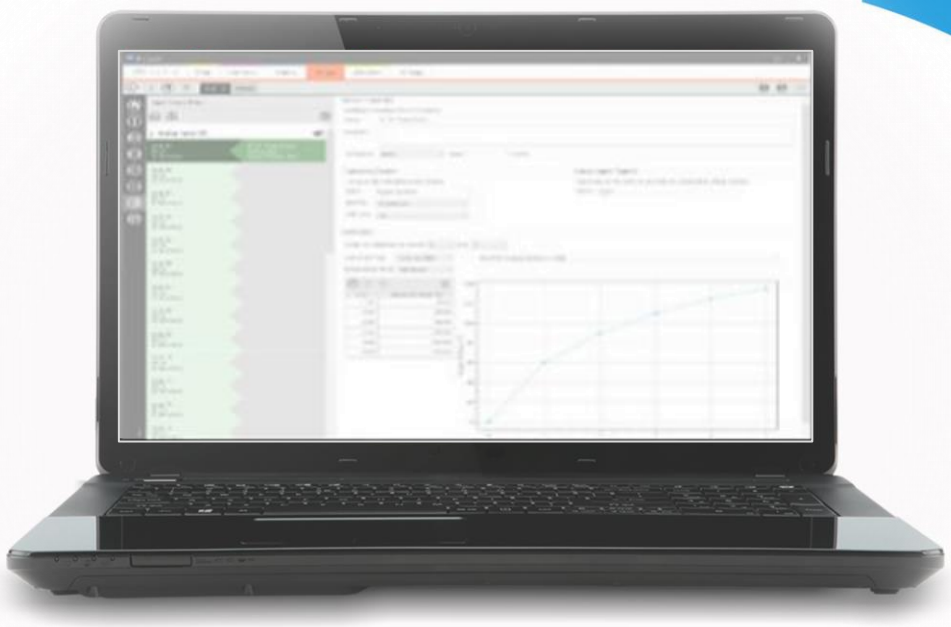
# Pi Toolset User Guide

Cosworth Electronics Limited
Brookfield Technology Centre
Twentypence Road
Cottenham  Cambridge
CB24 8PS
United Kingdom
Support: electronics.support@cosworth.com
Sales: sales@cosworth.com
Service returns@cosworth.com



COSWORTH
Enabling Greatness for over 65 years

# Contents

# 1. Installation

The latest versions of Toolset can be found on the Cosworth website (www.cosworth.com/motorsport/products/toolset/).

New firmware and included metadata require a minimum Toolset version. **Note**: Some championships are homologated to use specific versions of Toolset.

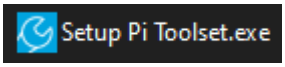
Before you install the latest version of Toolset make sure that you unplug any dongles, as the sentinel driver is updated during the process.

For versions of Toolset 10.X and above, the dongles are being deprecated. Cosworth are migrating to a software system which uses the encryption keys stored locally on the PC, rather than on the dongle.

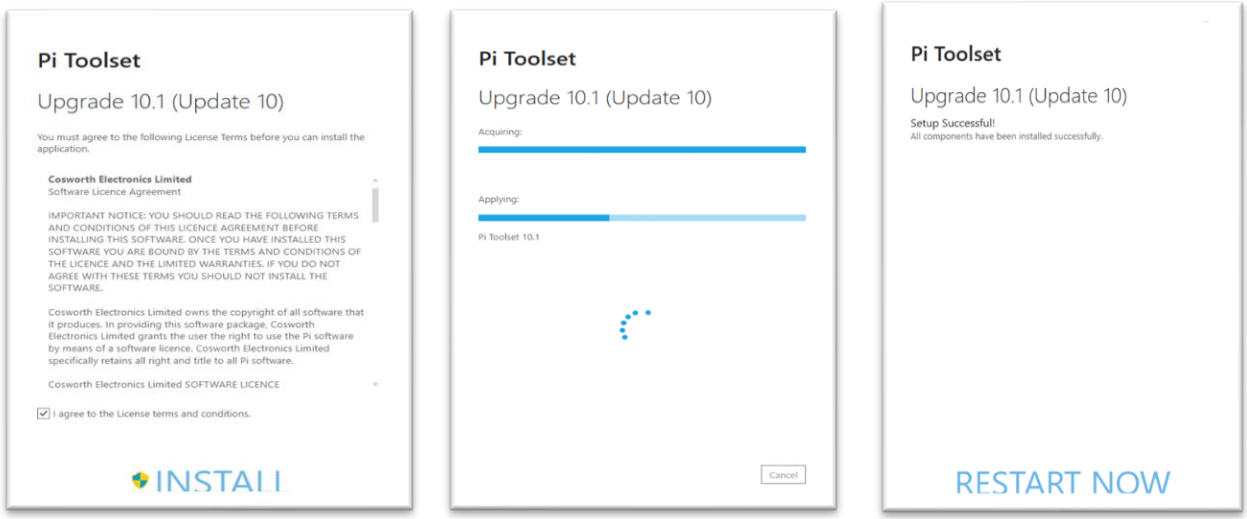
You can continue to use dongles, but make sure that the drivers are available.

You do not need to uninstall your existing Toolset version. Any new major version of Toolset is installed separately to the previous major version. Any minor updates overwrite the last version, with the same major version.

Double click the shortcut to the installation file.




When prompted, click **Next**, and then read the terms and conditions. To continue click **Agree**, and then click **Install**.




Because this a new installation, all devices seen previously by Toolset must be added to Toolset again. If the data offload file path and naming convection have been changed from the default settings, they need to be set up again.

### 1.1. Compatibilities

Although you can install different major Toolset versions alongside with minor versions (Toolset 7, 8 ,9, 10, 11 updates), these versions are installed over the top of existing installations.

Toolset setups do not support backwards compatibility between major versions of Toolset. For example, a setup created in Toolset 10 will not open in Toolset 9.

When you export or import channels, make sure that the same (or older) version of Toolset is used before importing the same file.

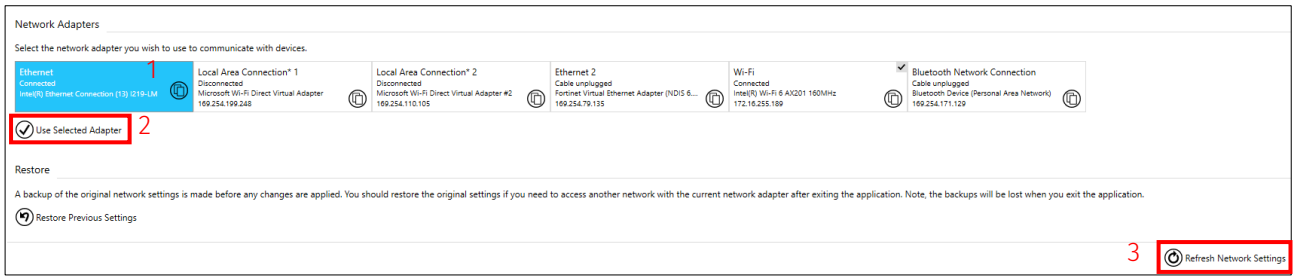
If a newer version of Toolset is used, then, if you import into an older major version, the *.tlf* file is greyed out and you cannot import it.

# 2. Get started and connect to your device

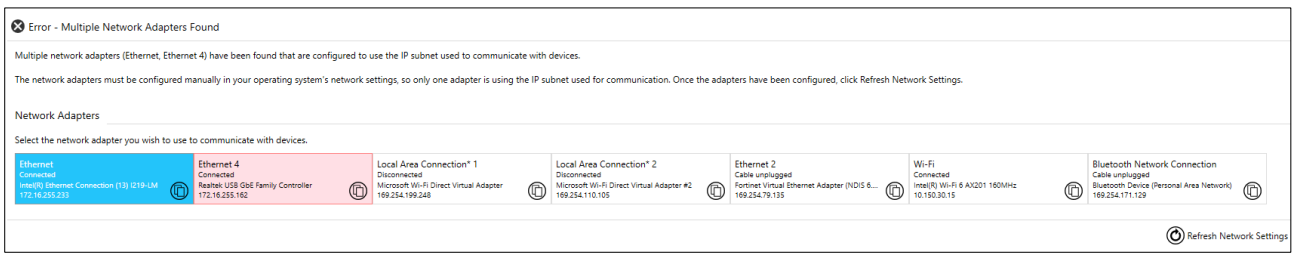
### 2.1. Configure the network (Ethernet) connection

Once you have installed Pi Toolset, you need to connect to the device using the Ethernet communication lead, and then set up the network connection on the **Settings** page.

On the **Network** page, click the required adapter, confirm that it is connected (1), and then select **Use Selected Adapter** (2). If the adapter is not immediately displayed, click **Refresh Network Settings** (3).



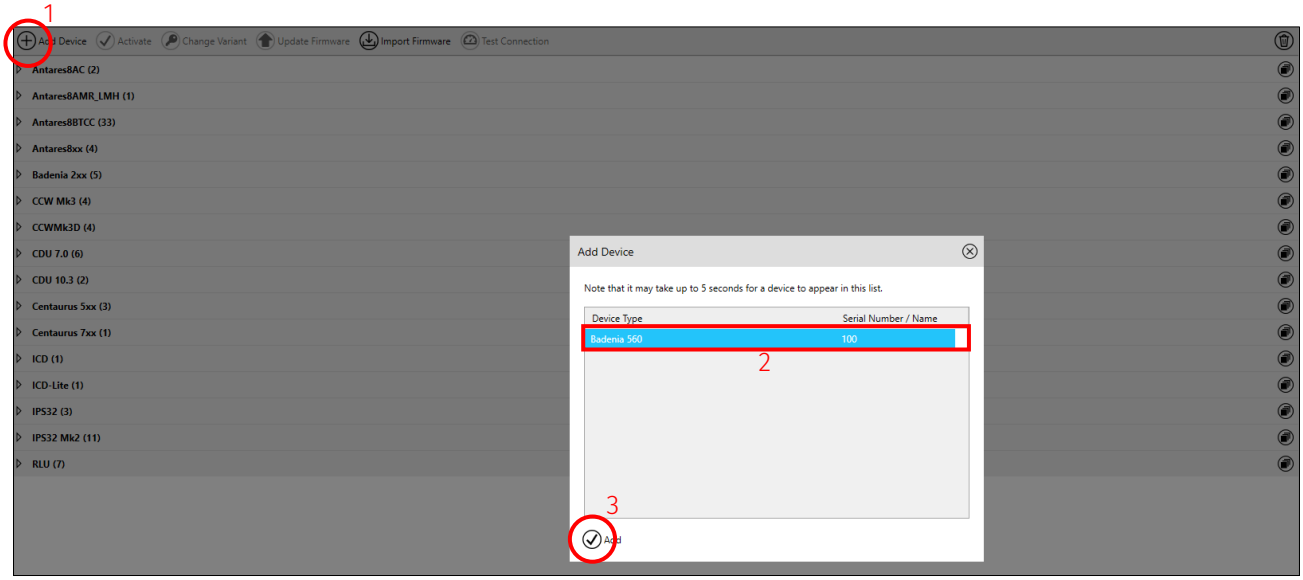

If there is a network error (for example, multiple network adapters are found), an error message is displayed. Make sure that the ethernet adapter is connected before you continue.

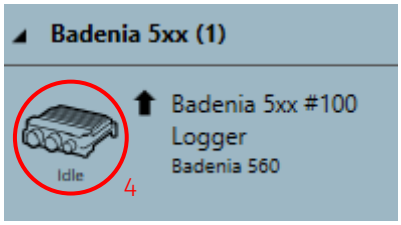



When network settings are configured, return to the **Devices** page.

## 2.2. Add a device

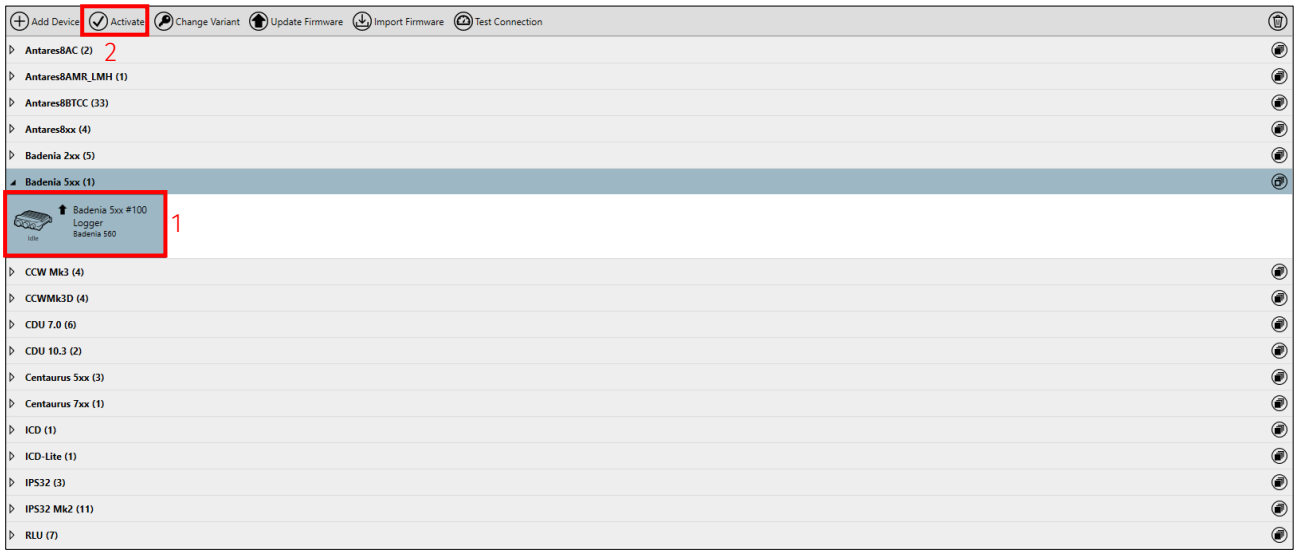
On the **Devices** page, use the **+** button to add a new device (1). The device is displayed in the menu. Select the device (2), and then click **Add** (3). Wait for the device to initialise and reach the 'Idle' state (4).




## 2.3. Activate a device

To read live data and send setups to the device, the device must be activated. Select the device (1), and then click **Activate** (2). To distinguish the active device if there are multiple known devices, the active device appears in a grey box.



# 3. Variants

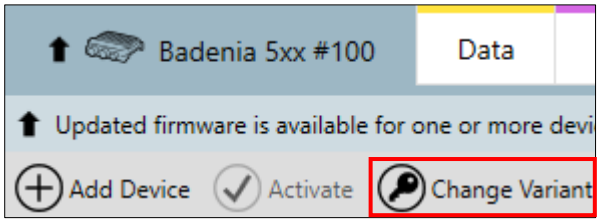
## 3.1. What is a variant?

Pi Toolset uses TCP communications over Ethernet to configure products (called 'devices'). This makes Toolset very versatile, but you must configure each device carefully before use.

To allow different specifications of device software, 'tokens' are used to enable or disable certain functions (such as analogue/digital inputs, CAN ports, Autocoding, and so on). Each specification of a device is called a 'variant'. The token forms part of the setup. The device variant and setup variant must match. When the setup is sent, the device variant is set.

## 3.2. Set the device variant

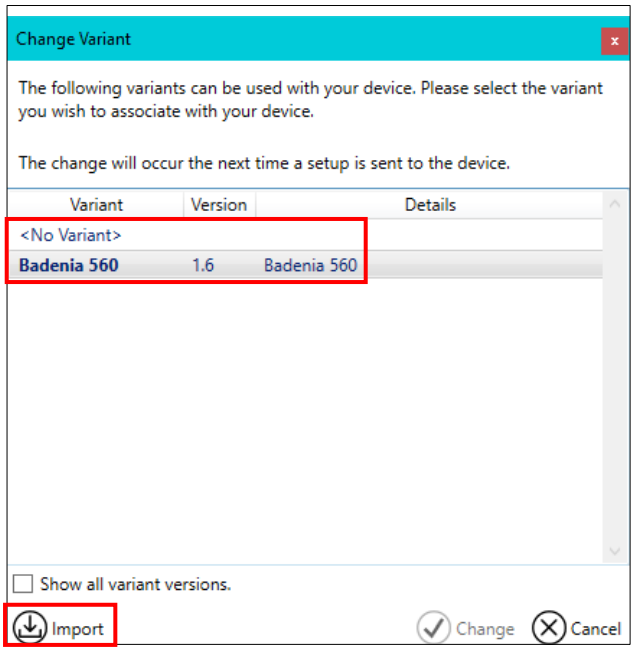
In the event of a device not having a variant, such as a new device, you must set the device variant before you send a setup. To set the device variant, activate the device, and then click Change Variant.




The Change Variant dialog box is displayed where you can control the tokens.

The dialog shows the available cached tokens. If there are no tokens displayed, you need to import a token file.

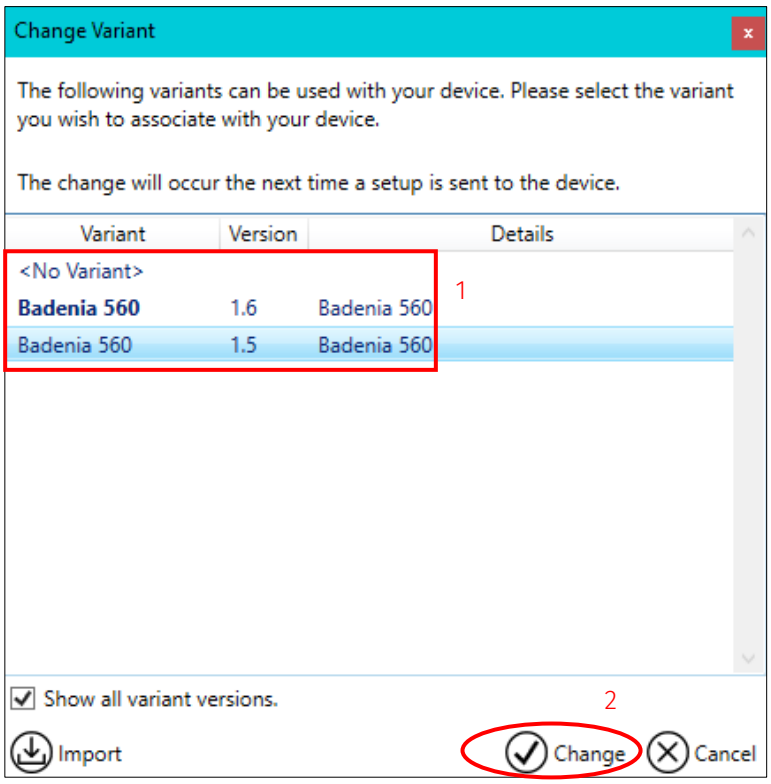
Click Import and navigate to where you downloaded and saved the token file. When the token is imported, it is stored and cached in Toolset, so you only need to do this once.



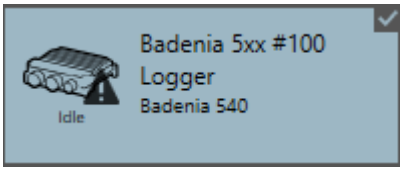
If you do not have a variant for the device, please contact: sales@Cosworth.com and make sure you state the serial number of the device, any developer requirements, and a specific version of variant if required.

The screenshot below shows a variant for a Badenia serial number 100.

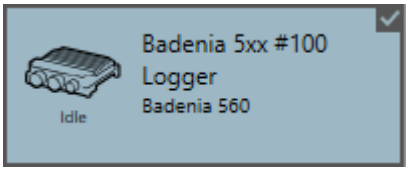

Tokens are required to control the corresponding Toolset options that match that specification of device. Select a token file, select the appropriate variant (1), and then click **Change** (2). You can also edit variants to make base level products if required.




This sets the device variant. The small warning triangle indicates that the setup containing the token file is yet to be sent to the device.




Once a valid setup containing the token file is sent to the device, the device icon is displayed without a warning triangle.



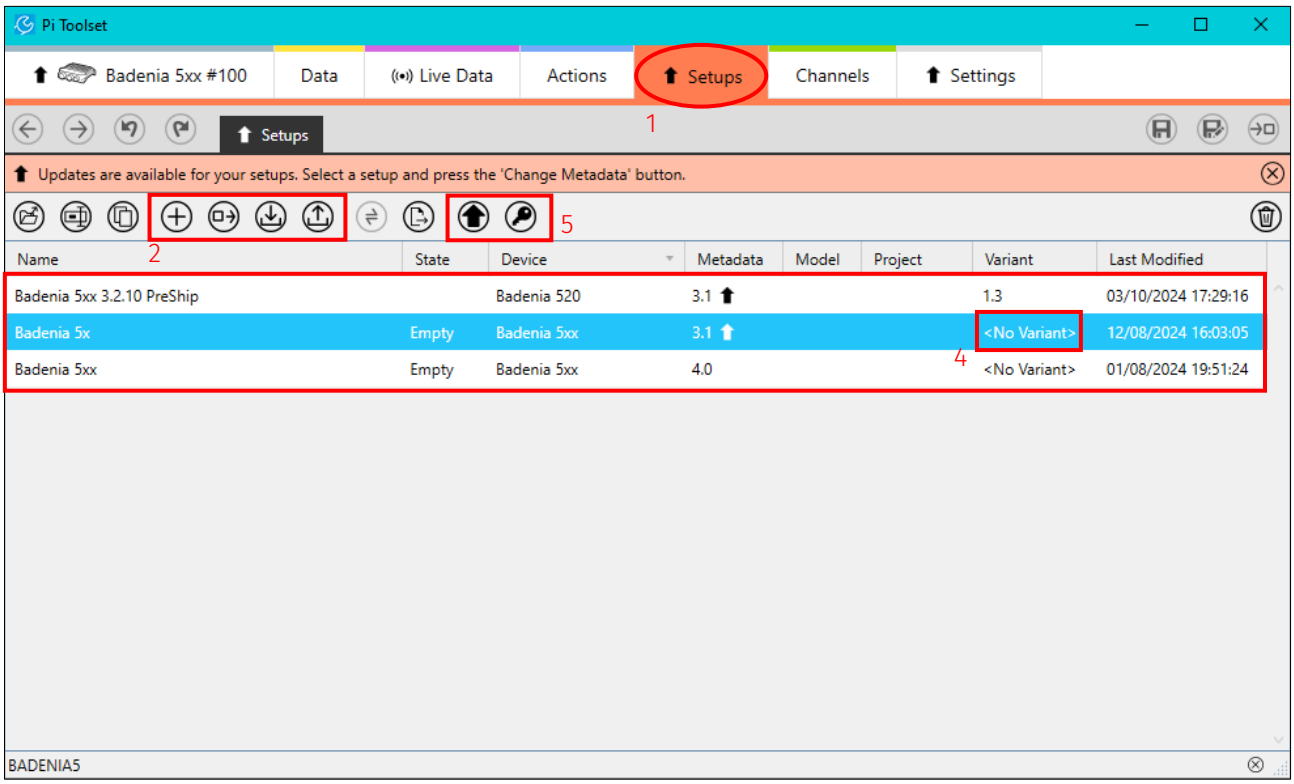
### 3.3. Set the device variant in Setups

To set the setup variant to match the device variant, click the **Setups** tab (1). All device setups are displayed, with name, device type, device variant, and a last modified timestamp.

If this list is empty you need to get a setup (2), either by importing the *.toolset* file supplied by Cosworth, or by reading it from the active device.

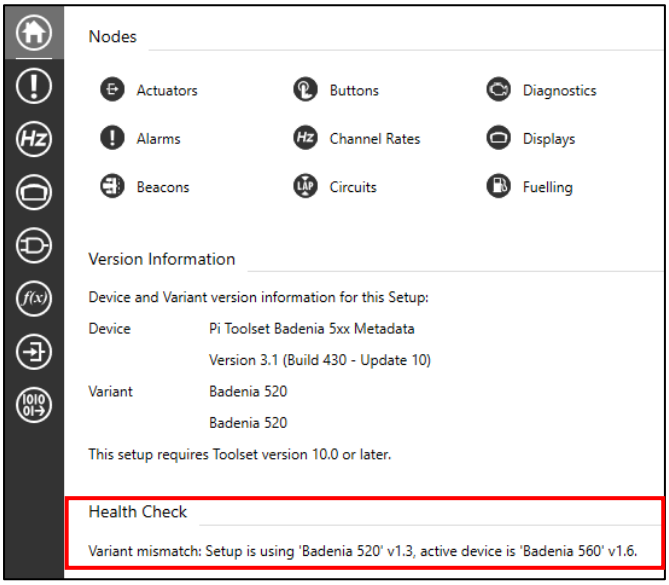
Once you have setups (3) to work with, it is important to make sure that the setup variant matches the device variant. In the screenshot below a setup that has no variant is shown (4). This might be an imported base level setup for that device. Therefore, it is important to set the variant to enable the purchased features.

On the **Setups** page you can set the setup variant via the **Change Variant** option (5) in a similar manner as changing the device variant. A similar dialog box to the device variant selection method is displayed.

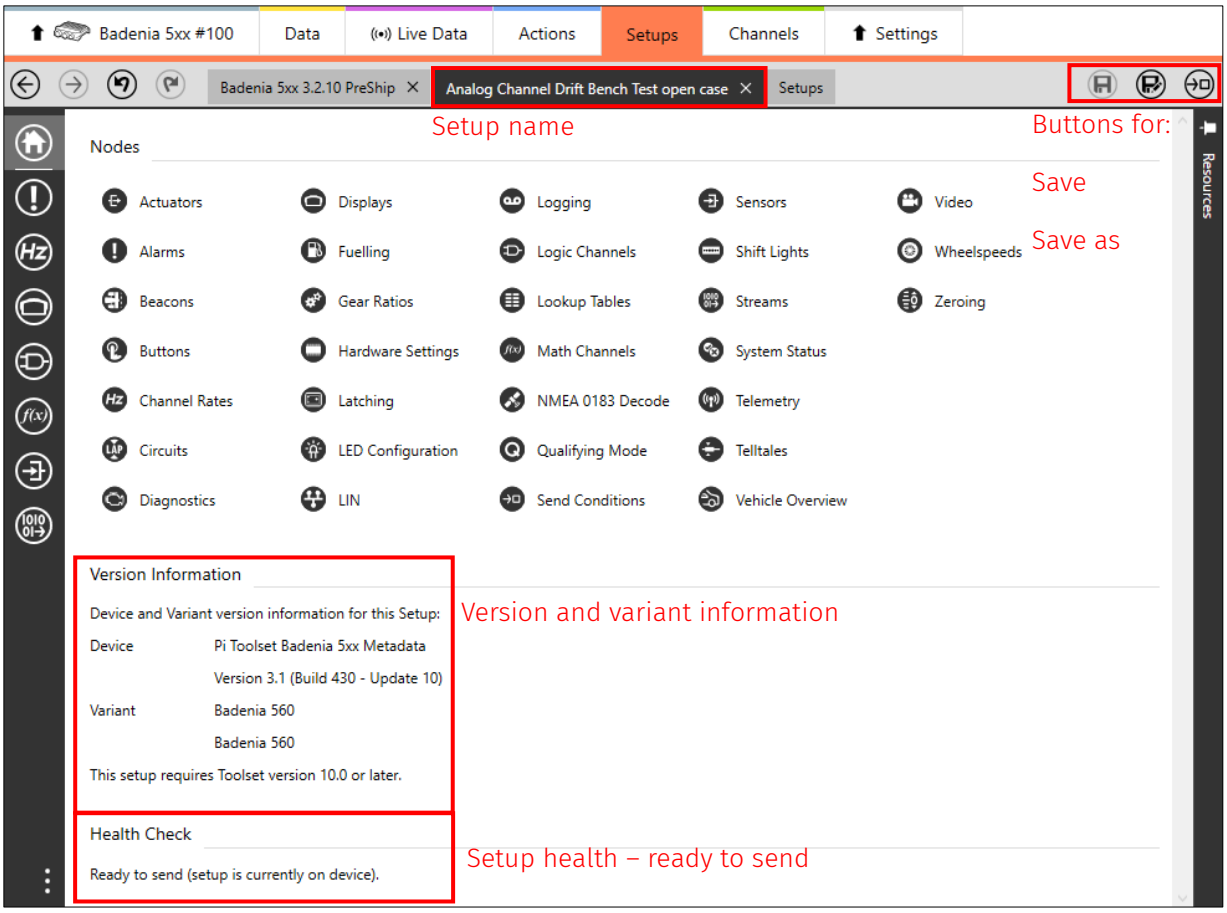



You can also open a setup. When you send the setup, a dialog box is displayed that prompts you to update the setup to match the new firmware on the device. Select **Use Device Variant**.

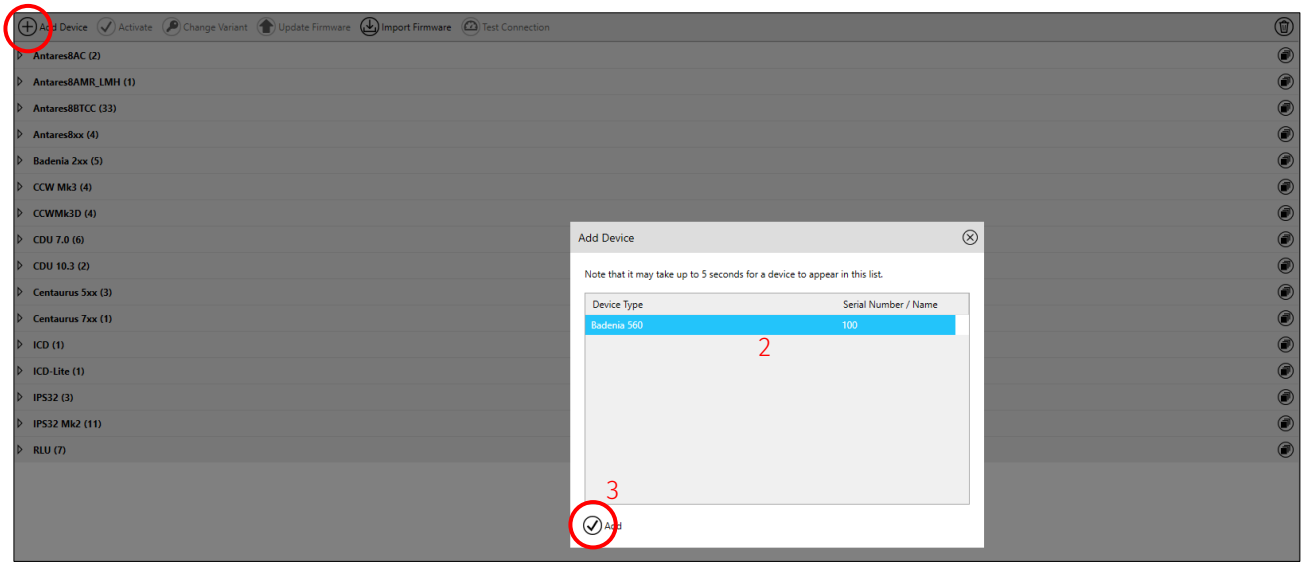
On the home page of the open setup, version information is displayed. If the device variant and setup variant do not match, a warning is displayed in the **Health Check** section.



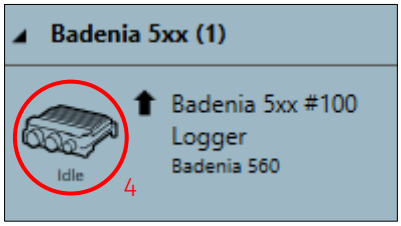

Once the error is corrected and the device variant and setup variant match, the following screen is displayed.



## 3.4. Add a device

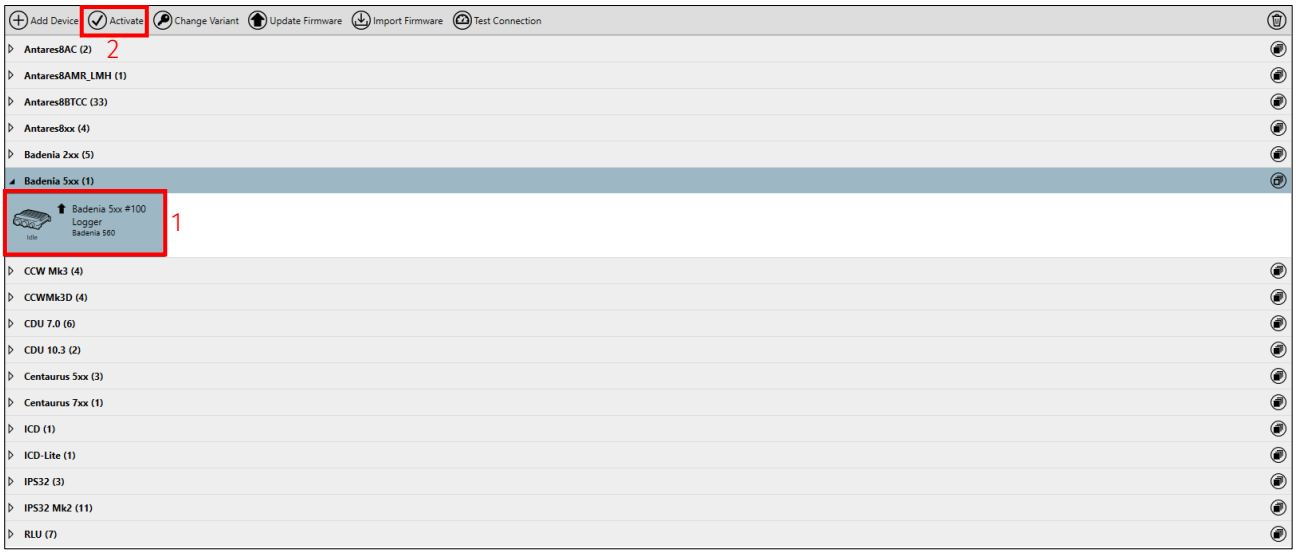



On the **Devices** page, use the **+** button to add a new device (1). The device is displayed in the menu. Select the device (2), and then click **Add** (3). Wait for the device to initialise and reach the 'Idle' state (4).




## 3.5. Activate a device

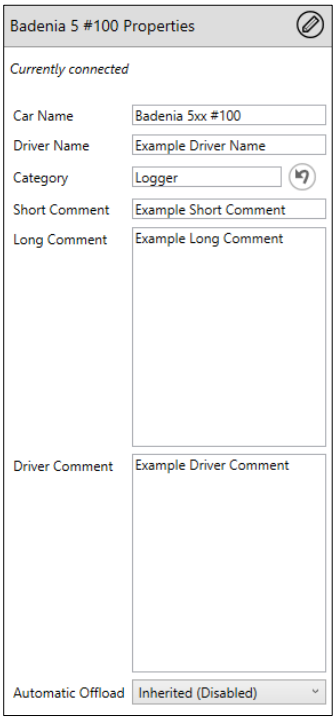
To read live data and send setups to the device, the device must be activated. Select the device (1), and then click **Activate** (2). To distinguish the active device if there are multiple known devices, the active device appears in a grey box.



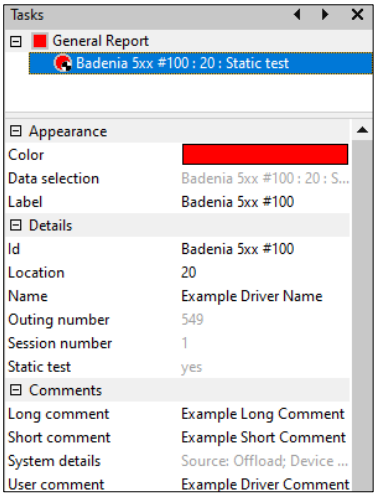
# 4. Device properties

You can view and configure the properties of a device on the **Devices** page. Select a device to display its properties on the right-hand side of the page.

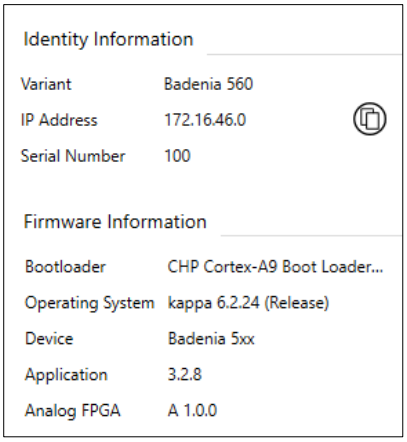
These device properties allow you to name the device by vehicle and/or drive, and apply comments about the device, session, or day, and so on.




These details are stored in the file metadata when the data is offloaded. You can view the data in Toolbox to help distinguish between data files.

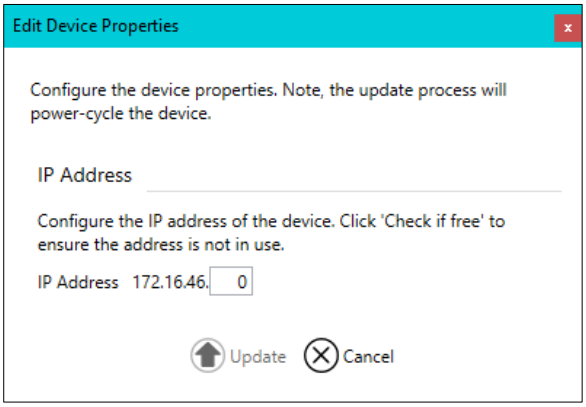


You can view device identity information, including the device type, IP address, serial number, and firmware details.




## 4.1. Configure the device IP address

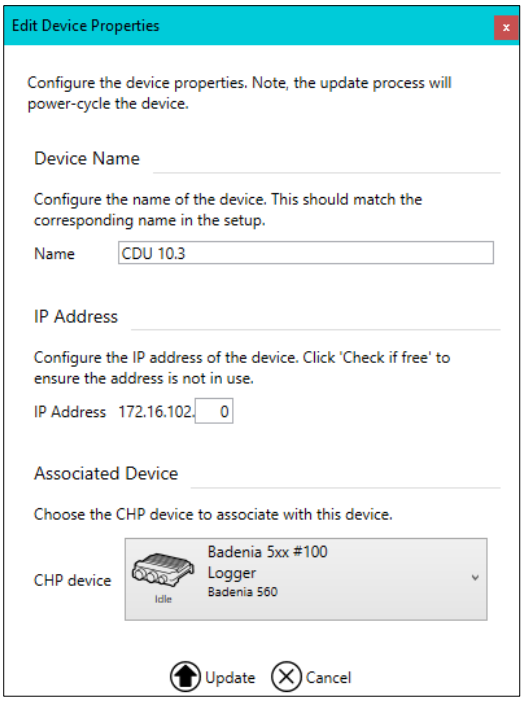
You can edit the suffix of the device IP address. Connect to the device, and then activate it. Click the pencil icon at the top-right to display the **Edit Device Properties** menu where you can edit the IP address suffix.




## 4.2. Associate auxiliary devices and displays

Devices such as the Remote Logging Unit (RLU) and Configurable Display Unit (CDU) are Ethernet devices and need to be associated with a primary device, such as an Antares, Badenia, or Centaurus.

To associate an auxiliary device, add the device to the devices page, select the device, and then click the pencil icon at the top-right to display the **Edit Device Properties** menu where you can name the auxiliary device and select the primary device with which to associate it.


Edit Device Properties
Configure the device properties. Note, the update process will power-cycle the device.
Device Name
Configure the name of the device. This should match the corresponding name in the setup.
Name
CDU 10.3
IP Address
Configure the IP address of the device. Click 'Check if free' to ensure the address is not in use.
IP Address
172.16.102.
0
Associated Device
Choose the CHP device to associate with this device.
CHP device
Idle
Badenia 5xx #100
Logger
Badenia 560
Update
Cancel

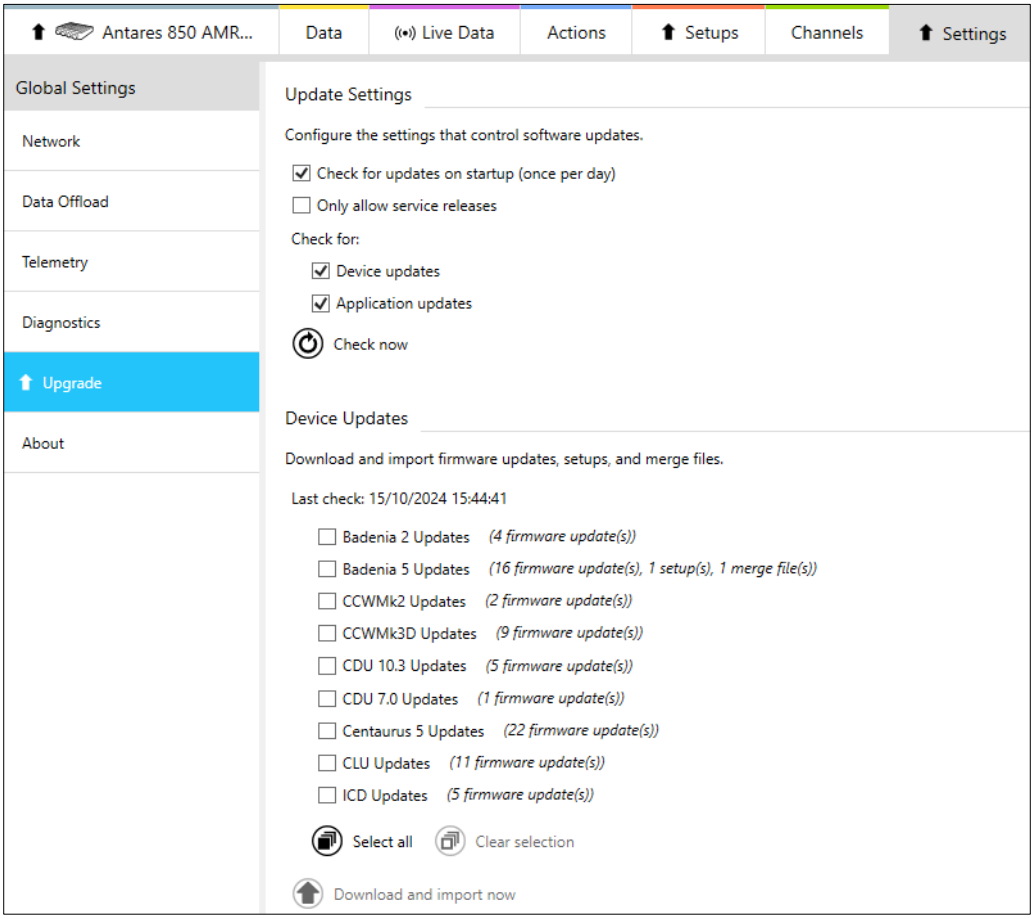
# 5. Device firmware

Update device firmware to make sure that all new features and functionality of your hardware are up-to-date.

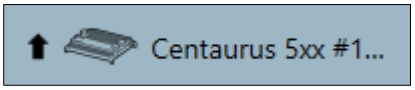
New firmware can require a minimum version of Toolset. You can find the latest versions of Toolset on the Cosworth website (https://www.cosworth.com/motorsport/products/toolset/).

## 5.1. Download firmware

You can download the latest device firmware from the **Upgrade** tab of the **Settings** page when a PC is connected to the internet.



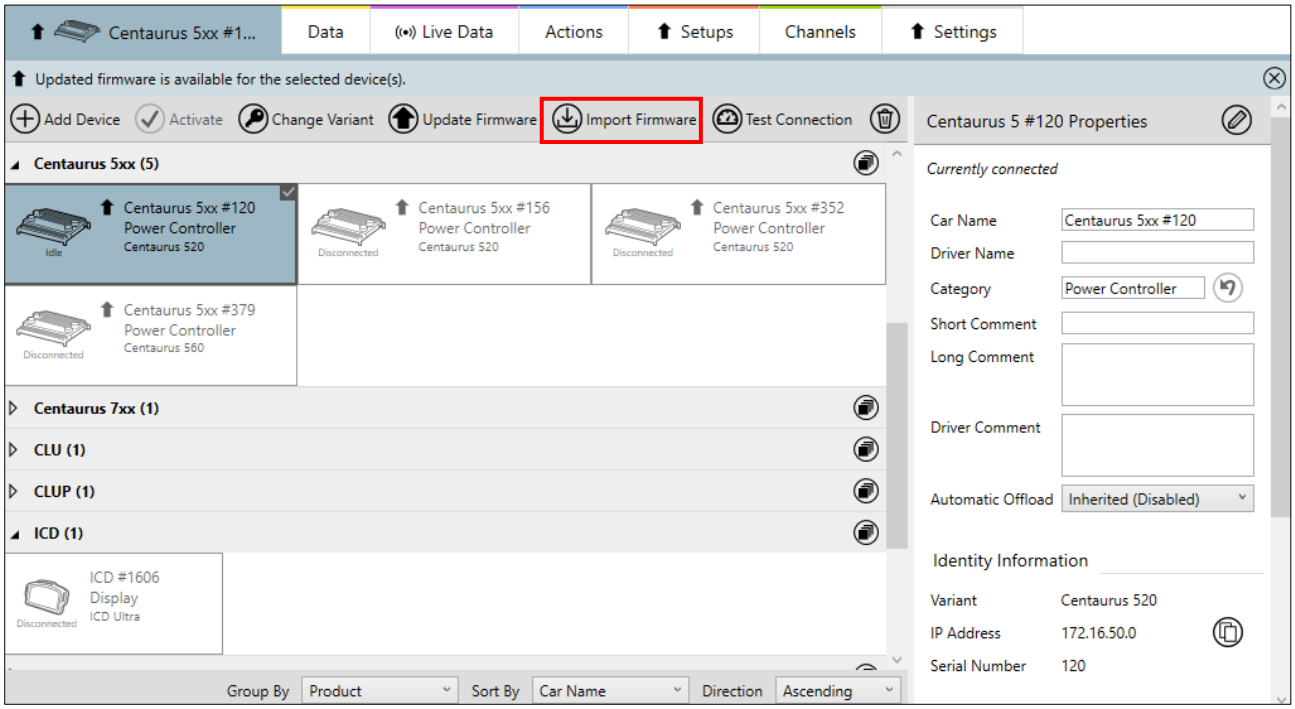

An upgrade prompt (up arrow) is displayed next to the device on the **Devices** page when new device firmware is available.



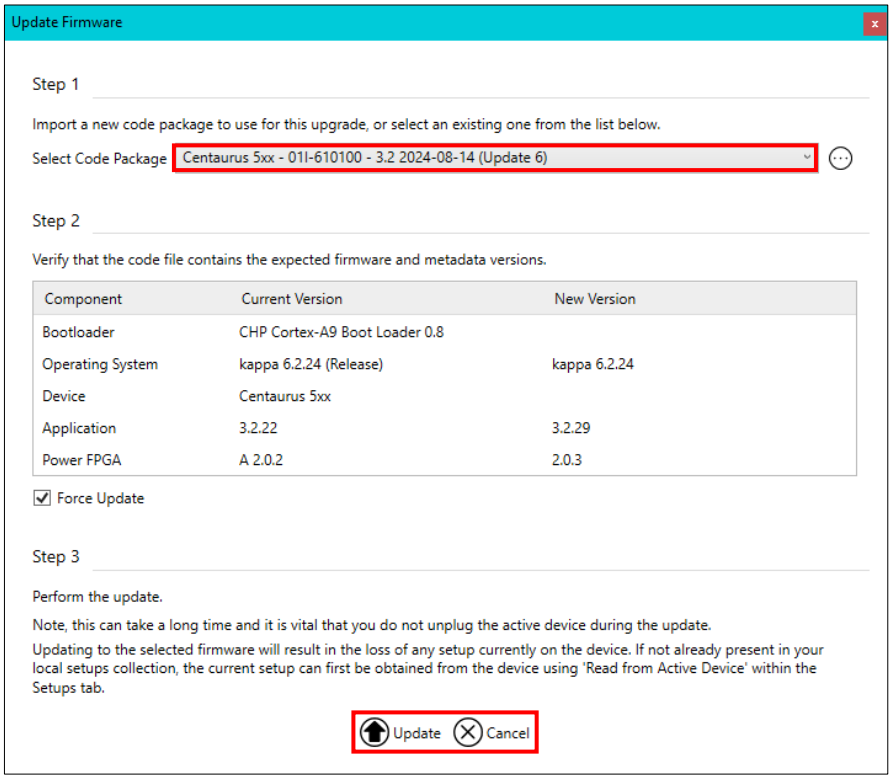

## 5.2. Update firmware

Download the latest firmware version, and then connect to, and activate, the device. Make sure that the device is in the 'Idle' state and active before you update the firmware.

**Note:** Updating firmware removes the setup from the device, together with any stored NV RAM variables (for example QM mode). Make sure that a copy of the device setup is saved, or **Read from Active Device** on the **Setups** page is selected, before you update the firmware. This saves the current setup stored on the device in your list of setups.



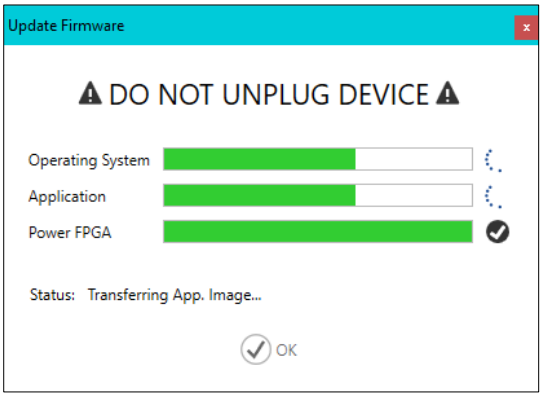

The following window shows the current firmware on the device.




**Step 1.** Browse to the location of the *.code* package required for the update.

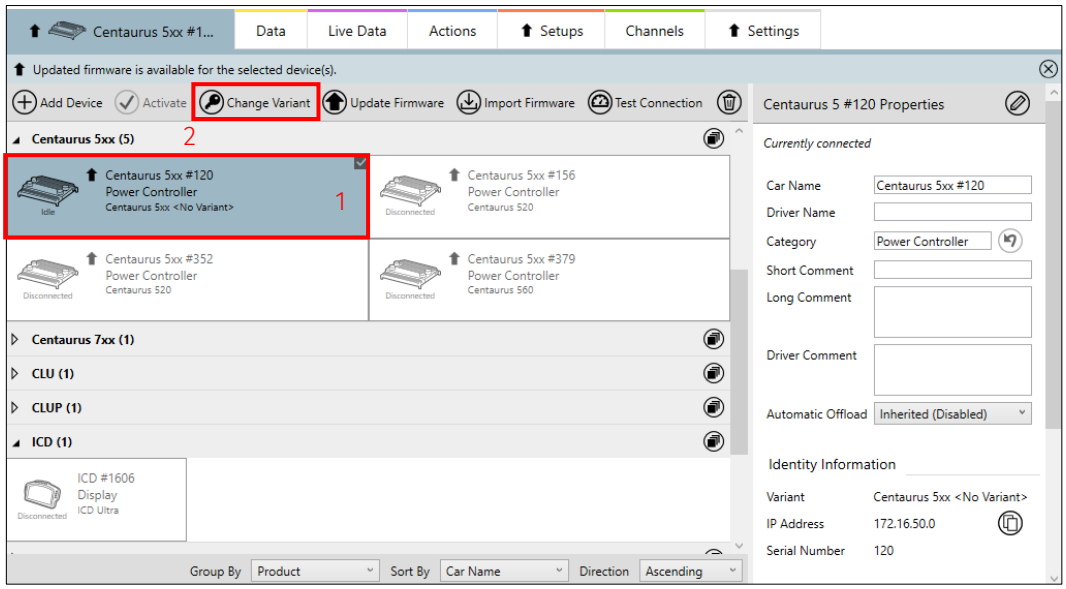
**Step 2**. Select the package. Toolset shows the differences between the two code packages.

**Step 3.** Confirm, and then click **Update** to begin installation of the new package. The **Update Firmware** window shows progress of the installation. When installation is complete, click **OK**.

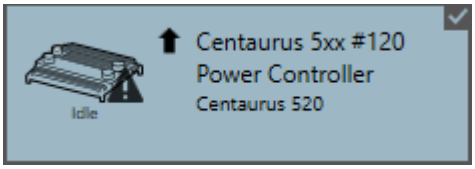



After the update, the device shows as a blank device in Toolset, indicated by the '<No Variant>' text in the device information box (1).

You now need to send the token and setup to the device. Click **Change Variant** (2).

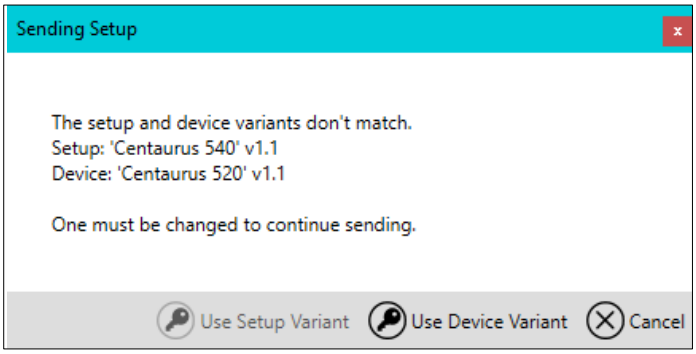



When prompted, select the required variant token, and then click **Change**. When the token is ready to be sent to the device, an exclamation mark in a warning triangle is displayed next to the device.

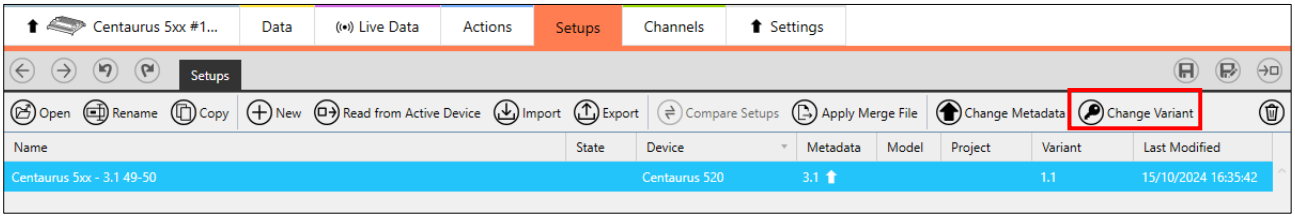



On the **Setups** page, open the setup, and then click **Send** to transfer the setup to the device.

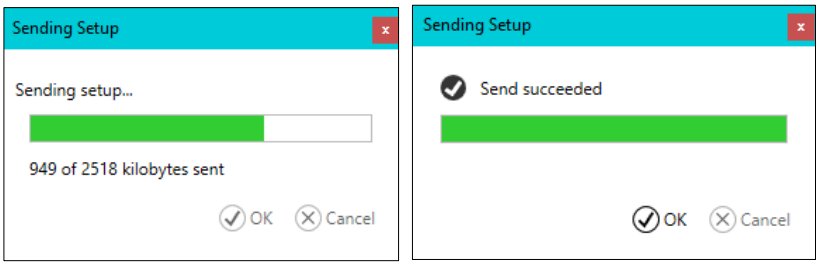
A dialog box is displayed to prompt you to update the setup to match the new firmware on the device. Select **Use Device Variant**.




You can also change the setup variant on the main **Setups** page via the **Change Variant** option to avoid the 'Use Device Variant' prompt.




When the setup is complete, the token is uploaded to the unit.




After an update, the device is displayed in an 'Idle' state on the **Devices** tab. The device firmware information is updated in **Device Properties**.

# 6. Data

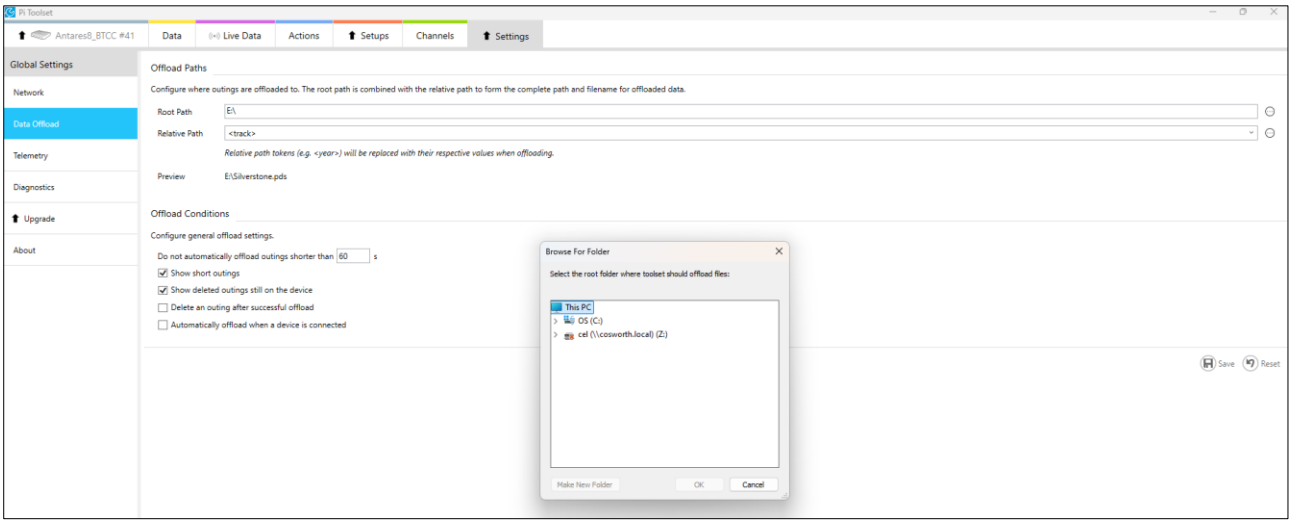
The **Data** page allows you to offload all logged data from the device.

## 6.1. Set up data offload paths

Before logged data is offloaded, you need to set up the offload paths on the **Settings** page.

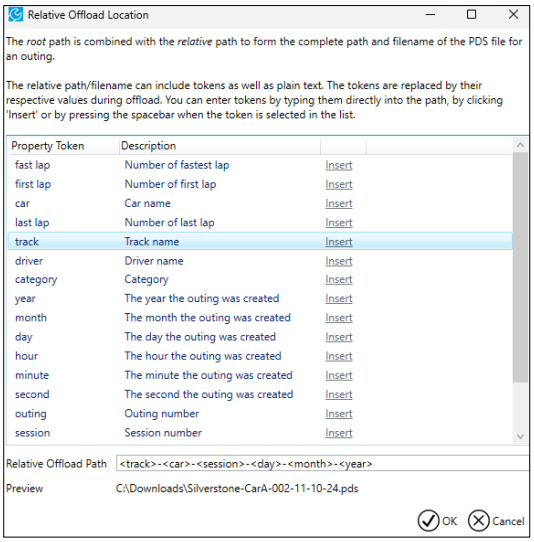
## 6.2. Set the offload path

On the **Settings** page, the data offload 'Root Path' is the file storage location on your device, or on an external device. Select the three dots (...) icon to browse and select the storage folder. Or you can copy and paste the file path from Windows File Explorer into the 'Root Path'.



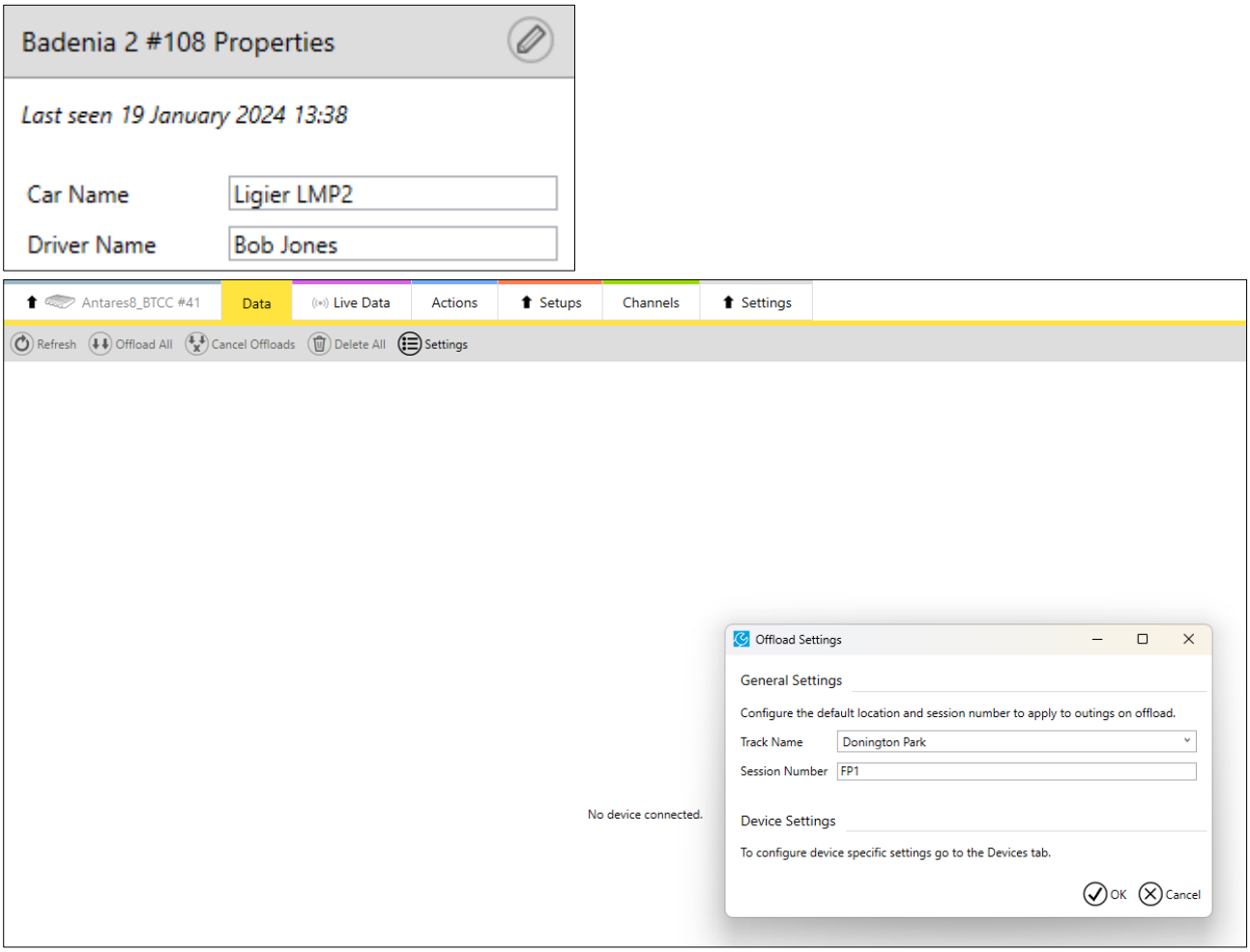

## 6.3. Set the relative path

The relative path is the naming convention used for the offloaded data. This identifies data by offloading files in a specific format or name.



Click the three dots next to ‘Relative Path’ to set the naming convention to include track name, driver name, car number, session number, day, month, year, and so on.

For example, the track name and session name/number are sourced from the settings on the **Data** page or the outing properties after you select an outing when connected to the device.

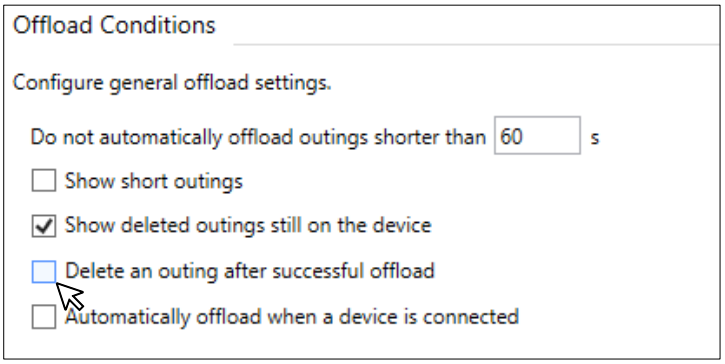



The car and driver name are sourced from the device properties on the **Devices** page. Other parameters such as lap number, date, and time are sourced from the logged data.

**Note:** Make sure that you save data offload settings using the **Save** option before you offload data.

## 6.4. Set offload conditions

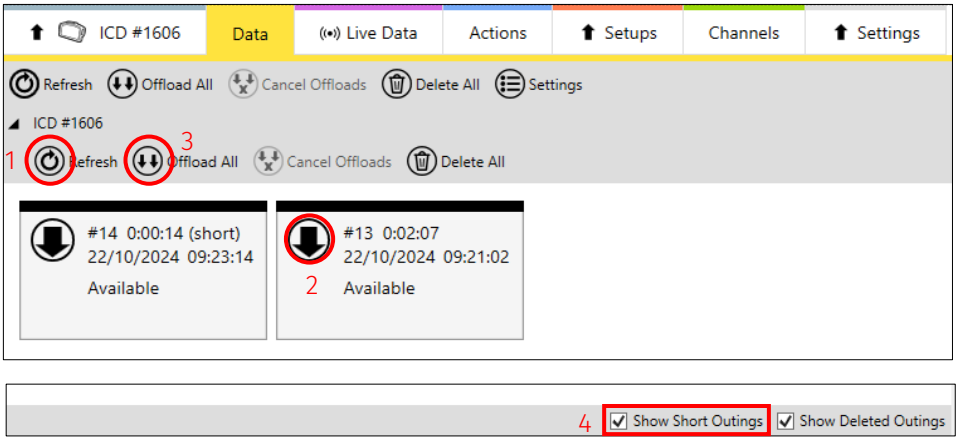
Offload conditions can be checked to automatically, hide, download, and delete outings.
For example, you can configure Toolset to automatically offload data when the device is connected, and to delete the outing after a successful download.



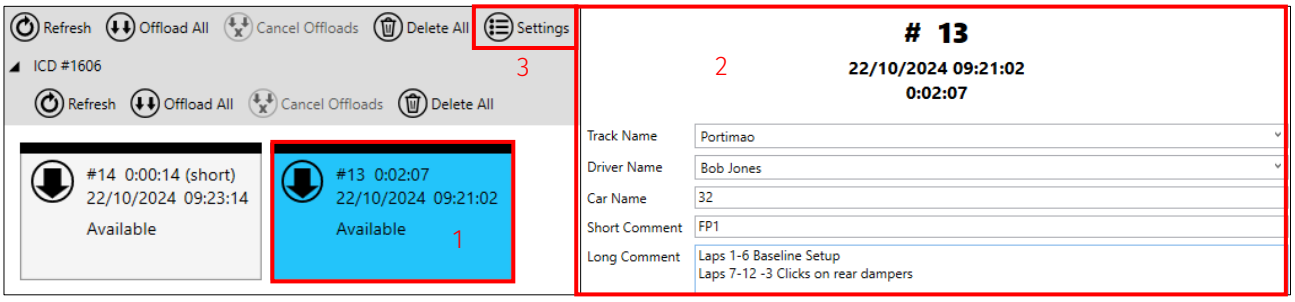

## 6.5. Offload logged data

Logged data (also known as outings) is displayed on the **Data** page. Outings from each device are shown under the name of the device. If outings are not displayed, click **Refresh** (1). To offload individual outings, click **Offload** (2) on each outing file. You can also offload all outings via the **Offload All** option (3).

**Note**: If no outings are displayed when you log short outings, check that the **Show Short Outings** option is selected (4). You can configure the duration of a short outing on the **Settings** page.

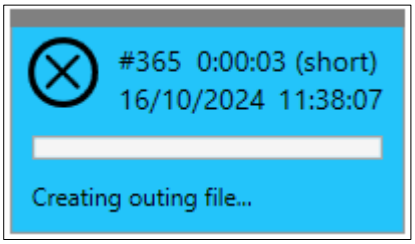



Click on an outing file (1) to display its properties on the right-hand side of the window. These include outing number, date and time, track name, driver name, car name, and comments, all of which you can edit (2). These details are stored in the outing file for reference.

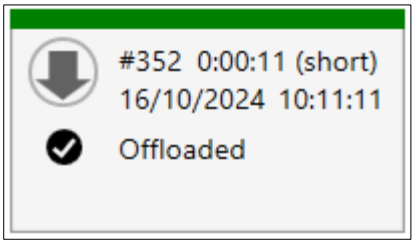


To offload multiple pieces of data with the same properties, select **Settings** (3), and then configure the relevant properties. These properties are applied to all selected offloads.

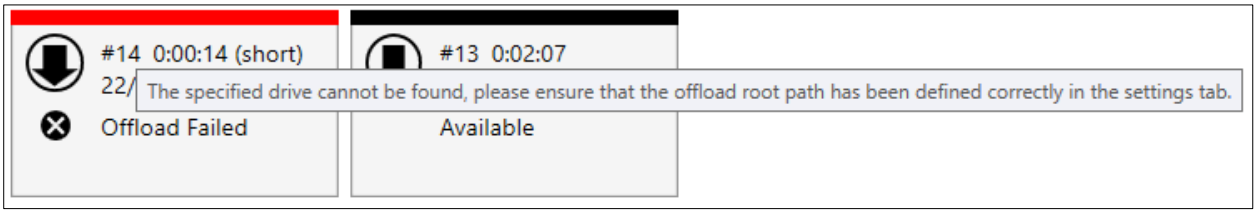
Click **Offload** to start the download from the device to a specified location. You can select multiple outings to queue for delivery. The progress and status of the download is displayed.




When successfully downloaded, the outing is marked as 'Offloaded' and highlighted in green.

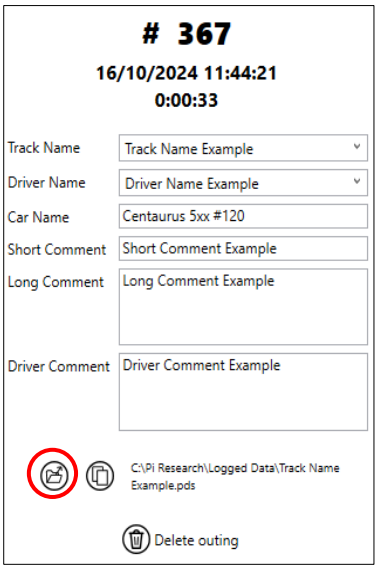



If an outing offload fails, the outing is marked as 'Offload Failed' and highlighted in red. Hover over the outing to display the cause of the failure.




If errors continue, close Toolset, and then navigate to: *C:\Pi Research\Toolset\[Toolset Version]\Logs\Pi Toolset\Log*. Open the log file and check for specific device errors. If you cannot identify the errors, then please send a message to: electronics.support@cosworth.com.

From the **Outing Properties** window, you can open the outing directly in Toolbox if the application is open.



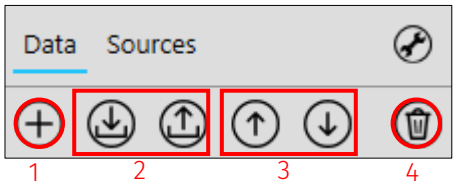
# 7. Live data

When connected to the device, you can view channel data in real time on the **Live Data** page.

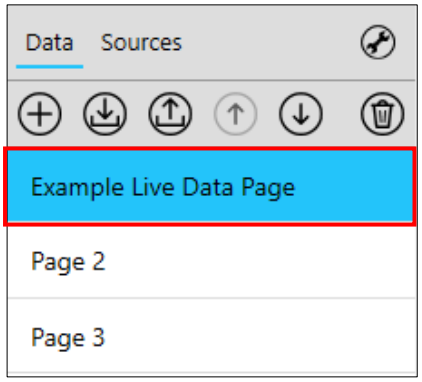
## 7.1. Live data pages

### Add and name live data pages

You use the **+** tool in the **Data** menu (1) to add live data pages. You can use the import and export tools (2) to import and export live data pages between Toolset versions and PCs. If you add multiple live data page you can use the reorder buttons to reorder items in the menu (3). Use the 'bin' tool to delete live data pages (4).

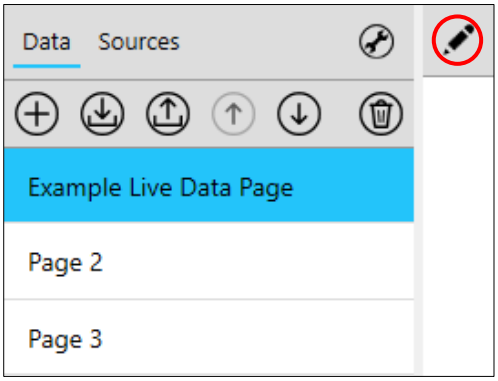



Click the name text box to rename a live data page.




### Configure live data pages

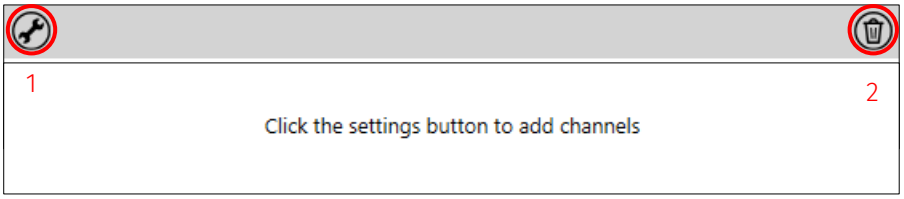
To configure live data pages, click the 'pencil' icon to enable 'edit mode'.



**Live data display types**

In edit mode you can add four display types:

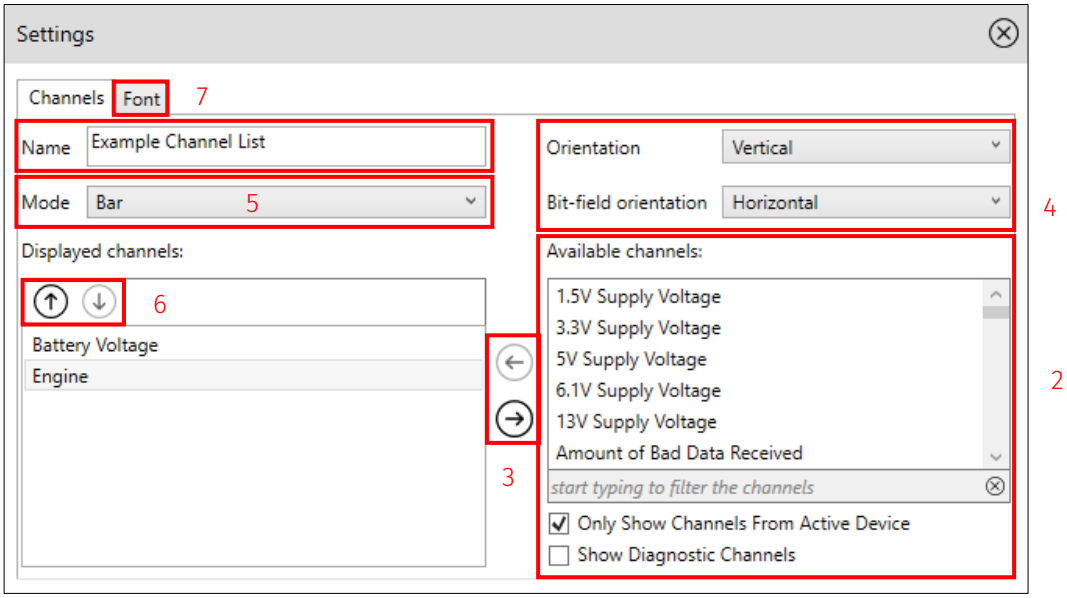
- Channel list – Used to display channel values as raw values or bar graphs
- Map control – Used to display a map
- Tabular outing report – Used to display telltales (see Telltales)
- Chart recorder – Used to display channel values as line graphs



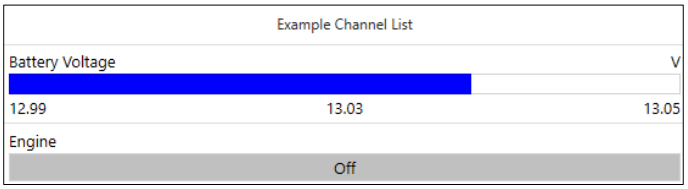

Channel List

Once you add a channel list, use the 'wrench' tool to open the display settings to add channels to the display (1). Use the 'bin' tool to delete the display (2)

From the display setting menu you can define the name of the display (1). A list of the available channels is displayed and can be searched (2). Double-click on a channel to add or remove it from the display or use the insert and remove tools (3). You can select the display mode (bar or value) from the dropdown menu (4). Select the orientation of standard channels and bit-field channels (horizontal or vertical) from the dropdown menus (5). Use reorder tools (6) to change the order of channels in the display You can select the display font, size, and colour from the Font menu (7).

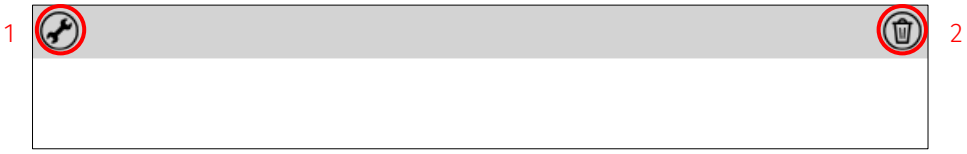


Close the display settings menu and exit edit mode to view the list of configured channels. Enumerated bit-field channels are displayed according to their enumerated statuses.

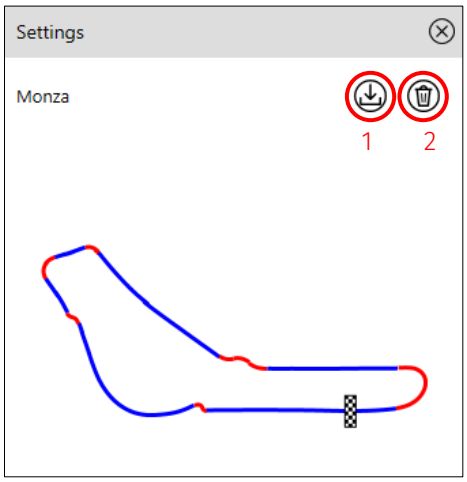



## Map control

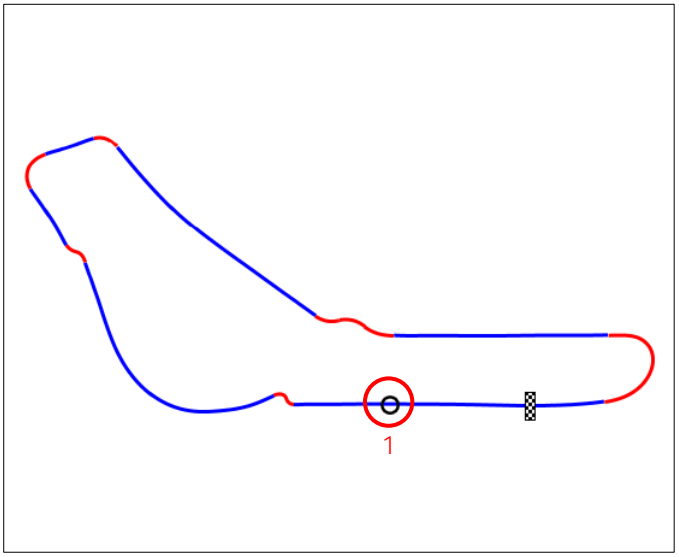
Once a map control display has been added, use the 'wrench' tool to open the display settings to add channels to the display (1). Use the 'bin' tool to delete the display (2).




When the display setting menu is open you can import a track map (*.pxt* file format) (1). You can use the 'bin' tool to delete a track map (2).

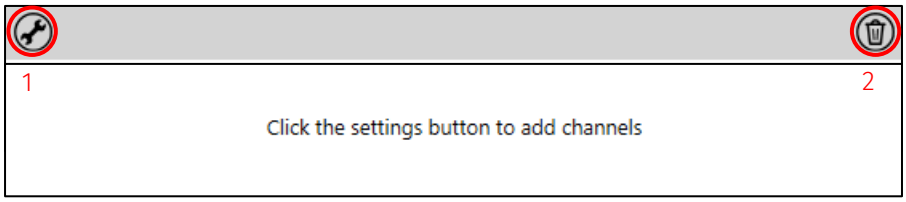


Close the display settings menu and exit edit mode to view the Map Control display. When connected to the device via telemetry the vehicle position is displayed on the map (1).

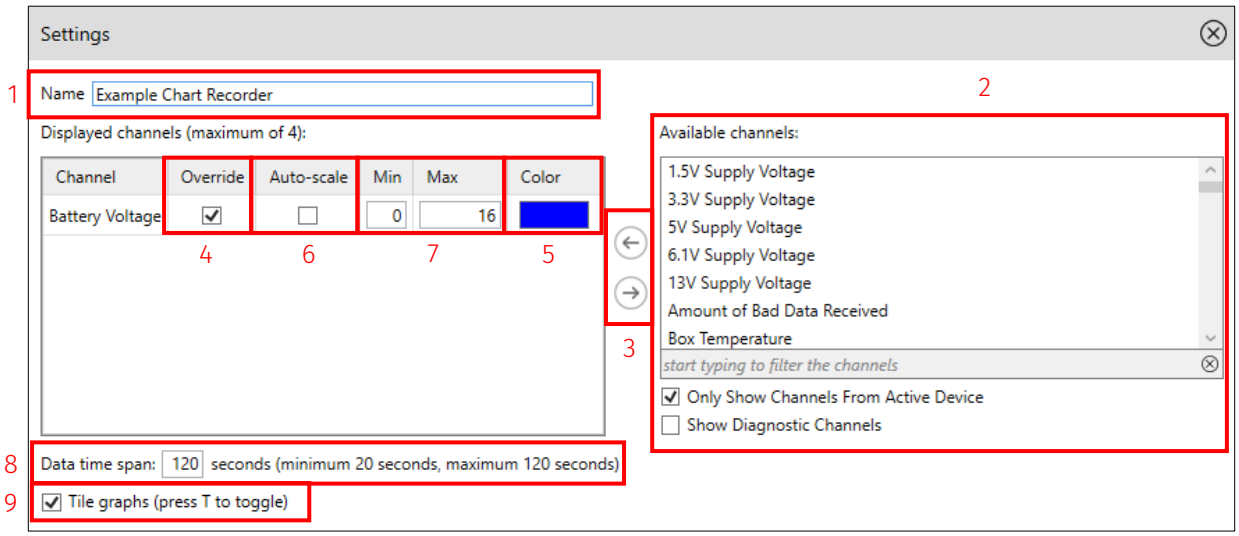



## Chart recorder

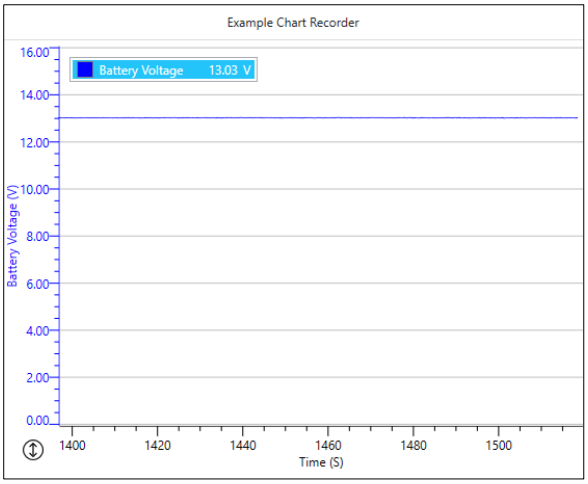
Once a chart recorder display has been added, use the 'wrench' tool to open the display settings to add channels to the display (1). Use the 'bin' tool to delete the display (2)




When the display setting menu is open, you can define the name of the display (1). A list of the available channels is displayed, and you can search specific channels (2). Double-click a channel, or use the insert and remove tools, to add or remove a channel (3). You can change the default display colour (4) and select another colour from the **Color** menu (5). You can auto-scale the display Y axis (6), specify the Y axis minimum and maximum (7), the X axis time span (8), and whether the channels are overlaid or tiled (9).



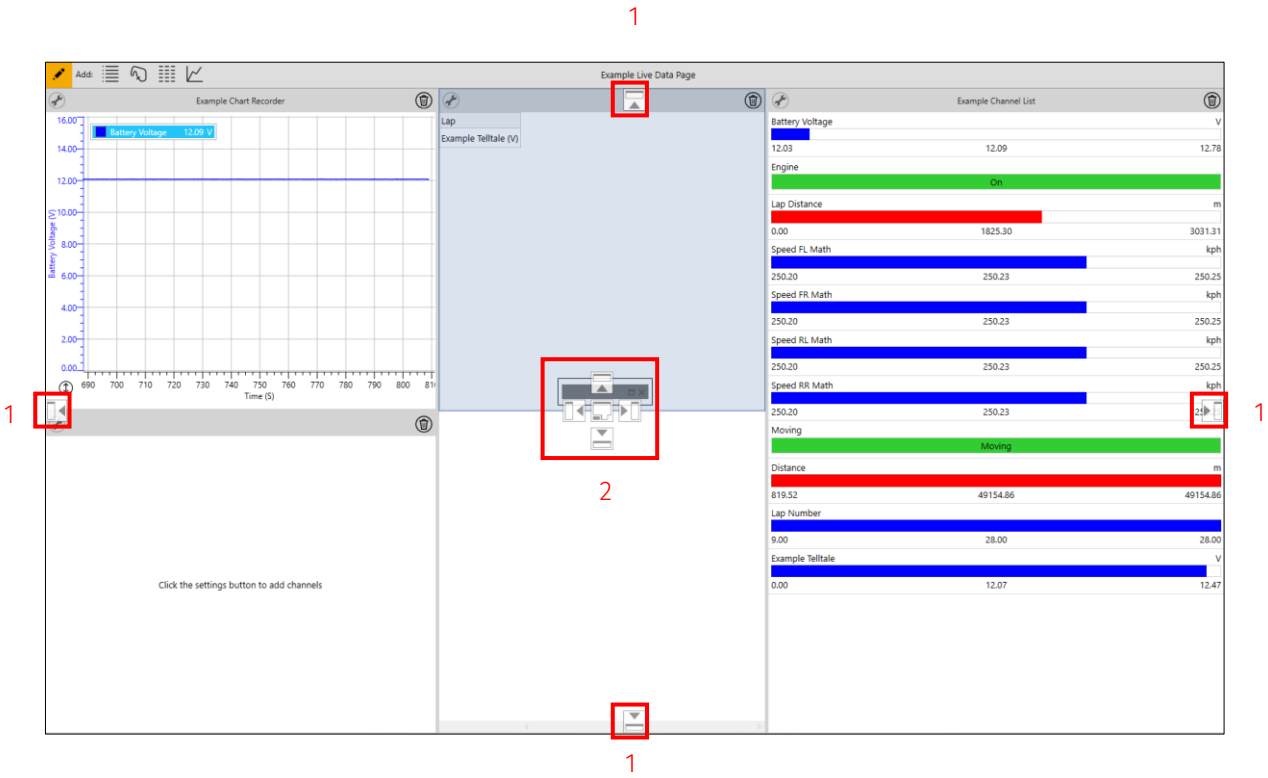
Close the display settings menu and exit edit mode to view the configured chart recorder display.



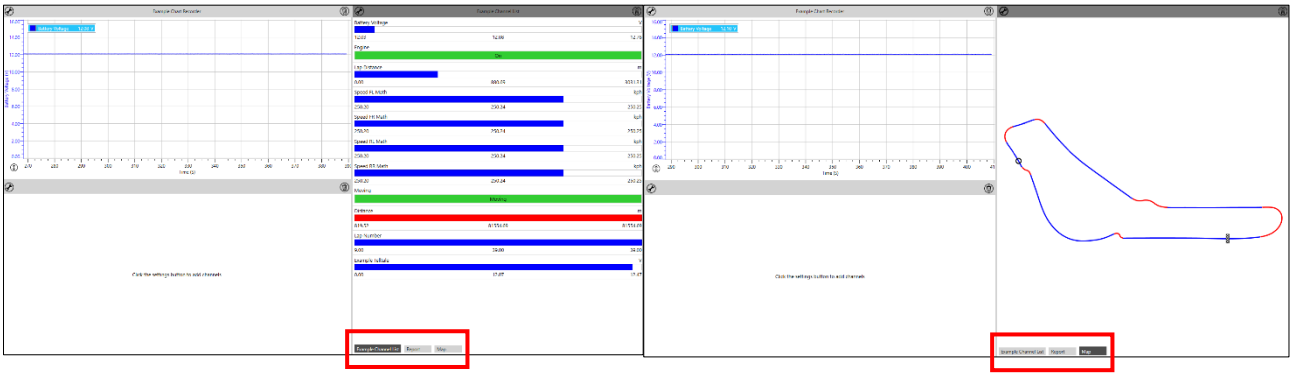

### Arrange live data pages

You can rearrange live data displays. In edit mode you can adjust the display width and height. Drag the edge of the displays to the required width and height (1).

To reposition live data displays, click the top of a display and drag it to one of the available positions (2).



You can also lay displays on top of each other to occupy the same area. The display for the area can be chosen by the user by selecting the display from the list at the bottom.



## 7.2. Sources

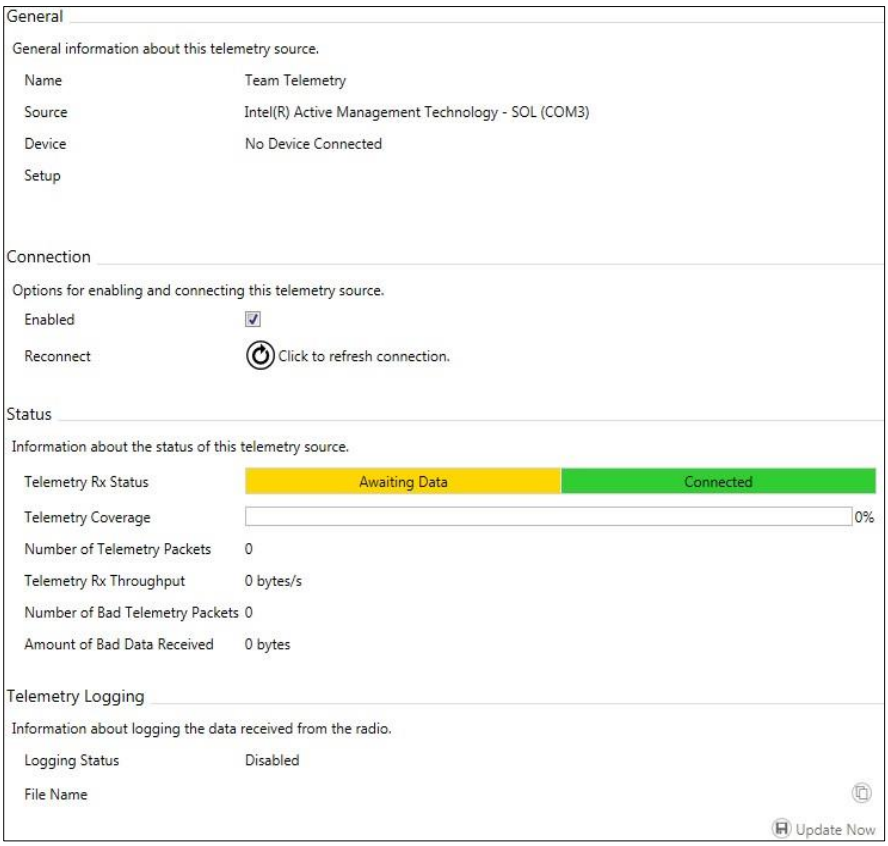
The **Sources** menu displays the status of the telemetry network connection and the number of clients on the telemetry network. You can check the state of the telemetry link and view the target directory for the logged telemetry data.

The **General** section shows the name of the source, the device, and the setup.

The **Connection** section shows if your source is enabled, together with a **Click to refresh connection** option.

Under the **Status** section you can see the status of your telemetry connection together with information about your connection.

Under the **Telemetry Logging** section, you can see the status and destination of the logged data.



# 8. Change lap numbers

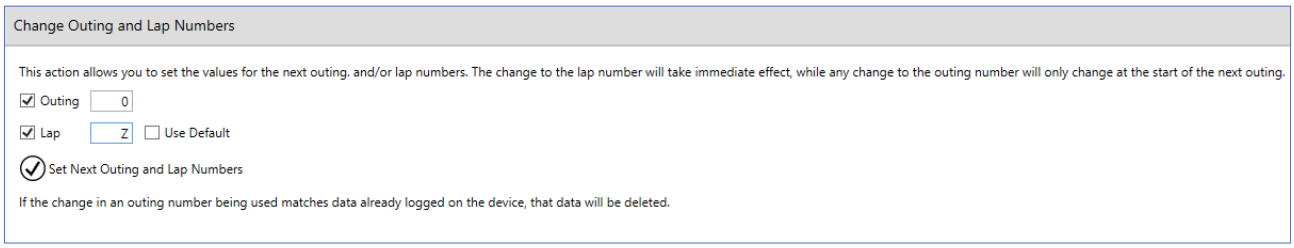
You can reset outing and lap numbers on the **Actions** tab when connected to a device.

## 8.1. Actions tab

The **Lap** node on the **Actions** tab allows you to reset the lap number and the outing number on the device. Select the **Outing** and **Lap** check boxes, and then enter the required outing and lap numbers in the text boxes.

Click the **Set Next Outing and Lap Numbers** button to set the target value. The newly defined outing and lap numbers are included in the next logged outing.

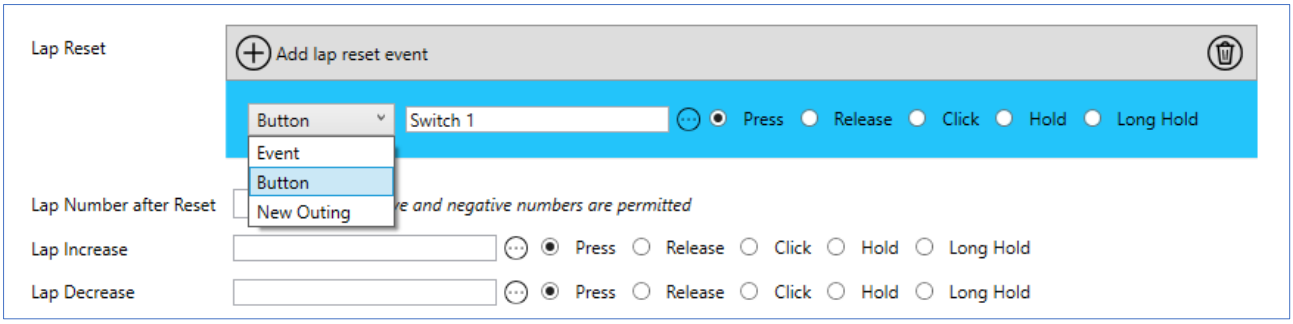
**Note**: The action does not reset the value of the current outing. You must stop logging and restart or power cycle the device to reset the outing number.




## 8.2. Circuits Node

### Lap reset

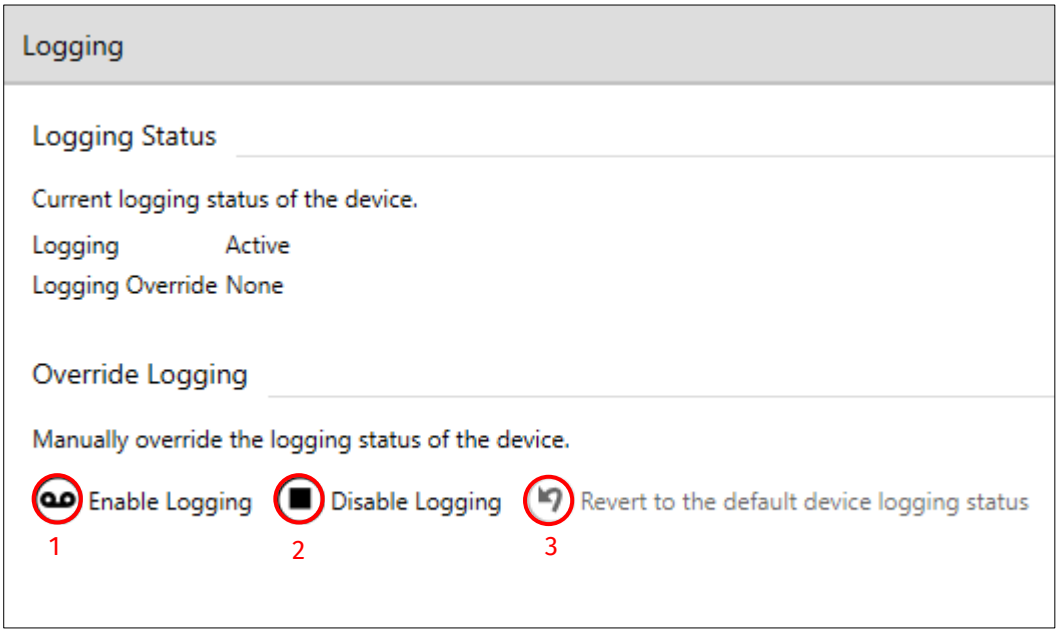
In the **Circuits** node (see Setups - Circuits) you can create multiple conditions to reset the lap number from a button press (See Setups – Buttons), an event, or at the start of a new outing.



# 9. Logging

The **Logging** node on the Actions tab allows you to manually enable and disable device logging when connected to a device. The current logging status of the device is displayed under **Logging Status**.

Select **Enable Logging** to manually start logging (1). Select **Disable Logging** to manually stop logging (2). If any override is selected, select **Revert to the Default Logging Status** to clear any manual logging overrides and use the standard logging conditions configured for the device (3).




The current **Logging** status (Active or Inactive) and **Logging Override** status (None, Force On, or Force Off) are displayed in the **Logging Status** section.

# 10. Beacons

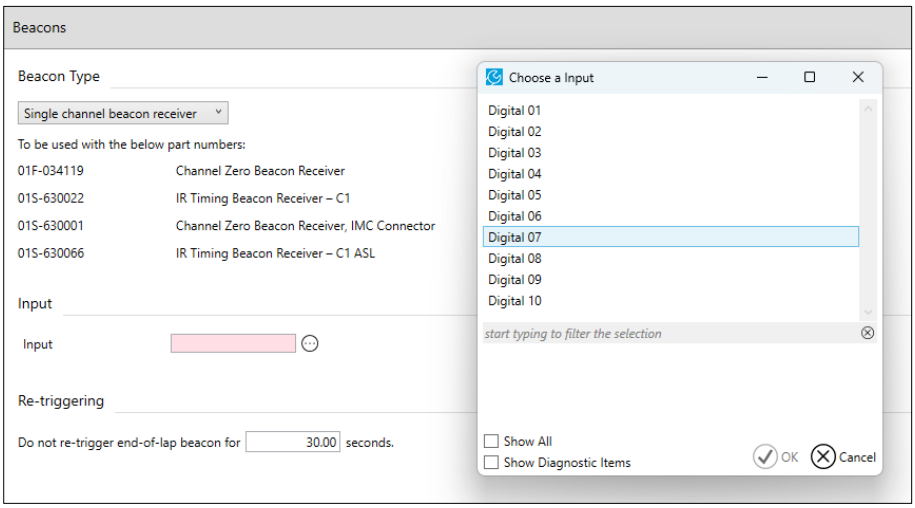
The **Beacons** node is used to configure the end of lap marker. The end of lap marker is used to divide *.pds* data files into laps. The node allows simple integration of a beacon to the setup with high flexibility for a range of multiple beacon types, including infrared beacons, GPS beacons, and virtual beacons:

- Single channel beacon receiver
- 10-channel beacon receiver
- 10-channel ASL beacon receiver
- 32-channel beacon receiver
- GPS beacon receiver
- Virtual beacon receiver

Select a beacon type to display a list of compatible parts. Contact Cosworth Electronics via the **Support** page on Cosworth website or e-mail electronics.support@cosworth.com for copies of individual product information sheets.

