

CANStream user guide

Table des matières

Introduction	5
Home	5
General overview	6
Getting started	7
Computer requirements	7
License agreement	8
Installation	10
Registration	14
PEAK USB CAN adapter driver	15
Main interface	16
CAN Controller panel	17
CAN controller modes	18
Spy & Manual mode	18
Cycle mode	19
CAN controller panel tool bar	19
Menu strip	21
CAN Configuration	21
Cycle	22
Recording	23
Tools	24
View	25
CAN Frames configuration	25
Single bus configuration	26
Bus properties	27
CAN message properties	28
CAN signal properties	29
CAN Configuration tree view	31
Multiplexed parameter	32
Message map	33
Virtual CAN signal	34
Vector CAN data base (dbc) import	36
Multiple buses configuration	38
CAN Bus information	41
Configuration import & export	42
Manual mode	43
Data Tx	44
Engineering format data Tx	45
Raw format data Tx	46
Data Rx	48
Engineering format data Rx	49
Raw format data Rx	50
Rx data graphic	51
Cycle mode	53
Cycle playing	54
Cycle RX data panel	56
Cycle starting and ending points setting	57
Cycle loop number setting	59

Virtual CAN signals in cycle	60
CAN controller appearance options	61
Control layout	61
Manual control layout	62
Cycle control layout	64
Grids contents control	65
Grid rows	65
Grid columns	66
Manual control grid columns	67
Cycle player grid control	69
Cycle creation	71
Easy cycle creation	72
Cycle data association	74
Advanced cycle creation	78
Record replay	84
Data analysis	87
Data browser	88
Analysis window	89
Analysis window presentation	90
Analysis window tool bar	91
Analysis window shortcut keys	92
Data file	93
Analysis functions	94
Main cursor	94
Reference cursor	96
Zoom	98
Statistics	102
Legend	104
Snapshot	107
Print	108
Graph window settings	111
Channel list	111
Graphic layout	113
Graphic window configuration	115
Generic series properties	117
General properties	120
Graphic grids properties	123
Abscisse properties	124
Cursors properties	126
Graphic serie detailed configuration window	128
Serie general properties	130
Serie format	133
Serie Y Axis	136
Serie custom grid	138
Serie reference lines	141
Appendix	144
Line styles	144
Virtual channels	145
Virtual channel edition	146
Virtual channel library properties	148

Virtual channel properties	149
Virtual channel testing	151
Virtual channel built-in functions	153

Introduction

Created with the Personal Edition of HelpNDoc: [Create HTML Help, DOC, PDF and print manuals from 1 single source](#)

Home

First of all, thanks for using CANStream !

This document is dedicated to the countless lost hours of both people having to search for simple answers, and people having to answer questions that shouldn't have to be asked in the first place. Documentation is always hard to get, good documentation nearly impossible.

This markup is based in the default appearance of Microsoft Windows. This can be different on your own system, but the names (text) should always the same. This is especially important on the screen shots. They are only for reference, your appearance of CANStream can be quite different.

Also, when talking about the left and right mouse buttons, the logical left and right is meant. These are the same as the physical left and right mouse buttons in the case of a right-handed setup. However, if you have a left-handed setup you will probably have the buttons swapped so act accordingly. This means in general you do what you normally do for most actions, CANStream mostly follows the standard Windows application behavior.

Likewise, the layout is discussed as it is displayed in a left-to-right order.

Created with the Personal Edition of HelpNDoc: [Easily create HTML Help documents](#)

General overview

CANStream is a .NET application intended to be used for CAN (Controller Area Network) bus communication development, testing and validation. CANStream allows to extensively test any system using CAN communication by sending and receiving CAN frames. Thanks to its powerful built-in mathematical expression evaluator, CANStream can also behave as a real control system feeding back the test device or commanding a third party device with context sensible data. Extended data logging and data analysis features of CANStream provide a comprehensive solution for testing and results analysis.

CANStream extensively uses the PCAN-Basic API developed by PEAK System for the PCAN-USB adapter. For more information about PEAK System: <http://www.peak-system.com>

You will therefore need to have PCAN-USB adapter and at least one free USB port available to make a complete use of CANStream.

CANStream can manage up to eight PCAN-USB adapters at the same time, thus you will need eight USB ports available.

As any .NET application, CANStream needs a Microsoft .NET framework to operate. You might need to install or upgrade your .NET framework.

If so, please visit <http://www.microsoft.com/net> to download the latest version of the Microsoft .NET framework.

Created with the Personal Edition of HelpNDoc: [Create iPhone web-based documentation](#)

Getting started

Created with the Personal Edition of HelpNDoc: [Full-featured multi-format Help generator](#)

Computer requirements

Hardware	
CPU	Pentium 4 1.2 GHz
RAM	1 GB
Hard drive	25 MB free space
USB port	1 minimum
Operating system	
Any of those Windows version	2000, XP (32/64 bits), Vista (32/64 bits), Seven (32/64 bits), 8 (32/64 bits)
.NET Framework	4.0

Created with the Personal Edition of HelpNDoc: [Free iPhone documentation generator](#)

License agreement

DEFINITIONS

"Software" means CANStream and its components. "User" means the Software customer or end user.

OWNERSHIP

Cobalt Solutions is the owner of the Software and of its copyright.

LICENSE

The Software ownership is not transferred to the User. Cobalt Solutions grant the User a simple, not-exclusive license to use the Software only on limited numbers of workstations as defined in the appendix to the present agreement, for a period of time depending of type of license. During this period the User is entitled to receive services as described in appendix to the present agreement. When the license will expire, licensed features will be automatically disabled. If the license is not renewed the User may continue to use the Software under terms and conditions set forth in this "LICENSE AGREEMENT", but is not entitled to receive any services of any kind. If the license is not renewed within four (4) months from the expiry date, then the renewal option will be lost and the User needs to order a brand new license.

