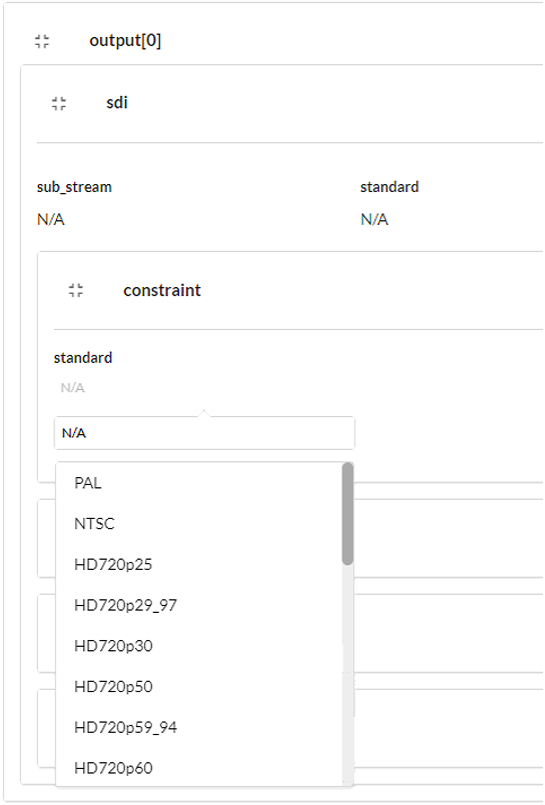
The next few pages describe each menu in more detail.

10.2.6 SDI Output Constraints

By assigning a constraint, you can prevent the sending of an incompatible video signal to the BNC connector.

1. Use the **standard** drop-down menu to make your selection.

If you choose **N/A**, then there is no limitation and all possible video source signals are permitted.

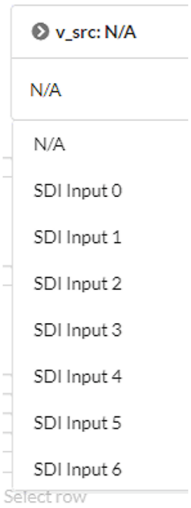


10.2.7 Assign Video Source

The **vid_src** menu can be used to connect an internal signal to the selected output.

Possible sources include SDI inputs, the Video Signal Generator or a Signal Processor (e.g. delay, color corrector, etc). This feature allows you to route a source to a SDI output without the need for a Video Crossbar.

1. Use the **> v_src:** menu to make a selection:



2. The **> t_src:** field defines the timing source.

By default, the PTP Clock is selected. This should be left unchanged unless otherwise instructed.

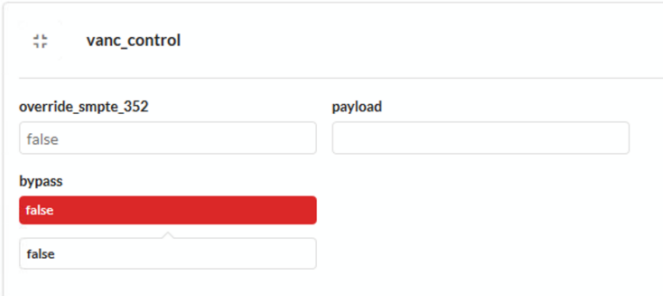
10.2.8 SDI Output VANC Control

The **vanc_control** menu can be used to bypass the vertical ancillary (VANC) data received from the source signal.

VANC data includes Closed Captions, Timecode, Active Format Description (AFD), Teletext and Dolby Metadata.

Ancillary data can be located in non-picture portions of horizontal scan lines. This is known as horizontal ancillary data (HANC). Ancillary data can also be located in non-picture regions of the frame. This is known as vertical ancillary data (VANC).

1. By default, the **bypass** option is set to **false**. This allows you to strip the VANC data from the incoming signal.



2. Set the option to **true** if you wish to enable the bypass function. In this instance, the VANC data is re-used from the original source to the transmitter.

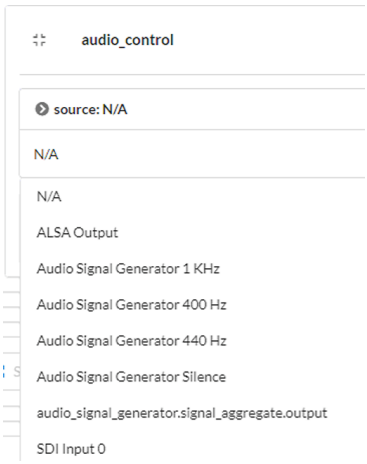
3. For more advanced setups, you can also override the SMPTE (set **override_smpte** to true) and force a **payload** (enter a value).

10.2.9 SDI Output Audio Control

The **audio_control** menu can be used to define the audio part of the SDI output.

Possible audio sources include the SDI inputs on the rear panel and the internal audio signal generator signals.

1. Use the **source:** menu to choose the audio source.



2. Then use the **group_enable** options to define the embedded channels.

Each group contains 4 mono audio channels which can be set to **Embed**, **Bypass** or **Off**.

- **Embed** = the group is enabled, and channels from the selected audio source are embedded into the output.
- **Bypass** = the group is enabled, and silent audio channels are embedded into the output.
- **Off** = the group is disabled.

So, to create a SDI output with 16 audio channels, you must set the first four **group_enable** options to either **Embed** or **Bypass**.

11. Streamer Configuration

This chapter describes how to configure the C100 as a signal streamer (TX).

11.1 Stream Formats

The C100 can stream signals in SMPTE ST2022-6 (SDI-over-IP) and SMPTE ST2110.

The advantage of using 2110 over 2022-6 is bandwidth optimization (between ±15 and ±40%), and the fact that 2110 supports separate streams for video, audio and metadata.

SMPTE ST2022-6 works more like an embedded SDI signal that has been converted to an IP stream. Thus, it requires a de-embedder on the receiver side to be able to use the audio/video/metadata components.

When using SMPTE ST2110, the video and audio transmitters must be created separately.

For more information about the two standards, please refer to the SMPTE documentation at https://www.smpte.org.

11.2 Configuring a Transmitter

Video transmitters can be created using either the configuration scripts or advanced page. In each case, you must choose a network interface, create a video transmitter and then configure the transmitter to define the stream format, video source and other options.

When determining how many transmitters to create, you must take care NOT to overload the bandwidth of the network interface. For example, the bandwidth required for a basic 1080p50 video signal is:

- 1080p50 @ SMPTE ST 2022-6 = 3,074Gbps.
- 1080p50 @ SMPTE ST 2110-20 = 2,39Gbps.

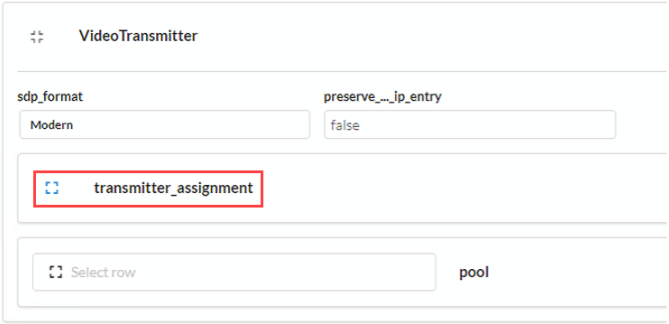
So, if you are using one 10GbE link, the maximum number of TX would be 3 for ST 2022-6 and 4 for ST 2110. For a 40GbE link, the maximum number of TX increases to 13 for ST 2022-6 and 18 for ST 2110.

The example which follows describes how to create a video transmitter manually from the advanced page.

11.2.1 Open Video Transmitter

1. Start by opening the advanced page in the usual manner.
2. Open **Video Transmitter** by clicking on its shortcut in the MODULES list, and then clicking on its icon.

Here you will find the **transmitter_assignment** menu which can be used to create a new video transmitter pool.



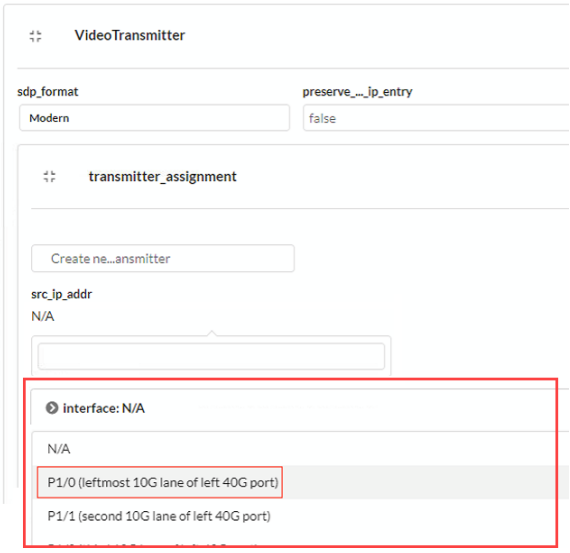
It is recommended to stay in **Basic** mode to keep things simple.

The **sdp_format** should be left as **Modern**.

11.2.2 Select Network Interface

1. Open the **transmitter_assignment** menu by clicking on its blue icon.
2. Use the **> Interface:** menu to choose the network interface you wish to use.

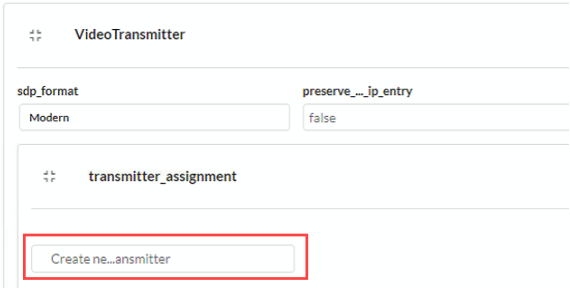
In our example, port **P1/0** is selected. This is the leftmost 10GbE lane of the first 40GbE port.



11.2.3 Create a Transmitter Pool

Once you have selected the streaming port, the next step is to create a transmitter pool.

1. Click on the **Create new transmitter** button to create a pool.

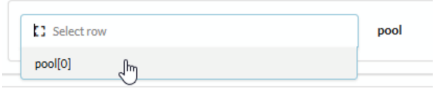


For our example, one pool is enough. A script can be used to create multiple pools if you wish.

Notice that the pools are counted from 0 upwards, so **pool [0]** is the first pool, **pool [1]** is the second pool, and so on.