## 10.1. Single channel beacon receiver

A simple digital input beacon set up on the **Hardware Settings** node.




## 10.2. 10-channel beacon receiver

10-channel receivers allow the flexibility to use up to ten different beacon codes to avoid conflicts with other teams using their own beacon transmitter locations. The 10-channel receiver is set up in the same way as the single channel receiver.

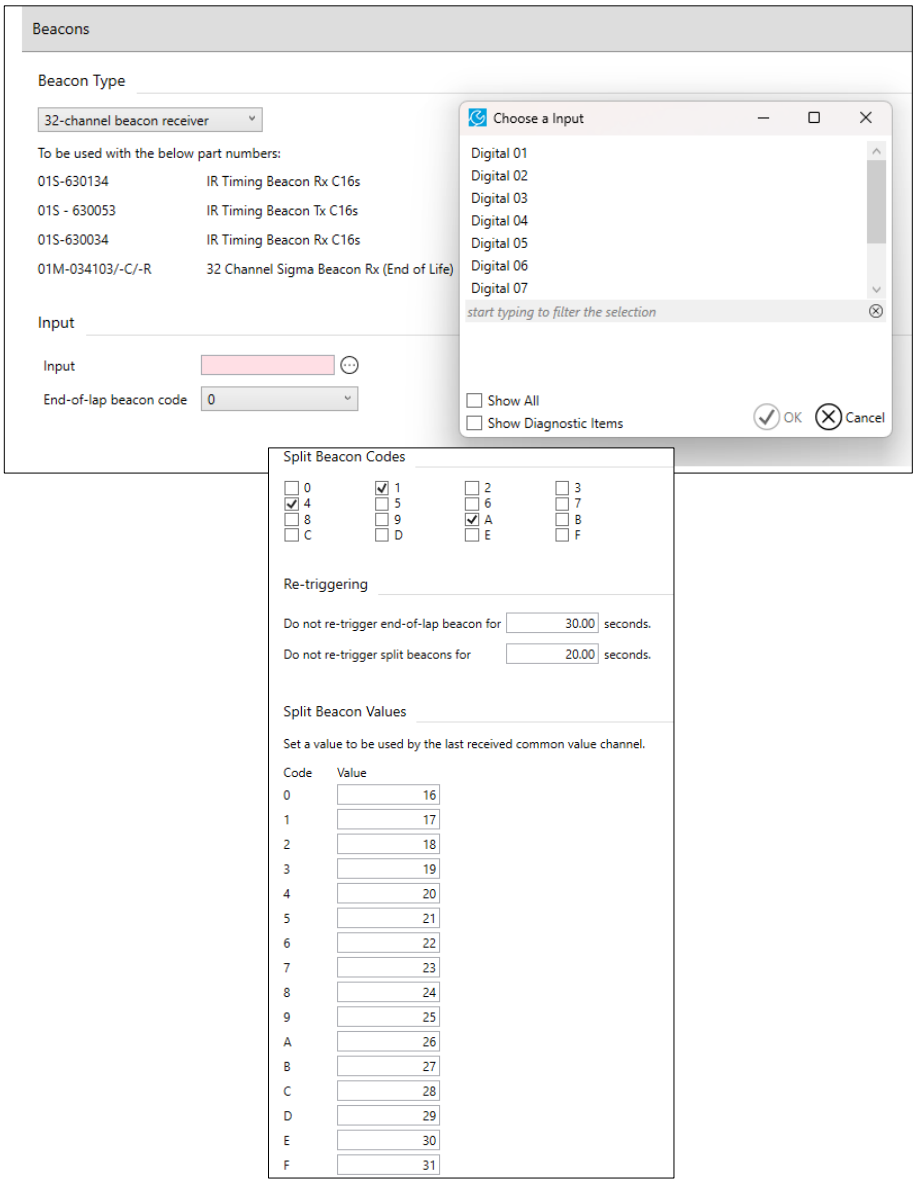
## 10.3. 10-channel ASL beacon receiver

The 10-channel ASL receiver is like the standard 10-channel beacon receiver, but you can configure the end of lap beacon code in the software.

## 10.4. 32-channel beacon receiver

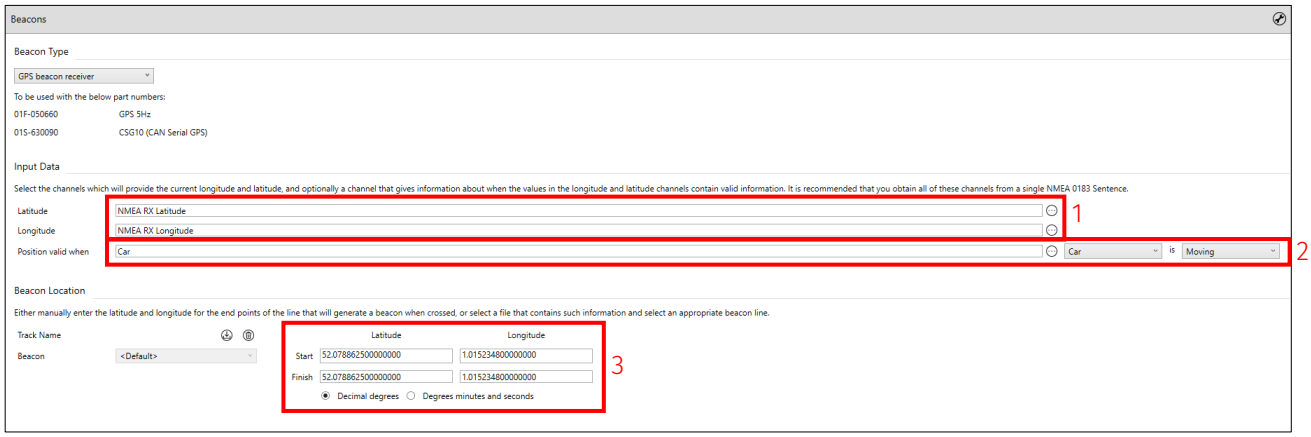
This option allows the flexibility to select from 16 different end of lap beacon codes, to avoid conflicts with other teams using their own beacon transmitter locations, plus provision for up to 16 split beacon codes to calculate accurate split times at multiple points around the track using several C16s split beacons.

## 10.5. GPS beacon receiver



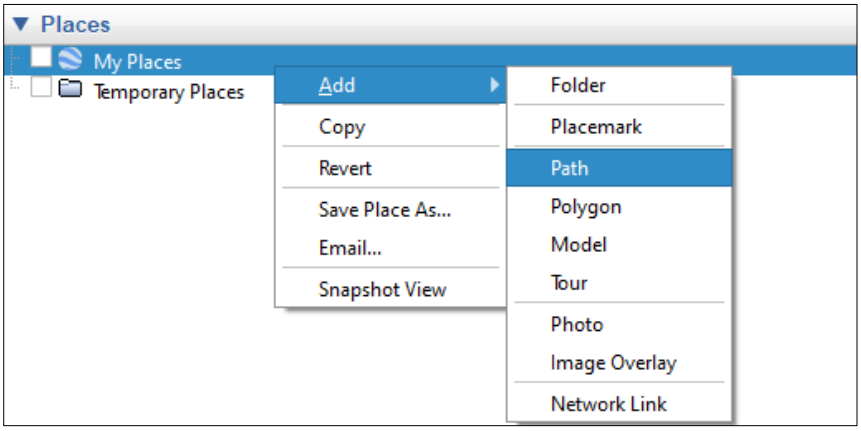
## 10.6. Standard GPS beacon

A GPS input configured in the **NMEA 0183 Decode** node can be used to configure an end of lap beacon. You can select **Latitude** and **Longitude** channels configured in the **NMEA 0183 Decode** from the browse menu for the **Latitude** and Longitude inputs (1). You must also define a strategy to indicate when the GPS position is valid (2) and enter the coordinates of the beacon location for the start/finish line (3).

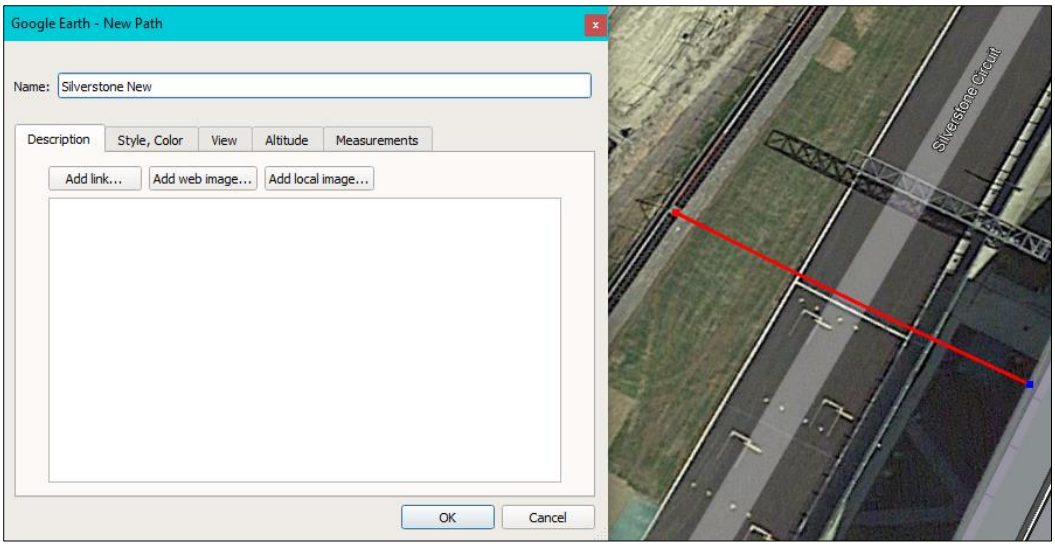



## 10.7. Create a GPS beacon

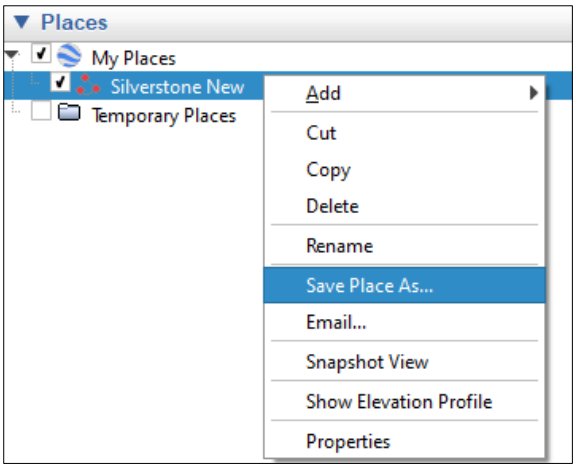
You can also use Google Earth to create a start/finish line to generate beacon events. Open Google Earth, and then right-click on **My Places > Add > Path**.



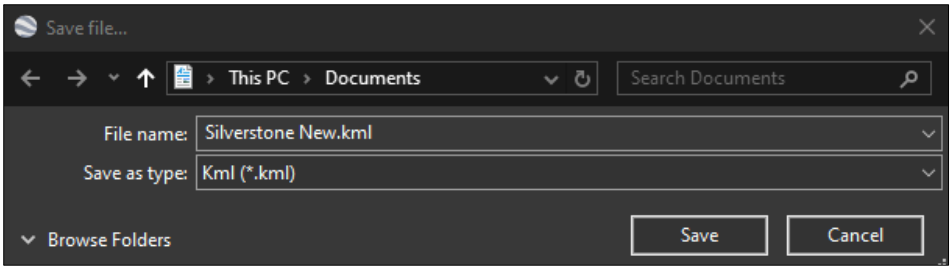

The **Edit Path** window is displayed. Enter a name for the start/finish line, and then use the mouse to draw the start/finish line on the map. This line must cross the track in the required location. A beacon is triggered every time this line is crossed.




When the new start/finish line is created, you must export it. Right-click **Path**, and then **Save Place As...**.

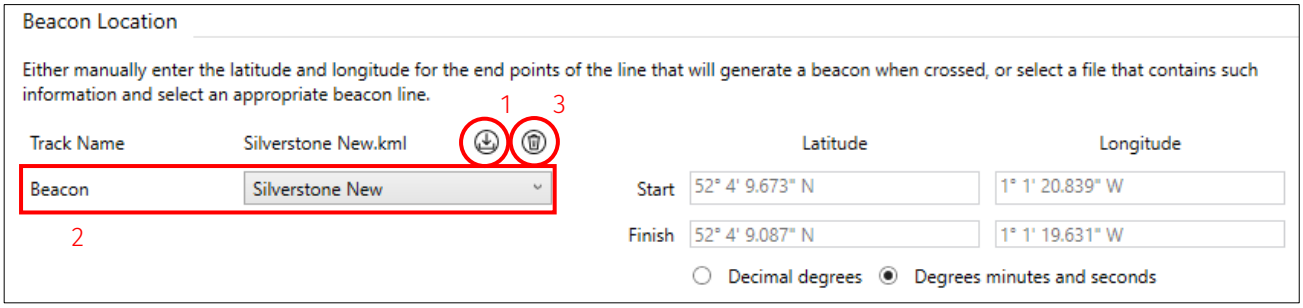



**Note**: Make sure that you select the *Kml (*.kml)* option from the **Save as type** dropdown menu.



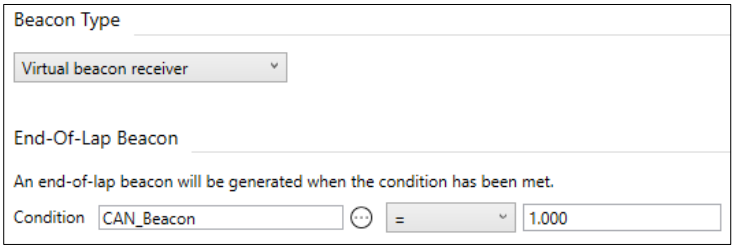
When you select the GPS Beacon Receiver from the **Beacon Type** drop down box in the **Beacons** node you can select the **Latitude** and **Longitude** channels configured from the **NMEA 0183 Decode** option from the browse menu for the Latitude and Longitude inputs (1). The strategy for when the GPS position is valid must also be defined (2).


Import the start/finish line file using the import tool icon (1), and then select the beacon path from the **Beacon** drop down menu (2). You use the 'bin; tool (3) to delete the start/finish line. The start/finish line coordinates are automatically populated from the *.kml* file.



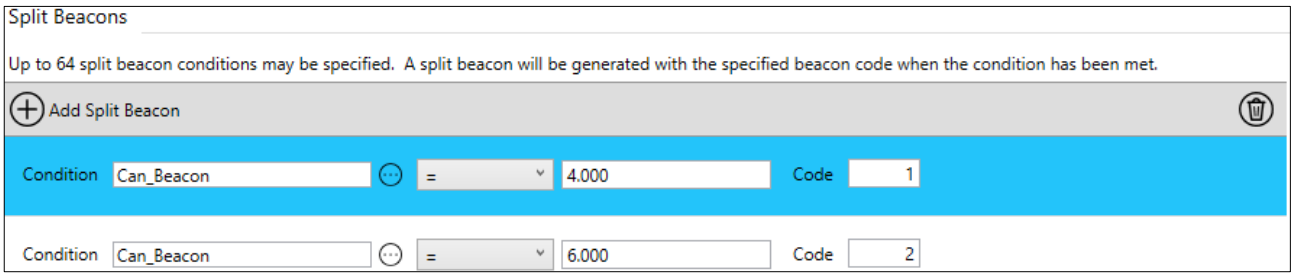

## 10.8. Virtual beacon receiver

You can configure virtual beacons in Toolset to use a user-defined channel as a beacon input. This could be a Maths, Logic or CAN channels.




## 10.9. Split beacons

There is also an option to add split beacons to the setup, triggered from a virtual source.

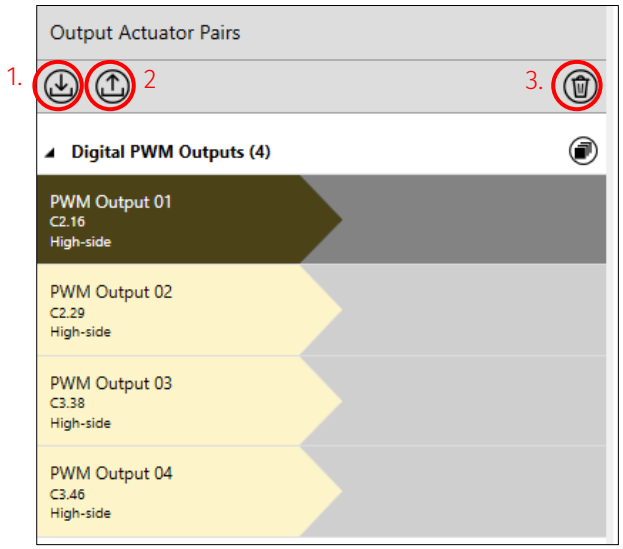


# 11. Actuators overview

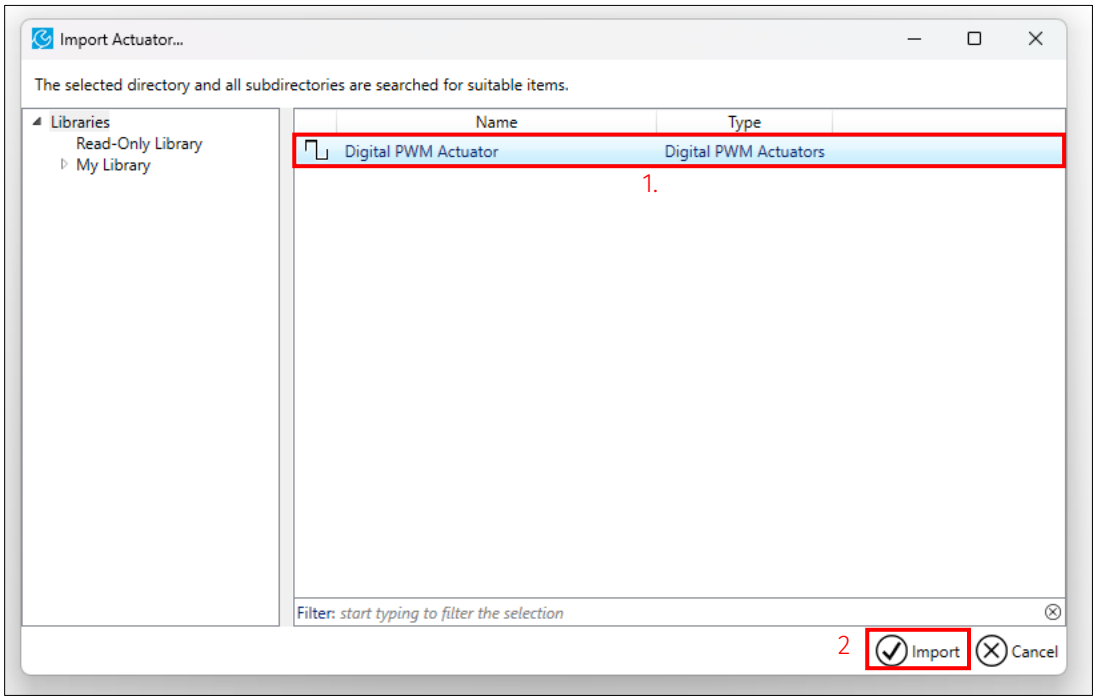
For devices such as the Badenia with High Side Driver outputs and the CCW Mk3 for Opto-Isolators, the **Actuators** node is used to configure the outputs.

## 11.1. Configure an actuator

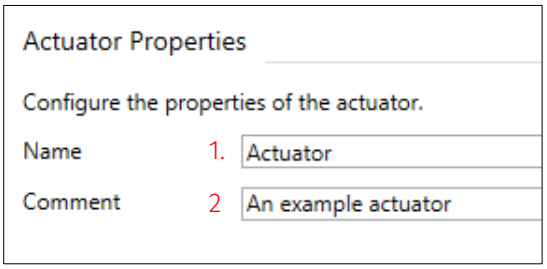
Use the **Import** option to import an actuator from the Toolset library (1). You can import and export previously configured actuators between existing setups (1/2). You can use the 'bin' option to delete actuators from the setup (3).




Select the actuator to import from the dialog box (1), and then click **Import** (2).



When the actuator is imported, you can configure the output duty. Enter a name for the actuator (1), and an optional descriptive comment (2).

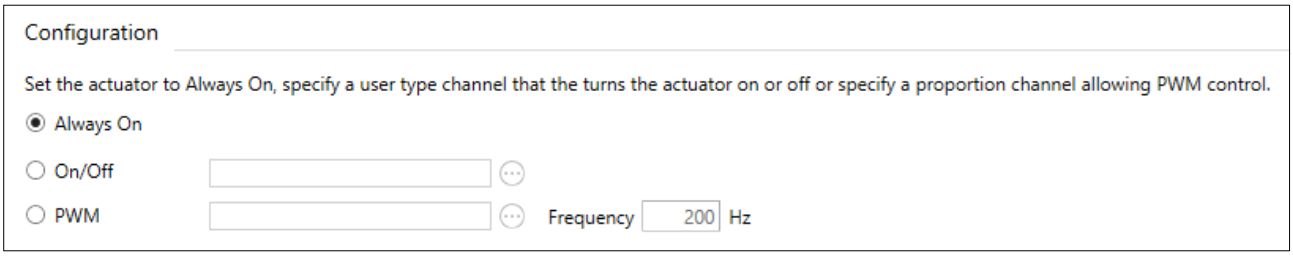



You can now configure the actuator output duty. There are three actuator configuration options:

Always On: 100% duty when the device is on.

On/Off: You can use a user defined channel (Maths, Logic, CAN, Alarm, and so on) to switch the output and configure its duty. For example, a register Maths channel or a Counter Logic channel can switch the output on, and the frequency of the channel determines the switching frequency.

PWM: You can use a user defined Maths channel with 'Proportion' units to configure the PWM duty for the output. The frequency of the output is configurable between 1 – 400Hz.

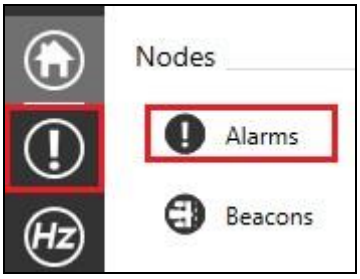


# 12. Alarms

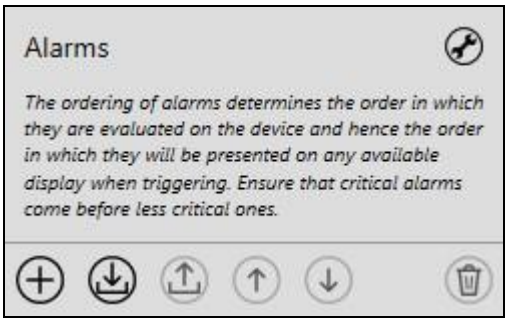
In Toolset you can use alarms to notify the driver/team when specific events or conditions are met.

## 12.1. Create an alarm

To create an alarm within a setup, click the **Alarms** node, or use the shortcut on the left of the screen.



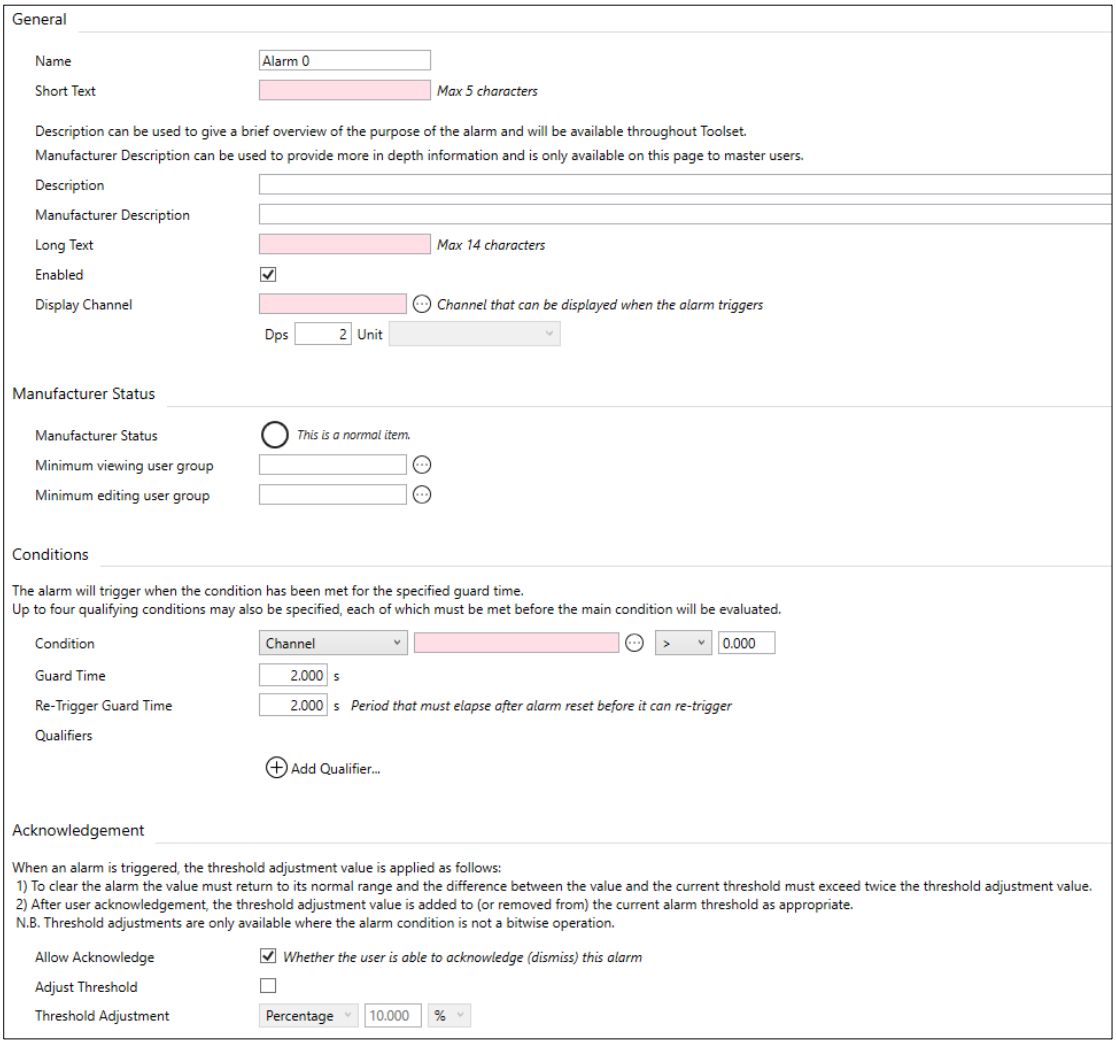

Click the **+** button at the top left to create an alarm. You can also import, export, prioritize (top to bottom), and delete alarms.




Click the 'wrench' icon to open **Alarm Settings**. You can select the option to acknowledge alarms and the rate at which they are evaluated.


Currently you can only trigger an alarm acknowledgement through a button event, and not a channel.

When an alarm is created, the alarm configurations are displayed on the right.




### General

**Name**: The name of the alarm channel. Use the format 'Alarm_Quantity_Channel_Qualifier' to group all alarms together.

**Short Text**: The short text used in display configurations.

**Long Text**: The long text used in display configurations.

**Enabled**: When this option is selected you can use the alarm you create in Math Channels, Displays, and so on.

**Display Channel**: When the alarm is on, the value of the channel is displayed on the dash. Select the decimal places (Dps) and the unit.

### Manufacturer Status

Refer to **Pi Toolset – Setup Locking** for details about this option.

### Conditions

Condition: This option allows you to define under which condition(s) the alarm is triggered. This can refer to a Channel (Oil Pressure, for example) or a Bitfield (Button for example). If a channel is selected, then a valued condition is required. If a bitfield is chosen, then a specific state is required.

Guard Time: The time during which a condition must be true to trigger the alarm.

Re-Trigger Guard Time: The time that must elapse after an alarm reset before it can re-trigger.

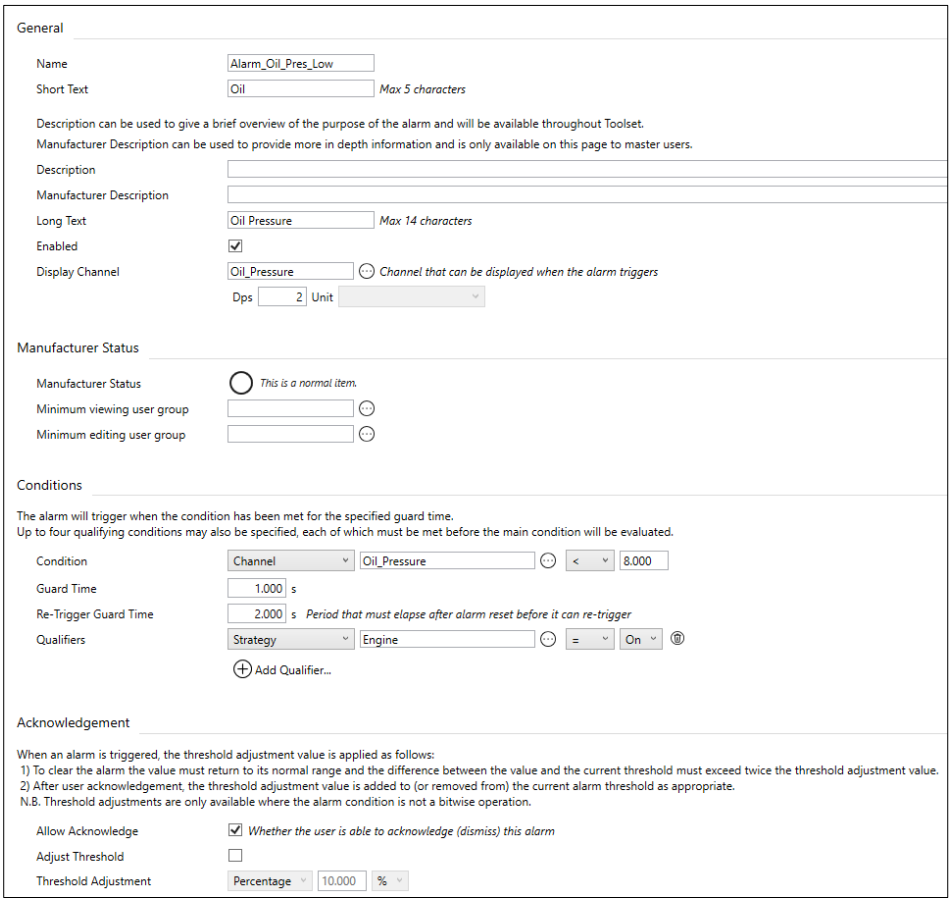
Qualifiers: You can add qualifiers to enable or disable alarms. The strategies available are automatically created by Toolset (Car, Engine, Logging, and Moving), or can be created by the user (see Setups – System Status for more information).

### Acknowledgement

Allow Acknowledge: If selected, the driver can acknowledge the alarm. If unselected, the alarm will always be active or inactive depending on its settings and cannot be over-ridden.

Threshold Adjustment: This option is only available when the Allow Acknowledge option is selected. You can change the threshold of the alarm when it is active, and the driver has acknowledged it. For example, if an alarm is set to display when Oil Pressure is low (less than 8 bar) and the Threshold Adjustment is Offset 2 bar, then after the alarm is acknowledged, it will not be active again until Oil Pressure is less than 6 bar. You can only reset the threshold adjustment when the car is stopped and powered off, or when the difference between the value and the current threshold is at least twice the threshold adjustment.

In the example of a Low Pressure Oil alarm below, the alarm is active when the channel Oil Pressure is less than 8 bar. It has a Guard Time of 1 second and a Qualifier that the Car must equal Running. The Display Channel is the channel that is associated with the alarm so the driver can see the Oil Pressure.  See the next section for details on how to use the alarm.



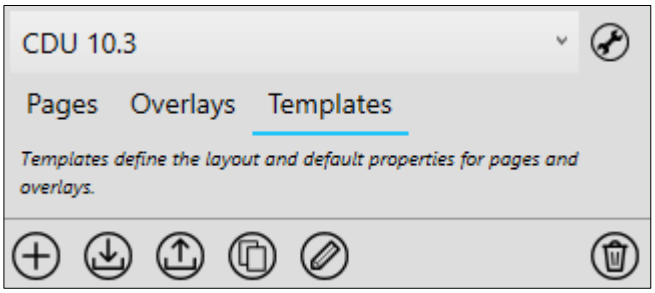

Alarms are also considered events. When you select from a list of events (for example on the **Fuelling** node, an event is selected for Reset Fuel Used, all alarms are displayed. This can be useful when a condition must be met for a Beacon, Fuelling Reset, and so on.

### Use an alarm

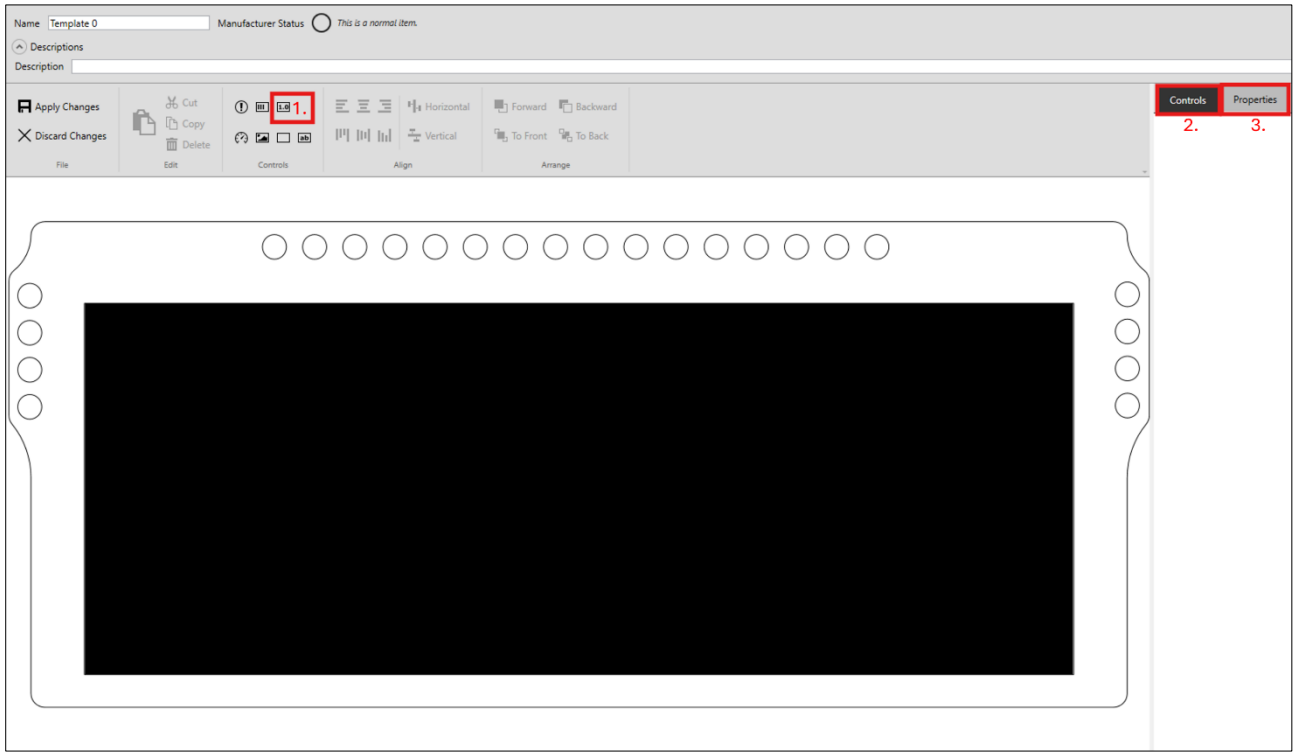
There are several ways to use an alarm on the display (CDU/CCW). You can use a Channel Control (Display Only), use an Alarm Control, or use an alarm to drive an overlay. Click **Displays (Ethernet)** or use the shortcut on the left.

### Use a channel control

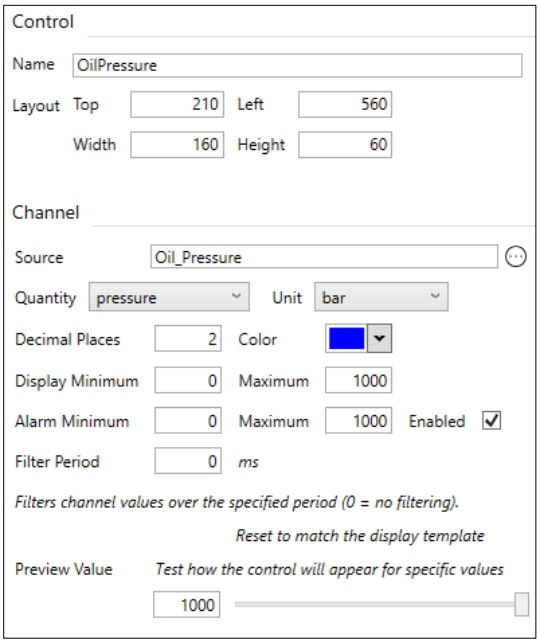
Click **Templates** to create a new template.



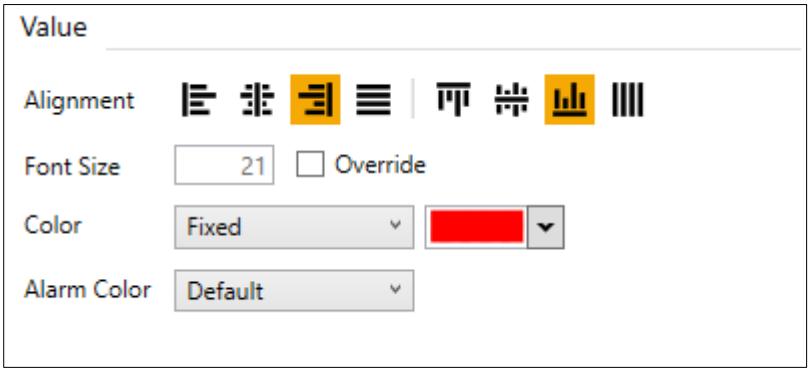

Click the **+** button to open a blank display. To add a channel control, click on the box (1) in **Controls** and drag it to the screen. When you click on the new box, the **Controls** tab (2) is populated with the box name (this can be left as the default). Click **Properties** (3) to open the channel properties.



From the **Properties** tab, navigate to the **Channel** section, and then select the required channel. In this example, Oil Pressure is added. Click **Enable** to select a display only alarm. This is separate from an alarm created in the **Alarms** node and is not displayed in data and is for display purposes only.



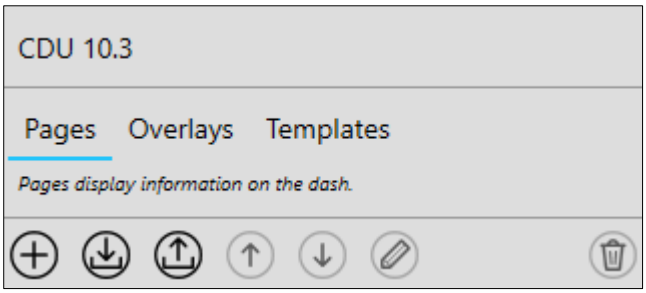

At the bottom, there is an option to select the font size, select the channel value colour, and select the alarm value colour. The text and value will flash in the alarm colour when the alarm is active. See **Displays** for more configuration details.



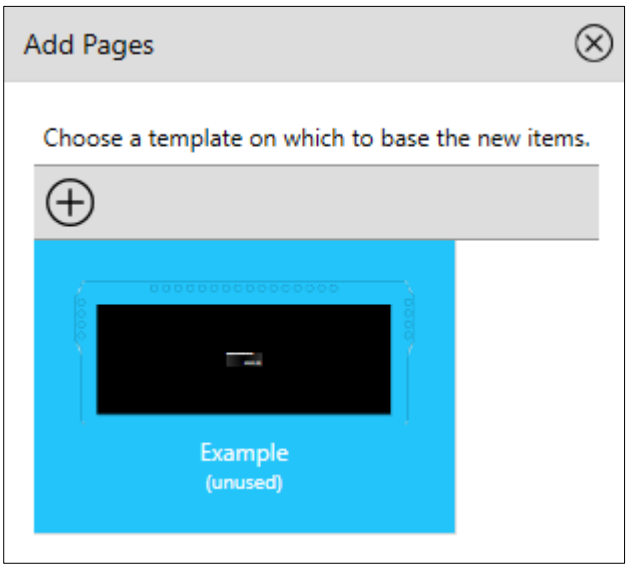

Click **Apply Changes** to save the template. You can use the template for a page or an overlay.

**Note**: It is recommended that you make templates generic enough to be used more than once if required and make all changes to the page/overlay rather than the template itself, as changes to the template may not propagate to the page/overlay correctly.

To create a page from a template, click the **+** button on the **Pages** tab from the **Display** node.

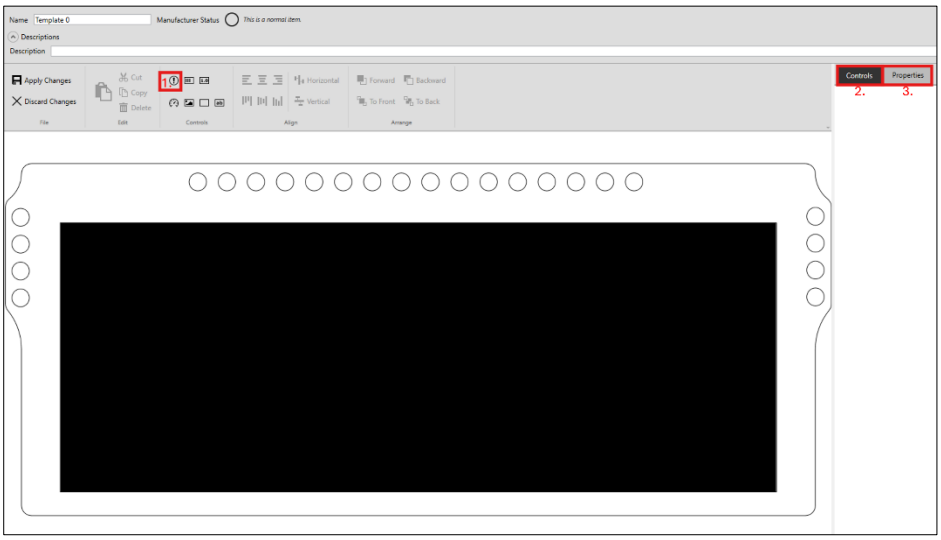



A new window displays all the available templates. Select the required template, and then click the **+** button.

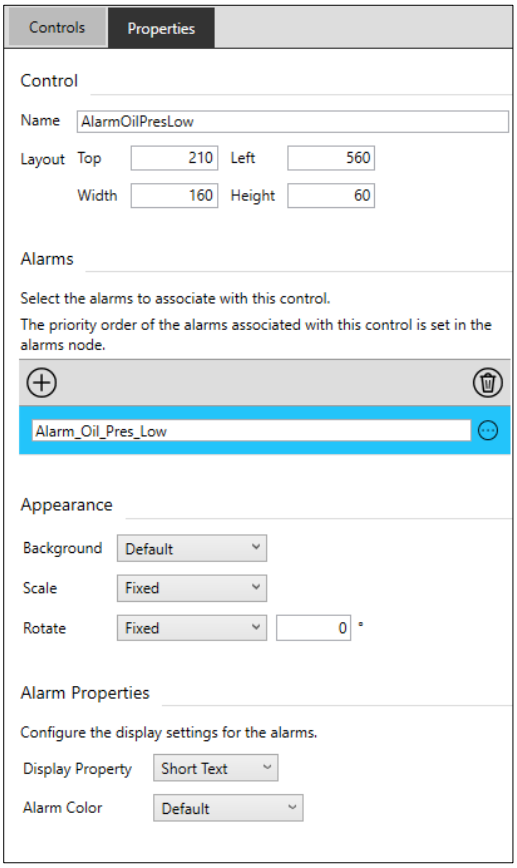



**Use an alarm control**

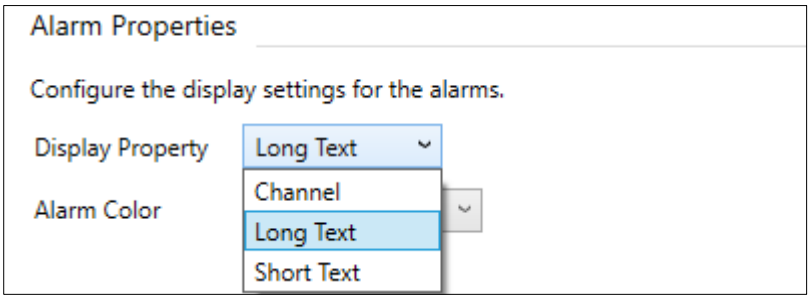
Create a new template, as described above, and then add an alarm control (as described below). These are only displayed when the alarm (from the **Alarms** node) is active, effectively functioning as a pop-up alarm. To add an alarm control, click on the box (1) in **Controls**, and then drag it to the screen. When you click on the new box, the **Controls** tab (2) is populated with the box name (this can be left as the default).



Click **Properties** (3) to open the Alarm properties. This creates an alarm control, like a channel control. Use the **+** to button to add an alarm to the alarm control.

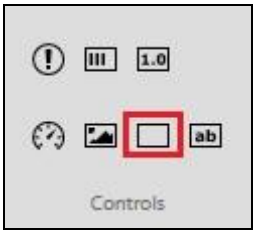



If the background is default, the box is hidden until the alarm is active. Under **Alarm Properties**, the **Display Property** can be the Channel (the 'Display Channel' in the **Alarms** node), Short Text, Long Text, or Channel value. These are configured on the **Alarms** node.

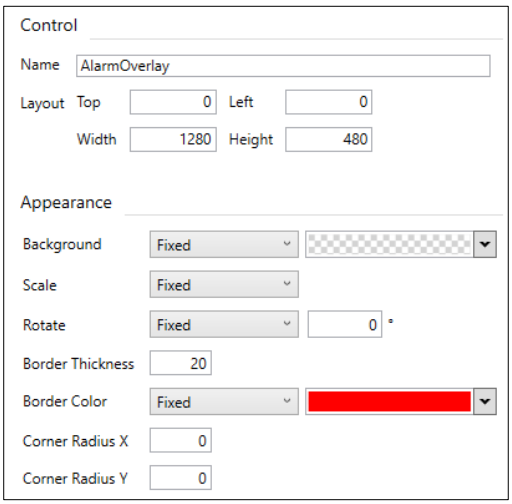


**Use an alarm to drive an overlay**

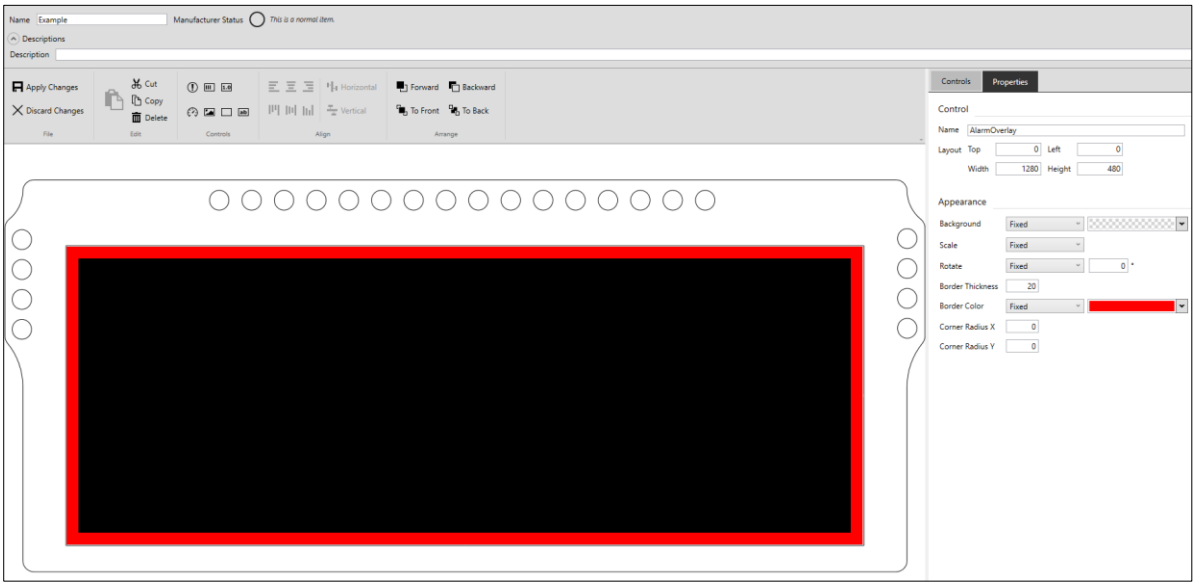
An example of generic overlay is shown. Create a new template, and then add a rectangle.



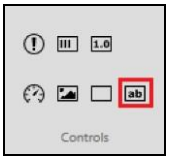

Click the **Properties** tab and change the **Fill Colour** to transparent (if black is selected, it covers everything on the screen). Change the **Border Color** to the required setting. Set the **Border Thickness** to ~20.



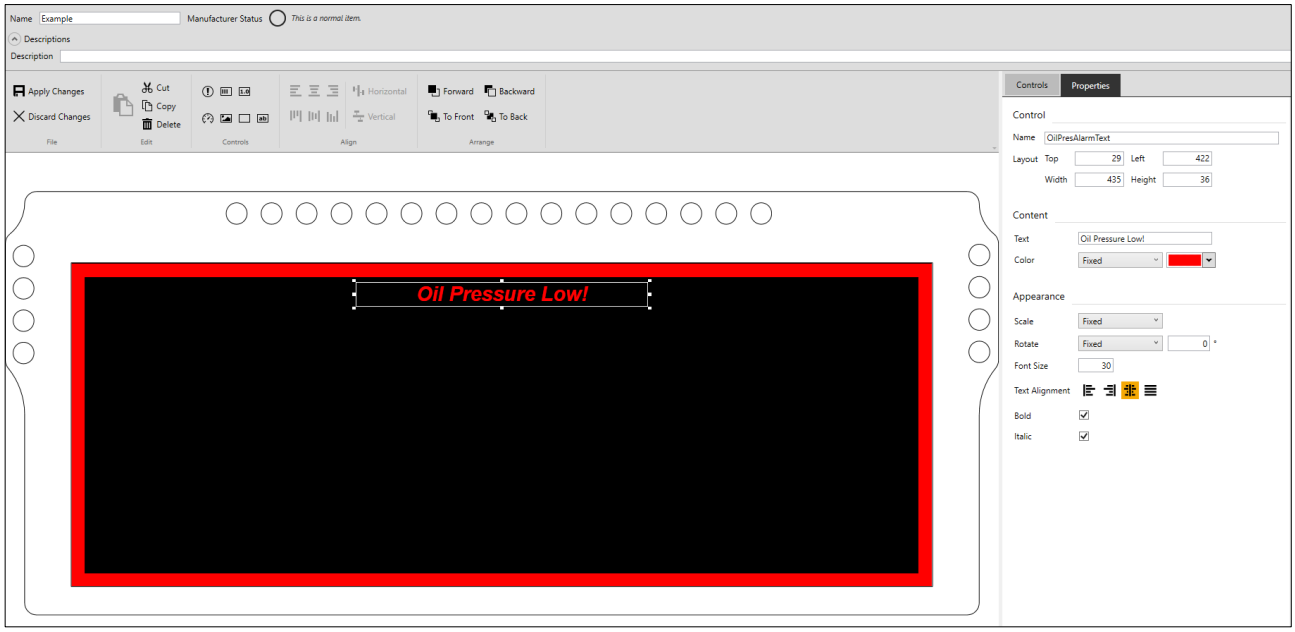

Position the rectangle to make a border around the display.



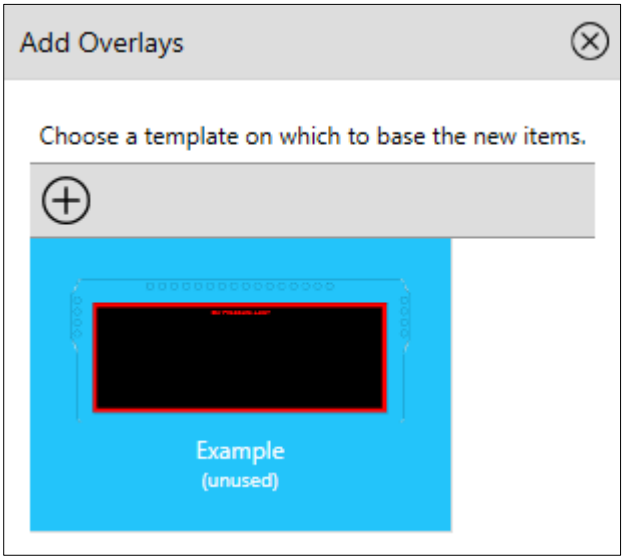
Next, you can add a text box to the screen.



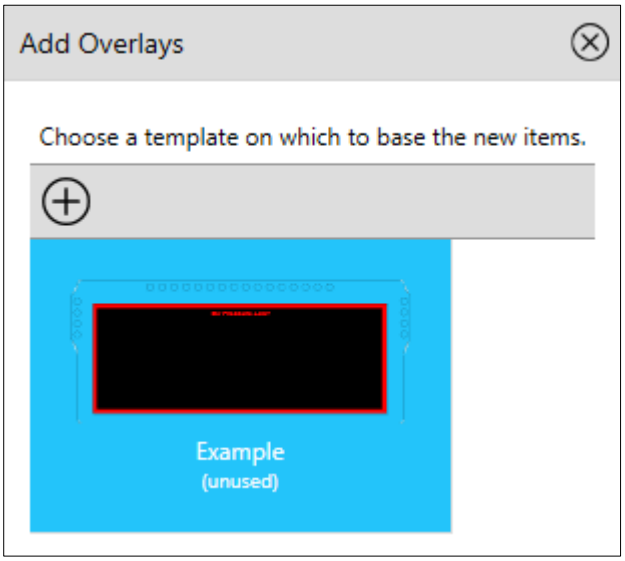

Enter ‘Oil Pressure Low!’ under **Label** on the **Properties** tab and position the text at the top of the screen.




Save the changes to the template and go to **Overlays**. Click the **+** button to add a template to overlays and select the new template.



Click the ‘pencil’ icon, and then select to add the alarm previously created. When the alarm is active this overlay appears on the display.



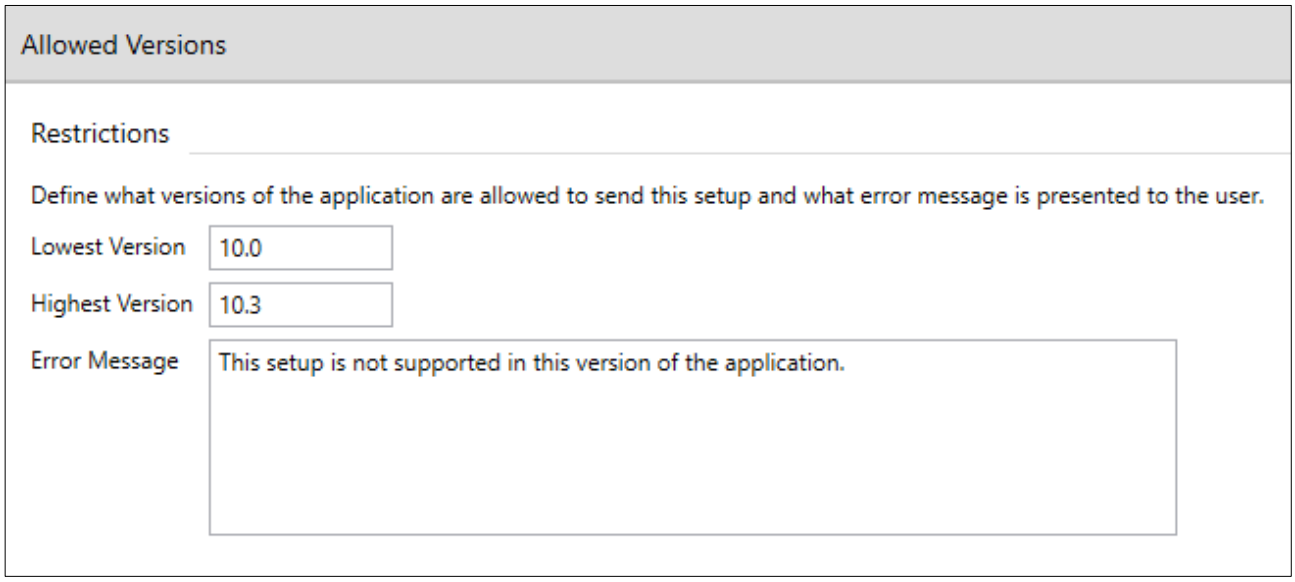
# 13. Allowed versions

You can use the Allowed Versions node to set the minimum and maximum version of Toolset allowed to modify setups. This feature is useful if a manufacturer wishes to lock the setup to a specific version (if the series/setup is homologated), so that customers can only use that version of software to modify the setup.

## 13.1. Setting the allowed version

You can configure the lowest and highest version of software, and then lock or hide the Allowed Versions node using the Setup Locking node to make it inaccessible to a customer.

You can also configure a message to be displayed if a customer tries to open the setup in an incorrect version of Pi Toolset.



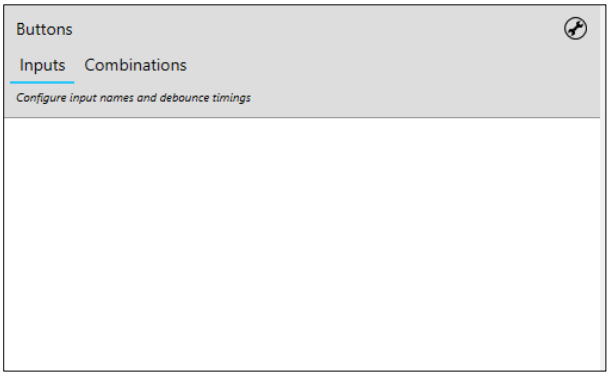
# 14. Buttons

Configuring buttons within Toolset is one of the best ways to expand functionality and allow flexibility when you create a setup. The following shows how you can configure buttons (with an emphasis on CCW & CSB), how buttons within Toolset are structured, and some examples of how to use them.

## 14.1. Configure button inputs

### Add buttons

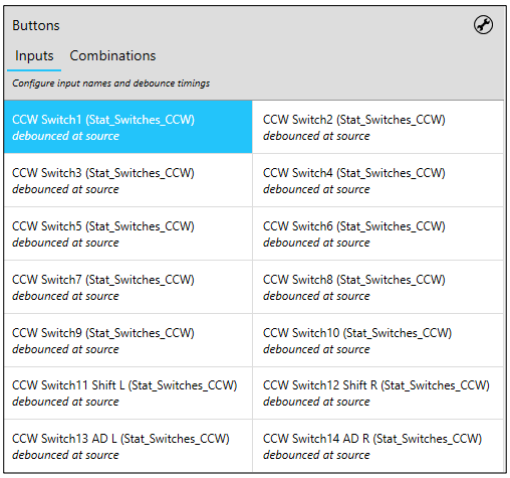
By default, on the **Buttons** node there are no entries on the **Inputs** page. Unlike a Maths channel or Logic channel, which are software defined, buttons are hardware defined. On devices such as the CCW and the CSB which communicate via CAN, the button properties are imported from the associated device CAN.




When you configure a CAN device such as the CCW or CSB, it comes with an associated CAN stream to transmit data between the primary device and the CAN device. Once the device stream has been imported to Toolset (see **Streams**), it automatically updates the **Buttons** node to show the available buttons.

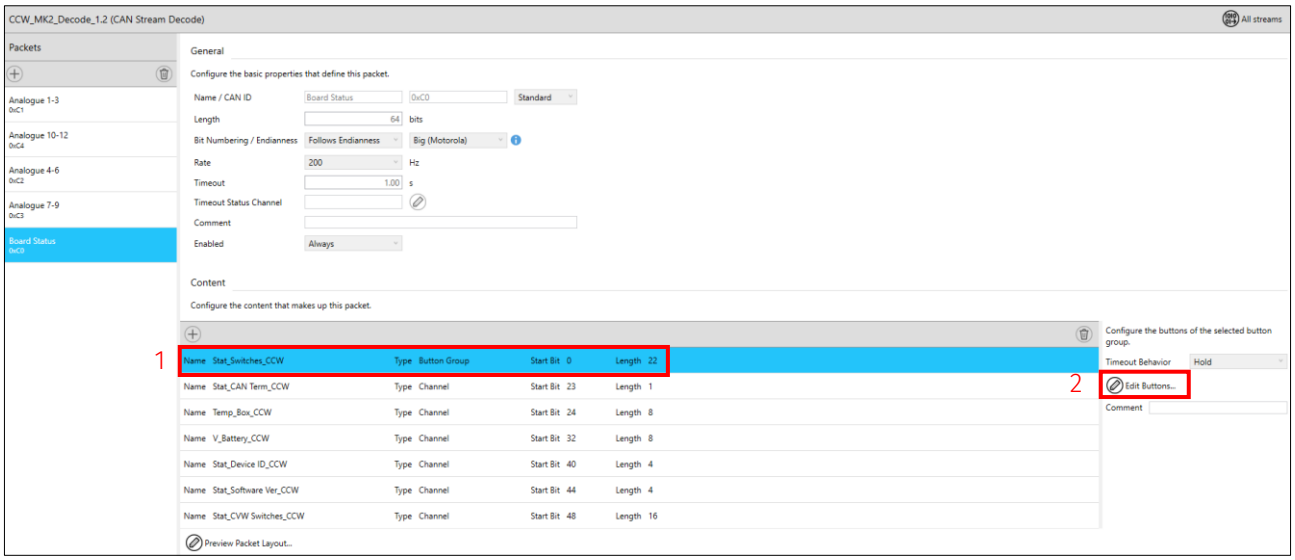
You can also generate your own virtual buttons via CAN streams and define any received channels as a 'Button Group'.

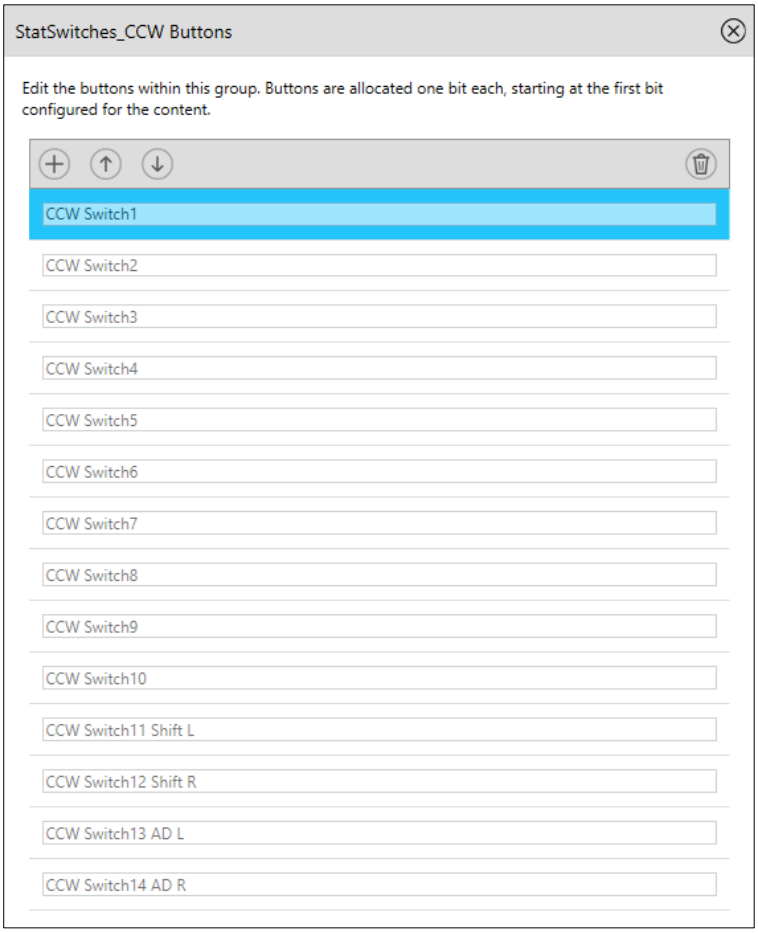
When the device stream is imported (a CCW stream in the example below), button inputs are shown on the **Buttons** node in the format of 'Name (Source)'.



### Name buttons

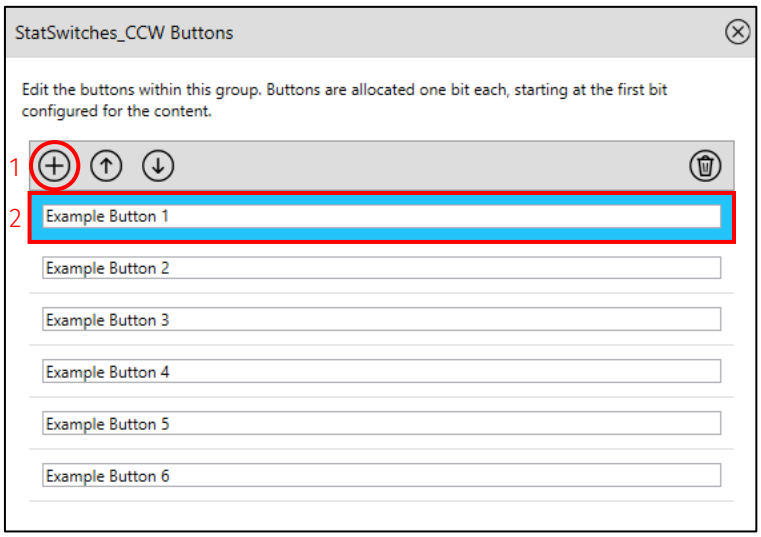
The name of the button is configured in the **CAN Streams** node. To edit the name of the button, select the 'Button Group' packet content (1), and then click **Edit Buttons** (2).



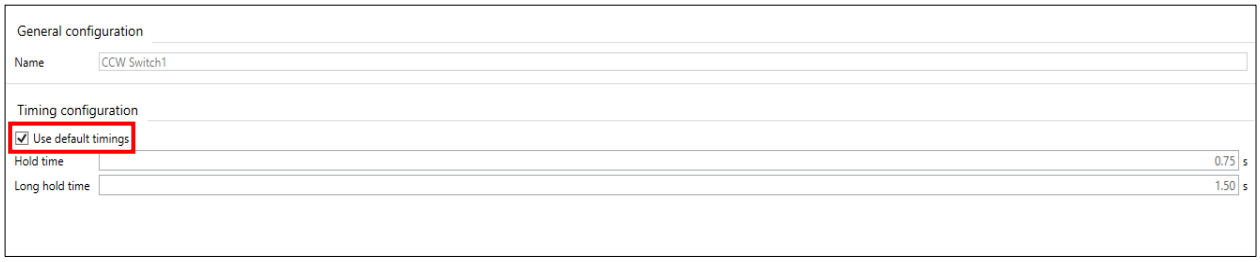

If a pre-compiled CAN stream for a device (that has been locked) is imported, you cannot edit the default button name, and it is greyed-out.

If a new CAN stream is compiled, you can add buttons by clicking the **+** icon (1), and then enter the button name (2).



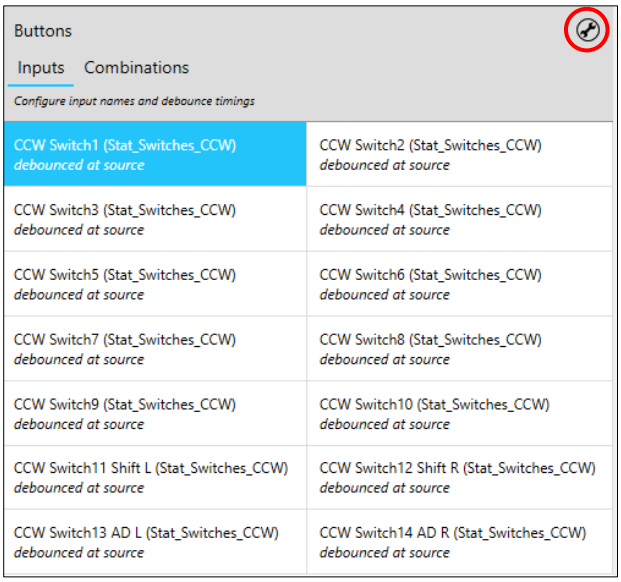

### Configure button properties

From the **Buttons** node, the **Button Configuration** option is displayed. Here the name of the button is displayed, and you can define the **Timing configuration**. Select or unselect the **Use default timings** option (1).

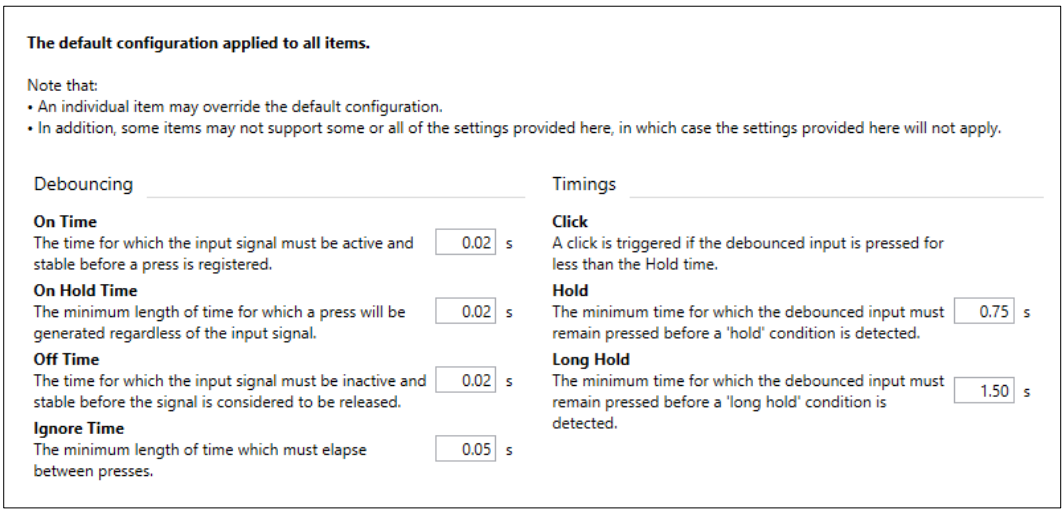



Click the 'Wrench' icon on the **Buttons** node to configure default timings.

Click the 'wrench' icon to display a dialog box where you can configure the default timing configuration.



You can configure timing configuration settings such as 'On', 'On Hold', 'Off, and 'Ignore' times under **Debounce**. 'Click', 'Hold', and 'Long Hold' timings are configurable under the **Timings** section.



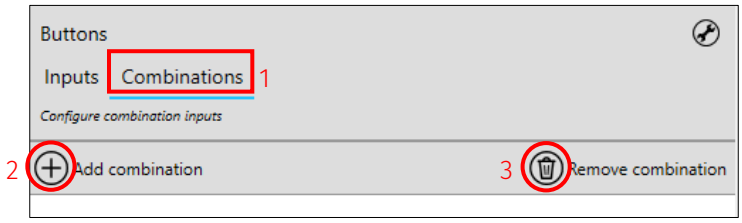

For most functions, the default values should work well.

**Note**: These settings are for all buttons on the **Inputs** tab. If you need to edit button settings, unselect the 'use default settings option.

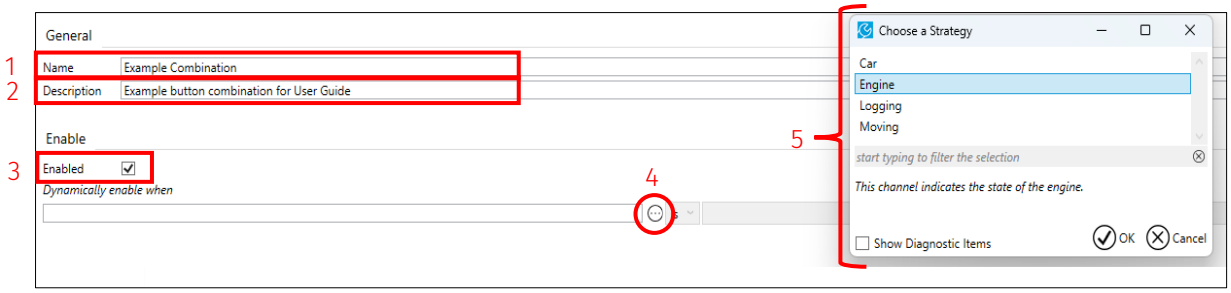
### Configure button combinations

On the **Combinations** tab, you can configure channels that react from combinations of button inputs. For example, you can configure a channel to be triggered when two buttons are pressed simultaneously. This channel can be used to drive miscellaneous functionality within the setup.

To create a new button combination channel, click the **Combinations** tab (1), and then click the **+** button (2). Combinations can be deleted with the 'bin' icon (3).

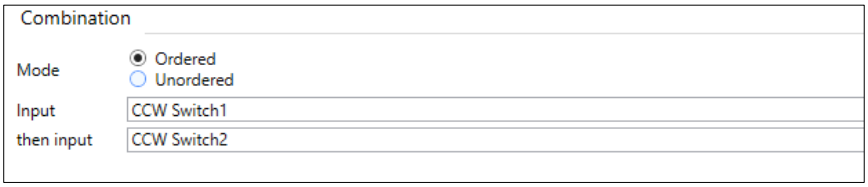



You can configure the new button combination. Name the combination (1), and then provide an optional description (2). You can select if the combination is enabled in the setup (3). You can then use the **Choose a Strategy** option to select the strategy condition (4/5.).

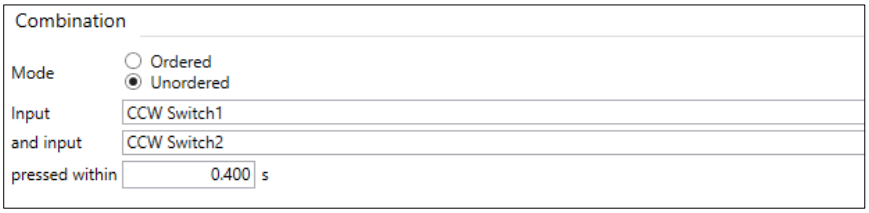


You can then define the button combination with options for the **Ordered** or **Unordered** button inputs:

- **Ordered**: the button needs to be pressed first, and then the second button to trigger the output channel.



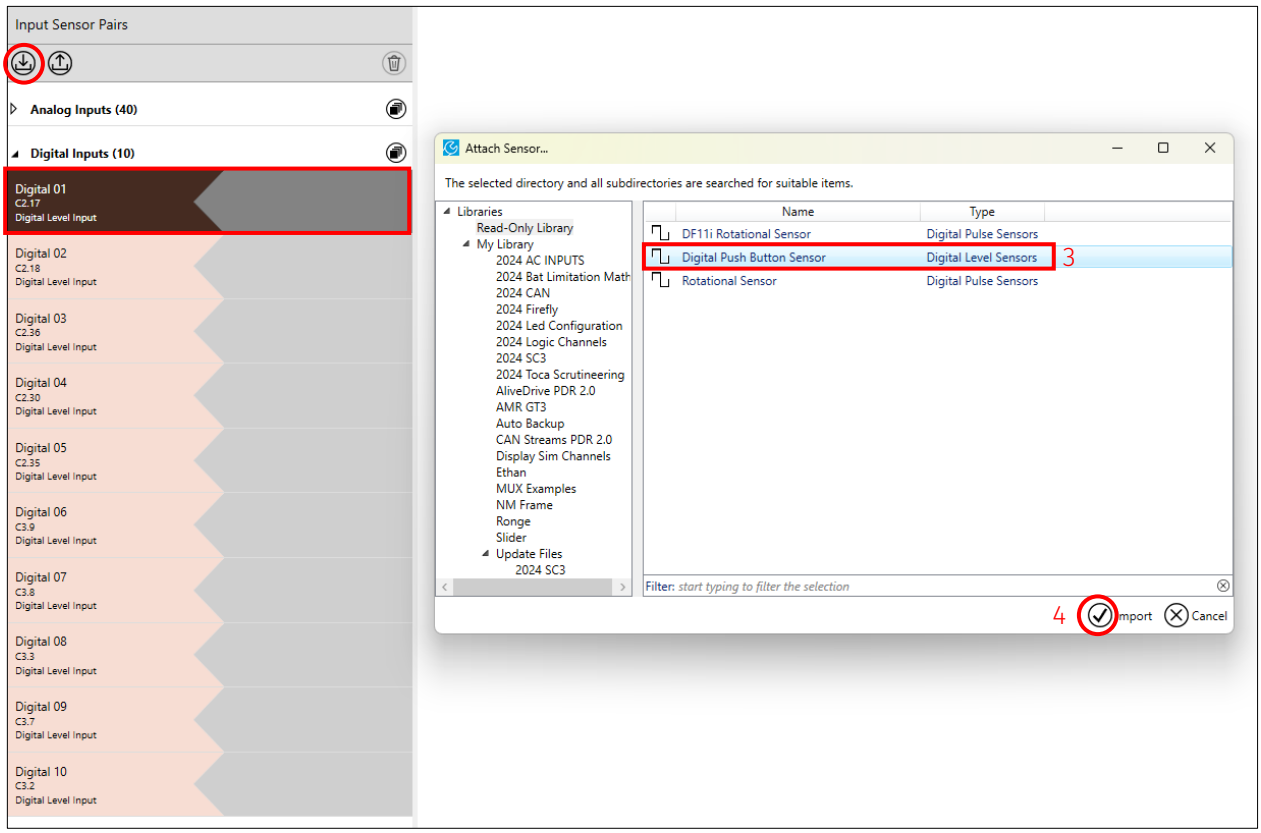

- **Unordered** – Two buttons must be pressed to trigger the output channel but a time threshold 'pressed within' must be defined to trigger the channel.



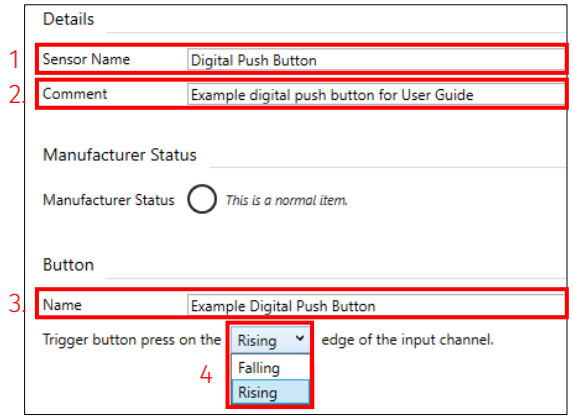

### Configure a digital button

A button can also be configured from a digital input rather than from a CAN device. This is useful if you need an additional button in the setup which is separate to any CAN devices.

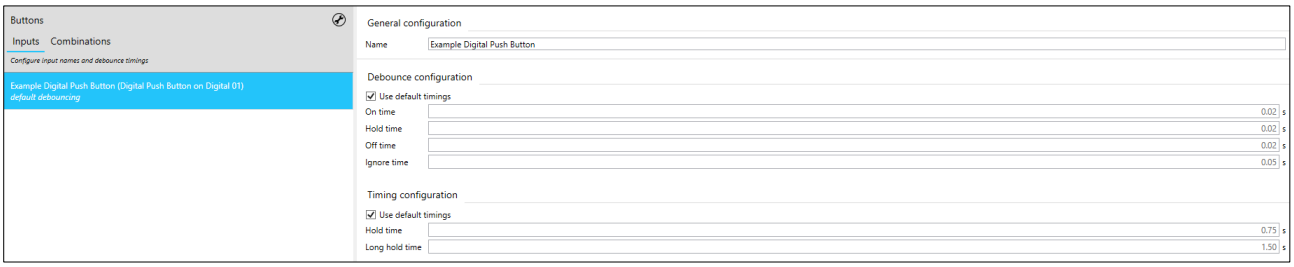
To configure a digital button, go to the **Sensors** node, and navigate to the available digital inputs. Click a digital input (1), click the import tool (2), select 'Digital Push Button Sensor' (3), and then click **Import** (4).



You can now configure the digital button. Enter a **Sensor Name** (1) and an optional comment about the digital button in the **Details** section (2). The actual button **Name** is configurable (3), as is the 'mode' (4), in the **Button** section. The 'mode' allows the user to configure if the button triggers on the rising or falling edge of the digital input channel.




Return to the **Buttons** node and acknowledge the creation of the digital button. The default button settings apply to this button, but default timings can be deselected. You can use the digital button in button combinations and in conjunction with CAN buttons.



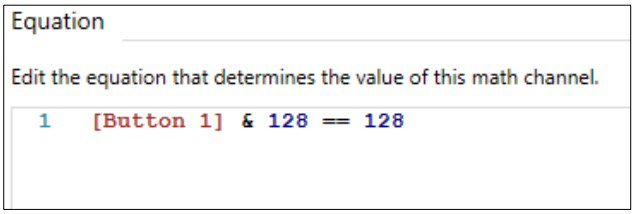
**Button structure and examples**

The CSB or CCW buttons consist of an 8-bit output to determine what type of button press is commanded. These are configured as bit-fields by default with bit 0 to the left, and bit 7 to the right. You might need to acknowledge specific buttons press types and ignore others. The table below shows the press type and the output bit number and decimal value:

| Press Type | Bit Number | Decimal Value |
|---|---|---|
| Raw | 0 | 1 |
| Debounced | 1 | 2 |
| Clicked | 2 | 4 |
| Held | 3 | 8 |
| Long Held | 4 | 16 |
| Click Latched | 5 | 32 |
| Latched Held | 6 | 64 |
| Latched Long Held | 7 | 128 |

For example, to command a Maths channel to enable on a 'Latched Long Hold', the channel syntax is configured as shown below:

For this channel, when the button value is both equal to 128 (the '& 128' portion of the equation) **and** equal to 128 (the '== 128' portion), then the button output is acknowledged. Therefore, there is no chance that an intermediary button state triggers the channel output. The computed Maths Channel generates a value of 1 or 0.

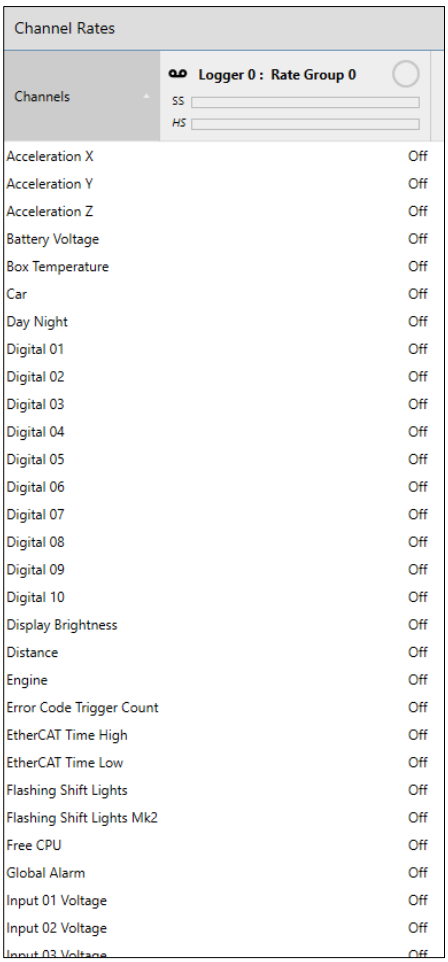


# 15. Channel rates

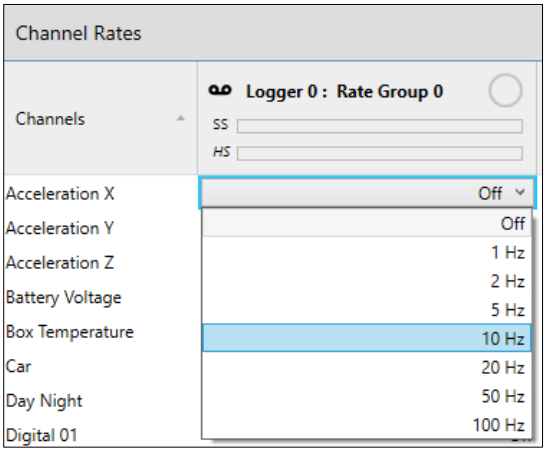
The **Logging Rate** node is used to define the frequency at which channels are logged. Before you set channel rates, a logging table must be defined (see **Setups - Logging**). You can access the **Channel Rates** node from the home page, the quick tabs on the left of the screen, and the **Logging** node.

## 15.1. Get started with logging rates

When you first open the **Channel Rates** node, a list of all the default/native channels is displayed in the default logging table. These channels are all set to 'Off' by default. The name of the logging table is configured in the **Logging** node.



Click ‘Off’ for any entry, and then select one of the available rates from the dropdown menu to set the channel rate.

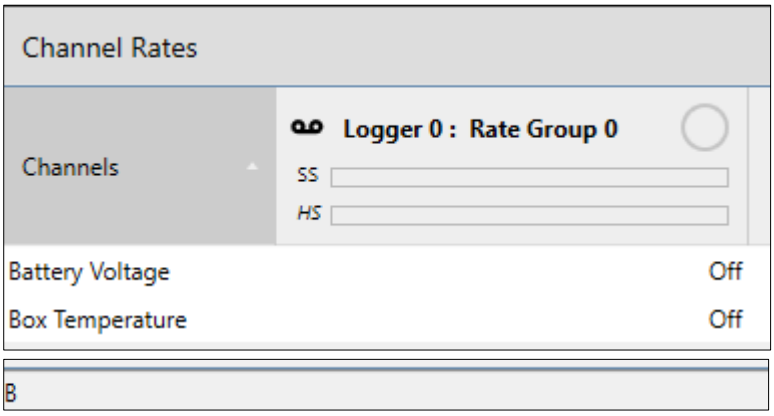



## 15.2. Search channels

When you work with large setups with many channels, you can search the channels through the search bar at the bottom of the screen.

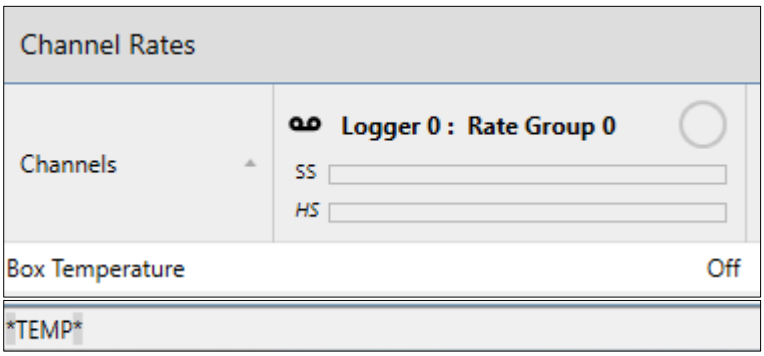

### Standard Search

Type in the node to filter the channel. By typing the channel name, the exact channel can be filtered. This search method must match the channel name exactly to show the result.

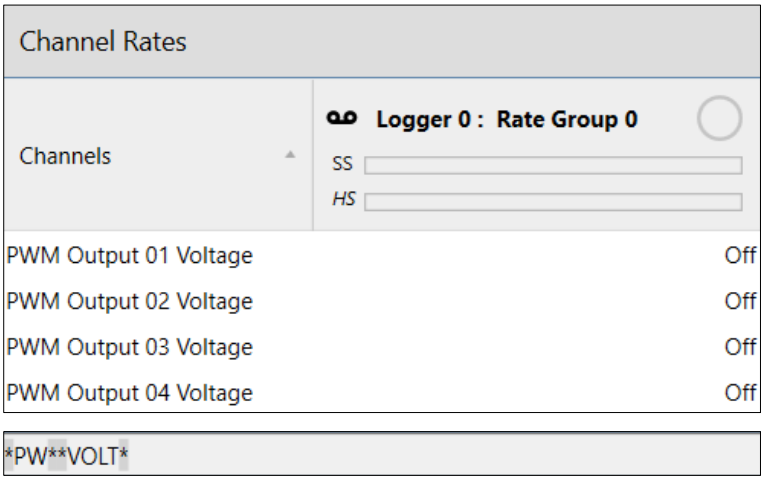


### Magic search

If the exact name of the channel is not known, you can use the magic search function. Magic search is a 'broad' search tool, which can display results that contain any string. Press CTRL + Shift + 8 simultaneously. A highlighted star indicates that the magic search is active. Type a specific word or phrase, and then press CTRL + Shift + 8 again to close the search. An example is shown below.

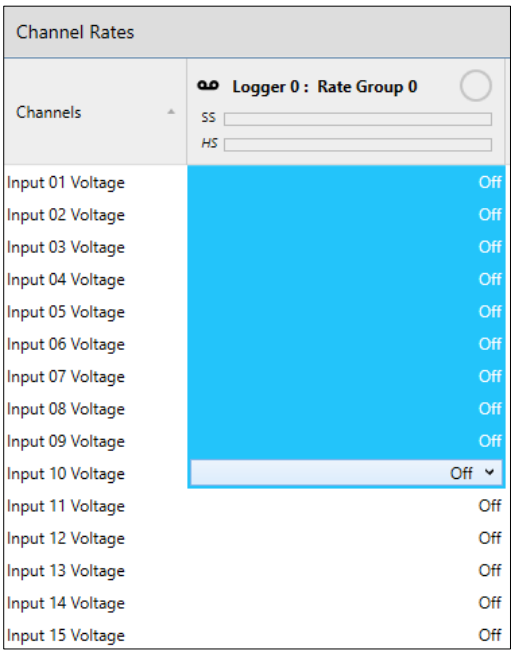



You can also combine multiple terms to search for multiple parts of a channel name in a single search.



### Set multiple channel rates

To define the same rate for multiple channels, select the first channel in the list and shift-click the last channel in the list to highlight all channels in between. Select a logging rate for one of the channels highlighted to apply it to all selected channels.

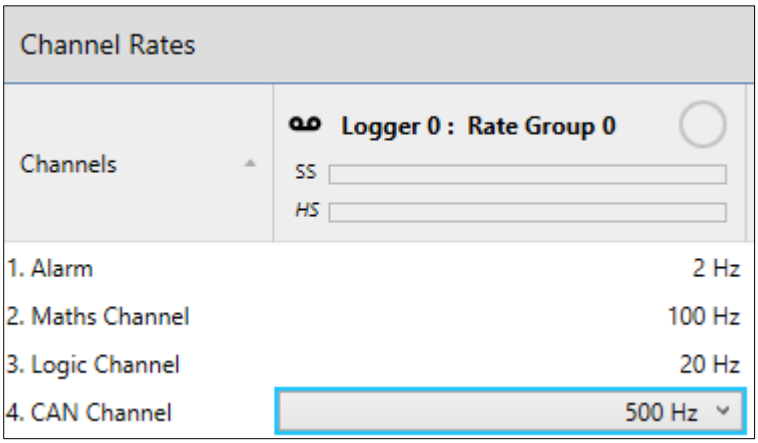



### Channel origins

Like the Resources menu, the channel rates list is an easy method to locate and find the origin of each channel. Double-click on a channel name to automatically direct you to the origin of the channel, where you can make edits.

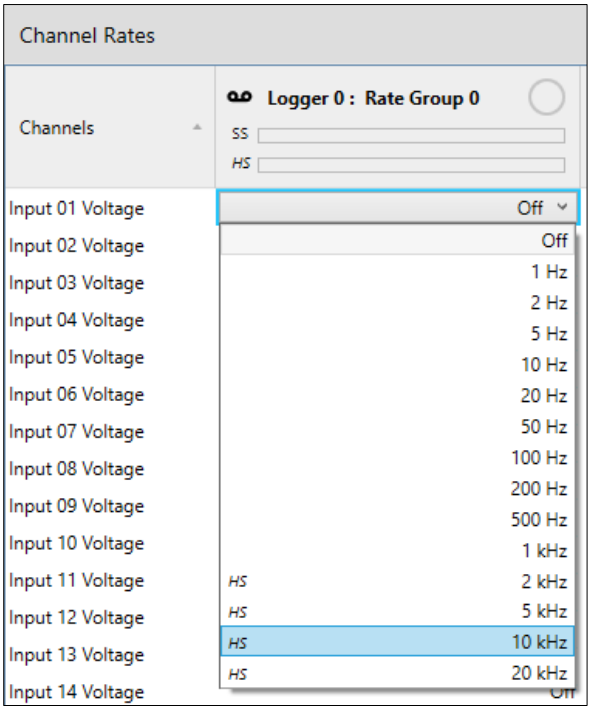
### Channel rates from non-native sources

You can configure the logging rate for channels originating from Alarms, Maths channels, Logic channels, and CAN channels on the Channel Rates node.



### High speed logging

On devices such as the Antares, Badenia and Centaurus with high-speed logging capability, the rates can be set for High Speed inputs. High speed rates are marked ‘HS’.



# 16. Displays overview

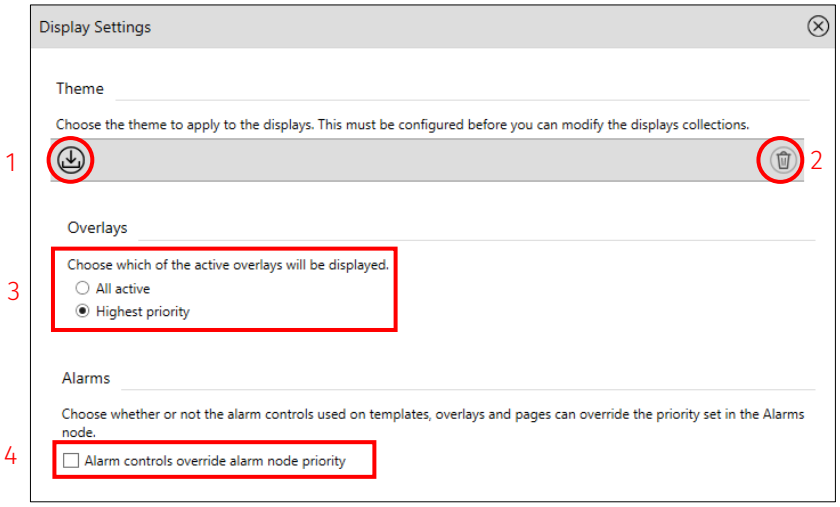
The **Displays** node is used to configure what is shown on display devices such as the CDU range and the CCW Mk2 and CCW Mk3. Toolset offers flexible display customisation with possible displays only limited by your imagination.

## 16.1. Get started with displays

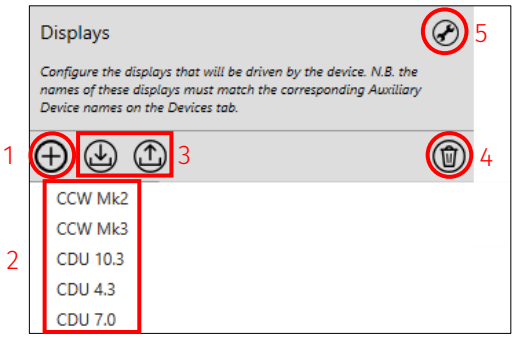
### Add a display

When you add a display for the first time you are prompted to select a display theme from the **Display Settings** menu.

Use the 'import' tool to import a new display theme (1). You can use the 'bin' tool to delete a theme (2). Select an overlay to display all active overlays or display the highest priority overlay (see Overlays) (3). You can also select whether an alarm control within a display can override the priority order of alarms on the **Alarms** node (See **Templates – Alarm Controls**) (4).




To add a new display, click the **+** tool (1), and then select the required display from the dropdown menu (2). You can import and export display configurations between existing setups (3). Use the 'bin' tool to delete unwanted configurations (4). Use the wrench tool to open display settings (5).

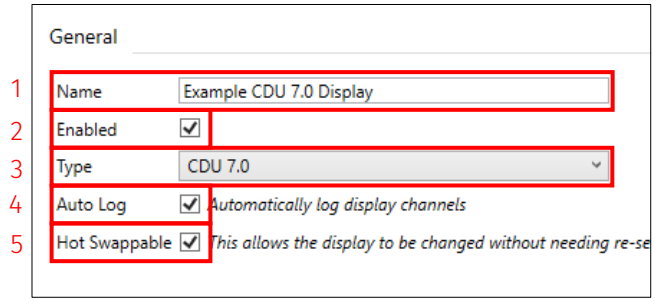


### General display configuration

You can name the display configuration (1), enable/disable the display without deleting the display from the setup (2.), and select the type of display (3). For example, if you are only using a CDU 7.0, then this option is not used. However, if a display is configured for a CDU 7.0, but you need to change the display type to a CCW Mk3, select CCW Mk3 from the **Type** menu to change the configuration with no other changes required.

You can set any channels used on a display page to automatically log, if they are not already assigned a logging rate (4). When a display channel is automatically logged, a new logging table is automatically generated, and the display channels are assigned a base logging rate that you cannot edit. This feature is useful for logging display channel items to help when diagnosing display configuration issues.

You can select whether the display configuration is 'hot swappable', which allows you to change the display in a car without re-sending the setup (5). This feature is useful if a steering wheel display such as the CCW Mk3, with different hand grips for different drivers, needs to be swapped during a driver change without the need to re-send the setup.