COPY RESTRICTIONS

The User is allowed to make one (1) backup or archive copy only for reserve. All copyright notices and any other proprietary legends on the original Software shall be reproduced on such copy. Both the original and the reserve copy of the Software are subject to the restrictions set forth in this license agreement and they shall be destroyed in case of license termination.

RESTRICTIONS

The User shall not rent or loan, distribute or sublicense the original or the backup copy of the Software and the manuals. User may not modify, translate, edit, decompile, reverse engineer, disassemble the Software or create derivative utilities based upon it. User may not edit the structure of the Software, nor ask to any other person to do so in order to modify its structure, dimensions, etc. THE USER SHALL NOT COPY THE MANUALS OR THE RELATED DOCUMENTATION, unless Cobalt Solutions gives written authorization. The license will immediately terminate without notice if the User fails to comply with any provision set forth herein.

DISCLAIMER OF RESPONSABILITY

The Software is provided "AS IS" and without warranty of any kind.

LIMITED WARRANTY

Cobalt Solutions reserves the right to grant the User the license for the Software, and he will replace the media on which the Software is furnished (disks and/or CD and/or password keys) which prove to be defective in material or workmanship under normal use. This warranty is good during a period of thirty (30) days starting from the date of purchase, and it does not apply to damages or problems resulting from misuse, abuse, accident or improper installation.

DISCLAIMER OF WARRANTY AND LIMITATION OF LIABILITY. EXCEPT AS SET FORTH ABOVE, NO OTHER WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING THE WARRANTIES OR CONDITIONS OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE APPLICABLE TO THIS PRODUCT. COBALT SOLUTIONS SHALL NOT BE LIABLE FOR ANY LOSS OF PROFITS, LOSS

OF BUSINESS, INDIRECT, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES, EXPRESSLY OR NOT SET FORTH HEREIN.

THIRD PARTY RESPONSIBILITY

The Software copyright is ownership of Cobalt Solutions. The User may be considered directly liable by Cobalt Solutions for any action concerning the use of the Software, made directly or indirectly, that does not comply with this "LICENSE AGREEMENT".

USER DUTIES AND MISUSE

The User agrees to make any effort in order to assure that his employees, consultants or other people that can access to the Software comply with the terms set forth in this "LICENSE AGREEMENT", including the prohibition to authorize anyone to use parts of the Software in order to identify its source code. The User shall immediately make any effort to stop any misuse that comes to his knowledge.

LICENCE TERMINATION

Cobalt Solutions reserves the right to terminate the license at any time if the User fails to comply with any provision of this "LICENSE AGREEMENT". In this case, the User must immediately destroy the Software and return the media and all the copies to Cobalt Solutions.

CONTROLLING LAWS

This agreement is governed by the laws of France, also for what concerns patent and copyright. In case of judicial acts or proceedings consequent to a demonstrated failure to comply with the conditions set forth in this "LICENSE AGREEMENT", all the legal costs will be at losing party expense. This agreement regulates any Software update and/or upgrade, and constitutes the entire agreement between the parties with respect to the use of the Software and the related documentation. Any dispute falls under the competence of the Law Court of Paris (France).

APPENDIX TO END USER LICENCE AGREEMENT

NUMBER OF INSTALLATIONS

Number of workstations on which the Software may be installed on CANStream: a finite number of installations defined by User at order placement.

IMPORTANT NOTE

This covers only installations of Cobalt Solutions proprietary software. Any license fees for third party software are not included in the Cobalt Solutions license fee.

SERVICES

For one year from the order of a new license or renewal of an existing one the User is entitled to receive the following services

- (i) supply of latest CANStream software release at the time of new license purchase or renewal;
- (ii) User is entitled to request, under payment, customization of CANStream software;
- (iii) Supply of upgraded features (at the sole discretion of MMM) and debugged Wintax4 software releases, if any, during a valid license period;

Created with the Personal Edition of HelpNDoc: [Easy EPub and documentation editor](#)

Installation

To install CANStream on your computer, just run the installation wizard and follow installation steps.

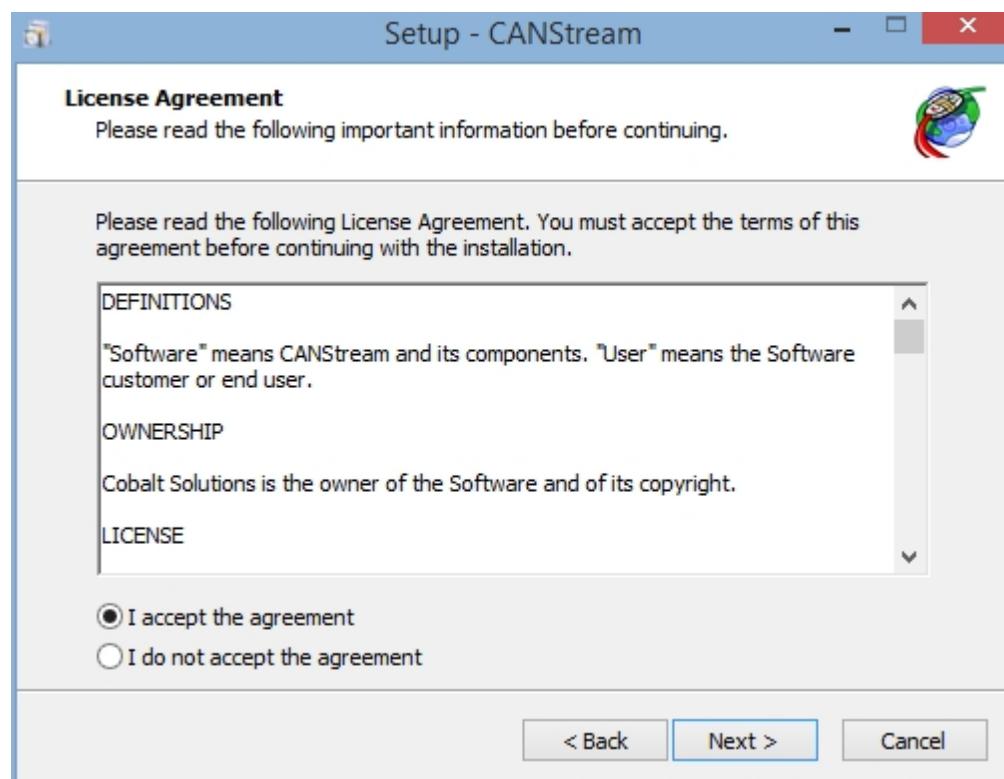
Depending of your system, Windows 32 bits or 64 bits, you have to run the appropriate installation wizard.