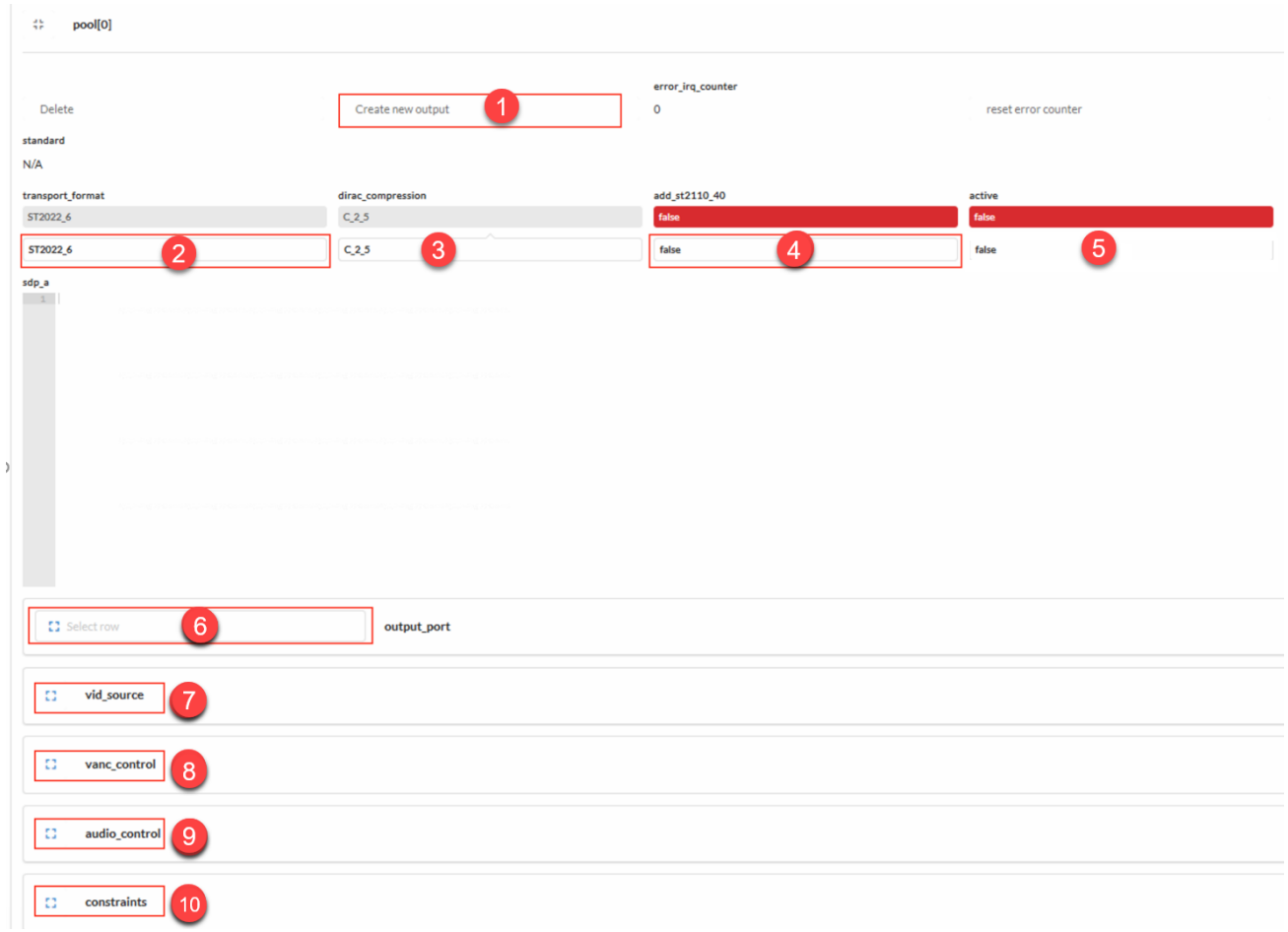
When determining how many pools to create, you must take care NOT to overload the bandwidth of the network interface (as described earlier).

2. Once a pool exists, it can be selected from the drop-down menu.



11.2.4 Configure a Transmitter Pool

1. Configure the selected transmitter pool as follows.



1 Create new output

By default, each pool comes with a single output which is enough for a non-redundant network.

To configure redundant streaming using SPS (Seamless Protection Switching), then two ports are required: one for the red network and one for the blue network. Use this button to create the second output port.

2 transport_format

This field defines the stream format. All of the possible options are described later. For now, select **ST2022_6** to configure a SMPTE ST2022-6 stream.

3 dirac_compression

This field defines the type of compression applied to the video. It does nothing when using ST2022-6 or ST2110, as these are uncompressed formats. It applies when using SMPTE ST2042, as described later.

4 add_st2110_40

When using SMPTE ST2110, this field can be set to **true** to activate the metadata.

5 active

This field is used to make the stream available to the network.

For now, leave the value set to **false** until all of the options are finalized. This ensures that all possible options remain available, and avoids publishing a "bad" stream to the network.

6 output_port

This menu selects an output port and then provides access to its stream settings.

7 vid_source

This menu defines the video source for the transmitter pool. The selection determines the video and audio content of the SDP.

8 vanc_control

This menu can be used to bypass the vertical ancillary (VANC) data received from the source signal.

9 audio_control

For a SMPTE ST2022-6 stream, this menu defines the embedded audio channels.

10 constraints

This menu can be used to apply constraints to the transmitter pool. For example, to limit the permitted video standard, set the maximum bandwidth or reserve bandwidth for UHD resources. It also allows you to enter custom-defined information into the SDP.

The next few pages describe the **vid_source**, **vanc_control**, **audio_control**, **constraints** and **output_port** menus in more detail.

11.2.5 TX Video Source

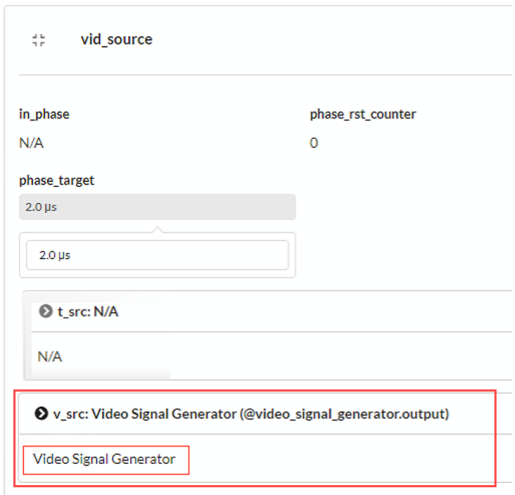
The **vid_source** menu defines the timing reference and video source used for the transmitter pool.

Possible video sources include SDI inputs on the rear panel and the internal video signal generator.

The available SDI inputs depend on the type of I/O module fitted to the C100 rear panel. For example, if the module is an SDI 18+2 there are 18 SDI inputs. Notice that these are numbered from 0 upwards, so **SDI Input 0** is the first input, **SDI Input 1** is the second input, and so on.

1. Use the **> v_src:** menu to make a selection.

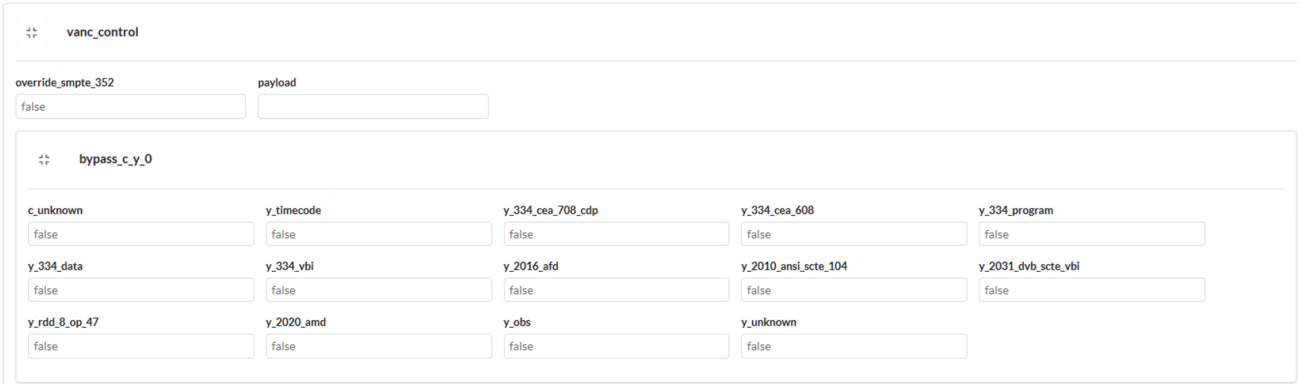
In our example, the **Video Signal Generator** is selected to be sure that there is a valid video signal.



11.2.6 TX VANC Control

The **vanc_control** menu can be used to bypass the vertical ancillary (VANC) data received from the source signal. The options work in a similar manner to the Output VANC Control described earlier.

Under **bypass_c_y_0**, you can select what types of metadata you wish to strip from the incoming signal. In each case, right-click on the true/false value fields to view more details.



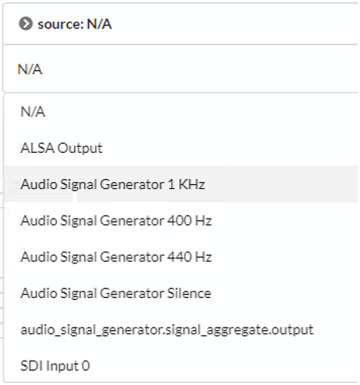
11.2.7 TX Audio Control

When using SMPTE ST2022-6, the **audio_control** menu defines the audio source for the stream.

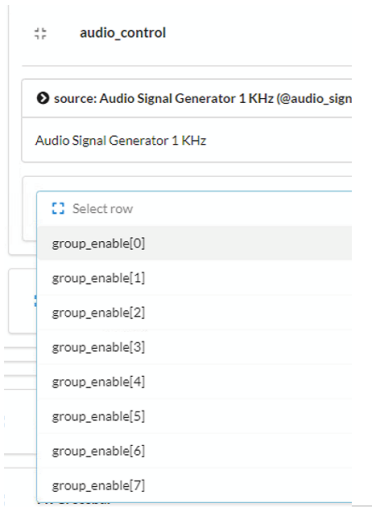
Possible audio sources include the SDI inputs on the rear panel and the internal audio signal generator signals.

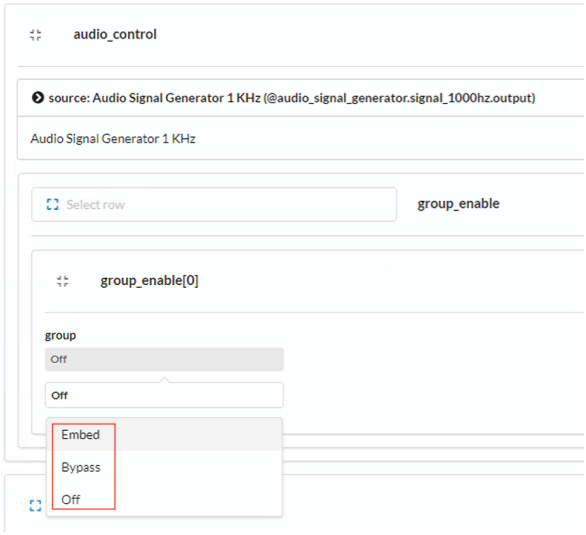
If you are setting up SMPTE ST2110, then you will need to use a separate audio TX.

1. Use the **source:** menu to choose an audio source.



2. Then use the **group_enable** options to define the embedded channels.





Each group contains 4 mono audio channels which can be set to **Embed**, **Bypass** or **Off**.

- **Embed** = the group is enabled, and channels from the selected audio source are embedded into the stream.
- **Bypass** = the group is enabled, and silent audio channels are embedded into the stream.
- **Off** = the group is disabled.

So, to create a TX stream with 16 audio channels, you must set the first four **group_enable** options to either **Embed** or **Bypass**.

To minimize the bandwidth used by the stream, you can disable all unwanted groups by setting the **group_enable** options to **Off**.

11.2.8 TX Constraints

The **constraints** menu can be useful if you wish to limit the permitted video standard, set the maximum bandwidth or reserve bandwidth for UHD resources.