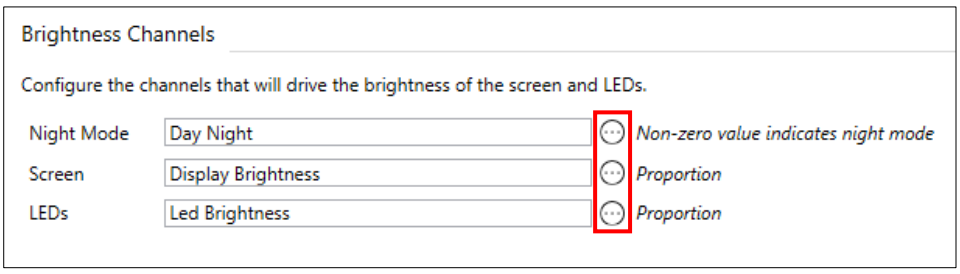



### Display brightness channels

You can configure the channels for 'Night Mode', screen brightness, and LED brightness in the **Brightness Channels** section. Use the 'browse' tool to select from the available channels. These channels can be Maths Channel, Logic Channels, CAN channels, Sensors, or Buttons.

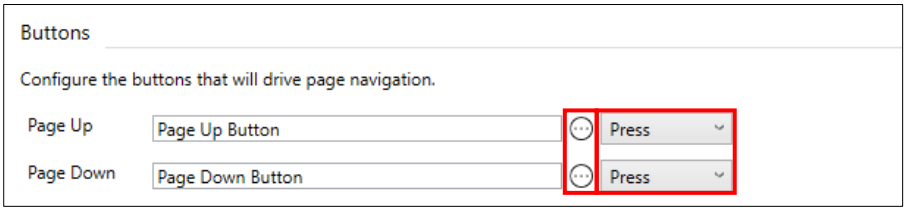
Night mode is a 'digital' channel where '0' is night mode OFF and, '1' (or above) is night mode ON.

Screen and LED brightness are controlled by 'Proportion' channels, where no display is '0' and full brightness is '100' (or above).



### Display page navigation

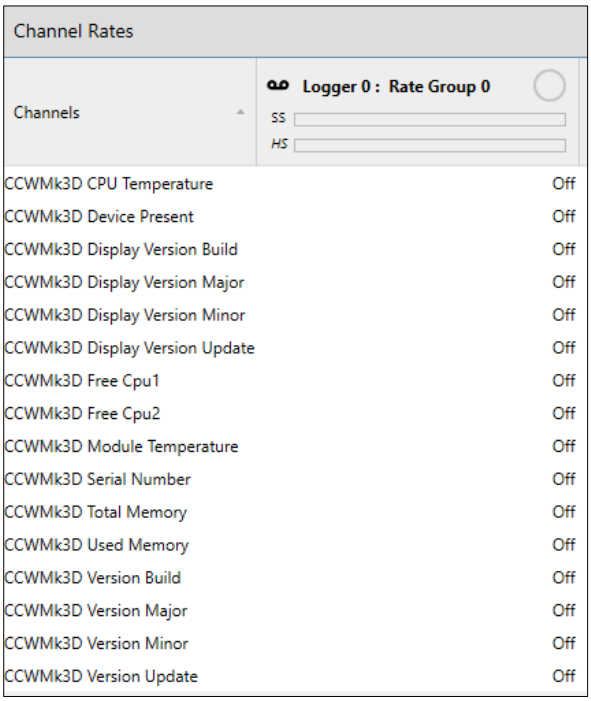
When multiple pages are added to a display, you must configure buttons to select to move up and down between pages (see Setups – Buttons). This can be a digital button input, a rotary switch, or a virtual button via CAN. Click the 'browse' tool to select from the available buttons (1). You can select the button press type (Click, Held, and so on) from the dropdown menus (2).




### Display generated channels

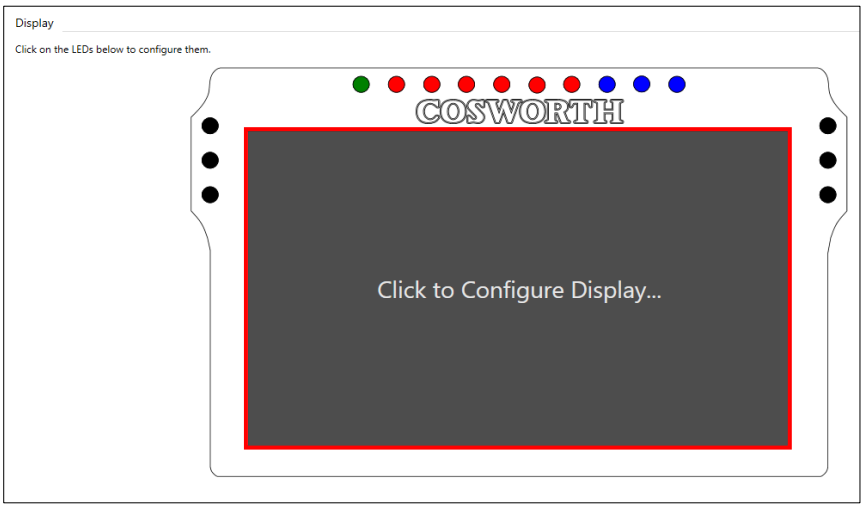
When a display is added (such as the CDU10.3 or CCW Mk3), display channels are automatically generated. These channels include device information such as temperatures, software versions, free CPU, and serial number.

Note: These channels are not automatically logged. You must set logging rates on the Channel Rate node to capture and offload these channels.



# 17. Configure a display

Click on the centre of the display to configure the display.

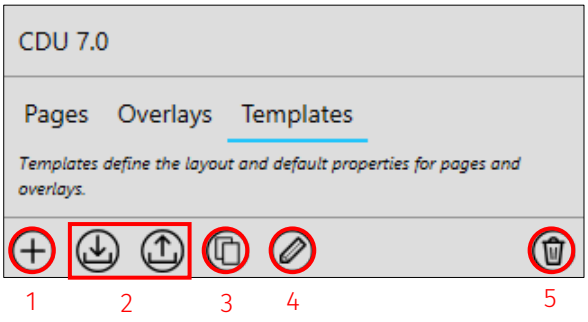



## 17.1. Templates

### Get started with templates

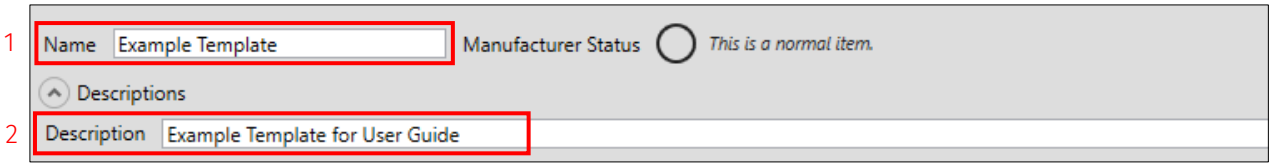
You must first create a display template. All pages and overlays are derived from templates. It is recommended that you to create an individual template for each page/overlay and to make changes to each template rather than to make changes to individual pages/overlays using the same template. This makes sure that each page reflects its own template to help diagnose any configuration issues.

Click the **+** tool (1) to add a new template. You can import and export templates between existing setups (2), either individually, or multiply as part of a group. Use the 'duplicate' tool (3) to duplicate a template. This can save time when creating multiple similar templates. Use the 'pencil' tool to edit templates (4). You can delete templates using the 'bin' tool (5).

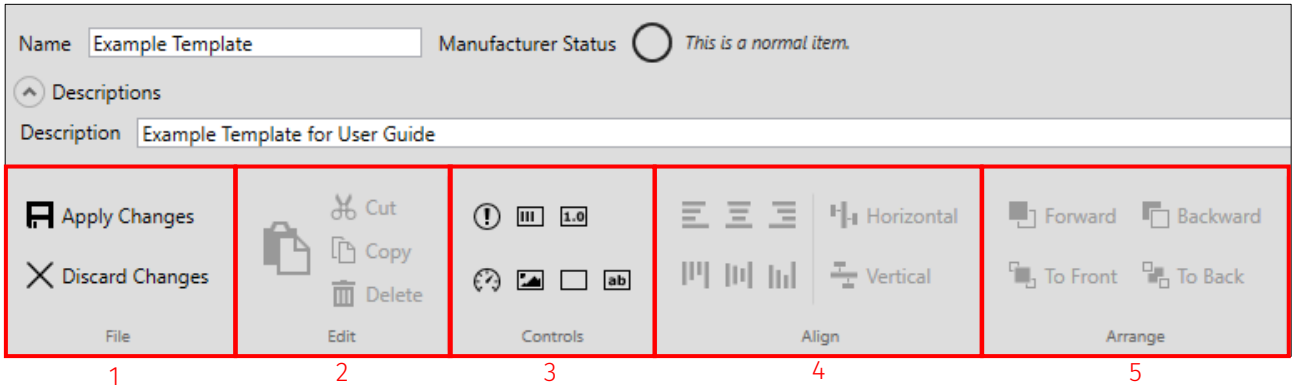



### Template editor

When a new template is added the template editor window is displayed. You can name the template (1) and add a brief description (2).



At the top of the template editor window is the template editor menu where you can select functions to configure the template.




1. File

- Apply Changes: Saves your changes.
- Discard Changes: Discards your changes.

2. Edit

- Cut/Paste/Copy/Delete: Click on the required option.

3. Controls

More information about display controls is available in the Controls section.

- Alarm: Use to display an alarm box. This box is only displayed when a specified alarm configured on the Alarms node is active.
- Bar: This control is used to display a channel value as bar.
- Channel: This control is used to display a channel value in a box.
- Dial: This control is used to display a channel value on a dial.
- Image: This control is used to display an Image. Takes a channel as an input and can change images based on the channel value.
- Rectangle: This control is used to display a rectangle. Useful for more efficient CPU usage compared to using an image. Takes no channel inputs.
- Textbox: This control is used to display a textbox. Displays user-defined text.

4. Align

When you select two or more controls (by clicking CTRL and control items), the Align menu is displayed. Use these to align controls on the display to the left, right, and centre of another control, and to space controls in the horizontal and vertical directions.

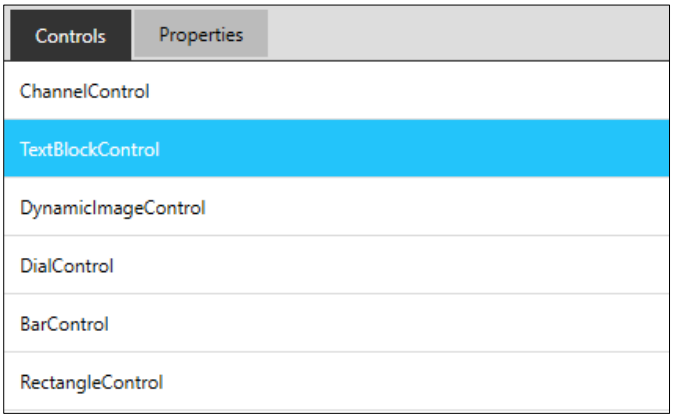
The click order of items dictates the alignment direction. In other words, if control 1 is clicked first, then control 2 and 3, controls 2 and 3 align relative to the position of control 1.

### 5. Arrange

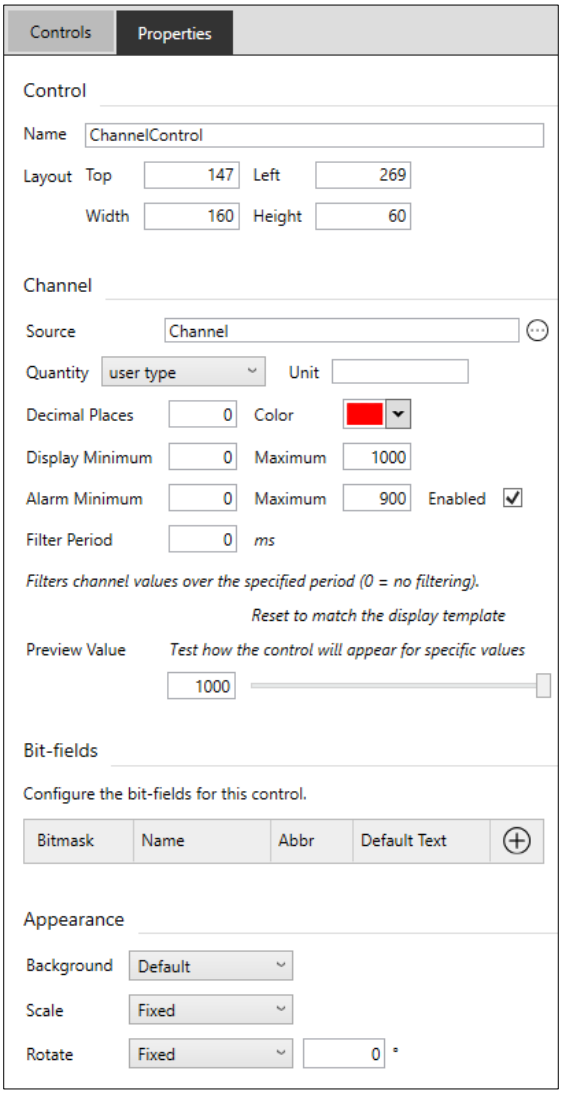
Used to layer controls and to bring a control forwards and backwards relative to each other on the display.

#### Controls list and properties

The **Controls** list allows you to visualise all the display controls on a template as a list and select controls here rather than from the template. This allows for easier selection of controls.



When you select a control, you can use the **Properties** menu to configure the control name, position, channel input, units, decimal places, displayed text, colour, size, and so on.



## 17.2. Controls

The properties of controls are mainly common between control types, as indicated below.

### Alarm

This control is used to display an alarm box. This box is only displayed when a specified alarm configured on the **Alarms** node is active.

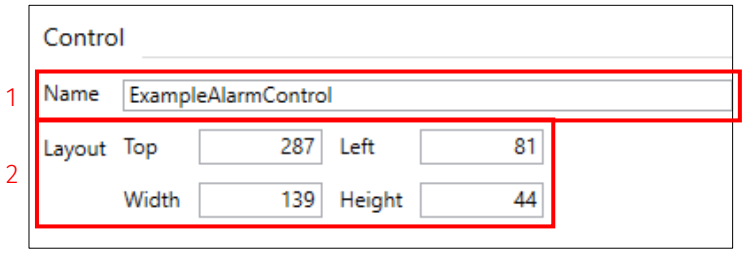
Add an alarm control (common)

To add an alarm control, drag and drop the alarm control from the **Controls** section of the **Template Editor** menu onto the page. By default, the box is blank. Click, drag, and drop to move the display. Click the tabs on the edges and corners of the control to resize it.

Click on the alarm control on the display, or from the **Controls** list menu, to edit the control **Properties**.

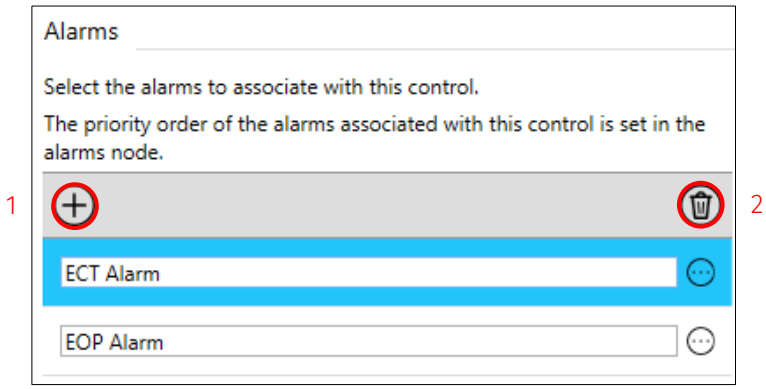
Control (common)

You can define the name of the control (1) and use the **Layout** options to enter values for the position and size of the control (2).




Alarms (alarm specific)

You can then add configured alarm channels from the **Alarms** node to the alarm control (1). You can add multiple alarms to the same alarm control. Use the 'bin' tool (2) to delete unwanted alarm controls.




**Note**: The priority order of the displayed alarms is derived from the hierarchy set in on the **Alarms** node, unless the **Alarm control priority overrides alarm node priority** checkbox is selected in the display settings.

Appearance (common)

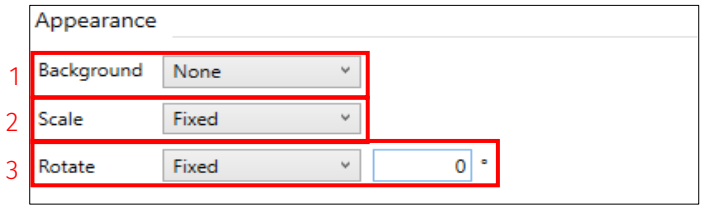
You can set the control background (1) to one of four options:

- **Default** – Black to silver gradient
- **None** – Transparent/No background
- **Fixed** – Select an RGB colour
- **Channel Value** – Dynamic RGB colour driven from the Maths Channel (see **Maths Channels – Color Control**)

You can set the scale (size) of a control to a fixed size specified by the control width and height or use the **Scale** option to set the size with units of proportion (2). The channel value scale resizes the control between its original size (100%) and shrinks the control down to hidden (0%), based on the channel value.

You can set the control to a user-defined fixed angle of rotation or dynamically rotate it through an angle based on a channel value (3). The value must be in units of angle and rotates the control through 0°-360° based on the channel value input.

**Note**: The channel value input can be in radians and the conversion to degrees takes place automatically.

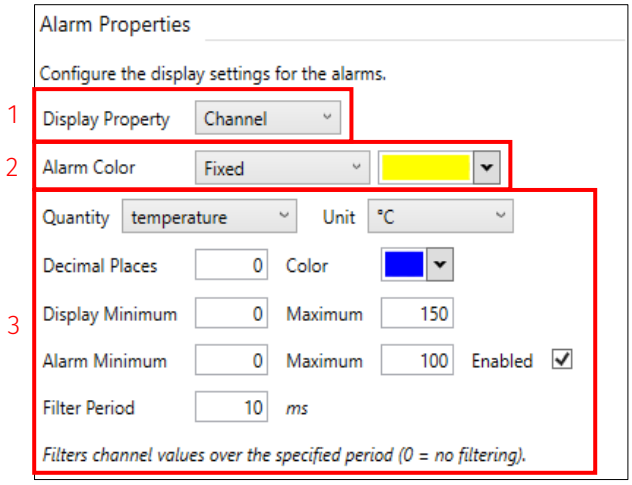



Channel / Alarm Properties (common)

The **Alarm Properties** section is where you configure the displayed information for an alarm control. The **Display Property** can either be the Short Text or Long Text configured on the **Alarms** node, or a channel value. (1)

You can set the **Alarm Color** for a triggered alarm to default (red) or user-defined fixed colour (2).

If 'Channel' is selected as the **Display Property**, then you can configure the channel units, decimals places, minimum and maximum values, and alarm trigger. You can apply a filter to remove noise from the channel that can cause the alarm control to trigger (3).



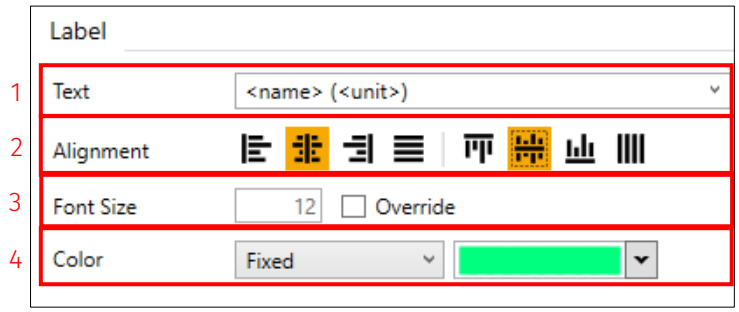
Label (common)

This is where you configure the label for the control. The **Text** box is used to configure the displayed label for the Control (1). You can insert text manually for a user-defined name or it can use <name> to display the channel name and <unit> to display the unit. Delete all text to display no label.

Use the alignment tools to align the label to the left, right, centre, top, middle, or bottom of the control (2).

The default font size is set to 12. Select **Override** to set the font size manually (3).

The label colour is set to white by default (using the default theme), but you can select the label colour (select fixed and an RGB colour). Select a Maths Channel using colour names (Channel) or a Maths Channel using an RGB value (Channel Value) to dynamically control the label colour (4).



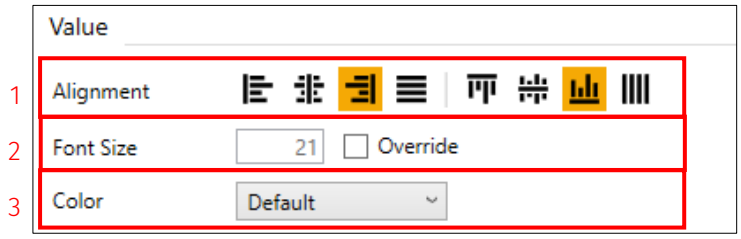

Value (common)

This is where you configure the value of the control.

Use the alignment tools to align the value to the left, right, centre, top, middle, or bottom of the control (1).

The default font size is set to 21. Select **Override** to set the font size manually (2).

The label colour is set to white by default (using the default theme), but you can select the label colour (select fixed and an RGB colour). Select a Maths Channel using colour names (Channel) or a Maths Channel using an RGB value (Channel Value) to dynamically control the label colour (3).

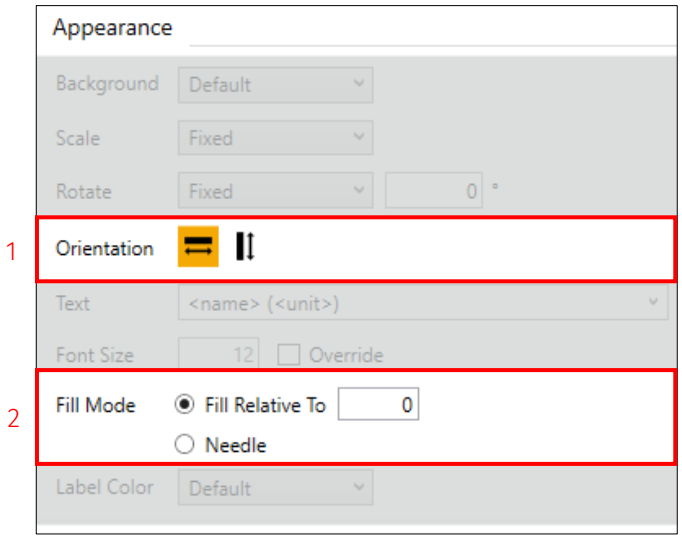


**Bar**

This control is used to display a channel value as bar.

Appearance (bar specific)

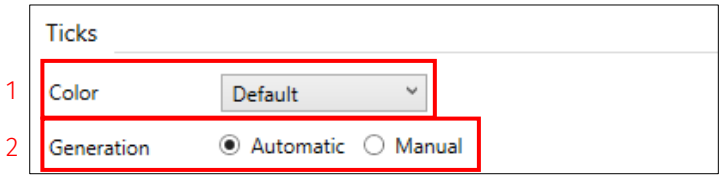
Under the **Appearance** section, you can set the orientation of the bar control to horizontal or vertical (1). The bar control can also be set to be display as a 'fill up to' value. Use the **Fill Relative To** option to set a value. Alternatively, you can configure the bar to display as a floating 'needle' over the current value (2).



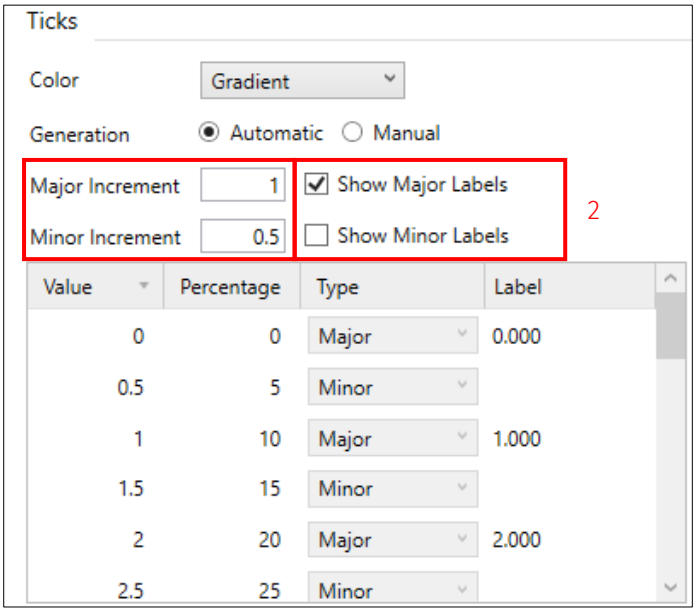

Ticks (common)

Here you can configure the tick markers for the channel value. You can select the colour of the ticks from five options: Default, Fixed, Channel, Channel Value, or Gradient (see the Gradient section) (1).

You can select whether the tick mark breakpoints are generated automatically or manually. Select either the 'Automatic' or 'Manual' Generation tick boxes (2).

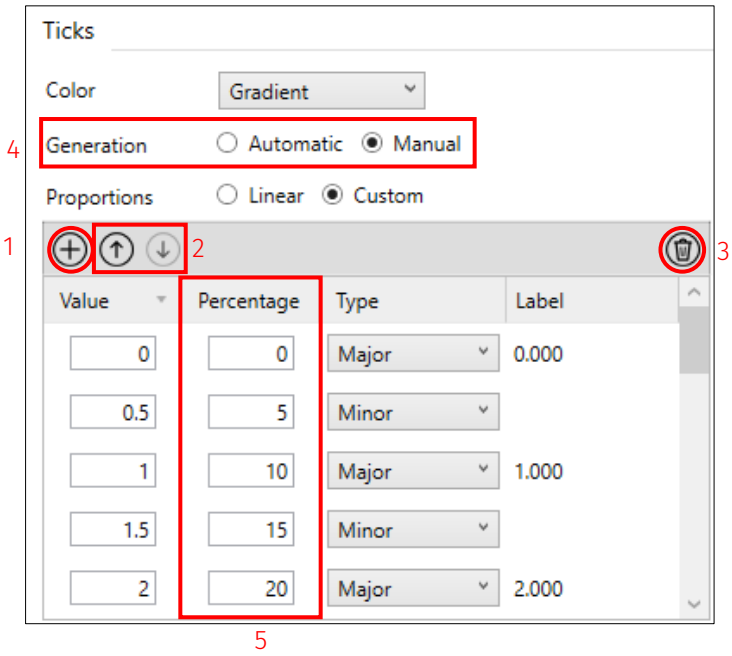


If automatic generation is selected, you can configure the major and minor breakpoints (1) and select whether to display major and/or minor tick labels (2) The User can rename the automatically generated label names, but cannot change the value, percentage or tick type.




If manual generation is selected, the User can configure the tick mark breakpoints. Use the **+** tool to add tick breakpoints. Reorder them using the reorder tools (2). Use the 'bin' tool to delete unwanted breakpoints (3).

You can select linear incremental proportions or custom incremental proportions (4). Selecting **Linear** prevents the percentage values from being configured. Selecting **Custom** allows you to configure the percentage values (5). You can adjust the tick mark value, type, and label via the text boxes.



You can select to 'snap' values from four modes:  Off, Closest, Floor, or Ceiling via the check boxes:

- **Off** – Tick mark displays the current channel value
- **Closest** – Tick mark rounds up or down to the nearest whole number
- **Floor** – Tick mark rounds down to the nearest whole number
- **Ceiling** – Tick mark rounds up to the nearest whole number

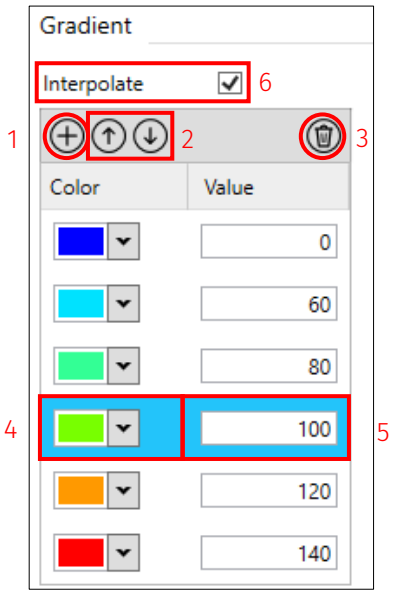

Gradient - Common

The Gradient function is used to set a colour gradient for the control. For example, this is useful for controls displaying a channel like temperature, where the working window can be shown in green, too hot could be shown in red and too cold shown in blue.

Click **+** to add a new gradient break point (1).  Use the reorder tools to reorder breakpoints (2). Use the 'bin' tool to delete gradient breakpoints (3)

Select the colour and value for each breakpoint from the dropdown menu (4) and enter a value in the **Value** boxes. (5)

A colour gradient is displayed on the control as discrete breakpoints. Alternatively, select the **Interpolate** check box to blend colours between breakpoints (6).

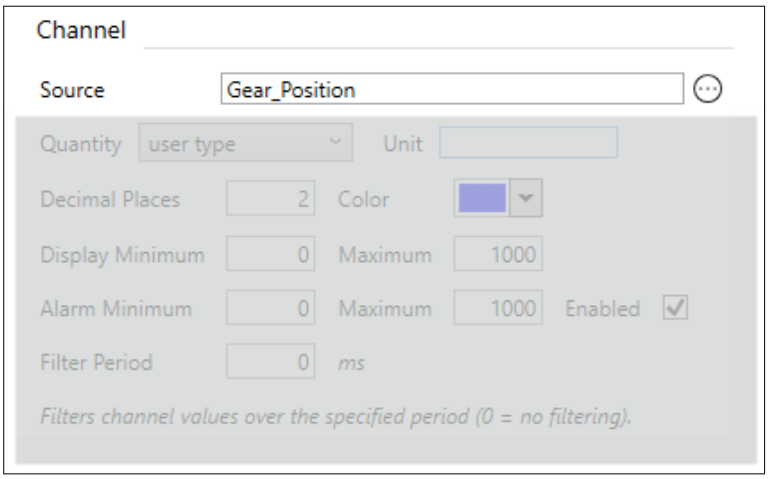


**Channel**

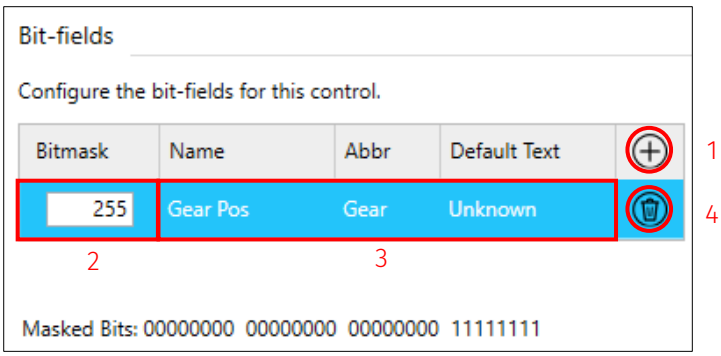
This control is used to display a channel value in a box and supports both standard channel values and bit-fields.

Bit-Fields – Channel specific

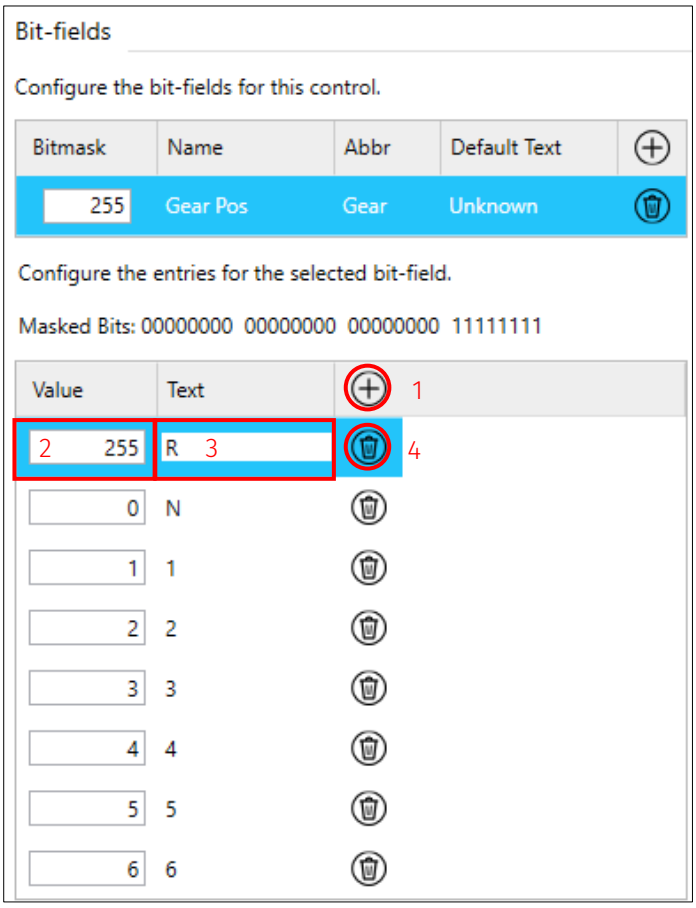
To configure a Bit-field to display, select the required input channel in the **Channel** section.




In the **Bit-fields** section, add a new bitmask (1) and set a value for it. This determines which bits are on/being monitored (masking on) (2). You can enter a name and abbreviated name for the bitmask and the default text to be displayed if a bit value input does not have an enumerated status (3). Use the 'bin' tool to delete an unwanted bitmask (4).



Next set the enumeration of the bitfield channel. Use the **+** tool to add a new bitfield entry (1), set the bit value (2), and then define the text displayed when that value is active (3). Use the 'bin' tool to delete a bitfield entry (4).

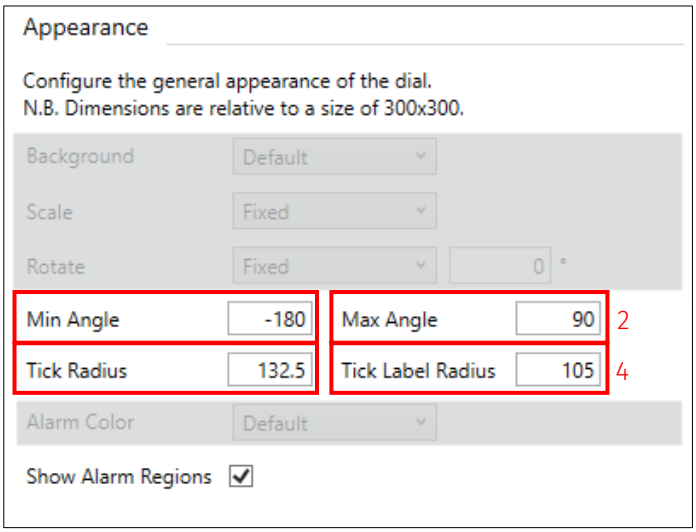


**Dial**

This control is used to display a channel value on a dial.

Appearance – Dial specific

The dial sweep angle and sweep radius is configurable. Use the **Min Angle** option to enter the starting angle for the dial start angle (1) and then use the **Max Angle** option to enter the dial end angle (2). Use the **Tick Radius** option to define the position of the tick marks (3.) and the **Tick Label Radius** option to position the labels on the dial (4). If an alarm value and colour is configured under the **Control Channel** section, then alarm regions can be displayed on the dial (5).



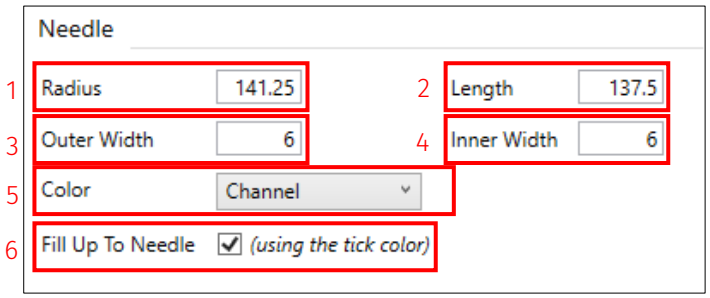

Ticks – Dial specific

Tick labels are displayed horizontally by default. Select the **Rotate Labels** check box to force the tick labels to follow the curvature of the dial

You can configure the needle size. The needle radius is the distance to outer tip of needle (1). The needle length the length from outermost tip to the centre (2). This allows you to configure a short needle that hovers over the ticks, rather than a needle that originates from the dial centre.
The outer width is the width of the needle tip (3) and the inner width is the width at the centremost tip (4).

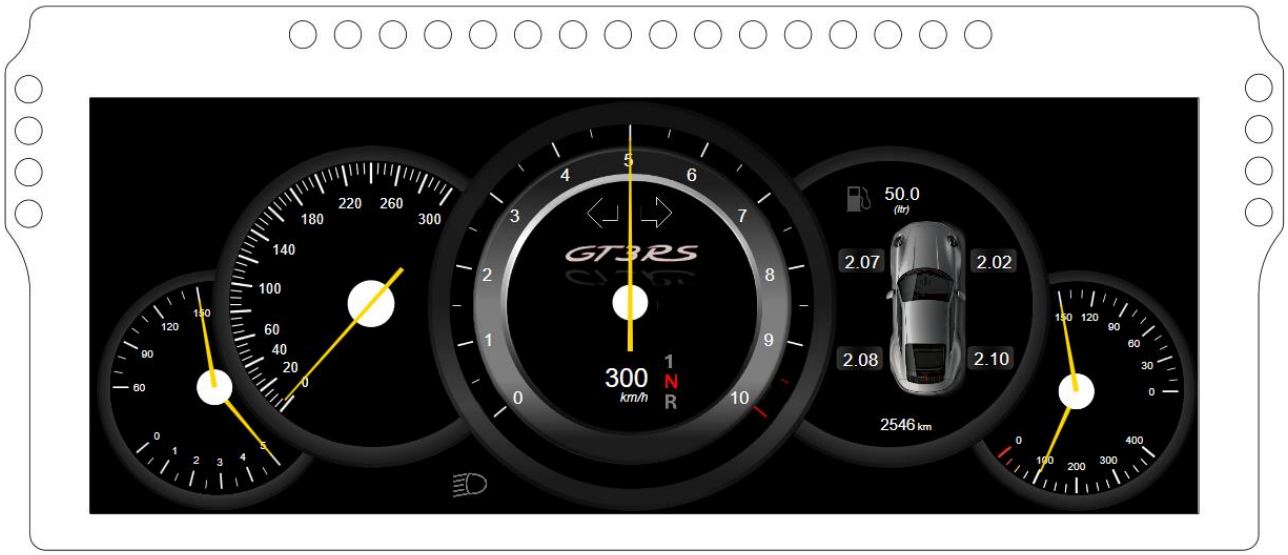
You can set the needle colour to be fixed or dynamic from a channel or gradient. Select from the following options: Fixed, Channel, Channel Value, or Gradient (5).

Use **The Fill Up To Needle** option to create a unique look for the dial control (6). The fill colour matches the needle colour. For example, you can create a dial control for RPM with a needle that dynamically changes colour from green to red as it approaches the RPM limit. You can use the fill as a shift light to create a unique appearance.



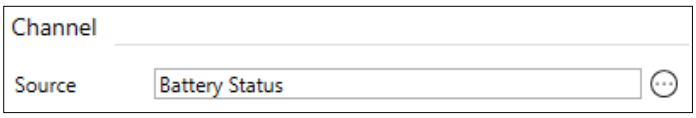

**Image**

This control is used to display an Image. It takes a channel as an input and changes images based on the channel value. For example, battery voltage status can be displayed as an image driven from a battery status channel. Images can be used to make more interesting display backgrounds and to create 3D effects.



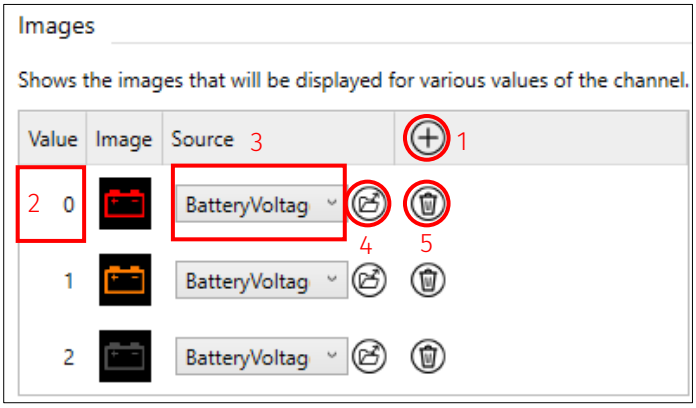
Images – Image specific

Select the channel to drive the image selection in the channel under the **Channel** section




In the **Images** section use the **+** tool to add a new image entry (1). Set the channel value for the required image (2), and then select the image to be displayed (3). A range of default images is available to select from the dropdown menu, but you can import your own .PNG and .GIF images using the browse tool (4). Use the 'bin' tool to delete an image entry (5).

**Allow Sending**: When this option is selected, you can send an image on the **Actions** tab to replace the current image.




**Rectangle**

This control is used to display a rectangle. Rectangles can create coloured borders, banners, and backgrounds since they are more efficient on CPU usage compared to using an image. The Rectangle control requires no channel inputs.

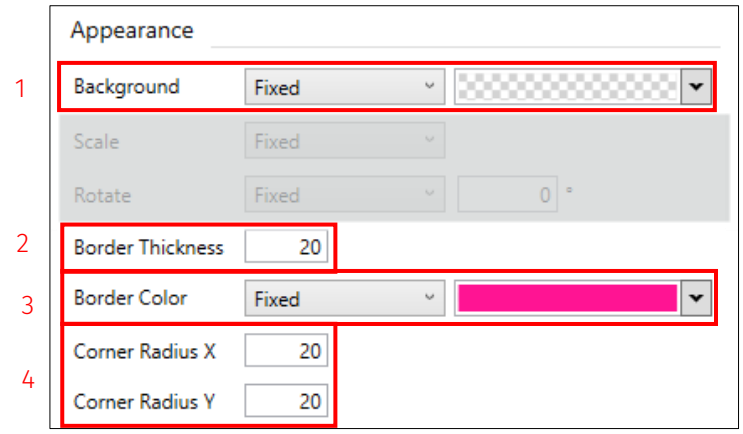
Appearance – Rectangle specific

The rectangle background (1) can be set to one of four options: Default, Fixed, Channel or Channel Value

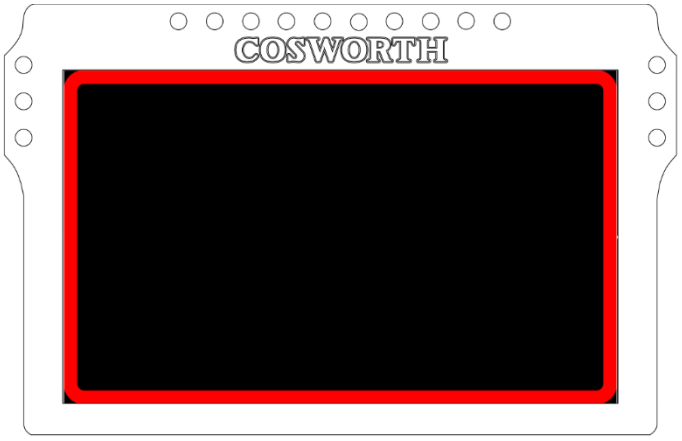
- **Default** – Transparent/No background
- **Fixed** – Select an RGB colour
- **Channel** – Dynamic RGB colour driven from a Maths Channel with named colours (see Maths Channels – Color Control)
- **Channel Value** – Dynamic RGB colour driven from Maths Channel with RGB values (see Maths Channels – Color Control)

The border thickness (2) and border colour can also be selected as Default, Fixed, Channel, or Channel Value (3).

The corners of the rectangle can be radiused. Configure the radius using the X and Y boxes (4).

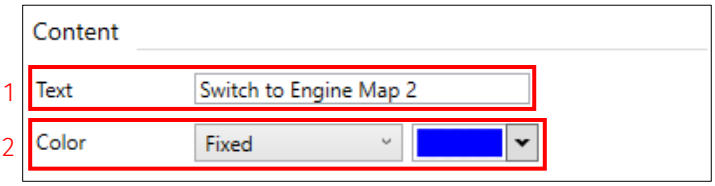



For example, you can use a rectangle control for a warning overlay to make the warning stand out for the driver.

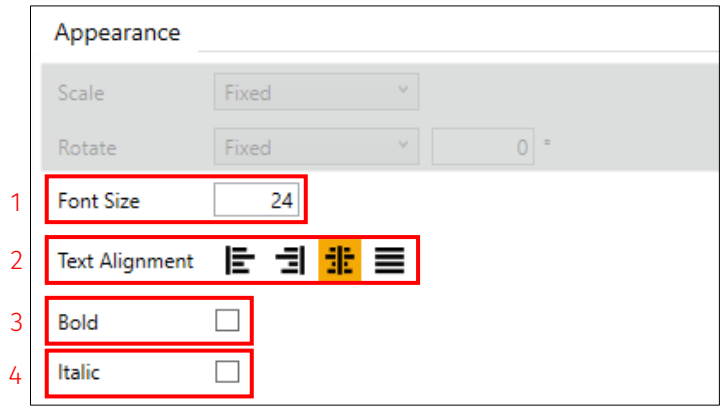



## Text

The **Text** control is used to display a textbox with your defined text. This allows for messages and instructions to be displayed in the content box (1). The colour of the text can be to one of four options: Default, Fixed, Channel or Channel Value (2).



You can change the font size (1), set the alignment to left, centre, right, or justify (2) and make the text bold (3) or italic (4).




## 17.3. Save a template

You must save configuration changes for a template before you can add the template to a page. Use the 'Apply Changes' tool to save any changes and the 'Discard Changes' tool to discard any changes.

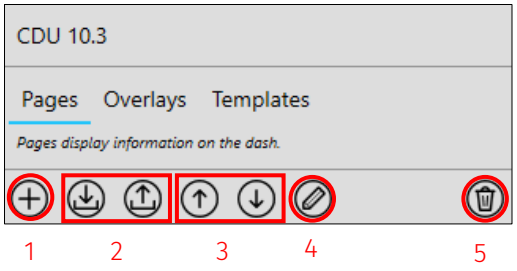
You must apply any changes to a template before you can send the setup. If a setup has unapplied changes an error is raised and displayed in the **Errors** menu.

## 17.4. Add a page

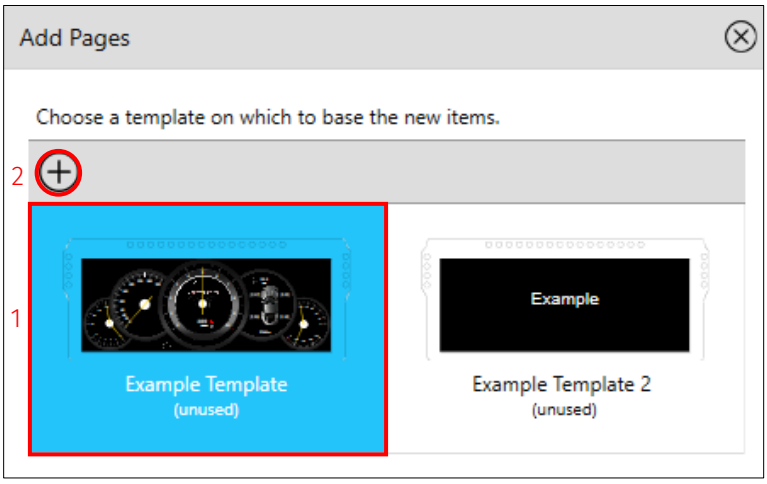
To add a new display page, navigate to the **Pages** tab and click the **+** button (1). Use the 'import' and export' tools to import and export pages from existing setups (2). You can import and export multiple display pages as groups for convenience.

Use the 'reorder' tools to sort the ordering of pages (3). The page hierarchy in the dash is ordered with the highest priority page at the top of the list and the lowest priority page at the bottom. When a new page is added, it is placed at the bottom of the list and lowest in the hierarchy. Use the 'reorder' tools to rearrange the list of pages to change the relative position of each individual page within the overall display hierarchy.

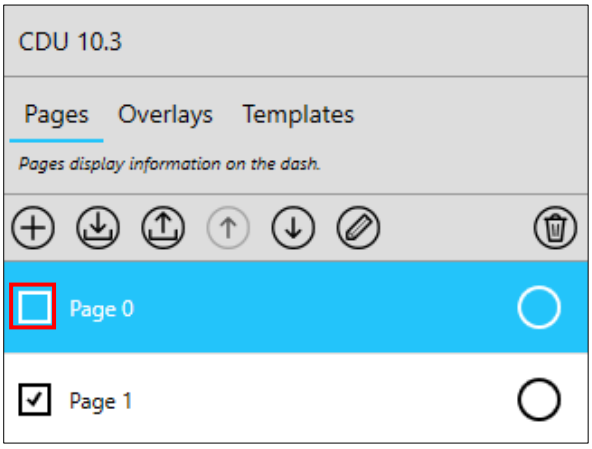
Use the 'pencil' tool to open the display page properties (4) (see Display properties and qualifiers). Use the 'bin' tool to delete an unwanted display page (5).



When you add a new page, you are prompted to select a template from the available templates menu. Select a template (1), and then click the **+** tool (2), or double-click a template to add it to the display page.




When you add pages, the tick boxes next to each page in the **Pages** menu allow you to enable or disable pages without having to delete the page from the setup. Select the tick box the enable or disable the display page. Any page that is unselected is not displayed until the setup is re-sent with the 'Enable' box selected.




**Display properties and qualifiers**

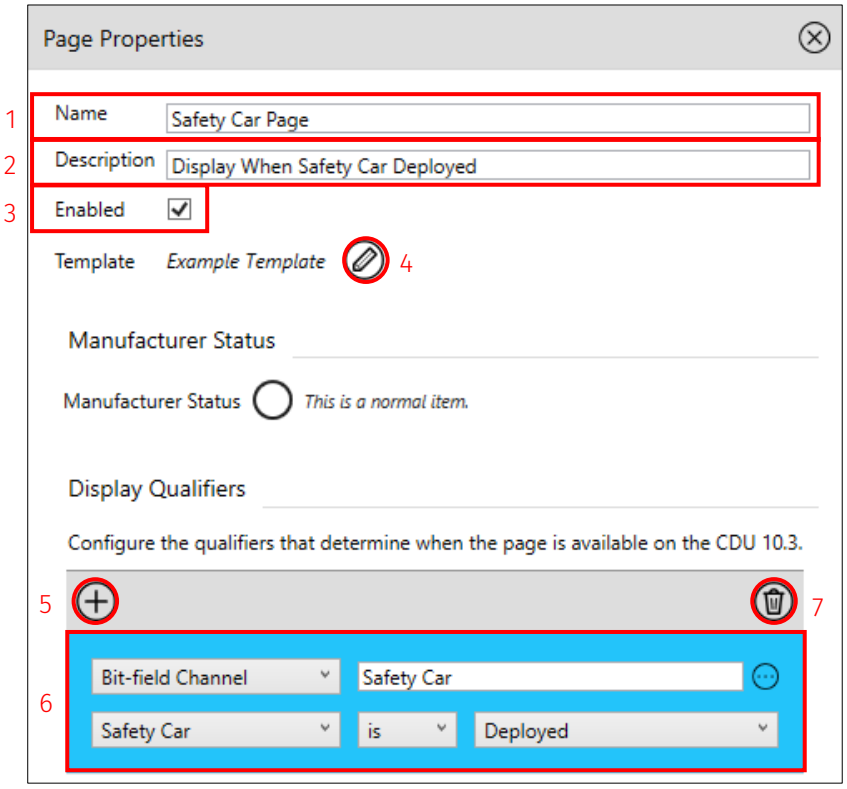
In the **Page Properties** menu, you can name the page (1) and add an optional description (2). You can also enable or disable a page (3).

Click the 'pencil' tool to change the template for the page (4).

You can define qualifiers for each page to determine when the page is made available to view on the display. If the qualifying conditions are met, the page is made available to view. Otherwise, the page is hidden and is not available to be displayed.

If multiple pages are enabled the device decides which page to display by first choosing which pages have their qualifying conditions met and then displaying the highest page in the hierarchy.

Click the + tool to add a new qualifier (5). You can add multiple qualifiers can be added. Select the channel type (bit-field, channel, or strategy) to configure the qualifying conditions from the available channels and define the condition to display when true (6). You can use the 'bin' tool to delete unwanted qualifiers (7).




## 17.5. Add an overlay

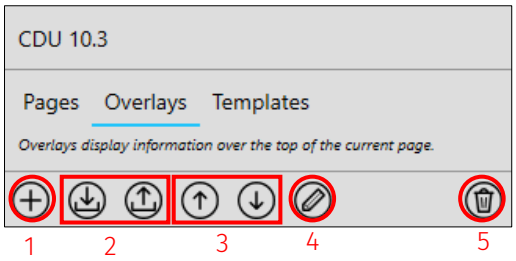
Overlays are like a page but can be used to display information over the top of the current displayed page. Unlike pages, multiple overlays can be displayed at the same time.

Overlays are used to create small, temporary items such as dialog boxes, displaying text instructions such as 'Change engine map' or information such as 'Pit Limiter Active'. Overlays are created and added in the same way as pages by using templates.

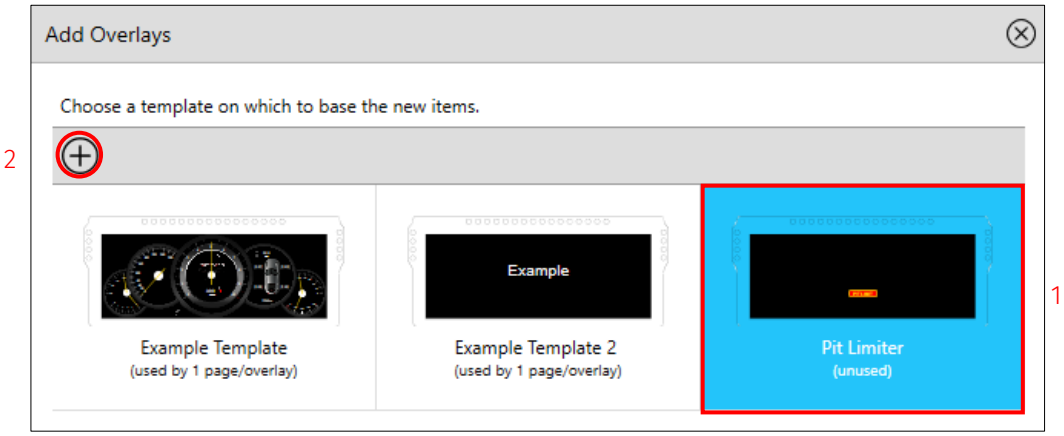
To add a new overlay, navigate to the Overlays tab and click the + button (1.). Use the 'import' and 'export' tools to import or export overlays from existing setups using the import and export tools (2). You can import and export overlays as groups for more efficient setup changes.

Use the 'reorder' tools to sort the order of overlays (3). The page hierarchy in the dash is ordered with the highest priority page at the top of the list and the lowest priority page at the bottom. When a new overlay is added, it is placed at the bottom of the list and lowest in the hierarchy. You can rearrange the list of overlays to change their relative positions in the overall display hierarchy with the 'reorder' tools.

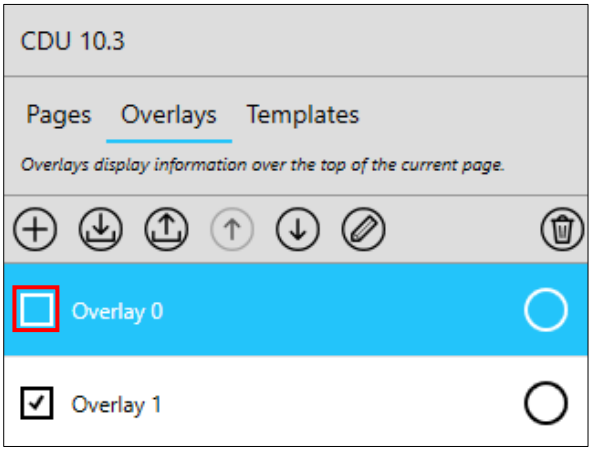
Use the 'pencil' tool to open Overlay Properties (4) (see Overlay properties, qualifiers and alarms). Use the 'bin' tool to delete unwanted overlays (5).




When a new overlay is added, you are prompted to select a template from the available templates menu. Click a template (1), and then click the **+** tool (2), or simply double-click a template, to add the template to the **Overlay** page.




When overlays are added, the tick boxes next to each page in the overlays **Pages** menu allow you to quickly enable or disable overlays from a setup without having to delete the entire overlay. Select the box to enable or disable the overlay. Any unselected overlay is not displayed until the setup is re-sent with the 'Enable' box selected.



**Overlay properties, qualifiers and alarms**

In the **Overlay Properties** menu, you can name the overlay (1) and add an optional description (2). You can also enable or disable the overlay from this menu (3). Click the 'pencil' tool to change the template used for the overlay (4).

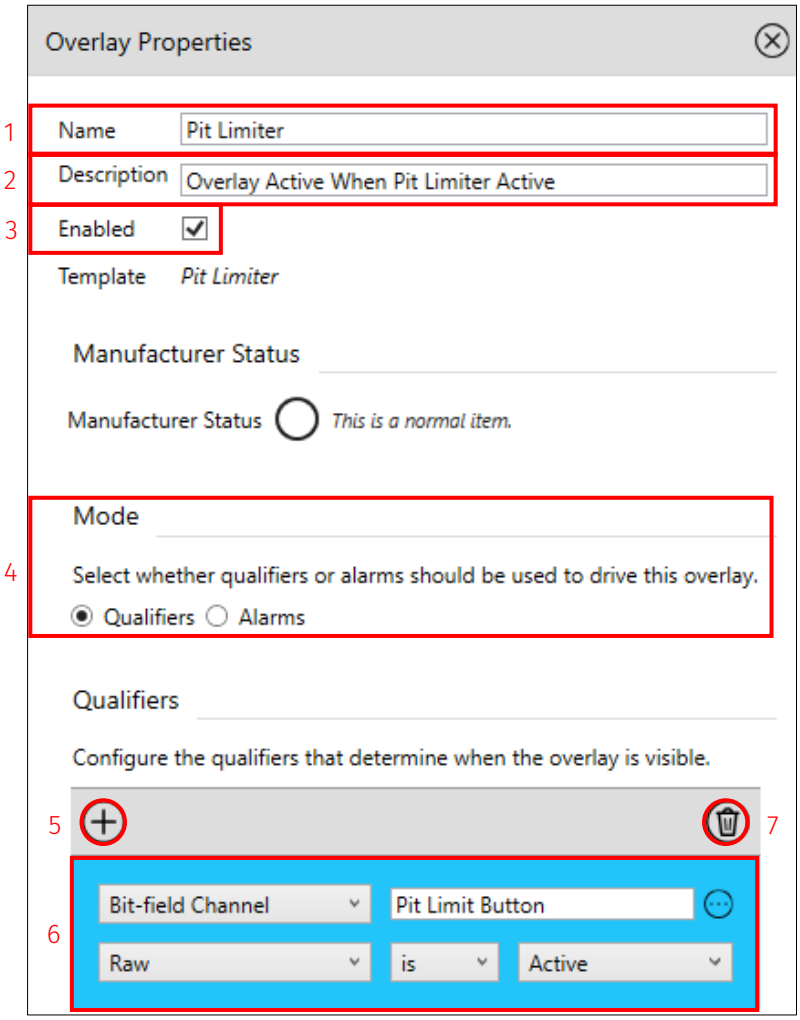
You can define qualifiers for each overlay to determine when the page is made available to view on the display. If the qualifying conditions are met, the page is available to view. Otherwise, the page is hidden and is not displayed.

You can also use alarms to drive overlays. When an alarm is triggered, the overlay is displayed.

Select **Qualifiers** or **Alarms** for the required overlay mode (4).

All overlays with their qualifying conditions or alarm triggers active are displayed.

Click the **+** tool to add a new qualifier or alarm (5). You can add multiple qualifiers or alarms. Select the channel type (bit-field, channel or strategy), by browsing from the available channels and define the condition to display when true (6). If using alarms, select the required alarm(s) from the 'browse' menu. You can delete unwanted qualifiers with the 'bin' tool (7).



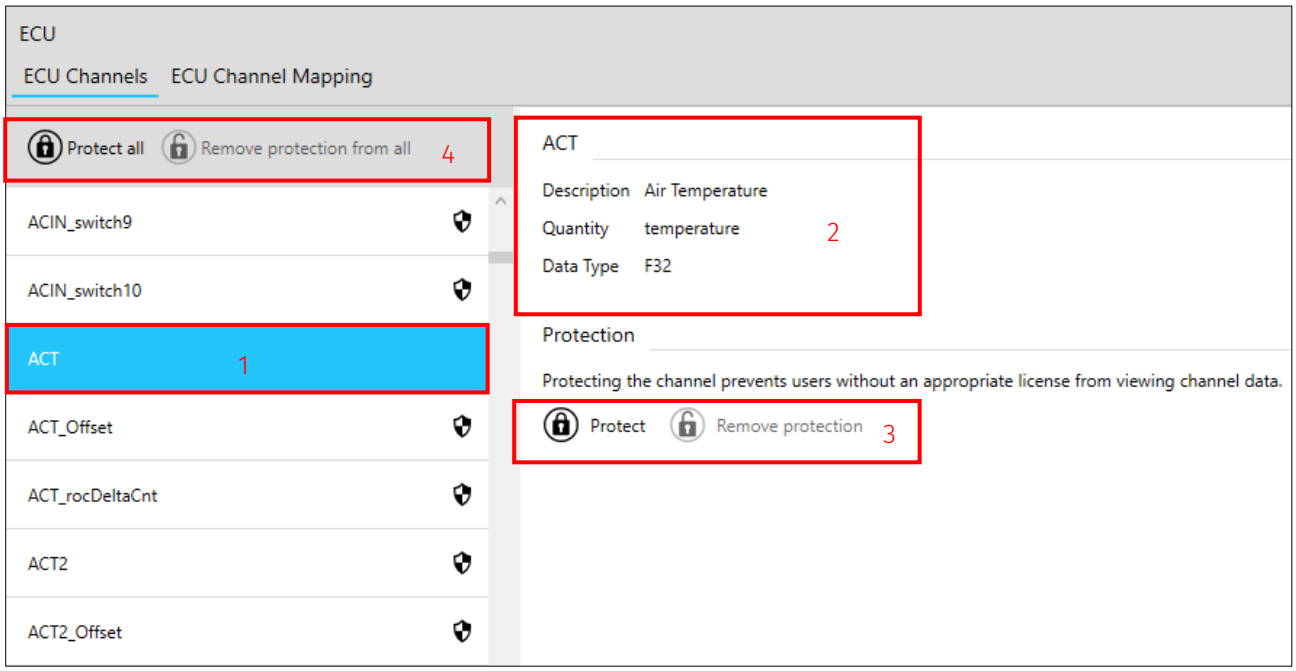
# 18. ECU

The **ECU** node is available in **Setups** for Cosworth ECU devices running CHP 2 (Metadata 3+). The **ECU** node is the interface for the exchange of channels between the ECU side (also known as the Pectel side) of the device and the logger side (also known as the Common Hardware Platform (CHP) side).

## 18.1. ECU channels

The **ECU Channels** menu is prepopulated with all the channels from the ECU side in the list of channels to be exchanged in the setup metadata. Click on the list of channels in the **ECU Channels** menu (1) to see a brief description about the channel (2). The setup owner (Manufacturer) can select whether to apply channel protection (see Setup Locking – Channel Protection) (3). For convenience, there is the option to apply and remove channel protection from all ECU channels (4).

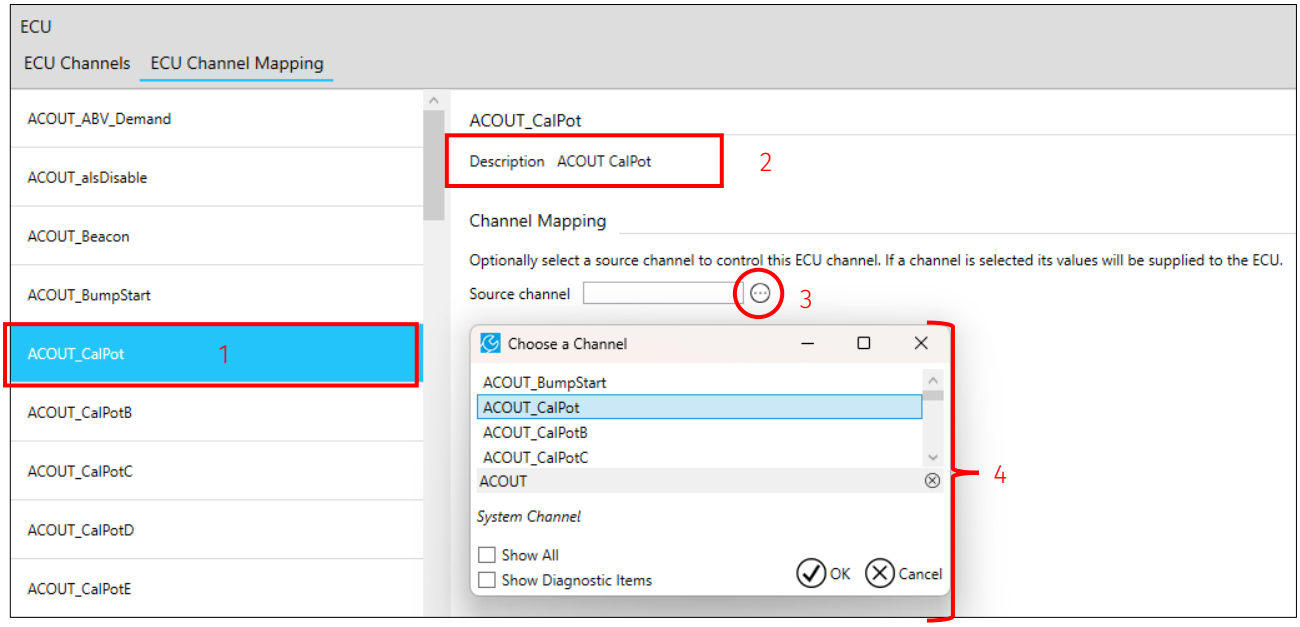
**Note** - Some channels may automatically be protected from an unlicensed user in the setup metadata. This is because the Pectel produced metadata contains protection at the point of build.



## 18.2. ECU channel mapping

The **ECU Channels Mapping** menu is prepopulated with all the channels that are produced from the control side (ECU) and can then be logged on the CHP side. This list contains Autocoding actuators for the ECU side, which enable control over ECU applications such as CalPots.

You can configure which CHP channel is mapped to be exchanged to the ECU side. Click on the list of channels in the **ECU Channels Mapping** menu (1), to see a brief description about the channel (2). To select a channel to be mapped, click the 'browse' icon (3), and then select an available channel from the Channels list (4).



# 19. Fuelling

The fuelling node is used to as an interface to configure fuel usage calculations and fuel level reset events. This node is particularly useful for configuring setups for endurance racing vehicles where it is important to monitor and analyse fuel usage. The fuelling node automatically generates fuel usage channels including:

- Fuel Consumption
- Fuel Lap Economy
- Fuel Level
- Fuel Remaining
- Fuel Used Last Lap

## 19.1. Configure fuel usage channels

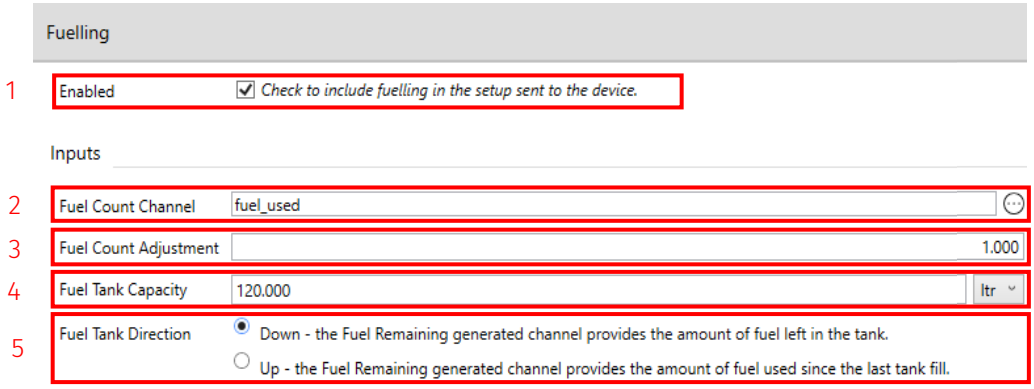
Select the **Enable** check box to enable the **Fuelling** node (1), and then select the **Fuel Count Channel** (2). For example, this can be a channel from the ECU via ECU node, CAN streams, or a Math Channel.

**Note**: You must configure the fuel count channel with **Units of Volume.**

You can apply a fuel count adjustment factor to convert the units of volume to cubic metres (3). You must also specify the vehicle fuel tank volume (4).

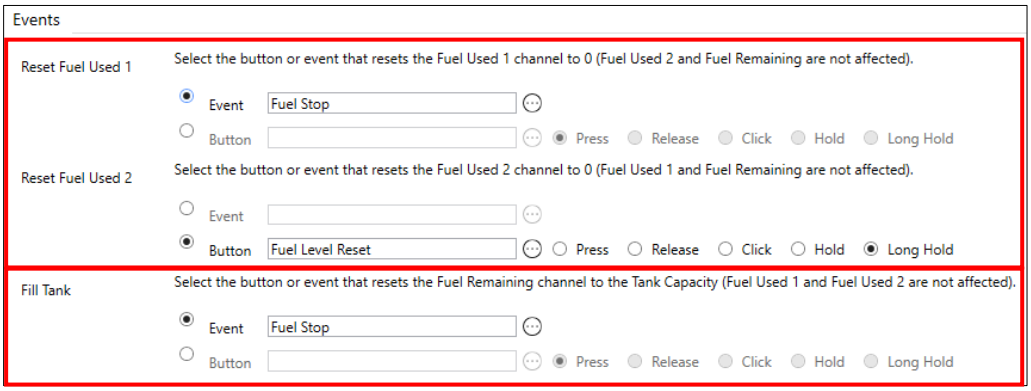
Finally, select the fuel tank direction (**Down** or *Up*) (5):

- **Down** – The generated fuel remaining channel provides the amount of fuel left in the tank.
- **Up** – The generated fuel remaining channel provides the amount of fuel used since the last fill.



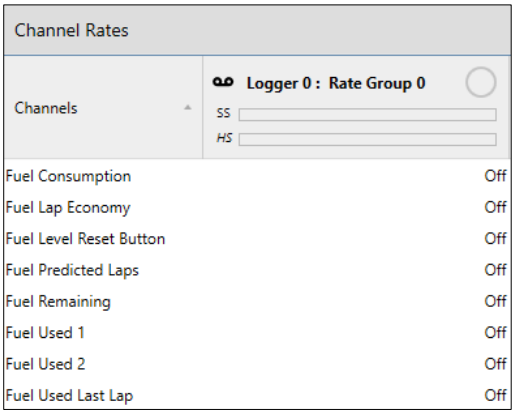
## 19.2. Configure fuel level reset events

To reset the generated fuel used channels to 0, you can configure two reset events (1). These can be an event (see Setups – System Status) or a button (see Setups – Buttons). The fill tank event is used to reset the generated fuel remaining channel to the maximum fuel tank capacity configured in the inputs, and can be either an event or a button trigger (2).




## 19.3. Generated channels

When you enable the **Fuelling** node, eight channels are automatically generated that you can use to monitor and analyse fuel usage. The channel rates for these generated channels must be set on the **Channel Rates** node (see Setups – Channel Rates) to be logged and displayed in the offloaded data.

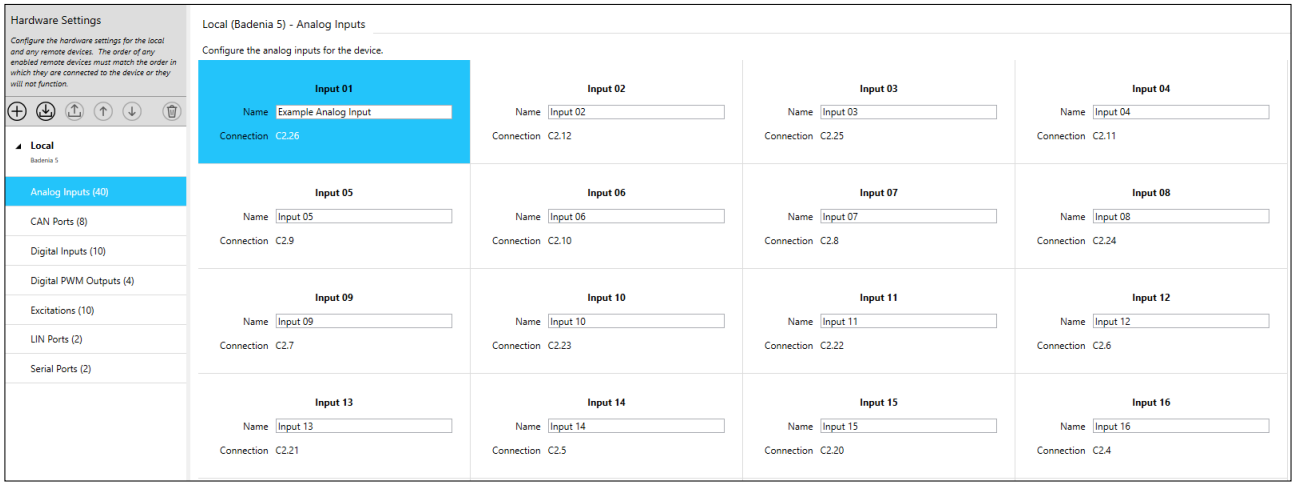


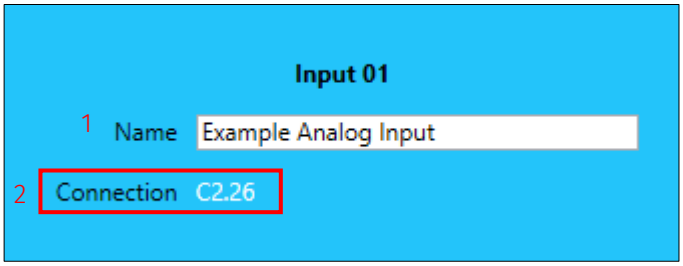
# 20. Hardware settings

The **Hardware Settings** node allows you to configure the settings for the hardware on the local device as well as any remote/secondary devices, such as an SJU.

## 20.1. Analog inputs

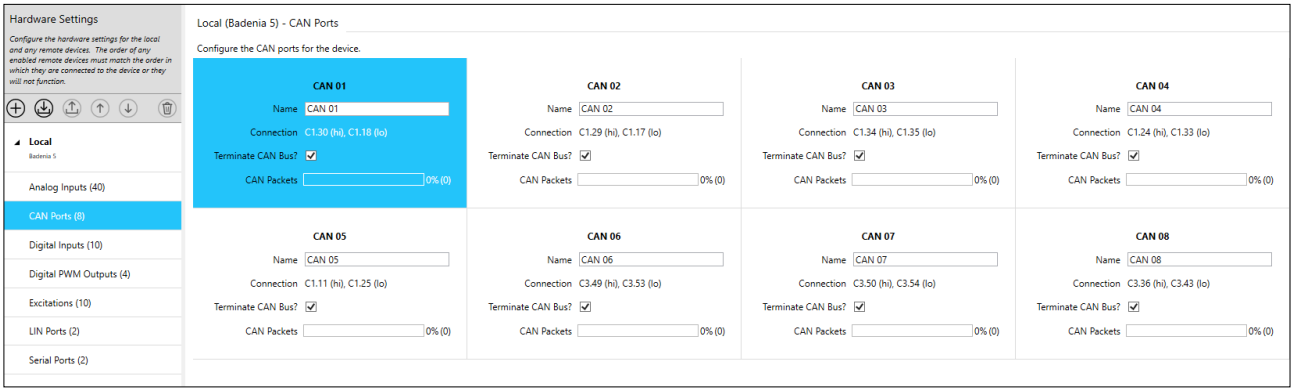
You can configure the name of analog inputs on the **Hardware Settings** node (1). Underneath each input label is a connection identifier in the format 'CX.Y', where X and Y are values that identify the connector and the pin of the device, respectively (2).

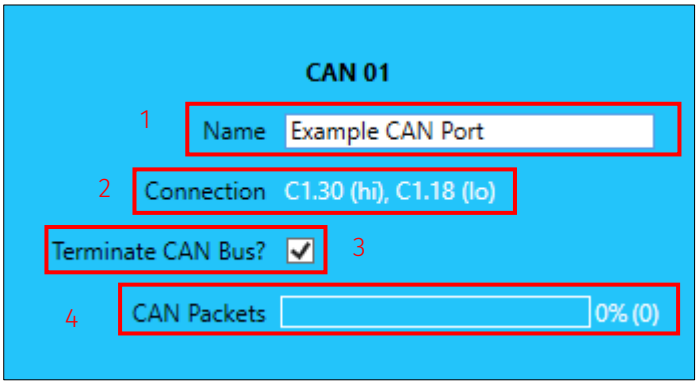


## 20.2. CAN ports

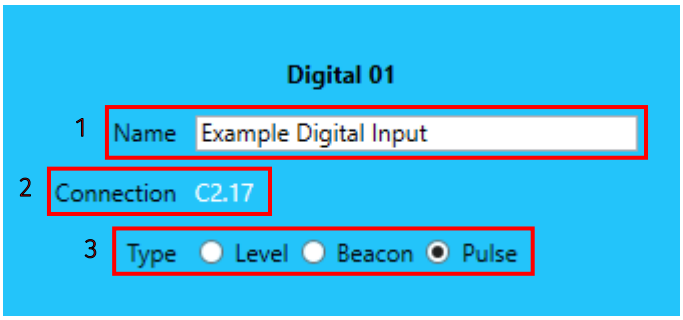
You can configure the name of the CAN ports on the Hardware Settings node (1). Underneath each CAN port label is a connection identifier in the format 'CX.Y', where X and Y are values that identify the connector and the pin of the device respectively (2). You can enable a selectable 120 Ohm CAN termination (3). The bar display shows the consumed bandwidth of the CAN port (4). This consumed bandwidth only represents the bandwidth allocation that Toolset can identify (configured in the CAN streams). Other messages on the bus are only visible to an external bus listener for the external devices.



## 20.3. Digital inputs

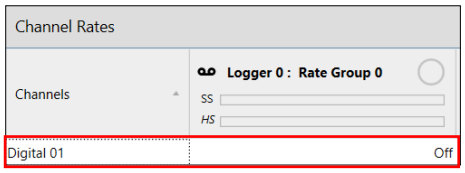
You can configure the name of digital inputs on the **Hardware Settings** node (1). Underneath each input label is a connection identifier in the format 'CX.Y', where X and Y are values that identify the connector and the pin of the device respectively (2). There are three digital input types (**Level**, **Beacon**, and **Pulse**) selected from the **Type** check boxes (3).



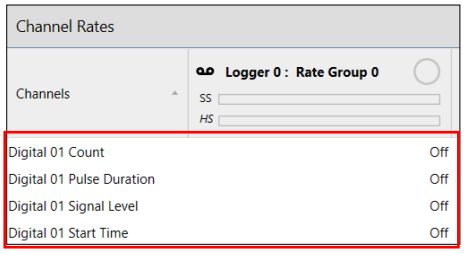

### Types of digital inputs

When you configure the **Hardware** settings of a digital input, you can select from three options based on the application type. You can define sensors as shown below, but within the **Hardware** settings you can also configure the debug channels created.

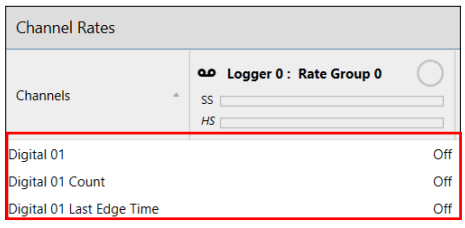
**Level**: Used for simple, digital push button, On/Off inputs. A standard configuration that allows you to generate channels to view the output level of the digital sensor.




**Beacon**: Used when you configure an end of lap timing beacon (see **Beacons**). You can generate multiple channels for debug from the digital pulse duration and start time. This is useful when you use an IR beacon to detect the start finish line, as a standard pulse time of ~12ms is usually required.

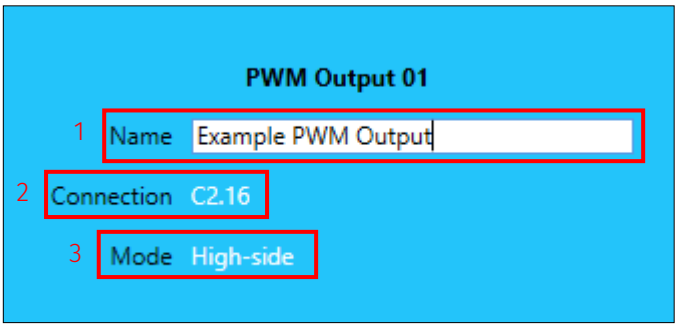



**Pulse**: Used when you configure rotational or DF11i wheel speed inputs where the number of pulses correlates to the number of revolutions. This generates frequency channels for rotational sensors and the time between teeth.



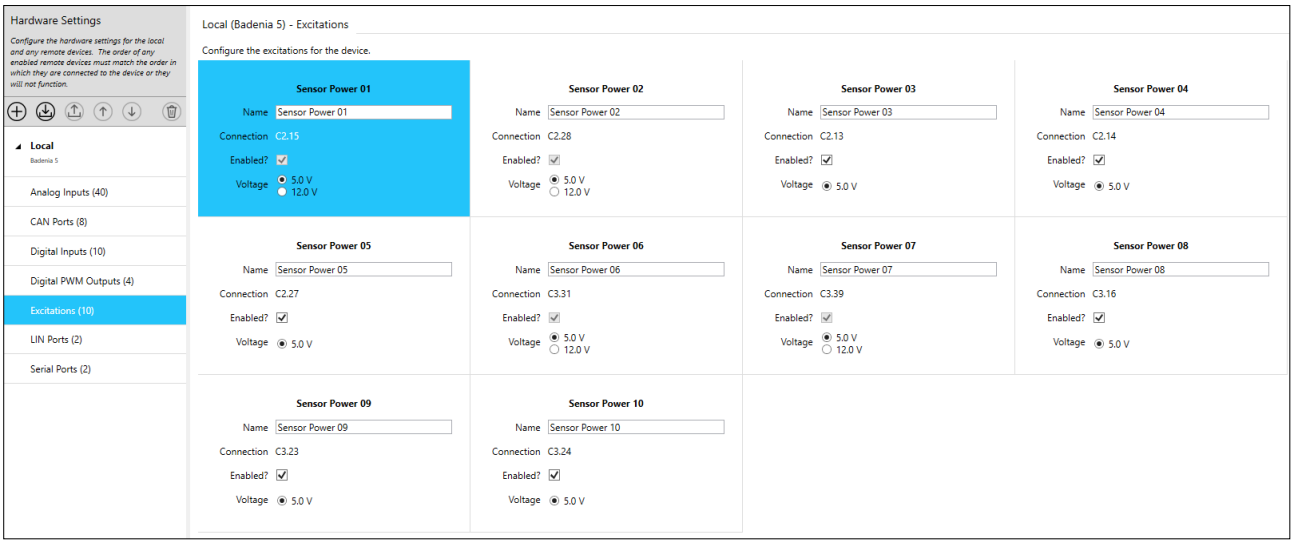
## 20.4. Digital PWM outputs

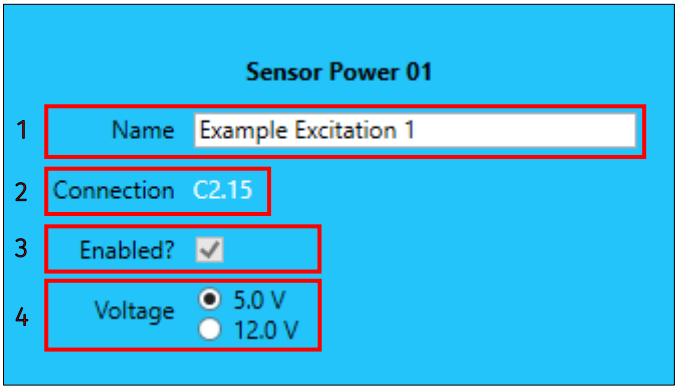

For devices with Digital PWM Output capability, you can configure the name of the Digital PWM Outputs on the **Hardware Settings** node (1.). Underneath each PWM Output label is a connection identifier in the format 'CX.Y', where X and Y are values that identify the connector and the pin of the device respectively (2). The PWM output **Mode** (high-side or low-side) is displayed (3).



## 20.5. Excitations

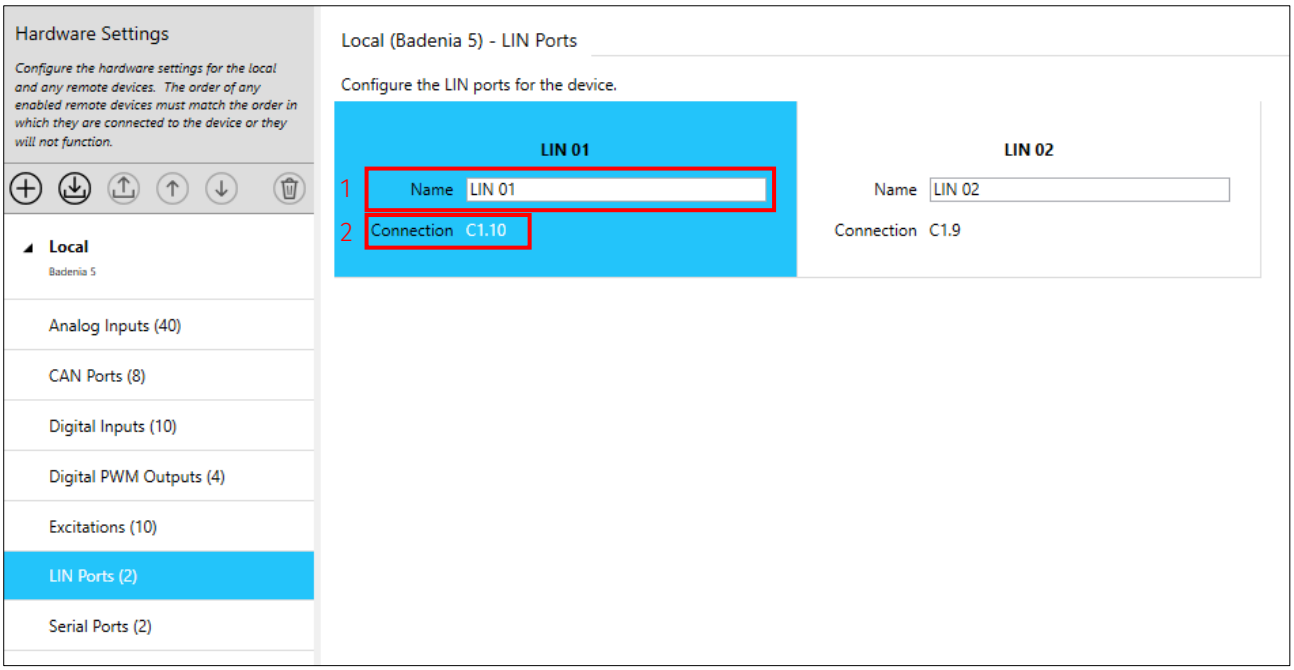
You can configure the name of power supplies (known as 'excitations') on the **Hardware Settings** node (1). Underneath each **Excitation** label is a connection identifier in the format 'CX.Y', where X and Y are values that identify the connector and the pin of the device, respectively (2). Select the **Enabled?** checkbox (3) to enable or disable the excitation by selecting the Some excitations offer variable output voltage capability. For these variable excitations you can click the required checkbox to set the output voltage (4).



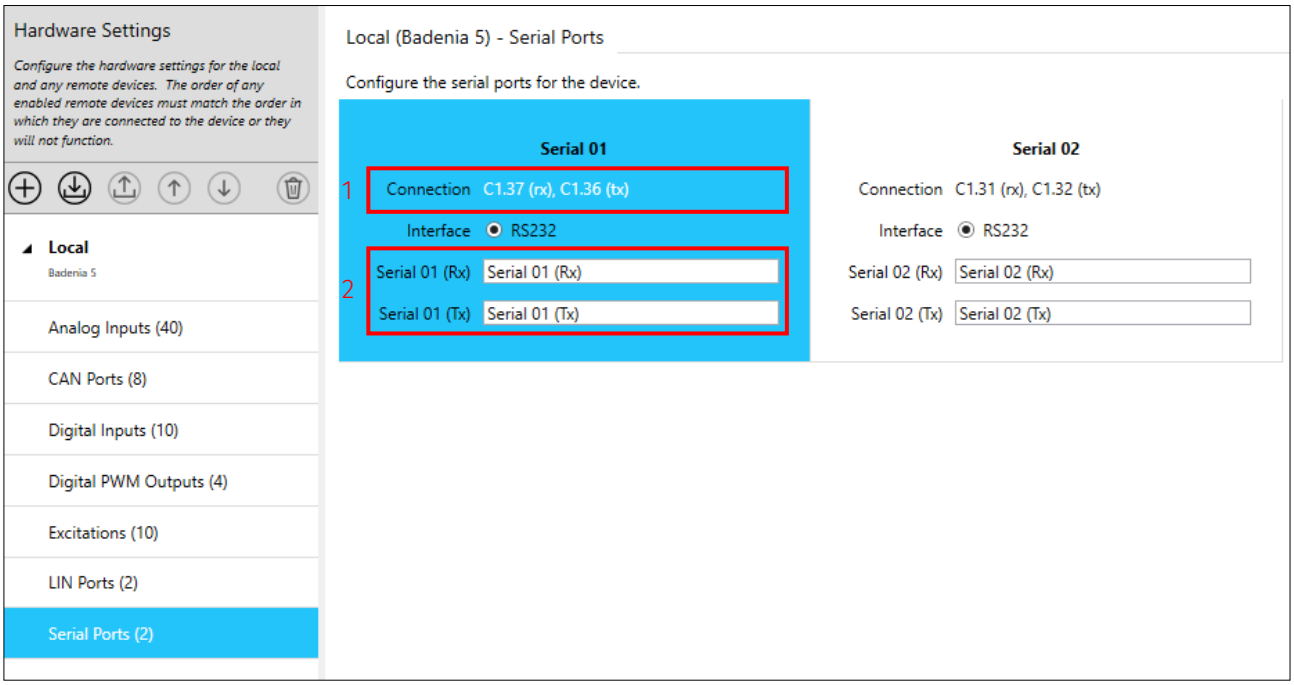
## 20.6. LIN ports

You can configure the name of the LIN ports on the **Hardware Settings** node (1). Underneath each LIN port label is a connection identifier in the format 'CX.Y', where X and Y are values that identify the connector and the pin of the device respectively (2).



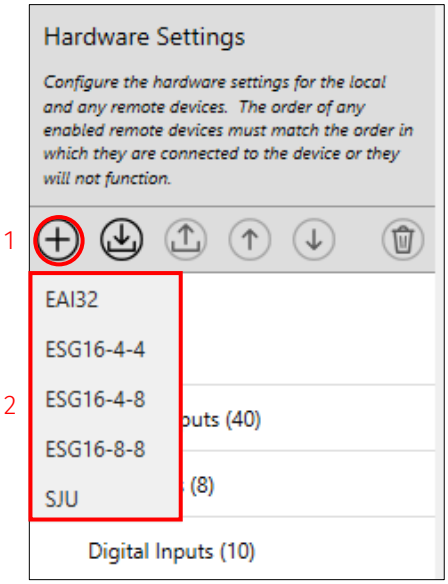

## 20.7. Serial Ports

On the **Hardware Settings** node, underneath each **Serial Port** label is a connection identifier in the format 'CX.Y', where X and Y are values that identify the connector and the pin of the device, respectively (1). You can configure the names of the **Serial Rx** and **Tx Ports** (2).

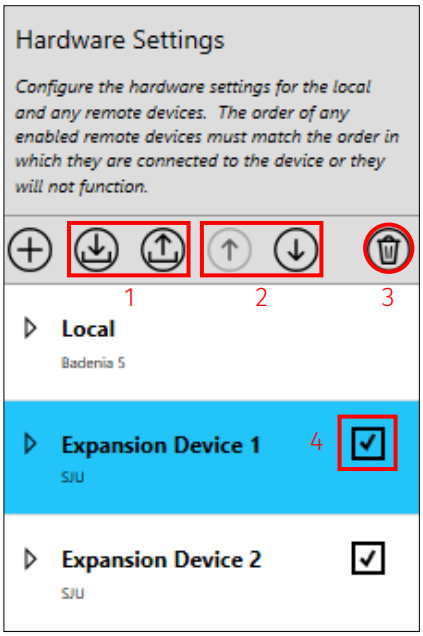


## 20.8. Configure an EtherCat expansion device

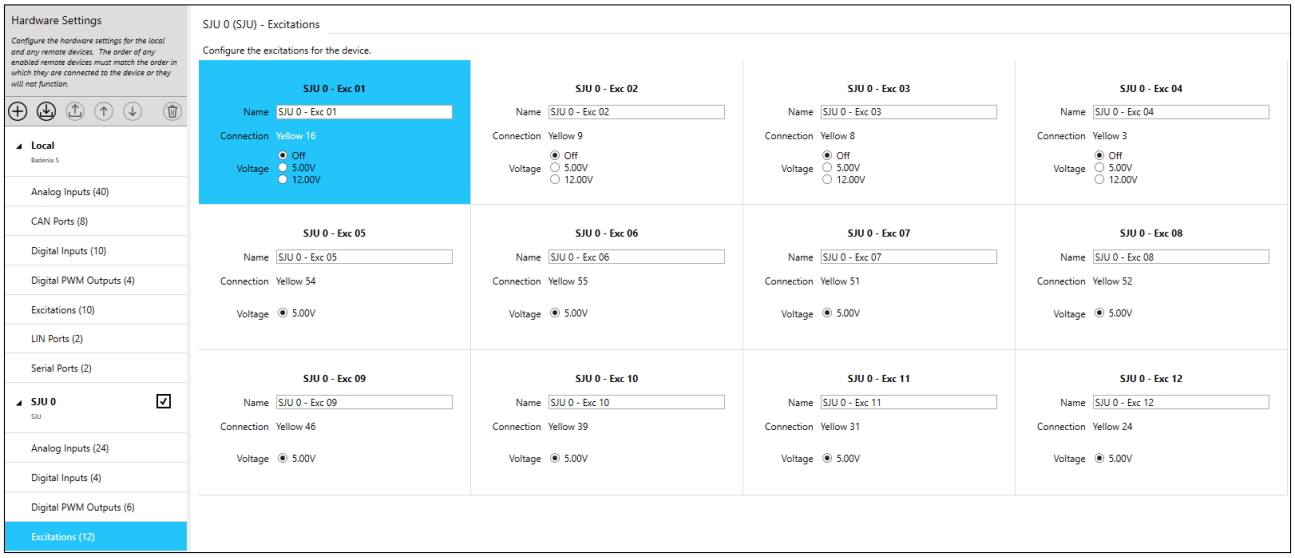
To add an EtherCat expansion device such as the EAI32 or SJU click the **+** button (1), and then select the device from the dropdown menu (2)



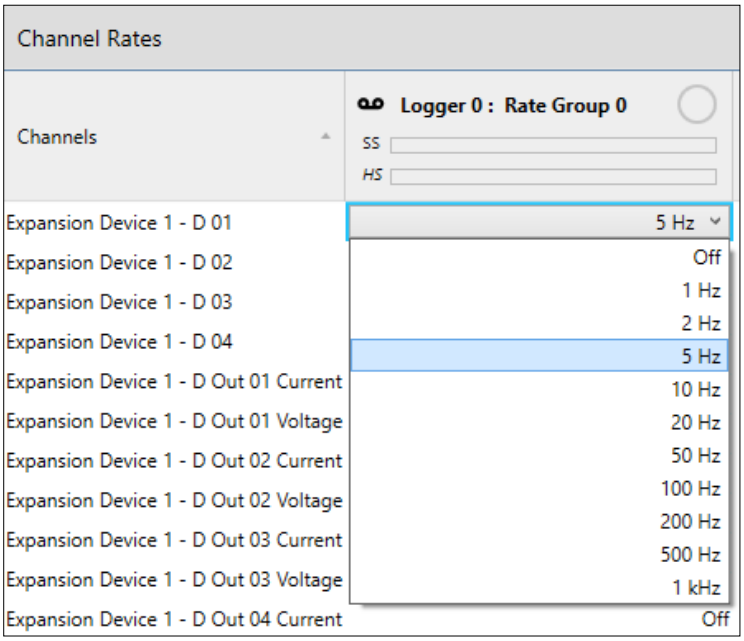

You can add multiple expansion devices and import and export previously configured expansion devices between existing setups (1). The order of the devices in the list must match the order in which they are physically connected on the EtherCat bus. Use the reorder tools (2) to change the order of the devices in the list up or down (2). Use the 'bin' option (3) to delete expansion devices from the list. If an expansion device is removed from the vehicle but you do not want to delete the device configuration from the list, deselect the 'enable' box (4).




The inputs and outputs for the expansion device appear in Hardware Settings in the same format and are configured in the same way as the local device hardware settings.




**Note**: When an expansion device is enabled, the expansion device channels are automatically populated on the **Channel Rates** node, but the logging rates must be set.