- Windows 32 bits: Setup_Win32.exe
- Windows 64 bits: Setup_x64.exe

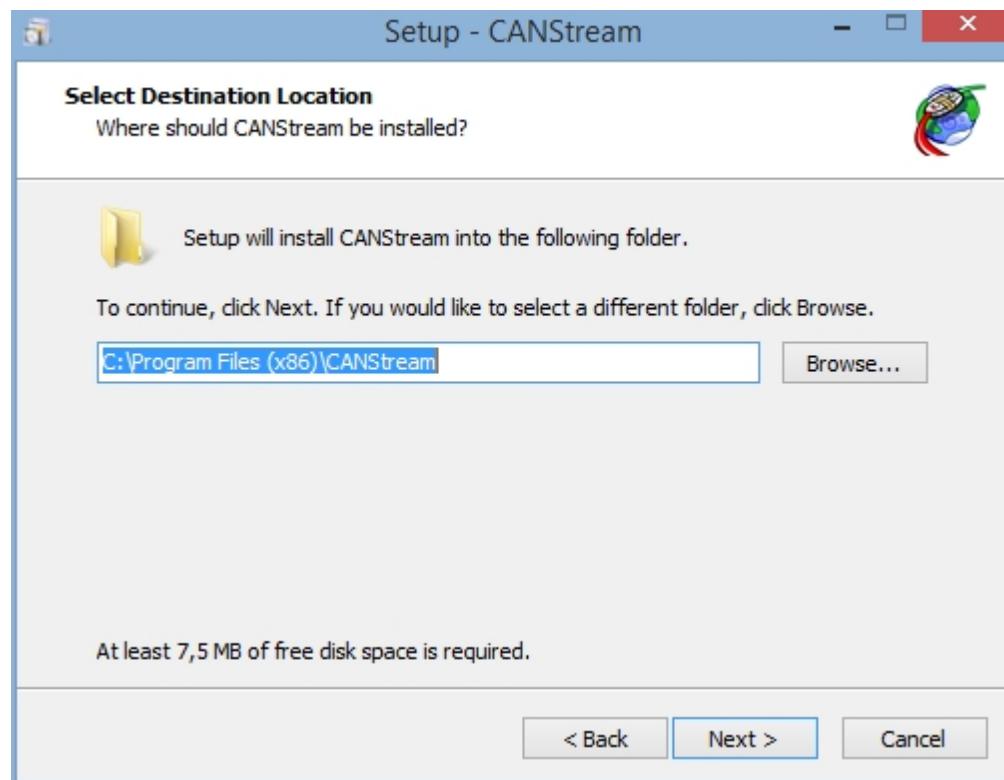
Just double-click on the file corresponding to your system to launch the procedure.



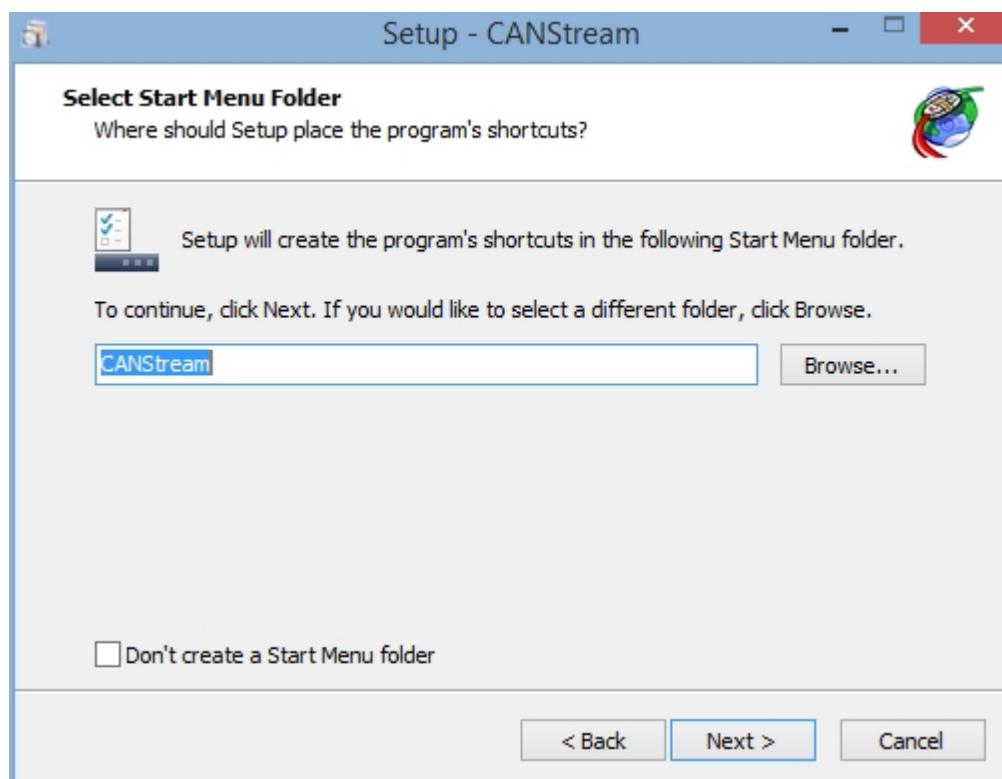
Click 'Next' to start the installation.