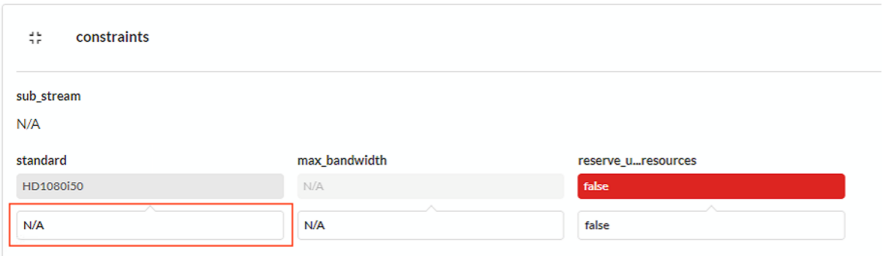
In most use cases it is best to set these options to **N/A** to prevent the blocking of signals by a constraint.

➢ **Limiting the Video Standard**

The **standard** shown in gray is the current detected video signal format.

1. Use the drop-down menu to apply a constraint.

If you choose **N/A**, then there is no limitation and all possible video source signals are permitted.



➢ **Setting the Maximum Bandwidth**

A **max_bandwidth** can be set to limit the bandwidth used by the stream.

1. Use the drop-down menu to apply a constraint. The options are **1.5Gb**, **3Gb** or **12Gb**.

If you choose **N/A**, then the bandwidth is not constrained.

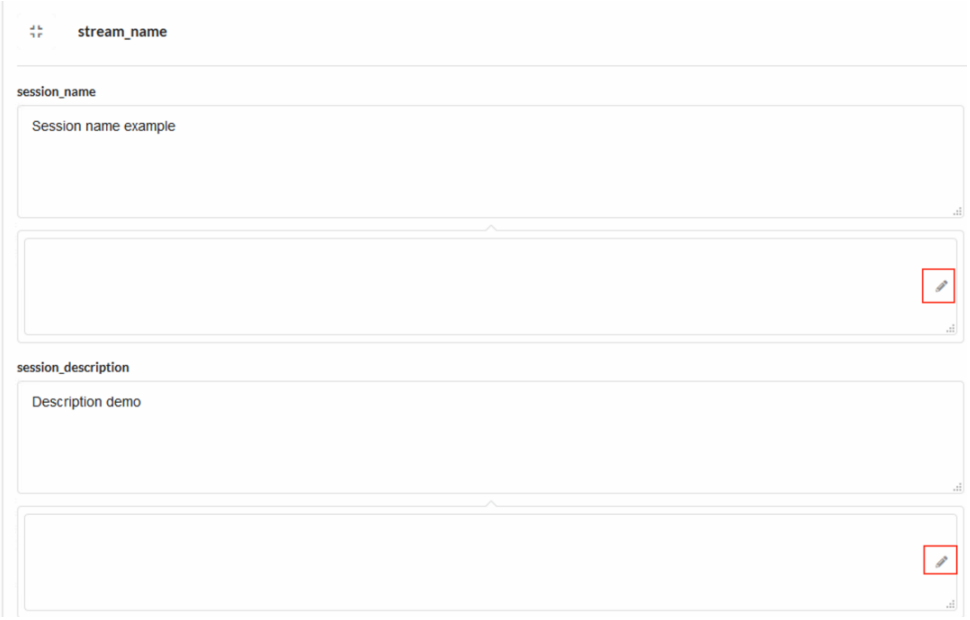
➢ **Reserving UHD Resources**

This option can be set to either **true** or **false**. When set to **true**, this transmitter will allocate enough resources to stream a UHD signal.

➢ Editing the Stream Name

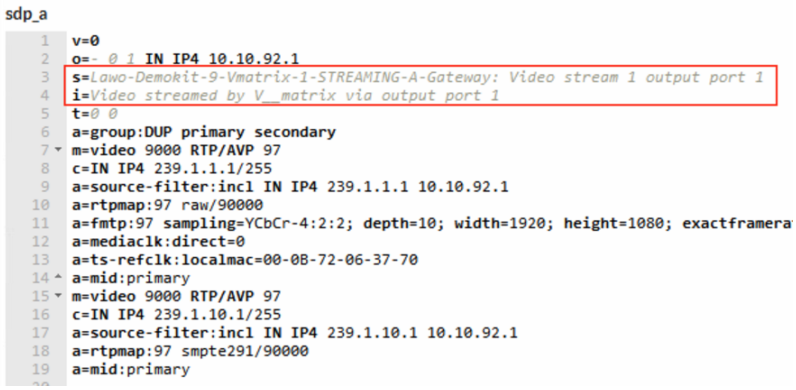
Under the **constraints** menu, you will find the **stream_name**. This can be used to enter a custom-defined session name and session description for the SDP.

1. Open the **stream_name** menu by clicking on its icon.



2. Click on the pencil icon to enter edit mode, and type in the required information.
3. Press SHIFT + ENTER to confirm and apply.

The entries are applied to the SDP once the stream becomes active.



11.2.9 TX Output Port

The **output_port** menu selects an output port, and then provides access to its stream settings: multicast IP and payload.

By default, each pool comes with a single output which is enough for a non-redundant network. To configure redundant streaming using SPS (Seamless Protection Switching), two ports are required: one for the red network and one for the blue network.

1. Use the drop-down menu to select the port you wish to configure.
2. Then use the **video_stream_data** and **meta_stream_data** sub menus to configure the stream settings.

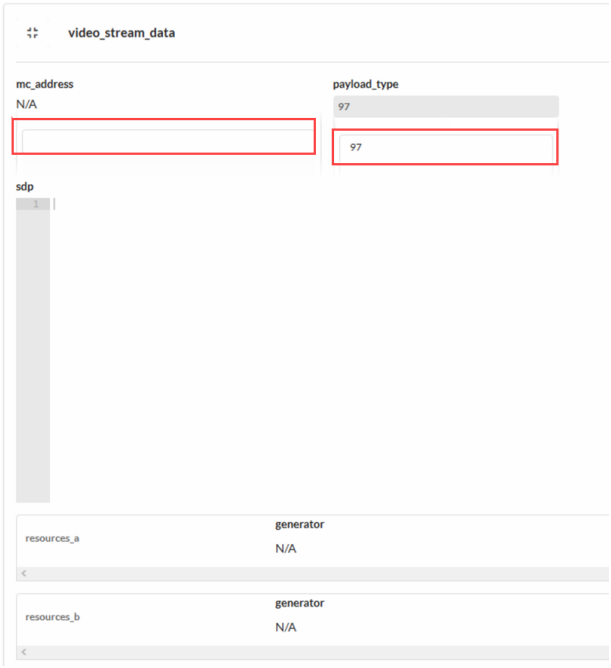
When using SMPTE ST2110, the video and metadata are streamed separately.

When using SMPTE ST2022-6, there is no separate metadata transmission.

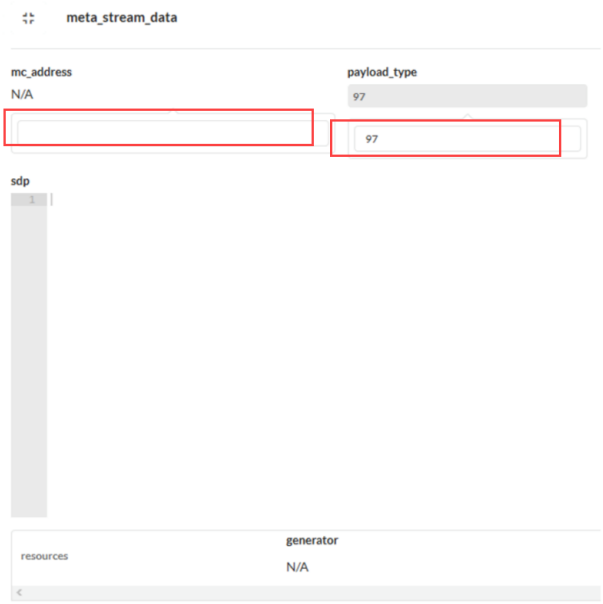
Define Multicast IP & Payload

1. Open each of the sub menus by clicking on its icon.

video stream data



meta stream data



2. Use the **mc_address** field to enter the multicast group to transmit the stream.

You can either type in the multicast IP address or use copy/paste. Then press Enter to apply. Once a valid address is entered, it replaces **N/A** (in the current value field).

If you are including the SMPTE ST2110-40 metadata in the same SDP, then it is best to find a logical way of defining the multicast list. For example:

- Video: 239.**1**.1.1
- Audio: 239.**2**.1.1
- Metadata: 239.**3**.1.1

The default port is 9000, but you can define a different port by adding the port number. For example: 239.1.1.1**:9001**

You can find more information about port numbers in the TCP/UDP Ports appendix.

3. Use the **payload** field to define the payload.

The default payload is **97** but this can be adapted if required. For example, to define different payloads for each signal type: video, audio and metadata.

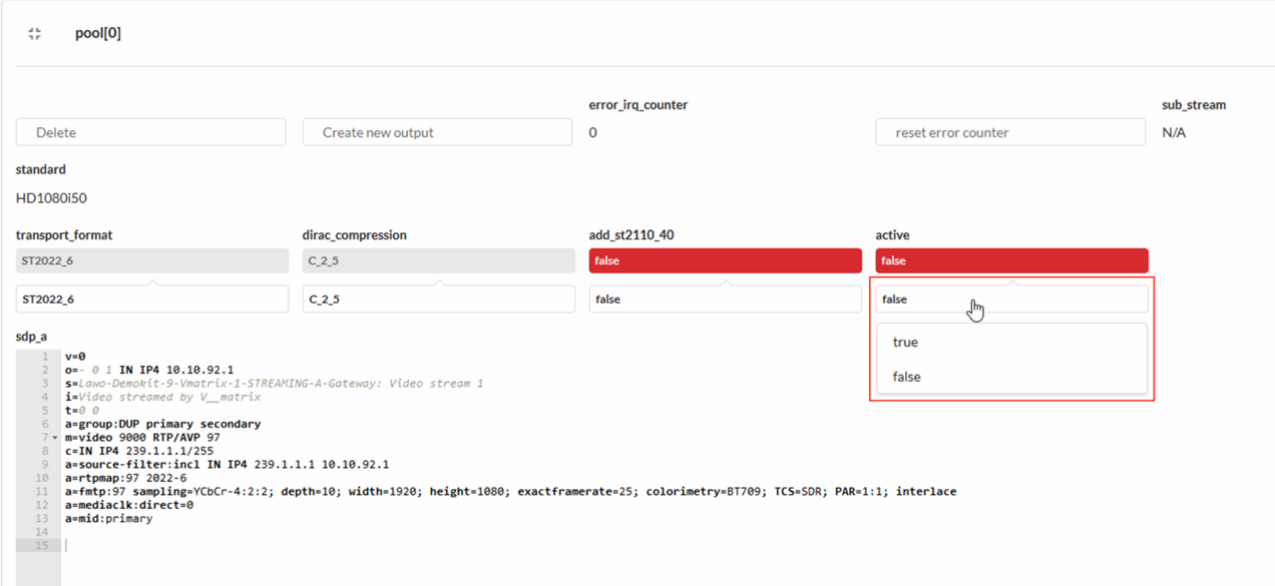
Usually, it is best to keep the default value for TX, and adapt the RX side as required.

If you do edit the TX payload, then make sure the payload matches in the RX.