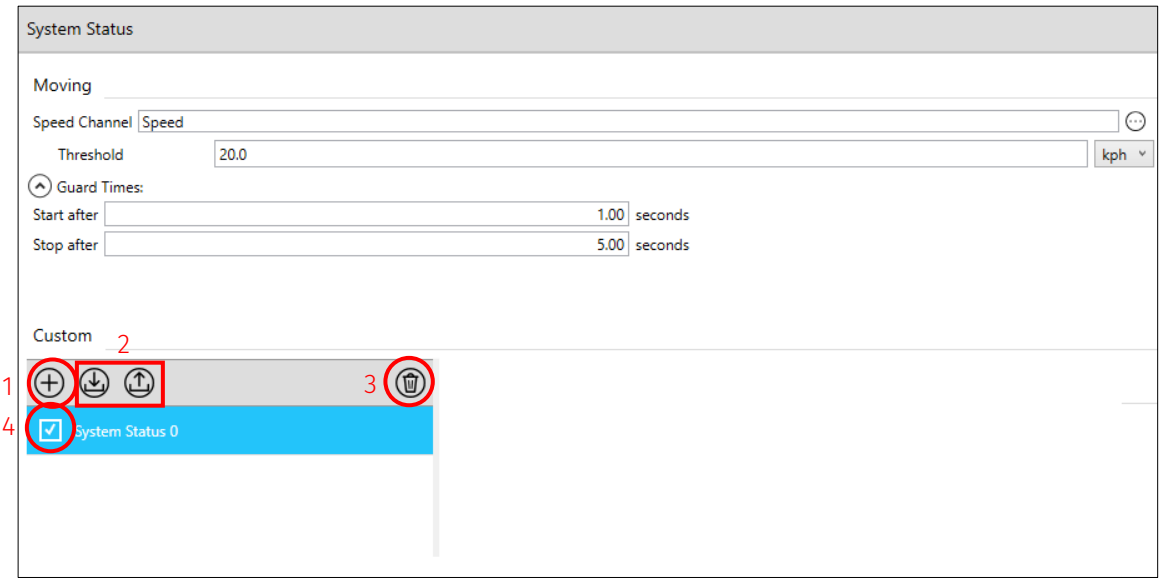
# 21. Latching

The **Latching** node in Toolset setups allows a device to retain a value over a power cycle after a specified event. You can create these events on the **System Status** node or use default events that are calculated on the device.

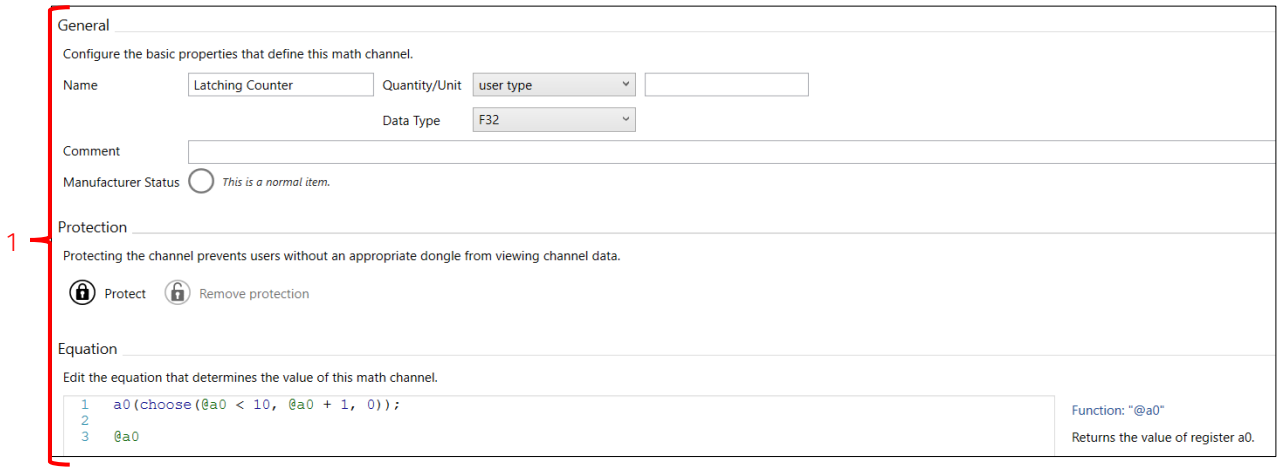
## 21.1. Latching values

### Generating custom events

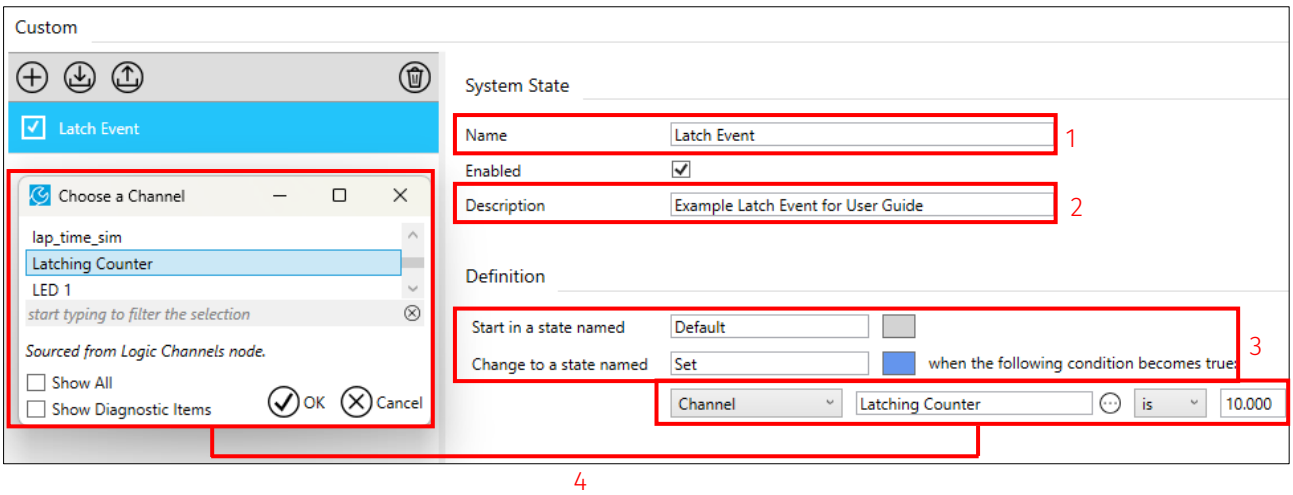
You can create custom events on the System Status node. Click **+** to create a custom event (1). You can import and export custom events between existing setups using the import and export tools (2). You can delete custom events with the 'bin' tool. To disable a custom vvent without deleting it from the setup, deselect the 'enable' box (4).




Multiple custom events can be added. In the example below, a custom event is created based on a Maths channel counter, so that repeatable events are present in the data (that is every 10 seconds). The channel used 'Latching Counter', simply generates value between 0-10 (1). On a vehicle, other events can be used such as 'Drive Out' or 'Car Halt', if more applicable.

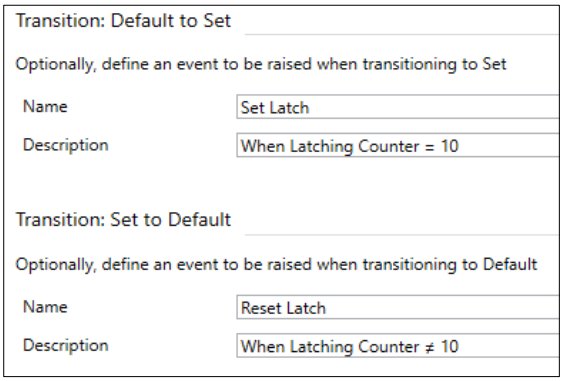


You can name the custom event (1) and add an optional description for the event (2). You can also name the bit-field definition states, apply a colour for each state (3), and define a trigger condition (4).




You need to supply a name for the 'transition' between 'Default to Set' and 'Set to 'Default'. The events channels are then generated.

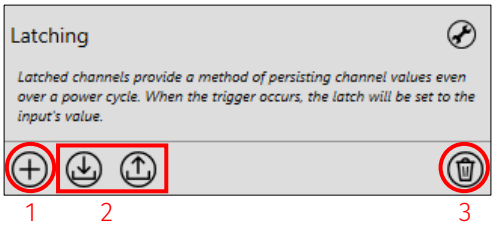
In the example below, when the latching counter reaches 10, an event called 'Set Latch' is generated when the transition from 'Default to Set' occurs as the latch conditions become true. Conversely, an event called



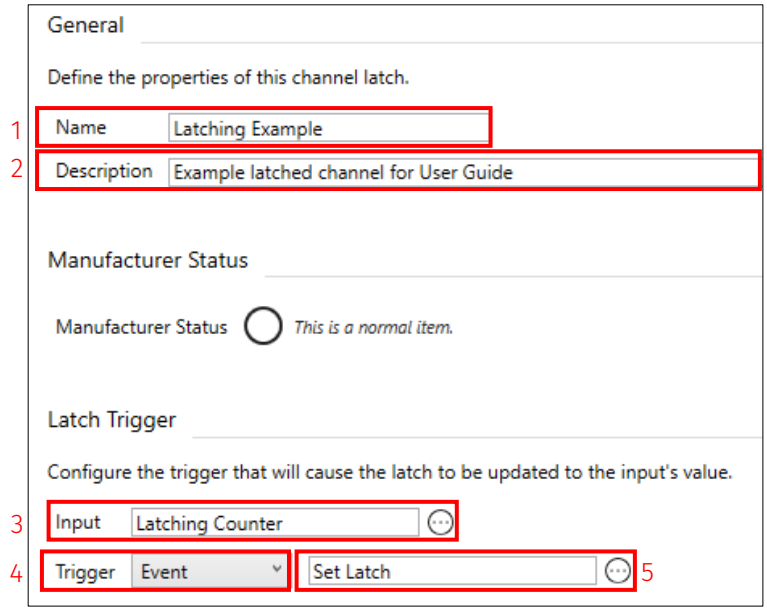

'Reset Latch' is generated when the latch conditions become untrue as the transitions from 'Set to Default' occurs.

### Add a latching channel

Once an event is being generated, you can create the latching channel. On the **Latching** node, click the **+** tool (1). Use the import and export tools (2) to import and export latching channels between existing setups. Use the 'bin' tool (3) to delete latching channels.



You can define a name for the latching channel (1) and add an optional description of it (2). Click the **Browse** menu to select an input channel for the latch trigger from the available channels (3). Select the required trigger (Button or Event) from the dropdown menu (4). From the **Browse** menu, select the latching event from the available events (5).




The latched value is updated at a maximum rate of 1Hz. Once the device is power cycled the non-volatile memory checks that the channel is equal to the latched channel value. If these values are different then the channel retrieves the latched channel value.

When the device is power cycled, the latching node retains the last latched value for each defined channel.

The Latch Status is a bit-field channel with the following definitions.

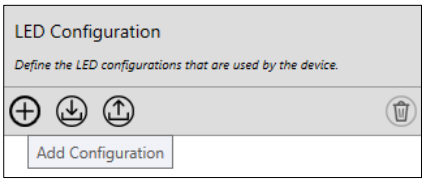

**Note**: The current max number of latched channels is 42. The logging rate is capped at 1hz, but the calculation rate is driven from the source channel.

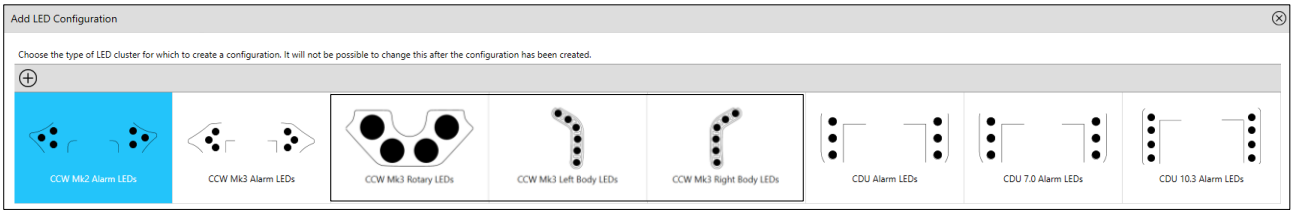
# 22. LED configuration

The **LED Configuration** node is used to configure the LEDs which are part of the Cosworth range of displays and steering wheels. You can use LEDs to alert drivers to system alarms, to display vehicle information like battery state of charge, or to feedback information like if turn signals are on.

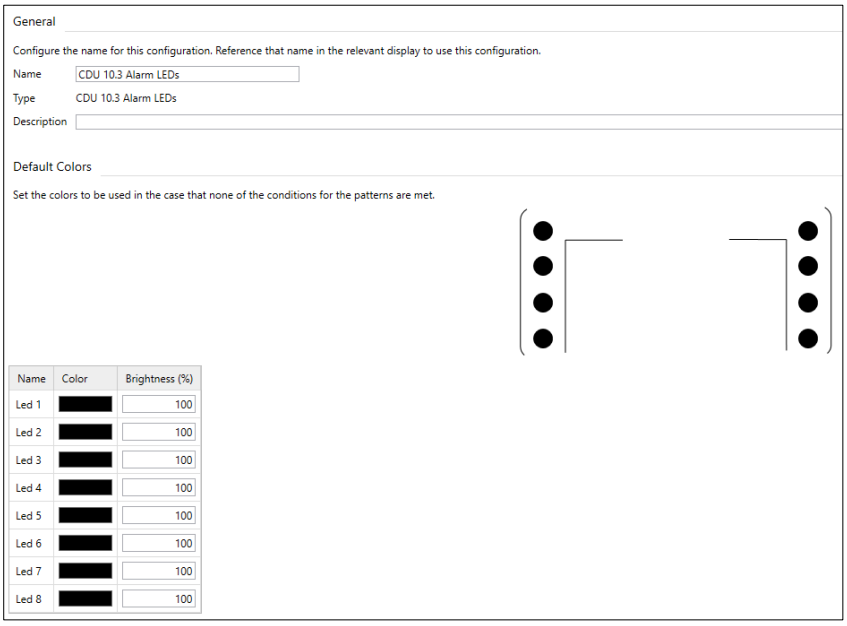
## 22.1. Add an LED configuration

Click the **+** button, to open the **LED Configuration** window, and then select the required LED cluster.



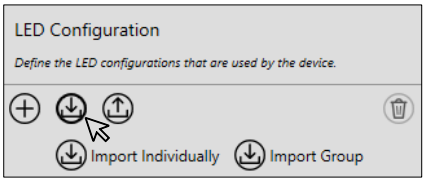




Enter a name and optional description for the configuration, and then select the **Default Colors** to be used if none of the conditions for the patterns are met.



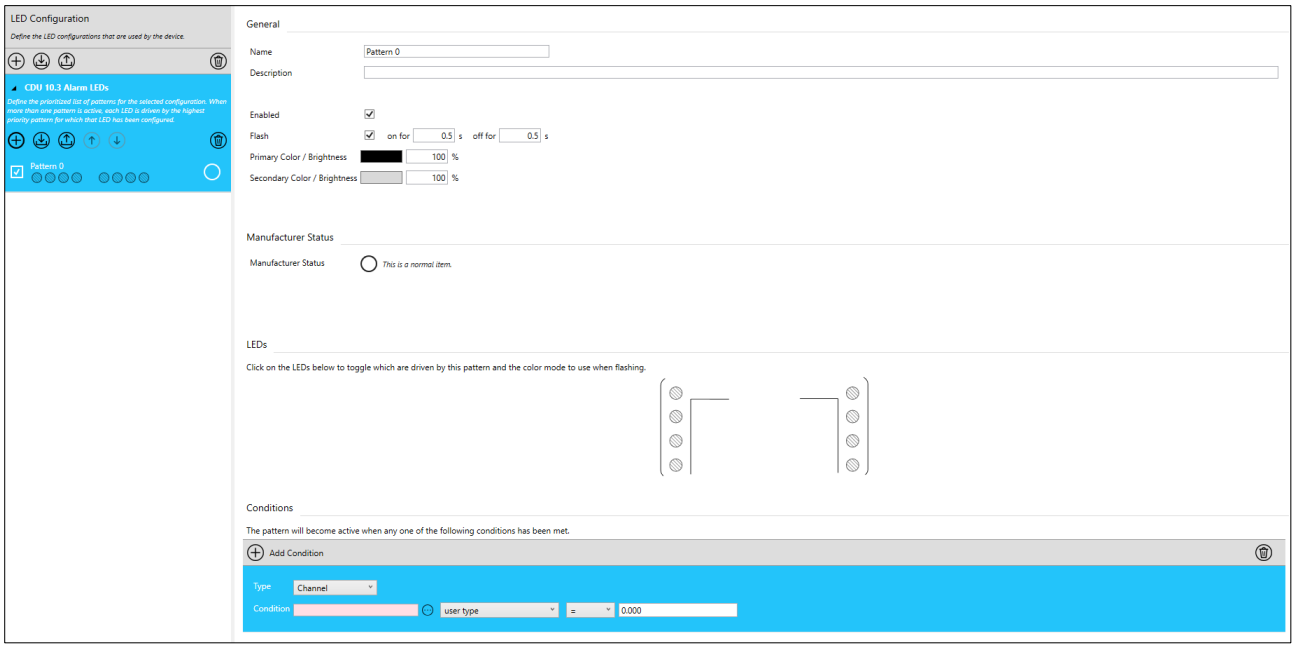
## 22.2. Import and export an LED configuration

You can import and export an LED configuration between setups, either individually or as multiples within a group.



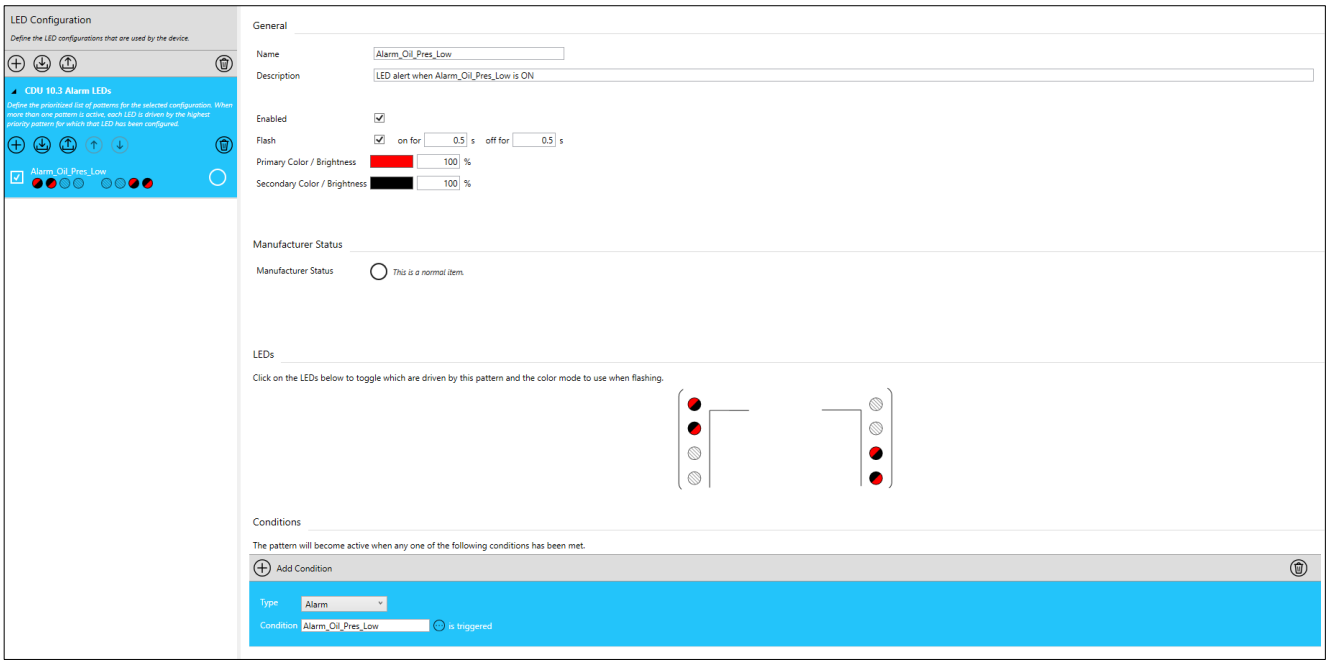

## 22.3. Configure display LEDs

When you select a display device, click **+** to add a new LED configuration.




Set up an LED configuration, as shown below:

1. Enter the name of the LED configuration.
2. Enter an optional description for the LED configuration.
3. Select to either enable or disable the configuration in the setup.
4. Configure the LEDs to flash, and then set the flash rate.
5. Configure the LED colours.
6. Click the LEDs to configure the LEDs that are active on the display. When you select the **Flash** option, you can click the LED to select whether it is active on the primary or secondary flash.
7. Configure the conditions to drive the LEDs – Alarm, Bit-field channel, Channel. or Strategy. You can add multiple conditions.

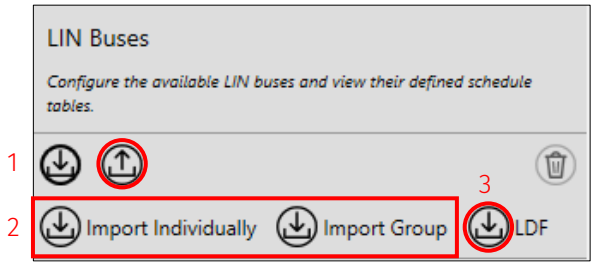


# 23. LIN

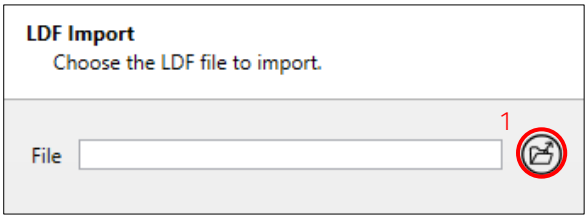
All Cosworth LIN-capable devices have LIN functionality to allow you to fully configure LIN buses and device behaviour through Toolset. The **Configurable LIN** node is compatible with LIN 2.0 and higher versions which require an *.ldf* file which matches this standard.

## 23.1. Configure a LIN bus

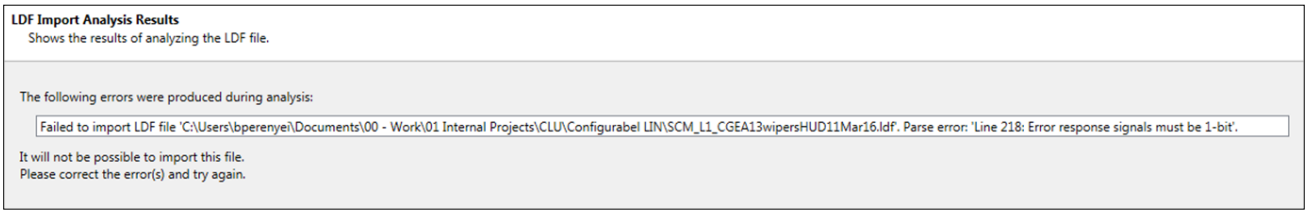
To configure a LIN bus, click import (1), and then select a Toolset Library File (*.tlf*) (2) or LIN Description File (*.ldf*) (3). You can use using the import and export tools (1 & 4) to import and export LIN buses between existing setups as a *.tlf* file.



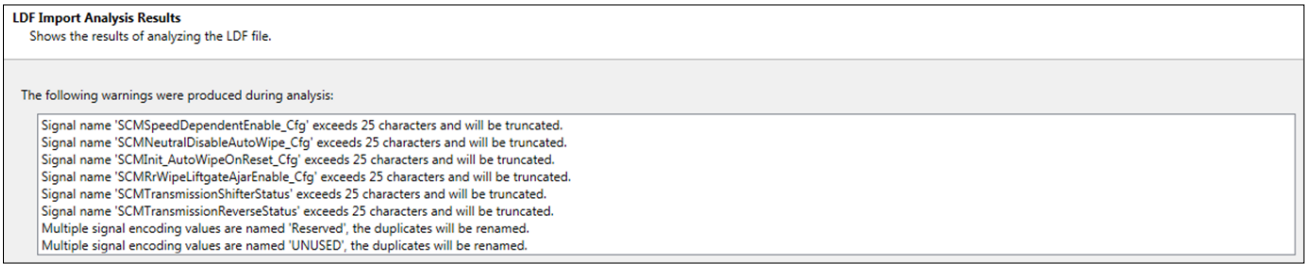

If the LDF import tool is enabled, you can browse to the file from any location on the PC (1).



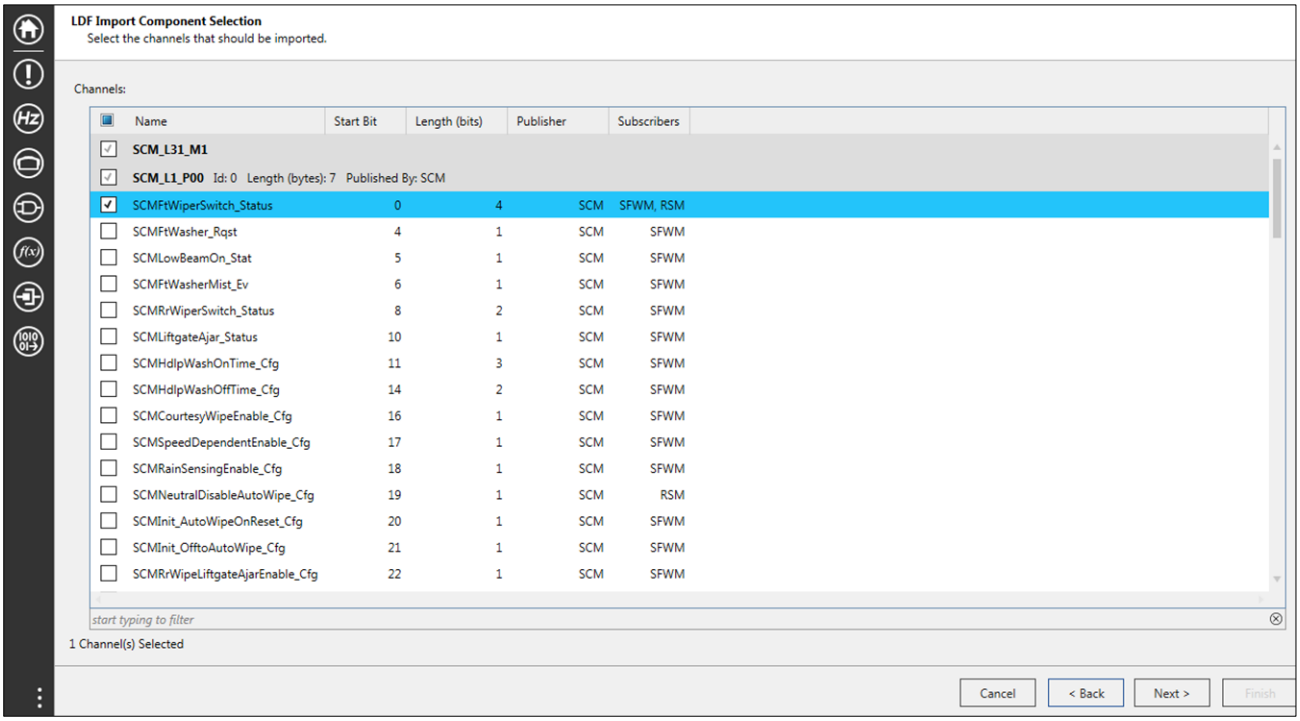

When you select the *.ldf* or *.tlf* file, Toolset checks the file to make sure it adheres to the LIN standard. Critical errors are displayed with their line position within the LDF. You must correct the errors to be able to import the LDF.



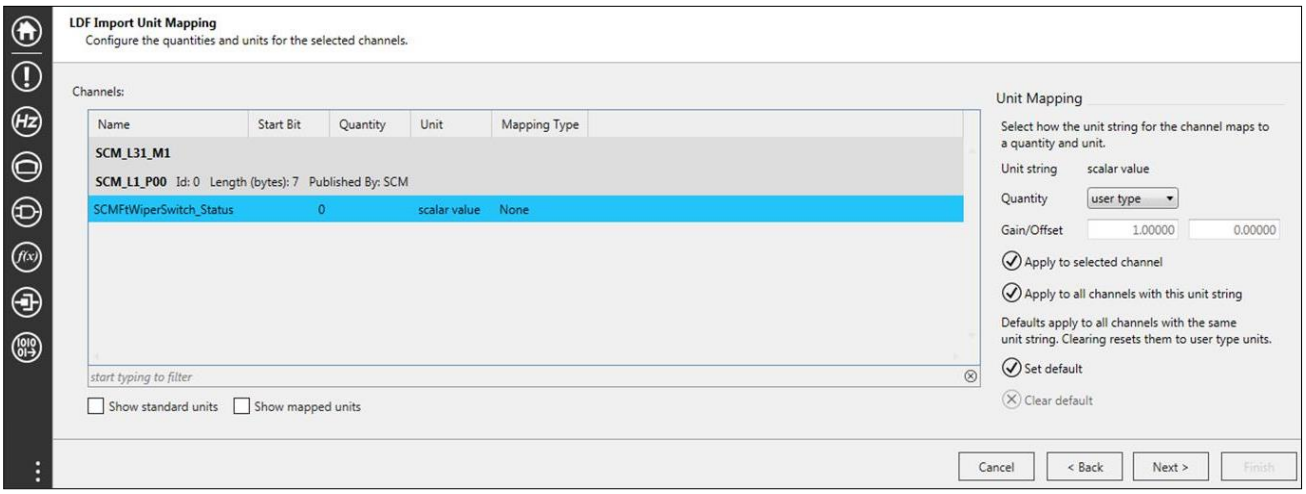

The LDF analysis also shows any warnings. Unlike errors, warnings do not stop the file from being imported into the setup, but can result in inconsistencies with the channel.



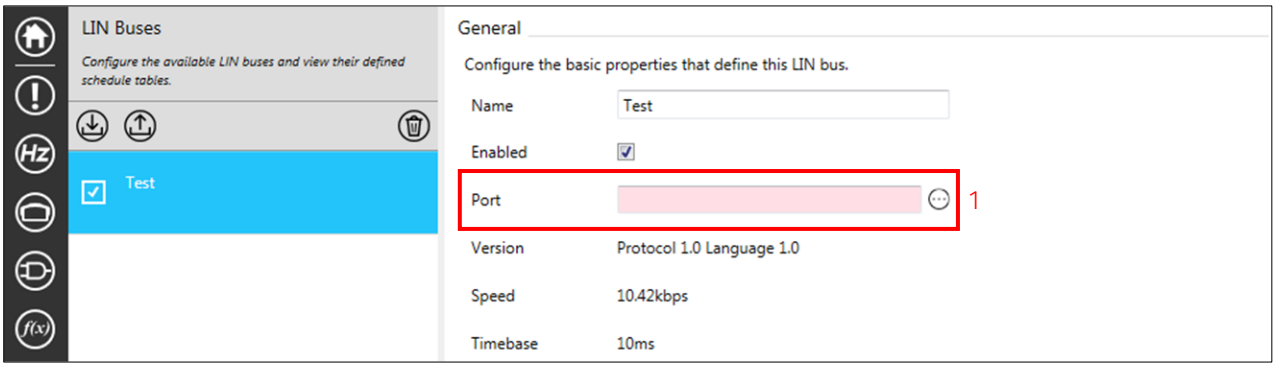
Click **Next** to display the **LDF Import Component** selection list. You can select/deselect schedule tables, frames, and channels here.



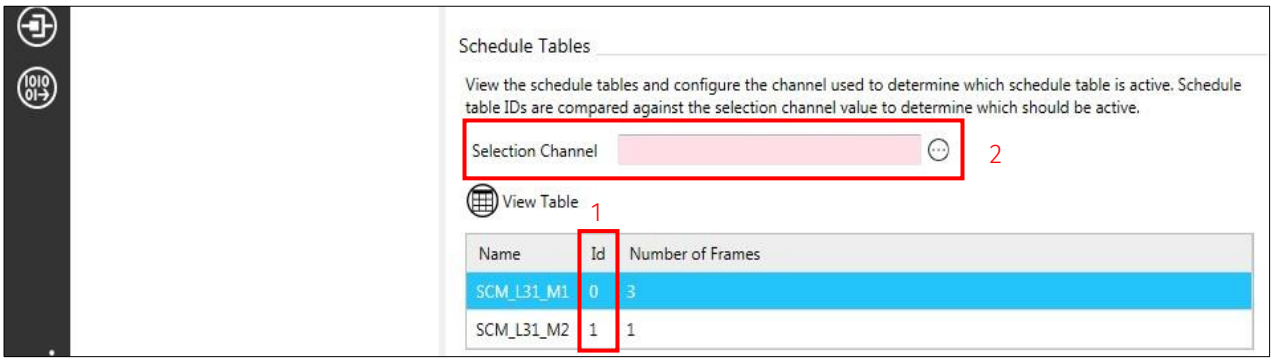

You can also set the unit mapping of each channel if required.



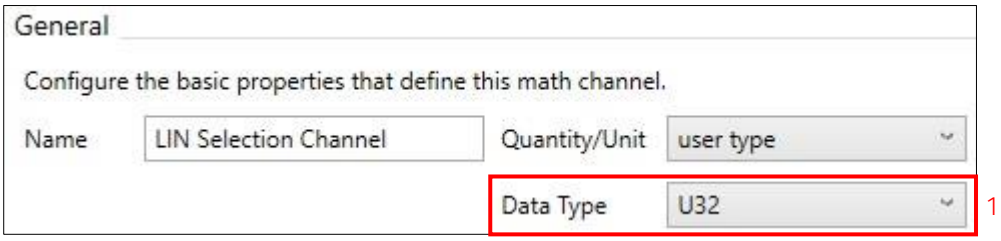
Once imported, the LIN schedule tables are displayed on the **LIN** node and you can select the required LIN port (1).



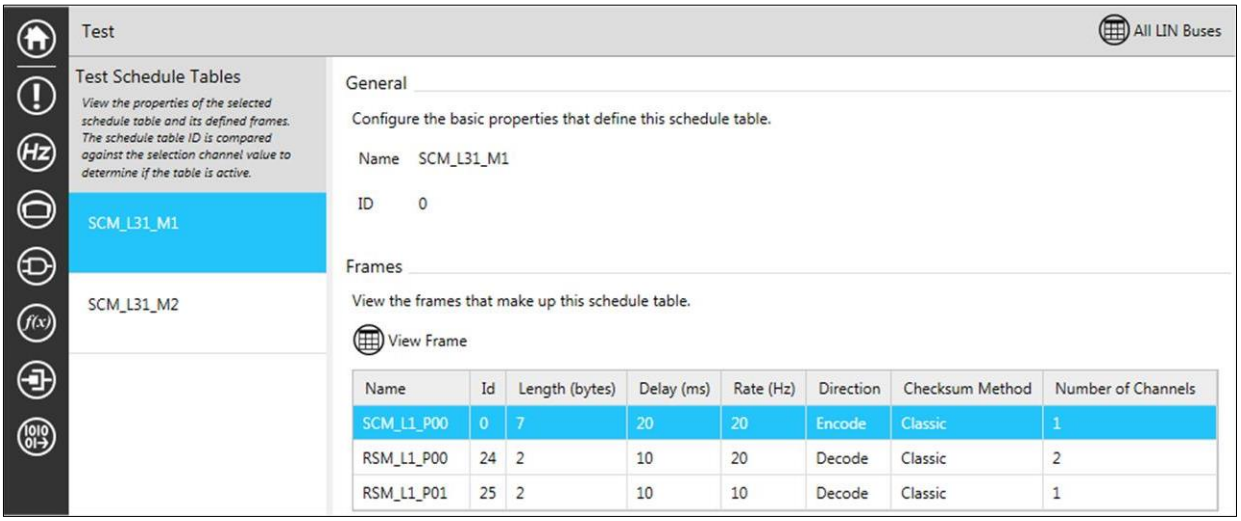

Imported schedule tables are assigned an ID number (1) by Toolset, starting from zero. The **Selection Channel** (2) controls which schedule table is sent out on the LIN bus.



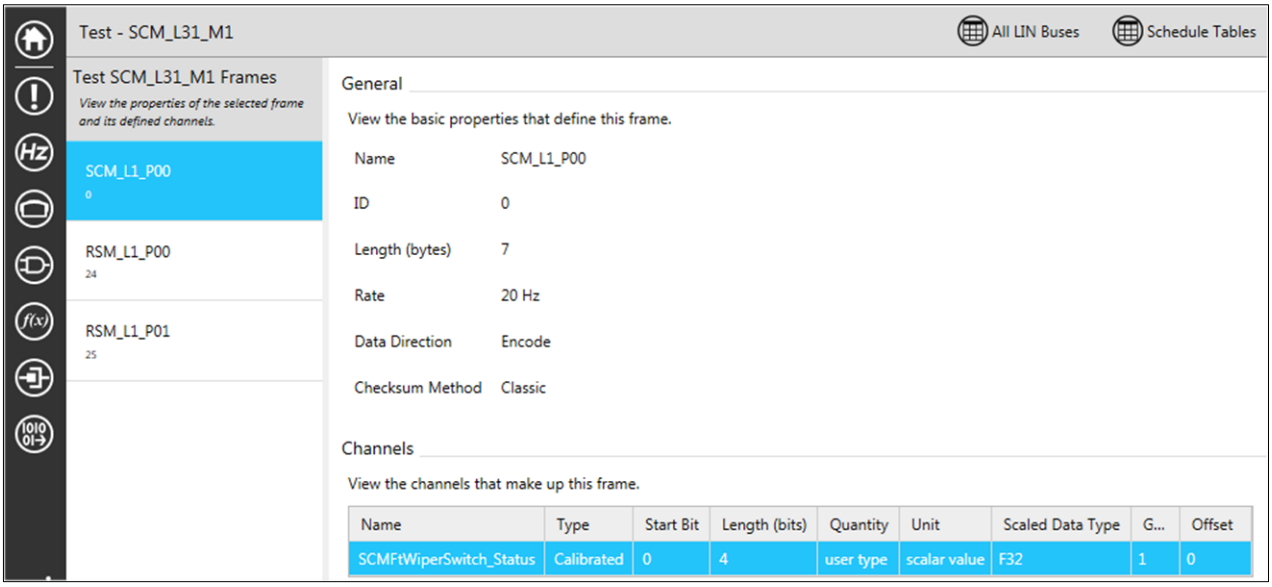

The selection channel should be configured as a maths channel within the setup (See Maths Channels). The maths channel intended to control the selection must be assigned as a U32 **Data Type** (1).



You can view each schedule table to show each frame contained within it.




You can also view each frame to show the individual channels present within it.




Channels that are encoded within the LIN bus also need to be defined within the setup as maths channels.

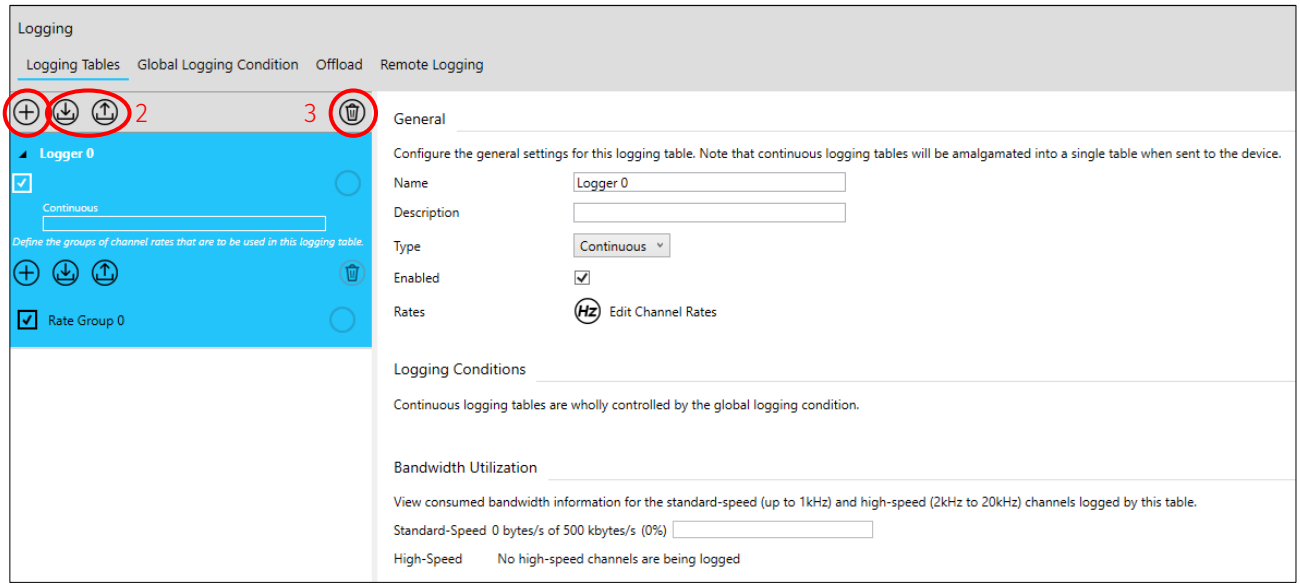
# 24. Logging overview

By default, all Cosworth logging devices are configured with one continuous logging table. The following explains how to configure and modify logging tables, configure logging and offload conditions, and configure remote logging devices.

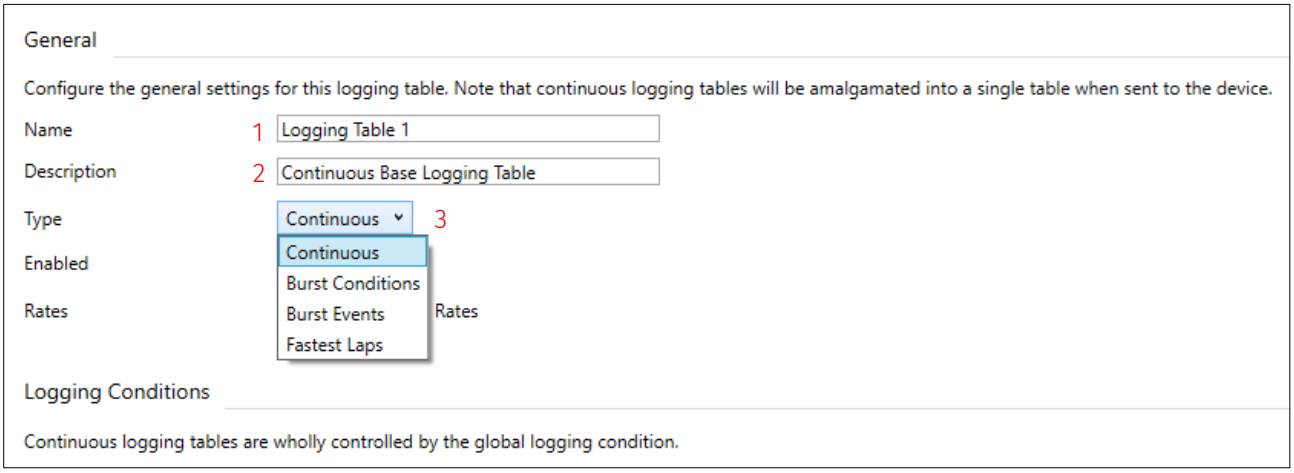
The **Logging** node is used to define the frequency at which channels are logged. Before you set channel rates, you must first define a logging table.

## 24.1. Configure a logging table

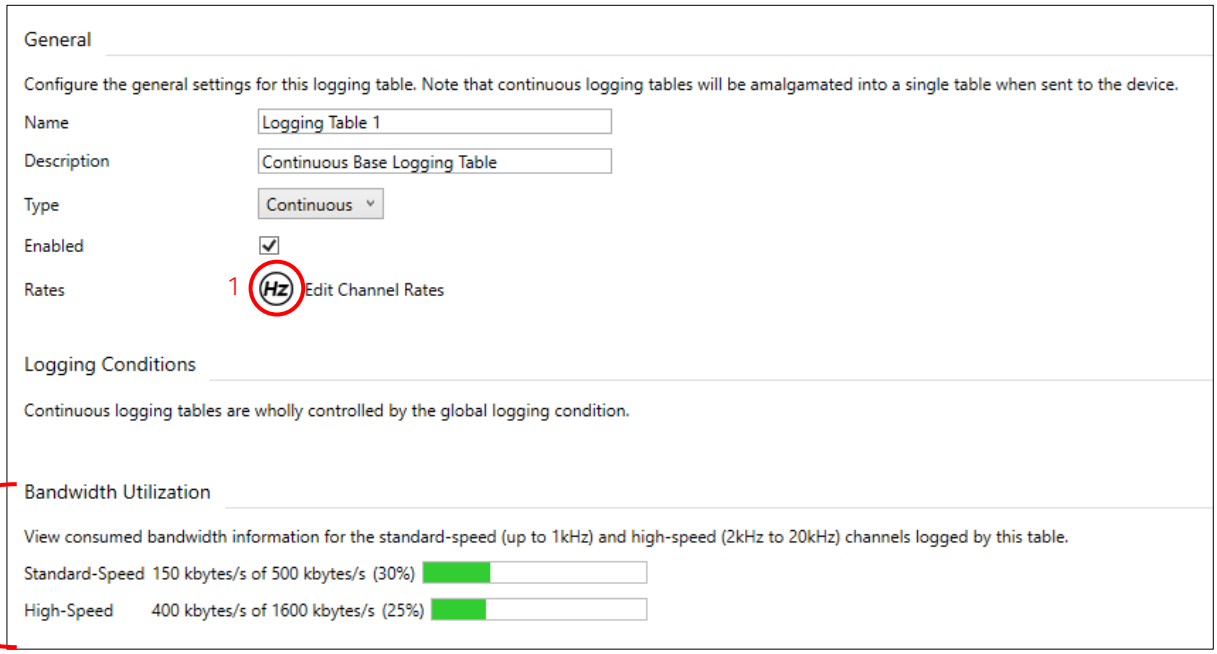
On the **Logging** node, all existing logging tables are shown on the left of the screen. One continuous logging table (Logger 0) is enabled as default. Click **+** (1) to add a new logging table. You can also import and export logging tables from existing setups using the import and export tool (2). You can delete logging tables with the 'bin' icon (3).




Enter a name for the logging table (1) and an optional description (2). Select the required type of logging table from the **Type** dropdown menu (3).



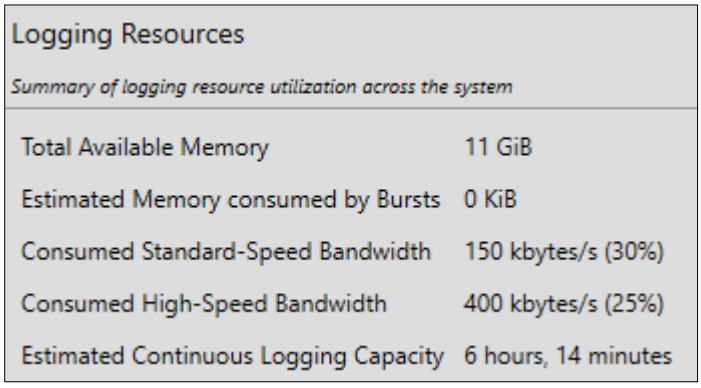
You can then configure the channel rates (1) for this logging table (see Channel Rates). The percentage bandwidth utilization is displayed in a bar graph (2).




### Logging resources

You can view the total logging resource utilization at the bottom left of the page. The Logging Resources section shows information about the bandwidth usage and the estimated time capacity available for continuous logging. This provides an estimate of the time that the device can log with the selected table. Make sure the available logging time is greater than your session time to collect all data.

Note: If the logger runs for longer than its capacity, then the earliest data logged is over-written.



## 24.2. Types of logging table

There are four types of logging table: Continuous, Burst Conditions, Burst Events, and Fastest Laps.

### Continuous

All channels in a continuous logging table are logged whilst the global logging condition is true. Continuous logging tables are wholly controlled by the global logging condition.

### Burst conditions

Ideal for short-term high-speed logging, burst condition logging is activated when a channel or bit-field channel meets the specified conditions.

Note: The global logging condition (for example, 'Engine ON') must also be satisfied before any logging takes place.

### Burst events

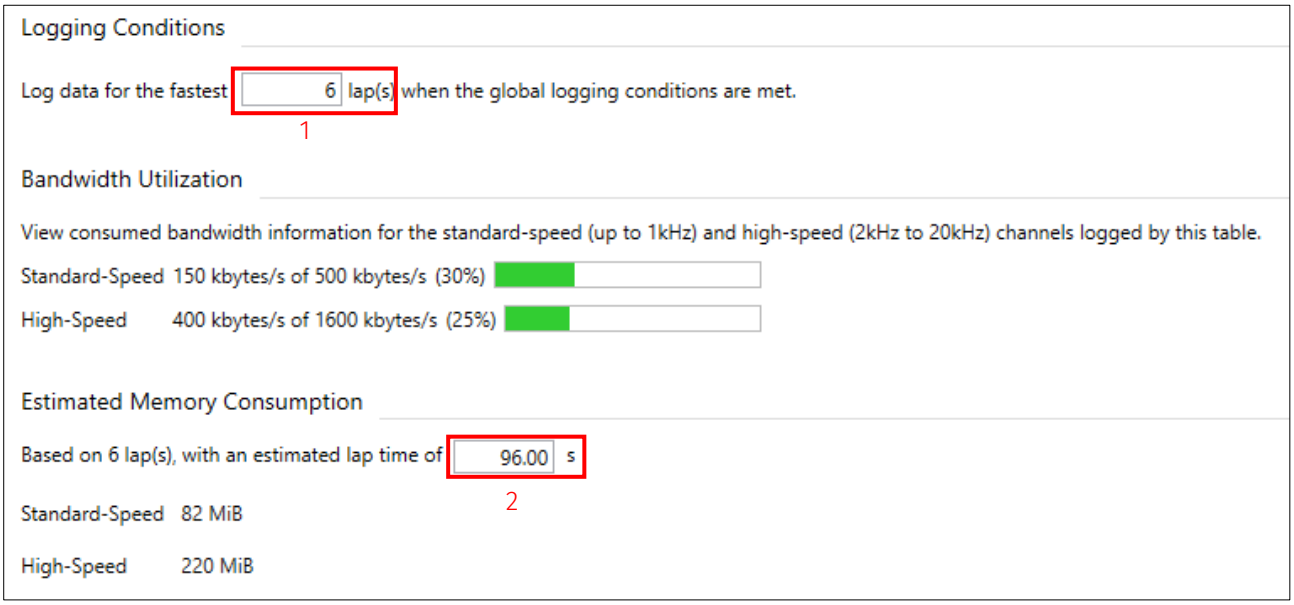
Ideal for short-term logging based on a specific event, such as an alarm being triggered. Logging can then stop 'based on' time from the start of logging or 'based on' a second event such as the alarm being reset.

Note: The global logging condition (for example, 'Engine ON') must also be satisfied before any logging takes place.

### Fastest laps

Stores data from the fastest lap(s) when the global logging condition is met to reduce memory usage. You can configure the number of fastest laps to store.

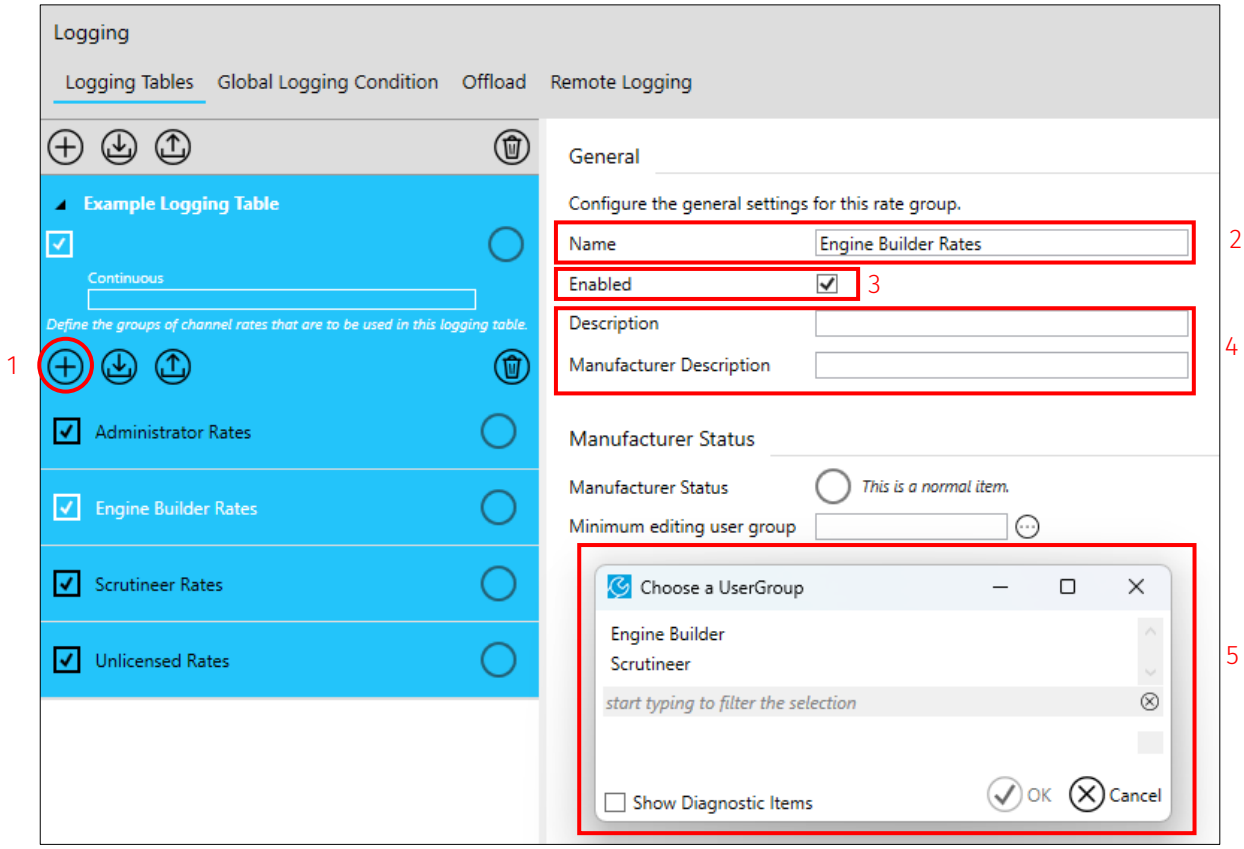
An estimated memory consumption is given from the configured number of fastest laps to store (1) and an estimated lap time (2).



## 24.3. Rate groups

Channels within the same logging table can be logged at different rates by adding rate groups. This is particularly useful when a manufacturer employs user groups (see Setup Locking – User Groups), or for viewing CAM & Crank data at high speed whilst cranking, but then reducing after idle, to optimise the logger capacity.

Click the + icon within a logging table to add a new rate group (1). You can name the rate group (2), enable or disable it (3), and add a brief description (4). After user groups are configured, you can define the minimum editing user group (5).



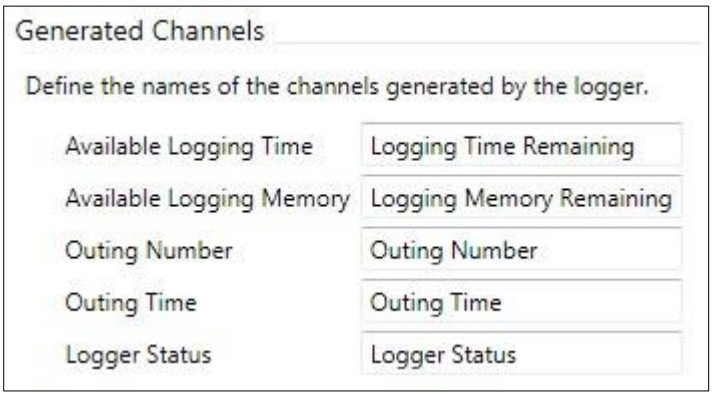

You can then configure the channel rates for each rate group on the Channel Rates node.

## 24.4. Generated channels

When you enable a logging channel, Toolset automatically generates five logging 'status' channels. These channels record:

- Available Logging Time
- Available Logging Memory
- Outing Number
- Outing Time
- Logger Status

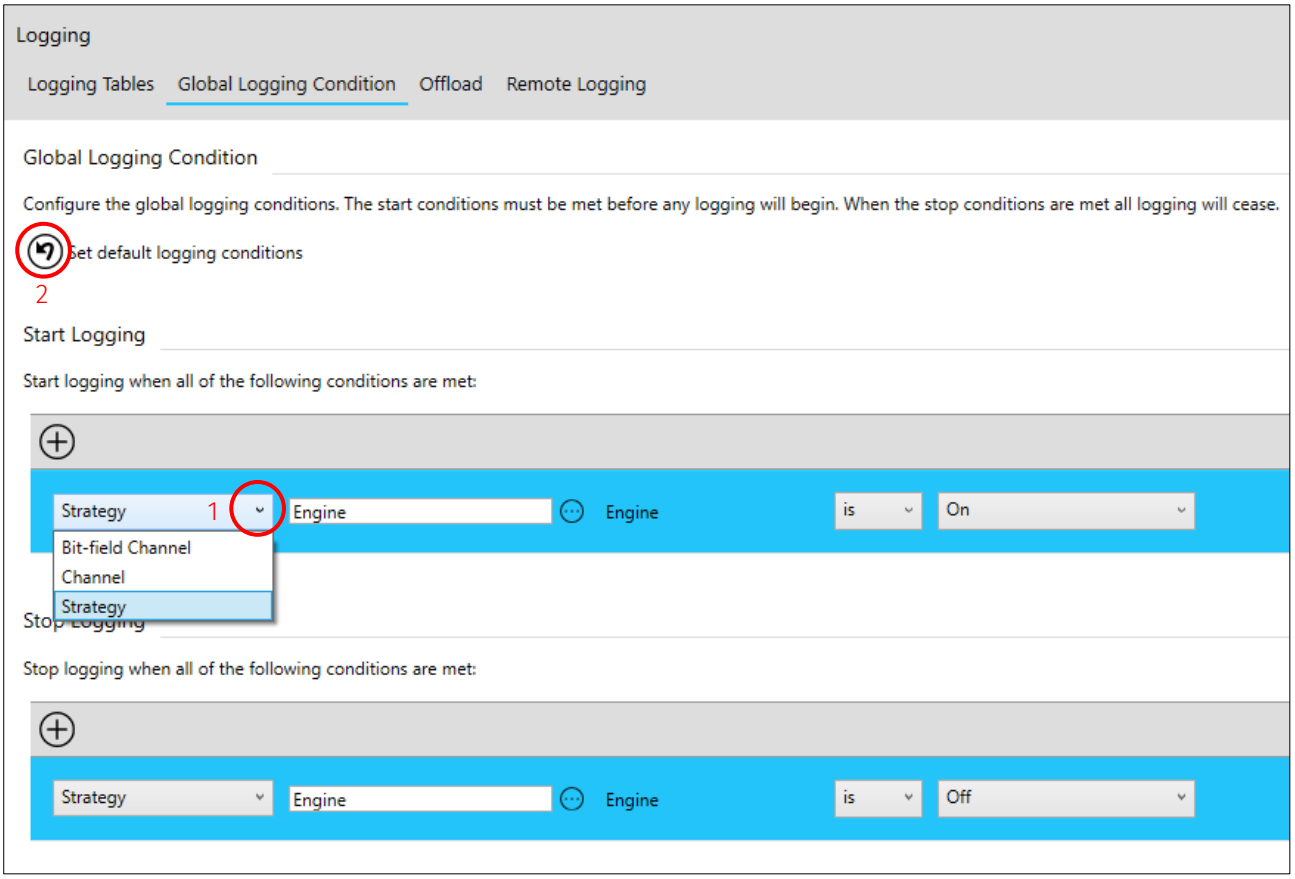
Click the 'wrench' icon at the top right of the window to change the names of these channels.




## 24.5. Global logging condition

The global logging condition is used to define the conditions that must be met before any logging starts and when all logging stops. By default, the global logging condition is set to start logging when 'Engine is ON' and stop logging when 'Engine is OFF'.
You can also configure the global logging conditions to start/stop based on another Strategy, Channel, or Bit-

field Channel selectable from the dropdown menu (1). You can use the ‘restore’ tool (2) to reset the global logging conditions to the default.

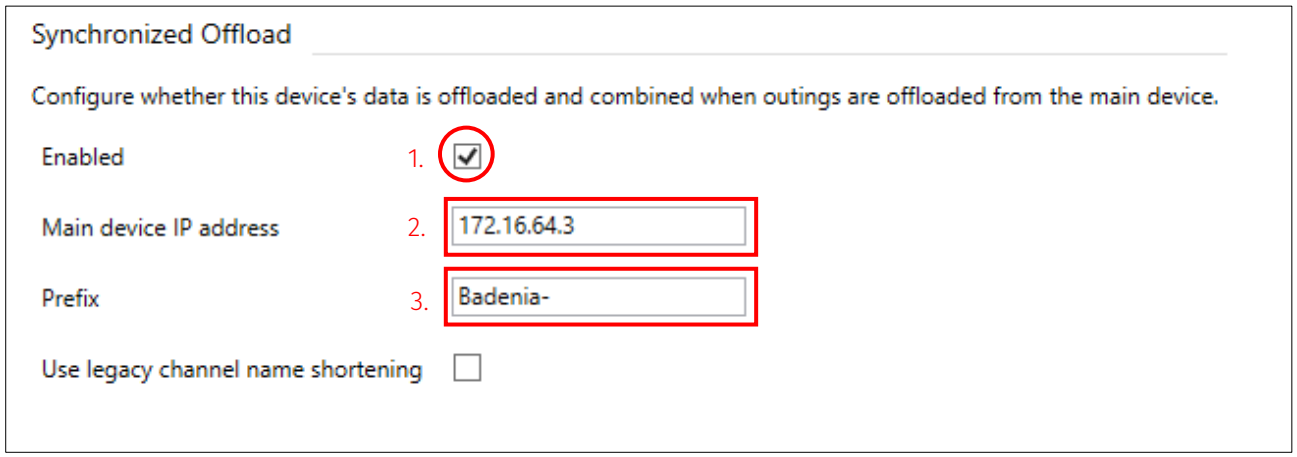



## 24.6. Offload

### Synchronized offload

If you enable synchronized offload (1) you can offload data from a secondary device with the data from the primary device. When you offload data from the primary device, both sets of channels are included, with those from the secondary device specified by a user-defined prefix.

For example, if you configure a setup for a Badenia as the secondary device and an Antares as the primary device, the Antares IP address is inserted in the primary address box (2). This allows the Badenia channels to be offloaded with the Antares channels with those from the Badenia device specified by a user-defined tag (for example, ‘Badenia-‘ (3).

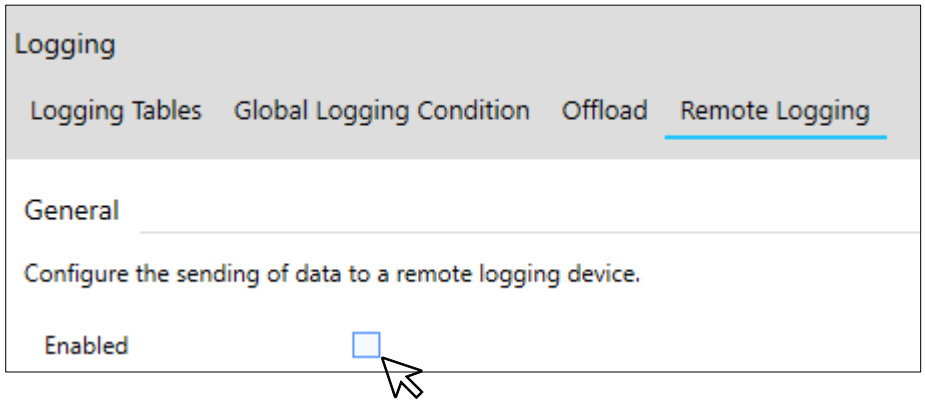



No channel data is sent between the devices, they both log locally, using their own logging conditions. It is only during data offload that the devices are 'synchronized' together. Toolset offloads from both devices individually and then combines the data in the *.pds* file.

## 24.7. Remote logging

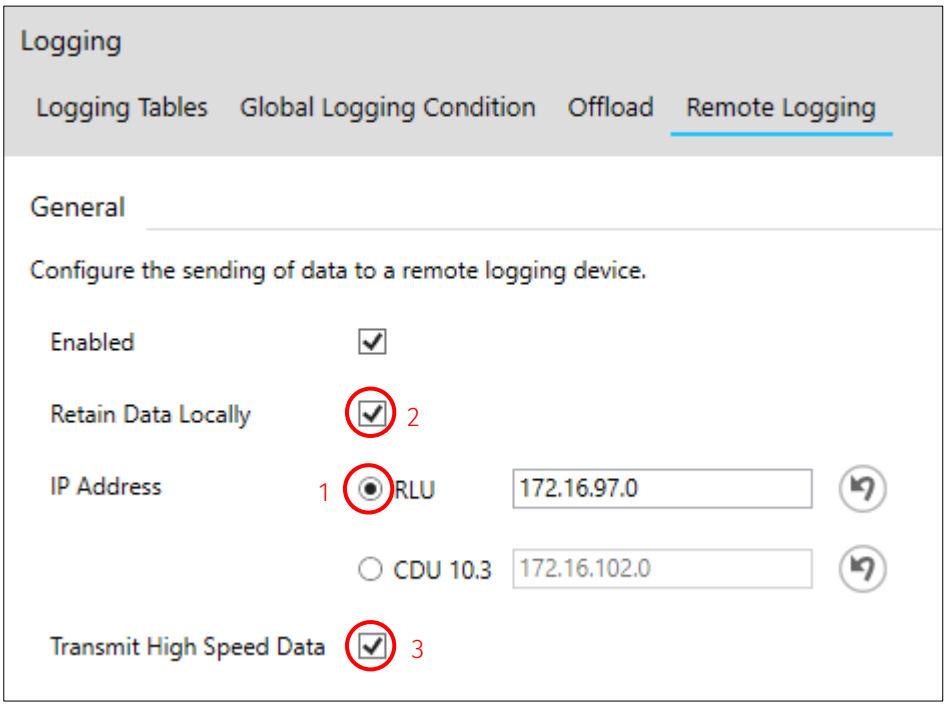
**Enable remote logging**

To log data remotely from a device, you must enable remote logging in the setup.



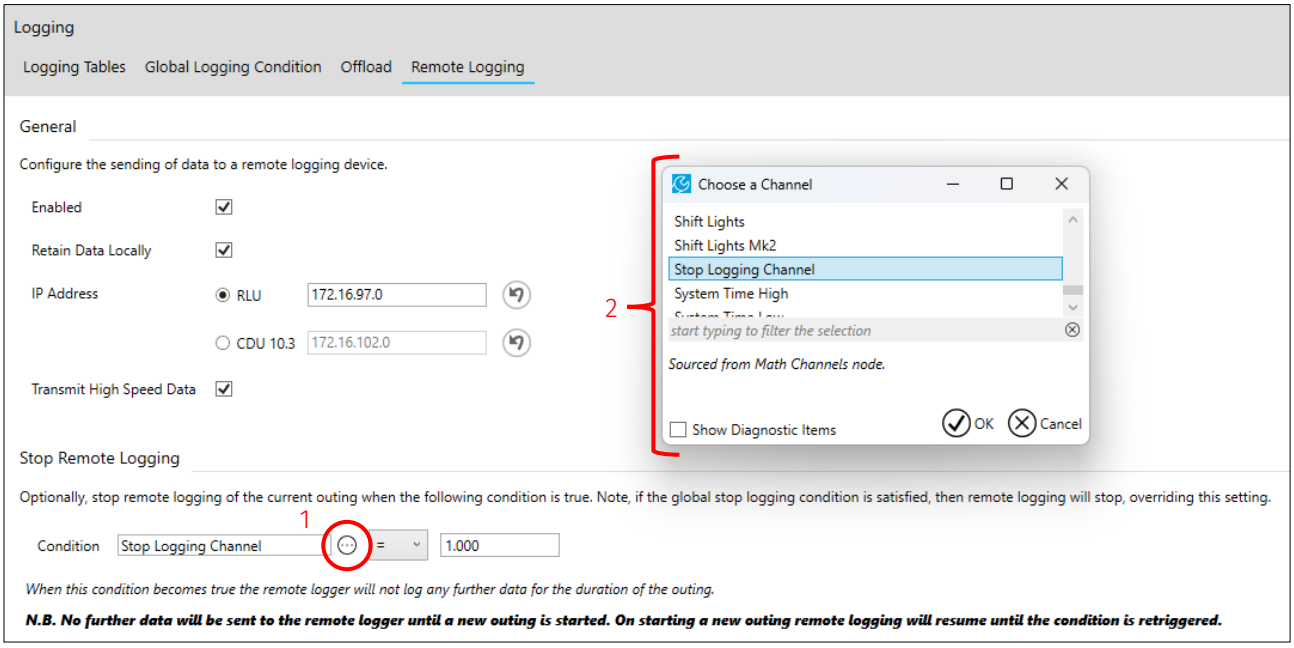
### Select remote logging device

When you enable remote logging, you can select the remote logging device (1). If you have modified the remote logging device IP address in **Device Properties** (see Devices – Device Properties) you can choose whether to retain the logged data on the primary device (2) and whether high speed channels are transmitted to the remote logging device (3).




### Optional stop logging on remote logging device

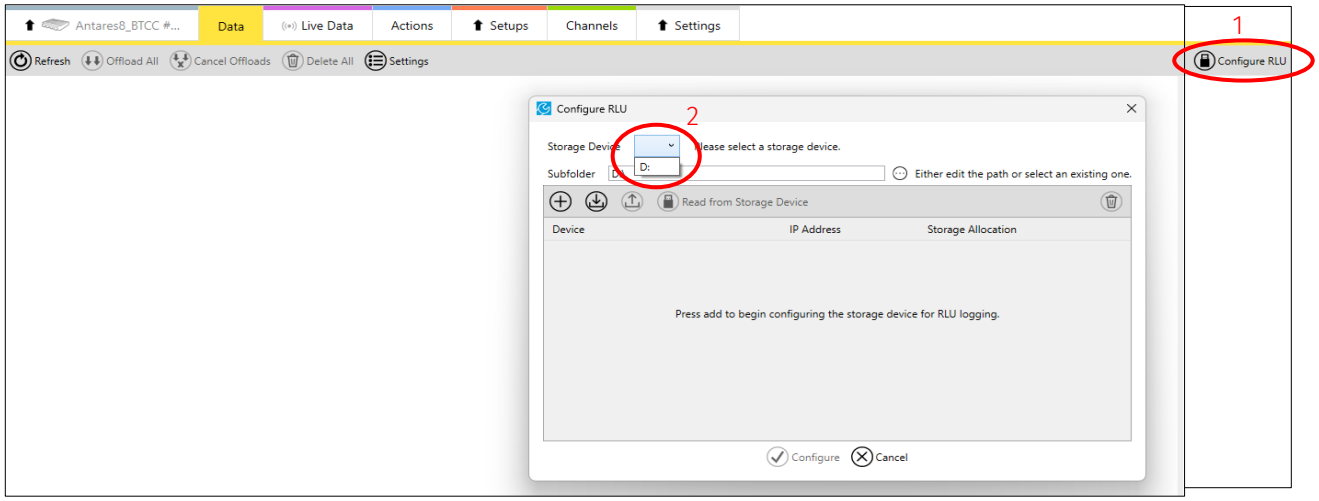
There is an option to stop remote logging, separate from the logging on the primary device. Click the three dots icon to open the dialog box (1), and then select a channel (2) to stop the remote logging.



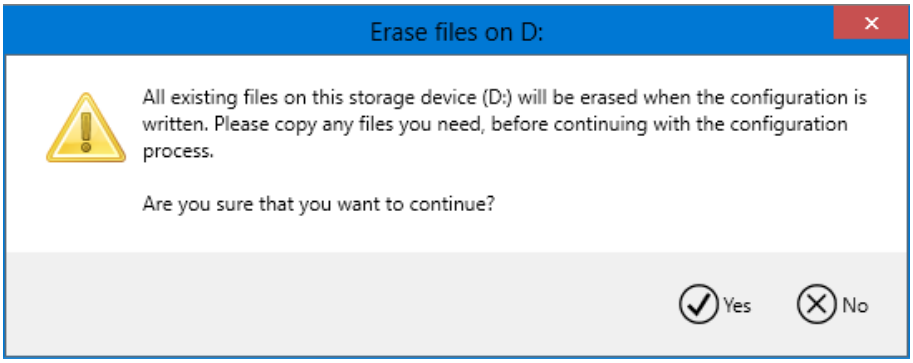
### Configure remote logging USB

You can use any USB stick to log data on a remote logging device such as the RLU or CDU10.3, but make sure the stick has sufficient storage capacity, otherwise data will be lost.

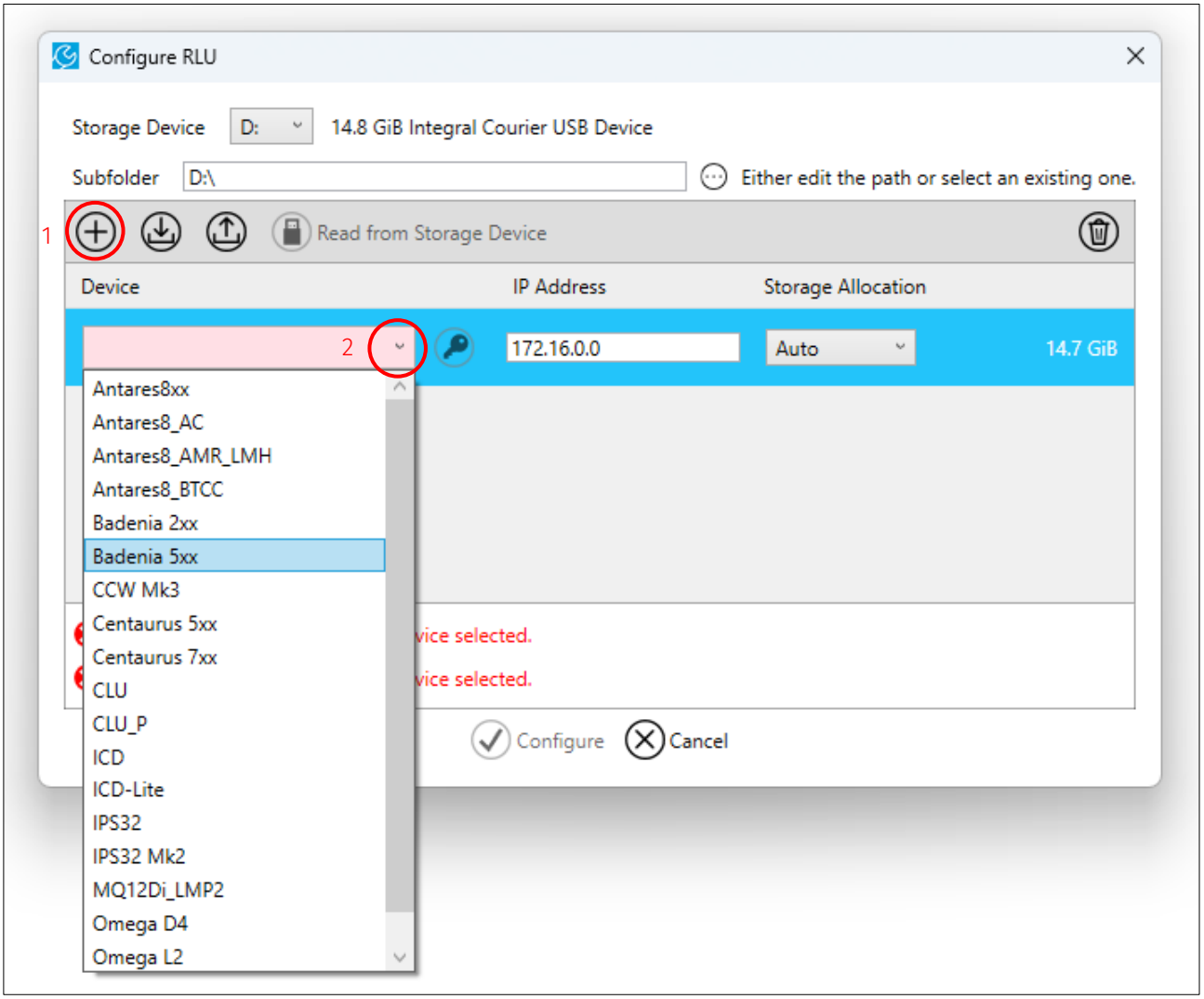
Plug the USB into a port on the PC, and then navigate to the **Data** tab in Toolset. Click **Configure RLU** at the top right (1), and then select the required device (2).



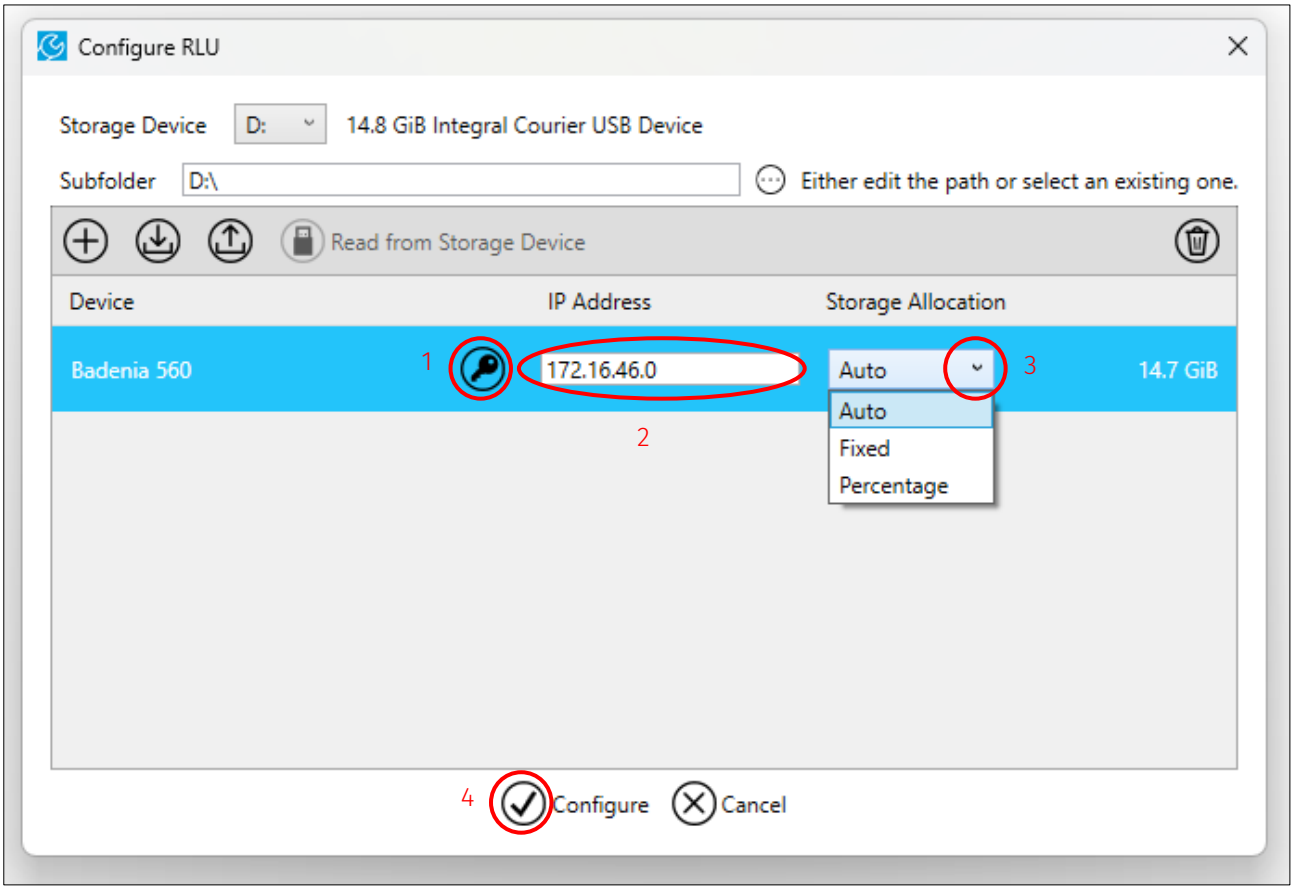

A warning dialog is displayed 'Formatting the USB for remote logging will erase all information currently on the USB.' Click **Yes** to proceed.



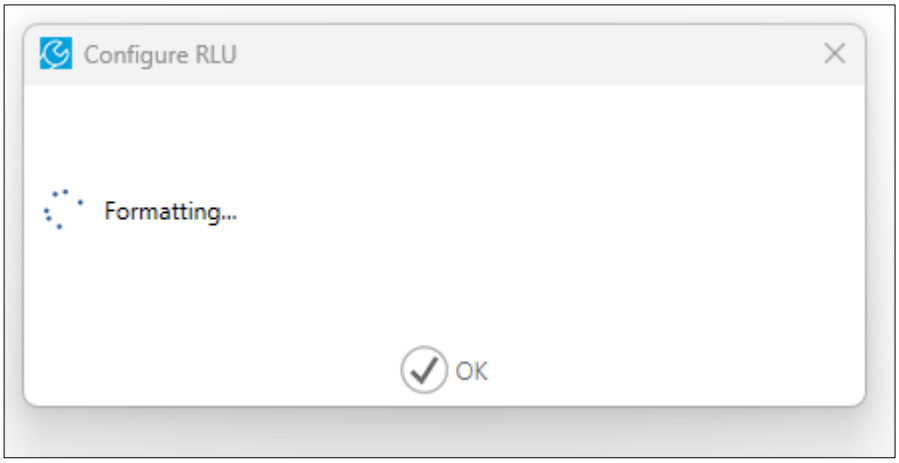
To add a new remote logging configuration, click **+** (1), and then use the dropdown menu to select the primary device (2).



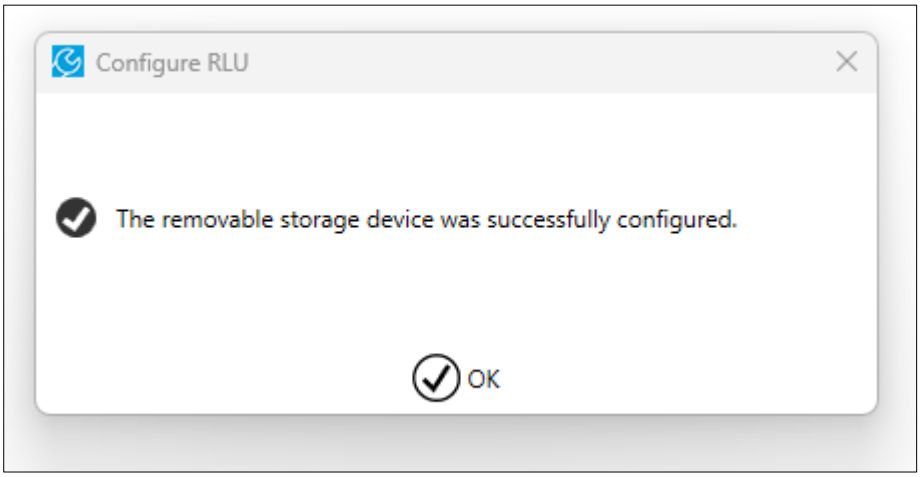
When you select the primary device, you are prompted to select the matching device variant (1). Make sure that the device IP address is correct for the primary device (2), select the storage allocation type – Auto, Fixed, or Percentage (3), and then click **Configure** (4).



Toolset starts to format the USB device for remote logging.

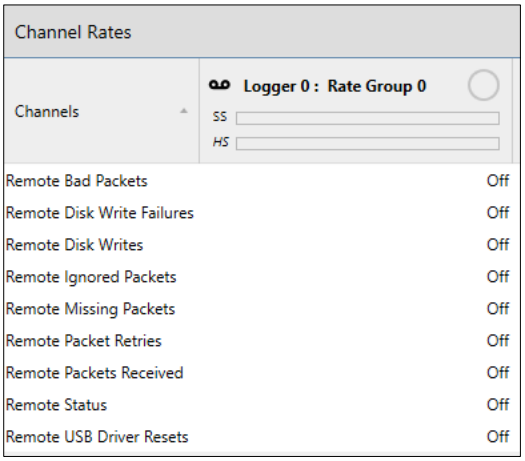



Once the USB device is formatted it is ready for remote logging.

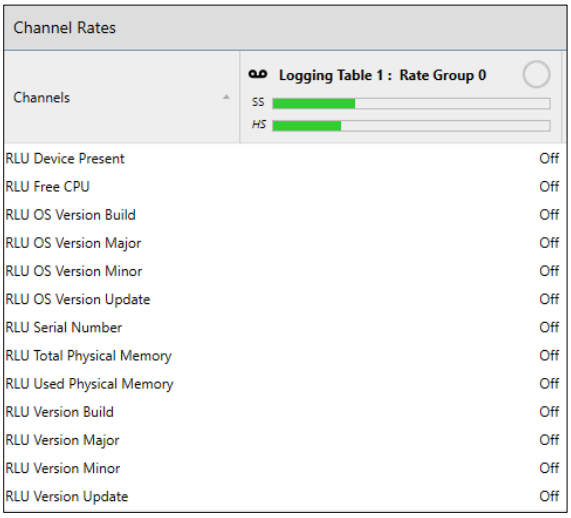


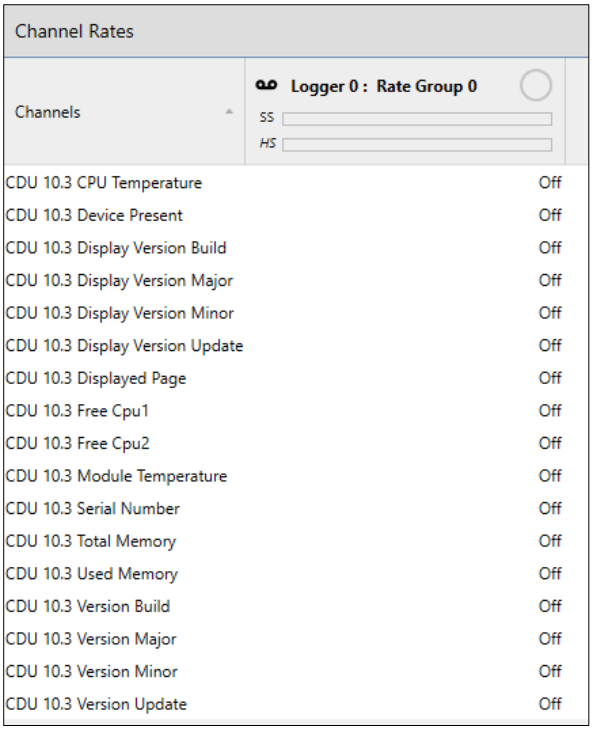

**Remote logging status channels**

When a remote logging device is enabled, Toolset automatically generates 9 logging ‘status’ channels.



Depending on the remote logging device selected (RLU or CDU10.3), device specific channels are also automatically generated.




**Note**: You must select the channel rates to enable the channels to be logged

# 25. Logic channels

The Logic Channels node offers an alternative approach to configuring control channels, simplifying the creation of control channels that are complex to create in maths channels. The following covers the two features of the Logic Channels node – condition and counter logic.

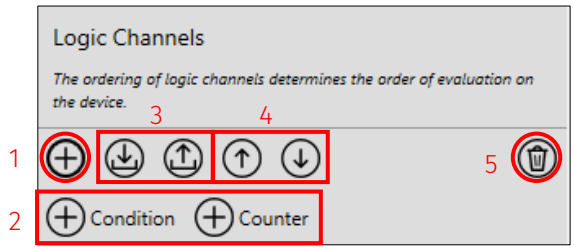
Note: Logic channels are limited to a fixed rate of 50Hz computation, but you can configure the computation rate of maths channels.

## 25.1. Add a logic channel

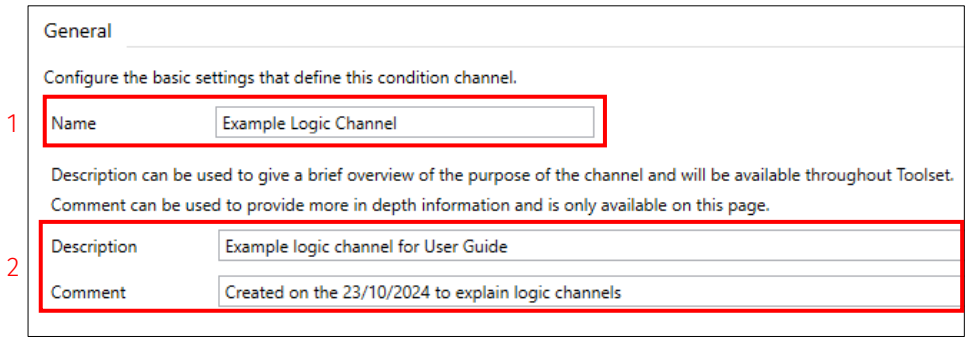
To add a new logic channel, hover over the + icon (1), and then click the required channel, either 'Condition' or 'Counter' (2). Use the 'import' and 'export' icons (3) to import and export logic channels between existing setups. If you add multiple logic channels, they can be reordered (4).

Note: - Logic channels are ordered in priority, so high priority and protected logic channels (see Setup locking – channel protection) must be at the top of the hierarchy.

Use the 'bin' tool to delete logic channels (5).




Enter a name for the channel (1) and an optional description and comment (2). Channel descriptions are used to provide a brief overview of the channel and are available to view when the logic channel is used elsewhere in Toolset. Channel comments are used to give more in-depth information about the channel and are only available to view on the Logic Channels node.

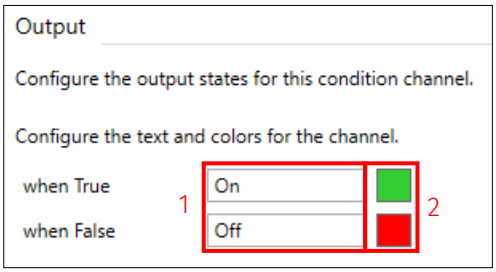



### Conditional logic

In each conditional logic channel there are two main elements - logic groups and logic gates. Conditional logic passes input channels through 'logic groups' of 'logic gates' (also known as conditions). When the groups of conditions are true, the output is true.

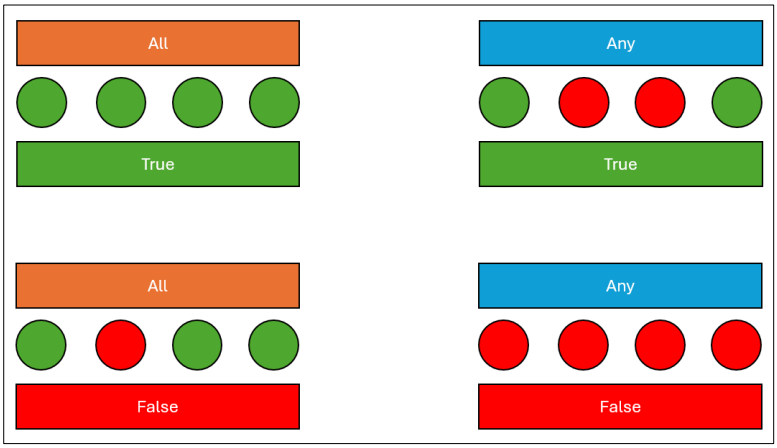
### Generated channels

When you configure the conditional logic channel, an enumerated bit-field channel is generated. You can rename the true/false enumerated states for the bit-field channel (1) and set a colour for the true/false states (2).




### Logic groups

Logic groups group conditions into ‘All’ or ‘Any’ to decide if all the conditions within a group must be met, or if only one condition within the group must be met, for the output to be true. In the logic tree, ‘All’ groups are highlighted in orange and ‘Any’ groups are highlighted in blue.




For example, the following logic channel is true when:

‘Battery voltage >=13.5V’ OR ‘Switch 1 is pressed And Switch 2 is not pressed’.

**Logic conditions**

These are the channel conditions that must be true for the output to be true.

There are many types of condition, summarised in the table below. In the logic 'tree', logic conditions are highlighted in green.

| Condition | Inputs | Description |
|---|---|---|
| Alarm Triggered | 1x Alarm | A logical output based on an alarm state (see Alarms) |
| Bitfield Comparison | 1x Bit-field Channel | A logical output based on a bit-field channel condition |
| Channel Comparison | 1x Channel | A logical output based on a channel condition |
| Strategy Comparison | 1x System State | A logical output based on a system state (see System States) |
| Logical AND | 2x Channels or System States | Comparing 2 channels (If Chan1 AND Chan2) |
| Logical OR | 2x Channels or System States | Comparing 2 channels (If Chan1 OR Chan2) |
| Logical XOR | 2x Channels or System States | Comparing 2 channels (If Chan1 XOR Chan2) |
| Bitwise AND | 1x Channels or System States | Compare a channel to a present hexadecimal value as either a binary or hexadecimal value |
| Flash | 1x Channels or System States | While a channel is active the output flashes (on/off) at a preset rate |
| Hysteresis | 1x Channels/ System States | Active when a set value is exceeded and deactivated when another different limit is reached |
| Pulse | 1x Channels or System States | On a channel edge the output will pulse for a pre-set time |
| Set/Reset | 2x Channels or System States | A channels edge drives this high and another drives this low |
| Toggle | 1x Channels or System States and 2 Override Channel inputs | A channel changes this state on each pulse. Channels can be reset to override this value |

Conditional logic channel example

The following example of a conditional logic channel is used to control a reserve fuel pump.

It is activated if ANY of these conditions are met:

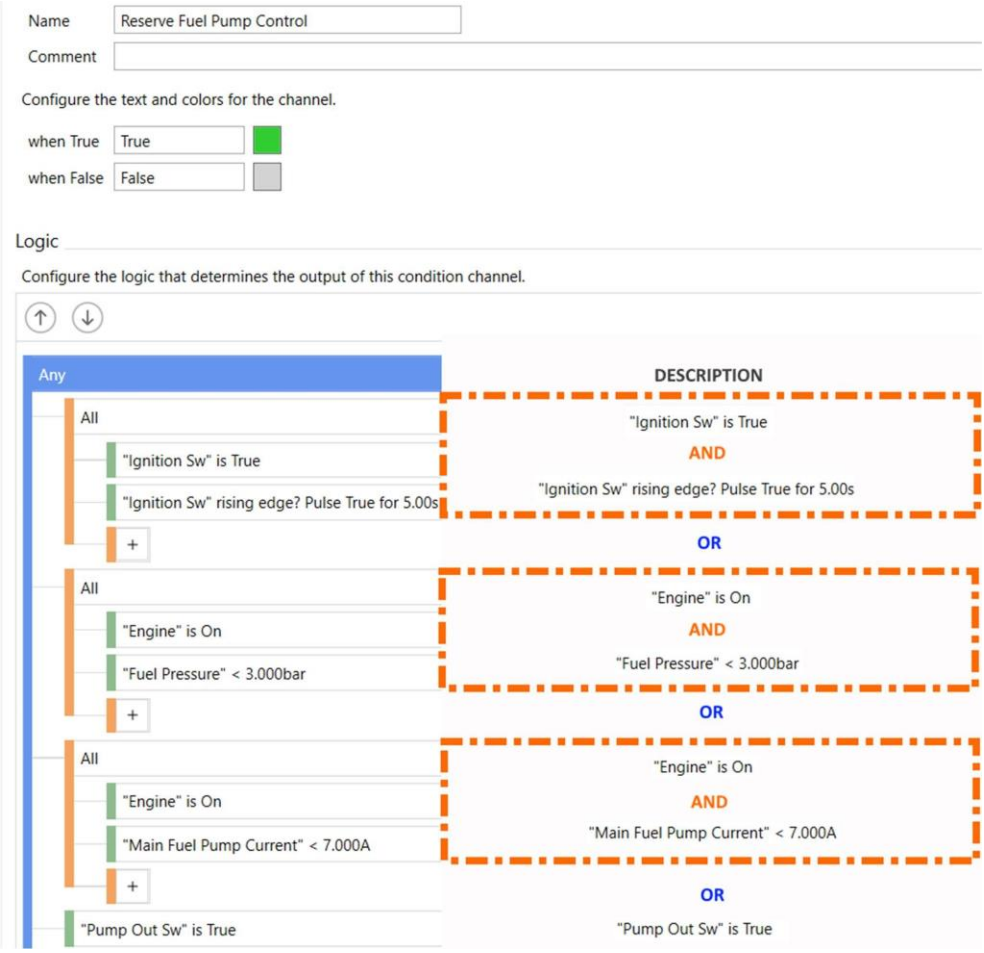
- The Ignition is pressed AND runs for 5 seconds.
- The Engine is running AND Fuel Pressure is less than 3 bar.

OR

- The Engine is running AND the Main Fuel Pump Current is less than 7 amps.

OR

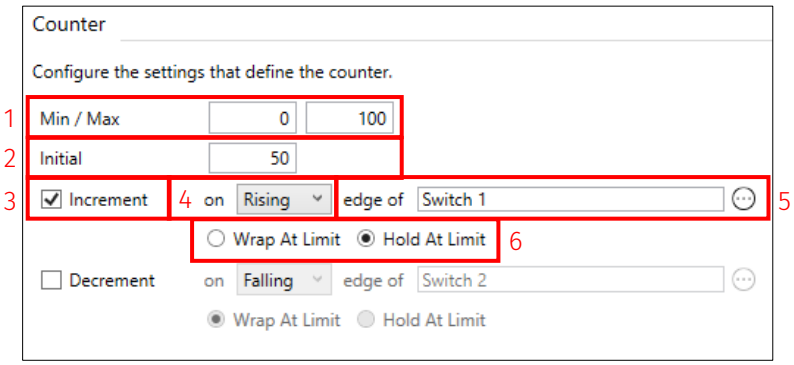
- The Pump Out switch is pressed.



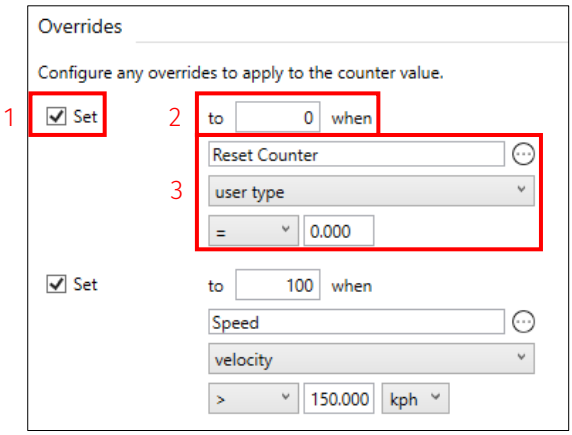

## 25.2. Counter logic

Counter logic channels offer an alternative approach for you to create a numerical counter and avoid register-based maths channels, which can be complex for new users. Counter logic channels allow you to define a counter to be incremented or decremented based on either the rising or falling edge of a channel, or system state inputs.

To configure a counter, you need to set minimum and maximum limits (0 to 65535) for the counter (1). You also need to see the initial starting value of the counter (2). You can choose to increment and/or decrement the counter by selecting either or both increment and decrement check boxes (3). Configure the counter to increment or decrement on the rising or falling edge of the channel or system state from the dropdown menu (4). You can then select the channel or system state from the 'channel browse' menu (5). Finally, you can configure the counter **Wrap At Limit** (to roll over the counter to minimum or maximum value when the maximum or minimum limit is reached), or **Hold At Limit** (hold the counter at the minimum or maximum limit when the limit is reached) (6).