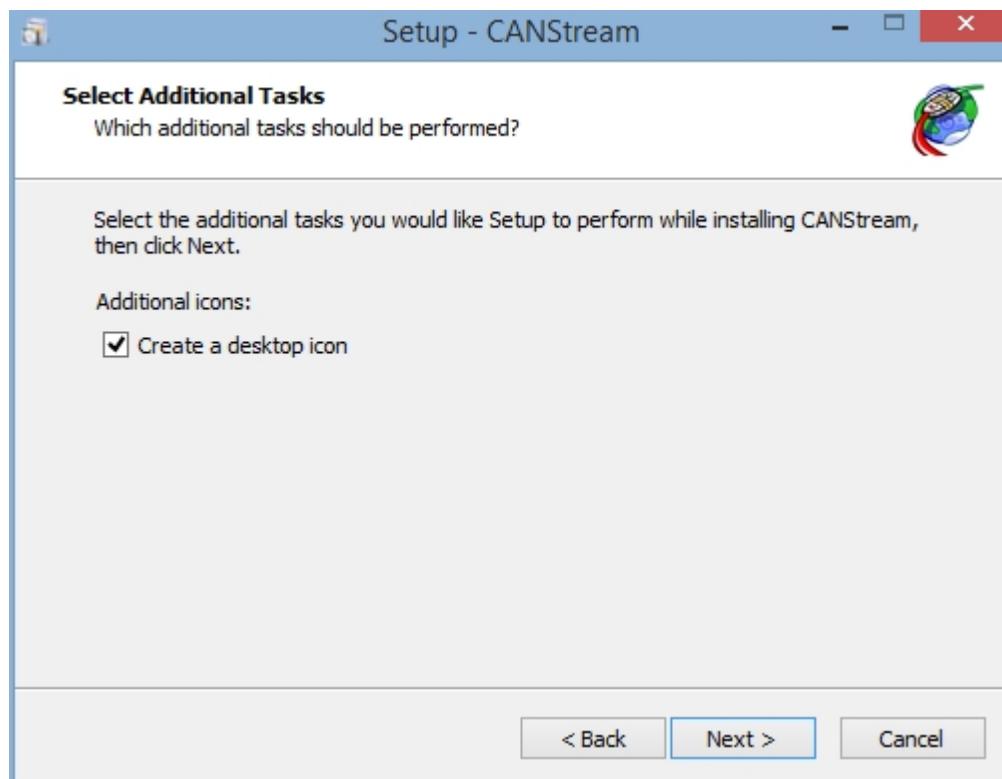
Click 'I accept the agreement' and then click 'Next'



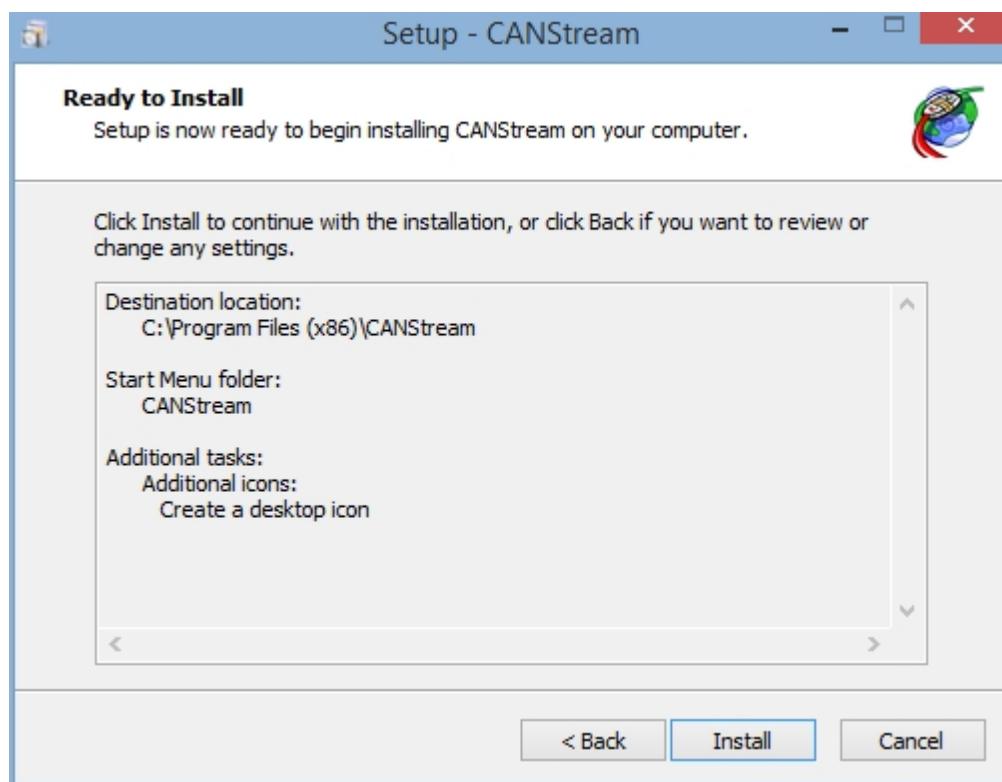
Check the installation folder, change it if you want by clicking the 'Browse..' button and then click 'Next'.