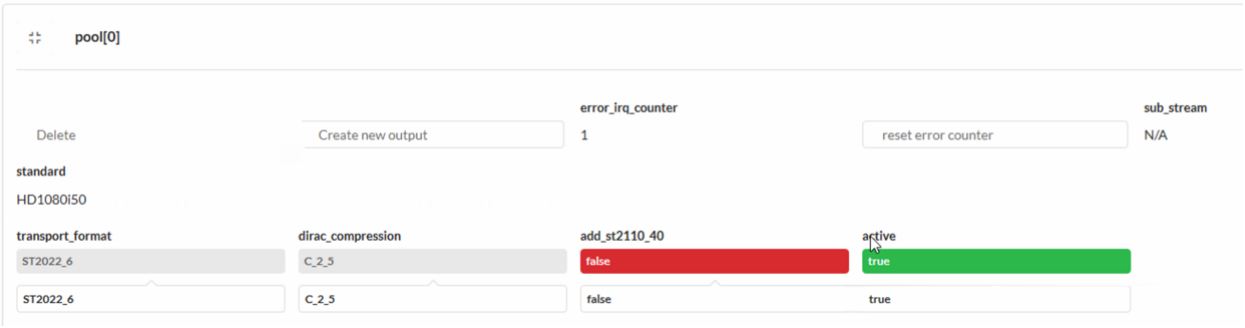
11.2.10 Activate Pool

Once all options are configured correctly, it is time to active the stream so that it becomes available on the network.

1. Return to the **pool** options.
2. Set the **active** field (on the right) to **true**.



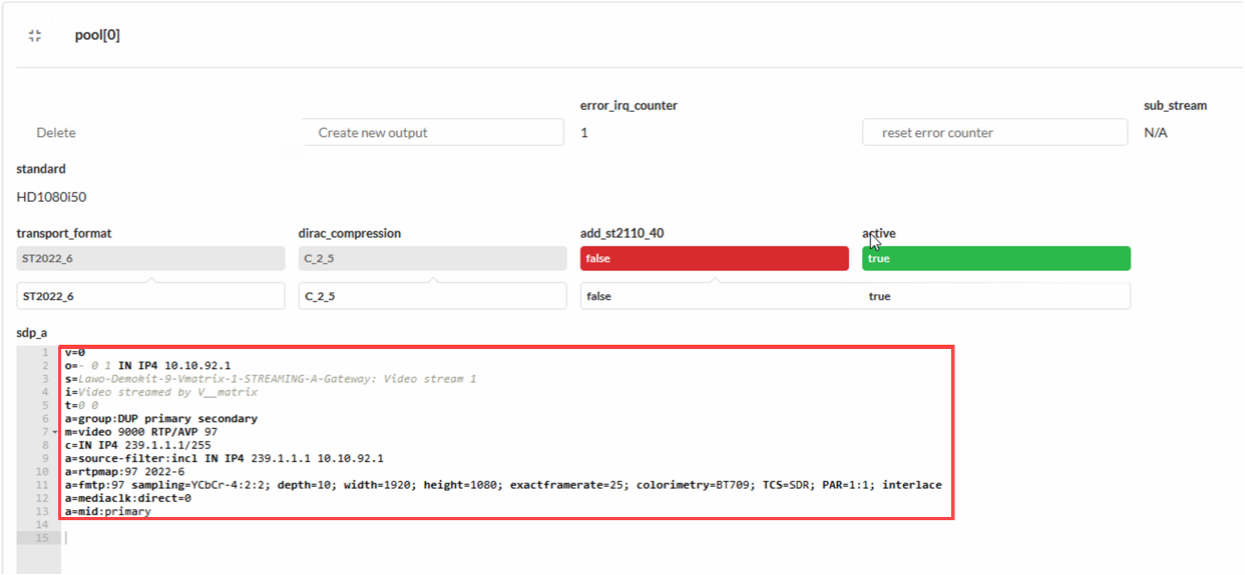
The **active** field turns green to show that the stream is active.



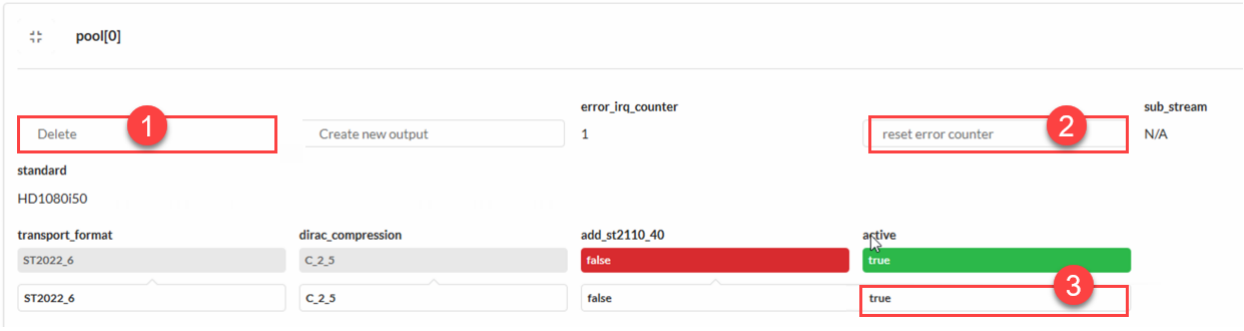
11.2.11 SDP & Other Functions

Copying the SDP

The SDP information is created from the detected video source. The text can be copied so that it may be pasted into a receiver.



Other Functions



1. If you need to delete the selected pool, click on the **Delete** button.

Take care as there is no confirmation or undo for this function.

2. Click on the **Reset error counter** button to reset the **error_irq_counter**.
3. To disable the stream, set the **active** field back to **false**.

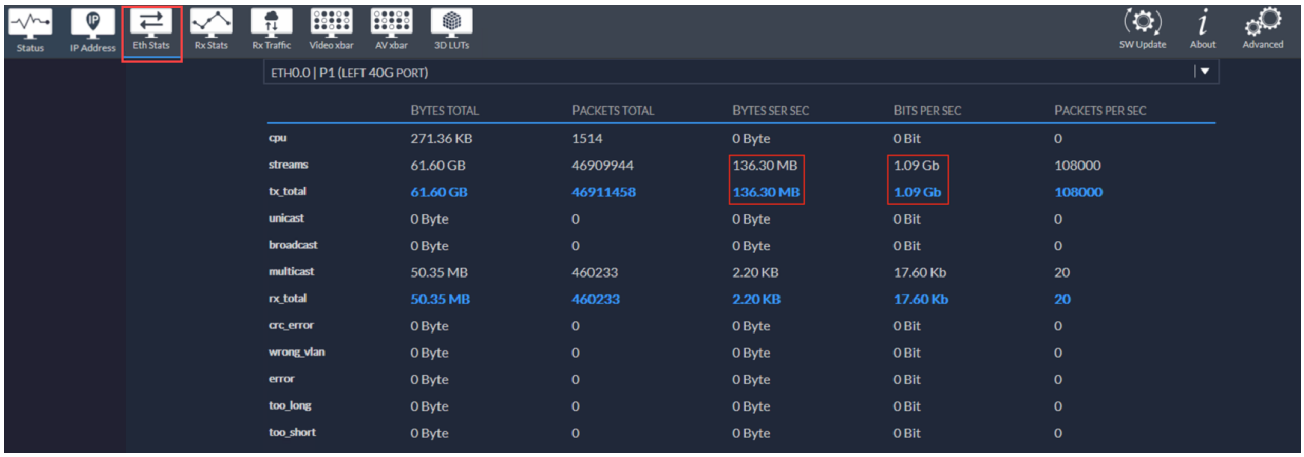
By toggling the **active** state to **false** and back to **true**, the stream can be stopped and restarted.

11.2.12 Monitor the TX Streams

You can now open the main web UI "Eth Stats" page to check the streams on the TX side.

1. Select the network port you wish to monitor.
2. Under tx streams, you can see the current used bandwidth.

The example below shows 1x SMPTE ST2110 stream carrying 1080i50 video only, and so the used bandwidth = 1.09Gb.



	BYTES TOTAL	PACKETS TOTAL	BYTES SER SEC	BITS PER SEC	PACKETS PER SEC
cpu	271.36 KB	1514	0 Byte	0 Bit	0
streams	61.60 GB	46909944	136.30 MB	1.09 Gb	108000
tx_total	61.60 GB	46911458	136.30 MB	1.09 Gb	108000
unicast	0 Byte	0	0 Byte	0 Bit	0
broadcast	0 Byte	0	0 Byte	0 Bit	0
multicast	50.35 MB	460233	2.20 KB	17.60 Kb	20
rx_total	50.35 MB	460233	2.20 KB	17.60 Kb	20
crc_error	0 Byte	0	0 Byte	0 Bit	0
wrong_vlan	0 Byte	0	0 Byte	0 Bit	0
error	0 Byte	0	0 Byte	0 Bit	0
too_long	0 Byte	0	0 Byte	0 Bit	0
too_short	0 Byte	0	0 Byte	0 Bit	0

Some other examples are:

- 1x SMPTE ST2022-6 stream carrying 1080i50 video only; used bandwidth = 1.55Gb
- 1x SMPTE ST2022-6 stream carrying 1080p50 video only; used bandwidth = 3.10Gb
- 1x SMPTE ST2110 stream carrying 1080i50 video only; used bandwidth = 1.157Gbps
- 1x SMPTE ST2110 stream carrying 1080p50 video only; used bandwidth = 2.315Gbps

V__matrix uses type N transmitters. There are 3 types of TX: Narrow (N), Narrow Linear (NL) and Wide (W).

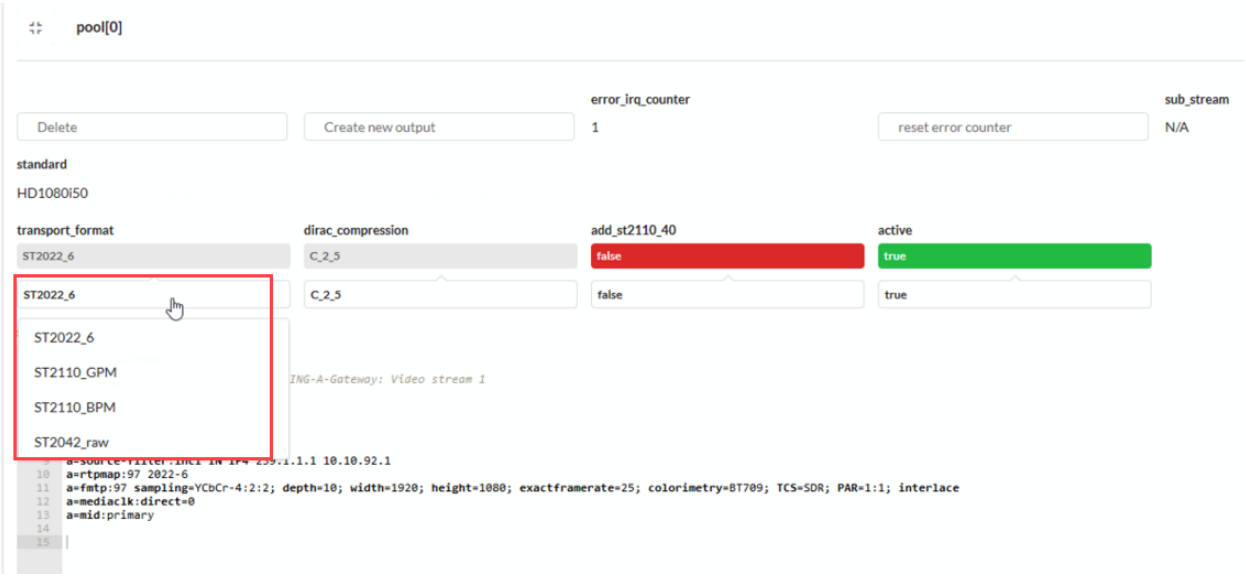
If you wish to dive more into the detail of Narrow/Wide receivers and transmitters, please refer to the following information from the IABM:

https://theiabm.org/deep-dive-smpte-st-2110-21-traffic-shaping-delivery-timing-video/

11.3 Changing the Stream Format

The **transport_format** field can be used to switch between SMPTE ST2022-6 and SMPTE ST2110 streams.