You can configure force counter values if defined conditions are met. Select the two from the **Set** check box (1). You can set the value to override the counter (2), and then define the channel condition when the override happens (3).

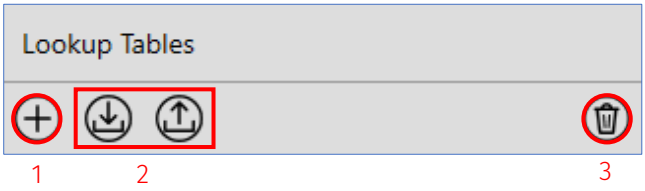


# 26. Lookup tables overview

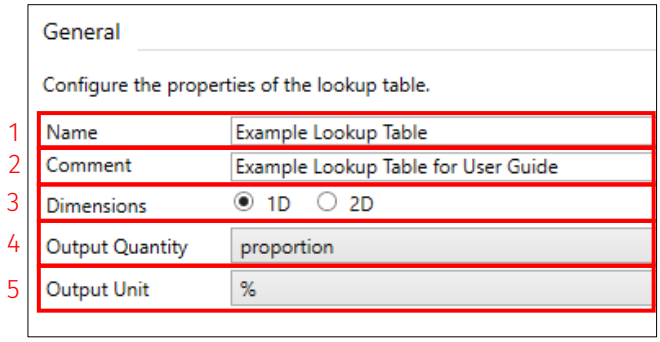
Lookup Tables (LUTs) are a useful way to use one or two input values to reference an output value. LUTs can then be used elsewhere in the setup through maths channels (see Setups – Maths Channels).

## 26.1. Add a lookup table

Click the + tool to add a LUT (1). Use the 'import' and 'export' tools to import and export LUTs between existing setups (2). Use the 'bin' tool to delete LUTs (3).




Once a LUT is added, you can configure the general settings. Add a name for the LUT (1) and an optional comment (2). Select the number of dimensions (inputs) for the LUT (either 1 or 2) from the check box (3). Select the output quantity (4) and units (5).




## 26.2. Interpolation mode

The interpolation mode determines how Toolset outputs a value when the input falls between specified discrete values. There are three options:

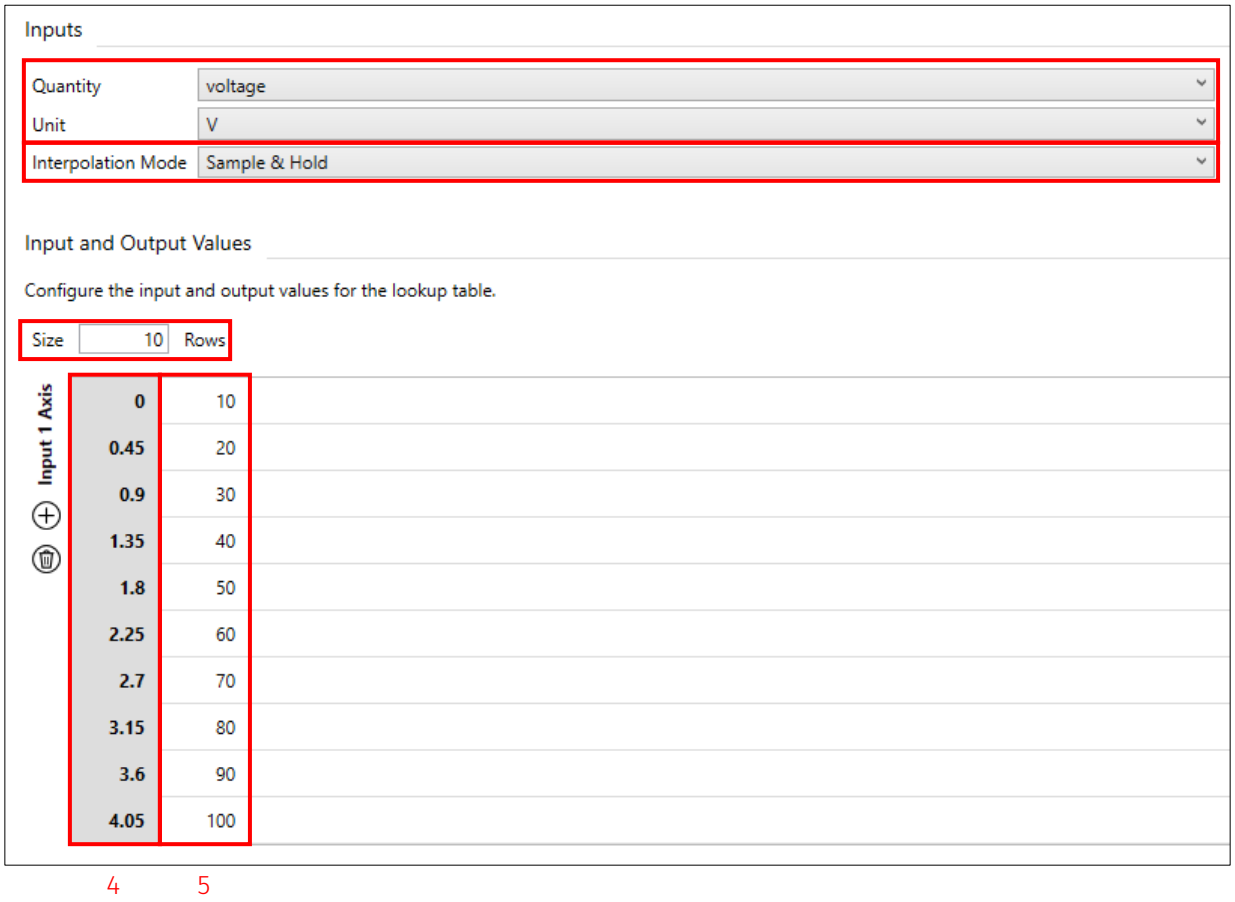
- Extrapolate – Used to estimate the output value when the input exceeds the limits of the specified input values, assuming a linear transition between after the last value.
- Interpolate – Used to estimate the output value when the input falls between specified discrete input values, assuming a linear transition between input values.
- Sample & Hold – Used to hold the output value until the next discrete input value is met.

## 26.3. One-dimensional tables

A one-dimensional table is used when one input channel is used to reference one output value.

For example, this could be rotary switch position (Volts) on a device such as the CCW Mk3, correlating to a screen brightness (%) setting.

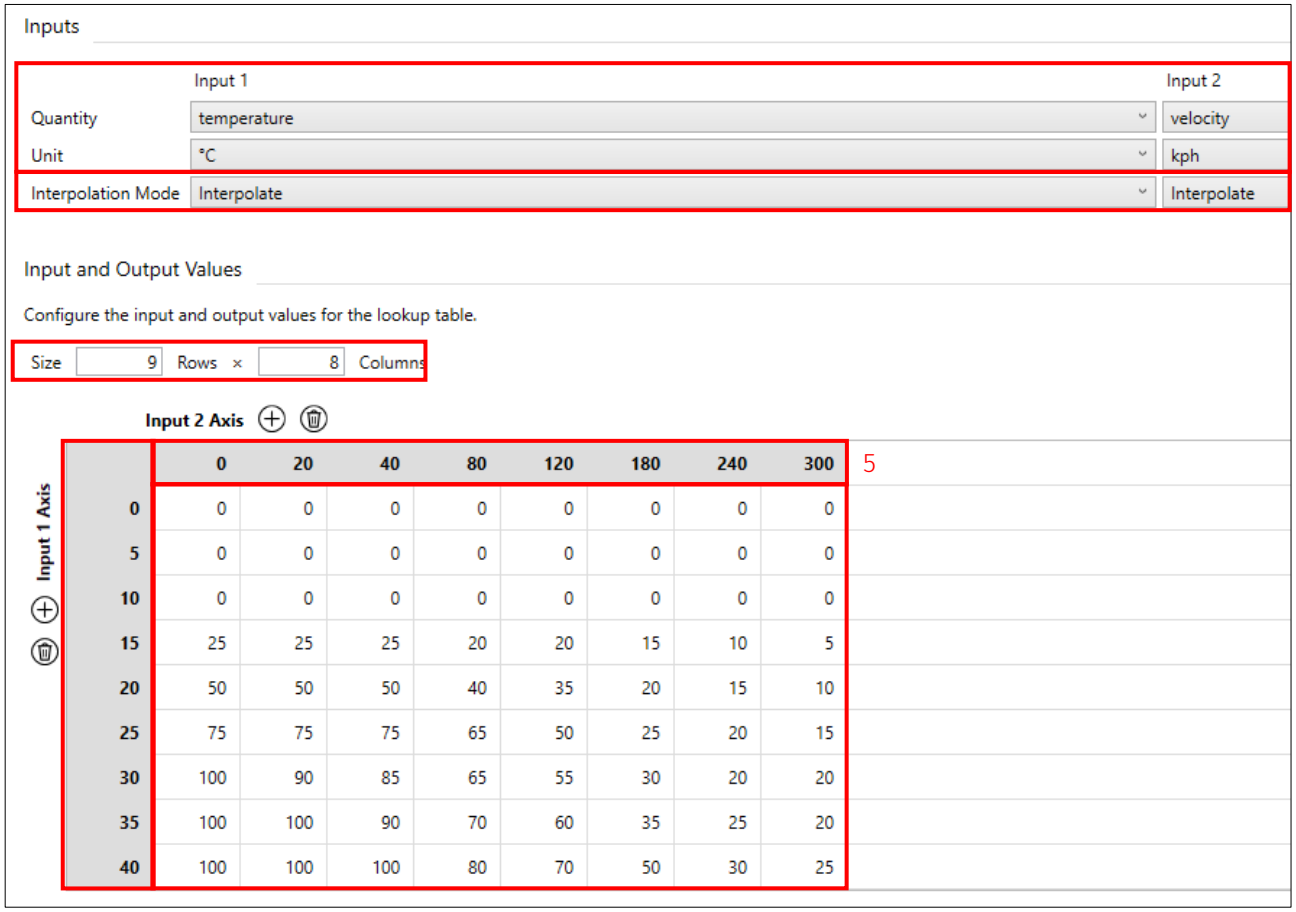
Set the input value units (1) and then the interpolation mode (2). Enter the required number of rows in the box (3). Finaly, fill in the input (4) and output (5) axis increments.




## 26.4. Two-dimensional tables

A two-dimensional table is used when two input channels are used to reference one output value.

For example, this could be used to control cooling fan duty (%) where ambient temperature (°C) is on input axis 1 and speed (kph) is on input axis 2.




Set the input value units for the two input axes (1),and then set the interpolation mode (2). Enter the required number of rows and columns in the boxes (3) Fill in the input axis 1 (4) and input axis 2 (5) increments. Finally, fill in the output values (6).

## 26.5. Create lookup tables in Excel

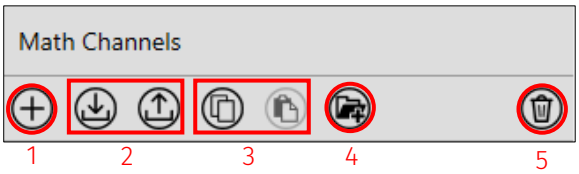
For large LUTs or complex LUTs that require calculations to define the inputs, it is possible to create the LUT in Excel and then copy and paste it Toolset. This can save a lot of time and makes creating LUTs simpler.

# 27. Maths channels

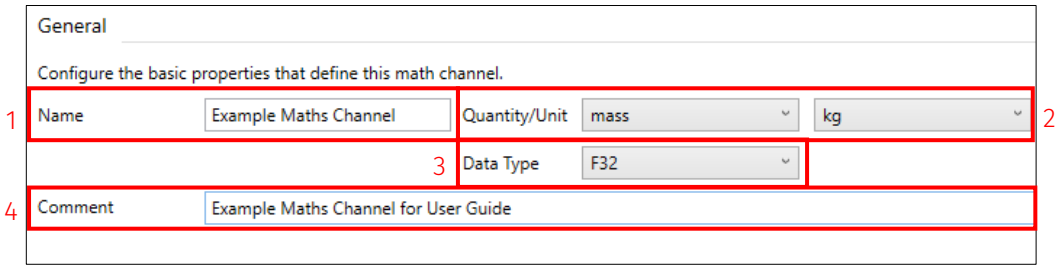
Maths channels allow you to configure mathematical inputs, together with mathematical manipulation of channels using mathematical functions. The following provides a generic overview and some examples of various typical maths channels.

## 27.1. Add a maths channel

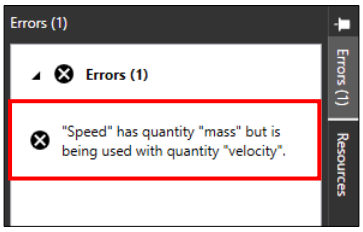
To create a new maths channel, click the **+** button (1). Use the 'import' and 'export' tools to import and export maths channels between existing setups. You can import and export multiple channels as a group (2). Copy and paste maths channels with the copy and paste tools (3.). You can add math channels to groups to aid setup organisation and structuring (4). Use the 'bin' tool to delete maths channels (5).



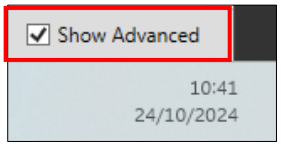

Once a maths channel is added you can configure the name of the channel (1), the units of the channel (2), select the data type (3), and add an optional comment about the channel (4).




Note 1: If maths channels are used for system channels, they may be required to have specific units or a specific data type. See the Errors menu to check for unit and data type errors.

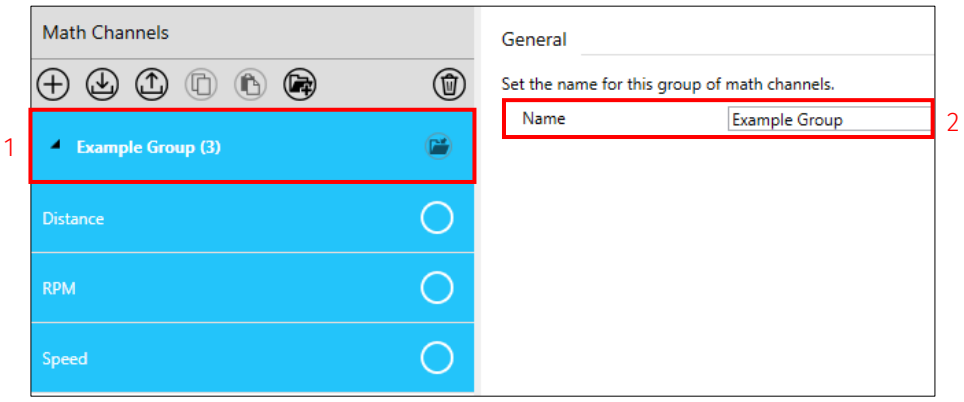



Note 2: If the Data Type menu is not shown, make sure that the Show Advanced check box at the bottom right is selected.

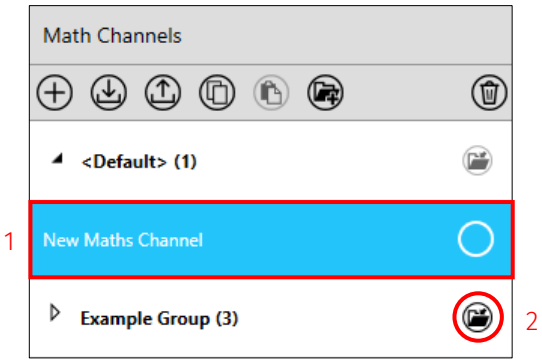


## 27.2. Group maths channels

When a maths channel is created, it is put in the <Default> group. To rename the group, click the group name (1) and enter a new name (2).




Select a maths channels (1) and use the 'Add to group' tool (2) to move that channel.

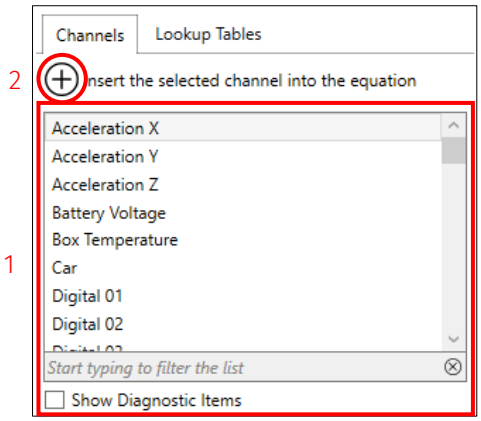



## 27.3. Maths channel equation

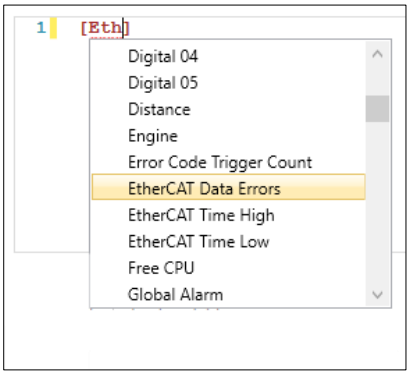
The equation box is where you enter channels equations. All the calculations are performed here. Equations can be as simple as a single value or a channel reference, or as complicated as several choose statements and registers with mathematical functions.

### Channels

You can add channels to the equation from the available channels list (1) and search a channel with the standard or magic search functions. Click the **+** tool to add the channel (2) or double-click a channel from the list.

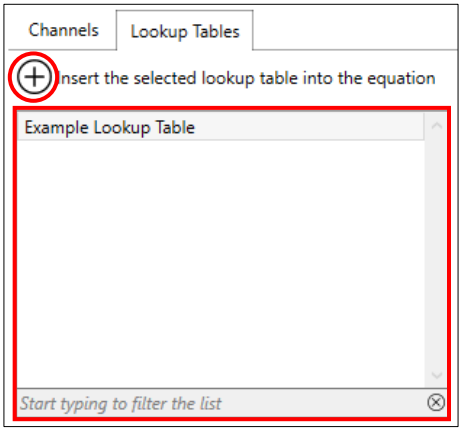



Tip: You can type '[]', and click 'CTRL + Space' and enter an equation within the brackets to add and search for a channel via the channel search menu. Enter a channel name to filter the search.




### Lookup tables

Lookup tables (LUTs) can be added to the equation from the available LUTs list (1). You can search for LUTs with the standard or magic search functions. Click the **+** tool to add the LUT (2) or double-click from the LUT list.




A LUT must be written in the following syntax:

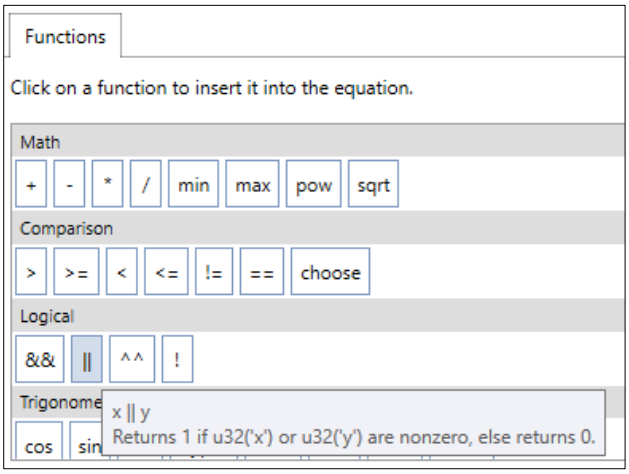
lut('*LUT Name*', '*First dimension reference*', '*Second dimension reference**')

*If using a 2D LUT.

### Functions

The **Functions** menu lists all the available mathematical operators to be used in the equation. The list of functions is extensive and includes general mathematical, logical, bitwise operators, and trigonometric operators.

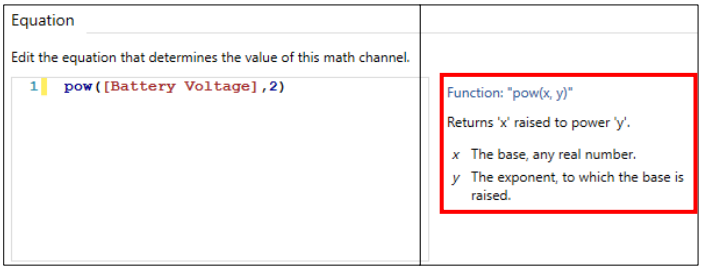
Hover over a function in the menu to display a brief description about the function. Click the function to add it to the equation. You can also type functions into the equation.




### Function help

Function help can help you to configure maths channels in the correct format and to help diagnose any syntax errors in the equation. The help displays a small description about the selected function in the equation and provides an example and breakdown of how the function syntax format (1).

**Note**: The yellow highlight next to the line number indicates a change since the maths channel was last opened.




## 27.4. Rates

The **Rate** section is used to configure the calculation rate of the maths channel. There are two settings:

- **Inherit Rate From** – The calculation rate is set by the rate of the selected channel.
- **Fixed Rate At** – You can define the calculation rate from a list of the available rates.

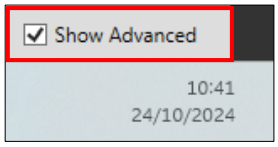
The **Consumption Rates** section is used to configure the maximum rates at which the **input** channel(s) are consumed. There are three key options:

- **My Rate** – Uses the calculation rate of the maths channel set in **Rate**.

- Auto – Uses the maximum rate of the input channel.
- 1Hz-20khz – You can select the input channel(s) at 1Hz-20kHz.

Note 1: If the Inherit Rate From input channel option is selected for the calculation rate, then the consumption rate for the input channel is automatically set to 'My Rate'.

Note 2: If the Data Type menu is not shown, make sure that the Show Advanced check box at the bottom right is selected.

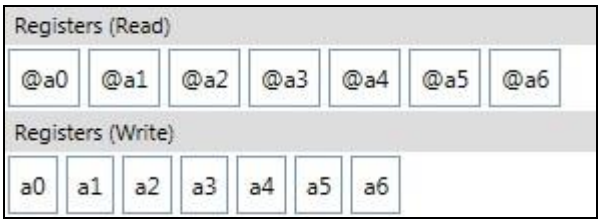



Tip: If for example, you want to log the Maths channel at 20Hz, containing an input channel that is being produced elsewhere on the device at 50Hz, then the calculation rate needs to be set to 100Hz (to satisfy both requirements). Set the calculation rate to be a common (or least common) multiple of all uses of the channel.

## 27.5. Registers

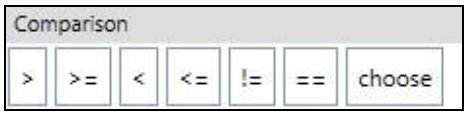
Registers act as short-term memory variables, enabling you to produce channels dependent on historic values.

In the Functions menu there is the option to add in up to seven 'read' and 'write' registers named from a0 to a6. These registers have a default value of zero whenever the device is reset (power cycle, new setup sent, and so on). The registers are evaluated at the calculation rate of the channel. For example, if a channel is set to 100Hz, the registers are updated every 10ms.




## 27.6. Choose statements

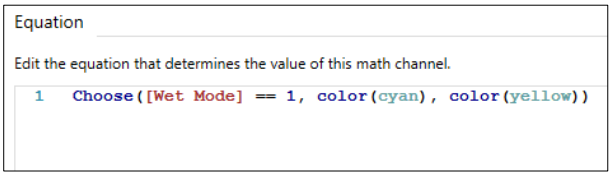
Choose statements are an 'if/else' statement. In other words, 'If' something is true, do X, otherwise do Y. Multiple choose statements can be 'nested' to support more complex comparison functions. The 'Choose' function is in the Comparison section of the Functions menu.



## 27.7. Color control

You can configure a maths channel can be configured for dynamic colour control of a display item or for shift lights. From the colour function in the equation, a colour can be referenced by its name (for example, cyan).

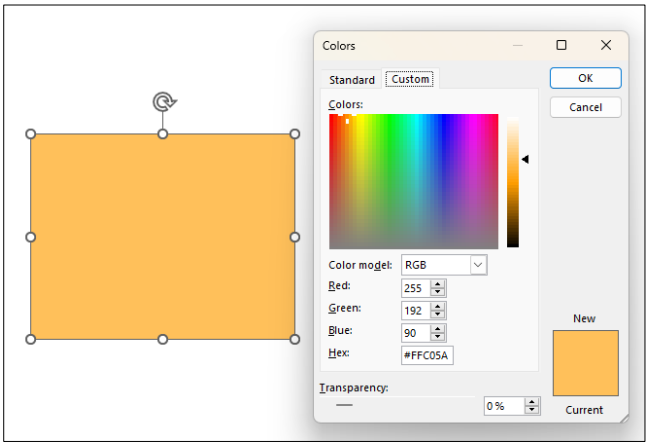
Note: The data type must be U32.




You can also select a colour by its RGB value as a U32 decimal number, where blue is the first byte, green is the second byte, red is the third byte, and brightness is the fourth byte.

For example, pure red at full brightness would equal 4,294,901,760


The following peach colour combining all three R, G and B portions at 50% brightness would equal 268,419,162.




You can also write the colour in bit-wise format where:

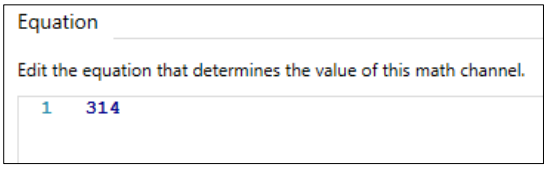
color = (("BRIGHTNESS VALUE"<< 24) | ("RED VALUE" << 16) | ("GREEN VALUE" << 8) | ("BLUE VALUE"))

color = (("127"<< 24) | ("255" << 16) | ("192" << 8) | ("90"))

## 27.8. Example maths channels

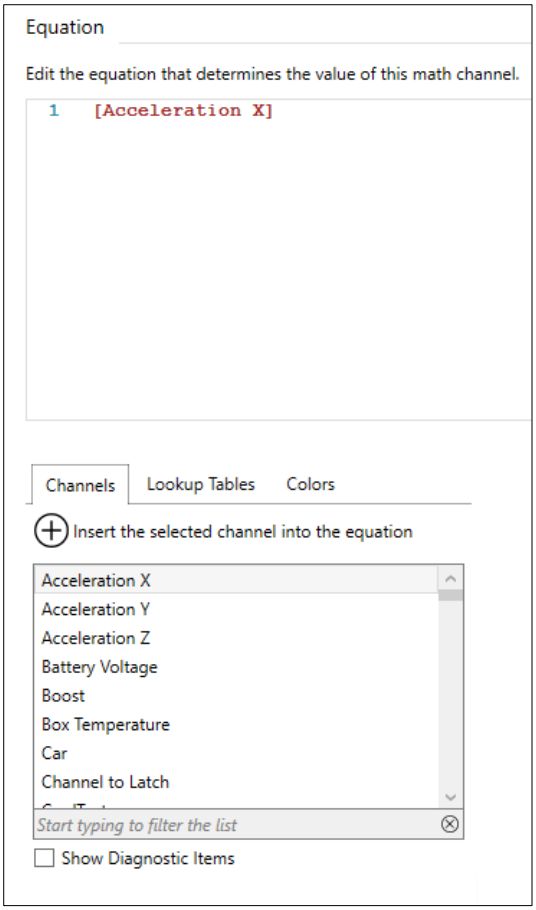
**Example 1 – Static value**

A maths channel can be a static value. This channel can be referenced and always outputs the configured number. A fixed rate of 1Hz is recommended for this type of channel because there is no computation, thus saving logger CPU processing.



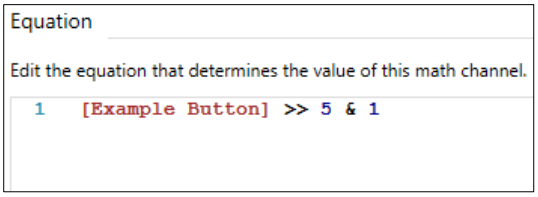

**Example 2 – Channel reference**

A maths channel can be another channel or event. You can type the channel name or select it from the list below. The syntax for using a channel is as follows: **[Channel Name]**. This type of maths channel can be used to rename native device channels to User defined name. It can also be used to increase the rate of logic channels (Logic Channels are calculated at a maximum of 50Hz).



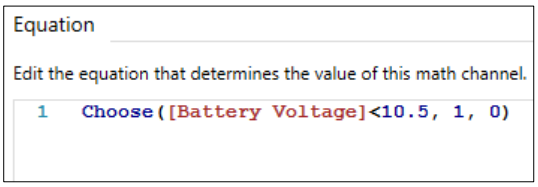
**Example 3 – Bitwise channel**

You can use a maths channel to 'Shift' and 'AND' a bit-field channel and return the value. For a button where a 'Click-Latched' option is bit 5 (see Setups – Buttons), it can be 'shifted right' by 5 and 'ANDed' with 1, to return a value of 1 when the 'Click-Latched' button is true.



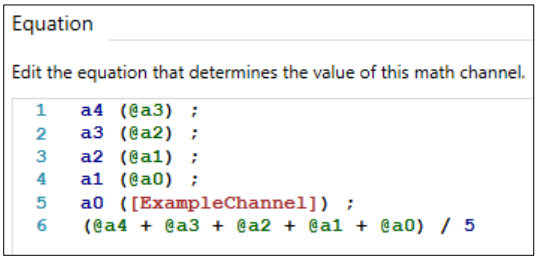

**Example 4 – Choose statement**

You can configure a maths channel to complete an 'If' statement decision, like a logic channel. Use the 'Choose' function to test a statement to output a value if true, and then output another if false.




**Example 5 – Registers – simple filter**

You can use a maths channel to apply an average filter to a noisy channel by using registers.




When this math channel is evaluated, it reads the functions from top to bottom, so register **a4** is evaluated first, then **a3**, and so on. The register name is given first, followed by the equation or value that sets the register value in parentheses. If you need to read the value of a registers value once it has been defined, the register name must be preceded by an **@**. The register name then turns green to show that it has been used. For example, the first line of this channel equation defines register **a4** as the previous value of register **a3**.

The fundamental function within this channel is line 5, **a0 ([ExampleChannel])**, where **[ExampleChannel]** represents the channel that requires filtering. This function sets **a0** as the instantaneous sample of the channel **[ExampleChannel]**. On the next channel sample, **a1** is populated with this first channel value and **a0** is replaced with a new channel value. Each channel value then propagates up the registers from **a0** to a4, where **a0** holds the most recent value and **a4** holds the most historic value (which in this case is four samples before **a0**). If this channel is calculated at 100Hz, then the filter response is like a 20Hz roll off but with a time (phase) delay of approximately 4ms, which, incidentally, may affect dynamic data analysis. Finally, the contents of the registers are summated, and divided by the number of registers used, to produce an average value.

**Example 6 – Registers - counter**

An alternative approach to a logic channel counter is to create a counter with registers in a maths channel. The following channel counts from 0 to 100 and resets to 0 when it reaches at 100.

Equation

Edit the equation that determines the value of this math channel.

```
1  a0 (choose( @a0 < 100 , @a0 + 1, 0)) ;
2  @a0
```

When this maths channel is first evaluated, a0 is equal to 0. In other words, the 'choose' statement means that, if a0 is less than 100, then increment a0 by 1, otherwise set a0 to 0. If the rate of this channel is set to 1Hz, then the output increments by 1 each second until it reaches 100 and then resets to 0. If the rate of this channel is set to 100Hz, then the output increments by 1 each 10ms second until it reaches 100 and then resets to 0.

**Example 7 – Registers – last beacon code seen**

This channel captures the last valid code seen by the beacon input. The following equation is used.

Equation

Edit the equation that determines the value of this math channel.

```
1  a3 (F) ;
2  a2 ( (@a2 * (@a3 > 65000) ) + (@a3 * (@a3 < 65000)) ) ;
3  @a2
```

The variable 'F' is the channel Beacon Raw. In this instance. a2 has two possible values in this function. If 'F' is greater than 65000, a2 remains at its original value. If a3 is greater than 65000, a2 is set to output the value of F' which is the new beacon code.

**Example 8 – Registers – beacon counter**

This is a monotonic count of all the beacons seen since the last logger reset. It can be easily modified to give just the number of end of laps beacons or the number of split beacons. The following equation is used where 'F' is again Beacon Raw.

Equation

Edit the equation that determines the value of this math channel.

```
1  a4( (@a4 + (@a4!=0)) * (@a4 < 100) ) ;
2  a1( @a1 + (@a4==2) ) ;
3  a4( @a4 + ((@a4==0) * (F < 65000)) ) ;
4  @a1
```

This can be thought of as a 'state engine' with 101 states. a4 holds the state (between 0 and 100) and a1 holds the beacon count. This channel creates the following state operation:

Do nothing. If 'F' is a valid code (<65000), then increment a4.

Increment **a4.** If **a4** is greater than 100, set **a4** = 0.

Increment **a1,** increment **a4**. If **a4** is greater than 100, set **a4** = 0.

Increment **a4**, if **a4** is greater than 100, set **a4** = 0.

Increment **a4,** if **a4** is greater than 100, set **a4** = 0 (which in this state is true so we return to state 0, that is **a4** is only incremented when **a4** is non-zero, or when **a4 i**s zero AND '**F**' is a valid code).

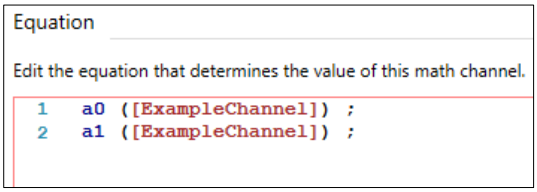
## 27.9. Advice For registers

### Synchronization

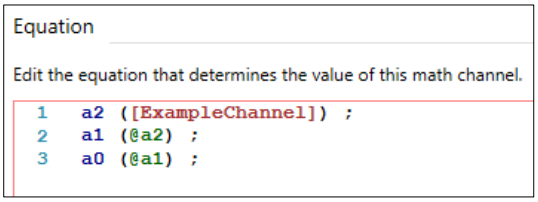
Maths channels are sampled sync executed channels which operate in a standard from top to bottom. Therefore, sampling the same channel at different functions (different lines of the equation) may produce different answers. This is because the functions are calculated as they are read, from top to bottom; they are not all calculated at the same time. As a result, cumulative channels can be influenced by a different start point for each sample.

### Input channel sampling multiple time per output channel

Sampling the same channel in more than one function (different lines of the equation) may produce different answers. This is because the functions are calculated as they are read, from top to bottom. In other words, they are not all calculated at the same time. As a result, cumulative channels can be influenced by a different start point for each sample.




Any input that changes its value midway through the calculation of the output channel can cause a problem. Careful consideration of the phase between channels must be taken when the sampling rates of the inputs and outputs are different. For example, **a0** and **a1** are not necessarily the same, as **[ExampleChannel]** may have been resampled between the calculation of **a0** and **a1** if the sampling rates are different. Whereas, the following equation ensures that all the registers hold the same value.



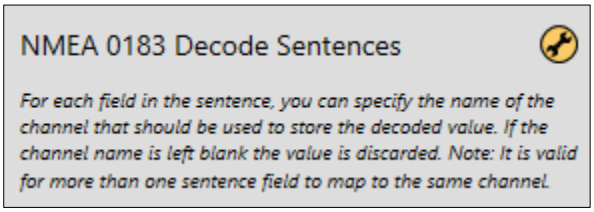

This is because once the value of **[ExampleChannel]** is assigned to **a2**, the values of the following registers take their value from **a2** rather than the channel itself. As the sampling rate of all the registers are held at the same rate defined by the output channel, this prevents the value of **[ExampleChannel]** being used throughout an equation from changing.

# 28. NMEA 0183

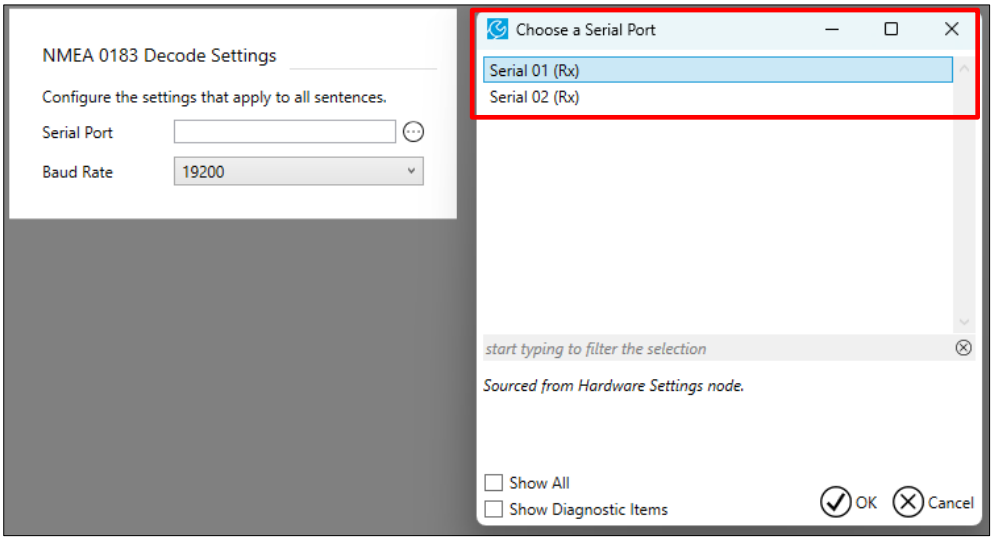
The **NMEA 0183 Decode** node is used to configure a GPS input.

## 28.1. Configure a GPS Input

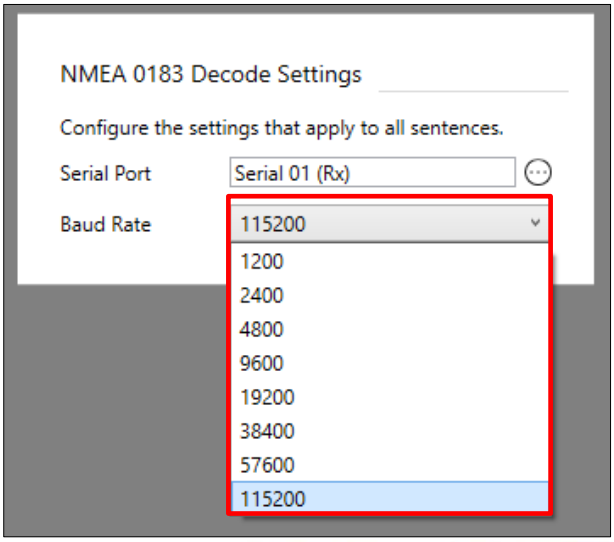
Open the **NMEA 0183 Decode** node and the 'wrench' tool to setup the GPS input



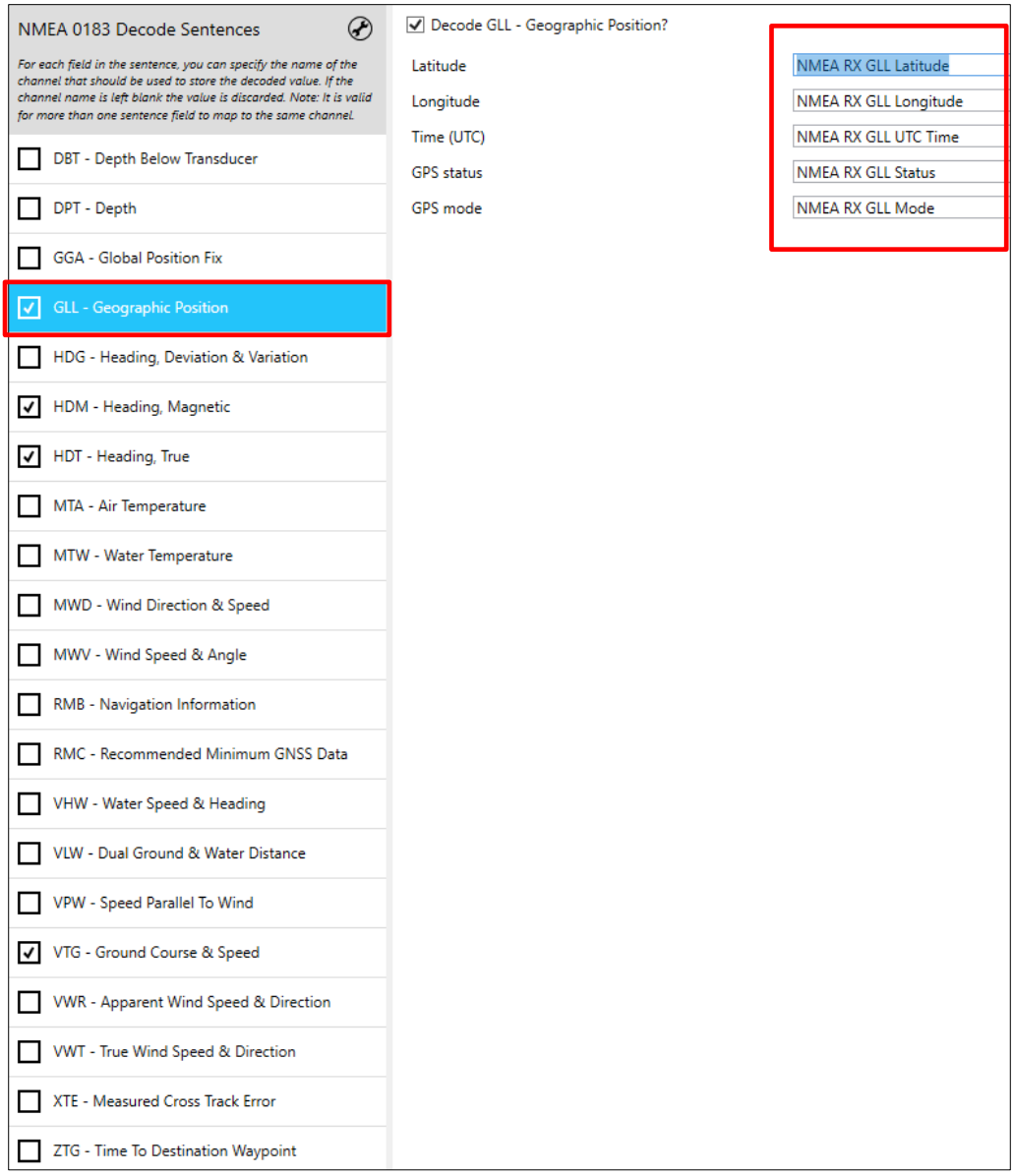

Select the **Serial Port** to which the GPS is connected.



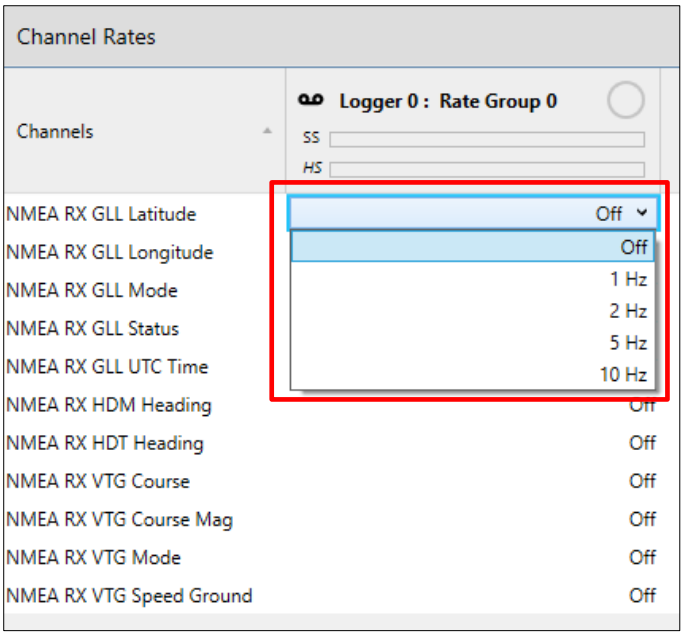

Next, select the **Baud Rate** for the GPS input from the dropdown menu.



You can then click the check boxes to select the NMEA sentences to be decoded (1). You can edit the channel names as required (2).




Once you select the NMEA sentences, the channels are displayed on the **Channel Rates** node. You must set the channel logging rate for the data to be logged (1).



# 29. Power outputs

The **Power Outputs** node is used to configure the power outputs for Power Box devices. Refer to the relevant PIS sheet for your Power Box device to find the ratings for each pin.

Centaurus : Centaurus Product Information Sheet

## 29.1. General Tips

Before you configure the power outputs pay attention to the following tips:

**Output type** – Check if the output is a high-side drive (HSD - switches the positive line) or a low-side (LSD - switches the ground line) output. On the Centaurus, CN1-4, CN1-11, CN1-12, CN1-19, and CN1-20 are low power low-side outputs.

**Current rating** – Each output has a maximum current rating. Overloading an output can damage the device. Choose outputs rated adequately for the devices they control. Group output pins to increase current rating if required (see Configuring a Grouped Pin Output)

**Continuous vs. Peak Current** – Some outputs can handle larger inrush current and longer surge times than others. Ensure the continuous and peak current consumption of the powered device is known before you configure outputs.

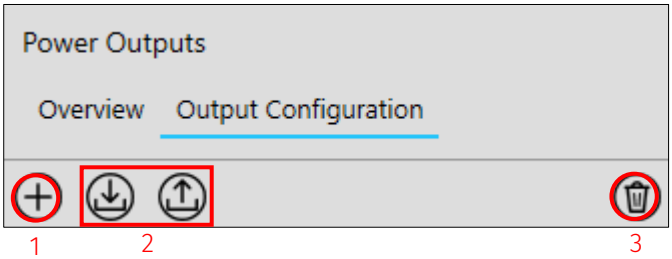
**Heat Dissipation** – Install the Power Box in a location with adequate ventilation and consider ambient temperature limits to avoid thermal shutdowns.

**Fail-safe Configurations** – For critical functions (for example., engine cooling fans or fuel pumps) consider adding redundant circuits or configuring fail-safes to make sure that these devices remain functional even if one output fails.

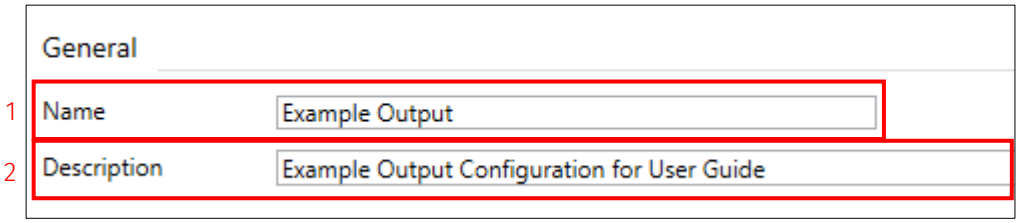
## 29.2. Output configuration

### Add a new output

To configure a new power out, navigate to the **Output Configuration** menu. Click the **+** tool to add a new output (1). You can use the 'import' and 'export' tools to import and export configurations between existing setups (2). Use the 'bin' tool to delete unwanted outputs (3).



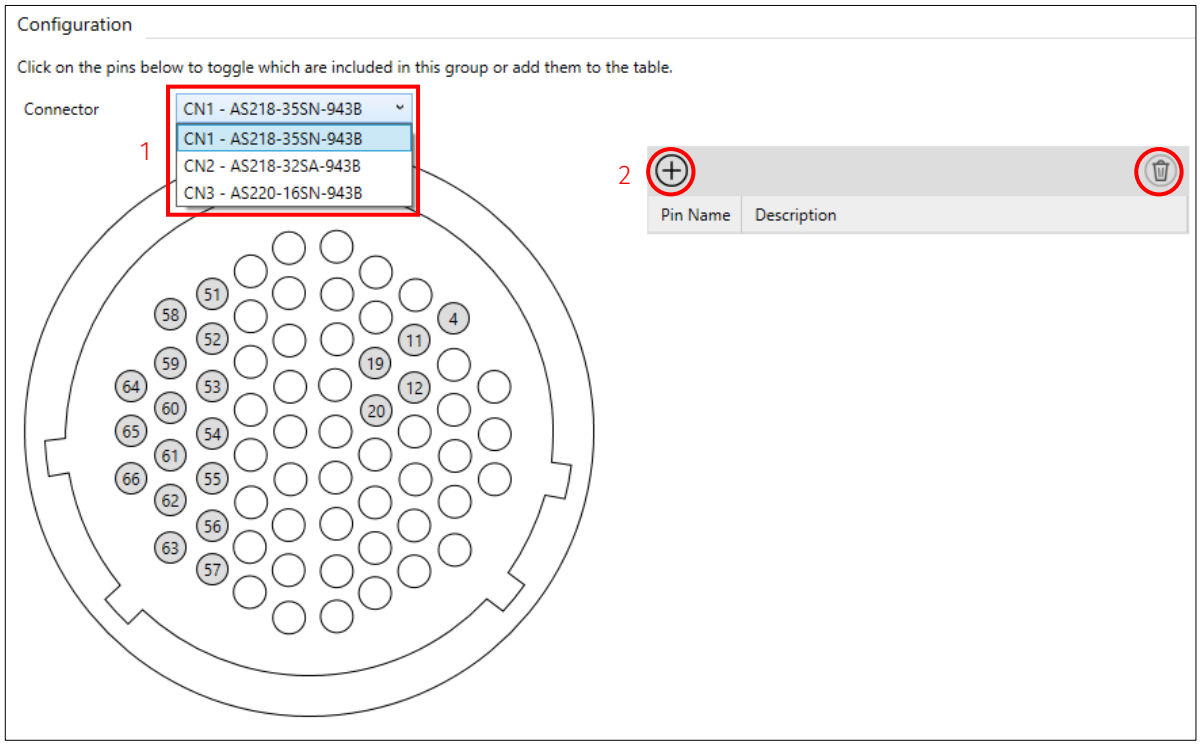
Once an output is created you can name it (1) and add a brief description about the output (2).



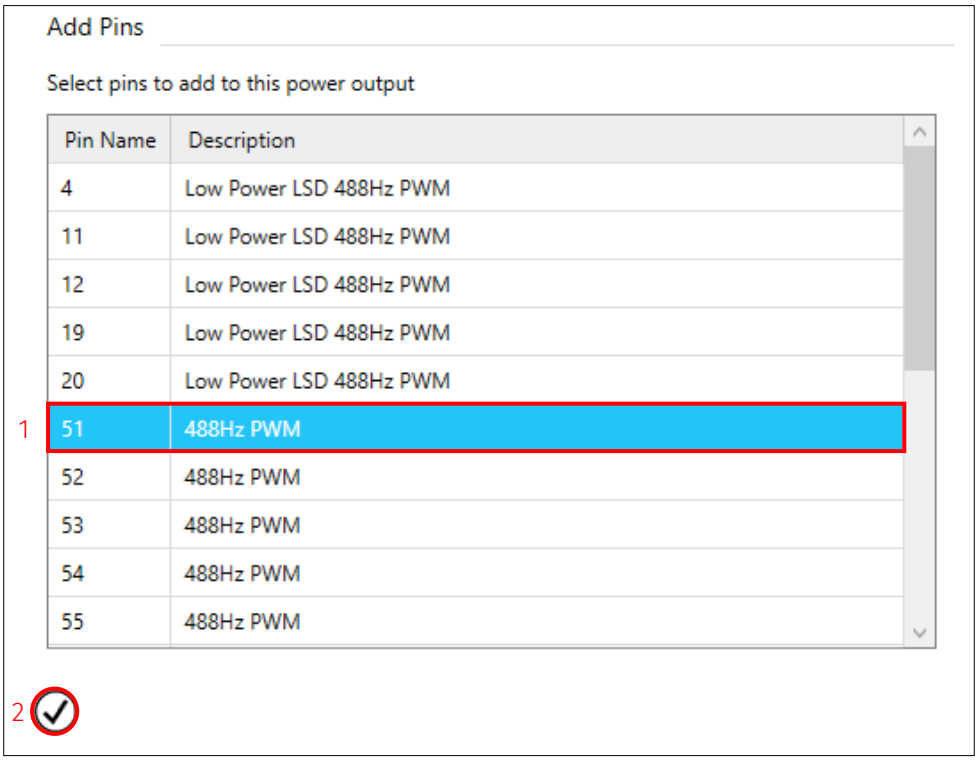

**Configure an output**

To configure an output, select which connector to use (1), and then click the **+** tool to add a new output pin (2). Use the 'bin' tool to delete unwanted pins (3).

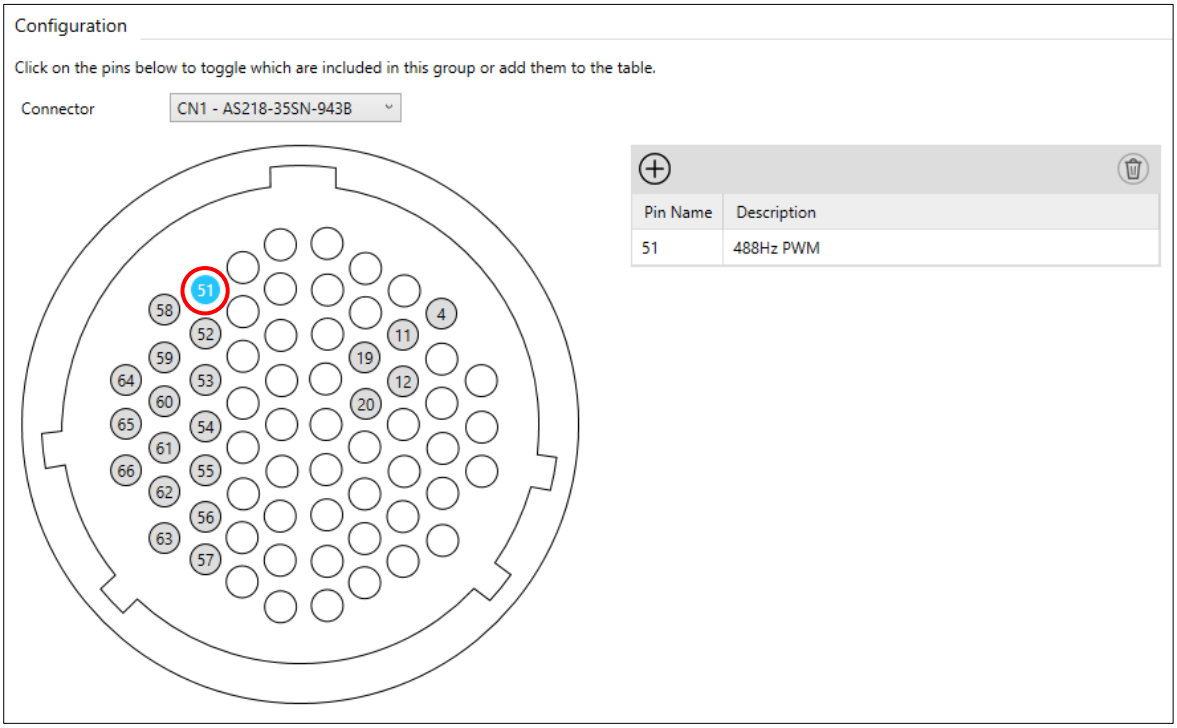
| | |
|---|---|
| Connector 1 CN1 (Low Current) : 1 | 6x 2.5A outputs |
| | 5x LSD outputs |
| Connector 2 CN2 (Medium Current) : | 32x 7.5A outputs |
| Connector 3 CN3 (High Current) : | 5x 12A outputs |
| | 2x 25A 'wiper' outputs |
| | 7x 25A 'standard' outputs |
| | 2x 25A 'high surge' |



You can then select a pin from the **Add Pins** menu. A description about each pin is shown. Click a pin in the menu (1), and then click the 'tick' tool to add (2), or alternatively double-click the pin name.

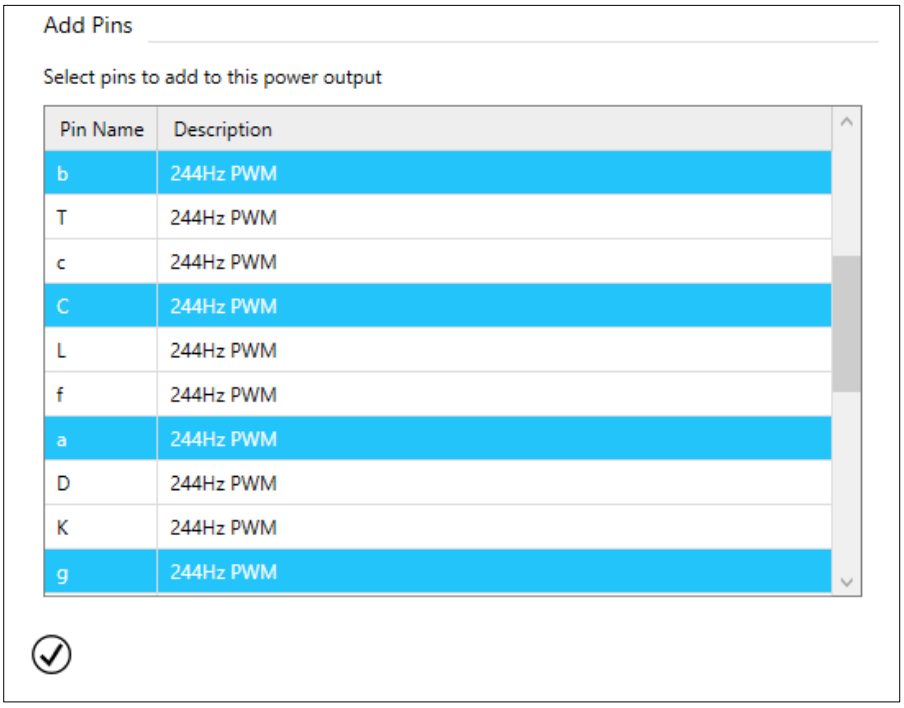



You can also click on pins in the displayed connector to select/deselect them. Selected pins are highlighted in blue.



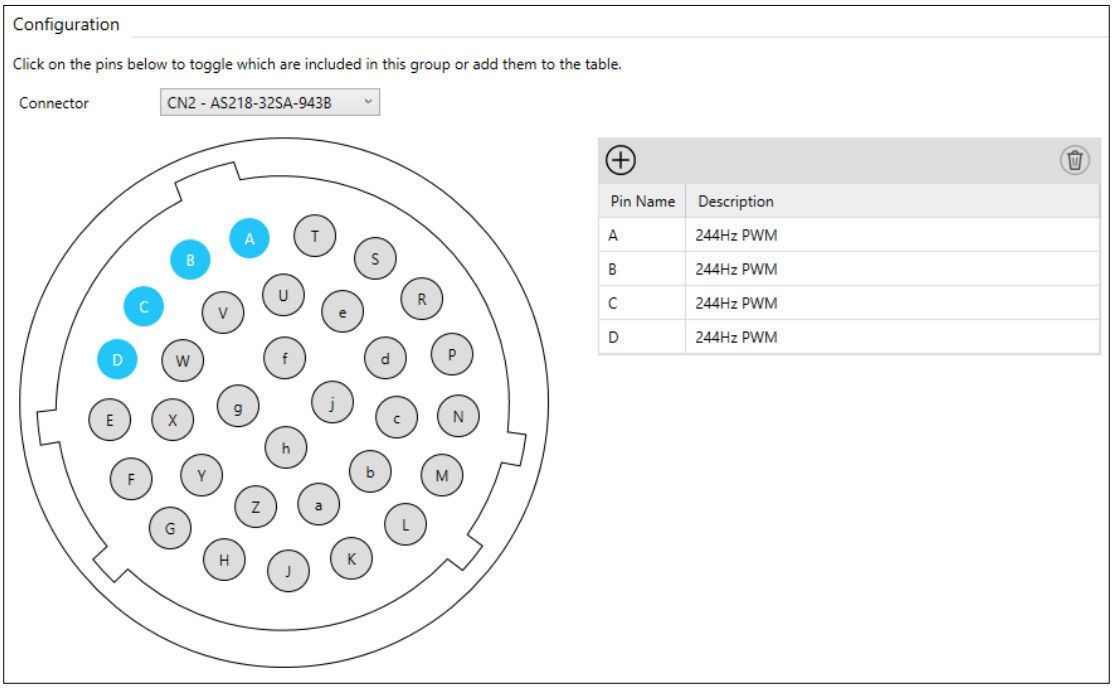
### Grouped pin outputs

You can select multiple pins to form grouped pin outputs. Grouped pins outputs enable increased output current rating by sharing current between pins. In other words, you can group 2x 2.5A pins to make a 1x 5A output and group 4x 7.5A pins to make a 1x 30A output.



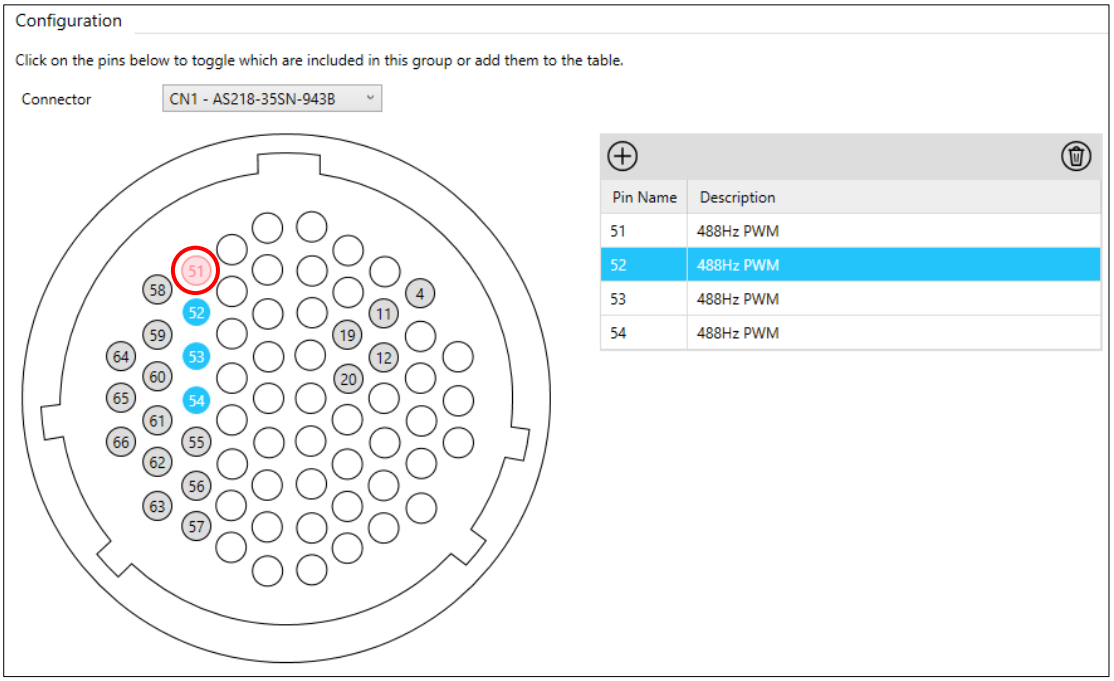

In the **Output Pins** menu, click pins + Shift to select multiple pins.

You can also click on pins in the displayed connector to select/deselect them. Selected pins are highlighted blue.



Note 1: Pins can only be grouped within a connector. Pins cannot be grouped between connectors.

Note 2 – A pin can only be used in one configured output. A pin cannot be used in more than one output. If a pin is used in more than one output, an error is raised, and the pin is highlighted in red.

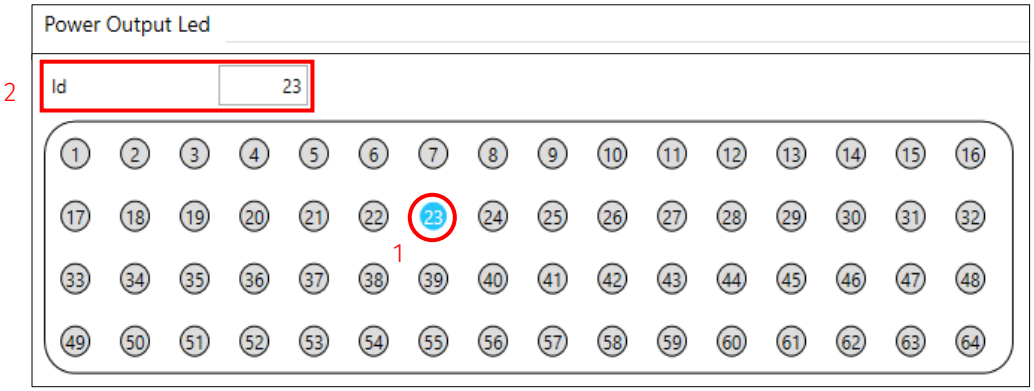



Note 3: Grouped pins must have the same current rating. In other words, a 12A pin and 25A pin cannot be grouped together.

### Power output LED

The Power Output LED section is used to configure which of the LEDs on the display are used to display the output status. When the LED is illuminated, this signals that the output is on. When the LED is not illuminated this signals that the output is off.

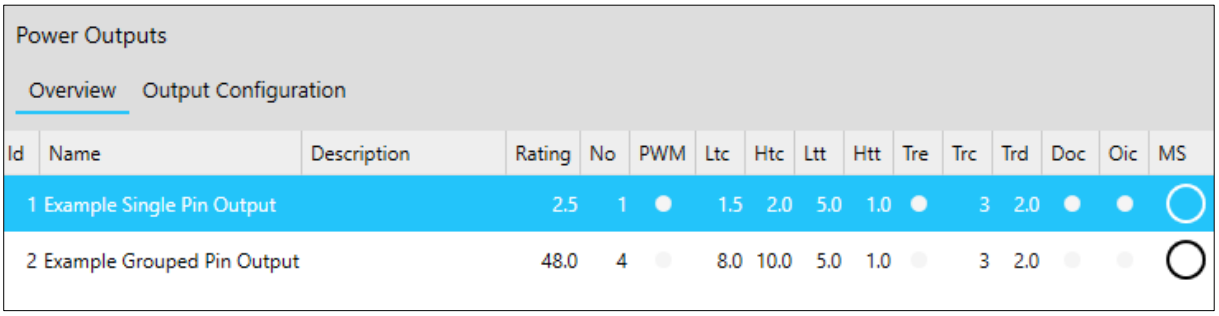
Click on an output LED from the displayed numbered LEDs (1) or type the LED Id number in the Id box to select the LED to display the output status. The selected LED is highlighted in blue.



## 29.3. Output overview

An overview of all the configured outputs is displayed on the **Overview** page.

By default, the **Overview** page is show in 'Grid' mode. This mode displays all the configured properties of the output(s).



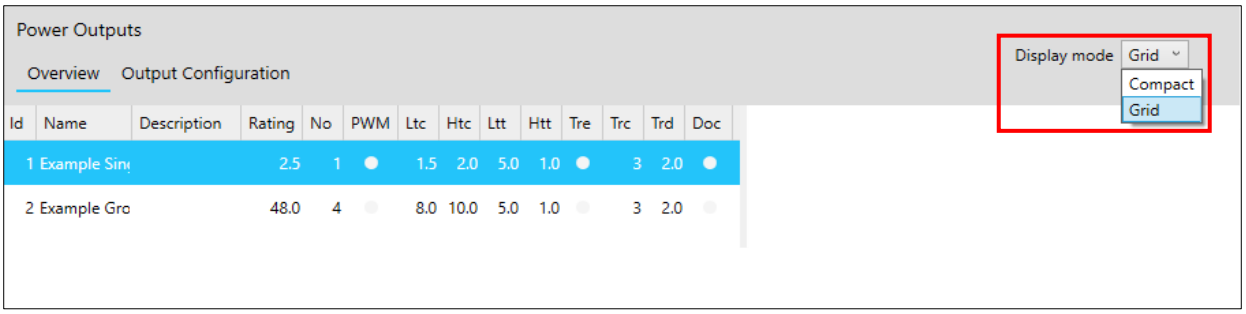

The Grid Overview Page displays:

- Id – Output LED ID number
- Name – Output name
- Description – Output description
- Rating – Current rating (will update for grouped pin outputs)
- No – Number of output pins assigned to this output
- PWM – PWM output enabled (Blue enabled, White disabled) *
- Ltc – Low trip current setting **
- Htc – High trip current setting **
- Ltt – Low trip time setting **
- Htt – High trip time setting **
- Tre – Trip retry enabled (Blue enabled, White disabled) **
- Trc – Trip retry count setting **
- Trd – Trip retry delay time setting **
- Doc – Default output condition (Blue ON, White OFF) *
- Oic – Output is controlled by a control channel (Blue true, White false) *
- MS – Manufacturer status (see Setups – Setup Locking)

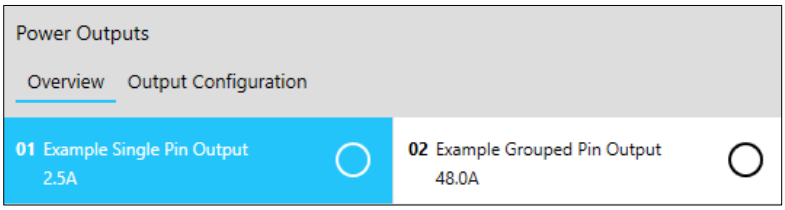
* (see Control Menu section)

** (see Trips Menu section)

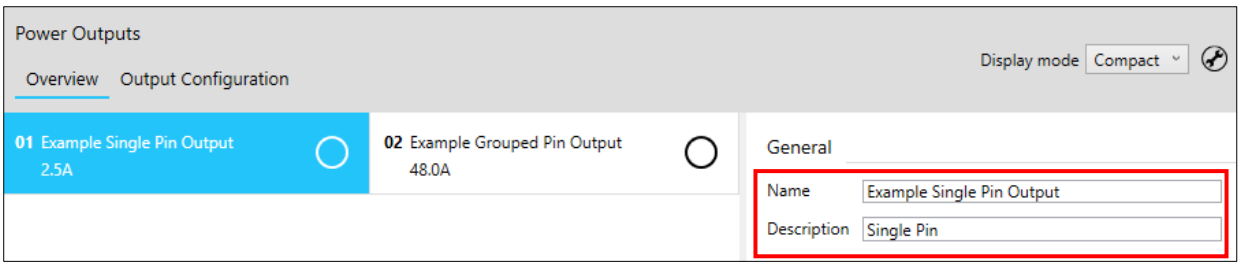
You can change the ‘Display Mode’ to ‘Compact’ to display information in a reduced format from the **Display mode** dropdown menu.




The ‘Compact’ display mode only shows the output name and its current rating.

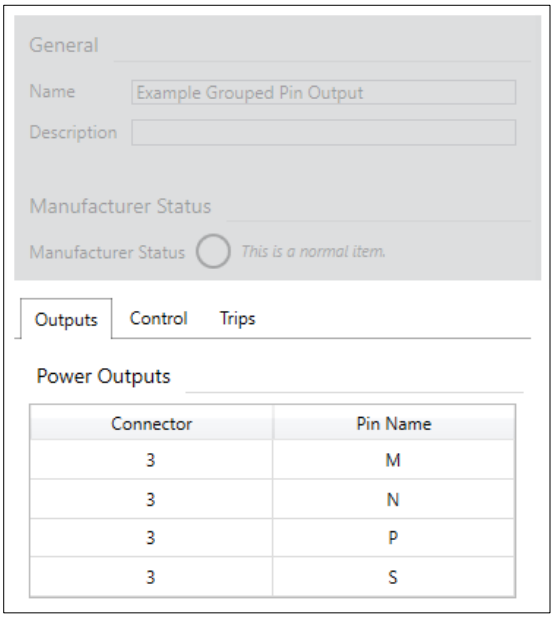



You can change the name and description of the output from the **Overview** page under the **General** section.



### Outputs menu

The **Outputs** menu displays the allocated connector number and pin name(s) for the selected output.



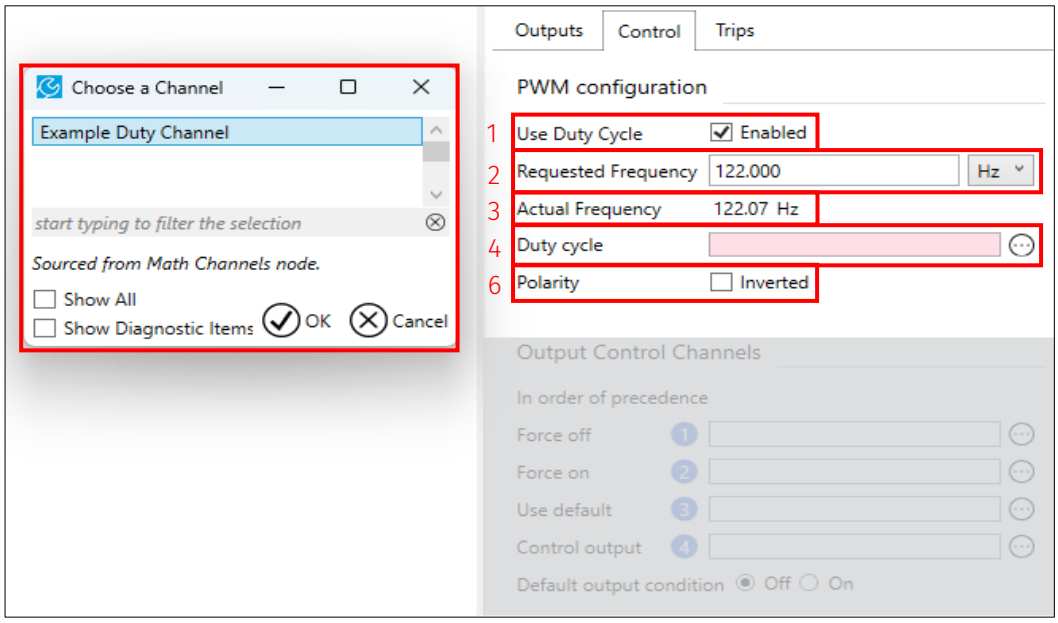

### Control menu

The **Control** menu is where the PWM configuration and output control channels are configured.

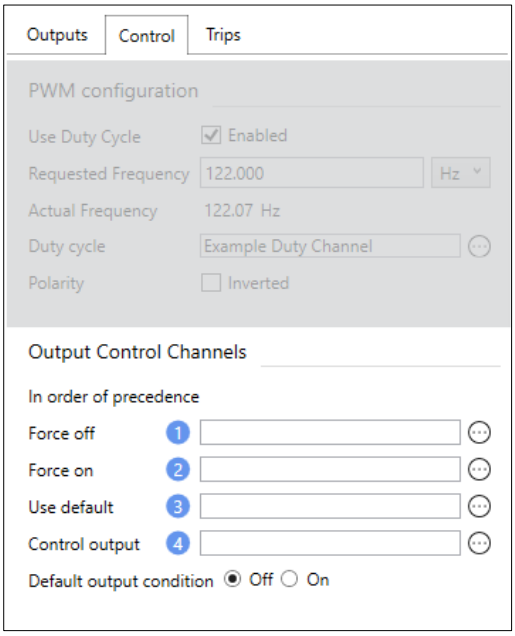
PWM configuration

To enable PWM for an output select the **Use Duty Cycle** check box (1). You can then set the requested PWM frequency (2).

**Note**: The actual frequency is capped at the maximum frequency of the pin(s). Use the 'browse' tool to select a channel with units of proportion to control the PWM duty cycle (3), and then select an available channel from the channels list (4). Click **OK** to confirm your selection (5). If required, select the **Polarity** check box to invert the polarity (6).



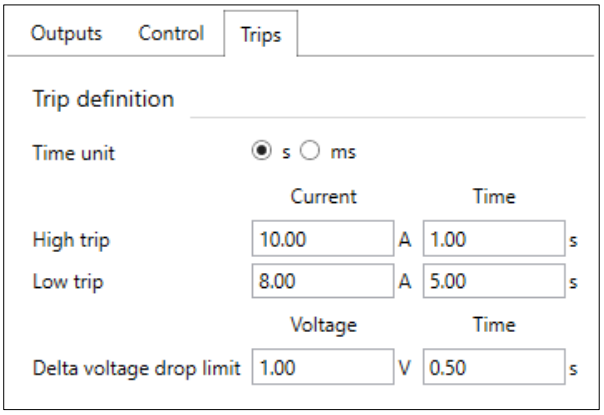
You can set the output to Off or On by default. Select the required **Default Output Condition** check box.




### Trips menu

In the **Trips** menu, you can configure 'High' and 'Low' trip definitions. To set the high and low trip definitions, type the current trip rating in the **Current** box (1) and the t ime duration during which the current must be greater or equal than for the trip to occur (2). Select the **Time units** (seconds or milliseconds) from the check boxes (3).

You can also configure a voltage drop to cause the output to trip.




The hard trip time is fixed by Cosworth but can be changed to suit specific applications. Contact the Electronics Support team (electronics.support@cosworth.com) for more information.

If the output trips, you can enable an optional 'Retry' to restart the output. You can configure up to nine attempted retries, with up to 30 seconds between each. Select a channel to trigger the trip reset from the 'browse' menu

**Note**: The trip reset channel must be a U8 data type.

## 29.4. Generated channels

Seven channels are automatically generated on the **Power Outputs** node and you can edit their names.

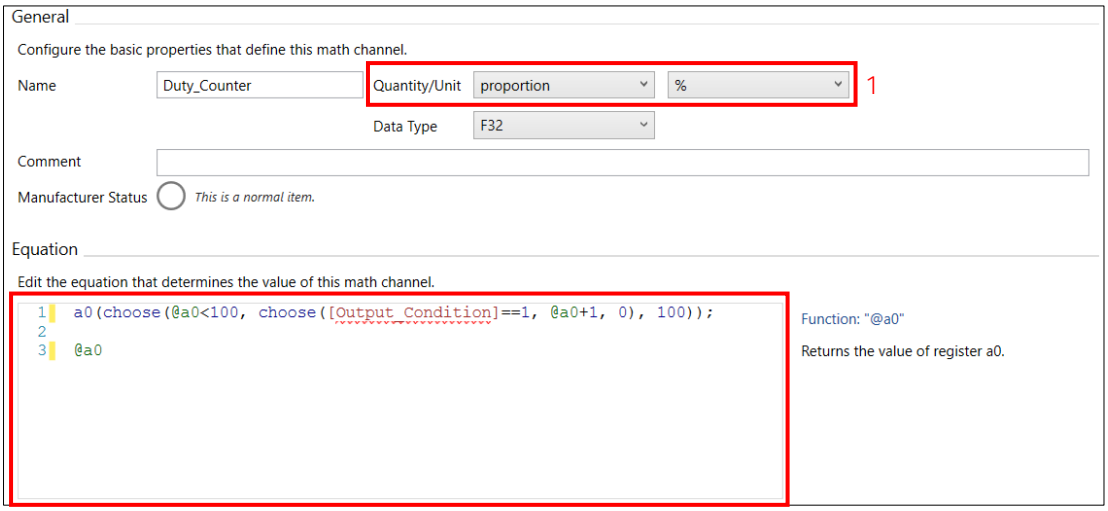
- Min Vbat Monitor:
- Max Vbat Monitor:
- Brownout:
- High Current Temperature 1:
- High Current Temperature 2:
- Low Current Temperature 1:
- Low Current Temperature 1:

These channels are not automatically logged but you can set a logging rate for these channels in the Channel Rates node.

## 29.5. Soft starts

It is possible to create a maths channel or logic channel with conditions that enable a device to slowly ramp up to the required output. This can reduce wear on components like lights and motors and avoid sudden inrush currents.

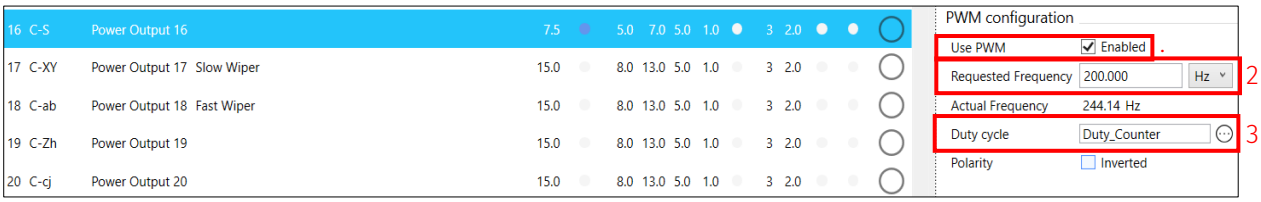
Create a maths channel that has the units of 'proportion' (1), and then create a counter as shown below (2).




This counter is enabled only once the **[Output_Condition]** is true (that is, the switch to activate the output is enabled). This channel needs to be named according to your switch/button that controls the output. Once the condition is met, the counter increments until it reaches 100 (100% duty).

The counter can be set to sped up/slowed down by modifying the evaluating rate of the 'Duty Counter' (for example, 1hz = 100 seconds, 100hz = 1 second, or 200hz = 0.5 seconds).

Select the configured output requiring ‘soft start’ and enable PWM (1). Set the **Requested Frequency** to the required frequency at 100% duty cycle (2). You can then select the new soft start ‘100% Duty Cycle’ channel (3).

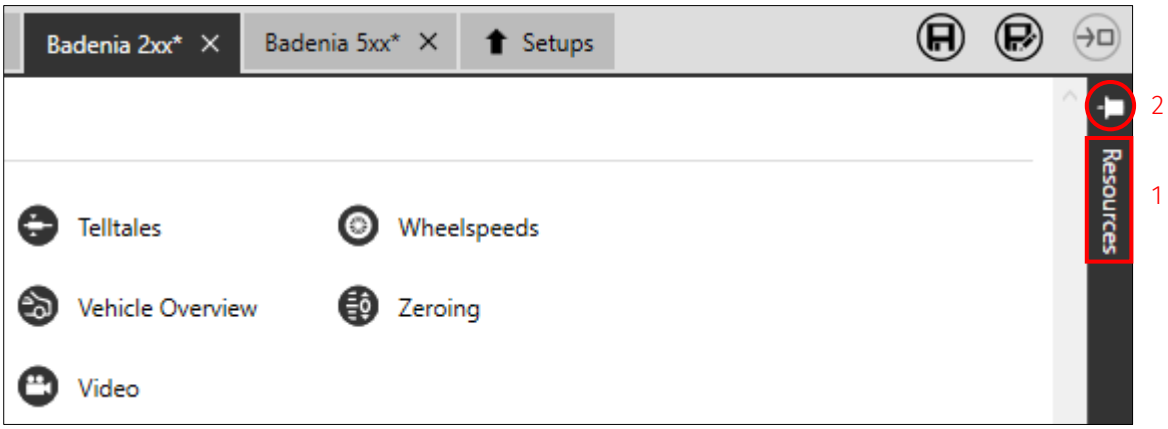


# 30. Resources overview

The Resources menu allows you to quickly navigate setups, allowing:

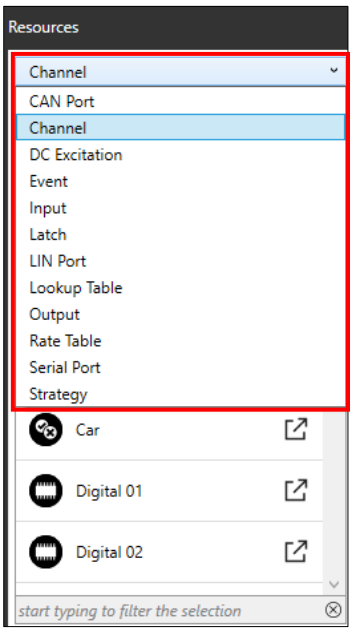
- Navigation to item sources
- Identification of item consumers

The resources menu is found on the right side of the Toolset window and is permanently displayed. Click the Resources menu (1) and use the 'pin' tool to keep the menu open (2).

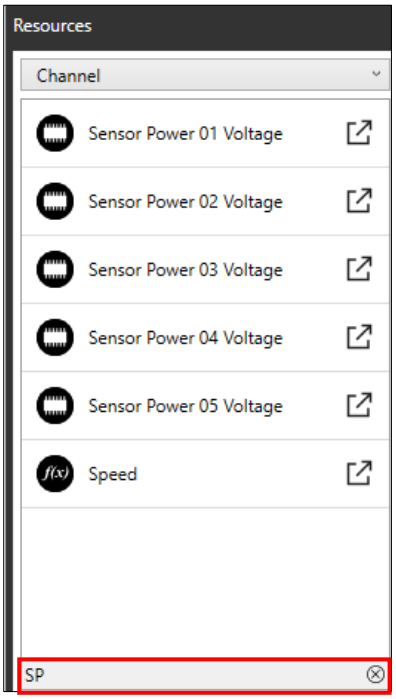



## 30.1. Navigate to item sources

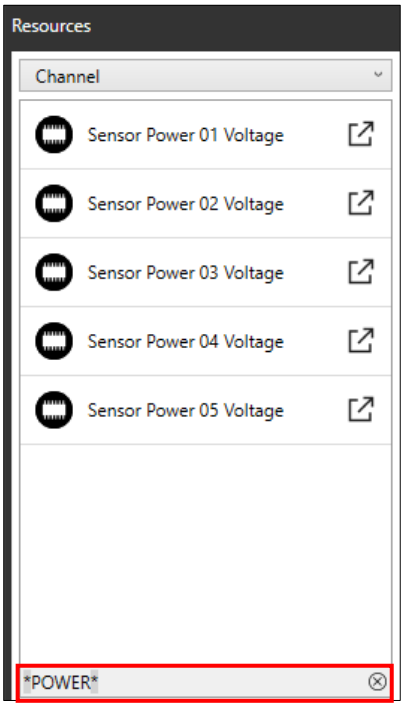
When the Resources menu is open, use the dropdown menu to search by item type.



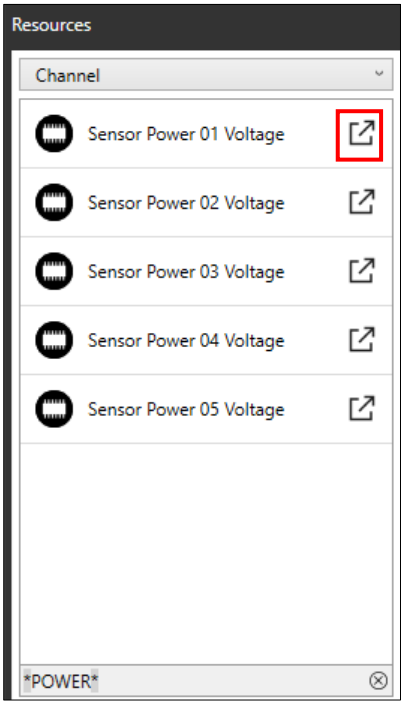
With the item type selected, you can use standard or ‘magic’ search tools to search or filter the item.




To use magic, press CTRL + Shift + 8 simultaneously. A highlighted star indicates that the search is active. Type a specific word or phrase, and then press CTRL + Shift + 8. to close the magic search.

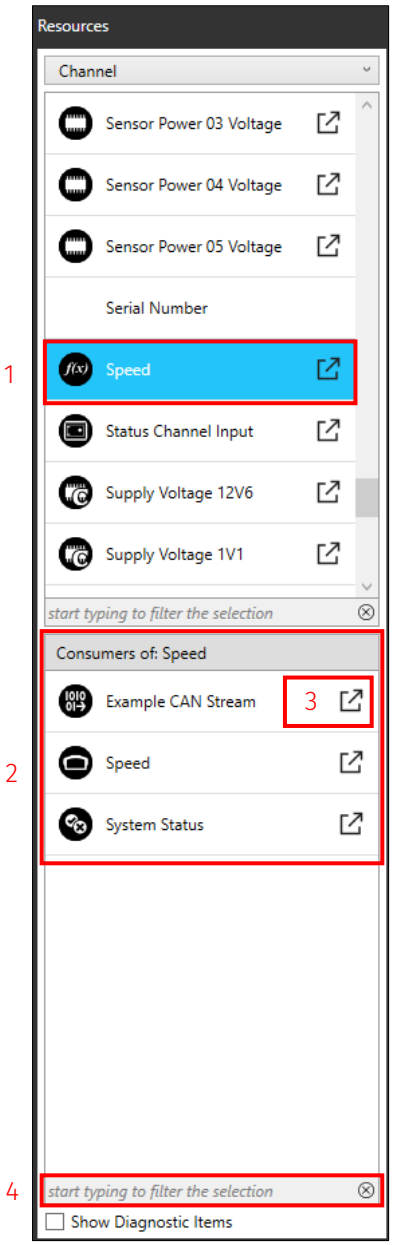


To navigate to the channel source, click the ‘Navigate to resource’ tool.




## 30.2. Identify item consumers

Click on an item to identify the consumers of an item (1). The consumers are displayed at the bottom of the Resources menu (2). Click the ‘Navigate to resource’ tool to go the consumer location in the setup (3). If there are many consumers, you can filter them with the standard search and ‘magic’ search functions (4).

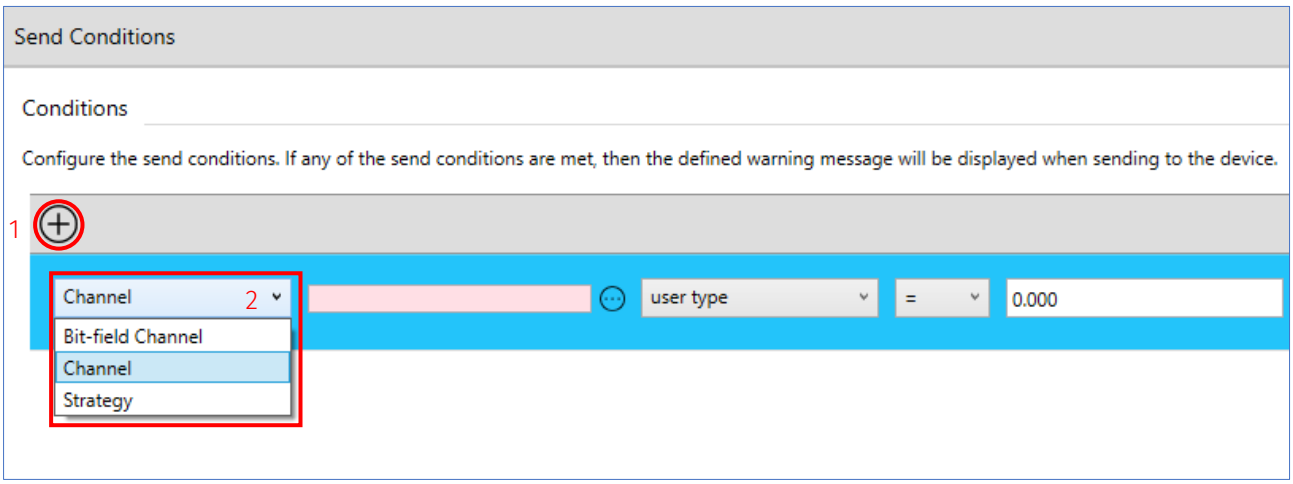


# 31. Send conditions

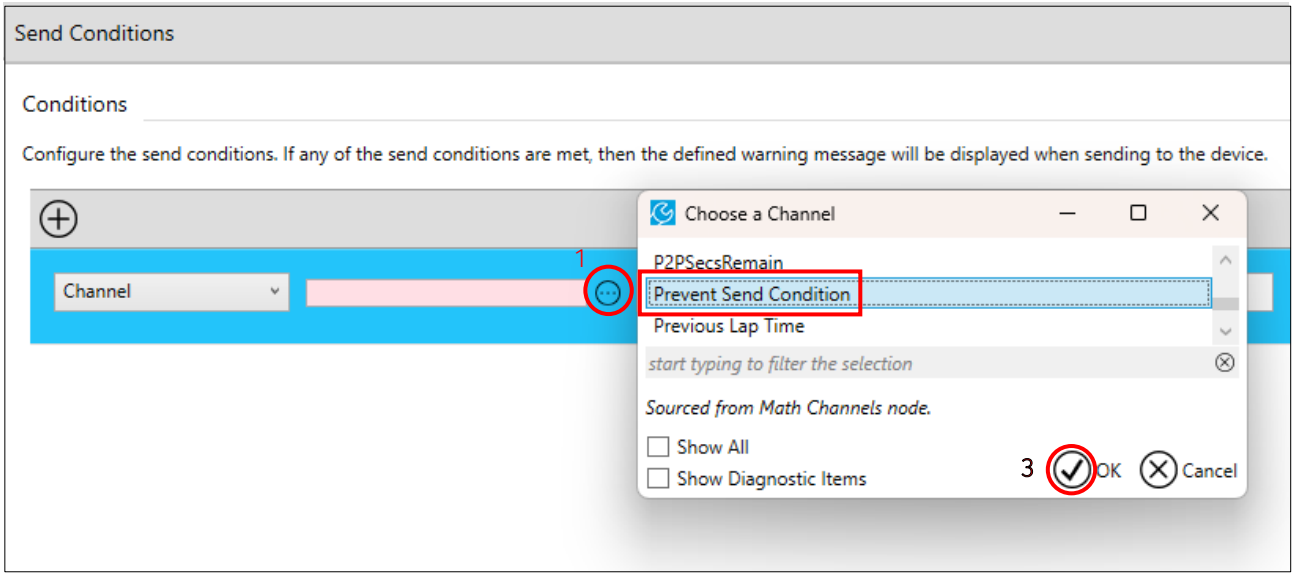
The **Send Conditions** node is used to configure any conditions that trigger a warning if send conditions have not been met before a setup is sent. If the send conditions are not met a pop-up warning is displayed. You can view details of the error and either cancel or proceed with the send.

## 31.1. Configure send conditions

To add a new send condition, click the **+** tool (1), and then select the type of condition (Bit-field Channel, Channel, or Strategy) from the dropdown menu (2). You can add multiple send conditions.

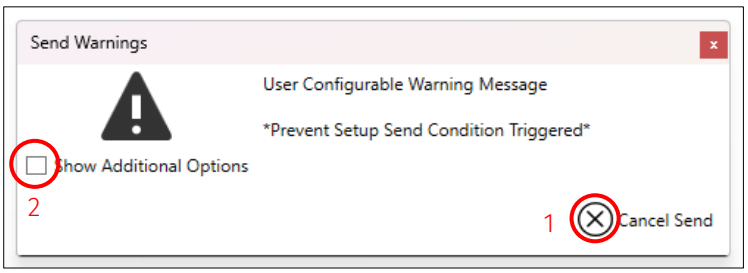



Once you select a condition type, click the 'browse' icon (1), select the channel from the dialog box (2), and click **OK** (3) to select the channel.

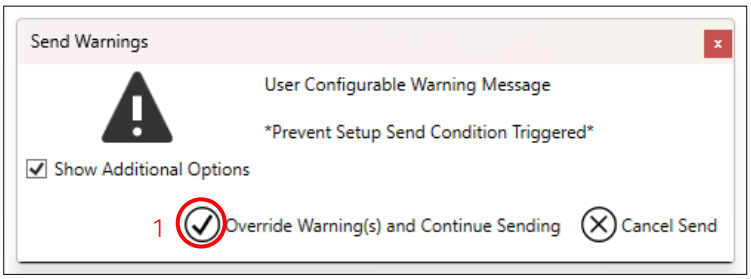


You can configure the condition that triggers a prevent send condition (1) and configure a custom warning message (2). Use the 'bin' tool to delete a send condition (3).


If you attempt to send a setup when one of the send conditions is true, a warning box is displayed with the warning message from the triggered send condition (see above). Make sure that this message suggests a solution so that the error can be corrected before you attempt to send the setup again. You can cancel the setup send (1.) or **Show Additional Options** (2).




If you select **Show Additional Warnings,** you have the option to 'Override Warning(s) and Continue Sending' (1).



# 32. Sensors

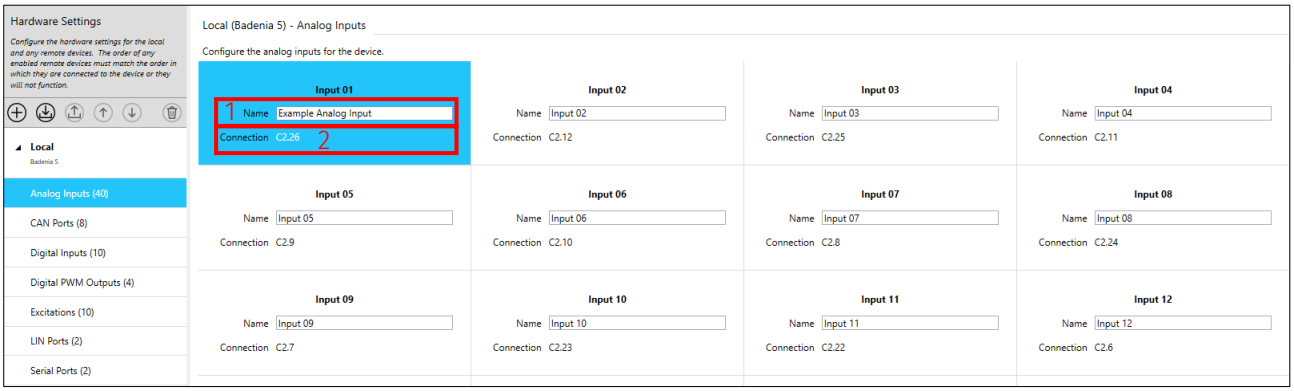
Configuring sensors within Toolset is an essential part of monitoring vital data imported from a car. The following provides an overview of how to configure analogue and digital sensors, and various configuration options.

Note: Refer to Beacons for configuring a digital beacon input.

## 32.1. Analog sensors

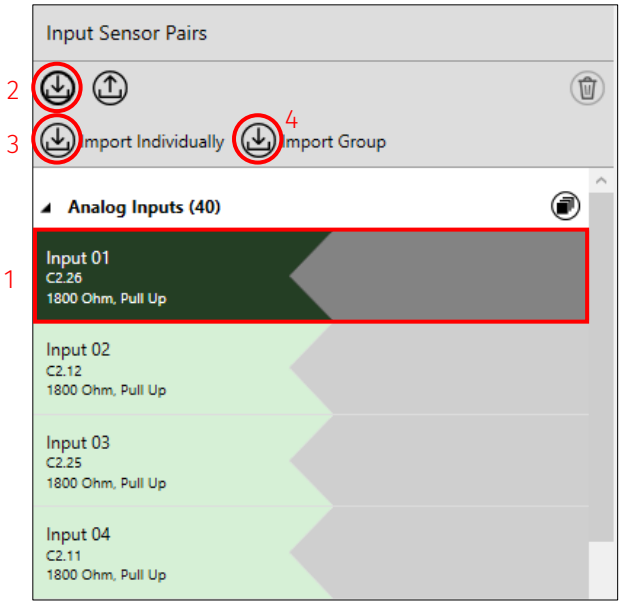
### Identify an analog input

You can configure the name of the analog inputs on the Hardware Settings node (1). Underneath each input label is a connection identifier in the format 'CX.Y', where X and Y are values that identify the connector and the pin of the device, respectively (2).