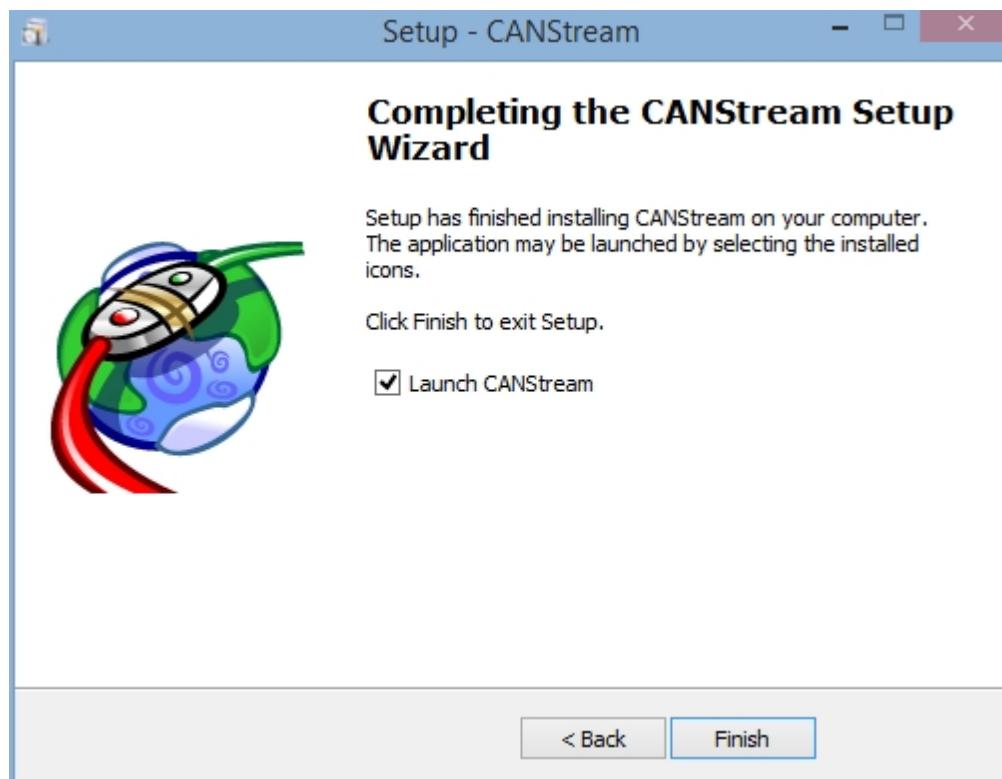
Check the name of the Start Menu folder that will be created and click 'Next'. If you don't want create any Start Menu folder, just tick the box 'Don't create a Start Menu folder' and then click 'Next'.



If you want create a desktop icon for CANStream, tick the box 'Create a desktop icon' and then click 'Next'.



If you are happy with all settings summarized, click 'Next' to start the actual installation of CANStream. Otherwise, click 'Back' to change those settings.



Here you are! CANStream has been installed. Just click 'Finish' to launch CANStream. If you don't want launch CANStream now, simply uncheck the 'Launch CANStream' box and click 'Finish' to exit the installation wizard.

Created with the Personal Edition of HelpNDoc: [Single source CHM, PDF, DOC and HTML Help creation](#)

Registration

blabla

Created with the Personal Edition of HelpNDoc: [Free help authoring tool](#)