Before changing stream parameters, make sure to deactivate the stream before applying the changes. The re-activate for the changes to take effect.



The drop-down menu shows all of the possible options:

- **ST2022_6** - creates a SMPTE ST2022-6 stream.
- **ST2110_GPM** - creates a SMPTE ST2110 stream with GPM packing.
- **ST2110_BPM** - creates a SMPTE ST2110 stream with BPM packing.
- **ST2042_raw** - creates a SMPTE ST2042 stream that supports VC-2 compression. When using this format, the correct option should be set in the **dirac_compression** field.

When using either of the ST2110 formats, remember to create the audio transmitters separately.

GPM vs BPM

The SMPTE ST2110-20 standard defines two packing modes for video signals: GPM (General Packing Mode) and BPM (Block Packing Mode). GPM is the more general case, while BPM is a restricted subset of GPM.

The standard requires all receivers to receive GPM, which means that all receivers are also capable of receiving BPM. It follows, therefore, that all formats can be defined as using GPM, and even if a specific sender emits BPM, signal interoperability is still achieved. Thus, there is no need to define specific formats that are BPM-only.

However, according to the ST2110-20 standard, each source is required to indicate the type of packing mode by including either “PM=2110GPM” or “PM=2110BPM” in their SDP definition.

12. Receiver Configuration

This chapter describes how to configure the C100 as a signal receiver (RX).

12.1 Configuring a Receiver

By default, there are no receivers in the configuration. They must be created using either the configuration scripts or advanced page. In each case, you must configure a session (to receive the stream) and then define its video and audio receivers.

When determining how many video receivers to create, you must take care NOT to overload the bandwidth of the network interface. This works in a similar manner to the TX described earlier. So, if you are receiving a basic 1080p50 video signal on one 10GbE link, the maximum number of RX would be 3 for ST 2022-6 and 4 for ST 2110. For a 40GbE link, the maximum number of RX increases to 13 for ST 2022-6 and 18 for ST 2110.

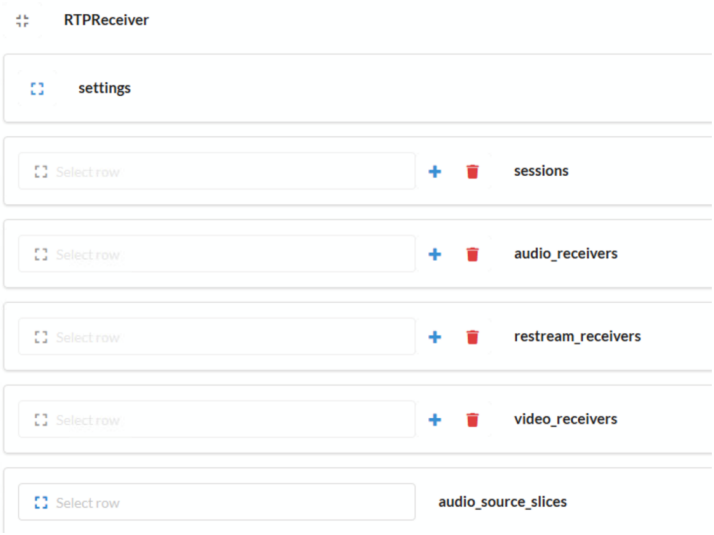
> It is not possible to use the same unicast IP to transmit and receive the same stream, and so you must use different network ports for the transmitter and receiver.
>
> This is important if you are thinking about using the same C100 to TX and RX. When using 4x 10GbE, you could use a different port with its own unique IP address. e.g. TX on Port0 and RX on Port1. However, the simplest solution is to use two separate C100 cards.

The example which follows describes how to create a receiver manually from the advanced page.

The method described here can be used to help understand the scripts and software options. This can be useful if you need to modify a script or fault-find the system.

12.1.1 Open RTP Receiver

1. Start by opening the advanced page in the usual manner.
2. Open **RTP Receiver** by clicking on its shortcut in the MODULES list, and then clicking on its icon.



The RTP receiver has four blocks of settings which are used as follows.

- **sessions** - activate the receiver; change or update the SDP; configure the stream switching mode.
- **audio_receivers** (required for SMPTE ST2110 only) - define the audio receiver and monitor its status.
- **restream_receivers** - will be skipped for now.
- **video_receivers** - define the video receiver and monitor its status.

3. By default, there are no sessions or receivers so they must be created by clicking on the **+** buttons.

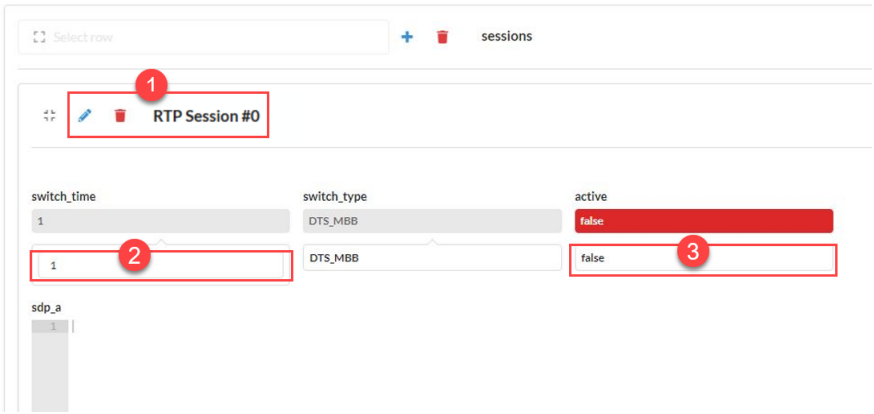
It is best to create a session first and then define the receivers.

The trash can icons can be used to delete ALL sessions or receivers that are not active. Take care as there is no confirmation or undo for this function.

4. Once a session or receiver exists, it can be selected from the drop-down menu.

12.1.2 Configure Session

1. Configure the selected session as follows.



1 Session Name

Optionally, the session name can be edited by clicking on the pencil icon. Remember to press Enter to apply the changes.

2 switch_time & switch_type

Whether and how a receiver switches from one stream to another is determined by the session's **SwitchTime** and **SwitchType** parameters as follows.

➢ **SwitchTime**

- A **SwitchTime** of exactly **0** is interpreted as DoNotSwitch: in this state, no transition is ever made from one stream to another.
- A **SwitchTime** of exactly **1** is interpreted as Patch: in this state, the **SwitchType** setting is ineffective and the receiver operates in patching mode. This results in a stream transition that is not frame-accurate (dirty switch).
- Any other **SwitchTime** defines an absolute switching time relative to the PTP epoch at which a clean switch of type **SwitchType** should be performed. We recommend to set this parameter to **2** for a clean switch that occurs as early as possible.

➢ **SwitchType**

- **DTS/MBB**: Destination-timed switching, make before break. In this mode, the new stream is received in parallel to the old stream for a very short amount of time (until the receiver completes the frame-accurate transition to the new stream). This results in the temporary doubling of bandwidth that should be considered before choosing this option.
- **DTS/BBM**: Destination-timed switching, break before make. In this mode, the new stream only gets subscribed to after the old stream is already unsubscribed. Thus, alleviating the need for temporary bandwidth duplication. To ensure a clean switch, the last frame of the old stream is played out from memory until the first frame of the new stream arrives.

3 active

This field is used to switch the receiver on or off.

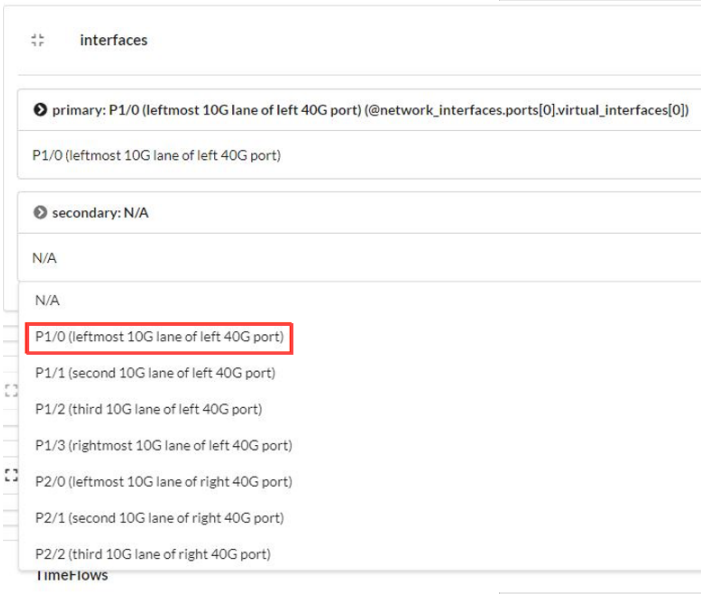
For now, leave the value set to **false** until the video and audio receivers are configured.

2. Open the **interfaces** menu to select the network interfaces.

In a non-redundant network, only one interface (**primary**) is required.

In a redundant setup, both interfaces must be specified: **primary** for the red network and **secondary** for the blue network.

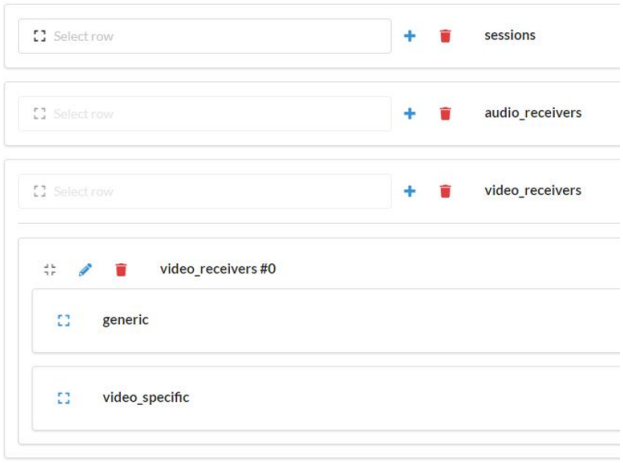
In each case, use the drop-down menus to select the network port.



12.1.3 Add Video Receivers

A number of video receivers can now be linked to the session. When determining how many video receivers, you must take care NOT to overload the bandwidth of the network interface (as described earlier).

1. Click on the **+** button to create a new video receiver.
2. Once a receiver exists, it can be selected from the drop-down menu.