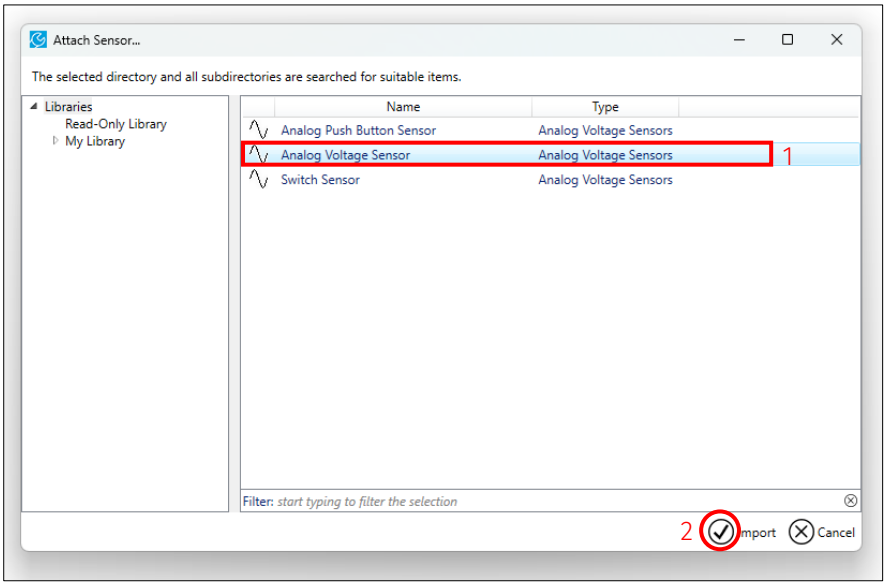



### Add an analog sensor

You can now configure the analog input on the Sensors node. To specify the desired sensor input, select an analog input (1), click the 'import' icon (2), and then click Import Individually (3). You can import multiple sensors as a group (4). You can import and export sensors between existing setups.



When you click the ‘import’ icon a popup is displayed that shows the Toolset sensor library. This allows you to select the required sensors. Select the required sensor (1), and then click **Import** (2).

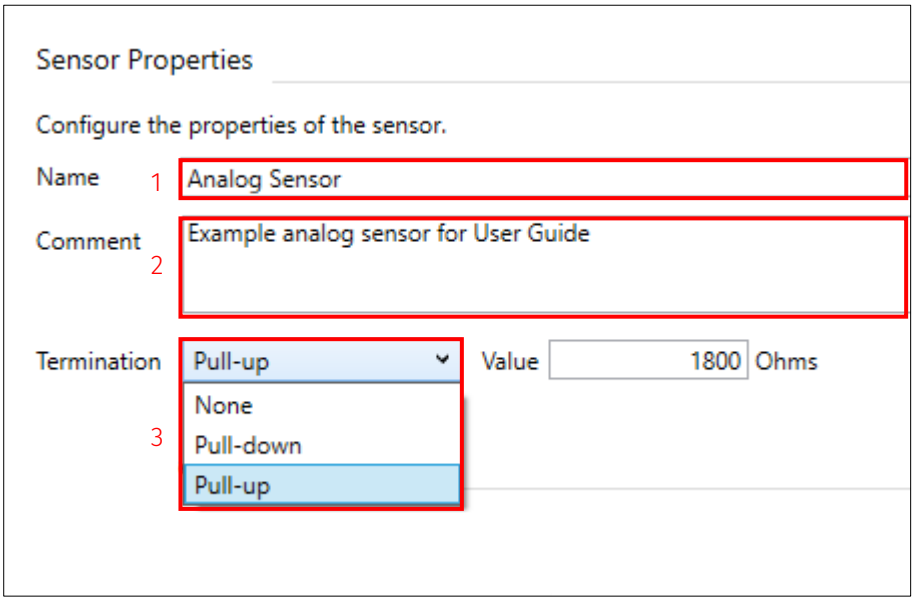



**Configure an analog voltage sensor**

Sensor properties

Once the sensor is imported, a new window is displayed to the right of the screen to allow you to configure the sensor.

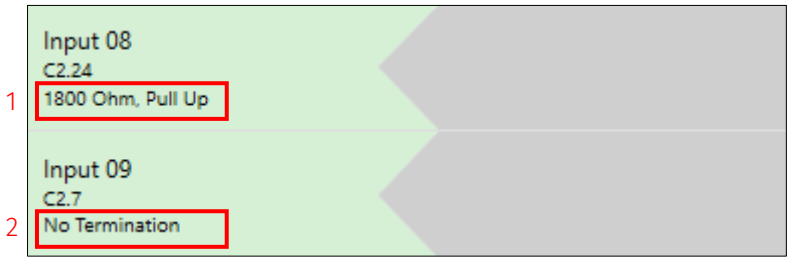
Enter a name for the sensor (1), add an optional comment about the sensor (2), and specify the termination type if required (3).




Terminations are available on specific inputs. You can see which inputs feature software selectable termination in the **Input Sensor Pairs** menu. The available termination for the input is denoted under the connector/pin identifier (1).

If no termination is available, ‘No Termination’ is displayed (2). If a ‘Pull Up’ termination is selected, the resistor must be specified to match the value in the Sensor Pairs menu.

The Pull Up termination is not available on all inputs. The Pull Down termination is not available on all analog inputs.

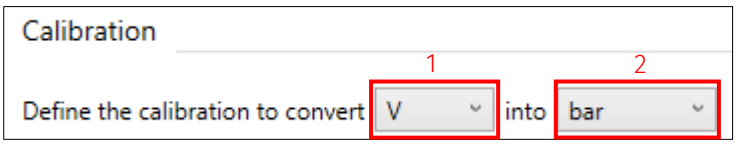



Calibrated channel

Enter the calibrated channel name for the sensor (1), the units for the output quantity (2), and specify the data type (3). You can set an optional uncalibrated channel name to generate a raw uncalibrated voltage channel (4).

Sensor calibration

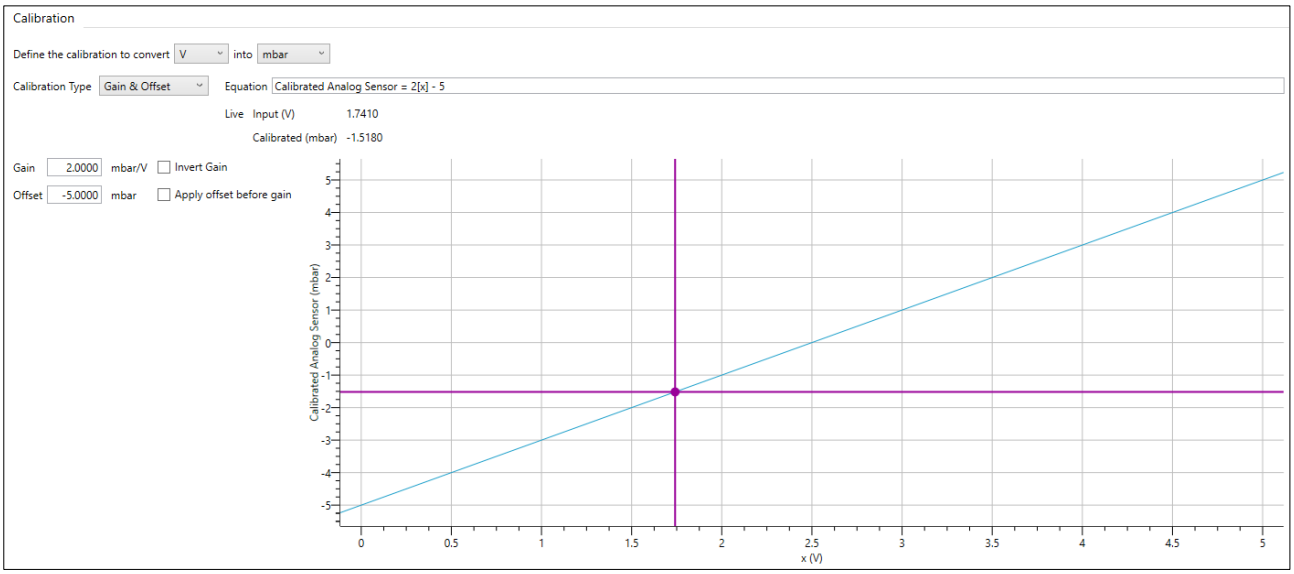
You can now calibrate the analog sensor input. You can define the conversion between voltage units (µV, mV, kV) (1) and the units of the selected output quantity (in this case the units of the selected quantity (pressure) are in bar) (2).




There are three calibration types available to select from the **Calibration Type** dropdown menu:

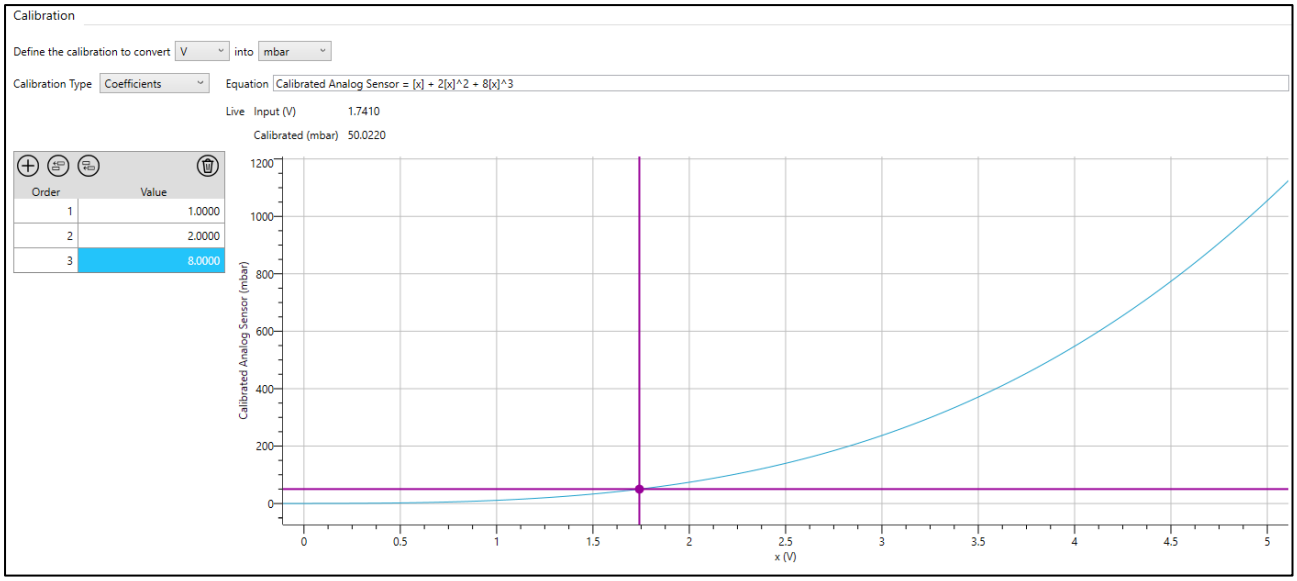
Gain & Offset

Used for proportional calibrations. You can apply a gain and offset to the raw analog voltage input to generate the calibrated output channel. You can insert the required gain and offset, invert the gain and the offset applied before the gain if required. The sensor curve is displayed on the X/Y chart and the equation of the line is displayed in the equation box. When connected to the device, the current sensor readout is displayed on the X/Y chart.



Coefficients

Used for exponential sensor curves. You can add and remove the number of orders from the equation with the = and 'bin' icons. Use the 'insert entry' tools to add orders before or after entries in the table. The sensor curve is displayed on the X/Y chart and the equation of the line is displayed in the equation box. When connected to the device, the current sensor readout is displayed on the X/Y chart.

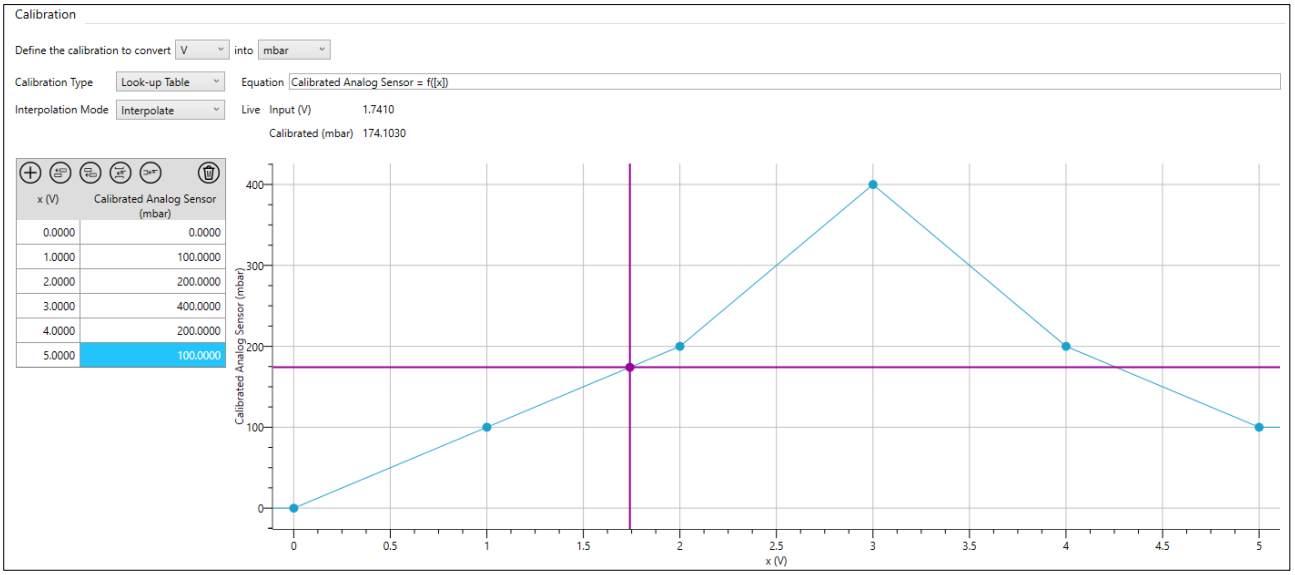


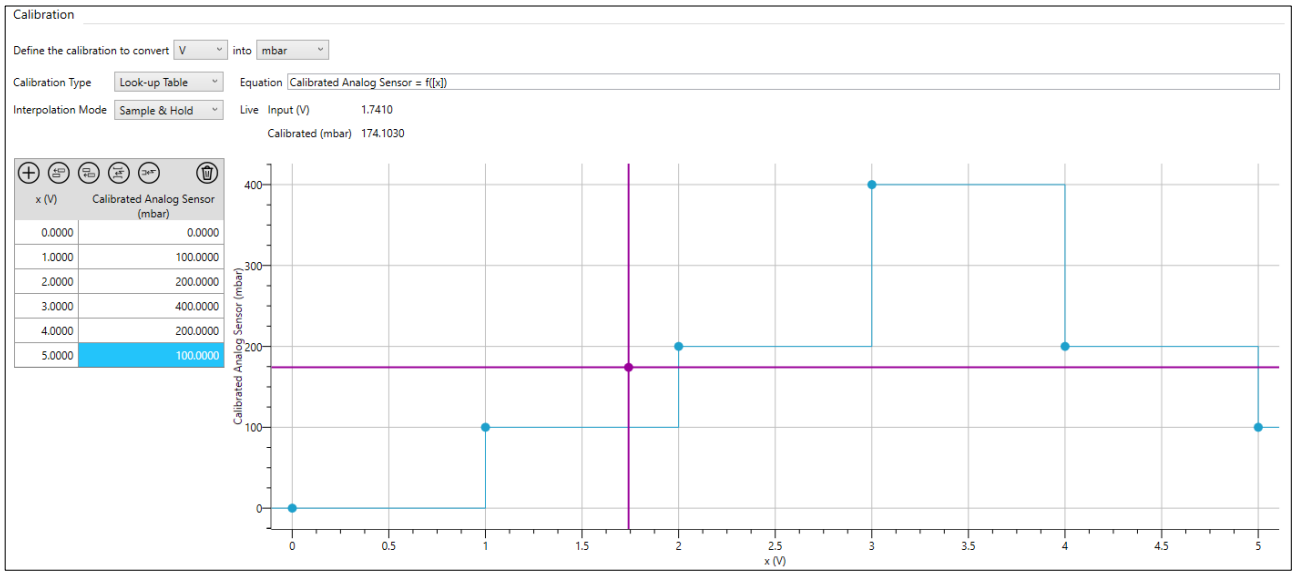

Look-up Table

Used to configure a sensor when the sensor curve is known from a datasheet or look-up table. In the look-up table you can associate certain values of the input channel to a corresponding output value. Toolset then offers three modes to interpolate input/output points in the look-up table:

- Extrapolate – Toolset estimates output values given an input value outside the defined range.
- Interpolate – Toolset calculates the output values between defined input value and corresponding output values.
- Sample & Hold – Toolset holds the output value of its corresponding input until the next defined input value is reached and the output is then updated

By default, the interpolation mode is set to Interpolate.



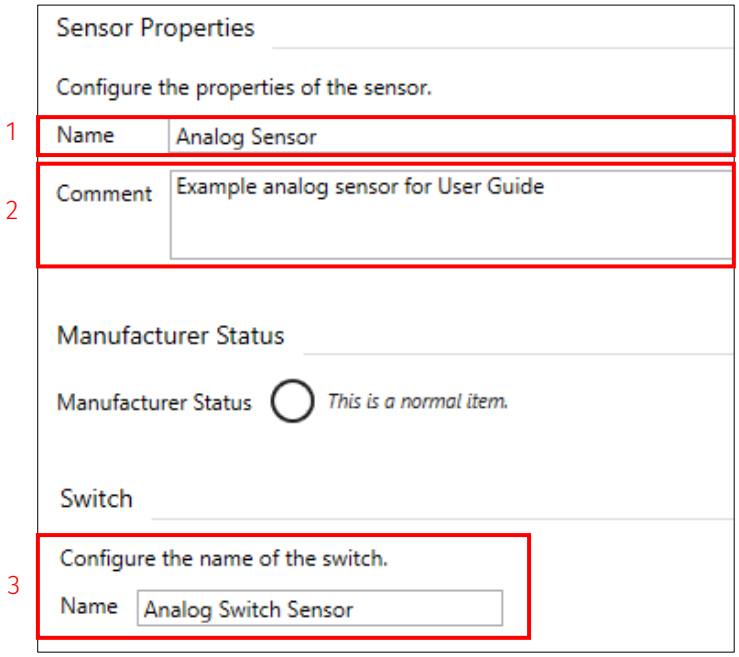

**Note**: The **Equation** display is a display of the mathematical function of the sensor curve only. The equation of the sensor curve cannot be written in the field. If you need to copy in a known equation, transfer that equation into a coefficient or table form, and then copy those values into Toolset. You can copy tables from external sources and then copy them into Toolset. Once the table is copied, the corresponding equation will match.

### Configure an analog push button sensor

The push button sensor provides a way to use the analog input as a push button/switch input. The AIN input has a maximum input range of 0-30V 'to battery'. However, if you select 'switch to ground' an external pull up resistor is required to generate the switch to the input logic.

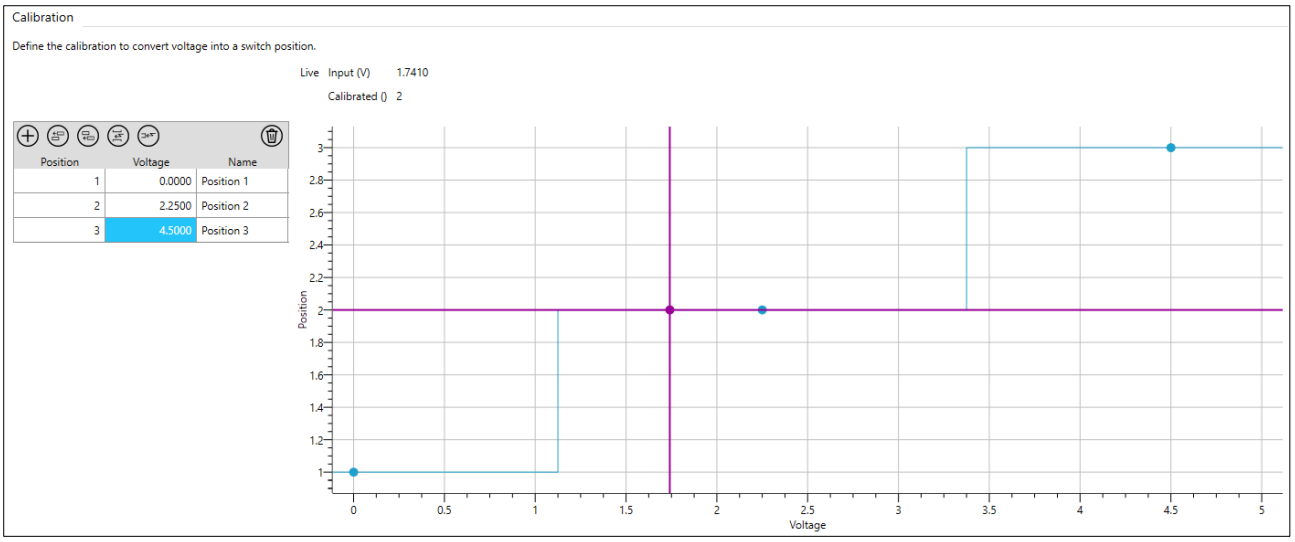

### Configure an analog switch sensor

Add a name for the switch sensor (1) and an optional comment about the sensor (2) in the **Details** section. The actual switch sensor name is configurable in the **Switch** section (3).




This sensor type enables a user-defined number of switches at various voltages. To modify these values, simply add additional rows to the table or change the existing values. The graph changes based on the number of positions and voltage set.

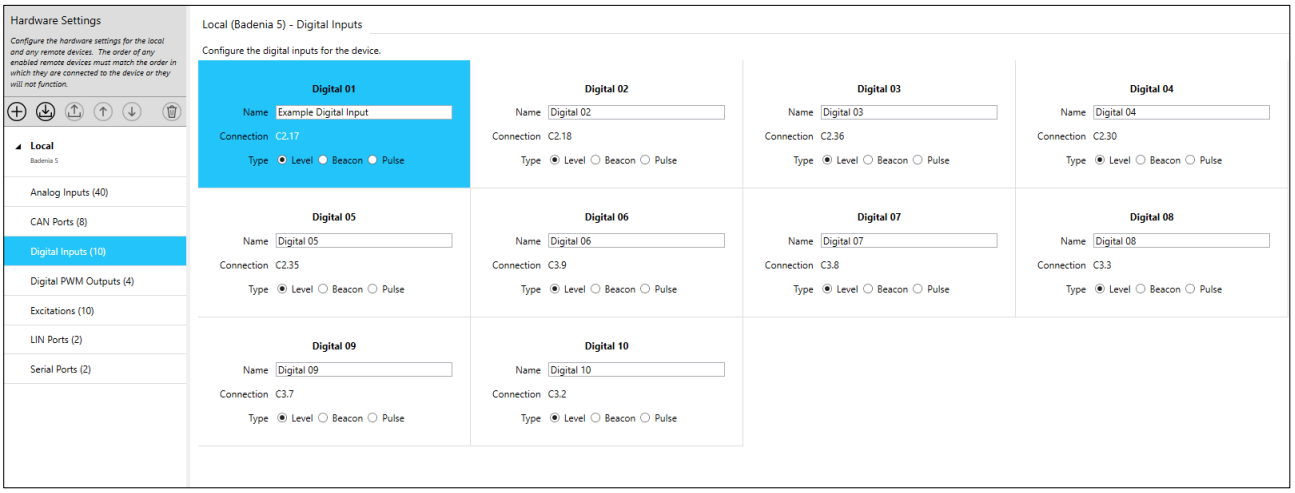
The order of the positions must be respective to the voltages, such that the voltages increase from 0V to 5V. If this condition is not satisfied, then Toolset displays an error and highlights the boxes in red.




## 32.2. Digital sensors

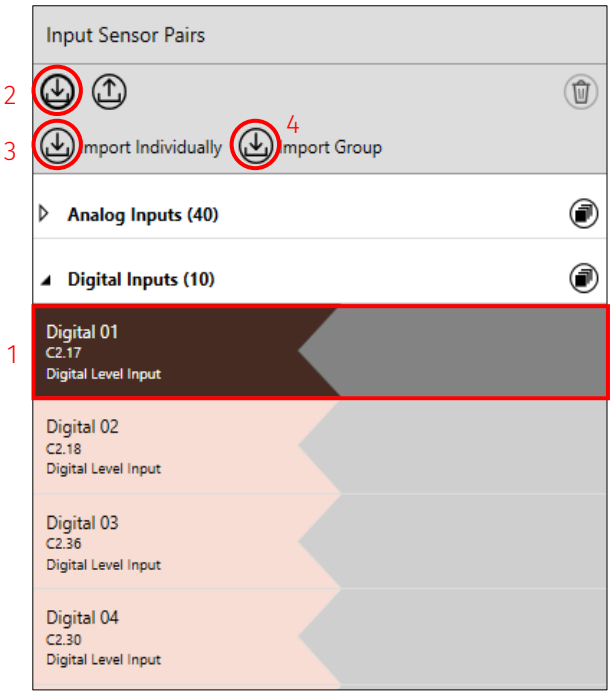
### Identify a digital input

You can configure the names of digital inputs on the **Hardware Settings** node, depending on the sensor configuration required.

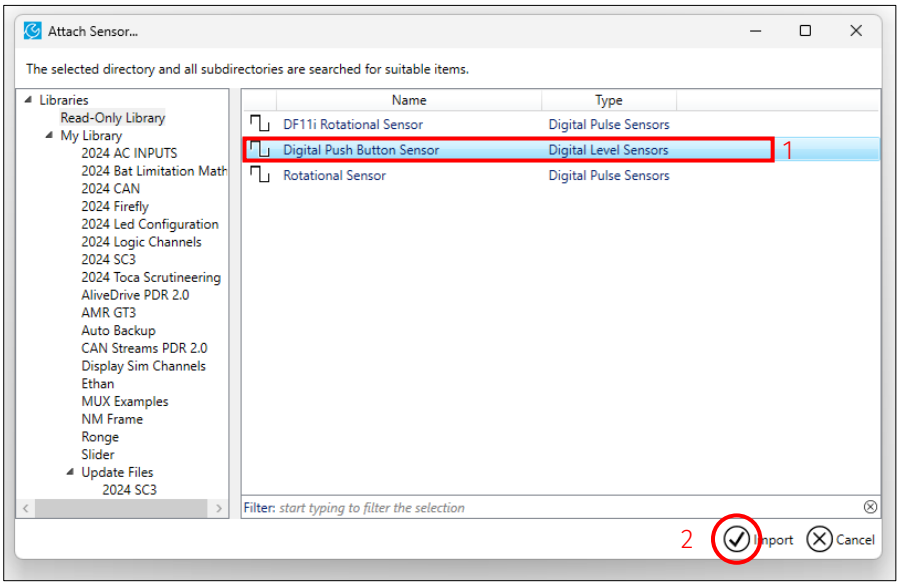



### Add a digital sensor

You can then configure the digital input on the **Sensors** node. To specify the desired sensor input, select a digital input (1), click the 'Import' icon (2), and then click **Import Individually** (3). You can import multiple sensors as a group (4) and import and export sensors between existing setups.

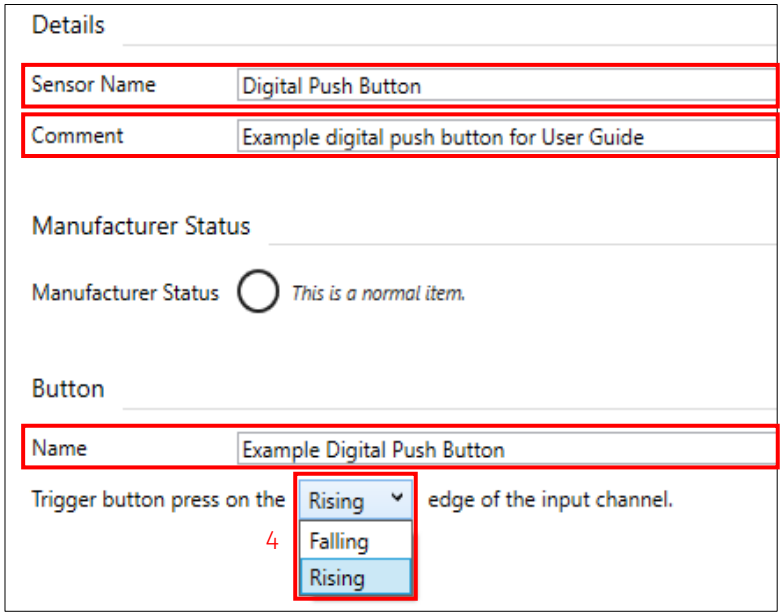



When you click the 'import' icon a popup is displayed that shows the Toolset sensor library. This allows you to select the required sensors. Select the required sensor (1), and then click **Import** (2).



### Configure a digital push button sensor

Add a name for the sensor (1) and an optional comment about the digital button (2) in the **Details** section. The actual button channel name is configurable (3), together with the 'mode' (4) in the **Button** section. The 'mode' allows you to configure if the button triggers on the rising or falling edge of the digital input channel.

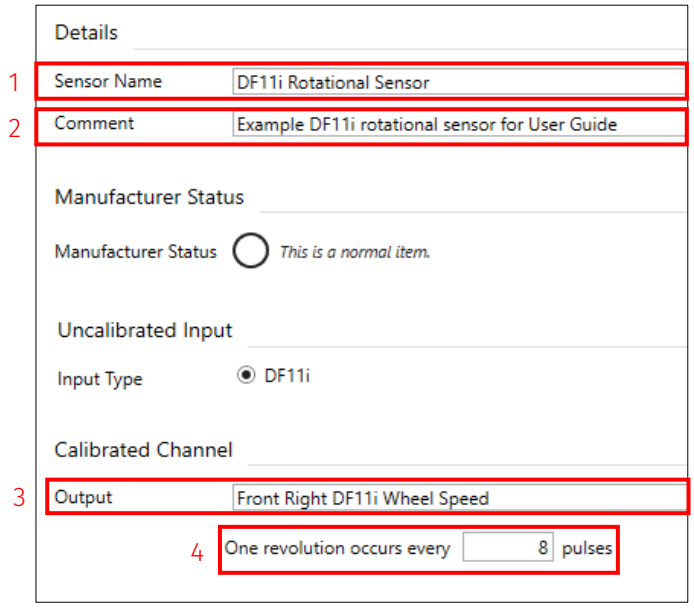



### Configure a DF11i rotational sensor

If a DF11i wheel speed sensor is attached to a digital input, you can configure the number of pulses that occur in a revolution of the sensor trigger wheel.

**Note**: For a 48 tooth reluctor wheel, the total number of pulses per revolution is 96.

You can enter a name for the sensor (1) and add an optional comment (2) about it in the **Details** section. You can configure the **Calibrated Channel** name (3) and set the number of pulses per revolution (4).



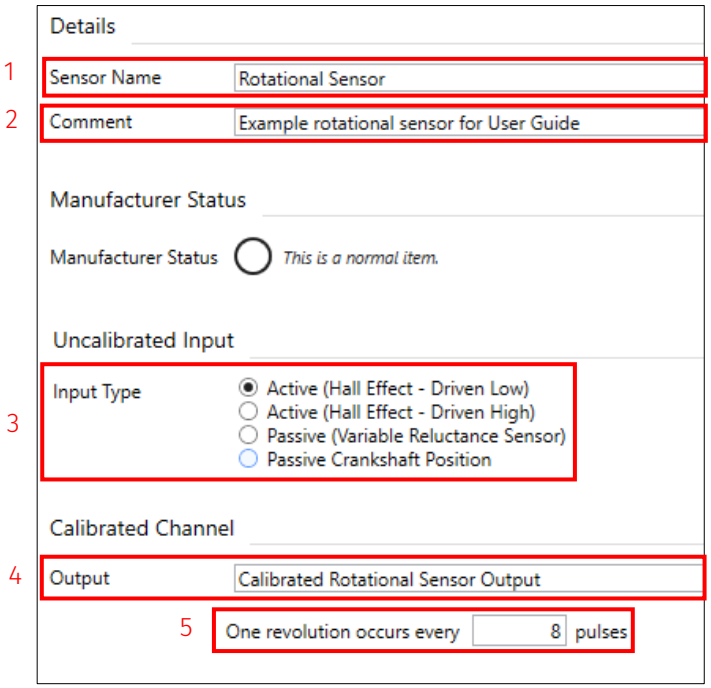
### Configure a rotational sensor

If another type of wheel speed or rotational sensor is used, then you must import the rotational sensor. There are four different options:

| Sensor Type | Specification | Description |
|---|---|---|
| Active | Hall effect – Driven Low | Low would assert an output of 1 when the input signal is Low |
| Active | Hall effect – Driven High | High would assert an output of 1 when the input signal is High |
| Passive | Variable Reluctance Sensor | Uses a two-wire sensor and magnetic pickup |
| Passive | Crankshaft Position | Uses a two-wire sensor and magnetic pickup |

You can enter a name for the sensor (1) and add an optional comment (2) about it in the **Details** section. Select the uncalibrated input type (3). You can configure the calibrated output channel name (4) and set the number of pulses per revolution (5). This is determined by the tooth count of the trigger wheel used to drive the digital input.

**Note:** If you create an input for a wheel speed sensor, the sensor is not defined internally as a wheel speed until it is set up within the **Wheel Speed** node. Until this is done, Toolset only recognizes it as a rotational sensor.



**Digital input information**

Maximum input voltage

The table below shows the maximum input voltage for the different sensor types:

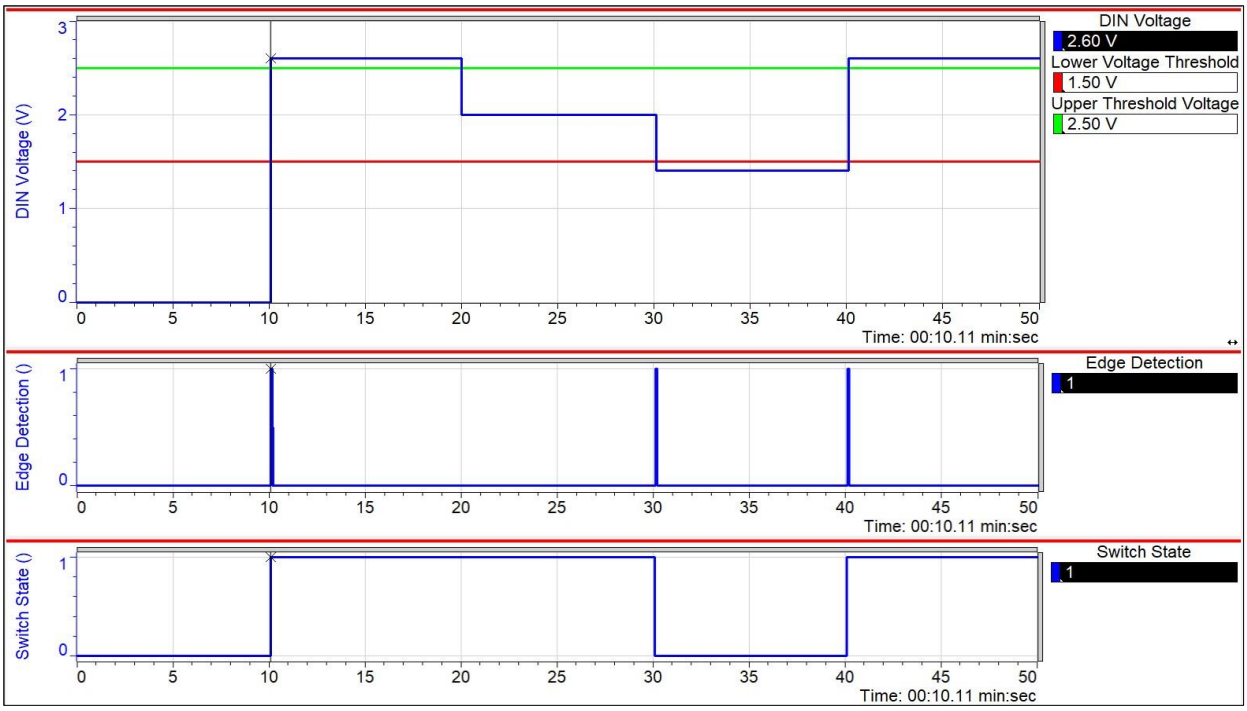
| Input type | Maximum input voltage |
|---|---|
| Digital Push Button Sensor | Battery voltage (up to 32V) |
| Rotational Sensor | Battery voltage (up to 32V) |
| DF11i Rotational Sensor | Only to be used with a DF11i Sensor. Any over-voltage could cause damage to the pull down resistor. |

Threshold voltages

The table below shows the threshold voltages for the signal edge detection. Both the upper and lower thresholds must be passed for an edge to be detected. The input remains registered until both the upper and lower threshold have been passed back through again.

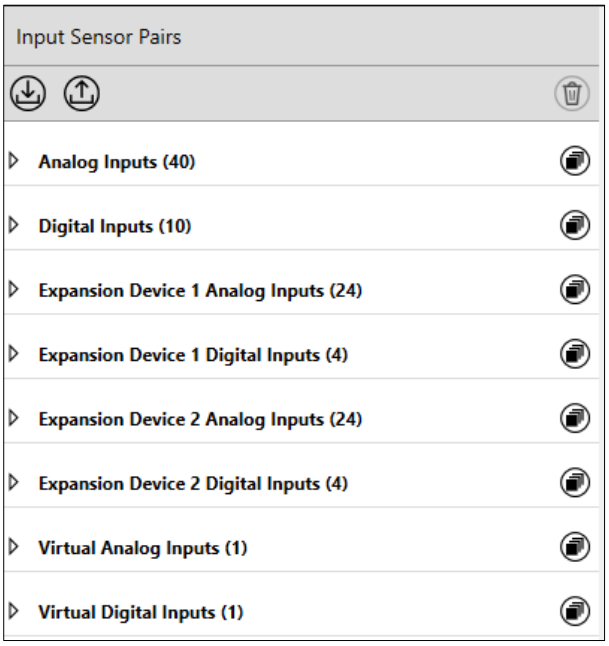
| Mode | Pull | Lower voltage threshold (V) | Upper voltage threshold (V) |
|---|---|---|---|
| OFF | None | 0 | 0 |
| Beacon/Level/Hall Effect | Pull Up | 1.5 | 2.5 |
| VRS | None | -0.5 | 0.5 |
| Current (DF11i) | Pull Down | 2.25 | 3.25 |

The diagram below shows the digital input edge detection logic. Both the lower and upper thresholds must be passed, in either direction, for an edge to be detected by the device.

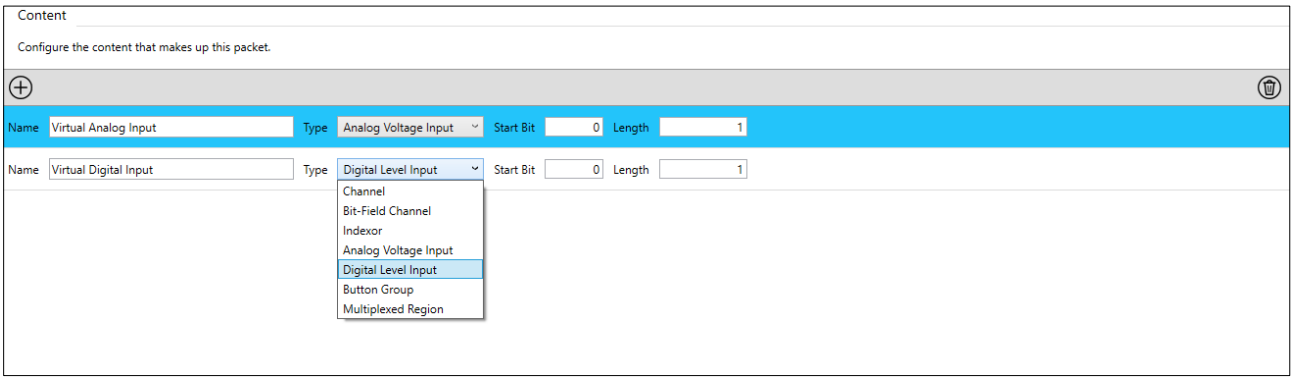



## 32.3. Virtual sensors

You can configure CAN channels (see **Setups - Streams**) to be an analog voltage input or a digital level input from the **Type** dropdown menu.



When you select an analog voltage input or a digital level input type, a virtual sensor is generated on the **Sensors** node.




A virtual sensor is configured in the same way as a standard analog or digital sensor.

# 33. Setup locking

When a device is supplied as a series specified component, the championship organiser or a manufacturer to lock setup items and channels for a team, customer, or other user groups. The **Setup Locking** node is used to configure what is hidden from the end user.

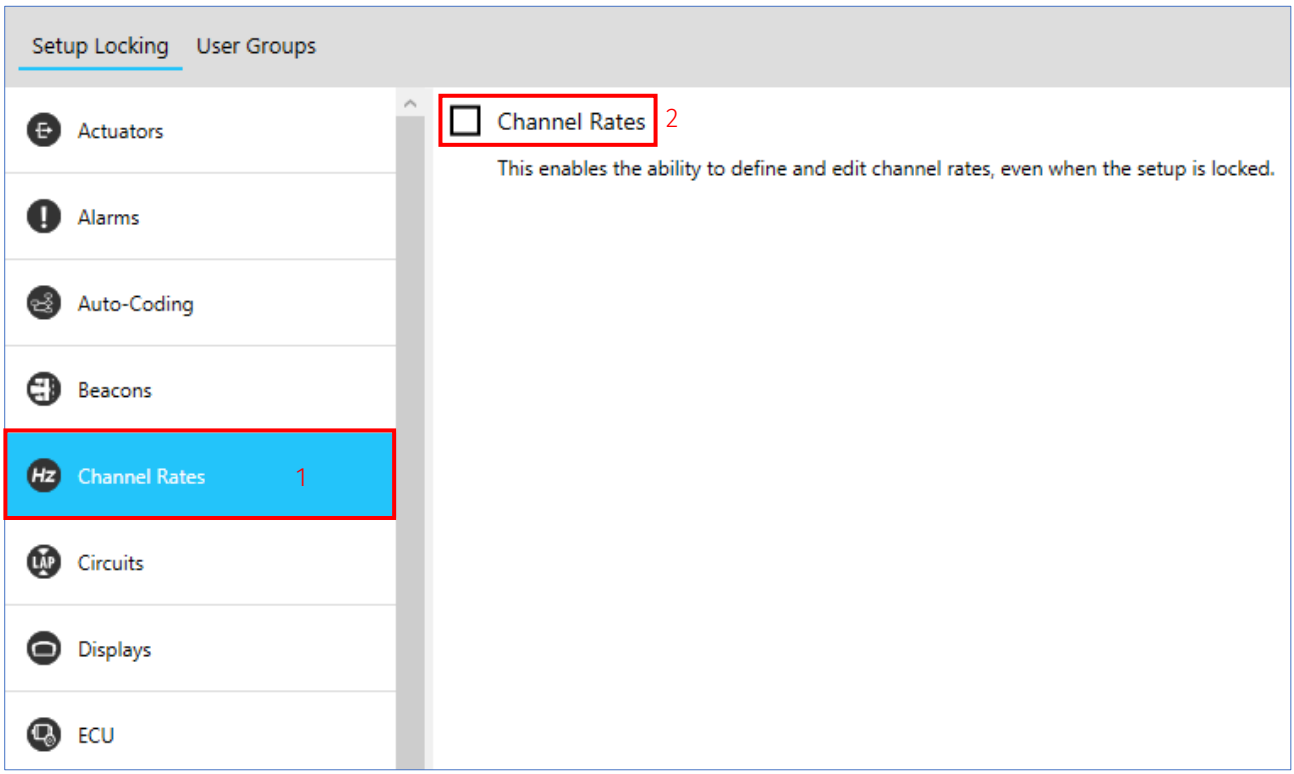
The **Setup Locking** node is made available with a manufacturer 'key' – a software license or a physical dongle. Contact sales@cosworth.com to request a quote.

The **Setup Locking** node contains two menus: **Setup Locking** and **User Groups**.

## 33.1. Setup Locking menu

The **Setup Locking** menu allows a manufacturer to select which nodes are available to the end user when the manufacturer key is not present. Unselected nodes are hidden from users without the manufacturers key and cannot be edited.

A manufacturer can view all the available nodes in the **Setup Locking** menu. Click on the node in the menu to unlock the node (1), and then select the check box (2).



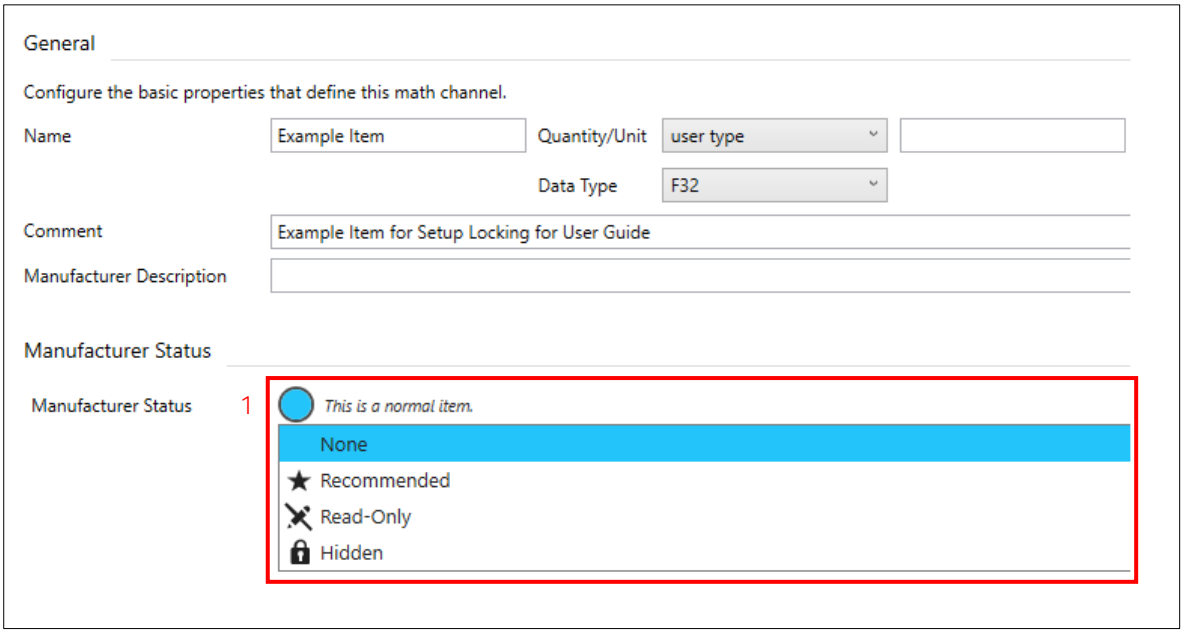
There are two distinct parts to setup locking: 'Manufacturer Status' and 'Channel Protection'.

### Manufacturer status

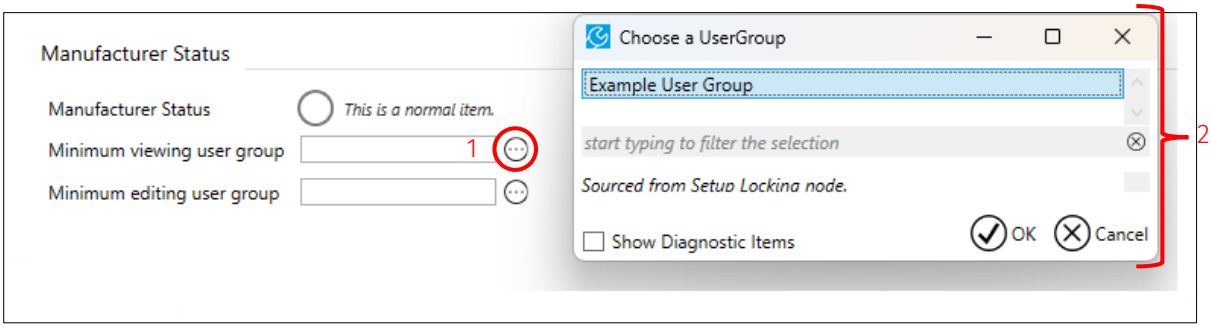
In some instances where a node contains items that a customer might need to edit, such as Maths channels, you can leave specific manufacturer items unmarked ('**None**'), marked as '**Recommended**', made '**Read-only**', or totally '**Hidden**' from the customer.

- **None** – The item is unmarked and is completely unprotected from an unlicensed user in the setup.
- **Recommended** – The manufacturer has recommended that the item is left in the setup, but the item is completely unprotected from an unlicensed user.
- **Read-Only** – The manufacturer has protected the item from one or more user groups so that it cannot be edited, but you can still view the item in the setup.
- **Hidden** – The manufacturer can completely hide the item from one or more user groups so that the item cannot be seen in the setup.

Click in the circle to select the **Manufacturer Status** from the dropdown menu (1)



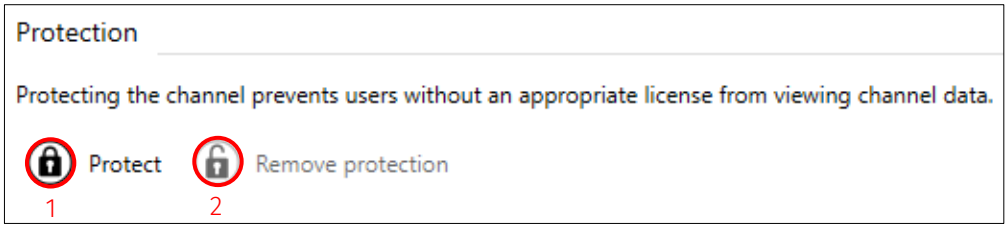
A manufacturer can also select the minimum viewing and editing user group for an item. Click the 'browse' icon (1), and then select the user group from the dropdown menu (2).




### Channel Protection

Channel Protection allows a manufacturer to hide channel data from one or more user groups. This means that when the data is opened by an unlicensed user, the channel is not shown.

Click **Protect** (1) to enable channel protection or **Remove Protection** to disable it (2).




### User Groups

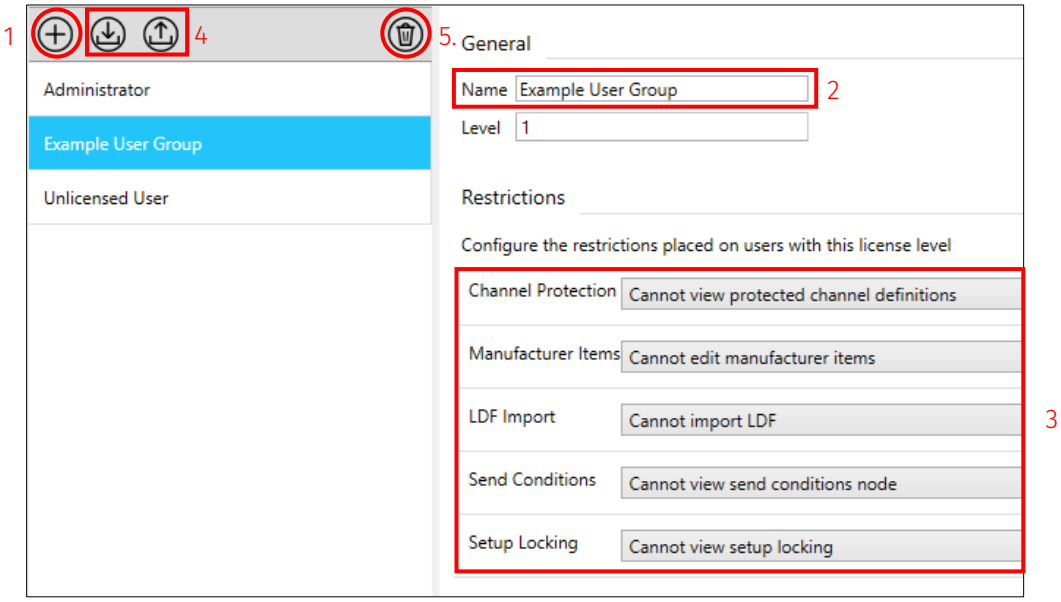
The **User Groups** menu on the **Setup Locking** node allows a manufacturer to configure the setup editing restrictions for one or more user groups.

By default there are two user groups: Administrator and Unlicensed User. The following restrictions apply:

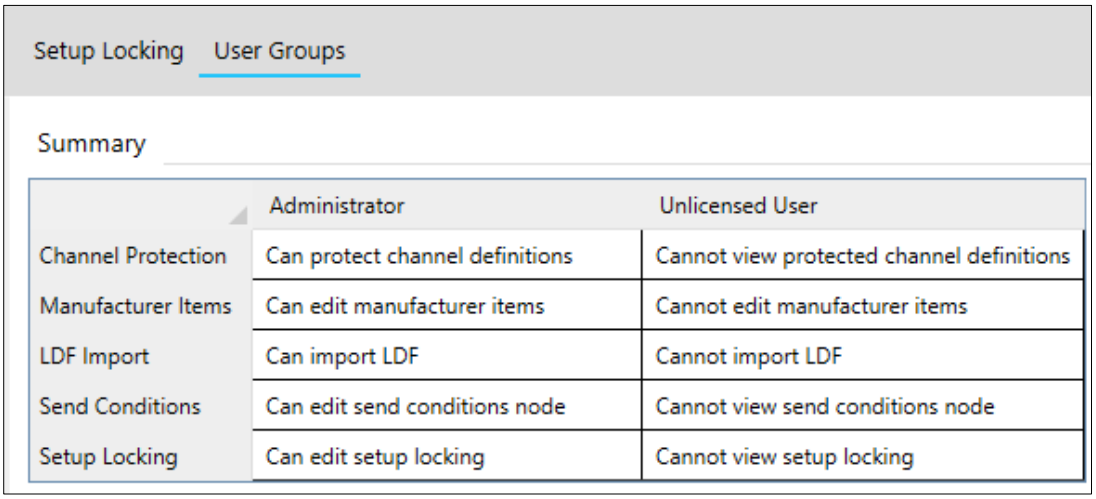
| | Administrator | Unlicensed User |
|---|---|---|
| Can protect channels | ✓ | ✗ |
| Can edit manufacturer items | ✓ | ✗ |
| Can import LDF | ✓ | ✗ |
| Can edit Send Conditions | ✓ | ✗ |
| Can edit Setup Locking | ✓ | ✗ |

A summary of the restrictions is shown at the top of the **User Groups** menu.

Click the + button (1) to create additional user groups. You can name the new user group (2) and set restrictions from the dropdown menus (3). You can import and export user groups between existing setups (4). You can use the 'bin' tool (5) to delete user groups.



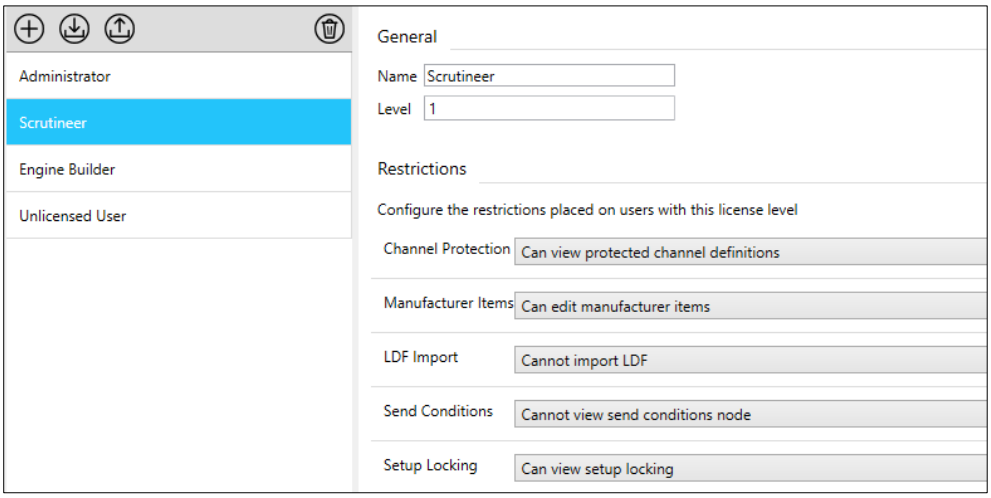

The user group 'Level' is used to define the hierarchy of user groups when a manufacturer has added one or more groups.

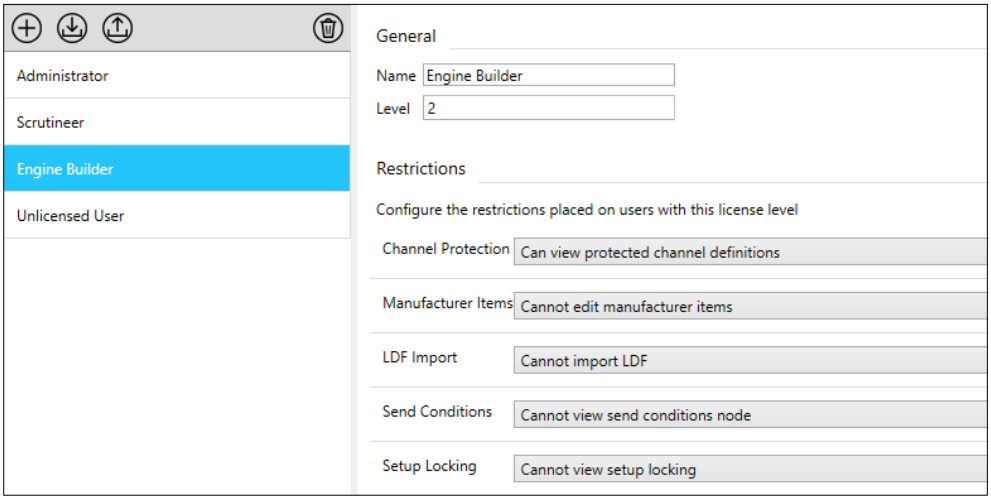



| | Administrator | Unlicensed User |
|---|---|---|
| Channel Protection | Can protect channel definitions | Cannot view protected channel definitions |
| Manufacturer Items | Can edit manufacturer items | Cannot edit manufacturer items |
| LDF Import | Can import LDF | Cannot import LDF |
| Send Conditions | Can edit send conditions node | Cannot view send conditions node |
| Setup Locking | Can edit setup locking | Cannot view setup locking |

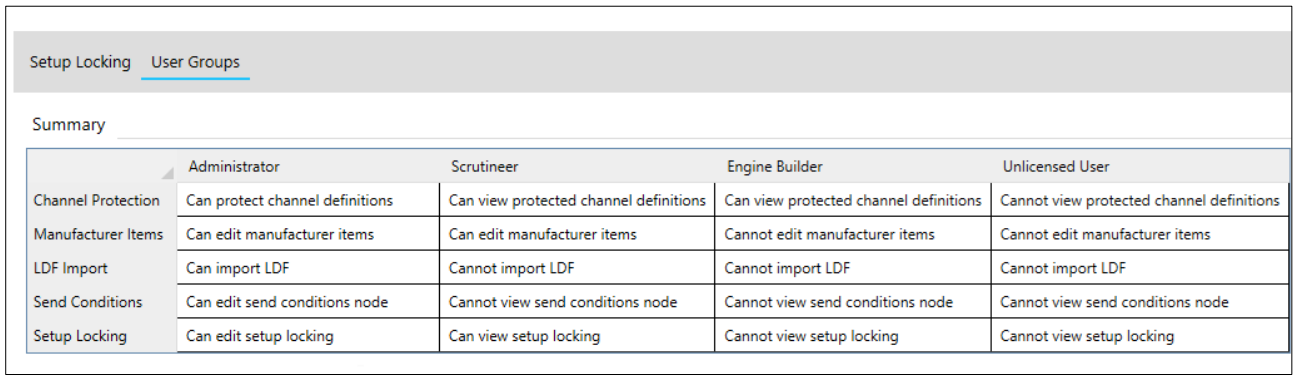
The example below shows additional ‘Scrutineer’ and ‘Engine Builder’ user groups with different restrictions applied. The Administrator is assigned ‘Level 0’ by default. A manufacturer can define the hierarchy for any new user groups. In this example ‘Scrutineer’ is assigned ‘Level 1’ and ‘Engine Builder’ is assigned ‘Level 2’. Therefore ‘Scrutineering’ is placed higher in the hierarchy.

Note: If a user group lower in the hierarchy is unrestricted from an item, then user groups higher in the hierarchy are automatically unrestricted too.



The summary of the restrictions for the user groups is updated and displayed at the top of the **User Groups** menu.




| | Administrator | Scrutineer | Engine Builder | Unlicensed User |
|---|---|---|---|---|
| Channel Protection | Can protect channel definitions | Can view protected channel definitions | Can view protected channel definitions | Cannot view protected channel definitions |
| Manufacturer Items | Can edit manufacturer items | Can edit manufacturer items | Cannot edit manufacturer items | Cannot edit manufacturer items |
| LDF Import | Can import LDF | Cannot import LDF | Cannot import LDF | Cannot import LDF |
| Send Conditions | Can edit send conditions node | Cannot view send conditions node | Cannot view send conditions node | Cannot view send conditions node |
| Setup Locking | Can edit setup locking | Can view setup locking | Cannot view setup locking | Cannot view setup locking |

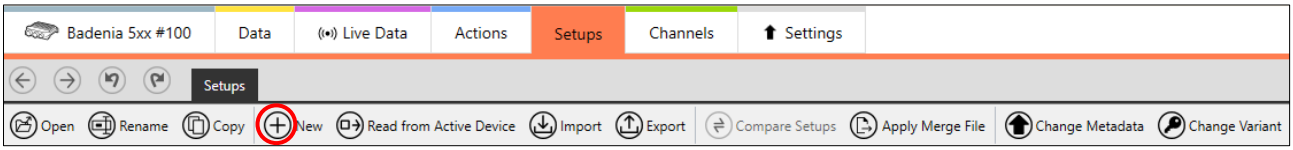
When user groups are configured, you can log channels in different tables and at different rates depending on the user group. See **Logging Tables – Rate Groups and Channel Rates – Multiple Rate Groups** for more information.

# 34. Setups page

The **Setups** page is the main hub of Toolset. From here you can find and configure all device setups.

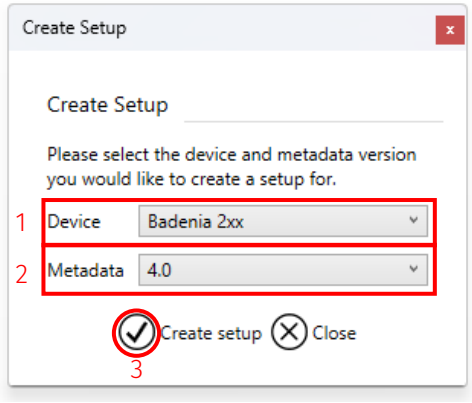
## 34.1. Create a setup

To create a new blank setup, click **New** to open the **Create Setup** dialog box.



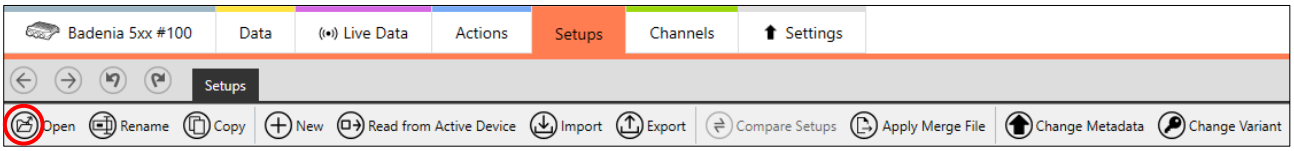

Select the device type (1) and metadata version for the setup (2). If the device type or specific metadata version is not shown, use the 'import' tool to import device metadata (see Import and Export a Setup).

You can locate device metadata in Settings – Upgrade or contact electronics.support@cosworth.com. Click **Create setup** (3) to create the new blank setup.



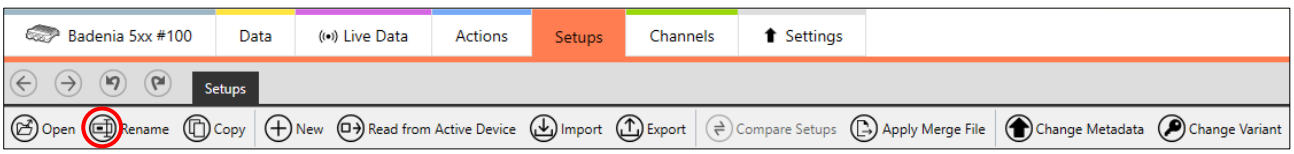

## 34.2. Open a setup

In the **Setups** menu, select a setup from the list and click **Open**, or double-click the setup.

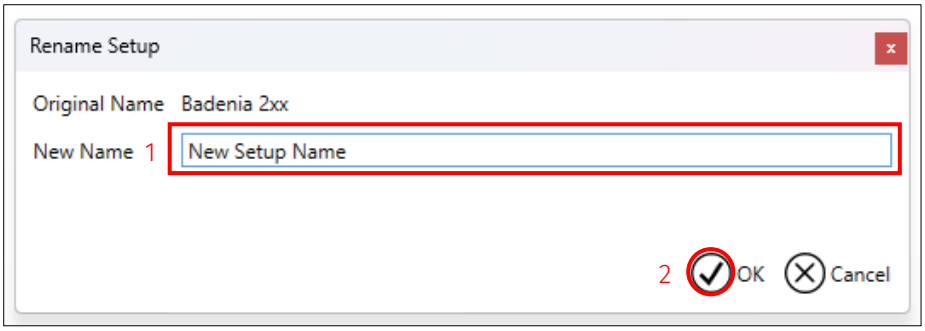



## 34.3. Rename a setup

Select a setup from the list and click **Rename**.

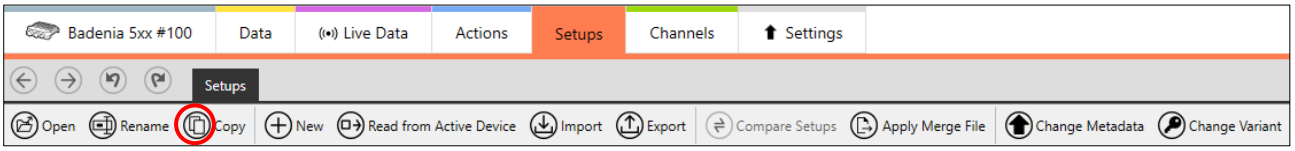


In the **Rename Setup** window enter a new setup name (1), and then click **OK** to confirm (2).




## 34.4. Copy a setup

This is useful for duplicating a setup to modify so that the original setup is preserved. Select a setup from the list and click **Copy**.




## 34.5. Delete a setup

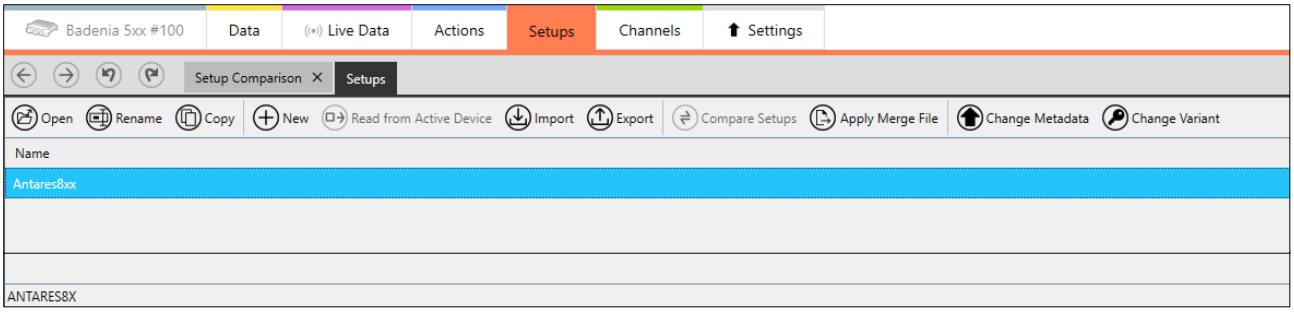
Select a setup and click the 'bin' tool.

## 34.6. Search a setup

If there are many setups on the **Setups** page, you can search and filter them with the standard search and 'magic' search functions.

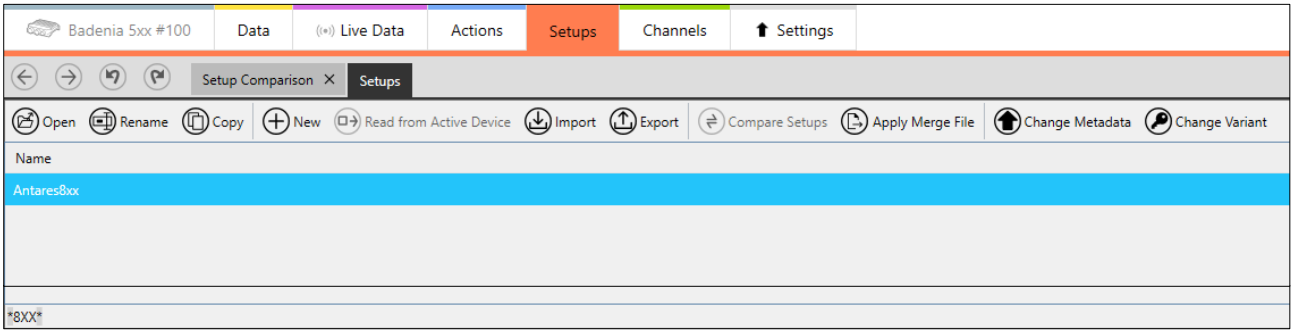
### Standard search

To use standard search, click in the **Setups** area and start to enter the setup name. The setups are filtered by the exact entered text. This search method must match the channel name exactly to show the result.

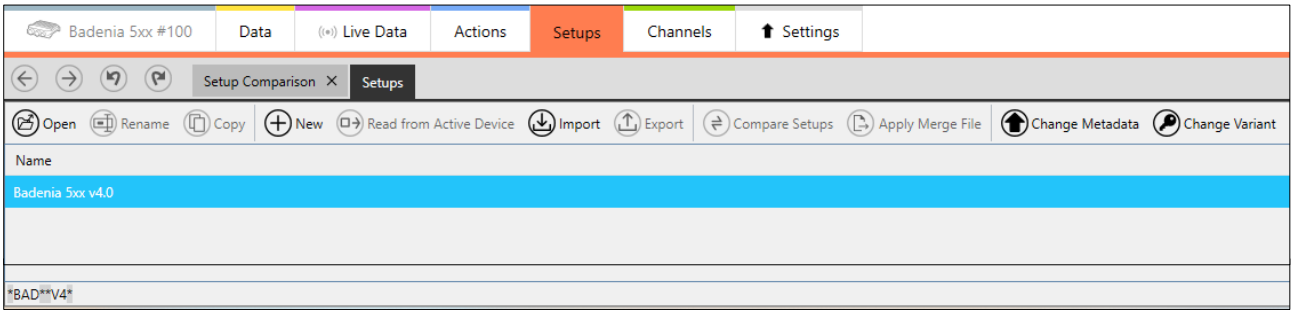



### Magic search

If the exact channel name is unknown, you can use the magic search function. Magic search is a 'broad' search tool, which can display results containing any string. Press CTRL + Shift + 8 simultaneously. A highlighted star shows the search is active. Type a specific word or phrase, and then press CTRL + Shift + 8 to close the search. An example is shown below.



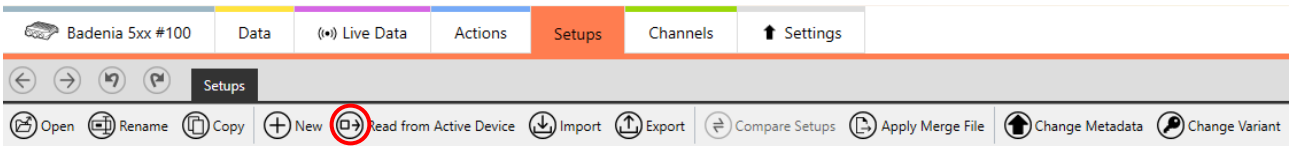

You can also combine multiple terms in the search, to find multiple parts of a setup name in a single search.




## 34.7. Read a setup from an active device

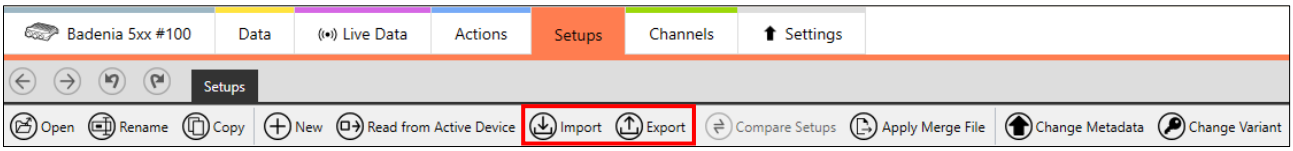
When connecting to a device with a setup already loaded, it is possible to read the setup from the device. This is useful for when the setup on the device is not already stored in the User setups page.

Tip: Always read the setup from the device to store an original copy of the setup before overwriting the setup, modifying the setup or updating device firmware and setup metadata.




## 34.8. Import and export a setup

You can use the 'import' and 'export' tools to import and export setup files (*.toolset*) between PCs and versions of Toolset.



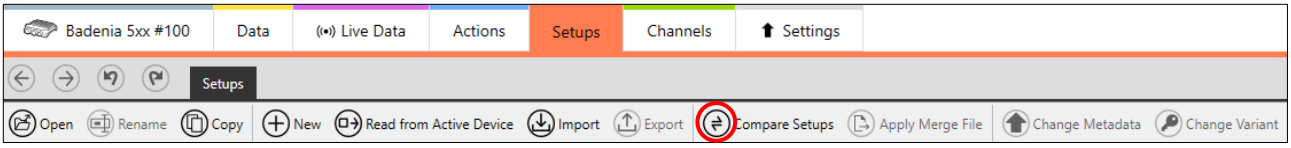

**Note:** Some versions of setup metadata are only compatible with specific versions of Toolset. If an incompatible setup is imported an error message is displayed.

## 34.9. Compare and merge setups

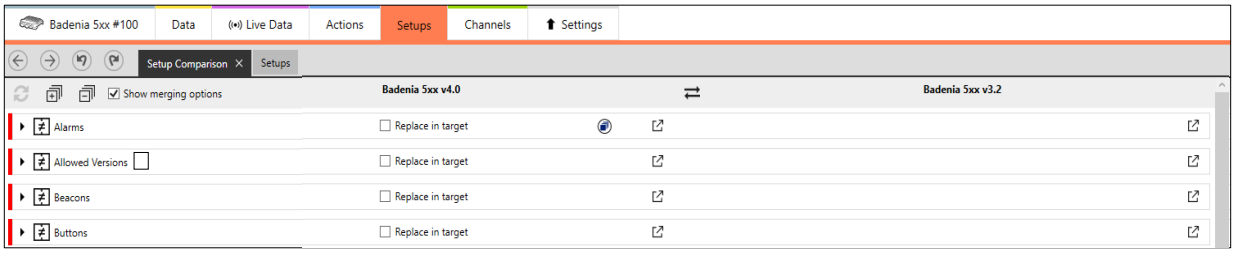
Toolset can compare setups which can be useful for identifying differences between setups. The **Compare Setups** tool also allows you to merge the setup differences from one setup to another, allowing efficient updating of setup configurations between setups.

To enable the **Compare Setups** tool, select two setups in the setup library and use the **Ctrl** key to highlight them.


Select the **Compare Setups** tool.



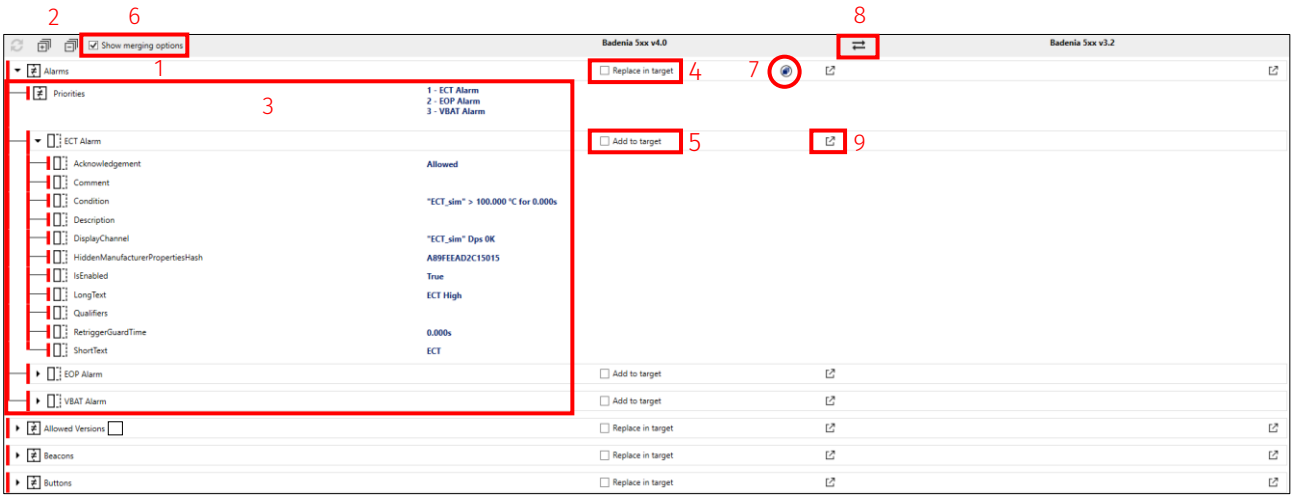

This opens a new tab on the **Setups** page where the difference between setups is displayed by node.



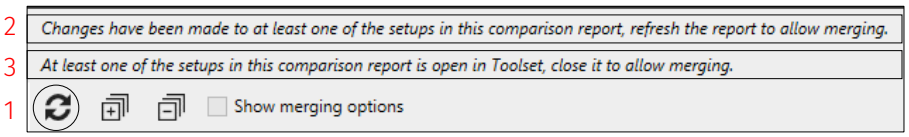
Click on a 'top level entry' (node) to open a lower level of 'children' (a list of items within the node that are different between setups) (1). You can open and collapse all entries with the 'Expand' and 'Collapse All' tools (2). Click on an 'child' item to view the individual item and its differences (3).

If an item exists in the target setup you can select the **Replace in Target** option (4). If an item does not exist in the target setup, then select the **Add to Target** option (5). If the no merging options are displayed make sure that the **Show merging options** box is selected (6). Use the **Select all children of this item** tool to select all merging options for an item (7).

The 'base' setup is displayed on the left of the **Compare** tool, with the target setup displayed on the right. If the home and target setups are not in the correct position, use the 'invert' tool to switch the home and target setups (8). Use the 'navigate' tools to explore setup items (9). This is useful for inspecting items before selecting them to merge.

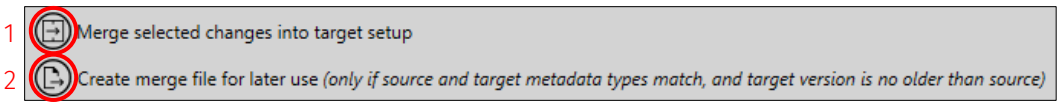



The 'navigate to item' tool opens the setup in a new tab. You can open setups in compare setups, but if any changes are made, use the 'refresh' tool (1) to update merging options. A warning message is displayed to remind you (2). You must close the setup after any changes are made to allow merging options to be redisplayed. Again, a warning message is displayed to remind you (3).



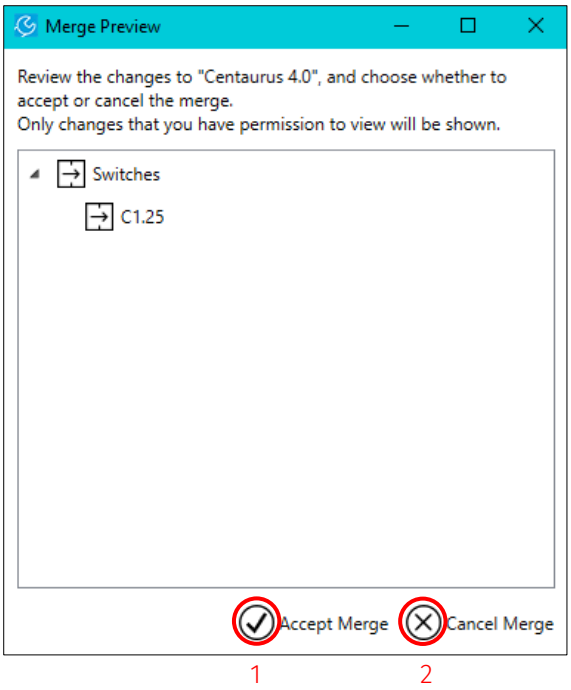

Select the required merging options. There are two options:

- Merge – Merge all the selected items now (1)
- Create merge file – you can create a file containing the changes and apply it later (2)

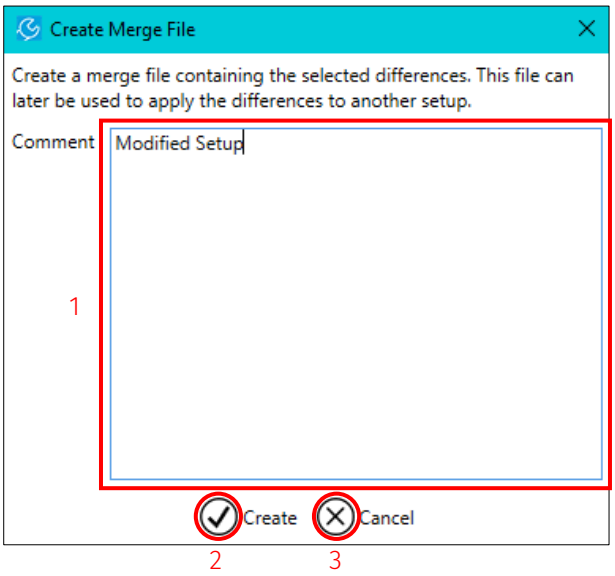



If **Merge selected changes into target setup** is selected, you are prompted to accept the merge.

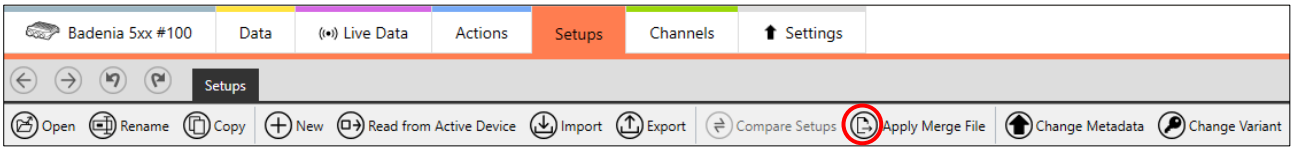
Click **Accept Merge** (1) to update the target setup with the changes. It is recommended that you make copies of setups when merging, so that the original setups are saved. Click **Cancel Merge** to cancel the merge if an error is identified in the preview (2).




If you select **Create merge file for later use** you are prompted to add a comment (1). Click **Create** to display a window that allows you to save the *.merge* file (2). You can cancel the creation of the merge file if an error is identified (3).

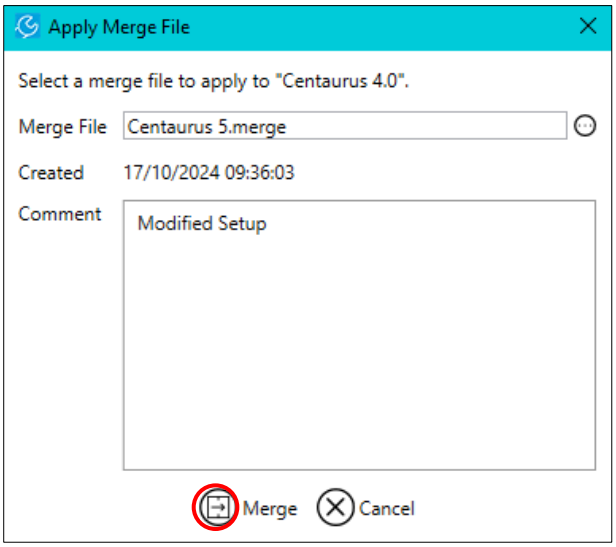


When you need to use the merge file, click on the target setup and select the **Apply Merge File**.

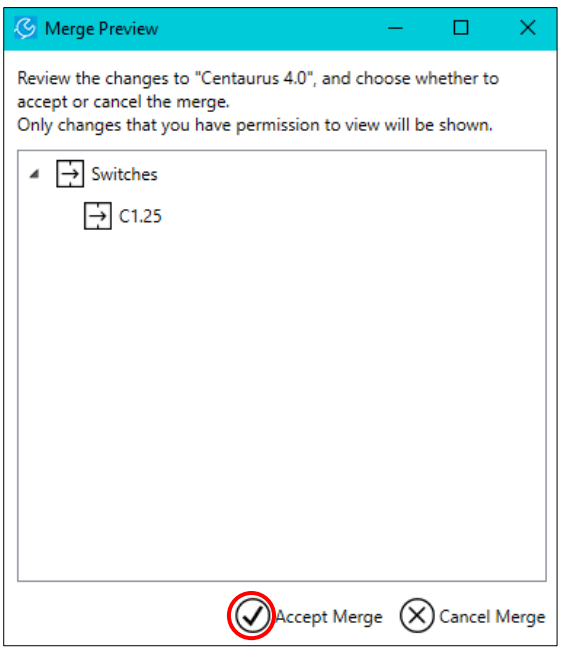



A popup window opens where you can search for the required *.merge* file. When selected the window displays the comment added during the merge creation.

Click **Merge** to display a preview which summarises the setup changes.

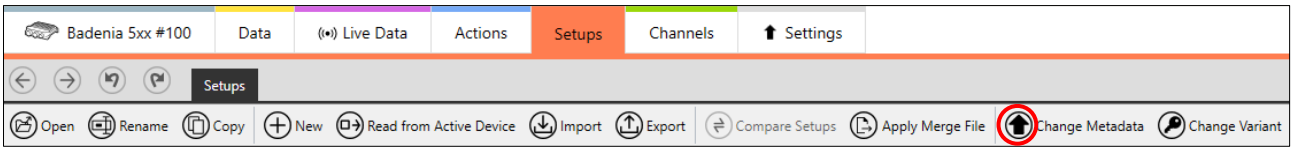



Click **Accept Merge** to add the changes to the target setup.



## 34.10. Change setup metadata

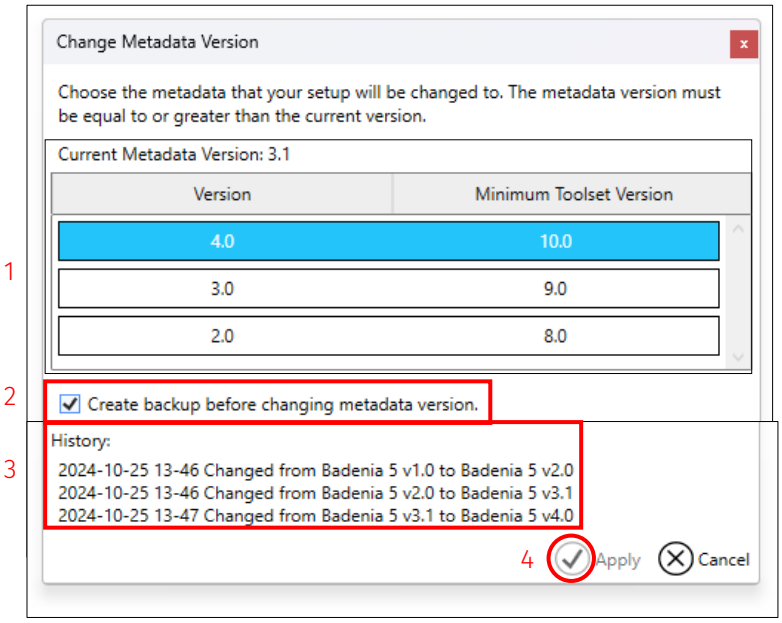
The setup metadata version must match the device firmware. If the device firmware is updated, then then you must also update the setup metadata before the setup can be sent. To change setup metadata, use the **Change Metadata** option.




The **Change Metadata Version** menu allows you to select from the available metadata versions (1).

If the required metadata version is not shown, you can use the 'import' tool (see Import and export a setup). Device metadata is located in Settings – Upgrade, or contact electronics.support@cosworth.com.

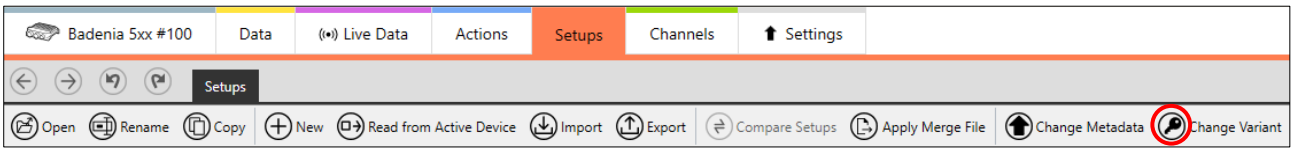
Make sure that you create a backup of the original setup before changing metadata versions. Select the **Create backup before changing metadata version** option to automatically make a save a copy of the original setup (2). The setup metadata history of the setup is stored so you can view the update history (3). Click **Apply** (4) to change the setup metadata version.




## 34.11. Changing setup variant

Like firmware and metadata versions, the setup variant must match the device variant. If a new setup is created, or a blank metadata file is imported, the setup variant must be selected before the setup can be sent.

To select the setup variant, use the **Change Variant** tool.



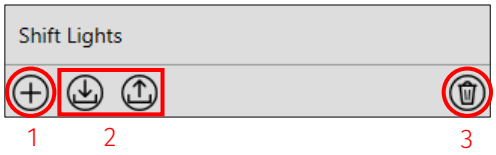

If the setup variant is not set before you attempt to send a setup, or the setup variant is different from the device variant, you are prompted to select 'Use Device Variant' or 'Use Setup Variant'.

# 35. Shift lights

The Shift Lights node is used to configure the shift lights on display devices such as the CDU range and the CCW Mk2 and CCW Mk3.

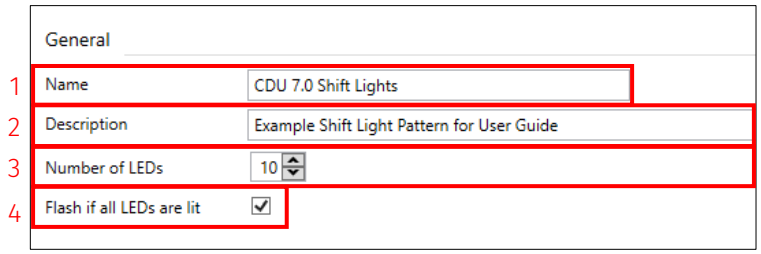
## 35.1. Add a new shift light pattern

The first step to add a new shift light configuration is to add a new shift light pattern group. Use the + tool to add a new shift light pattern group (1). You can import and export shift light pattern groups between existing setups (2). Use the 'bin' tool to delete an unwanted shift light pattern group (3).

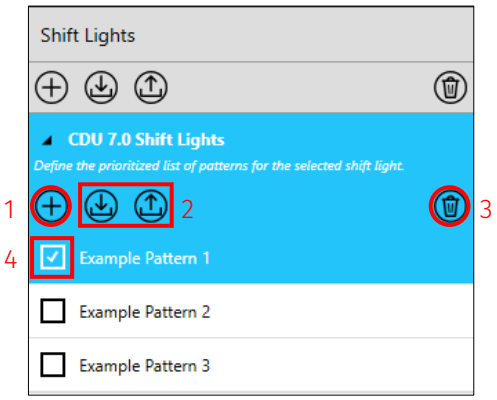



You can enter a name for the new group (1) and add an optional comment about the group (2).

The number of LEDs for the pattern group must be set (3). For example, a pattern group for CDU 7.0 with 10 shift lights is set to 10 LEDs. You can select an option to Flash if all LEDs are lit (4). This is useful for alerting the driver to shift gear.



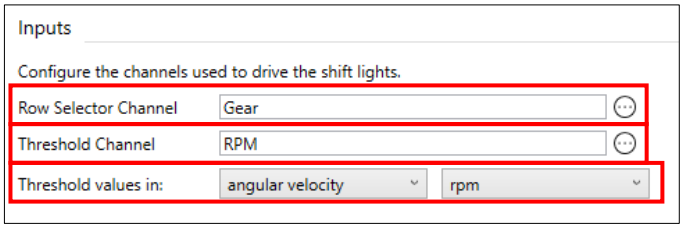

You can then add shift light patterns. Click the + tool within the pattern group to add a new pattern (1). You can import and export patterns between existing setups (2). Use the 'bin' tool to delete a pattern (3). Enable or disable patterns from the setup using the 'Enable' check box (4).



## 35.2. Configure shift light patterns

To configure a shift light pattern, click the shift light pattern group to select the input channels to drive the shift light pattern.

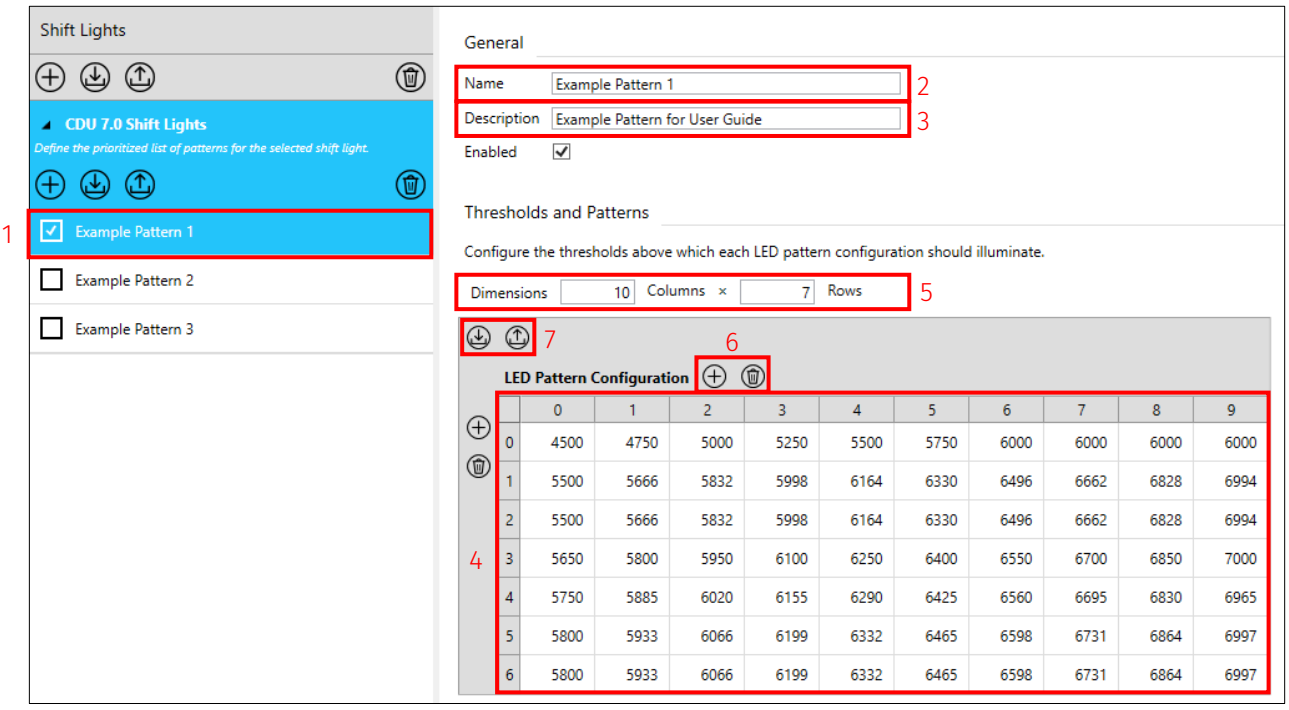
Two channels are required for the 'Row Selector Channel' and 'Threshold Channel'. The 'Row Selector' channel is most likely to be the 'Gear' channel (1) and the 'Threshold' channel is most likely to be RPM (2). Click on the 'browse' buttons to select another channel. You must configure the units for the Threshold Channel value (3).



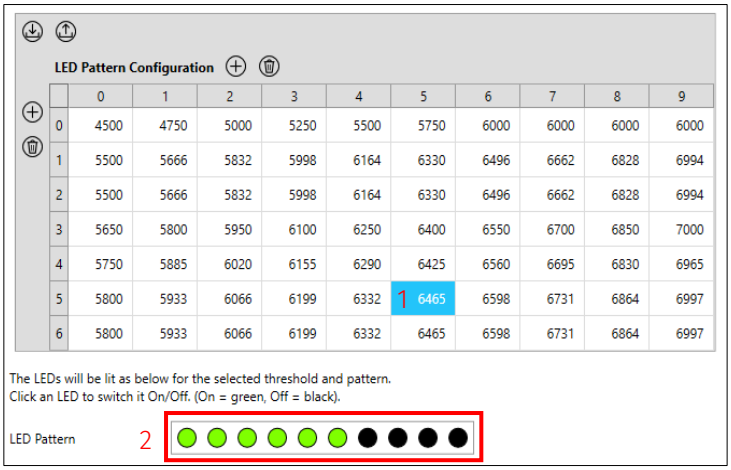

To configure the shift light pattern, select the new pattern from the menu (1). You can name the pattern (2) and add an optional comment (3).

The pattern is configured using the pattern table (4). Configure the table with the required number of rows (for example, gears) and thresholds/columns (for example, RPM breakpoints) (5). You can add rows and thresholds and delete them with the + and 'bin' tools (6). You can import and export Pattern tables between existing setups (7).

Note: You can create shift light pattern tables in Excel and copy and paste them into Toolset. This allows you to create complex tables that require calculations to define the thresholds, saving time and simplifying the process.