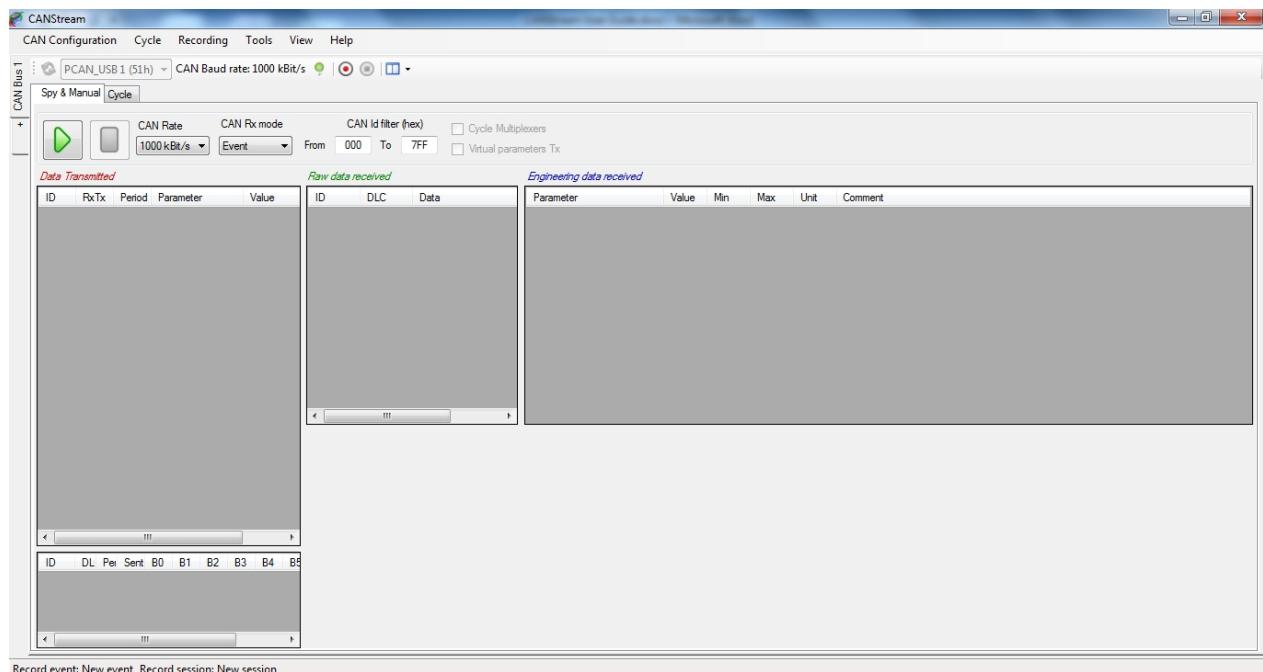
PEAK USB CAN adapter driver

blabla

Created with the Personal Edition of HelpNDoc: [Easily create PDF Help documents](#)

Main interface

At startup CANStream comes as the picture bellow.



The main CANStream interface is composed of different elements

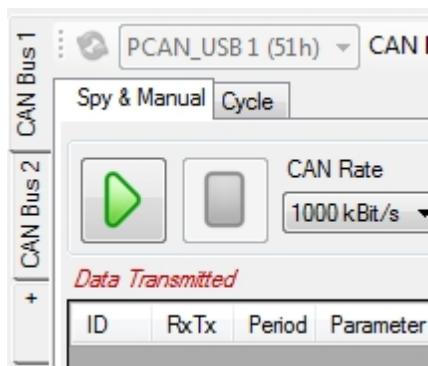
- CAN Controller panels using the most of the space
- The menu strip, on the top, regrouping all commands and tools accesses
- A status bar on the bottom showing different information such like the current record event and session, the current CAN configuration name and error flags

Created with the Personal Edition of HelpNDoc: [Free EBook and documentation generator](#)

CAN Controller panel

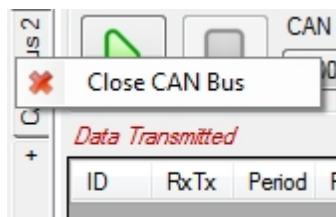
The CAN controller panel is where the CANStream magic happens!

There is one panel for each Peak PCAN-USB adapter you want to use. By default only one panel is opened, simply click the '+' tab to open a new panel and use an additional PCAN-USB adapter.



CANStream can handle up to eight PCAN-USB adapters, so it is possible to open a maximum of eight panels.

To close a panel and release its PCAN-USB adapter, stop the current panel operation (Manual control or Cycle player). Then right-click on the panel you want to close, and click the 'Close CAN Bus' command.

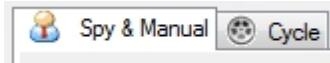


The CAN controller panel is composed of a tool bar and a multi-tab panel.

Created with the Personal Edition of HelpNDoc: [Produce Kindle eBooks easily](#)

CAN controller modes

A CAN controller has two operation modes, 'Manual & Spy' and 'Cycle'. Each mode has its own tab in the panel.



Created with the Personal Edition of HelpNDoc: [Easily create CHM Help documents](#)

Spy & Manual mode

The 'Spy & Manual' mode is intended to be used to spy the content of frames (or messages) circulating on the bus.

Using this mode, it also possible to send messages containing values set by the user.

Both reception and transmission handle 'Engineering' and 'Raw' formats of data. 'Engineering' means that user can read and write actual physical values of CAN signals while the 'Raw' format means raw byte values of the CAN messages.

ID	RxTx	Period	Parameter	Value	Unit	Comment
204	Tx	50	Signal_B1	0		
204	Tx	50	Signal_B2	0		
204	Tx	50	Signal_B3	0		
208	Tx	50	Virtual_Signal_B1	0		
208	Tx	50	Virtual_Signal_B2	0		
208	Tx	50	Virtual_Alarm_B1	0		
208	Tx	50	Virtual_Alarm_B2	0		
200	Rx	50	Signal_A1	0		
200	Rx	50	Signal_A2	0		
200	Rx	50	Signal_A3	0		
200	Rx	50	Virtual_Signal_A1	0		
200	Rx	50	Virtual_Signal_A2	0		
200	Rx	50	Virtual_Alarm_A1	0		
200	Rx	50	Virtual_Alarm_A2	0		