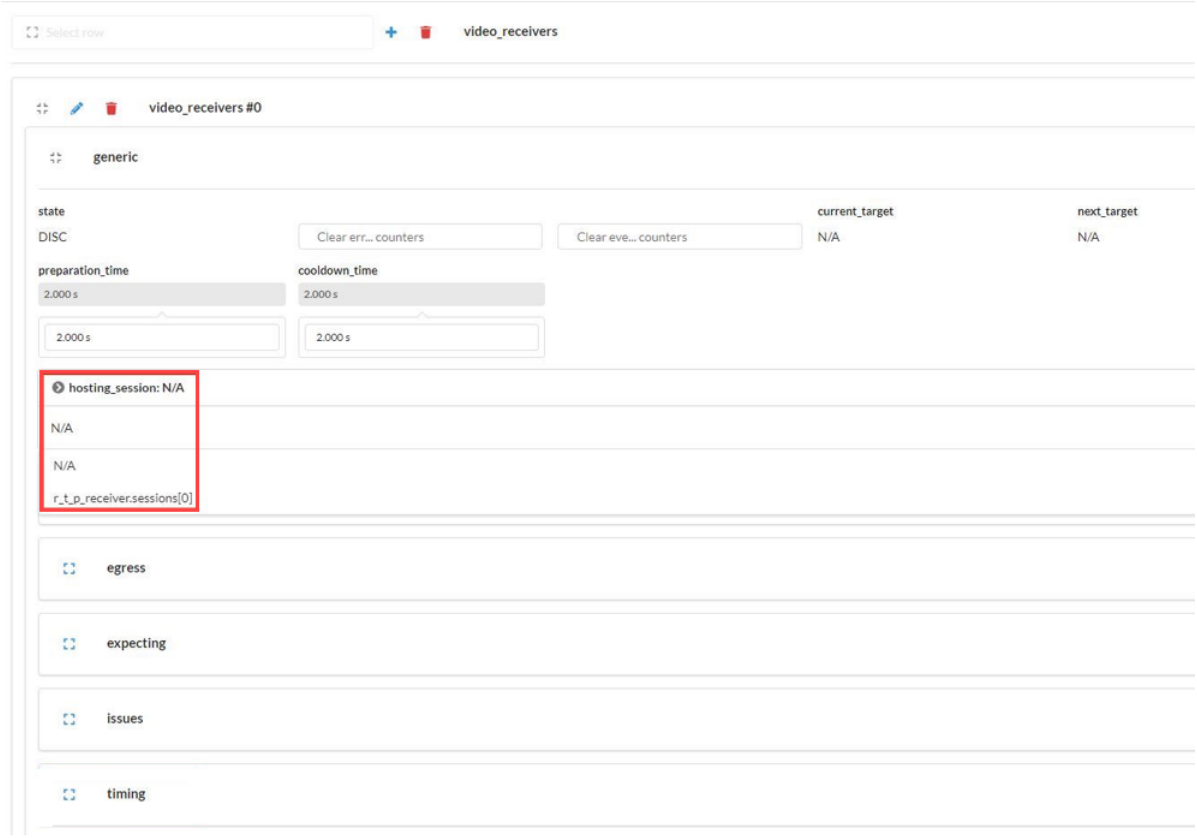
Optionally, you can change its name by clicking on the pencil icon. Remember to press Enter to apply the changes.

The video receiver has two sub menus which support the following options.

- **generic** - links the receiver to a session; adjusts timing.
- **video_specific** - supports options for ST-2042 and 12G; sets the frame shift policy.

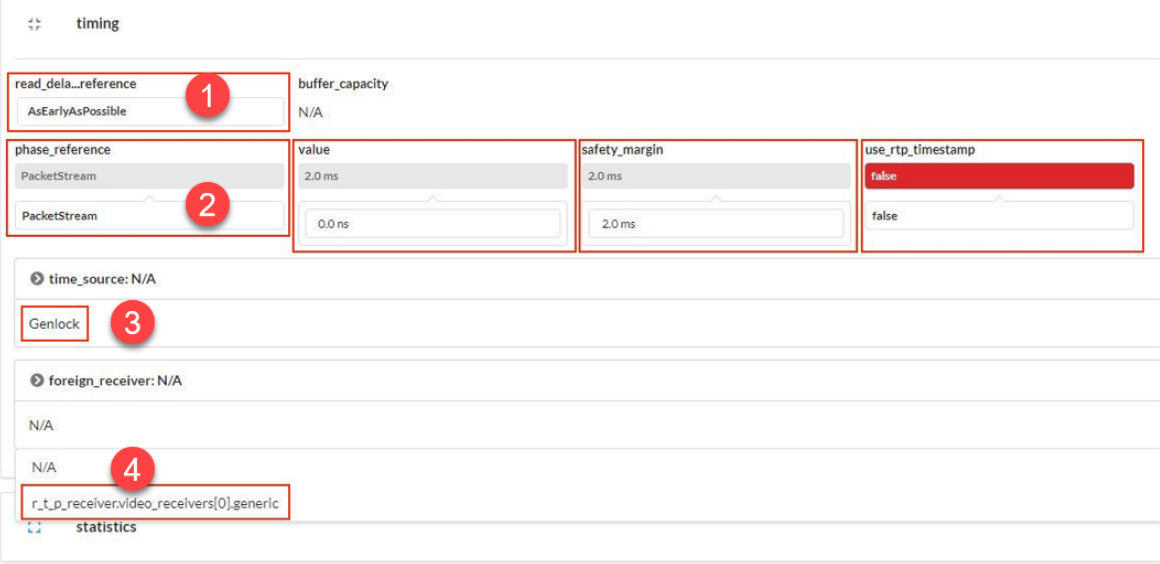
12.1.4 Configure Video Receiver

1. Open the **generic** menu for the selected receiver.
2. Use the **> hosting_session:** menu to link the receiver to a session.



3. Use the following sub menus for monitoring purposes:
   - **ingress** shows info about the incoming signal in the RTP receiver.
   - **egress** shows info about the outgoing signal of the RTP receiver.
   - **expecting** and **issues** show more details about the receiver.

4. Open the **timing** sub menu to adjust the following settings.



1 read_delay_reference

By default, the read delay is set to **As Early As Possible**.

2 phase_reference

By default, the phase reference is set to **Packet Stream**. In this instance, the **value** field defines how fast the receiver starts to decode the signal after it is received at the network port, and the **safety_margin** sets the minimum value. This is why the value in our example is set to 2.0 ms and not 0.0 ns. The phase reference can never be negative in **Packet Stream** mode.

The **use_rtp_timestamp** should be set to true when the phase reference is set to **Time Source** (except for multiviewers where Packet Stream works faster).

3 time_source

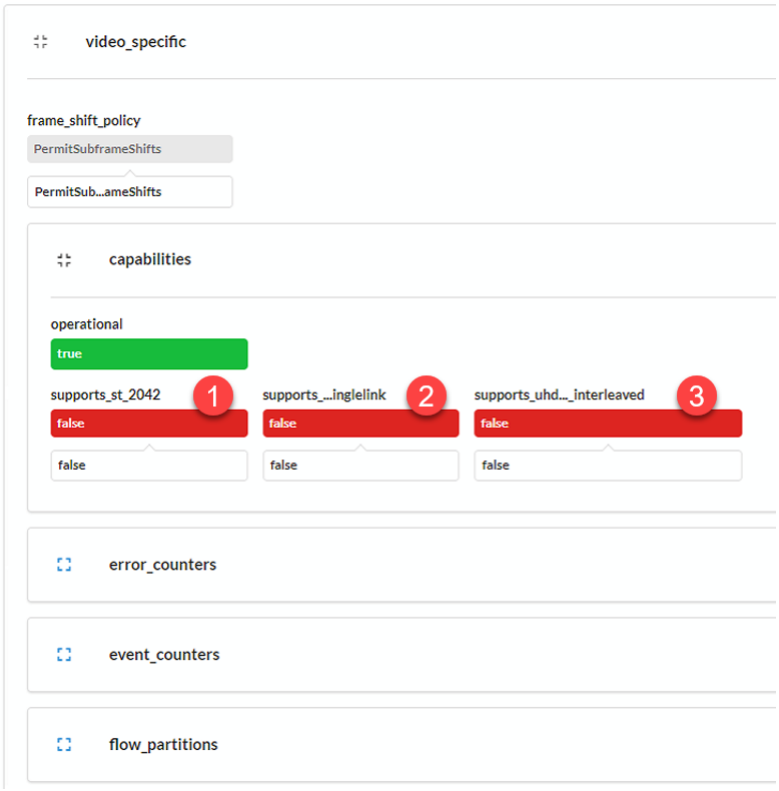
This field defines the time source when using Time Source mode. It should be set to **Genlock** to follow the PTP signal (as described earlier).

When using Packet Stream mode, the field is **N/A**.

4 foreign_receiver

This setting is optional.

5. Then open the **video_specific** menu for the selected receiver.



6. Under **capabilities** you can set the following options.

1 supports_st_2042

This option allows you to decode a VC-2 stream.

2 supports_singlelink

This option activates support for UHD 2110 single link streams, as opposed to quad links.

When using 12G you have one link for UHD. However, some devices do not support 12G and will use 4x 3G in quadrants to create a single UHD image.

3 supports_uhd_sample_interleaved

This option activates support for UHD Sample Interleave (2SI).

7. The **frame_shift_policy** can be set to one of three options:
   - **Permit Subframe Shifts** - the default.
   - **Prohibit** - used for RTP timestamps
   - **Permit Frame Shifts** - used for clean-switching.
8. The **error_counters**, **event_counters** and **flow_partitions** sub menus are used for monitoring purposes.

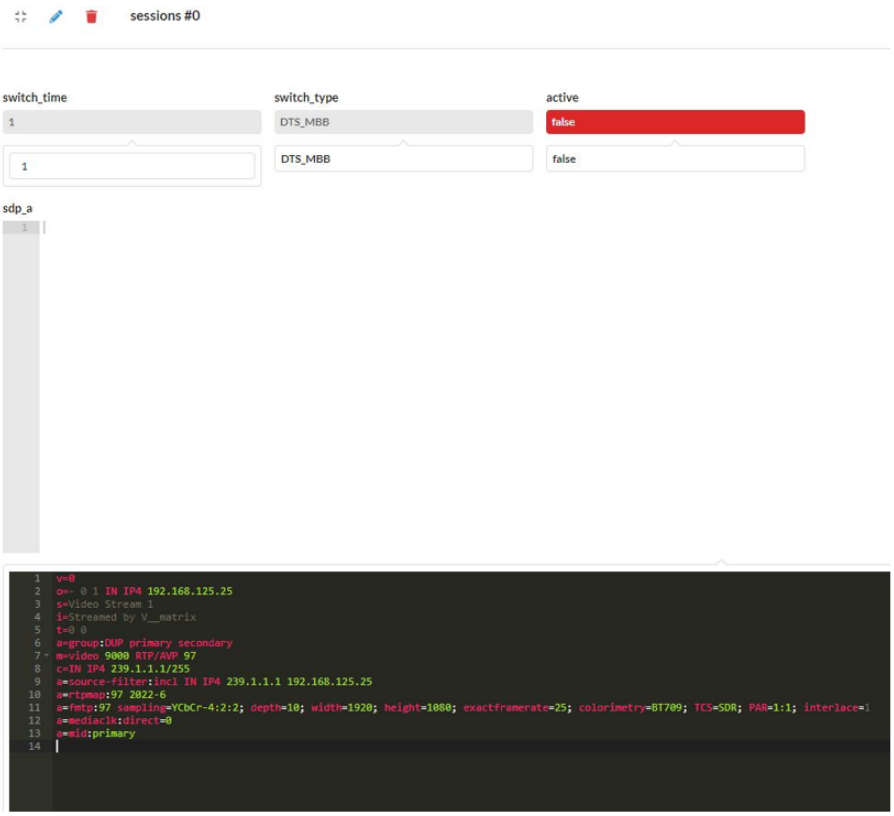
12.1.5 Activate Session & Paste SDP

To receive a stream, the receiver needs to have a valid SDP (Session Description Protocol). Usually this is provided to the C100 via API. However, it is also possible to copy the SDP from the transmitter and paste it into the receiver as follows.

1. Return to the **session** options.

The white part of the **sdp_a** window shows the current active "on-air" receiver. This is empty until the SDP is pasted and the session is active.