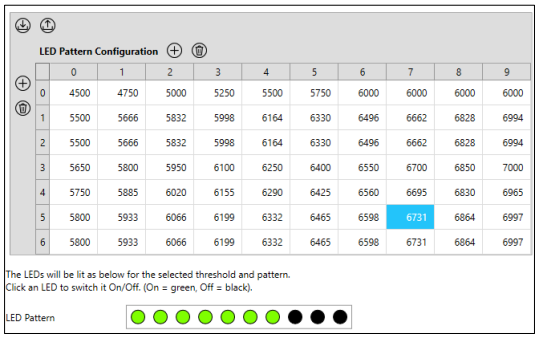
Click a cell in the table to configure the LED pattern (1). Select which LEDs are on for the cell by clicking LEDs within the LED strip (2). The LED strip shows a preview of how LEDs light up based on the thresholds. Green circles indicate LEDs that will be lit. Click them to toggle them on/off.




| | 0 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 |
|---|---|---|---|---|---|---|---|---|---|---|
| 0 | 4500 | 4750 | 5000 | 5250 | 5500 | 5750 | 6000 | 6000 | 6000 | 6000 |
| 1 | 5500 | 5666 | 5832 | 5998 | 6164 | 6330 | 6496 | 6662 | 6828 | 6994 |
| 2 | 5500 | 5666 | 5832 | 5998 | 6164 | 6330 | 6496 | 6662 | 6828 | 6994 |
| 3 | 5650 | 5800 | 5950 | 6100 | 6250 | 6400 | 6550 | 6700 | 6850 | 7000 |
| 4 | 5750 | 5885 | 6020 | 6155 | 6290 | 6425 | 6560 | 6695 | 6830 | 6965 |
| 5 | 5800 | 5933 | 6066 | 6199 | 6332 | 6465 | 6598 | 6731 | 6864 | 6997 |
| 6 | 5800 | 5933 | 6066 | 6199 | 6332 | 6465 | 6598 | 6731 | 6864 | 6997 |

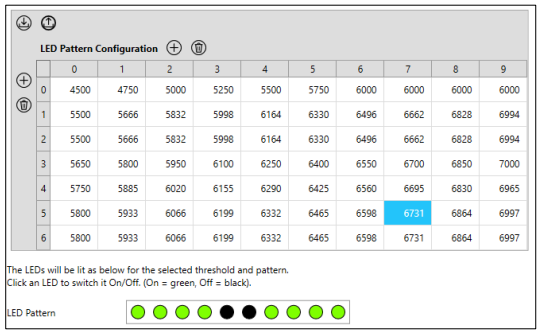
This allows you to configure various LED patterns. For example:

- Left to right increment



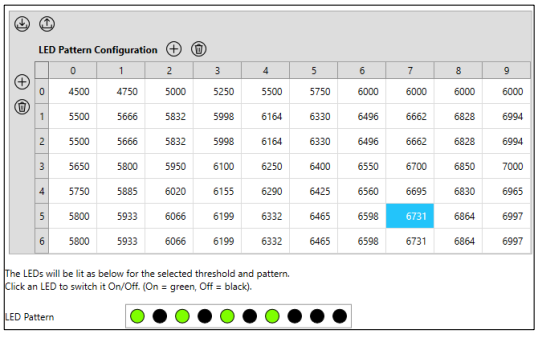

| | 0 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 |
|---|---|---|---|---|---|---|---|---|---|---|
| 0 | 4500 | 4750 | 5000 | 5250 | 5500 | 5750 | 6000 | 6000 | 6000 | 6000 |
| 1 | 5500 | 5666 | 5832 | 5998 | 6164 | 6330 | 6496 | 6662 | 6828 | 6994 |
| 2 | 5500 | 5666 | 5832 | 5998 | 6164 | 6330 | 6496 | 6662 | 6828 | 6994 |
| 3 | 5650 | 5800 | 5950 | 6100 | 6250 | 6400 | 6550 | 6700 | 6850 | 7000 |
| 4 | 5750 | 5885 | 6020 | 6155 | 6290 | 6425 | 6560 | 6695 | 6830 | 6965 |
| 5 | 5800 | 5933 | 6066 | 6199 | 6332 | 6465 | 6598 | 6731 | 6864 | 6997 |
| 6 | 5800 | 5933 | 6066 | 6199 | 6332 | 6465 | 6598 | 6731 | 6864 | 6997 |

- Increment to centre




| | 0 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 |
|---|---|---|---|---|---|---|---|---|---|---|
| 0 | 4500 | 4750 | 5000 | 5250 | 5500 | 5750 | 6000 | 6000 | 6000 | 6000 |
| 1 | 5500 | 5666 | 5832 | 5998 | 6164 | 6330 | 6496 | 6662 | 6828 | 6994 |
| 2 | 5500 | 5666 | 5832 | 5998 | 6164 | 6330 | 6496 | 6662 | 6828 | 6994 |
| 3 | 5650 | 5800 | 5950 | 6100 | 6250 | 6400 | 6550 | 6700 | 6850 | 7000 |
| 4 | 5750 | 5885 | 6020 | 6155 | 6290 | 6425 | 6560 | 6695 | 6830 | 6965 |
| 5 | 5800 | 5933 | 6066 | 6199 | 6332 | 6465 | 6598 | 6731 | 6864 | 6997 |
| 6 | 5800 | 5933 | 6066 | 6199 | 6332 | 6465 | 6598 | 6731 | 6864 | 6997 |

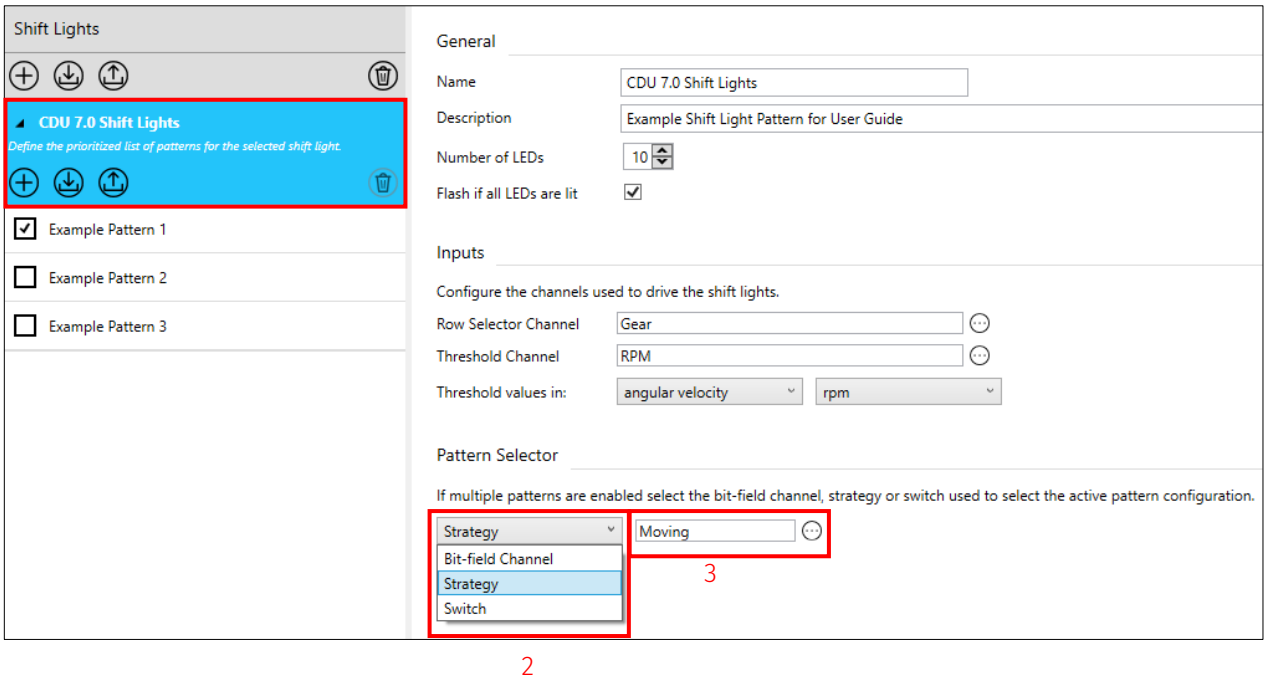
- Missing LEDs



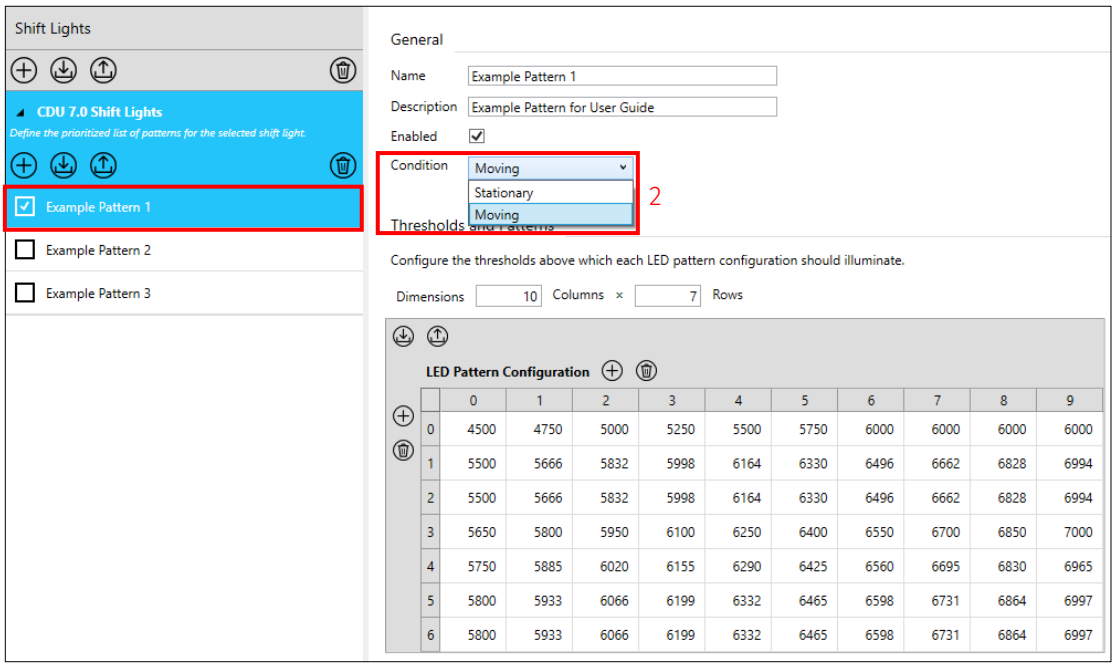

| | 0 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 |
|---|---|---|---|---|---|---|---|---|---|---|
| 0 | 4500 | 4750 | 5000 | 5250 | 5500 | 5750 | 6000 | 6000 | 6000 | 6000 |
| 1 | 5500 | 5666 | 5832 | 5998 | 6164 | 6330 | 6496 | 6662 | 6828 | 6994 |
| 2 | 5500 | 5666 | 5832 | 5998 | 6164 | 6330 | 6496 | 6662 | 6828 | 6994 |
| 3 | 5650 | 5800 | 5950 | 6100 | 6250 | 6400 | 6550 | 6700 | 6850 | 7000 |
| 4 | 5750 | 5885 | 6020 | 6155 | 6290 | 6425 | 6560 | 6695 | 6830 | 6965 |
| 5 | 5800 | 5933 | 6066 | 6199 | 6332 | 6465 | 6598 | 6731 | 6864 | 6997 |
| 6 | 5800 | 5933 | 6066 | 6199 | 6332 | 6465 | 6598 | 6731 | 6864 | 6997 |

## 35.3. Multiple shift light patterns

You can add multiple shift light patterns to a shift light pattern group. If you add multiple shift light patterns, then you must configure the shift light pattern selector on the **Shift Lights** pattern group page. Click the shift light pattern group (1), select whether a Bit-field Channel, Strategy, or Switch is used to select the pattern (2), and the select an available channel from the 'browse' menu (3).

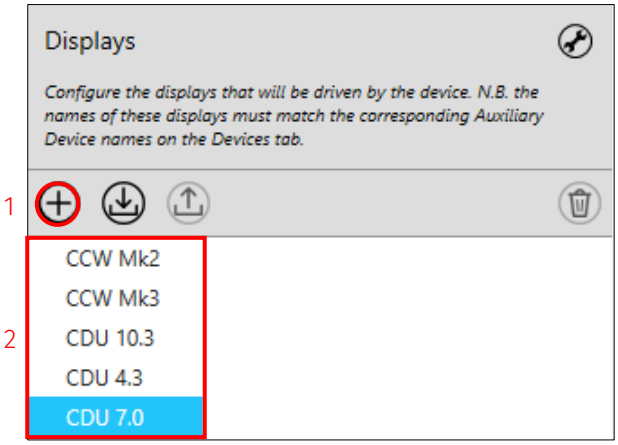



When you configure the pattern selector, the **Condition** box is displayed on the **Shift Lights** pattern page. You can the select the condition that triggers the shift light pattern.

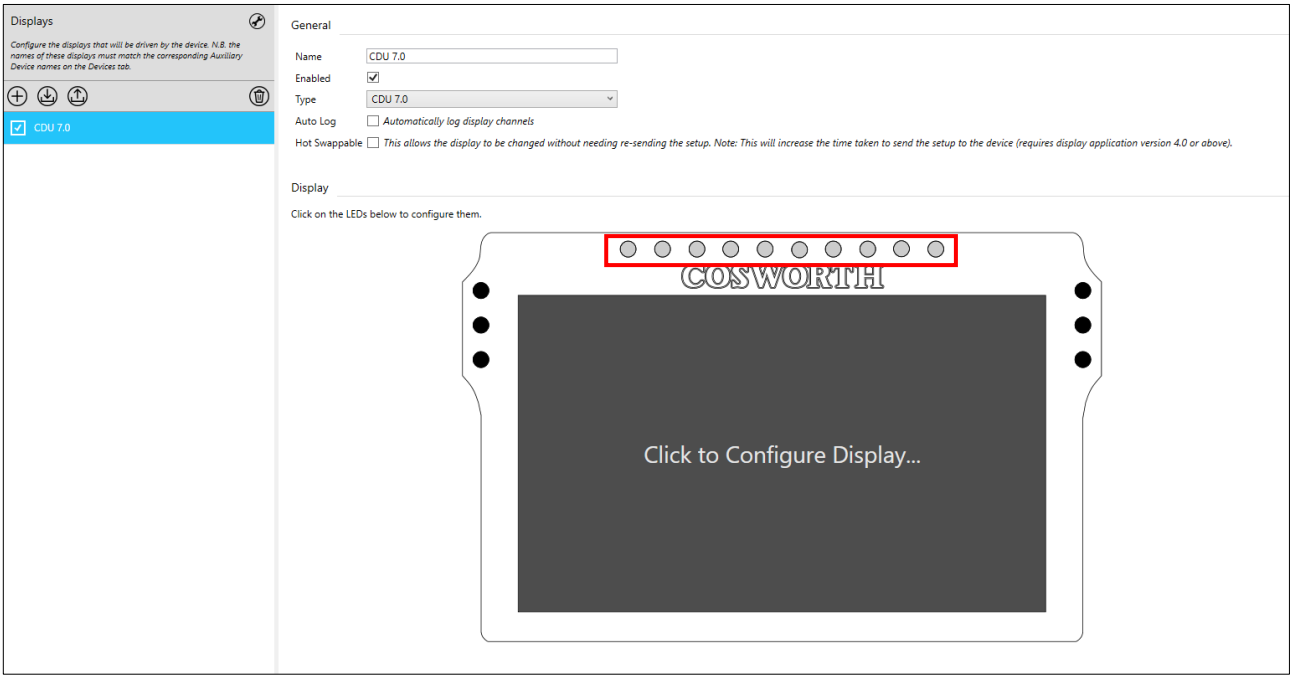


## 35.4. Apply shift lights to a display device

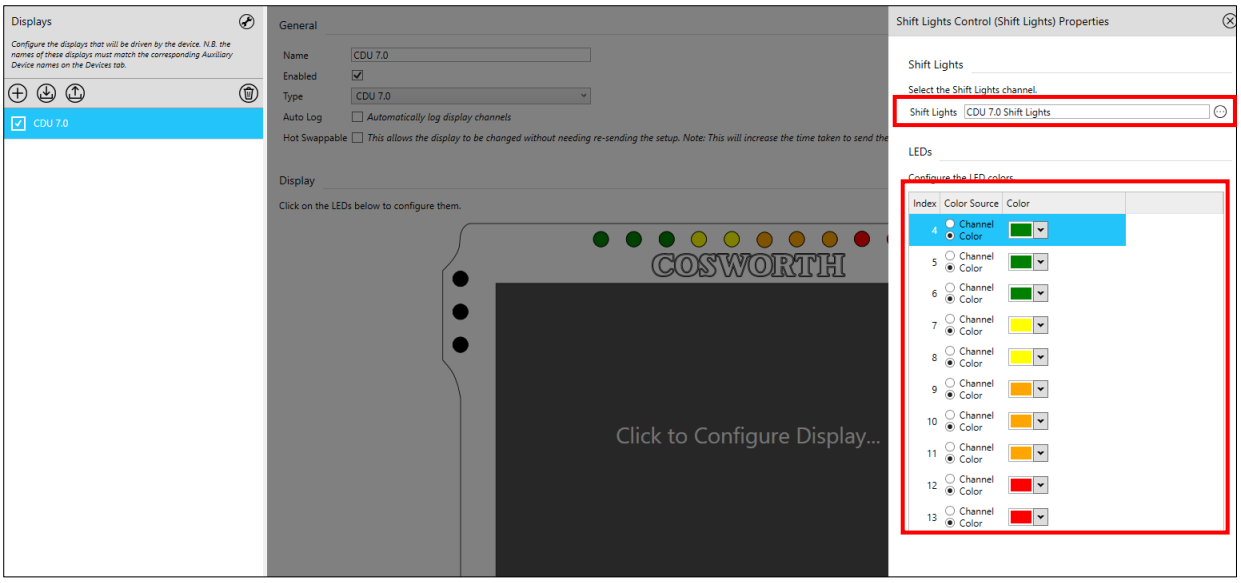
Once you configure a shift light pattern you can apply it to the display device. On the Displays node, use the + tool to add a display (1), and then select the required device from the menu (2).




Click the shift lights at the top of the display to open the **Shift Lights Control Properties** menu.

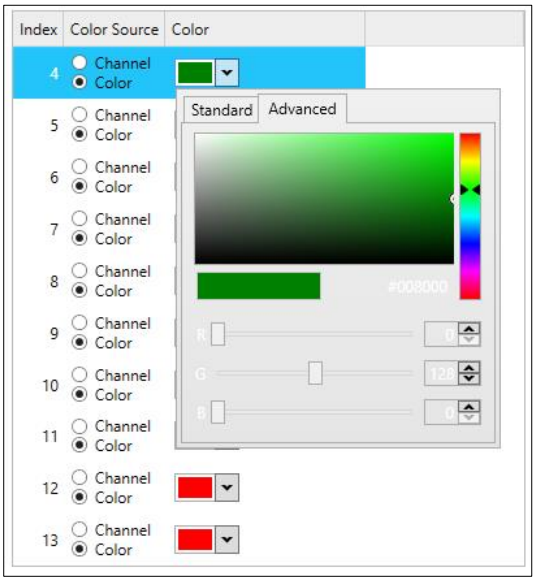



Select the shift light pattern from the menu (1) and configure the colour for each shift light (2).



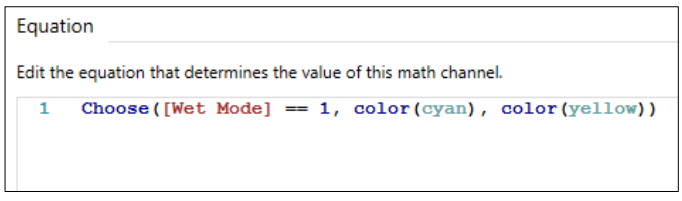

## 35.5. Shift light colours (static and dynamic)

You can select the colour of a shift light to be a static colour. Select the source as 'Color' and select the required colour from the Standard or Advanced colour menus.




You can also set a dynamic shift light colour where the colour changes according to one or more condition(s). Select 'Channel' as the source and a configured colour channel (see Maths Channels – Color Control for more information).

For example, the following maths channel sets a shift light to cyan if 'Wet Mode' is active, and yellow if 'Wet Mode' is not active.

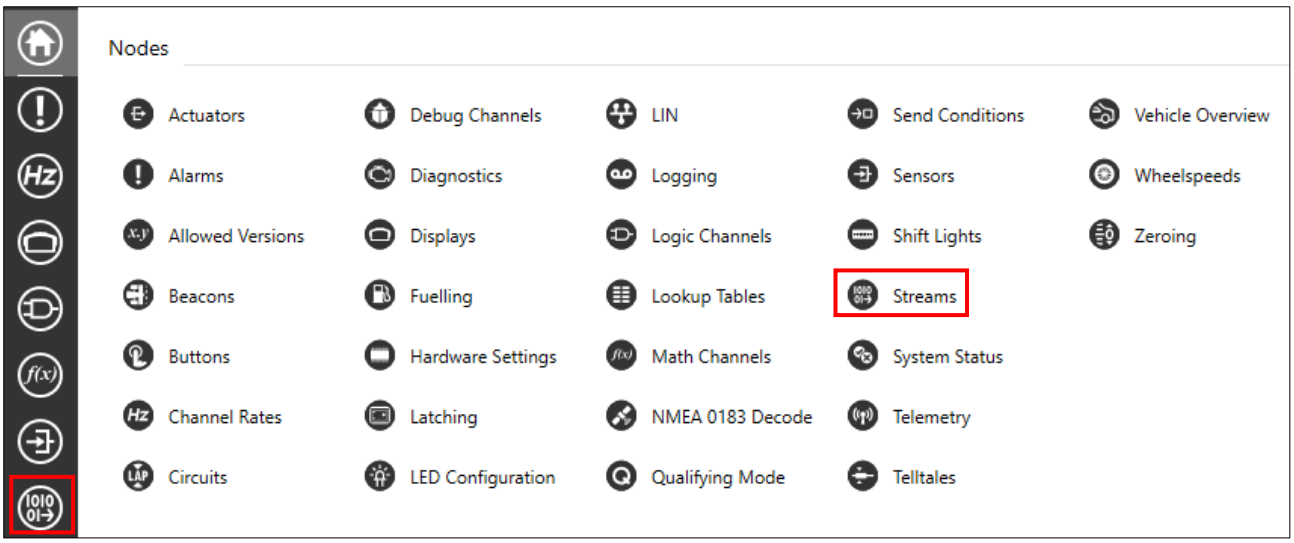


# 36. CAN streams

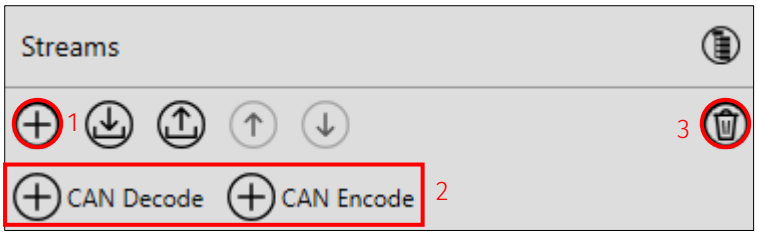
Control Area Network (CAN) is a widely used communication protocol for system integration. Most Cosworth devices support CAN communication and interfacing with both Cosworth and third-party CAN devices. The Streams node is used to configure CAN streams.

### Create a CAN stream

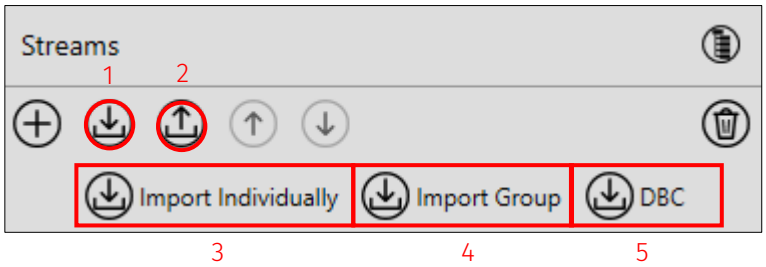
Within a setup, click on the 'Streams' option on the left side or use the Streams node to access all the CAN streams. The hierarchy of a CAN steam is as follows: Stream, Packets, Bytes, and Bits. Channels can be multiple Bytes or Bits.



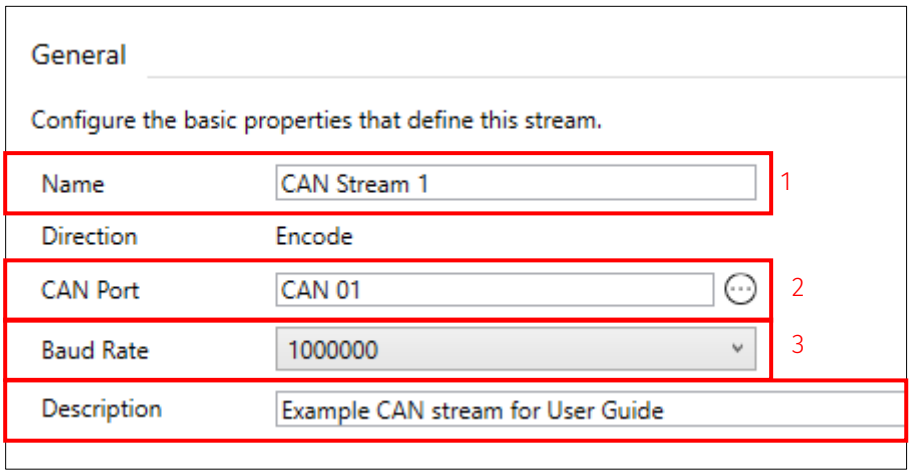

To create a new stream, hover over the + button (1) and select either CAN Decode or CAN Encode (2). Use the 'bin' tool to delete unwanted streams (3).




You can import (1) and export (2) streams either individually (3) or in multiples within a group (4) between existing Toolset setups in Toolset Library File (*.tlf*) format and import and export in the standard CAN DBC (*.dbc*) format (5).

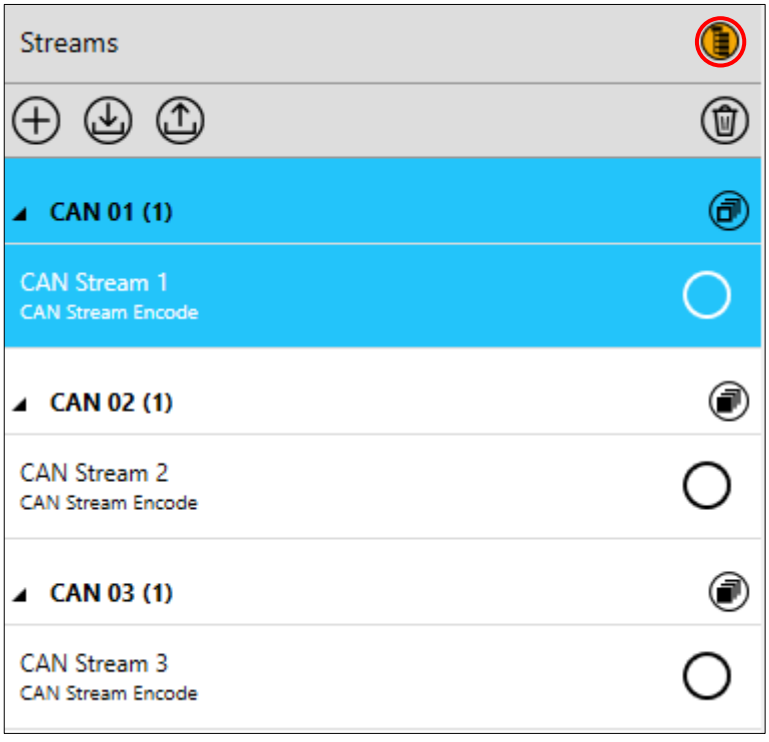


When you add a new stream, the stream appears in the streams list. Configure the stream at the right in the main window. In the General section you can configure the stream name (1), CAN Port allocation (2), Baud Rate (3) and a description (4).

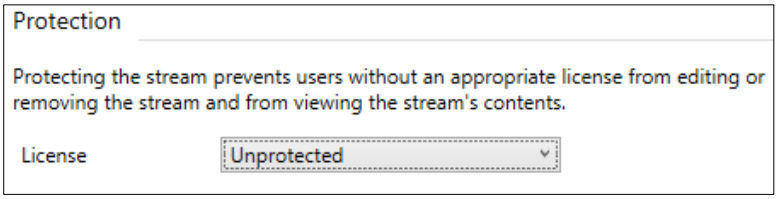



Note: You can configure the CAN Port name on the Hardware Settings node.

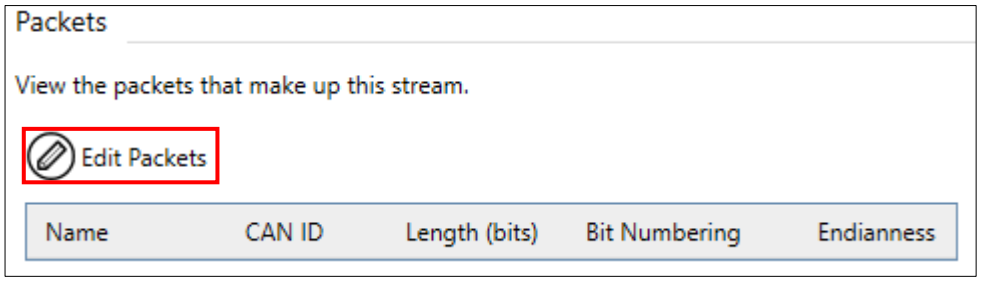
If you configure multiple streams, you can select the 'display view' option to display the Streams list in groups linked to the CAN port allocation for each stream.



In the **Protection** section you can add permissions to the stream via a dongle to allow you to edit or view the stream content. This feature is typically used by manufacturers to lock their setups, streams, and so on.




The **Packets** section hosts each packet in the stream. Click **Edit Packets** to begin to create a packet.



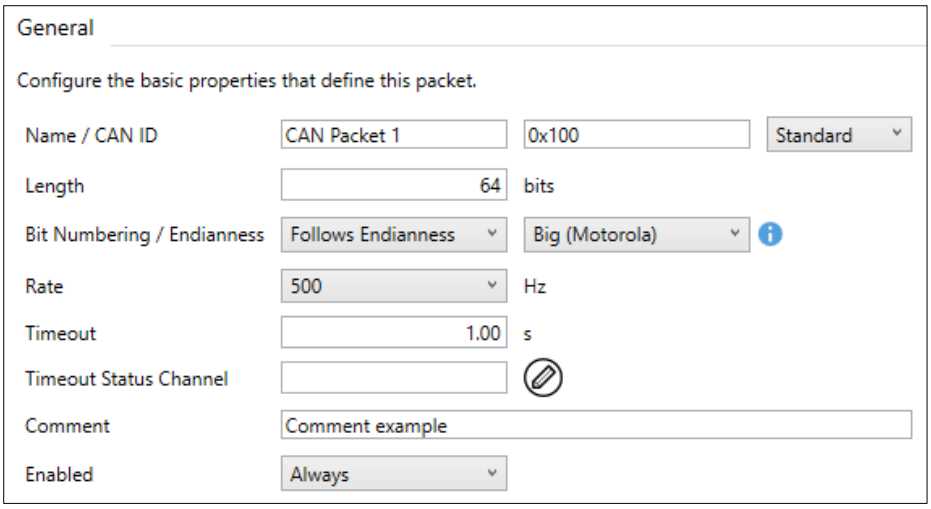

Click the **+** button at the top left to create a new packet. This populates the list below with a new packet. Use the 'bin' icon to remove a selected packet Click the + button (1) to add a new packet. Use the 'bin' tool to delete an unwanted packet (2).


Once a packet is added, you can complete configuration from the main window. In the **General** section you can edit the **Name/CAN ID**, **Length**, **Bit Numbering/Endianness**, **Rate**, **Timeout**, **Timeout Status Channel**, and **Comment**. The **CAN ID** needs to be unique on for that specific CAN Port.

The **Length** has a maximum of 64 bits. You can change **Endianness** according to preference, but it needs to correspond to the opposing Encode/Decode. For more information about Endianness/Bit Numbering methodologies click the **i** option.

The **Rate** selected is the limit for all channels in that packet.

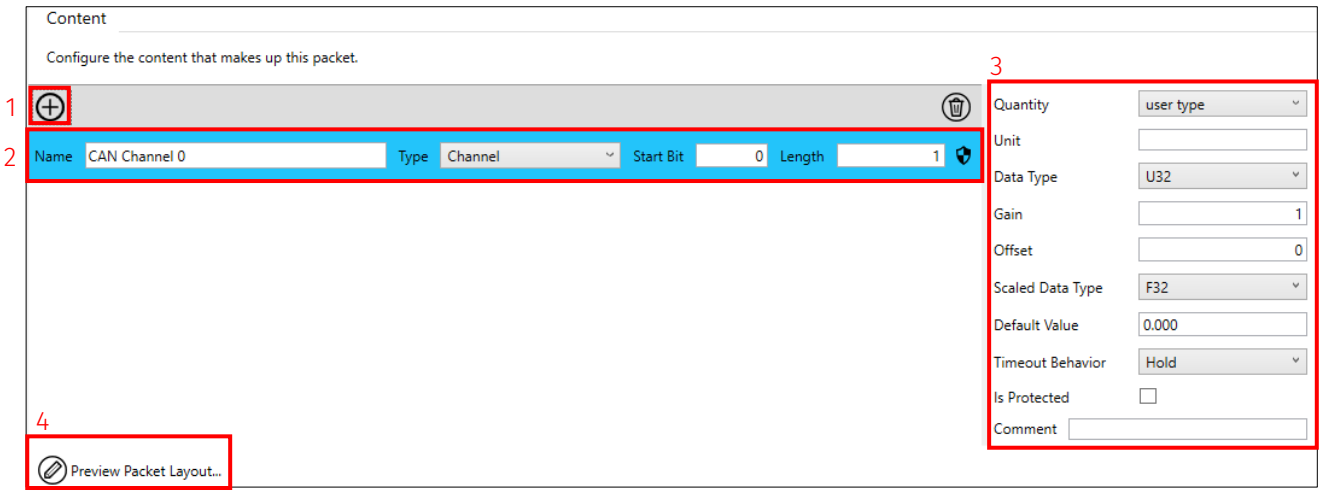
The **Timeout** option is used to set an interval after which the Encode/Decode triggers the Timeout Behaviour (explained later in this section). You can use the **Timeout Status Channel** to generate a channel that records the status of a timeout.




To add a channel to the packet, click the **+** icon (1) in the **Channels** section. This adds a channel to the list (2) and populates the channel configurations (3) to the right. The channel list contains the following information: **Name**, **Type**, **Start Bit**, and **Length**.

**Note 1**: Channel names are limited to 24 characters, including spaces.

**Note 2**: It is best to use a math channel when dealing with complex order of operations for Gain/Offset.



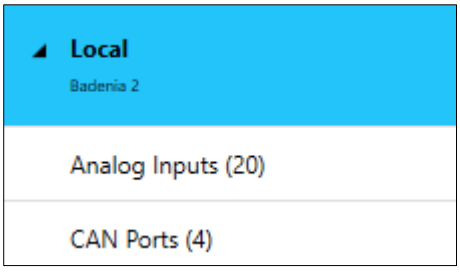
Click **Preview Packet Layout** (4), a visual representation of the packet layout is displayed. This enables an easier visualisation of the bit allocation of items within the packet. LSB means least significant bit and MSB means most significant bit (see below).

| | 7 | 6 | 5 | 4 | 3 | 2 | 1 | 0 |
|---|---|---|---|---|---|---|---|---|
| 0 | 63 | 62 | 61 | 60 | 59 | 58 | 57 | 56 |
| 1 | 55 | 54 | 53 | 52 | 51 | 50 | 49 | 48 |
| 2 | 47 | 46 | 45 | 44 | 43 | 42 | 41 | 40 |
| 3 | 39 | 38 | 37 | 36 | 35 | 34 | 33 | 32 |
| 4 | 31 | 30 | 29 | 28 | 27 | 26 | 25 | 24 |
| 5 | 23 | 22 | 21 | 20 | 19 | 18 | 17 | 16 |
| 6 | 15 CAN Chan... MSB | 14 | 13 | 12 | 11 | 10 | 9 | 8 |
| 7 | 7 CAN Chan... | 6 | 5 | 4 | 3 | 2 | 1 | 0 LSB |

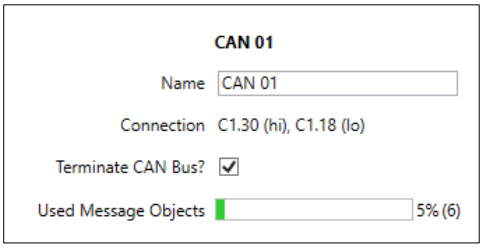
**Hardware limitations**

Each CAN Bus has a limit to the number of messages it can send or receive. Click on the Hardware Settings node to see details about CAN bus usage.

On the left side, a list is populated. Select **CAN Ports > Local** to view information about port usage.

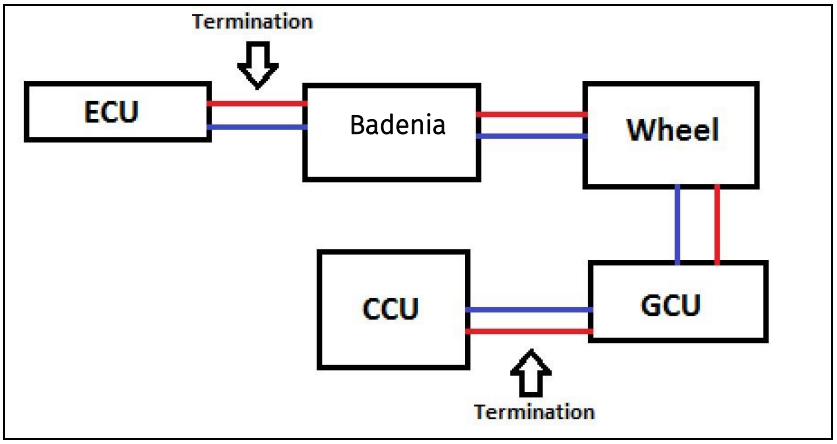


This displays each CAN Port and its usage. It is recommended that you keep the utilization at 80% or below. Oversaturating the CAN bus can lead to CAN errors and potentially missed data.




You can use the **Terminate CAN Bus** option to apply a software selectable CAN bus termination. This acts as a physical termination for that bus, if required.

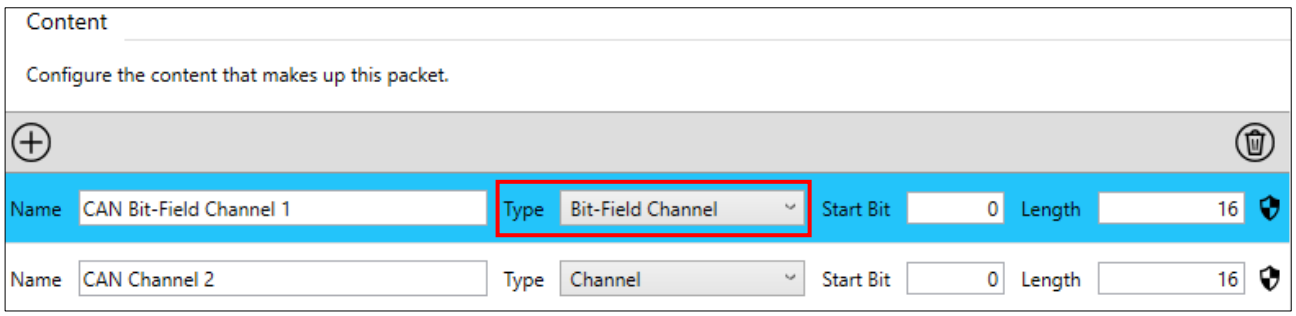
This is a complete CAN Bus setup that has two physical terminations.



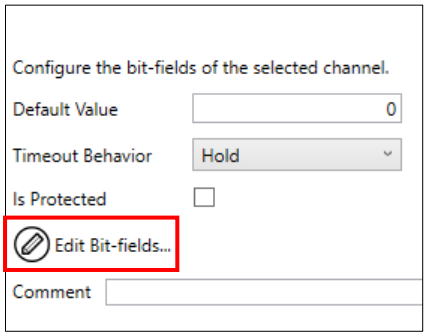

**Implement a Bit-masked CAN channel**

Bit-masking a CAN channel can be useful when information from each bit is beneficial.

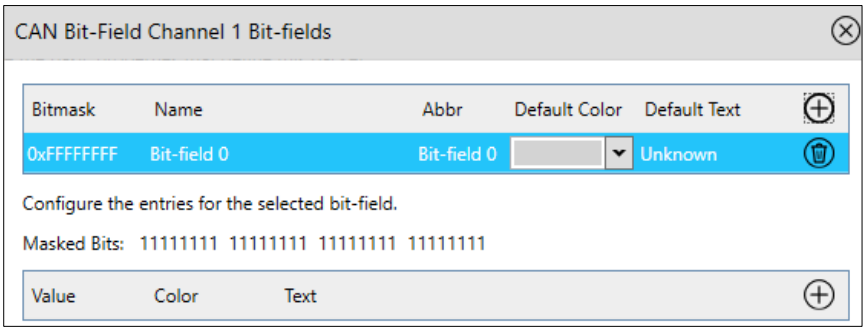
When you create a CAN Channel there is an option to add a Bitmask. The Bitmask ANDs with the channel and returns the bit entries that are true. To create a Bit-masked CAN Channel, create a normal channel as described above, but set the **Type** to *Bit-Field Channel*.



Select **Edit Bit-fields** to edit the Bit-fields.




A window is displayed where you can click the **+** icon to add a Bitmask. The top section relates to the Bitmask and the bottom section relates to bit-specific actions.

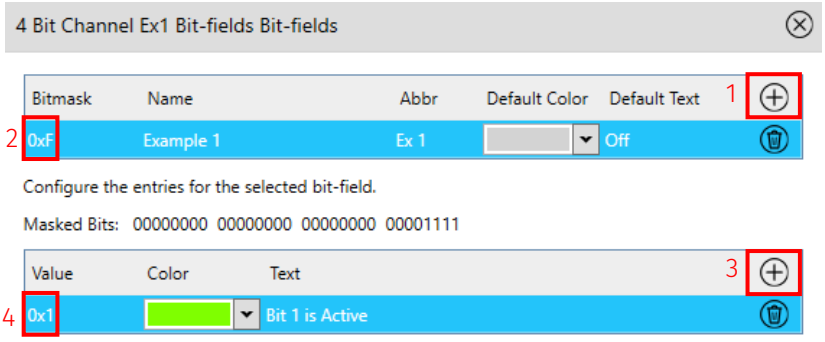



The top section shows the actual **Bitmask**, **Name**, **Abbr**(eviation), **Default Color**, and **Default Text**. The Bitmask is in Hexadecimal. In the bottom section there is each entry for the specified bits. There are two ways to configure Bit-fields and each depend on which view you want to see in live data (see examples below).

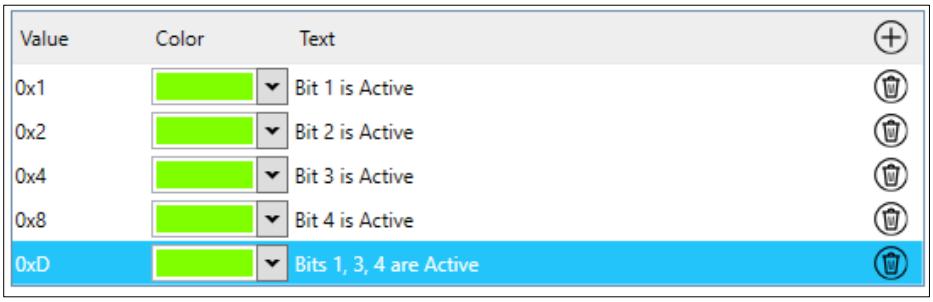
Example 1

In this example, a 4 Bit-masked channel is created, consisting of 1 Bitmask and 5 entries. Click the **+** button (1) to create a Bitmask and change it to the desired value (here 0xF (2)). Click the lower + button to add a bit entry (3). The value (4) needs to equal the bit value being requested.

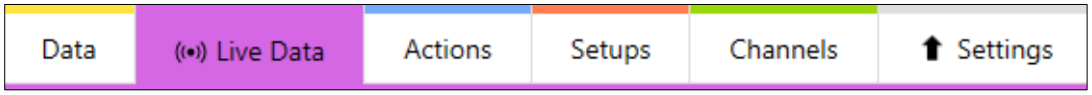
**Note:** The value needs to be in Hexadecimal. The first entry added is active when bit 1 in the channel is 1.



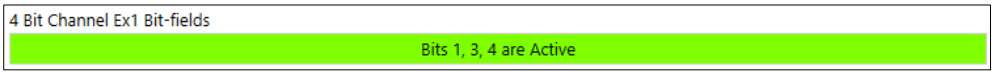
Four more entries are added. Value 0x2 activates when Bit 2 is active. Value 0x4 activates when Bit 3 is Active. Value 0x8 activates when Bit 4 is active, and Value 0xD activates when Bits 1, 3, 4 are Active.




You can view the output of this channel on the **Live Data** tab.



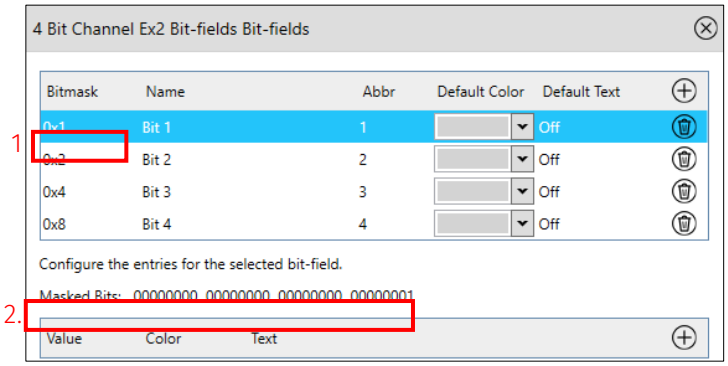

Add the channel 4 Bit Channel Ex1 Bit-fields (from above) to the **Live Data** page. The channel is displayed as seen below. The default value of this channel is set to 13 (0xD). Since there is no data coming across, the value reverts to the default.



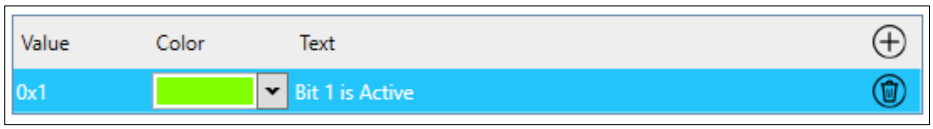

Example 2

In this example, a 4 Bit-masked channel is created. There are 4 Bitmasks and 1 entry per Bitmask. Create 4 Bitmasks, but make each of them specific to each bit. The first mask (1) has a value of 0x1. This creates a mask for bit 1. (2)

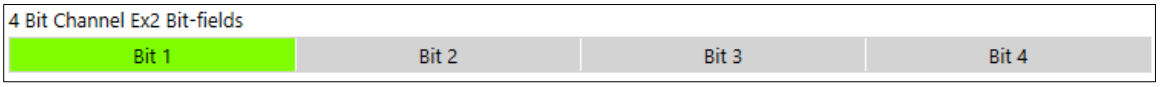
**Note**: No Bitmasks can overlap.



For each mask, you must dd an entry that has the same value as the mask. For the first mask, an entry with the value of 0x1 is added.




Do this for each corresponding Bitmask. Go to the **Live Data** tab to view the channel. The channel is displayed as seen below. This method of Bit-masking is useful when several bits in a channel have independent importance. The default value of this channel is set to 1 (0x1). Since there is no data coming across, the value reverts to the default.




**Implement multiplexed CAN channels**

You can use multiplexed CAN Channels to compress additional data into a single CAN packet/stream CAN packet.

To create a Multiplexed CAN Encode, two math channels are needed to locate and associate the CAN Channels in the packet. The first math channel is the Indexor. To create a math channel, click on the **Math Channels** node.

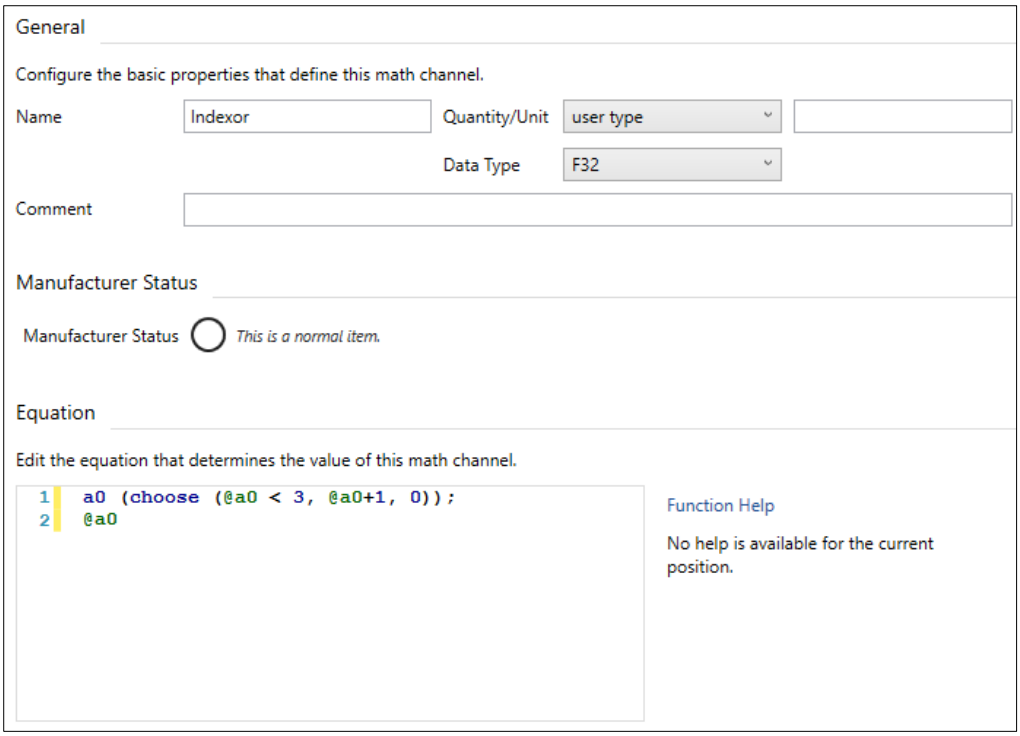
Click the **+** button (1) at the top left to create a new math channel. By default, this is added to the <Default> group of channels. To add channels to a group, highlight the required channels, click 'add group' (2), and name the group.


The first math channel is named Indexor. This is a counter and records the number of channels being multiplexed. In this example, three channels are being multiplexed. The equation required is:

a0 (choose (@a0 < 3, @a0+1, 0));

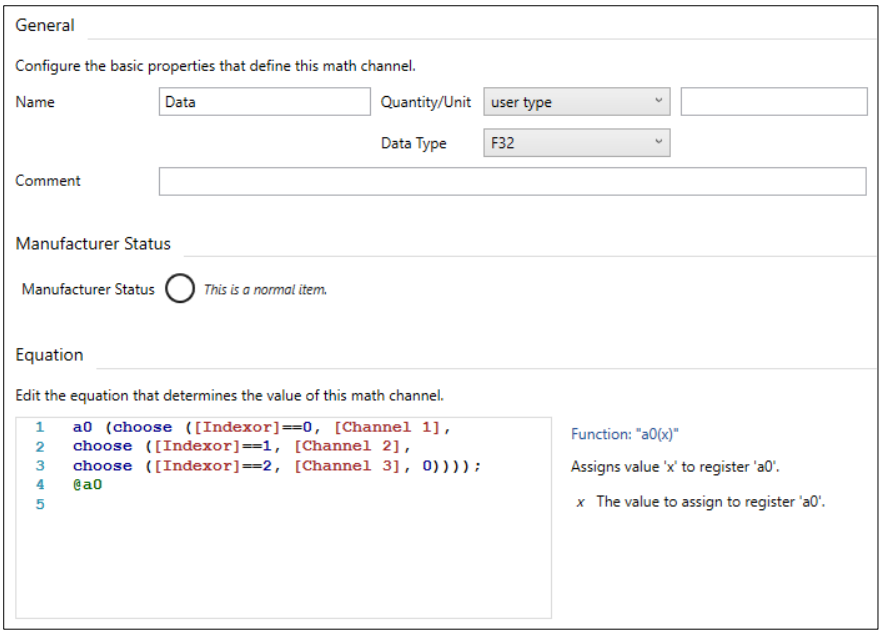
@a0

Set the rate to 100Hz and this channel counts to 2 from 0 at 100 increments per second.



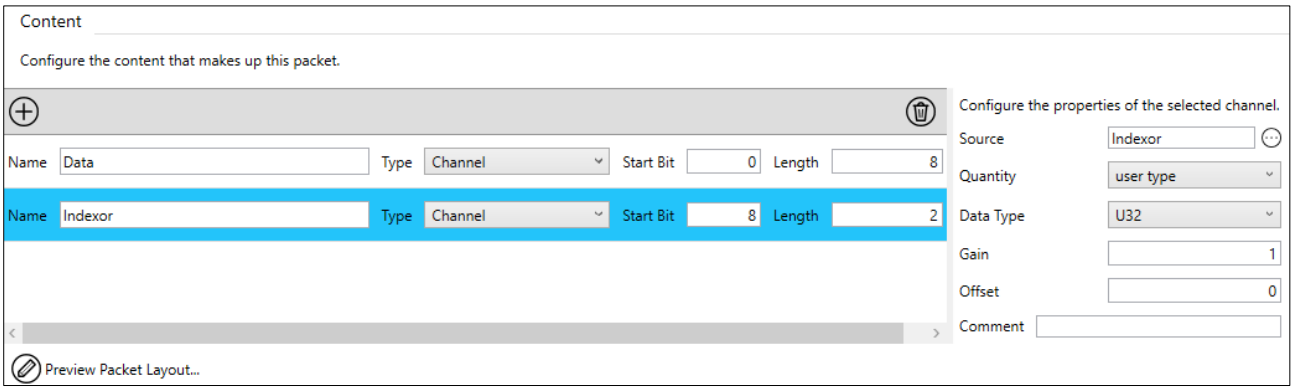

The second channel needed is used to correlate the value from the Indexor to the channel being sent. In the example code below, when the Indexor is equal to 0, the output is Channel 1. When it is 1, the output is Channel 2, and so.

```
a0 (choose ([Indexor]==0, [Channel 1],
choose ([Indexor]==1, [Channel 2],
choose ([Indexor]==2, [Channel 3], 0))));
@a0
```



Set the rate to 'Inherit rate from Indexor' and this math channel uses the same rate as the channel Indexor. The channels that are requested need to match the rate of the Indexor/Data channels.

Create a new CAN Stream Encode and add two channels to the packet. The source for the channels is the two math channels created above. In this example, data starts at 0 and is 8 bits long. The Indexor starts at 8 and is 2 bits long. The **Length** needs to be long enough to support the number of states. If the number of states is 3, then the length needs to be 2 bits (3 in binary is 11).

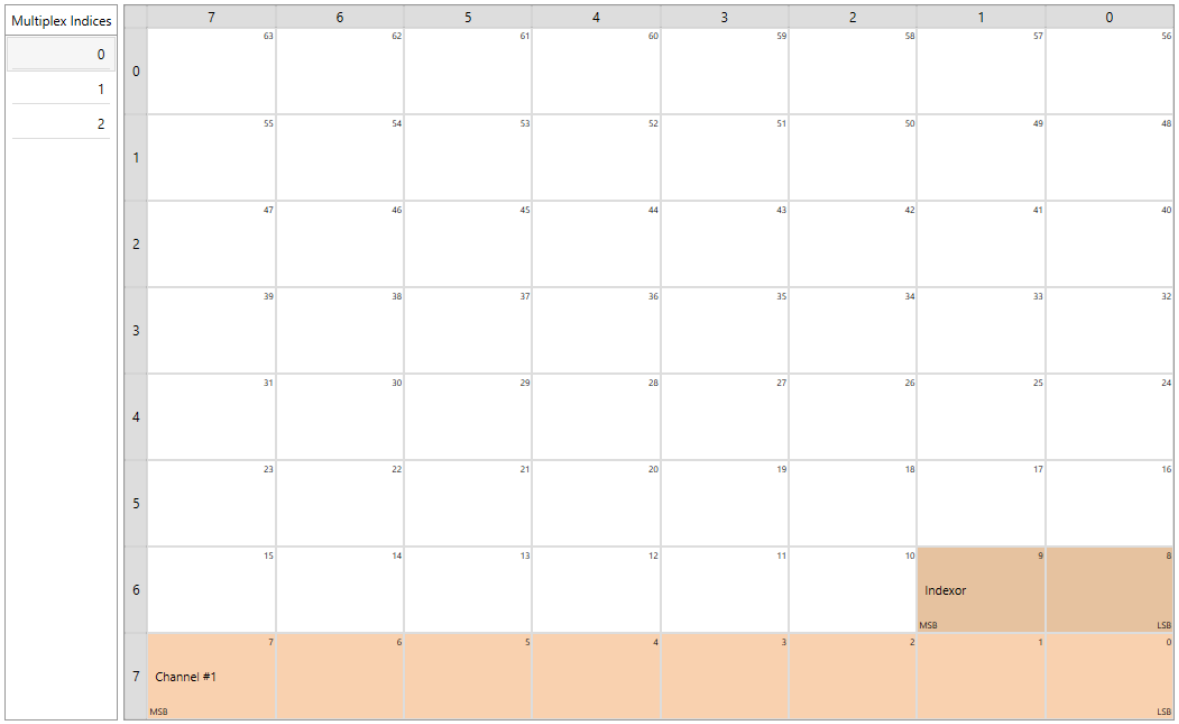



When this is done, the channels added to the Data math channel are transmitted across CAN in a multiplexed message.

To create a Multiplexed CAN Decode, create a CAN packet and add an Indexor channel. Under channel type, select 'Indexor'. To the right, enter the **Number of States**. For this example, this 3. Select the **Start Bit** and the **Length**.

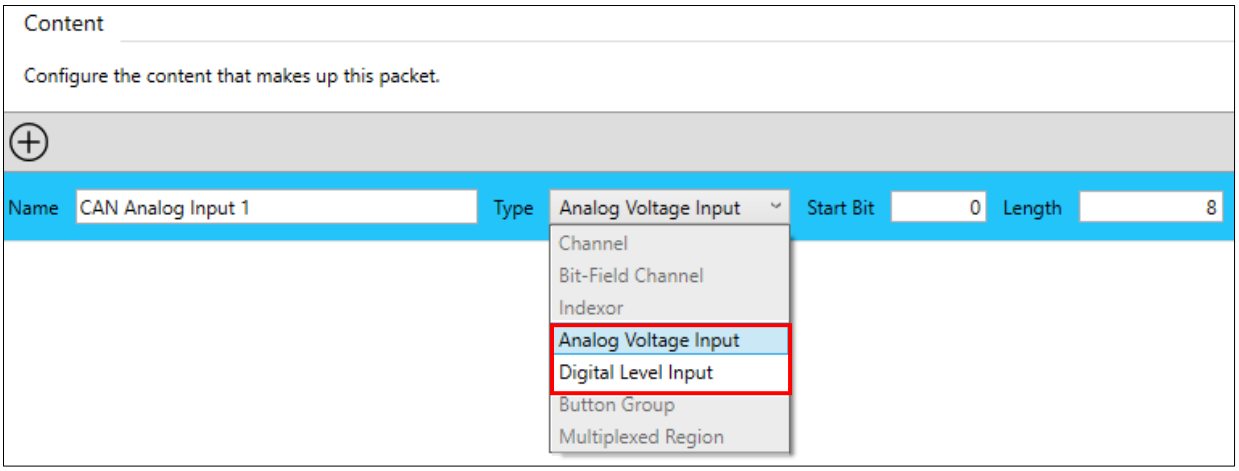
Next add the required channels. Create a new channel and set its type as 'Multiplexed Region'. In this example, add Channel #1, Channel #2, and Channel #3. The **Start Bit** and **Length** are the same for all three. Each channel will have its own Mux Index (0-2).

Click on **Preview Packet Layout** in the **Channels** section to view the packet layout. On the left, the Mux Indices are displayed. Click on an index to display each Mux and the layout for that packet.

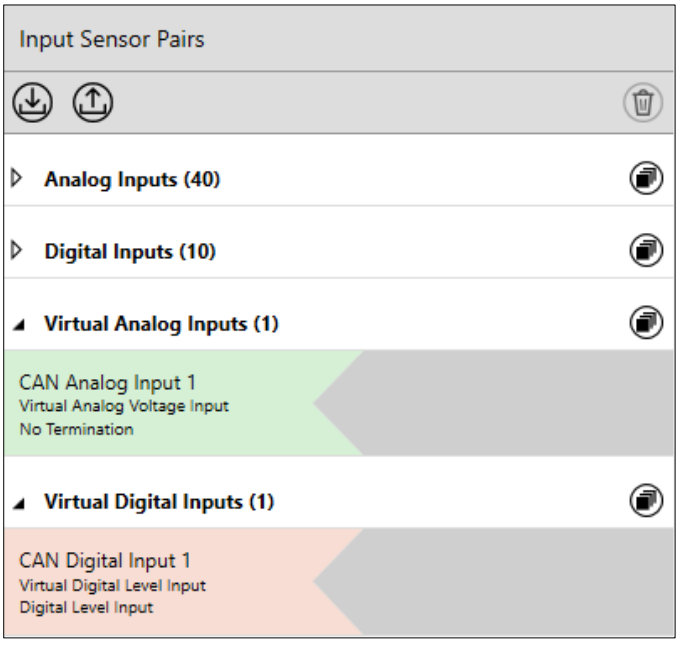


### Virtual analog and digital inputs

You can use CAN messages as virtual analog and digital sensor inputs. In other words, when the message within a packet is configured as either an 'Analog Voltage Input' or 'Digital Level Input' then the value transmitted over CAN is interpreted as an analog or digital input, respectively.

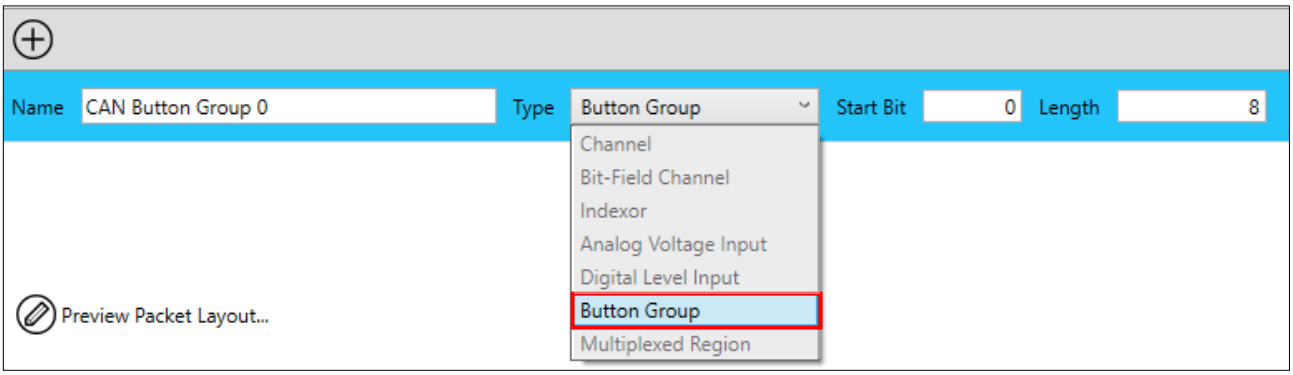



Once the CAN message is configured as either an 'Analog Voltage Input' or 'Digital Level Input', they appear as a virtual input on the Sensors node. You can then configure the virtual sensor and calibrate it like a standard analog or digital input (see Setups – Sensors).

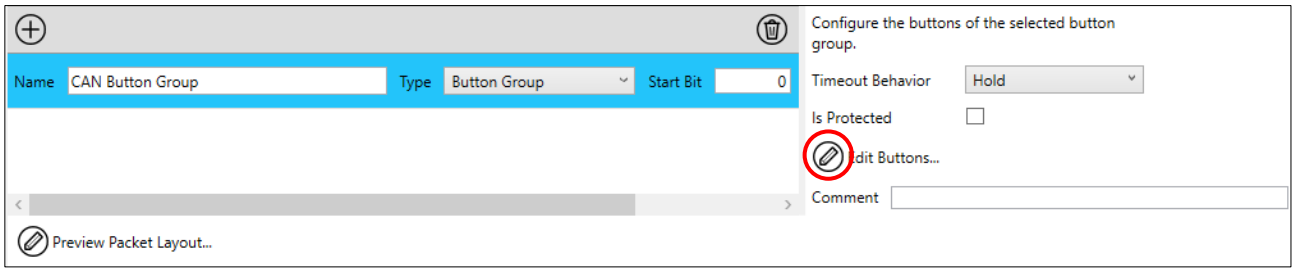


### Buttons groups

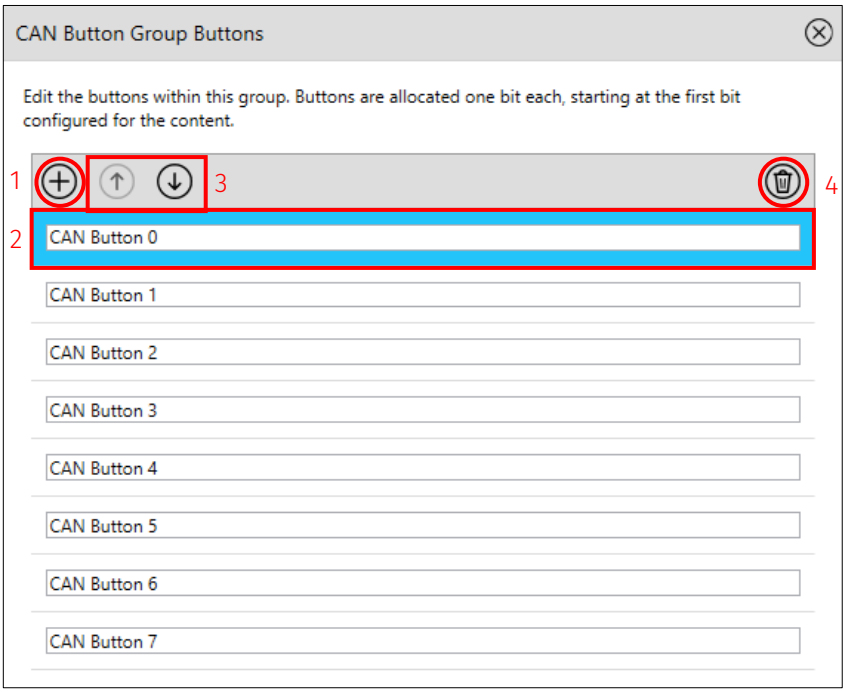
You can also configure CAN messages to be interpreted as button presses. For example, bits within an 8-bit message can be configured to act as different button press types (for example clicked, click latched, held, and so on).




Once a CAN message is configured to be a 'Button Group', click **Edit Buttons...** to add buttons.




You can then use the + button (1) to add button inputs and name them in the text box (2.) Use the 'reorder arrows' (3) to change the button order, Use the 'bin' tool to delete buttons (4).




Refer to Setups – Buttons for more information about how to configure a CAN button.

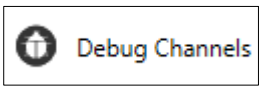
## 36.1. CAN CHP2 debug channels

| | |
|---|---|
| <CAN Device Name>_RxPackets | Increments by one when a packet received is read (that is, at the receive rate) |
| <CAN Device Name>_TxPackets | Increments by one when a packet being sent is written (that is. at the transmit rate) |
| <CAN Device Name>_BusOffCount | Increments each time a device is detected in the bus off state and is restarted |
| <CAN Device Name>_BusState | Bit encoded see below:<br>• Bit 0 : Error State - Set when the device is ERROR_ACTIVE (as per the CAN specification)<br>• Bit 1 : Bus Warn - Set when bus is heavily disturbed (as per CAN spec one of the CAN Error counters exceeds 96)<br>• Bit 2 : Bus Off - Module is in the bus off state<br>Cleared when the device is ERROR_PASSIVE (as per the CAN specification) |

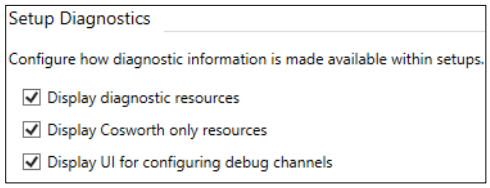
| | |
|---|---|
| <CAN Device Name>_LastErrorCode | Numerically encoded see below:<br><br>• Value = 0 - Received or transmitted a packet successfully<br>• Value = 1 - Bit Stuff Error – More than 5 equal bits in sequence received where this is not allowed<br>• Value = 2 - Format Error - A fixed format part of a received frame has the wrong format<br>• Value = 3 - No ACK – A frame transmitted has not been acknowledged by another node<br>• Value = 4 - Dominant Bit Error – During the transmission of a packet (except for the arbitration field) the device wanted to send a recessive level, but the monitored bus level was dominant<br>• Value = 5 - Recessive Bit Error – During the transmission of a packet, or acknowledge, or active error flag, or overload flag the device wanted to send a dominant bit, but the monitored bus level was recessive<br>• Value = 6 - Received CRC error – The CRC of the packet received does not match the calculated CRC<br>• Value = 7 - No CAN Bus errors detected since last update |

**Debug channels setup**

To enable these channels first create them in the setup. On the **Debug Channels** node, select **Advanced**.

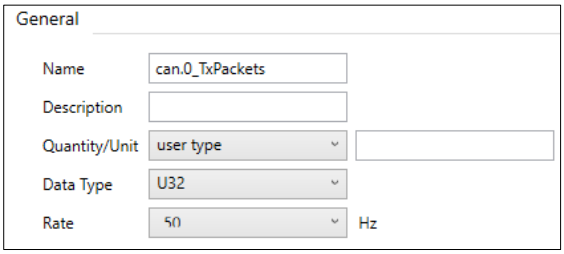



**Note**: If this node is not available on the setup, enable it on the **Settings** tab under **Setup Diagnostics**.

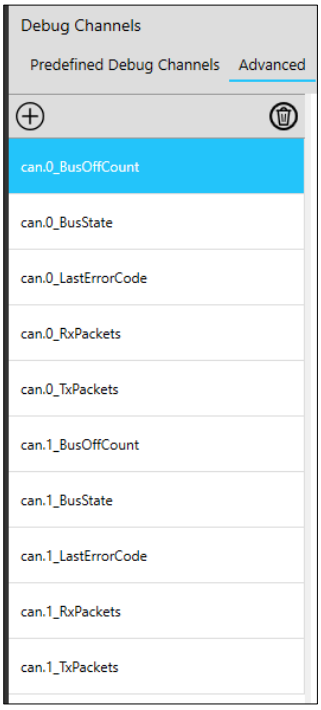



Click the **+** button to create a channel. The **Name** must match the name used in the above table (<CAN Device Name> = can.0 (for CAN port 1), <CAN Device Name> = can.1 (for CAN port 2), and so on).

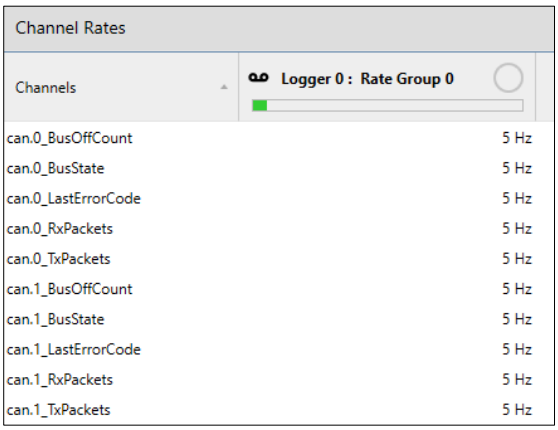
Set **Quantity/User** to ‘user type’, **Data Type** to’U32’, and **Rate** to ‘5Hz’, as shown below.




For two CAN ports there should be these channels.




The channels are then available to log:




**Note**: When You create the channel, it might display as empty. To save the setup, close it, and then re-open it.

### Allowable rates

When multiplexing, only certain rates are allowed for the packet. This value is dependent on the number of multiplexing states. This table defines what is allowed in Toolset for all of Cosworth devices.

The chart below details allowable channel rates for multiplexed CAN streams. The individual channels must be sent at legal rates, defined by dividing the packet rate by the number of channels. Only certain integers are allowed for the channel rates.

Green is an allowable rate
Red is an illegal rate.

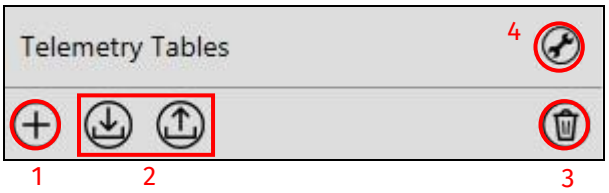
| Packet Rate | Number of Channels in Packet | | | | | | | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 |
| 1 | 1.00 | 0.50 | 0.33 | 0.25 | 0.20 | 0.17 | 0.14 | 0.13 | 0.11 | 0.10 | 0.09 | 0.08 | 0.08 | 0.07 | 0.07 | 0.06 | 0.06 | 0.06 | 0.05 | 0.05 |
| 2 | 2.00 | 1.00 | 0.67 | 0.50 | 0.40 | 0.33 | 0.29 | 0.25 | 0.22 | 0.20 | 0.18 | 0.17 | 0.15 | 0.14 | 0.13 | 0.13 | 0.12 | 0.11 | 0.11 | 0.10 |
| 5 | 5.00 | 2.50 | 1.67 | 1.25 | 1.00 | 0.83 | 0.71 | 0.63 | 0.56 | 0.50 | 0.45 | 0.42 | 0.38 | 0.36 | 0.33 | 0.31 | 0.29 | 0.28 | 0.26 | 0.25 |
| 10 | 10.00 | 5.00 | 3.33 | 2.50 | 2.00 | 1.67 | 1.43 | 1.25 | 1.11 | 1.00 | 0.91 | 0.83 | 0.77 | 0.71 | 0.67 | 0.63 | 0.59 | 0.56 | 0.53 | 0.50 |
| 20 | 20.00 | 10.00 | 6.67 | 5.00 | 4.00 | 3.33 | 2.86 | 2.50 | 2.22 | 2.00 | 1.82 | 1.67 | 1.54 | 1.43 | 1.33 | 1.25 | 1.18 | 1.11 | 1.05 | 1.00 |
| 50 | 50.00 | 25.00 | 16.67 | 12.50 | 10.00 | 8.33 | 7.14 | 6.25 | 5.56 | 5.00 | 4.55 | 4.17 | 3.85 | 3.57 | 3.33 | 3.13 | 2.94 | 2.78 | 2.63 | 2.50 |
| 100 | 100.00 | 50.00 | 33.33 | 25.00 | 20.00 | 16.67 | 14.29 | 12.50 | 11.11 | 10.00 | 9.09 | 8.33 | 7.69 | 7.14 | 6.67 | 6.25 | 5.88 | 5.56 | 5.26 | 5.00 |
| 200 | 200.00 | 100.00 | 66.67 | 50.00 | 40.00 | 33.33 | 28.57 | 25.00 | 22.22 | 20.00 | 18.18 | 16.67 | 15.38 | 14.29 | 13.33 | 12.50 | 11.76 | 11.11 | 10.53 | 10.00 |
| 500 | 500.00 | 250.00 | 166.67 | 125.00 | 100.00 | 83.33 | 71.43 | 62.50 | 55.56 | 50.00 | 45.45 | 41.67 | 38.46 | 35.71 | 33.33 | 31.25 | 29.41 | 27.78 | 26.32 | 25.00 |
| 1000 | 1000.00 | 500.00 | 333.33 | 250.00 | 200.00 | 166.67 | 142.86 | 125.00 | 111.11 | 100.00 | 90.91 | 83.33 | 76.92 | 71.43 | 66.67 | 62.50 | 58.82 | 55.56 | 52.63 | 50.00 |

# 37. Telemetry

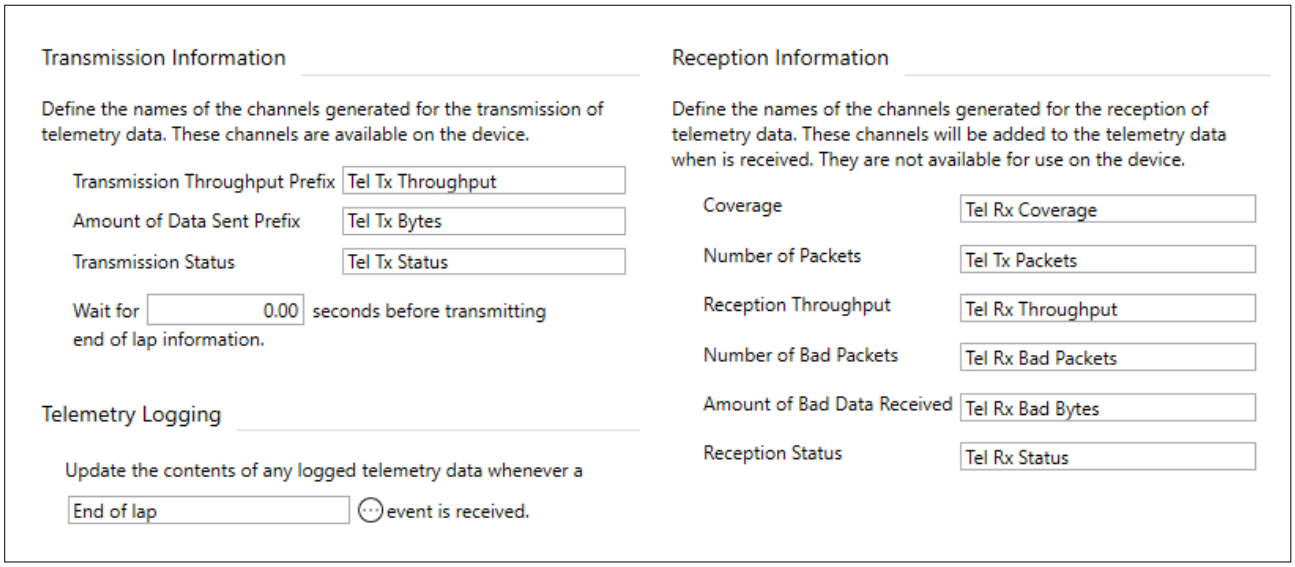
Within Toolset you can configure a telemetry stream to allow a proper connection to the car.

## 37.1. Configure device telemetry transmit (Tx)

For telemetry to be supported, you must enable it via the **Telemetry** node. Click **+** to add a new telemetry table. You can use the import and export tools to import and export existing telemetry tables between setups (2). Use the 'bin' tool to delete unwanted telemetry tables (3). When a telemetry table is added, channels are automatically generated to report the telemetry transmission status. Click the 'wrench' tool to configure these generated channels (4).

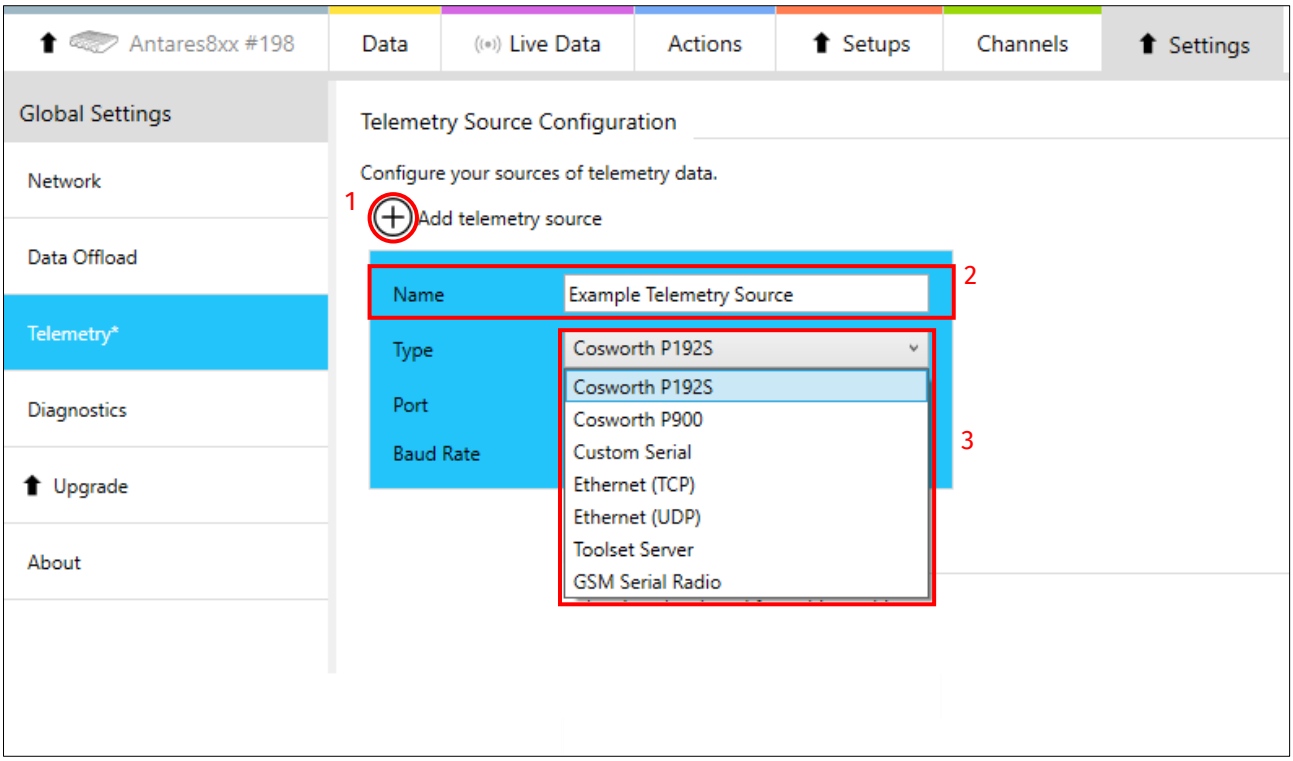



This allows you to configure the automatically generated transmission and reception status channels. Here you can modify the interval before EOL information is transmitted (this is useful if you know there is poor reception at the EOL, allowing you to postpone the transmission until you are in a better reception area.) The EOL repeat count and interval are fixed at 4s and 0.5s respectively.

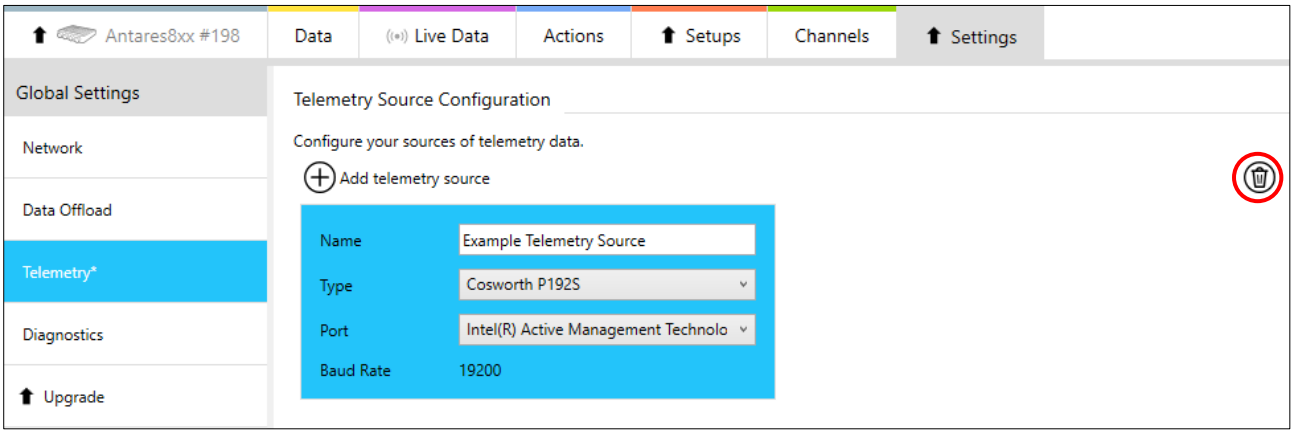


### Configure PC telemetry receive (Rx)

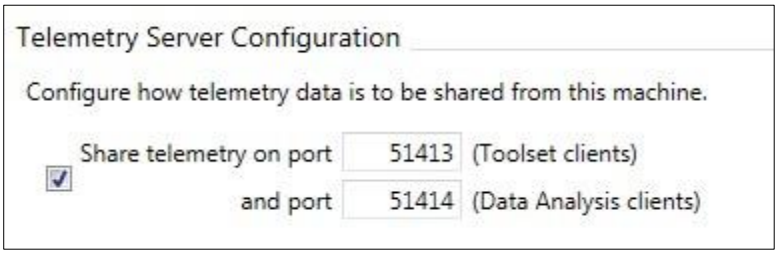
You must define the source used, together with where the data is downloaded via **Settings > Telemetry**. Click **Add telemetry source** (1), rename the telemetry source (2) and select the telemetry source type from the dropdown menu (3).



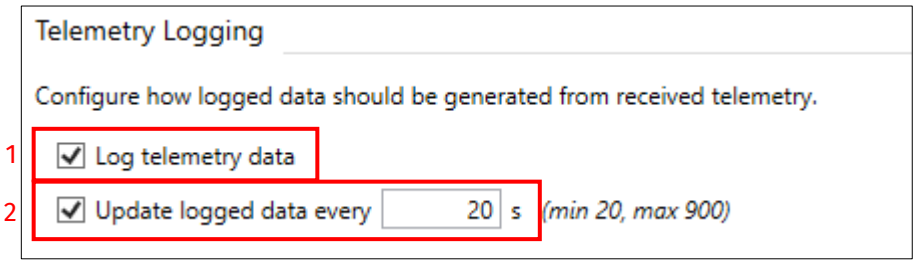

When a source type is added, you are prompted to configure the telemetry port settings. Some telemetry source type settings are predefined, such as the Cosworth P192. You can configure custom serial and Ethernet ports. Select the required port to connect to your pit stand receiver. Select a telemetry source, and then click the 'bin' icon to delete it.



If you need to allow others to connect to your machine to view telemetry, select the **Share telemetry** option in the **Telemetry Sever Configuration** section. Keep the default values for both ports.



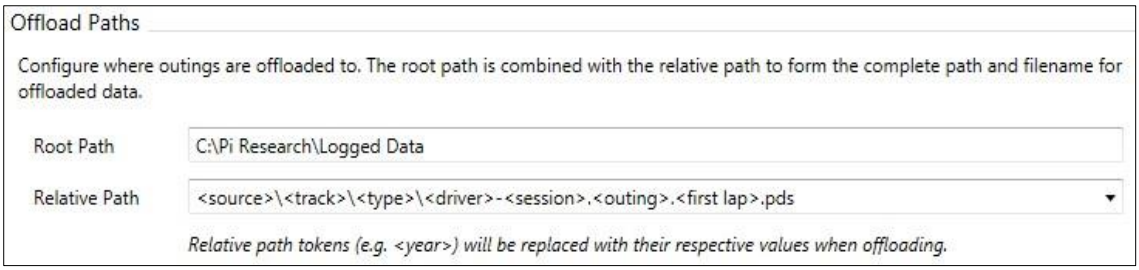

You can add several telemetry sources with a single machine as the server PC (see **Error! Reference source not found.**). You can choose to log the data that is received from the telemetry stream as well as the frequency at which it is updated. Select the **Update logged data every...** option and enter a value between 20 and 900 seconds.




Once sources are configured you must save the configuration. Click **Save**. You can also select **Reset** to return to the default configuration.

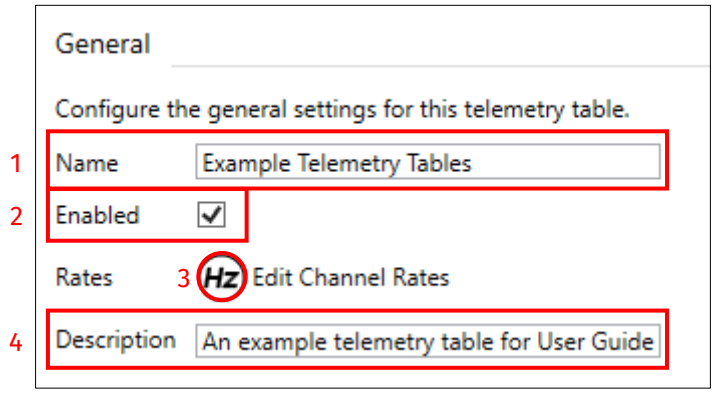

**Configure logged telemetry data**

To change the relative path of logged telemetry data, select the **Settings** tab and click **Data Offload**. Look in the **Offload Paths** section to find the **Relative Path**. At the beginning of the path add '<source>\'. This creates a specific folder for telemetry data and logged data. For more information about saving logged data, see the Settings guide.



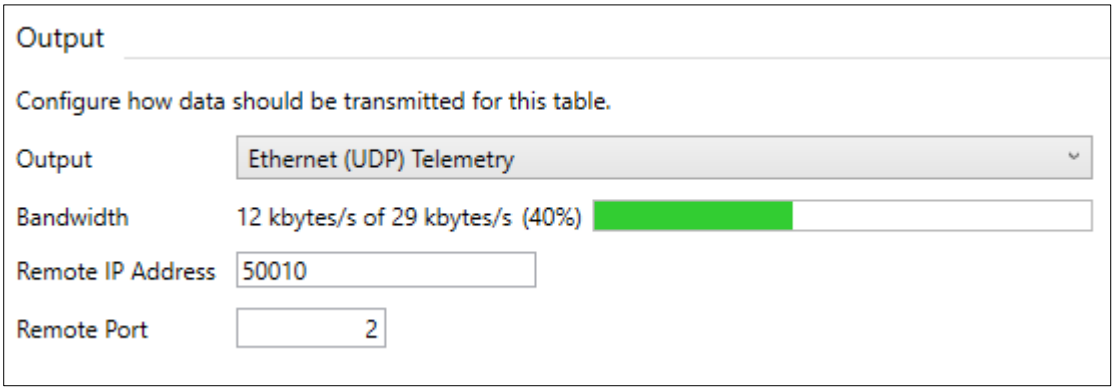

Once a telemetry table is added on the **Telemetry** node of the device setup, enter a name for the table (1) and select **Enabled** (2). Deselect this option to disable a table without deleting it.

**Note**: You must configure the channel rates for the channels within the telemetry table to allow them to be transmitted and received. Click **Edit Channel Rates** to navigate to the **Channel Rates** node (3). You can enter a description of the table (4). Multiple telemetry tables can be added to a setup.



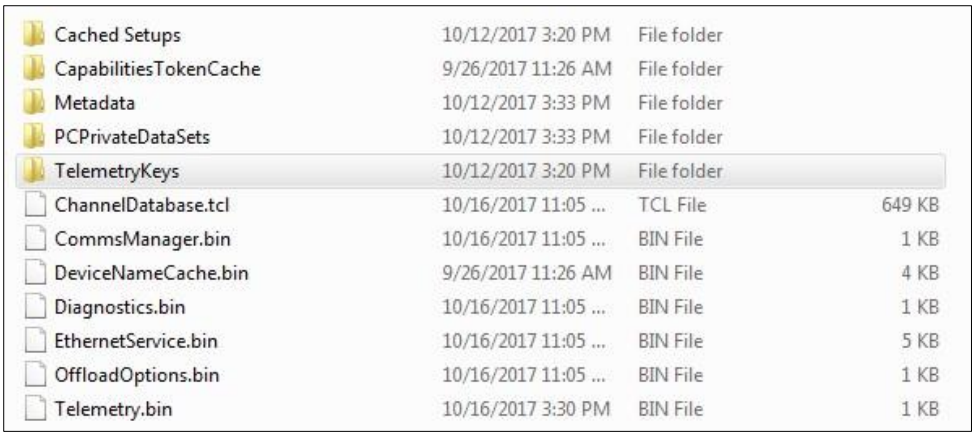

You can then configure the type of telemetry from the **Output** dropdown menu. If a Cosworth serial telemetry system is selected, you only need to configure the serial port. If custom serial telemetry is selected, then additional information such as the transmission baud rate and data bits need to be configured. Cosworth devices also support ethernet (UDP) telemetry, which you can configure using the remote IP address and remote port.

The consumed bandwidth of the telemetry table based on the usage of the telemetry logging table is also displayed.




Once the setup is sent to the device, the telemetry key is written to the Toolset app data folder:

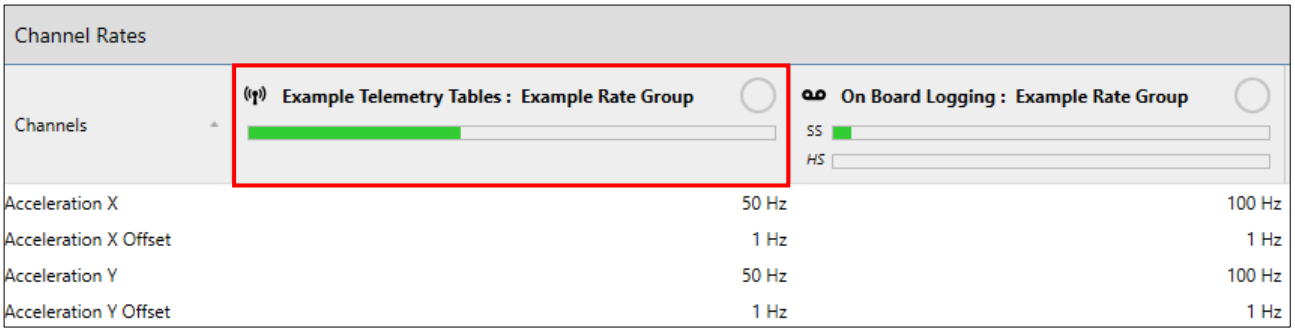
C:Users**[YourAccount]**\AppData\Roaming\Cosworth\Toolset\[Toolset Version]"\TelemetryKeys.




### Configure telemetry channel logging rates

After you configure your PC telemetry Rx and Toolset telemetry Tx, you must assign rates to the channels transmitted over telemetry. This is done on the **Channel Rates** node.

The telemetry table is identified by its user-defined name and the 'radio' icon, rather than the device on-board logging 'tape' icon.




### Watch telemetry channels

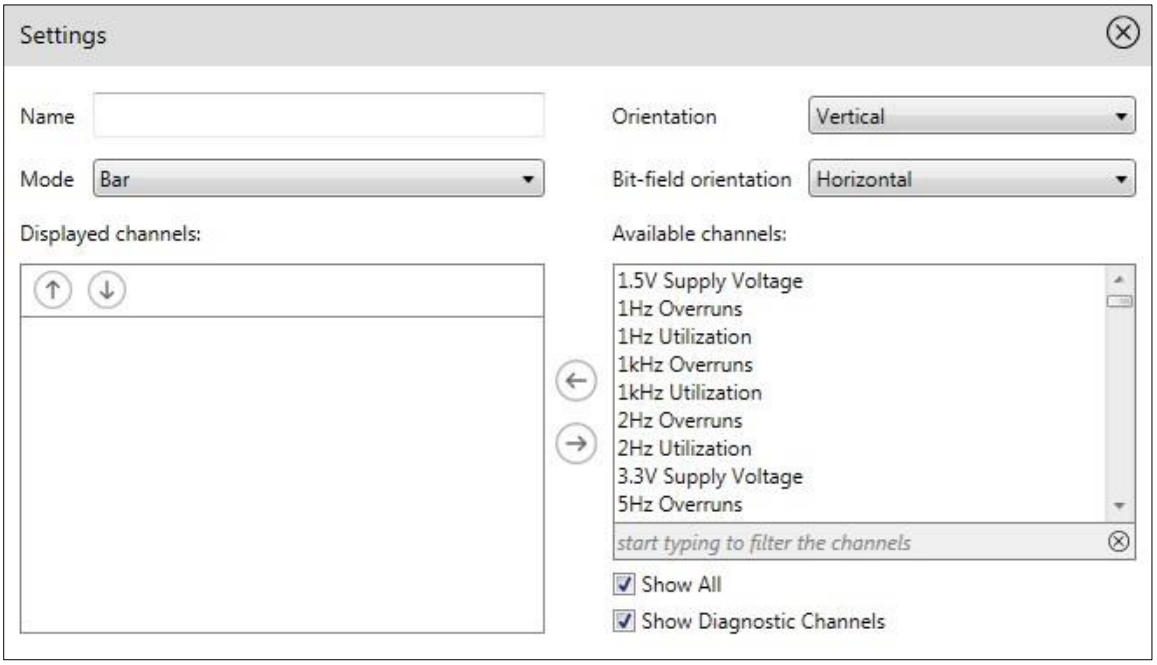
To watch telemetry channels, select the **Live Data** tab, and then **Data**. Click **Sources** to check the telemetry connection and information.

**Note**: the 'radio icon' next to **Live Data** changes from grey to black when there is a working telemetry connection.

Click the + button to add and populate a page. Click on a page name to rename the page. Click the 'pencil' icon to display more buttons (from left to right):

- Add Channel List
- Add Map Control
- Add Tabular Outing Report.

Use the 'Add Channel List' option to add telemetry channels. When you add a page, click the 'wrench' icon to add required channels by selecting them from the **Available channels:** and moving them to the **Displayed channels:** section.