ID	Data	Period	Count

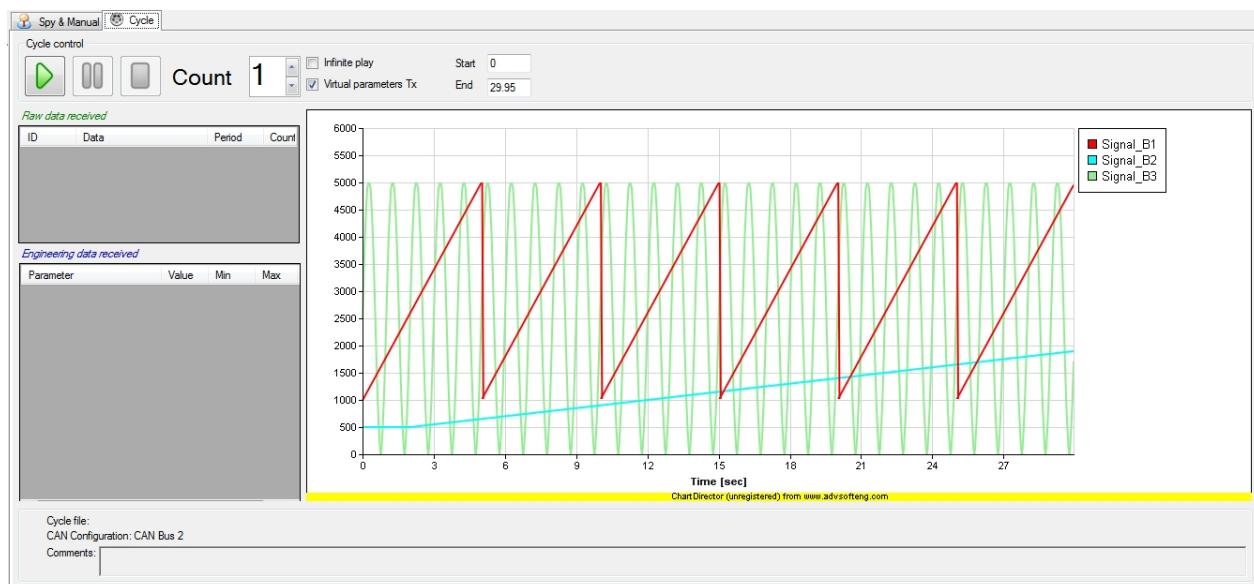
Parameter	Value	Min	Max	Unit	Comment

ID	DLC	Period	Sent	B0	B1	B2	B3	B4	B5	B6	B7

Created with the Personal Edition of HelpNDoc: [Full-featured Documentation generator](#)

Cycle mode

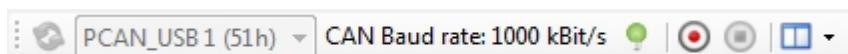
In 'Cycle' mode CANStream is sending a pre-defined sequence of message. This mode allows simulation of real usage conditions in which values have a very high dynamic that couldn't be replicated by a human.



Created with the Personal Edition of HelpNDoc: [Create HTML Help, DOC, PDF and print manuals from 1 single source](#)

CAN controller panel tool bar

The CAN controller panel tool bar contains common commands for both 'Spy & Manual' and 'Cycle' modes

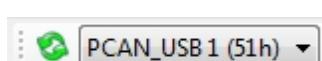


At the CANStream start up or on new controller opening, if a PCAN-USB adapter is connected and free (not used by another application) CANStream takes the control of this adapter and tool bar is as shown above.

By default the CAN Baud rate is set to 1000 kBit/s, however Baud rate can be changed in the CAN Configuration editor.

The 'Stop connection' button permits to stop the connection and to release the PCAN-USB adapter in order to make it available for another controller panel or another application. Click the 'Start connection' button to restart the connection using the same PCAN-USB adapter.

When the connection stops, 'Refresh PCAN USB Devices list' command and the PCAN USB device list are enabled.



Click the 'Refresh PCAN USB Devices list' button to refresh the 'PCAN USB Devices list'. Click in the list to select the PCAN-USB device that you want use.



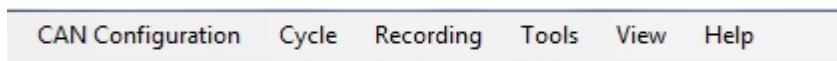
The 'Start stream recording' button starts the data recorder. When it is clicked, the 'Stop stream recording' button becomes enabled allowing the stop of data recording.

Command 'View' contains several sub-commands to customize the appearance of the 'Spy & Manual' panel. See the 'Manual control layout' section for more details.

Created with the Personal Edition of HelpNDoc: [Easily create PDF Help documents](#)

Menu strip

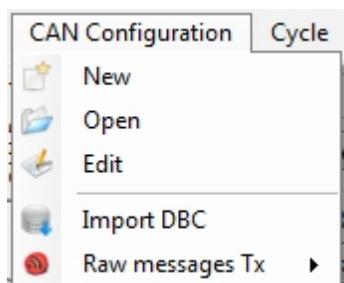
The menu strip gives access to all CANStream functions and tools.



Created with the Personal Edition of HelpNDoc: [Free help authoring tool](#)

CAN Configuration

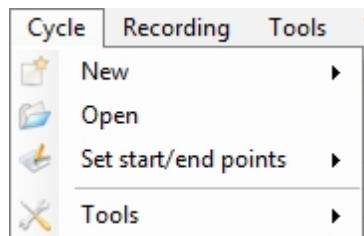
This menu contains all function related to the CAN configuration of the current controller.



- >New: Create a new CAN configuration.
- Open: Open a CAN configuration file.
- Edit: Edit the current CAN Configuration.
- Import DBC: Import a DBC file and use it as current CAN configuration.
- Raw messages Tx: Management of transmitted raw messages list.

Cycle

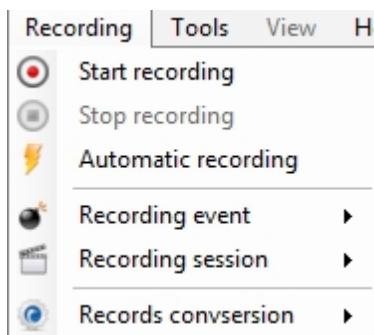
This menu contains all functions related to the cycle player.



- ☝ **New:** Create a new cycle. Check the 'Cycle creation' section for more details.
- 📁 **Open:** Load a cycle into the cycle player of the current CAN controller.
- 🔧 **Set start/end points:** Access to cycle starting and ending points control functions. Check the 'Cycle playing' section for more details.
- 🛠 **Tools:** Access to cycle creation tools. Check the 'Cycle creation' section for more details.