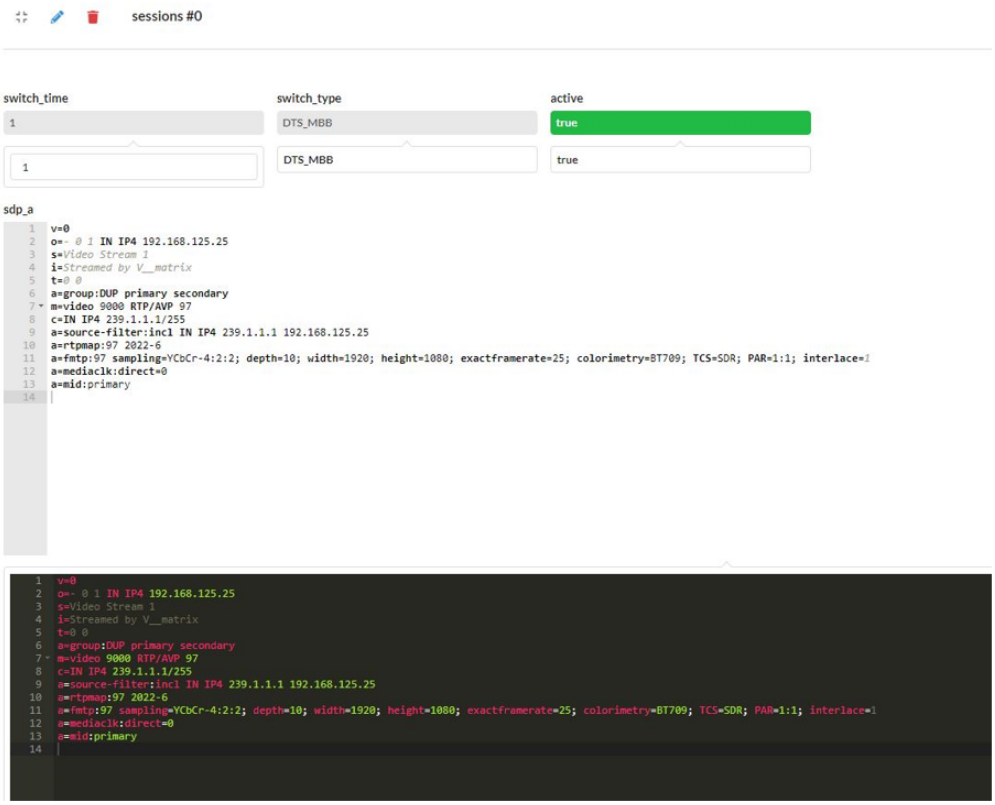
2. Paste the copied SDP into the black part of the **sdp_a** window. Then press SHIFT + ENTER.
3. Set the **active** field (on the right) to **true**.



The **active** field turns green to show that the session is active, and the white part of the **sdp_a** window updates accordingly.

4. If you need to adapt the SDP, then edit the contents of the black part of the **sdp_a** window, and press SHIFT + ENTER to apply.

It is wise to de-activate the stream before editing, but this is not mandatory. Whether the changeover occurs cleanly or not, is determined by the selected **switch_type** and **switch_time** (as described earlier).

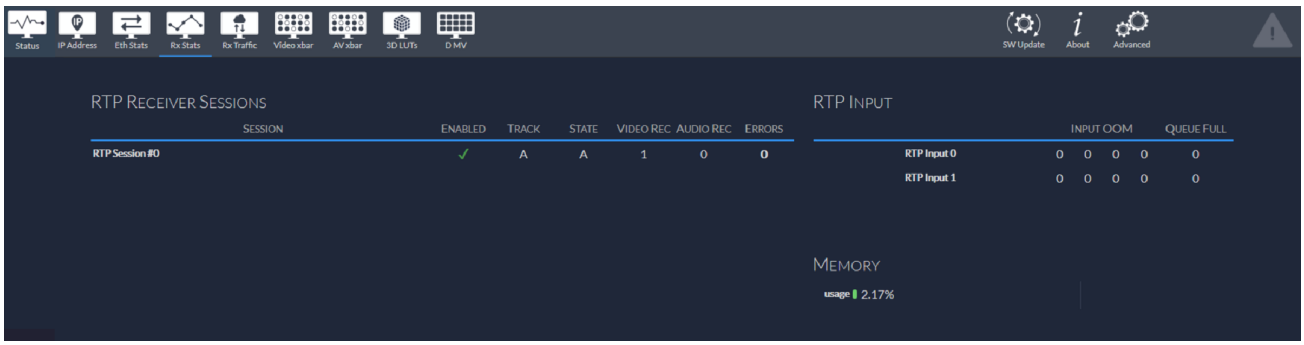


12.1.6 Validate the Stream

You can now open the main web UI to validate the stream.

1. Open the "Eth Stats" page and select the network port you wish to monitor.
2. The highlighted area shows the rx stats of the multicast.

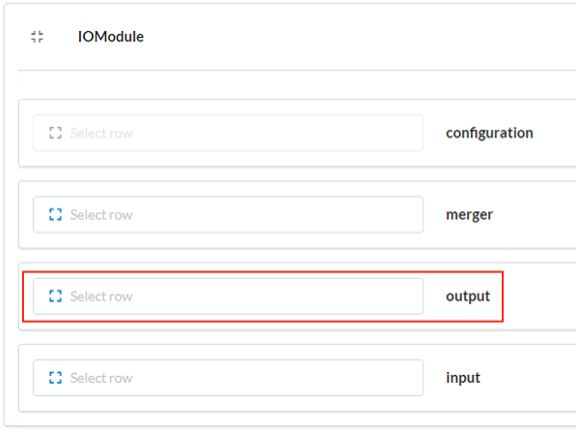
3. Open the "Rx Stats" page to view the sessions per stream. In our example, there is only one.



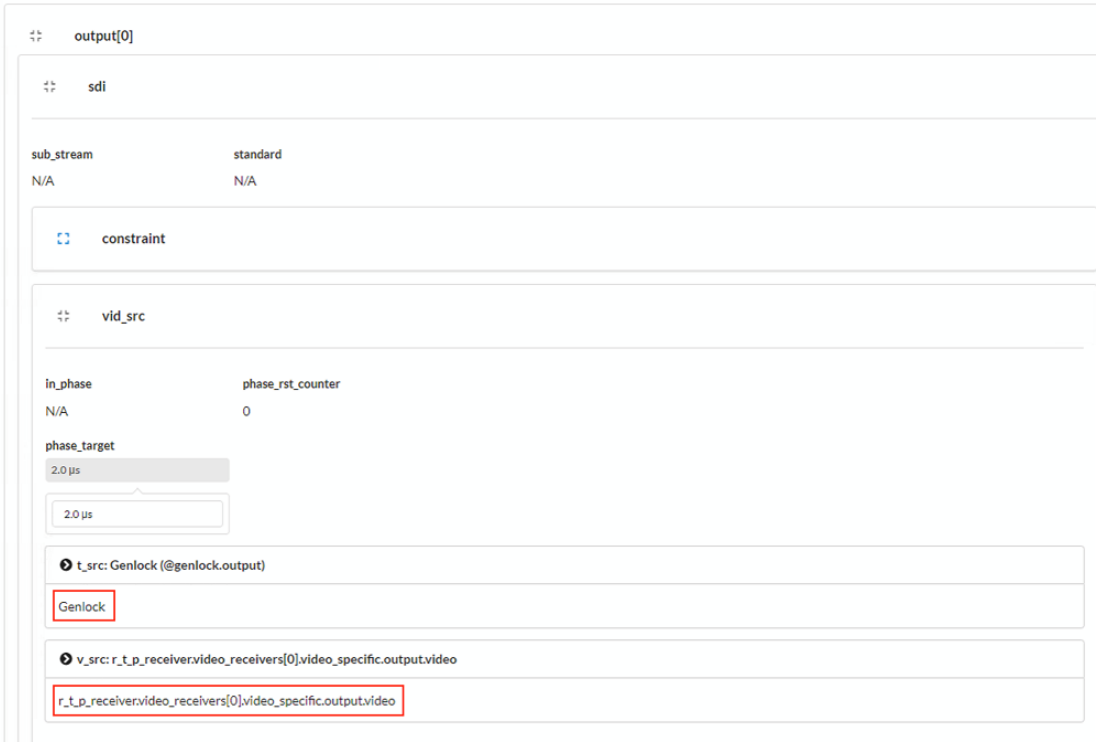
12.1.7 View Stream on SDI Output

If you wish to view the received stream on an SDI monitor, then you can use the rear I/O Module.

1. Open **I/O Module** by clicking on its shortcut in the MODULES list, and then clicking on its icon.
2. Open the **output** sub menu.



3. Select the output you wish to use - e.g. **output[0]**.
4. Use the **t_src** and **v_src** fields to define the timing source and video source.



When the Tx and Rx are both active, you will see the video out via the first BNC connector.

12.2 Receiving SMPTE ST2110 Streams

When using SMPTE ST2110, you will have a separate SDP for audio and video. This requires two RTP_receiver sessions to be configured, one for the audio and one for the video.

The video receiver must be linked to the video RX session, and an audio receiver linked to the audio RX session.

You can then check the signals by assigning the video to an SDI video out, and the audio to an SDI audio out.

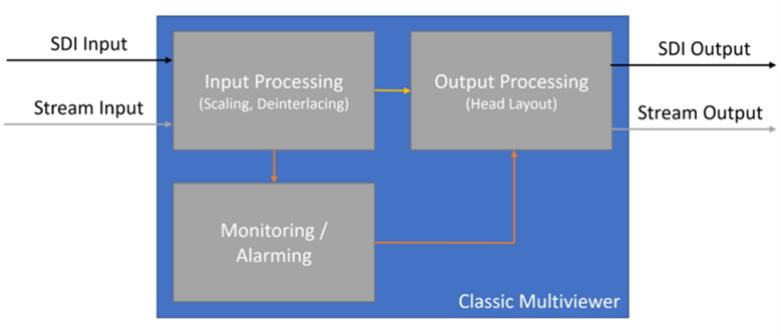
13. Distributed Multiviewer (DMV)

This chapter describes how to use the C100 as a distributed multiviewer (DMV).

13.1 Introduction

The C100 supports two types of multiviewer: the classic multiviewer (vm_mv) and distributed multiviewer (vm_dmv). The former runs on a single C100 card for stand-alone use, while the latter runs on multiple cards to form a cluster of nodes.

Classic Multiviewer (MV)

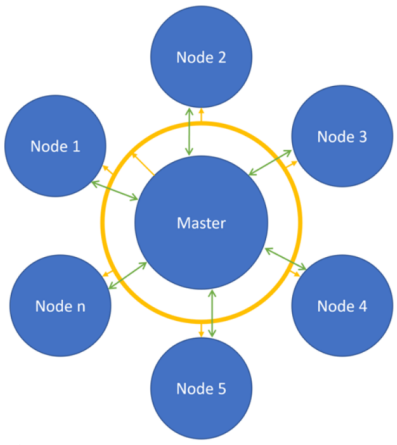


In the classic multiviewer, all necessary tasks are handled on a single C100 card:

- Input of source signals (via stream or SDI).
- Signal processing (scaling, de-interlacing, monitoring, alarming, etc.)
- Creation and layout of the multiviewer heads.
- Output of multiviewer head signals (via stream or SDI).

If a source is needed on multiple heads (processed by different cards), then it needs to be available on each card.