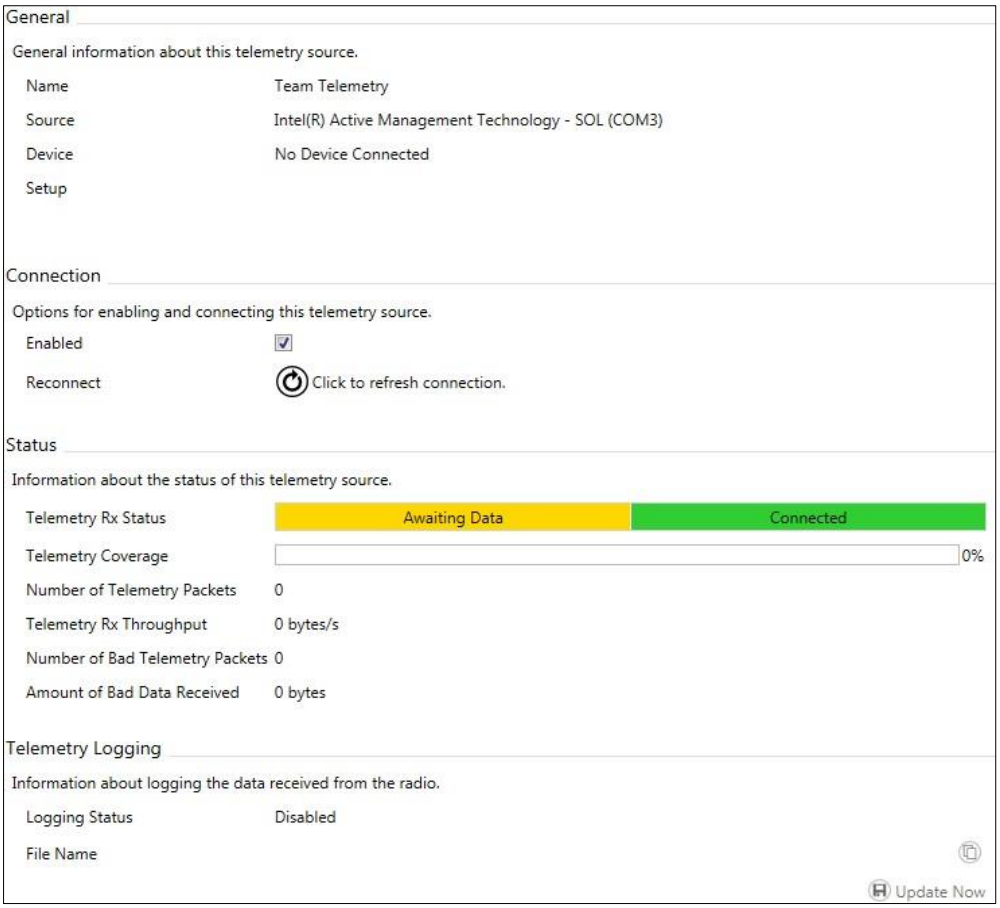

In the **Sources** section you can check the state of the telemetry link and view the target directory for the logged telemetry data.

In the **General** section you can view the name of the source, the device, and the setup.

In the **Connection** section you can see if your source is enabled and have the option to refresh the connection.

In the **Status** section you can view status of the telemetry connection and information about the connection.

In the **Telemetry Logging** section, you can view the status and destination of the logged data.



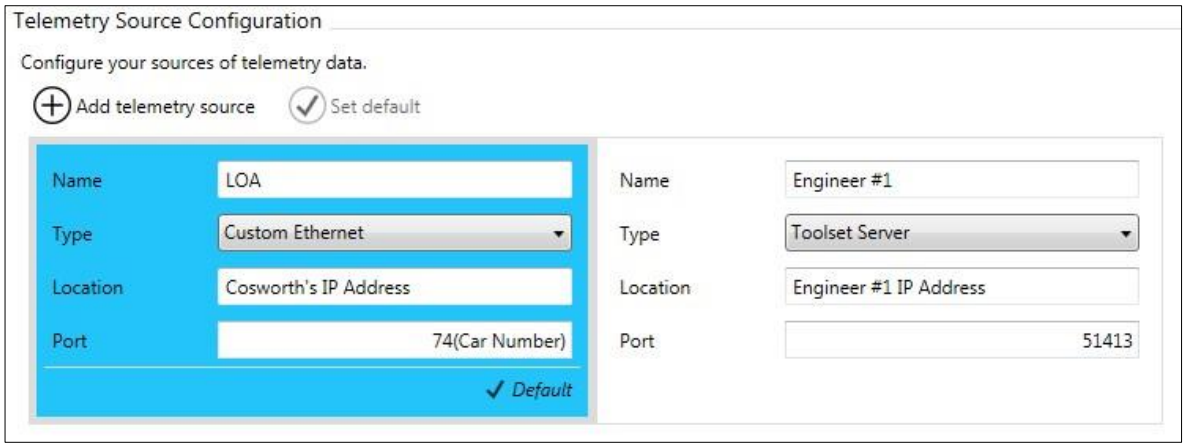

**Use Cosworth Live On Air (LOA) as a telemetry source**

It is recommended that you use the pit stand as a common connection point for all connected users on Toolset. Point the pit stand to Cosworth to establish connection to the stream, configure it as a Toolset server connection, and then point the stream to each engineer connected and using telemetry. This allows connected users to receive telemetry through the desktop.

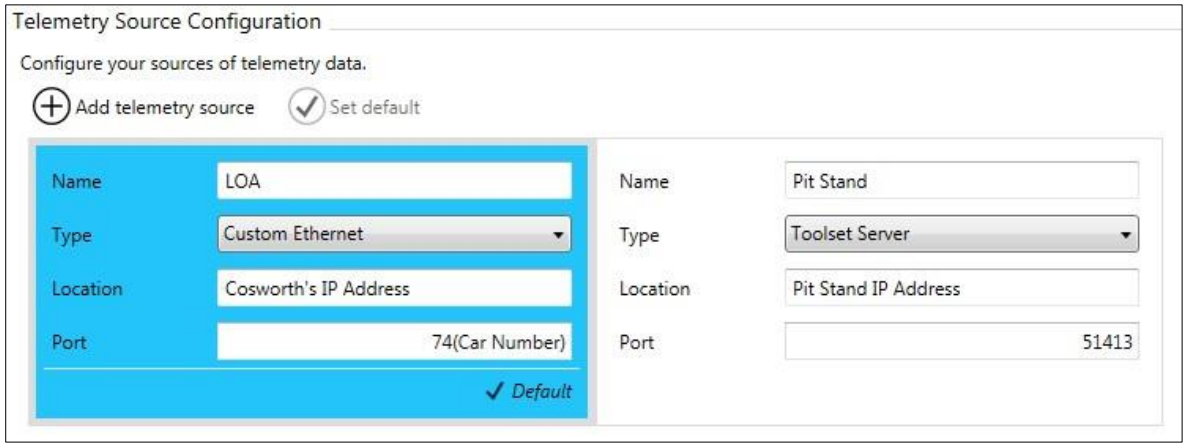
With the pit stand configured, the DAG points to Cosworth to establish the initial telemetry connection (if a telemetry key is available). The DAG then connects to the pit stand as a Toolset server. This distributes the telemetry key to the pit stand and provides it to connected engineers to avoid loss in telemetry if the DAG disconnects. Finally, each engineer can point their device to Cosworth directly to make sure that they also have a direct connection to the stream in case other devices disconnect or fail.

On the pit stand desktop, go to **Settings > Telemetry**. Click **Add telemetry source** and select 'Custom Ethernet' from the **Type** dropdown box. The **Location** is 'Cosworth's IP Address', and the **Port** is '74', followed by the 'Car Number'.

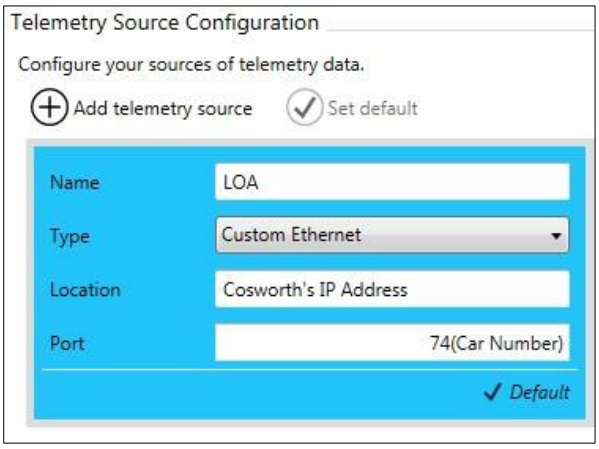
Next, add a new source for each engineer on the team who use telemetry. Each of these sources is a Toolset server connection. The **Location** for each engineer is their IP address, and the **Port** is '51413'. This completes the desktop setup and the configuration is saved for future use.




The DAG also needs a custom ethernet connection to Cosworth. The **Location** is 'Cosworth's IP Address' and the **Port** is '74', followed by the 'Car Number'. The DAG also points a Toolset server connection to the pit stand desktop. The **Location** is the 'Desktop's IP Address' and the **Port** is '51413 (Default)'. This pushes the telemetry key to the desktop, and then to each engineer.

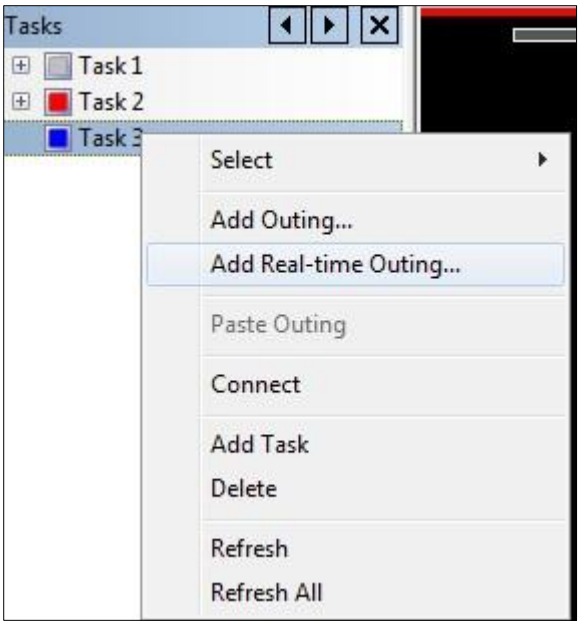



Each engineer only needs a 'Custom Ethernet' connection to Cosworth. The Location is 'Cosworth's IP Address' and the **Port** is '74', followed by the 'Car Number'. There is no need for a server connection because the pit stand pushes the key to each engineer.

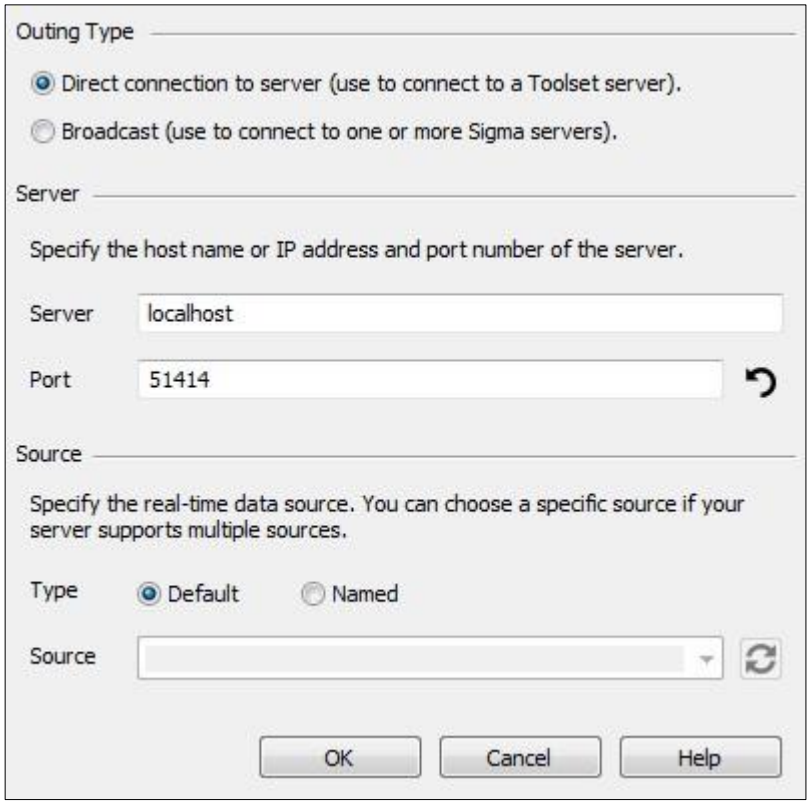


**Add a real-time outing to Toolbox**

With an active telemetry connection open Toolbox and create a new task. Right-click the task and select **Add Real-time Outing**.




If you have telemetry via Toolset, the default settings will work. The **Server** is your own machine (localhost) and the **Port** is 51414 (the default).




Once this is complete, a real-time outing is created and Toolbox telemetry established.

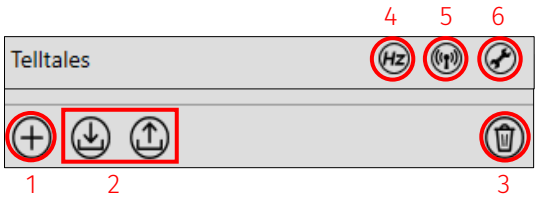
# 38. Telltales overview

Telltales are channel statistics that are published at end of a lap over telemetry. You can add telltales to the **Tabular Outing Report** display on the **Live Data** page (see Live Data).

**Note**: To view the telltale over telemetry you must add the output channel named after the telltale to the telemetry table (see Telemetry).

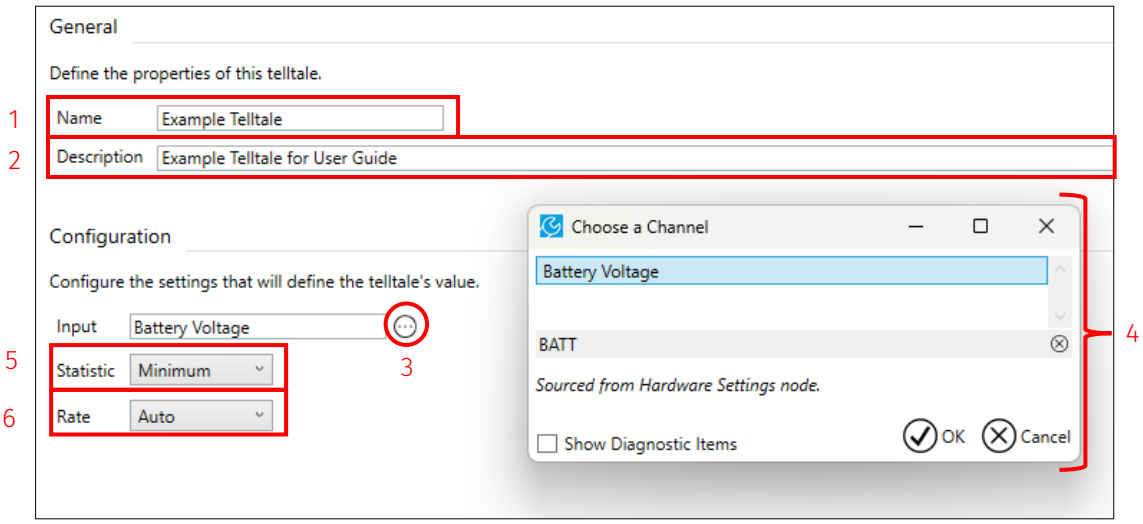
## 38.1. Add a telltale

Click the **+** tool to add a new telltale (1). You can import and export telltales between existing setups with the 'import' and 'export tools' (2). Use the 'bin' tool to delete telltales (3). Use the channel rates shortcut (4) and the telemetry shortcut (5) to quickly navigate to the **Channel Rates** and **Telemetry** nodes. Use the 'wrench' tool (6) to configure telltale reset events.




## 38.2. Configure a telltale

Add a telltale and enter a name and optional description (1/2). Click the 'browse' icon (3) to select the input channel from the available channel list (4). Select the channel statistic (minimum, maximum, mean, difference, or end) from the **Statistic** dropdown menu (5) and the telltale calculation rate from the **Rate** dropdown menu (6). Select 'Auto' to use the rate from the input channel.



# 39. Wheelspeeds overview

Distance is one of the most important channels used in Toolbox and Toolset and is fundamental for features like Qualifying Mode. The value of distance is derived from the integral of speed. Therefore, the setup of wheelspeed strategies is critical to make sure that the speed and distance values generated are consistent and correct.

The Wheelspeed node is used to configure the wheelspeed inputs and the speed and distance outputs.

## 39.1. Select wheelspeed input type

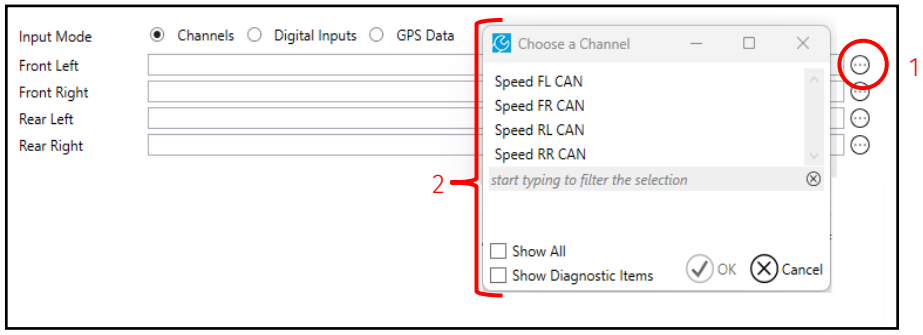
First, configure the wheelspeed inputs. Select the source of speed for each wheel from either a Channel or Digital Input, or vehicle speed from GPS data.

- Channel – Use for a channel with units of 'speed', such as a channel being received over CAN from an ECU or other device.
- Digital Inputs – Use if rotational sensors or DF11i wheelspeed sensors are configured on the Sensors node.
- GPS Data – Use if speed is being decoded from GPS data sourced from the NMEA 0183 Decode node.

## 39.2. Channel inputs

### Configure inputs

If you use channel inputs, click the 'browse' tool to select the channel inputs (1), and then select the required channel from the menu (2).



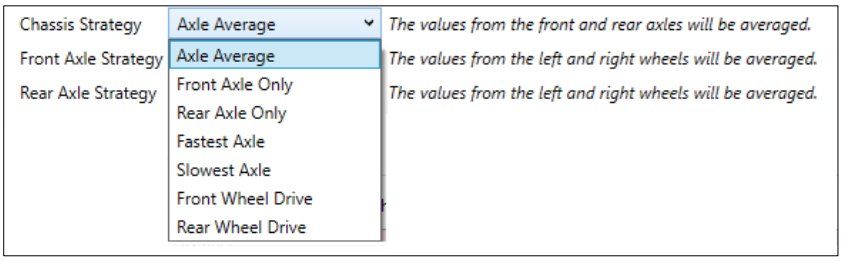

Note: – You must configure the channel with units of 'speed'. Channels with other units are not available to select.

### Processing

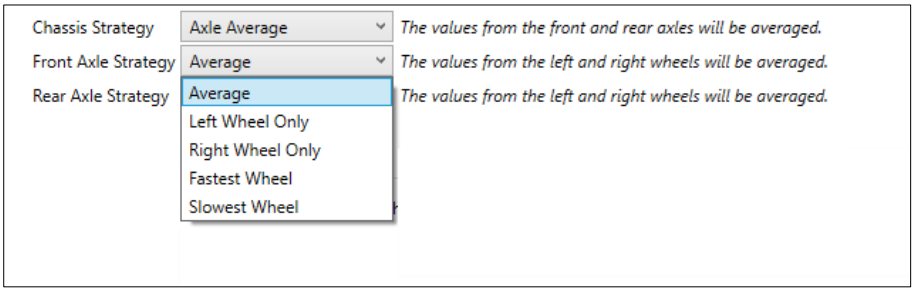
Next, select how the individual wheelspeed inputs are combined to calculate the vehicle speed and distance. Individual wheelspeeds are first combined to produce front and rear axle speeds. The front and rear axle speeds are combined to produce overall speed and distance channels.

See Wheelspeed strategy for more information about wheelspeed processing strategy.

Select the chassis strategy (the values from which the front and rear axles are averaged) from the **Chassis Strategy** dropdown menu.




Then select the front and rear axle strategies from the dropdown menus.



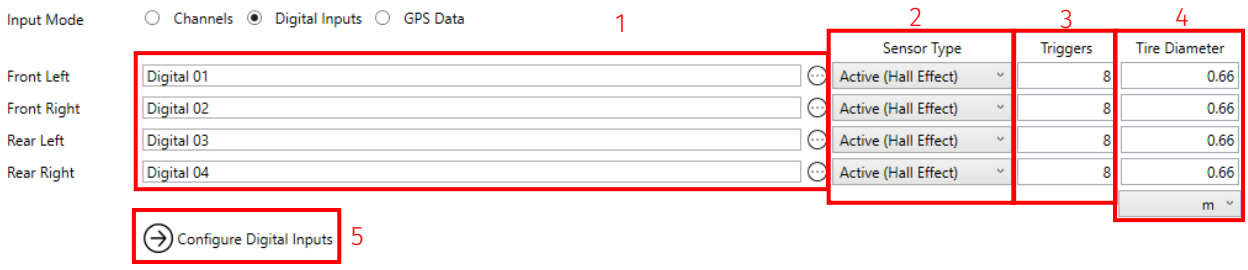

#### Generated channels

Enter a name for the distance and speed channels generated from the processed inputs in the text boxes.

## 39.3. Digital inputs

#### Configure inputs

If you use digital inputs, you must select the inputs for each wheel (1), specify the sensor type (2), the number of triggers per rotation (3), and the tire diameter (4). If digital inputs are not configured, you can use the shortcut to the **Hardware Settings** node (5) (see Hardware Settings).

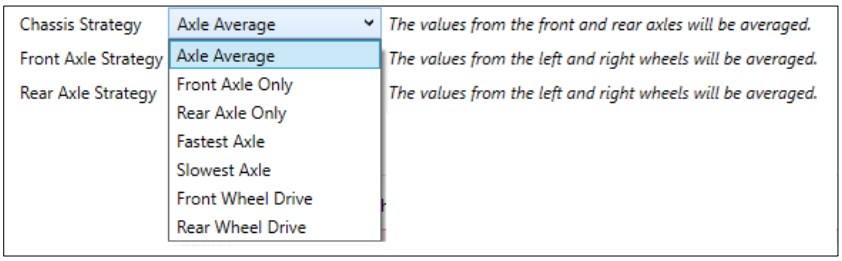



#### Processing

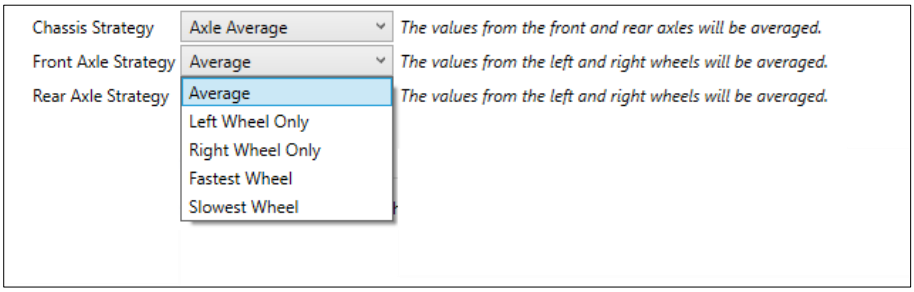
Next select how the individual wheelspeed inputs are combined to calculate the vehicle speed and distance. Individual wheel speeds are first combined to produce front and rear axle speeds. The front and rear axle speeds are combined to produce overall speed and distance channels.

See Wheelspeed strategy for more information about wheelspeed processing strategy.

Select the chassis strategy (the values from which the front and rear axles are averaged) from the **Chassis Strategy** dropdown menu.




Then select the front and rear axle strategies from the dropdown menus.



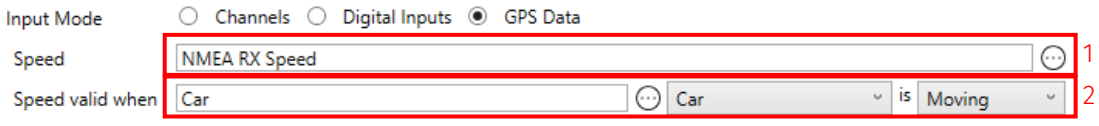

### Generated channels

You can then name the channels generated from the processed inputs. When you use digital inputs, there is greater functionality and, therefore, more generated channels. Enter names for the output channels for the wheel speed, angular velocity, triggers, and tire diameter of all wheel inputs.

## 39.4. GPS data

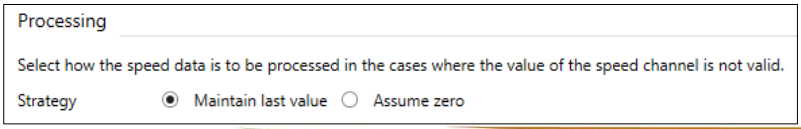
### Configure inputs

If you use GPS data, select the GPS speed channel (sourced from the NMEA 0183 Decode) (1) and configure the strategy for when the speed is valid (2).




### Processing

When the GPS signal is invalid, use the **Maintain last value** option to determine whether the speed value is held at the last known value, or select zero speed with the **Assume zero** option.



**Generated channels**

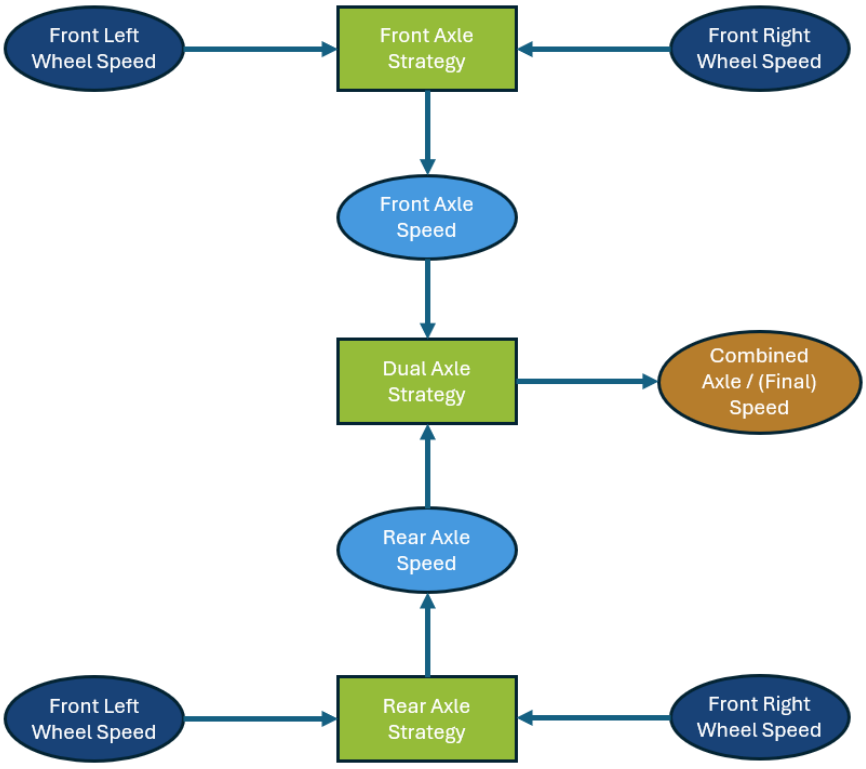
You can then name the distance and speed channels generated from the processed inputs.

## 39.5. Odometer

An odometer channel (a total distance channel stored in non-volatile memory) is also generated. Enter a name for the odometer (1). Click **Add Trip** to add a 'trip meter' channel (2 and add a name for the channel (3). You can also use a to reset the trip meter (see Buttons) (4). Use the 'bin' tool to delete a trip meter channel (5).


## 39.6. Wheelspeed strategy

The following flow chart shows how the wheel speed strategies are combined to output the final speed value. Ellipses represent a true, measured, speed and rectangular shapes represent numerical processing or a calculation.

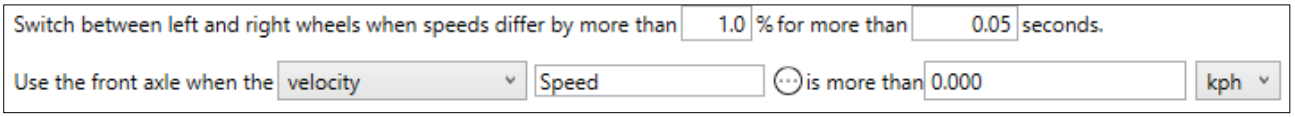


A table of the processing strategies available in Toolset is shown below:

| | Strategy | Description |
|---|---|---|
| Single axle strategy | Left Wheel | Uses the value of the axle left wheel only |
| | Right Wheel | Uses the value of the axle right wheel only |
| | Fastest Wheel | Uses the axle fastest wheel |
| | Slowest Wheel | Uses the axle slowest wheel |
| | Wheel Average | Uses the average of the two wheels |
| Dual axle strategy | Front Axle | Uses the value of the front axle only |
| | Rear Axle | Uses the value of the rear axle only |
| | Fastest Axle | Uses the fastest axle value |
| | Slowest Axle | Uses the slowest axle value |
| | Axle Average | Uses the average of the two axles |
| | Front Wheel Drive | Uses the front axle value under a specified condition (braking), then the rear axle value at all other times |
| | Rear Wheel Drive | Uses the rear axle value under a specified condition (braking), then the front axle value at all other times |

**Switch strategies**

Some older versions of setup metadata allow switching between strategies (for example, between the left and right wheel on an axle, or between the two axle speeds). In these versions of metadata there is a section to set the switching thresholds.




The percentage difference in speed is the percentage difference between the fastest and slowest wheel. The percentage difference to the fastest wheel is chosen and not percentage different to slowest wheel because this provides greater accuracy (see the table below):

| FL Wheel Speed (kph) | FR Wheel Speed (kph) | Difference (kph) | Difference to Fastest (%) | Difference to Slowest (%) |
|---|---|---|---|---|
| 10 | 5 | 5 | (5/10)*100 = 50 | (5/5)*100 = 100 |
| 100 | 101 | 1 | (1/101)*100 = 0.9901 | (1/100)*100 = 1 |
| 165 | 170 | 5 | (5/170)*100 = 2.9412 | (5/150)*100 = 3.3333 |

The time factor is the duration for which the difference statement must be true before the speed channel switches. It is advisable to set the time factor quite low. A good starting setting is below 1-2% and below 0.2 seconds and then amend the settings as required.

**Select a strategy**

The optimal wheelspeed strategy for a vehicle varies depending on many different variables, and there is no suggestion for the best all round strategy.

The table below outlines some vehicle characteristics together with a suggested approach (actual optimal strategies may vary).

| Vehicle Characteristic | Description | Suggested Approach |
|---|---|---|
| Rear wheelspin likely | Rear Wheels could spin to a value higher than the vehicle speed | Try to use front wheels, depending on Front locking likelihood. Fastest Front Wheel will account for single wheel lockups but not two wheel lockups. |
| Front wheelspin likely | Front Wheels could spin to a value higher than the vehicle speed | Try to use rear wheels, depending on Rear locking likelihood. Fastest Rear would account for rear jacking / single rear wheel lockups. |
| Front locking | Front wheels could drop below representative vehicle speed | If single lockups, can use fastest front wheel, however for double front lockups, the rear wheels should be used if possible. |
| Rear locking / jacking under braking | Rear wheels could drop below representative vehicle speed | If single lockups, can use fastest rear wheel, however for double front lockups, the front wheels should be used if possible. |

Double lockup events tend to affect the speed trace more than wheelspin events, so for a car with both lockups and wheelspin, it can be better to select the axle with wheelspin rather than lockups.

# 40. Wipers

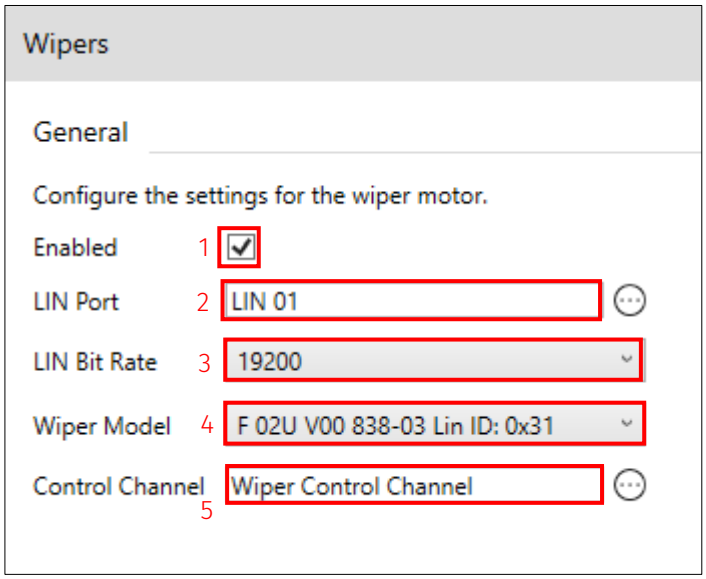
Windshield wipers are a crucial part of a race car and cannot afford to fail when rain falls during a race. You can configure the wiper control output from a power box, such as the Centaurus, or an older device such as the IPS, in many ways. This guide provides an overview of how to configure a wiper control output from the **Wipers** node, as well as other 'traditional' methods using maths and logic channels.

## 40.1. Wipers node

You can configure wiper control outputs from the **Wipers** node. Enable the wiper control output in the **Wipers** node (1), and then select the **LIN Port** to which the wiper is connected (2). Select the **LIN Bit Rate** from the dropdown menu to match the LIN bus bit rate (3.). You can select one of three common wiper models from the dropdown menu (4).

Finally, use the 'browse' button to specify the channel to drive the control (5).

**Note**: The wiper control channel can be a logic channel, a Maths channel, or a channel received over CAN, but the data type must be in U32 format.




## 40.2. Manual configuration of wiper output control

### Wiper basic

Analog wipers usually require a minimum of four wires to be connected to control individual motor windings. These wires normally consist of:

- Power
- GND
- Slow coil
- Fast Coil

Unlike LIN wipers, where the logic is controlled on the wiper motor itself, the logic to control the wiper motor needs to be integrated into the setup of the device controlling the motor. In this case, a Centaurus 5 is being used as an example.

The basic requirements to control an analogue wiper motor are outlined in this document, but you can implement further logic to add features such as intermittent settings.

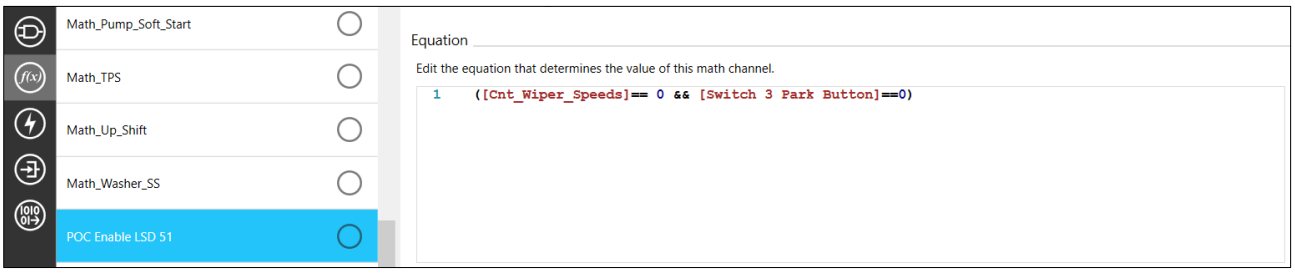
In this example, the control logic is written on the **Logic Channels** node to create a simple setup, but this logic can also be written in the maths channels if required.

The logic for the wiper is triggered on the button press of a switch panel. This can be via an RSP or CAN input from external devices.

### Maths channels

POC Enable LSD 51

The 'Park' feature of the Centaurus (the LSD) needs to be activated via the maths channel **POC Enable LSD 51** option. This channel name must match to invoke the LSD.

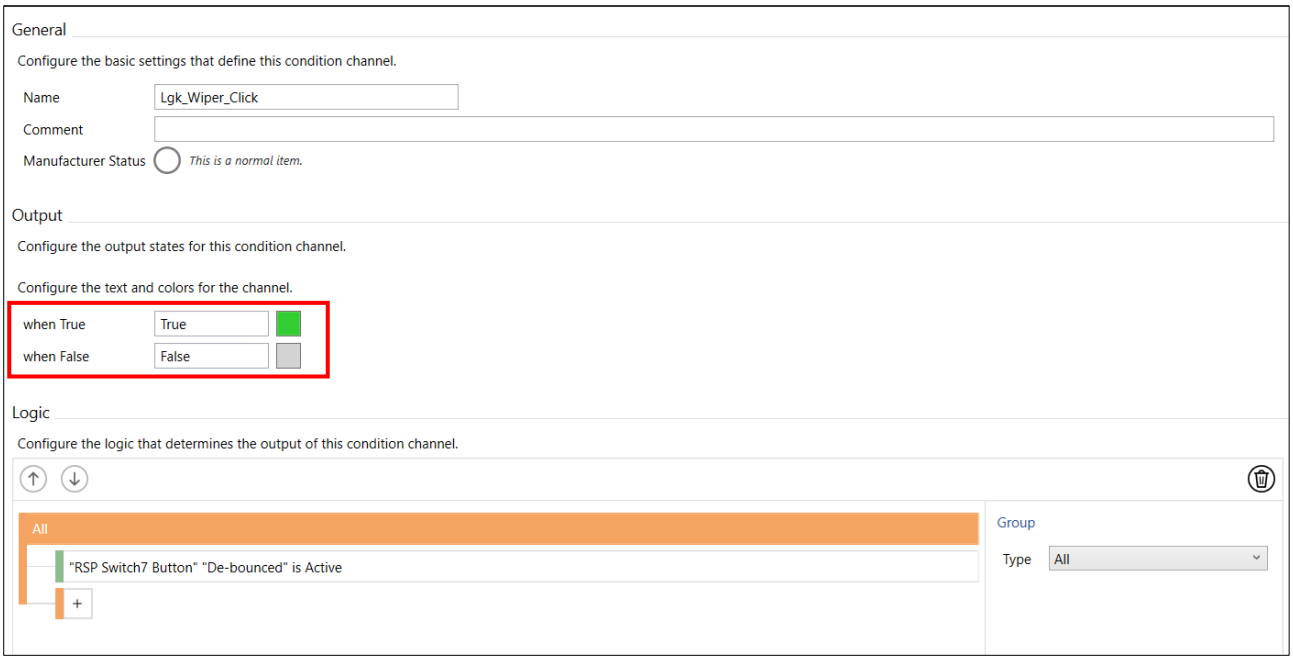



The example maths above show, the two conditions of the wiper speeds and the switch park. These conditions are fully customisable but, in this case, a physical button press is being used. These messages can come from anywhere, for example from the CAN bus or from digital inputs.

### Logic channels

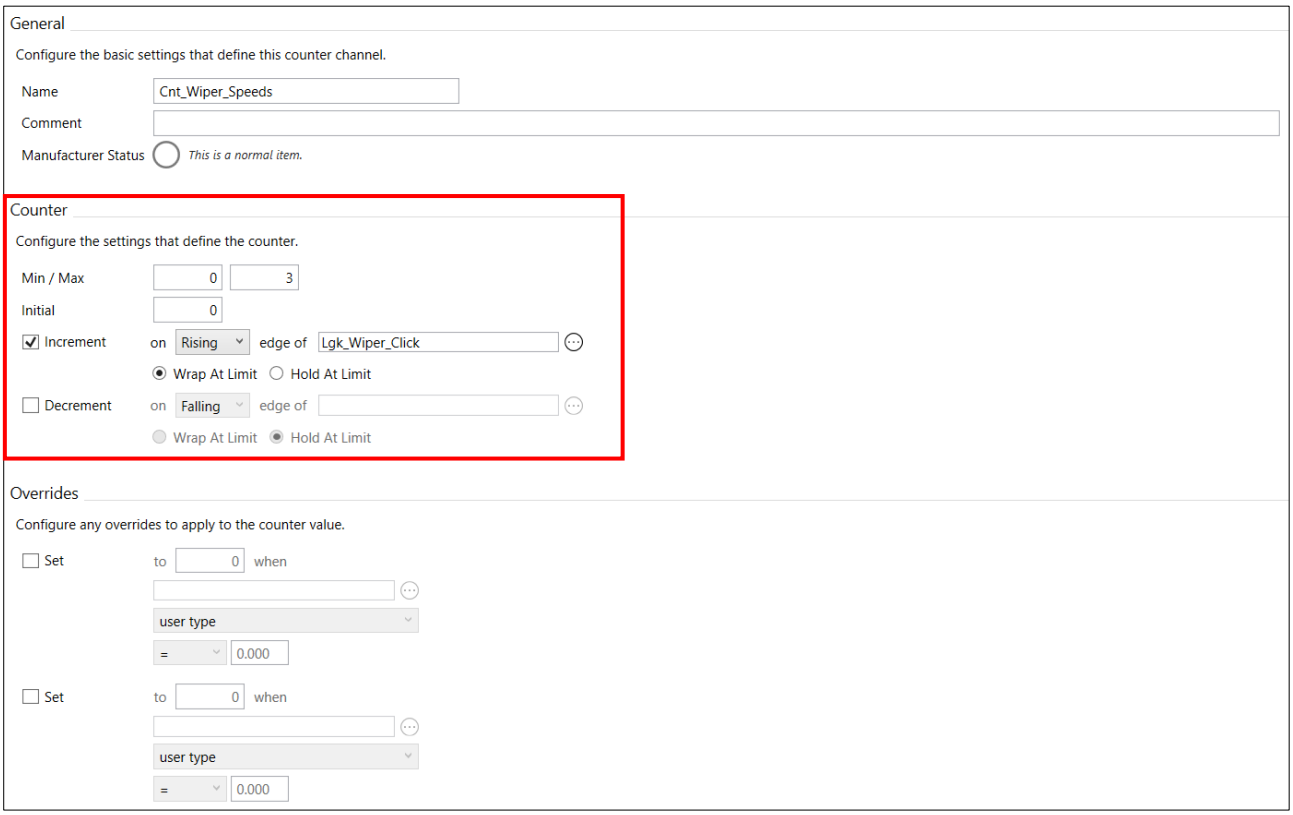
Wiper button press

This channel receives the input from the physical button. When the button is pressed, the logic channel returns 'True'.




## Wiper counter

This channel is a counter which determines how many times the button has been pressed. It is set to increase on each button press and determines at which speed to run the wipers.



In this example you can only press the button three times for the three different states:

1. Intermittent Wiper
2. Slow Wiper
3. Fast Wiper

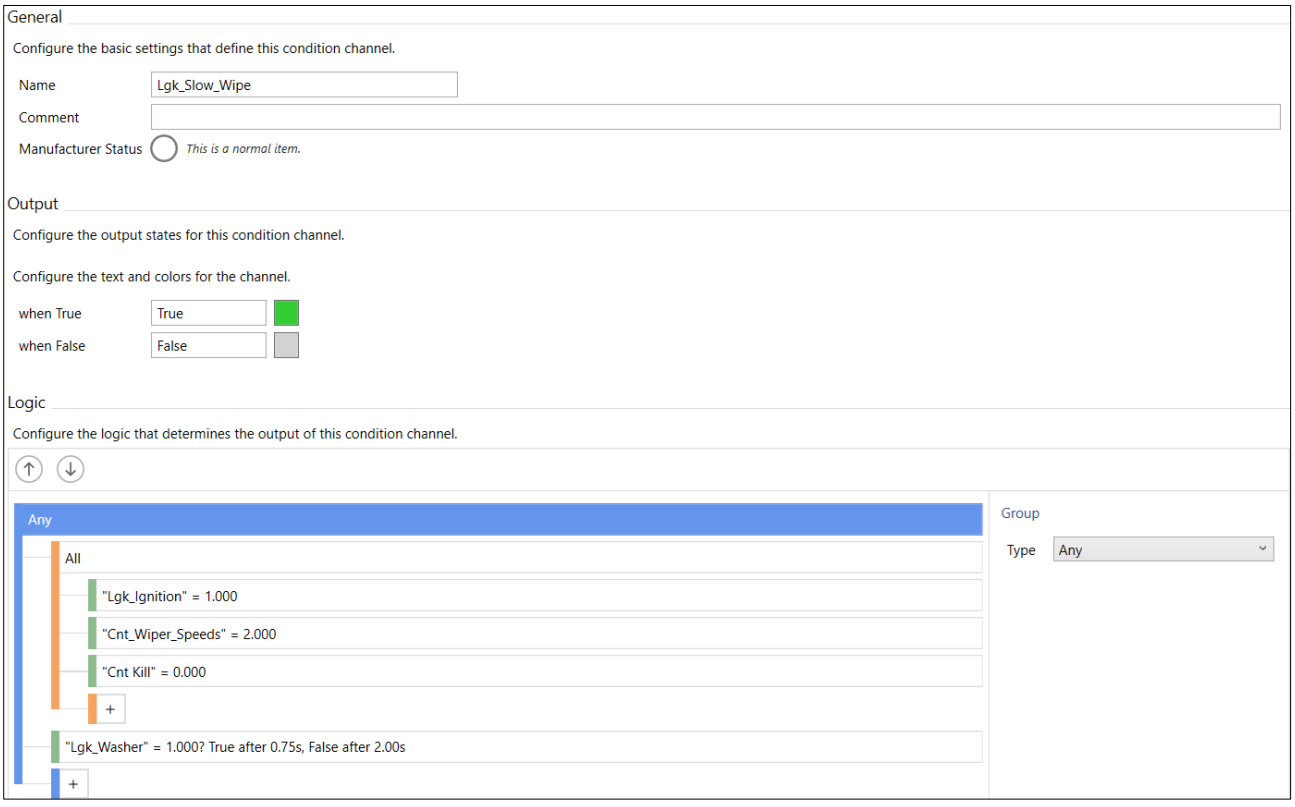
Intermittent wiper

To control the intermittent setting, a timer needs to be used to set the active high status. You can define the timer, specific to your requirements.

Slow wiper

When this logic returns 'True', the wipers activate in slow mode. This is controlled by the counter and by setting it to return 'True' when 'Cnt_Wiper_Speeds' is equal to 1. Other conditions can be added to make sure that wipers do not run when the kill switch is active and when the car is powered off.

Alternatively, if you press the washer button this automatically enables the 'Slow wipe' and bypasses the other parameters.

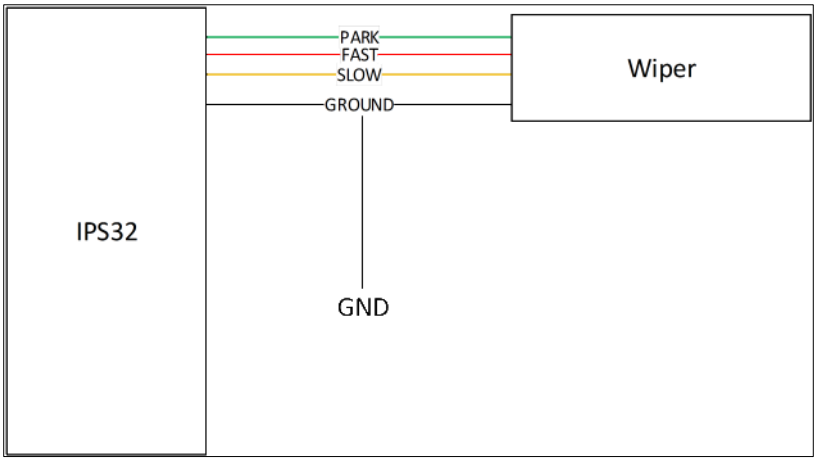



## 40.3. Wipers advanced

Note: This guide describes how you can configure analogue wipers for a IPS32 or IPS48 device. These are only guidelines on how to set up your IPS32 with wipers and the precautions to take. You can modify these guidelines to meet customer requirements, but setups should always be tested before being implemented on a car.

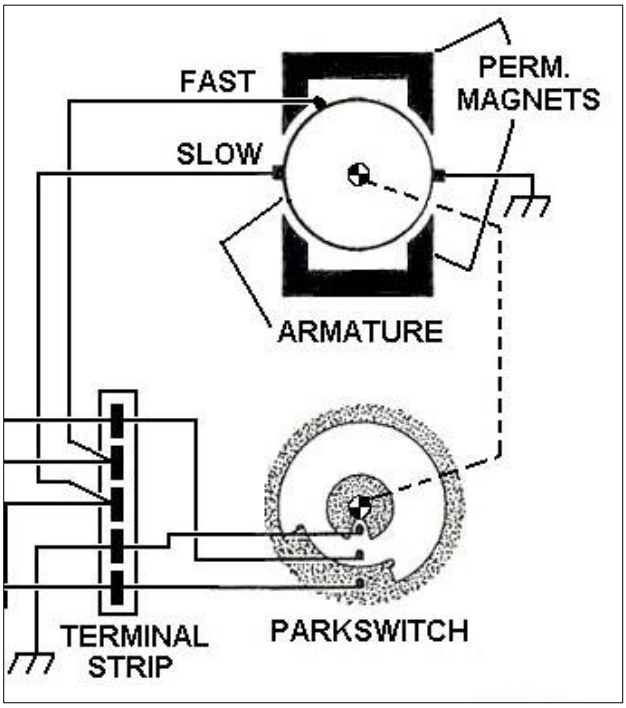
**General principle**

Unlike the LIN wiper, the analogue wiper does not have any inbuilt 'intelligence'. The controller (in this case the IPS) must tell the wiper when to go slow, go fast, or stop. The loom between the IPS and the wiper is usually composed of 4 wires.




The ground link is usually already routed directly to the chassis. The FAST and SLOW lines are the 'power' lines which supply 12V to the fast and slow motors, respectively. The SLOW line supplies current to the full length of the motor windings, whereas the FAST line supplies current to half the length of the motor windings. This is how the two different wipers speeds are controlled.

The PARK signal is a signal from the wiper system that the IPS uses to calculate when to stop and apply the brake. It is usually either a tooth or missing tooth on the wiper motor track. As the motor rotates it passes this (missing) tooth once a rotation and a mechanical switch linked to this track outputs a digital signal depending on whether a contact is registered.



**Precautions when you connect to the power box**

To stop the wipers in a controlled and accurate manner, the motor can be 'braked' by removing the live current supply to the motor and connecting it directly to ground. Outputs 1 and 17 on the IPS have this feature, so it is recommended that these outputs are used to supply the slow motor. To apply this brake, either the channel '[POC Enable LSD 1]' or the channel '[POC Enable LSD 17]' must output 1.

Without this feature you can only supply live current to the wiper, so stopping the wiper in the correct position is difficult to do accurately, relying on the friction applied to the wipers and the inertia of the motor to slow it to a halt. With the added complication of varying inertia on the wipers themselves (depending on the condition of the surface of the windscreen and the speed of the vehicle), an active stopping mechanism is required to make sure that the wipers stop in the 'park' position.

Note: You must take care when you apply the 'brake' as the IPS is not protected against accidentally supplying live current to the output and connecting it to ground at the same time. This could cause serious damage to both the IPS and the wiper.

**Control the wipers**

In this example, five math channels control analogue wipers. Below is an explanation of the function of these five math channels and the names assigned to them:

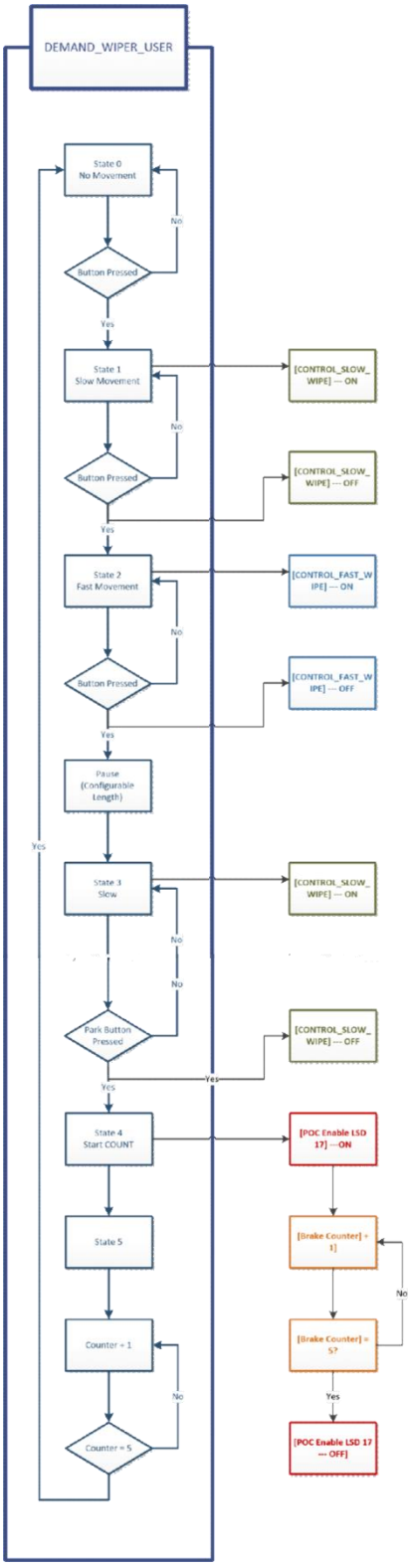
1. One master channel runs as a 'state machine' and controls the state of the system [DEMAND_WIPER_USER]
2. One that controls the 'Slow' wipe - [CONTROL_SLOW_WIPE]
3. One that controls the 'Fast' wipe - [CONTROL_FAST_WIPE]
4. One that controls the 'Brake' - [POC Enable LSD 17]
5. One to run as a counter for the duration that the 'Brake' is applied - [Brake Counter]

The driver in this example has one button (usually on the steering wheel) enables them to switch between SLOW/FAST/OFF.

Note: This is only an example, and might not suit every customer (for example, the sequence means that the wiper operates in the following order to be turned off, which may not suit some applications): SLOW > FAST > OFF.

When the system is running Fast and a Stop is requested, it returns to Slow before stopping at the park signal. This means that the process sequence is: SLOW > FAST > (SLOW > BRAKE > OFF).

The flowchart below explains the state machine process that the system follows, with conditions for each change. 'State' represents the value of the output from the channel 'DEMAND_WIPER_USER':



**Link to the outputs**

The output channels to power the two speeds are '[CONTROL_SLOW_WIPE]' and '[CONTROL_FAST_WIPE]'.

Allocate '[CONTROL_SLOW_WIPE]' to either output 1 or 17 (the channels in this example are set up to use output 17).

Allocate '[CONTROL_FAST_WIPE]' to output 18. The loom should have pins connected from these sockets to the power inputs on the wiper motor.

Set up the Park signal, received as a digital input from the wiper motor, as a button named '[WIPER_PARK Button]'.

Set the wiper control button as a button named '[WIPER Button]'. The ground for the wiper must go directly to ground.

Appendix

Master wiper channel - DEMAND_WIPER_USER

This is the channel that controls the changing state of the wiper system, on which output functions rely and into which other channels feed information. To keep all these synchronised, it is recommended that all wiper channels are calculated at the same rate (for example, 50Hz).

DEMAND_WIPER_USER

```
a5 ( [WIPER_PARK Button] & 1 );

a6 ( choose ( ( @a5 > @a4 ) , 1 , ( choose ( (@a5< @a4 ) , 2 , 0 ) ) ) ) ;

a4 ( @a5 );

a2 ( choose ( @a2 == 4 , 5 , @a2 ) ) ;

a0 ( choose(((shr([WIPER Button] , 1 ) ) & 1 ) > @a1, 1 , 0 ) ) ;

a1 ( (shr ( [WIPER Button], 1) & 1 ) ) ;

a2 ( choose ( ( ( @a0 ) == 1 ) ,choose ( @a2 == 3 , 3 , choose ( @a2 < 4 , @a2 + 1 , 4 ) ) , @a2 ) ) ;

a2 ( choose ( ( @a2 == 3 ) && ( @a6 == 2 ) , 4 , @a2 ) ) ;

 a3 ( choose ( ( @a2 == 4 ) , 0 , @a3 ) ) ;

a3 ( choose ( ( @a3 < 6 ) , @a3 + 1, 6 ) ) ;

a2 ( choose ( ( @a3 == 5 ) , 0 , @a2 ) )  ;

@a2
```

There are seven registers in this channel (a0, a1, a2, a3, a4, a5, a6, a7). Each of these has a different function in the calculation of the channel output. To explain the purpose of each register, from the start of the channel:

- a4, a5, and a6 are all used to calculate whether a rising or falling edge has been received from the 'WIPER_PARK Button'.
- a5: This register takes the value of the wiper park button. When the system is in the park position (once per revolution) the value of a5 is 1. At all other times it is zero.
- a4: This register displays the value of a5 from the previous cycle to allow the two to be compared by a6.
- a6: This register detects whether a rising edge or falling edge of the WIPER_PARK Button is received. This is done by comparing a4 and a5: if a5 is larger than a4 (as this is a digital signal, this means that as a5 is 1 and a4 is 0), then a rising edge is detected, as the current value is 1 and the previous value is 0. Conversely, if a4 is larger than a5, then a falling edge is detected. a6 displays 1 for a rising edge, 2 for a falling edge, and 0 if no change is seen.
- a0 and a1 are used to recognise a rising edge of the WIPER Button.
- a0: This register compares the current value from the WIPER Button (1 or 0) with the previous value (held by a1). If a rising edge is detected, the current value is greater than the previous value and a0 outputs a 1, otherwise 0.
- a1: This register holds the previous state of the WIPER Button for comparison with a0.
- a2: This is the final output from the channel, which has six states.
- a2 = 0: Initial state of the system, no power applied until a button is pressed.
- a2 = 1: Button has been pressed once, power applied to slow motor.
- a2 = 2: Button has been pressed again, power applied to fast motor and not to slow motor.
- a2 = 3: Button has been pressed again and a stop requested. This button press sets in motion the remaining states. The system returns to powering the slow motor only and remains in this state until a rising edge of the WIPER_PARK Button is detected (a6 = 1).
- a2 = 4: WIPER_PARK Button rising edge is detected, power is removed from slow motor, brake applied, counter is started, and state moves to a2 = 5.
- a2 = 5: Brake is held on for set duration, then a2 moves back to state 0, with wipers stopped and no power to any motors or brakes.
- a3 is a counter to keep track of whether the brake is on or off.
- a3: This register counts from 0 to 6. It starts from a3 = 0 and a2 shifts to 0 when it detects that a3 = 5; a3 resets to 0 once it reaches a3 = 6. This register is required to make sure that the brake is not applied at the same time as power is applied to one of the motors.

Initially the channels were set up so that the brake came on once it detected that '[DEMAND_WIPER_USER]' was at state 4, stayed on for a set amount of time, and then '[DEMAND_WIPER_USER]' moved to state 5 once it detected that the brake was off. However, this was not possible because the two maths channels could not both be dependent on each other. Therefore, two counters were created which worked simultaneously. They both start when they detect that '[DEMAND_WIPER_USER]' is at state 4, but one turns off the brake when it reaches a certain value, and the other moves '[DEMAND_WIPER_USER]' from state 5 to state 0 when it reaches that same value. Register a3 is the counter which controls the change of state from 5 to 0. A separate channel, 'Brake Counter' controls turning off the brake. '[Brake Counter]' is displayed below:

Brake Counter

```
a1 ( choose ( ( [DEMAND_WIPER_USER] == 4 ) , 0 , choose ( ( @a1 < 6 ) , @a1 + 1 , @a1 ) ) ) ;
@a1
```

This counter is reset to zero when it detects that DEMAND_WIPER_USER = 4, then increments up by one each cycle until it reaches 6, then rests at 6 until it detects DEMAND_WIPER_USER = 4 again.

The brake is controlled by the channel 'POC Enable LSD 17' (shown below):

**POC Enable LSD 17** ***(Alternatively 'POC Enable LSD 1' if using output 1)***

```
a1 ( choose ( ( [DEMAND_WIPER_USER] == 4 ) , 1 , ( choose ( ( [Brake Counter] < 5 ) , 1 , 0 ) ) ) ) ;
@a1
```

This channel applies the brake whenever it detects that either DEMAND_WIPER_USER = 4 or Brake Counter is below 5. Therefore, as soon as Brake Counter = 6, the brake is off.

Power Outputs

The remaining two channels are the outputs from the IPS to control the slow and fast motion of the wipers.

These are CONTROL_SLOW_WIPE and CONTROL_FAST_WIPE: **CONTROL_SLOW_WIPE**

```
a0 ( choose ( ( [DEMAND_WIPER_USER] == 1 ) , 1 , 0 ) ) ;

a1 ( choose ( [DEMAND_WIPER_USER] == 3 , choose ( @a1 == 5 , 5 , @a1 + 1) , 0 ) ) ;

a2 ( choose ( ( @a0 == 1 || @a1 == 5 ) && ( [POC Enable LSD 17] == 0 ) , 1 , 0 ) ) ;

@a2
```

This channel engages the slow mode of the wiper motor when DEMAND_WIPER_USER = 1 and when DEMAND_WIPER_USER = 3 (after a delay of 5 cycles). In both cases, POC Enable LSD 17 must be zero for the motor to be engaged.

CONTROL_FAST_WIPE

```
choose ( ( [DEMAND_WIPER_USER] == 2 ) && ( [CONTROL_SLOW_WIPE] == 0 ) && ( [POC Enable LSD
17] == 0 ) , 1 , 0 )
```

This channel engages the fast mode of the wiper whenever all of the following conditions are met: DEMAND_WIPER_USER = 2, CONTROL_SLOW_WIPE is off and POC Enable LSD 17 is off.

Note: Some logical AND (&&) and logical OR (||) functions are used in these math channels. Care must be taken when you use these. It is strongly recommended that parenthesis are used wherever appropriate to separate each member of a logical operation to avoid miscalculations:

( xxxxxx > 1 ) && ( yyyyyy == 2)
( ( xxxxxx == 0 ) || ( yyyyyy > 4 ) ) && ( zzzzzz != 0 )

Function explanations

| Function | Description | Syntax |
|---|---|---|
| *choose* | The choose function is a logical 'if', which gives one output if the statement is true, and another output if the statement is false. | choose('Statement' , Output if statement true , Output if statement false ) |
| Registers | Variables to which values can be set. Up to 7 registers can be used in any channel, identified by their number<br>(ranging from 0 - 6) | To set register value: a1(***value)***<br><br>To reference register in other function: @a1<br><br>Available registers:<br><br>a0<br>/@a0<br>a1 /<br>@a1<br>a2/<br>@a2<br>a3<br>/@a3<br>a4<br>/@a4<br>a5<br>/@a5<br>a6<br>/@a6 |
| | | Example:<br><br>a2 ( 7 ) ; ---> set a2 value to 7<br><br>choose ( @a2 == 7 , 1 , 0 ) ; ---> If a2 is equal to 7, output 1, otherwise output 0 |
| == | Logical 'is equal to' | choose ( @a4 == 1 , 3 , 4 ) ; ---> If register @a4 is equal to 1 then output 3, otherwise output 4 |
| < | Logical 'is less than' | choose ( @a1 < @a5 , 9 , 2 ) ; ---> If register @a1 is less than the register @a5, then output 9, otherwise output 2 |

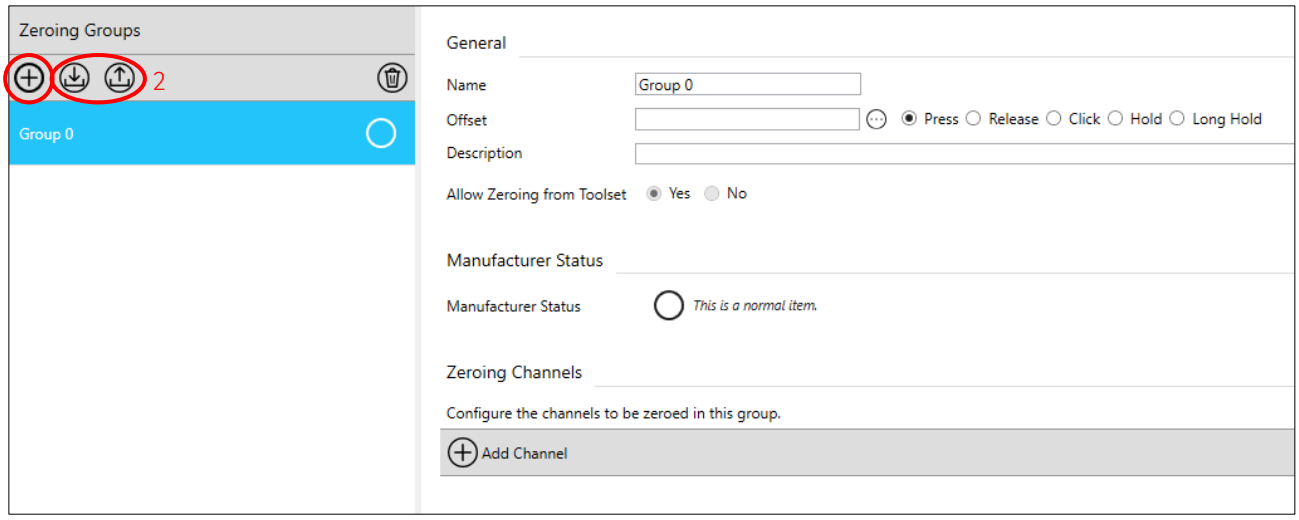
| | | |
|---|---|---|
| > | Logical 'is greater than' | choose ( @a1 > @a5 , 9 , 2 ) ; ---> If register @a1 is greater than the register @a5, then output 9, otherwise output 2 |
| + | Mathematical 'plus' | ( @a6 + 1 ); |
| && | Logical 'and' | choose ( ( @a2 == 3 ) && (@a4 == 4 ) , 6 , 2 ) ; ---> If register @a2 is equal to 3 and register @a4 is equal to 4, then output 6, otherwise output 2 |
| \|\| | Logical 'or' | choose ( ( @a3 == 3 ) \|\| ( @a3 == 4 ) , 8 , 5 ) ; ---> If register @a3 is equal to 3 or register @a3 is equal to 4, then output 8, otherwise output 5 |
| shr | Shifts byte right by X bits, inserting zeroes to the left | shr ( [Input 4] , 3 ) ; ---> Shifts the input from [Input 4] right by 3 bits, with 3 zeroes inserted to the left |
| & | Returns a U32 'ANDed' with another<br><br>U32 | [Input 7] & [Input 6] ; ---> Returns 1 if [Input 7] and [Input 6] have the same value |

# 41. Zeroing

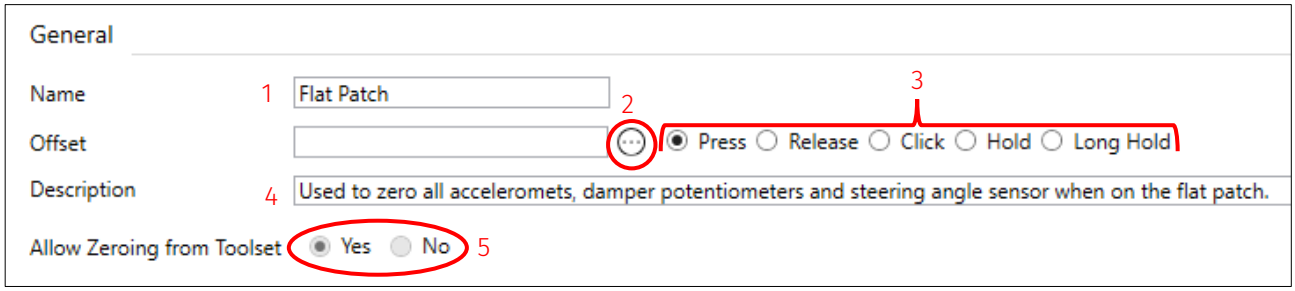
The **Zeroing** node enables you to configure channels to be zeroed. You can zero channels individually or in groups.

## 41.1. Create a zeroing group

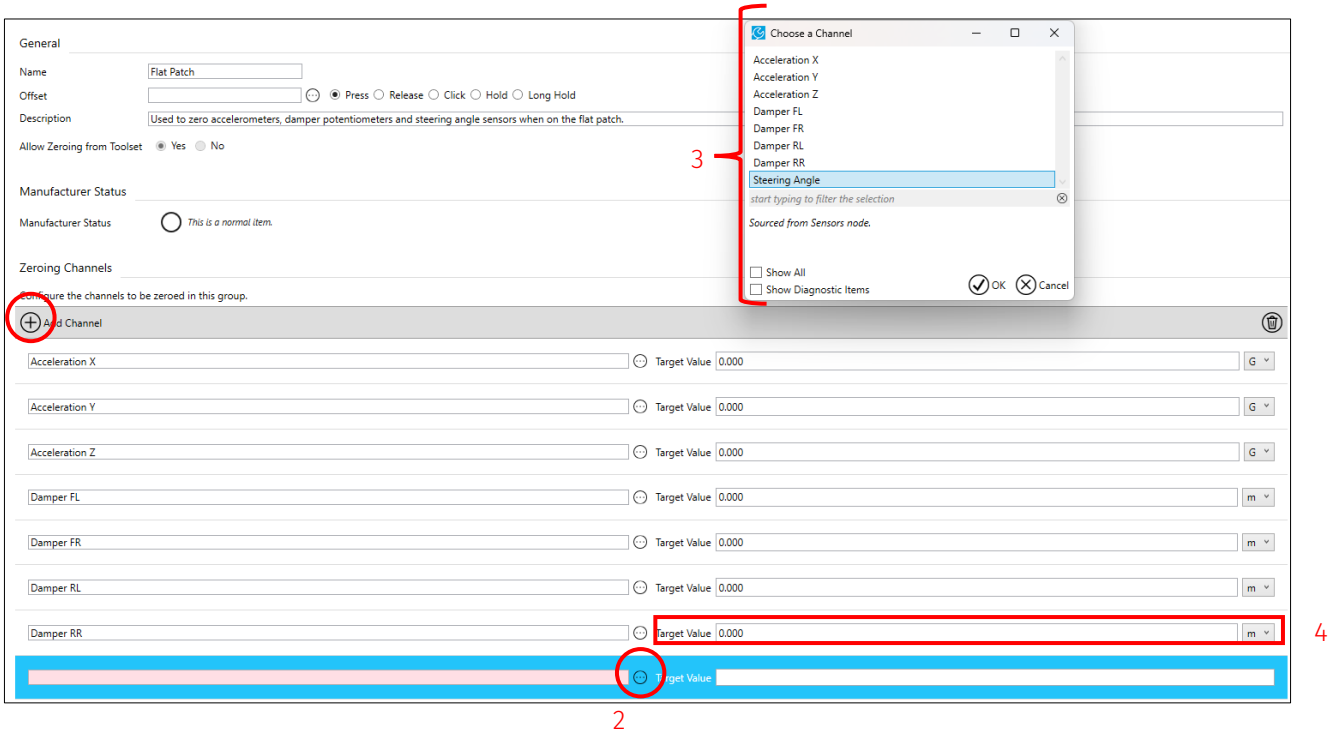
Use the **+** tool to add a new zeroing group (1). You can import and export zeroing groups from existing setups (2).



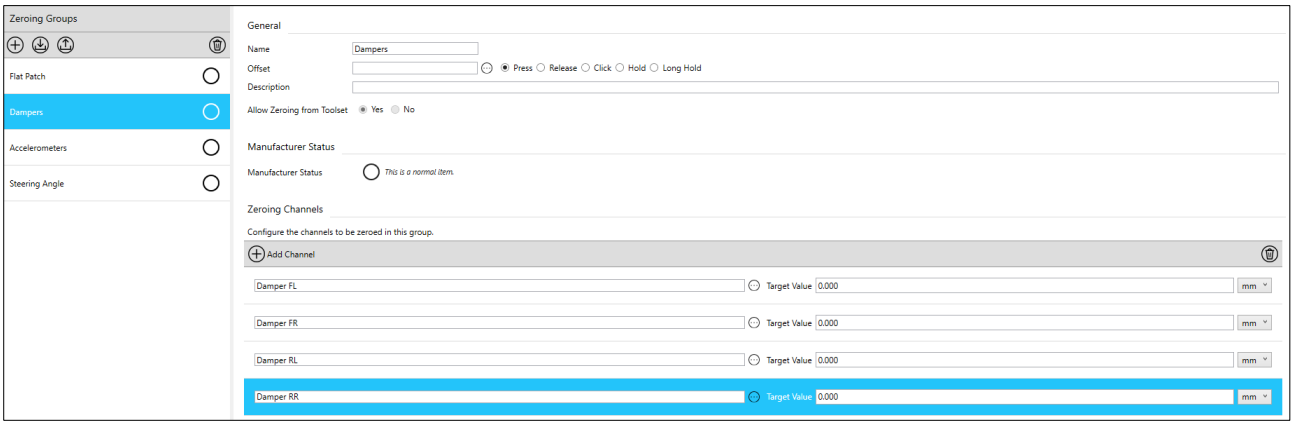

You can name the zeroing group (1) and add an optional description (4). Use the 'browse' tool to select offset buttons (see Setups - Buttons) (2), and then the button click type (3). Select the **Allow Zeroing from Toolset** option (5) to enable zeroing from a PC from the **Actions** tab in Toolset (see Actions - Zeroing).



You can then add channels populated from the **Sensor** node (see Setups - Sensors) to the zeroing group (1) by selecting available sensor channels from the dialog box (2/3). You can also set the target value and unit for the channel (4).

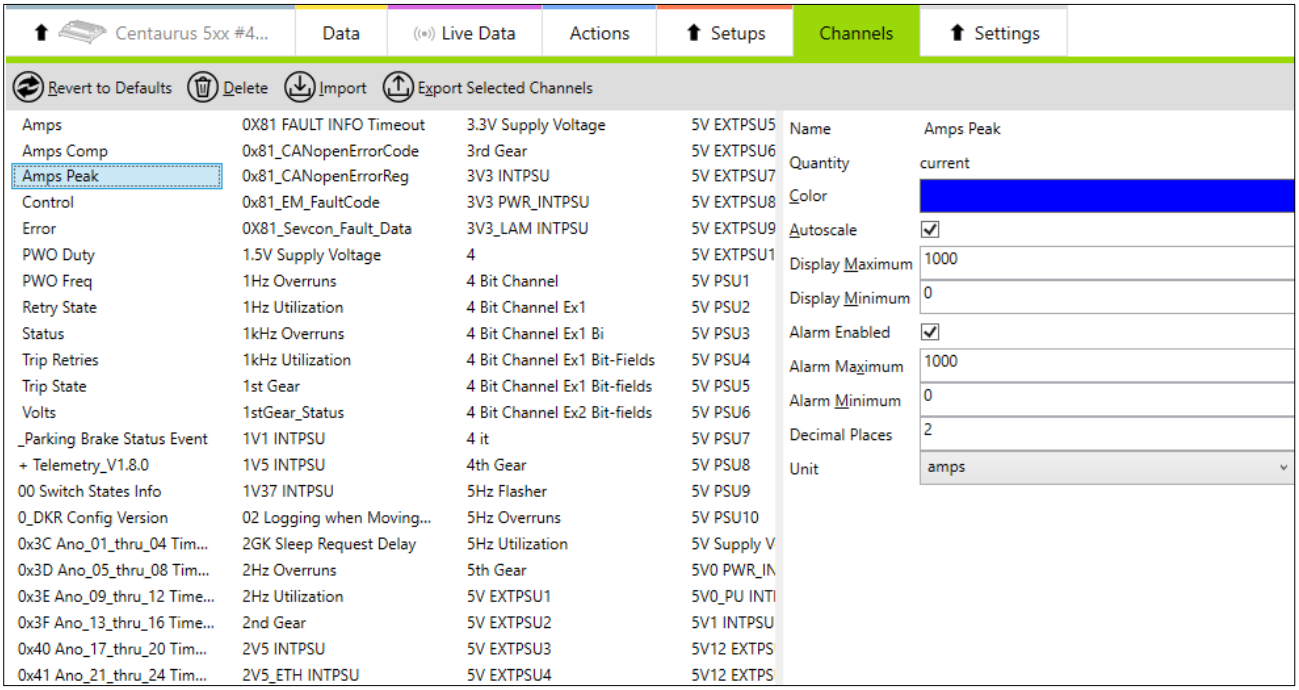



You can add multiple zeroing groups to zero sensors exclusive of each other. See Actions - Zeroing for more information about how to set the zero from Toolset.



# 42. Channels configuration overview

The **Channels** page displays all the channels created or used in Toolset. It enables you to customise the display and alert settings for each channel display for efficient monitoring. For example, you can convert channels displayed in live data between units using the **Channels** page.

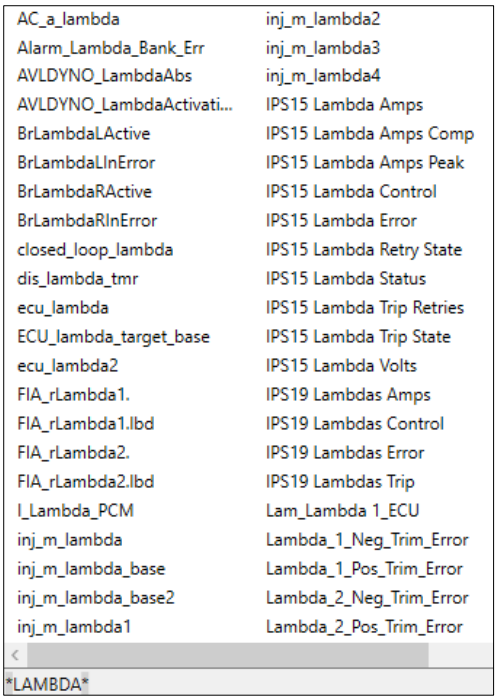



## 42.1. Toolbar options

- **Revert to Defaults**: Resets all settings to their default values.
- **Delete**: Removes the selected channel.
- **Import/Export Selected Channels**: Allows you to import or export configurations for specific channels in *.tcl* format (Toolset **Channel List**).


## 42.2. Channel list

The list on the left displays available channels. Whenever a new channel is created or imported, they are listed in this section. You can perform a magic search (ctrl+shift+8) to find any specific channels, as shown in the example below:



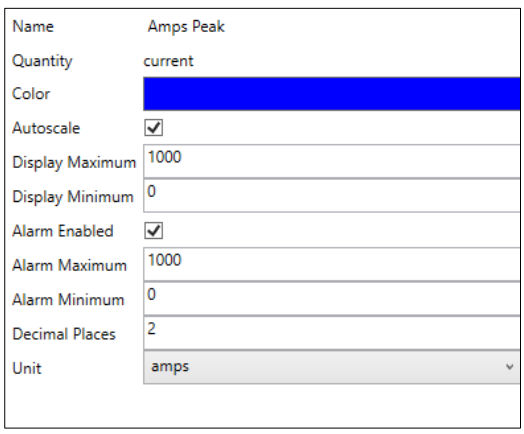

In this example, any channel that contains 'LAMBDA' in its name is shown, making it easier to search large sets of data.

Select a channel to display its configuration options on the right.

**Channel configuration options**

For the selected channel (for example., *Amps Peak*):

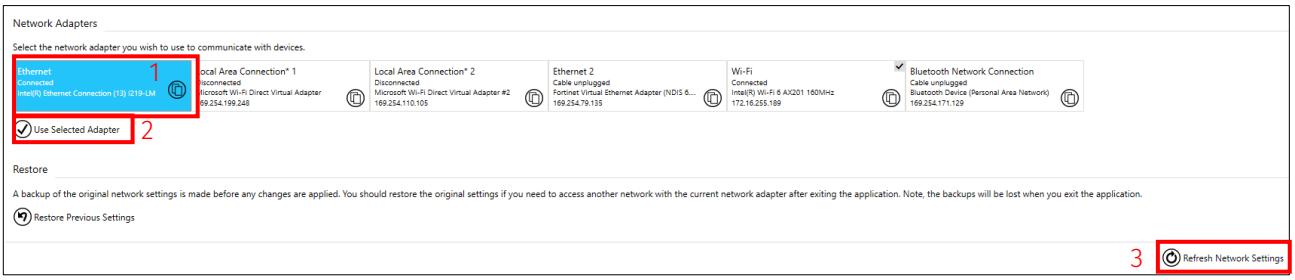
- **Name**: Displays the name of the channel.
- **Quantity**: Defines the measured type (for example, *Current*).
- **Color**: Sets the display colour for the channel (for example, *Blue*).
- **Autoscale**: Automatically scales the display for data values.
- **Display Maximum/Minimum**: Sets upper and lower display limits.
- **Alarm Enabled**: Toggles alarm notifications for the channel.
- **Alarm Maximum/Minimum**: Sets alarm threshold values.
- **Decimal Places**: Defines the number of decimal places for displayed values.
- **Unit**: Specifies the unit of measurement (for example, *Amps*).



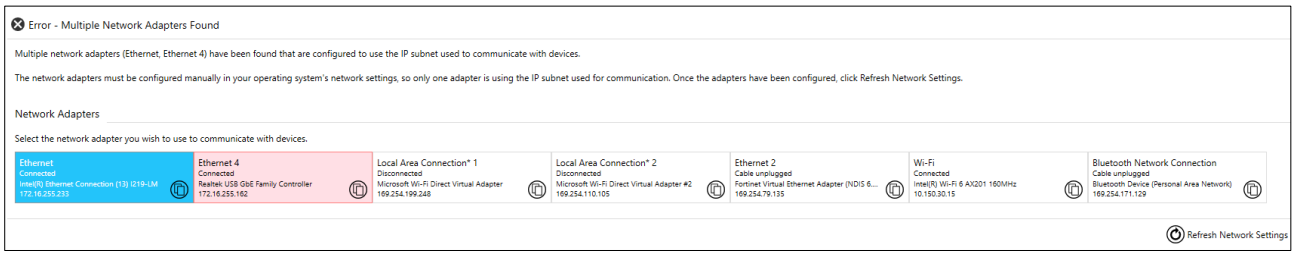
# 43. Settings

## 43.1. Network

The **Network** page is used to configure the connection to the device. All PC network connections are displayed in the menu. Click the required network adaptor and confirm that it is connected (1). Select **Use Selected Adaptor** to confirm the chosen network adaptor (2). If your network adaptor is not displayed, select the **Refresh Network Settings** option (3).

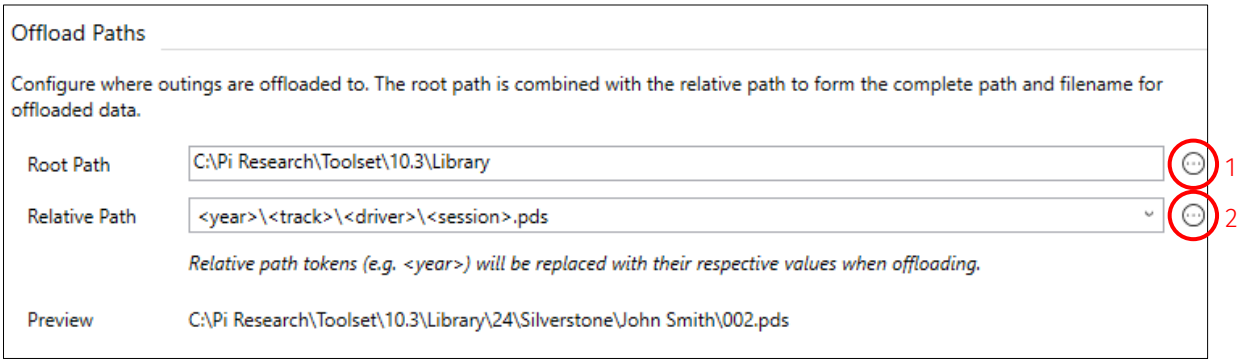



If there is a network error, for example, multiple network adaptors are found, an error is displayed with a brief description of the error. Make sure that the ethernet adapter shows as **Connected** before you try to connect to a device.

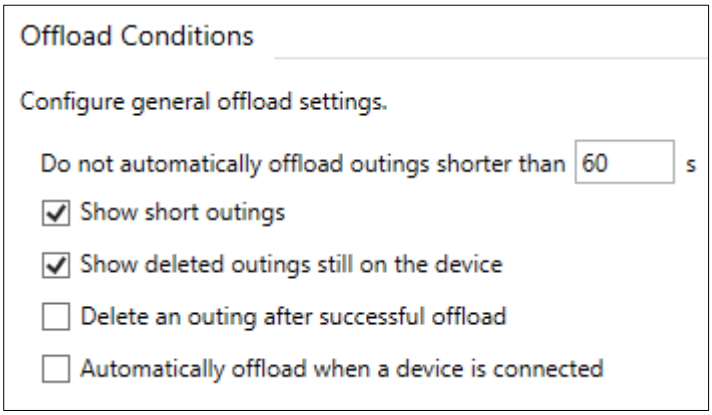



## 43.2. Data offload

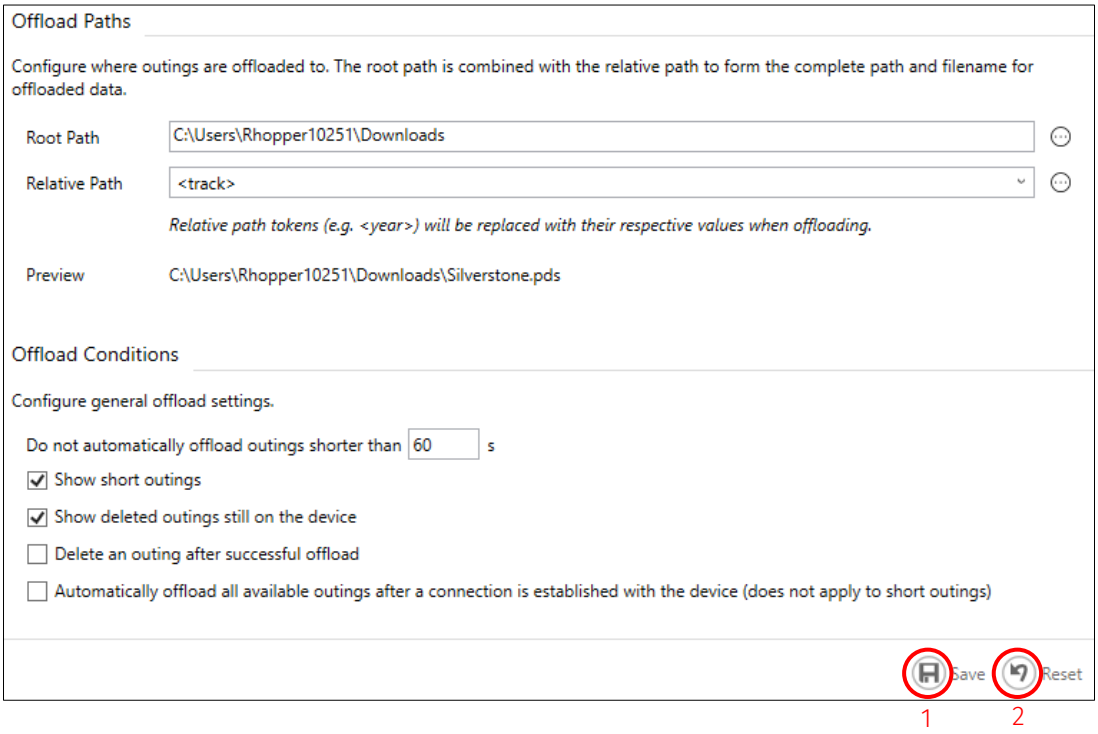
On this page you can configure the root path to where outings are offloaded, as well as the name of the PDS file for the outing. You can copy the file location from the file explorer or select it via the browse button (1). You can configure this filename with property tokens, which are automatically replaced by their respective values during offload. Use the 'token browse' option to select relative path tokens (2).



On the same page you can also configure further offload settings, such as automatic offloading and deletion, to enable a more efficient workflow.




Note: Save any changes for them to take effect (1). Click Reset to restore the default settings (2).



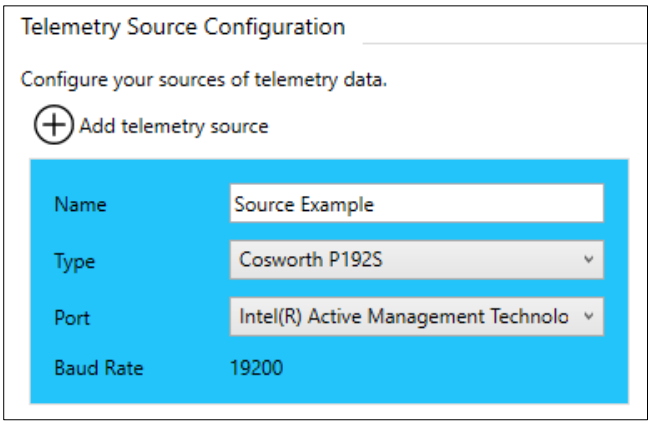
## 43.3. Telemetry

This page allows you to configure telemetry data sources, telemetry server settings, and telemetry logging options.

### Telemetry source configuration

This section is where you can add and configure telemetry data sources.

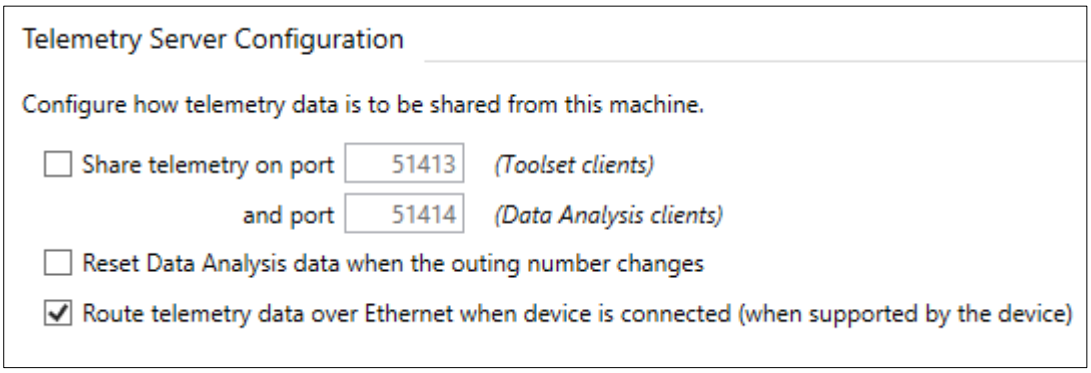
- Add telemetry source: Click the + button to add a new telemetry source.
- Name: The name assigned to the telemetry source.
- Type: Select the type of telemetry device from the dropdown list.
- Port: Enter the communication port to which the device is connected.
- Baud Rate: The baud rate for the connection.




### Telemetry server configuration

This section allows you to configure how the telemetry data is shared over a network.

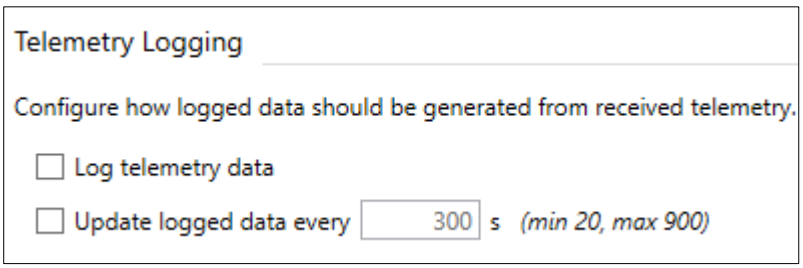
- Share telemetry on port:
  - *Toolset clients*: Specify the port number for toolset clients (for example, *51413*).
  - *Data Analysis clients*: Specify the port number for data analysis clients (for example, *51414*).
- Reset Data Analysis data when the outing number changes: Enable this option if you want to reset data analysis after each new outing.
- Route telemetry data over Ethernet when the device is connected: Select this option to enable telemetry data transmission via Ethernet where the device supports it.



#### Telemetry logging

This section allows you to configure how logged data is handled.

- Log telemetry data: Select this option to enable logging of telemetry data.
- Update logged data every X seconds: Specify the interval in seconds at which the logged data is updated (for example, 300 seconds). The minimum value is 20 seconds, and the maximum is 900 seconds.

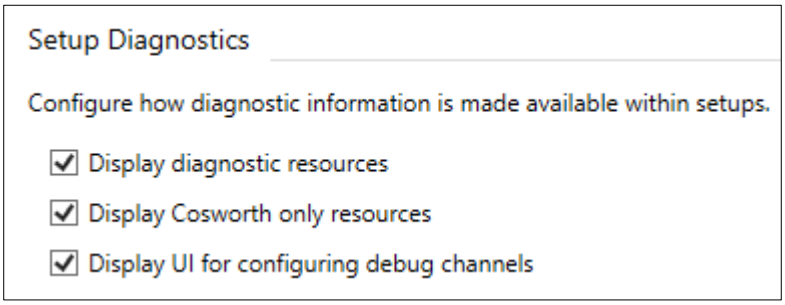



## 43.4. Diagnostics

This page allows you to configure the diagnostics display and rendering options for the application.

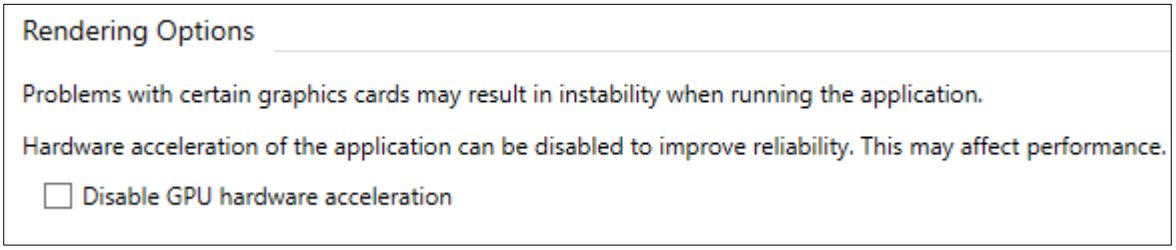
#### Setup diagnostics

- Display diagnostic resources: Enable to show general diagnostic tools.
- Display Cosworth only resources: Display diagnostics specific to Cosworth devices.
- Display UI for configuring debug channels: Enable this option to show the interface to set up debug channels.




#### Rendering options

- Disable GPU hardware acceleration: Turn off hardware acceleration to improve stability, especially if there are issues with the graphics card. Note that this might affect application performance.

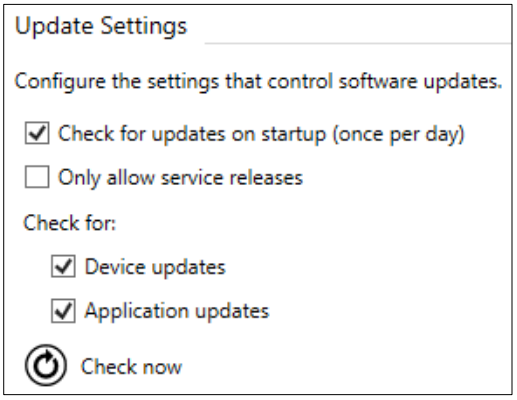


## 43.5. Upgrade

This page allows you to manage software updates and migrate settings from a previous version of the toolset.

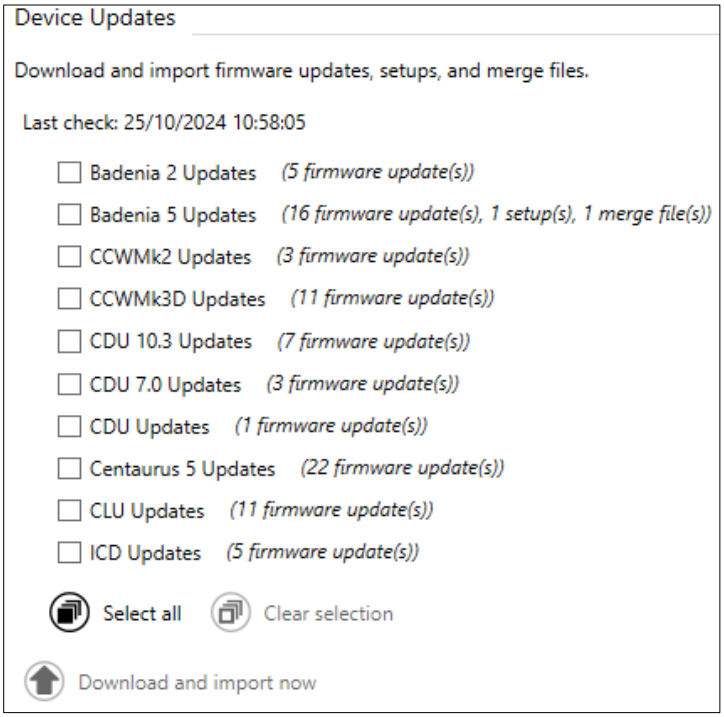
**Update settings**

- Check for updates on startup: Enable this to automatically check for updates once a day.
- Only allow service releases: Limit updates to stable service releases.
- Check for: Select whether to check for device or application updates.
- Check now: Manually trigger a check for updates.



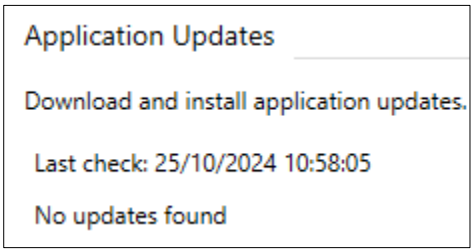

**Device updates**

- View and select firmware updates, setups, and merge files for various devices.
- Last check: Shows the date and time of the last update check.
- Use the checkboxes to select updates for specific devices.
- Select all or Clear selection: Select all updates or reset the selections.
- Download and import now: Click to download and save selected updates.



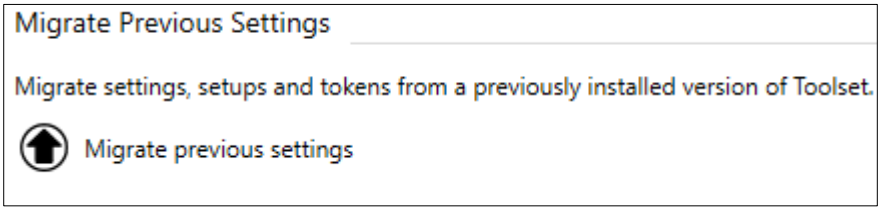
#### Application updates

- Last check: Shows the last time application updates were checked.
- If updates are found, this section provides the option to download and install them.




#### Migrate previous settings

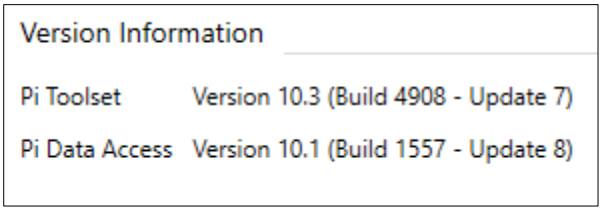
- Migrate previous settings: Import settings, setups, and tokens from an older version of Toolset.




### 43.6. About

#### Version Information

On this page, the current versions of Toolset and Data Access are displayed. If there is an active license, it is shown on this page.




Note: If an asterisk is displayed next to the page title (for example, DataOffload*), it indicates that some settings have been changed without being saved.

Document: Pi Toolset User Guide (Complete Version)

**Rev:** 1.0 - Released

All Information in this document is correct as of 23/07/2025

**T (UK)** +44 (0) 1604 598 300

**T (US)** +1 (317) 644 1037

**E** enquiries@cosworth.com

**W** www.cosworth.com