Distributed Multiviewer (DMV)

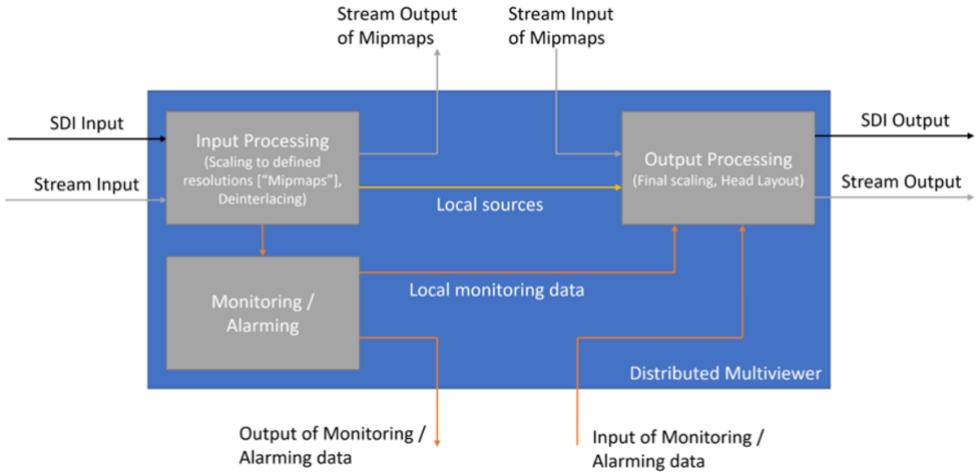


In the distributed multiviewer, a group of C100 cards operate together to form one or more clusters of nodes:

- One of the nodes in every cluster is elected as the master while the others are slaves.
- The configuration database is always shared to all nodes in a cluster.
- The DMV has input stages, monitoring objects and output stages (independent head outputs with PIPs, PPM meters, CC boxes, TC-boxes, Clocks, etc.)

13.2 Inputs & Outputs

Input and output stages are simply a logical separation. A C100 configured as a DMV cluster node acts as an input stage and output stage at the same time (no separation of hardware into inputs and outputs).



Input stage (video, audio, data)

The input stage provides:

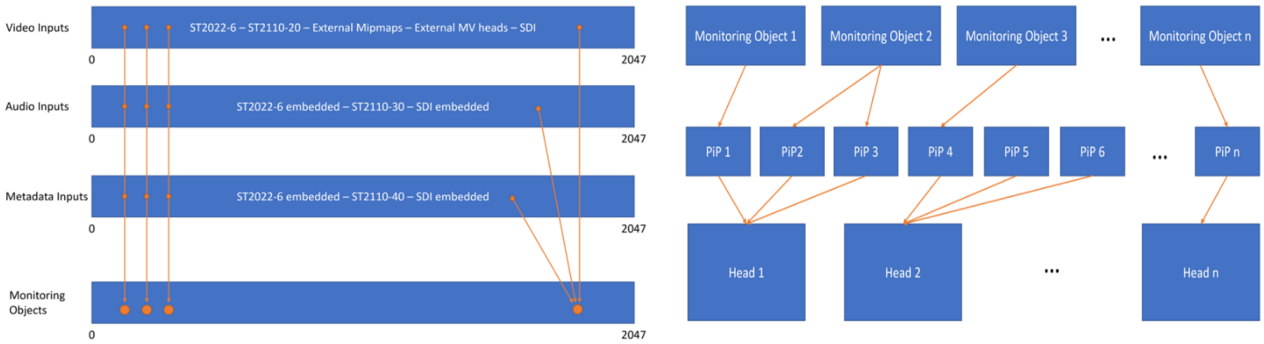
- Input of source signals (via stream or SDI).
- Pre-scaling of signals.
- Monitoring of signals (video loss, audio silence, etc.)
- Output of scaled versions and the corresponding monitoring to the network.

Output stage

The output stage provides:

- Input of pre-scaled signals and corresponding monitoring data.
- Creation and layout of the multiviewer head.
- Alarming based on monitoring data.
- Output of multiviewer head signals (via stream or SDI).
- Picks the mipmaps from the network in accordance with the final scale of the picture on the output head.

13.3 Monitoring Objects



The monitoring objects consist of:

- video, audio, ANC-data sources.
- video, audio, ANC-data alarms
- video, audio, ANC-data alarm settings.

The cluster master performs the following tasks:

- manages the assignment of monitoring objects to a PIP (Window) of any head output in the cluster.
- maintains a global list of monitoring objects which are synced to all cluster slaves.
- The master has TCP/IP WebSocket/JSON connections to slave nodes for subscriptions, and RTP/UDP multicast database synchronization of slave nodes.

13.4 Mipmaps

The scaled-down version of the original signal generated by the input stage are called “Mipmaps”.

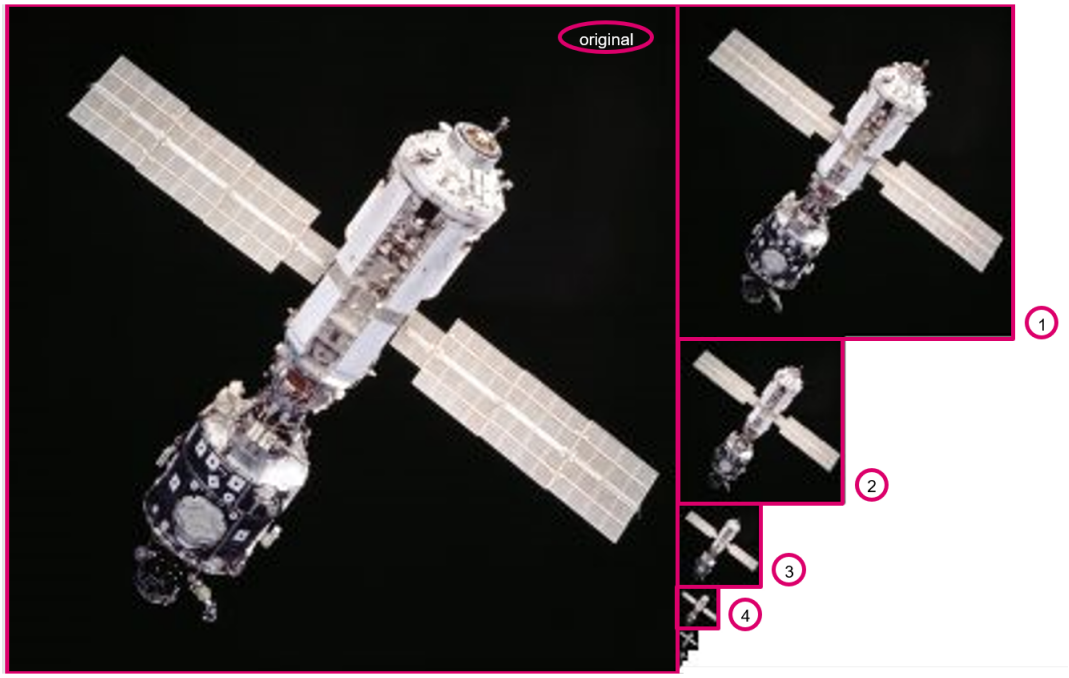
The input stage generates four resolution levels, scaling down the resolution to one-fourth of the previous resolution, as follows:

Original	3840 x 2160	1920 x 1080
Resolution Level 1	1920 x 1080	960 x 540
Resolution Level 2	960 x 540	480 x 270
Resolution Level 3	480 x 270	240 x 135
Resolution Level 4	240 x 135	120 x 67

The mipmaps generated by the input stage are formatted according to RFC4175. This means that they can be consumed by any device capable of understanding RFC4175 (which is also the basis for SMPTE ST2110-20).

The example below shows the mipmapping technology. Each source gets multiple down-scaled (not compressed) resolutions of the original. This is a real-time process that does not introduce noticeable delay.

vm_dmv mipmapping (example)



13.5 Input Sources

The list of input sources is a flat list which contains all input sources that the distributed multiviewer can consume.

Video sources can include:

- SMPTE ST2022-6
- SMPTE ST2110-20
- Compressed video via the SMART compressed input processor (CIP)
- Externally generated Mipmaps
- SDI inputs
- Multiviewer head outputs

Audio sources can include:

- Embedded audio from SMPTE ST2022-6
- SMPTE ST2110-30/-31
- Embedded audio from SDI inputs

Metadata sources can include:

- Embedded VANC data from SMPTE ST2022-6
- SMPTE ST2110-40
- Embedded VANC data from SDI inputs

13.6 Reference Design Switches

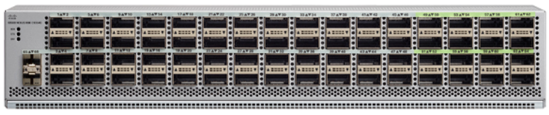
Arista 7280QR-C72-F

16 x 100GE + 56 x 40GE QSFP+

Cisco 9364C

64 x 100GE QSFP+ & 2 x SFP+

100/40 GE



Arista 7050CX3-32S

32 x 100GE QSFP+

100/50/40/25/10 GE

Cisco 9336C-FX2

32 x 100GE QSFP+

100/40 GE

14. Appendices

This chapter includes further information which you may find useful.

14.1 TCP/UDP Ports

The ports used in TCP/IP networking are divided into several categories. It is important to understand the numbering so as to avoid the ports registered with the IANA (the Internet Assigned Names and Numbers Authority).

This appendix provides a short overview of the port categories.

System or *Well-known* Ports

Ports 0-1023 are the so-called *system* or *well-known* ports.

They should only be used by important system services which are registered with the IANA. In many operating systems, using a port number below 1024 requires superuser privileges.

Some examples are:

- tcp/20-21 (FTP)
- tcp/22 (SSH)
- tcp/23 (Telnet)
- tcp/25 (SMTP)
- udp/53 (DNS)
- tcp/80 (HTTP)
- udp/80 (HTTP/3)
- tcp/110 (POP3)
- udp/123 (NTP)
- tcp/143 (IMAP)
- tcp/161-162 (SNMP)
- udp/319-320 (PTP)
- tcp/443 (HTTPS)
- udp/443 (HTTP/3-over-TLS)
- udp/514 (Syslog)

Registered Ports

Ports 1024–49151 are the so-called *registered* ports.

They should only be used for services that are registered with the IANA, or for protocols that have a stable, public protocol specification.

Unlike system ports, they do not require superuser privileges (hence why you often see web servers on tcp/8008 or tcp/8080).

Some examples are:

- tcp+udp/1194 (OpenVPN)
- tcp+udp/1293 (IPSec)
- udp/1701 (L2TP)
- tcp/1723 (PPTP)
- tcp+udp/1883 (MQTT)
- tcp/2375-2377 (Docker)
- tcp+udp/3389 (RDP)
- tcp/4222 (NATS)
- tcp+udp/5060 (SIP)
- udp/5353 (mDNS)
- tcp/5671-5672 (AMQP)
- tcp+udp/5900 (VNC)
- tcp+udp/5938 (TeamViewer)
- udp/6343 (sFlow)
- tcp/8008 and tcp/8080 (alternate for HTTP if 80 not available)
- tcp/9000 (EmBER)

Ephemeral, *Private* or *Dynamic* Ports

Ports 49152–65535 are the so-called *ephemeral*, *private*, or *dynamic* ports.

These are the ports that should be used for ad-hoc defined or private services, and for automatically allocated ports. These port numbers cannot be registered with the IANA.

One example used by Lawo's V__matrix DMV is the udp/49999. This port distributes MO labels, MO tally, and PPM metering data between the cluster nodes.