Recording

This menu contains all functions related to the data recorder.



Start recording: Start the data recorder.

Stop recording: Stop the data recorder.

Automatic recording: Enable or disable the automatic recording. Check the 'Data recording' section for more details.

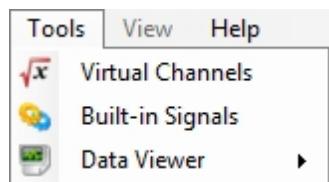
Recording event: Access to recording event control functions. Check the 'Recording event' for more details.

Recording session: Access to recording session control functions. Check the 'Recording session' for more details.

Recording conversion: Access to recording conversion control function. Check "Recording conversion" for more details

Tools

This menu contains all CANStream tools access



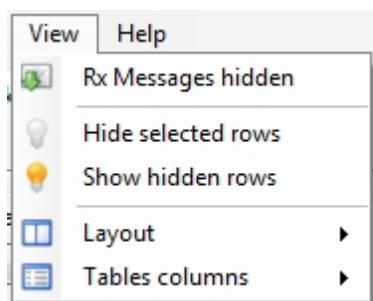
Virtual Channels: Open the virtual channels editor

Built-in Signals: Open the built-in signals editor

Data Viewer: Open the data viewer

View

This menu contains layout configuration functions for the manual mode of the current CAN controller.



Rx Messages hidden: Hide Rx messages in the transmission panel

Hide selected rows: Hide the selected rows of the active grid

Show selected rows: Show hidden rows of the active grid

Layout: Access to the control layout configuration functions.

Tables columns: Access to the grid columns configuration functions

CAN Frames configuration

You can use CANStream to send and receive CAN messages in raw format (hexadecimal byte values) but if you want use all features of CANStream and get the maximum out of it (and I'm sure you do), the first thing you have to do is to create a CAN configuration file.

A CAN configuration file contains the description of all messages (frames) circulating on the CAN bus and all properties of all parameters (nodes) for each message. Such configuration doesn't need to be exhaustive, you can setup only messages and parameters which are of your interest and simply ignore those you don't want work with.

CANStream actually handles two kinds of CAN configuration file: single bus configuration and multiple buses configuration.

A single bus configuration simply contains descriptions and properties of CAN messages and parameters for a single CAN bus. This the configuration you have to use if you are using only one Peak PCAN-USB adapter.

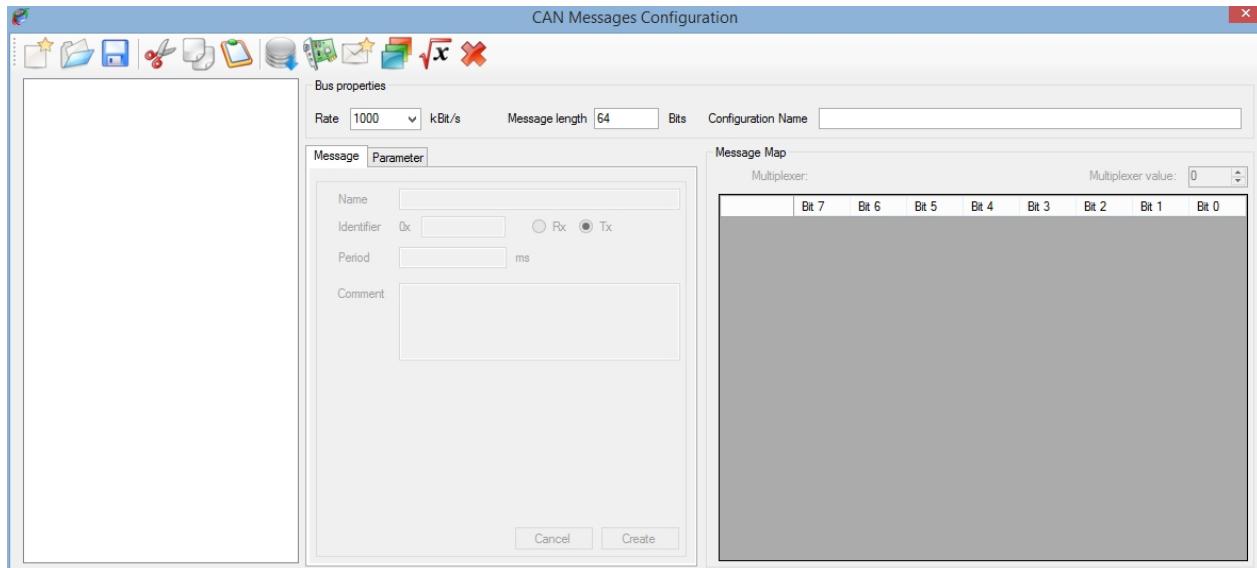
A multiple buses configuration does the same but it can handle up to height CAN buses at the same time. This is the kind of configuration which is intended to be used when you are using simultaneously two, three or more (up to eight) Peak PCAN-USB adapters.

Created with the Personal Edition of HelpNDoc: Produce electronic books easily

Single bus configuration

To create a CAN configuration, click on the 'CAN Configuration' menu of the menu strip and then click on 'New'.

The CAN configuration editor should appear as bellow.



Use button 'New CAN configuration' to create a new CAN configuration.

Buttons 'Open CAN configuration' and 'Save CAN configuration' permit to edit an existing CAN configuration and save the current configuration in a file (*.xcc)

Created with the Personal Edition of HelpNDoc: [Generate EPub eBooks with ease](#)

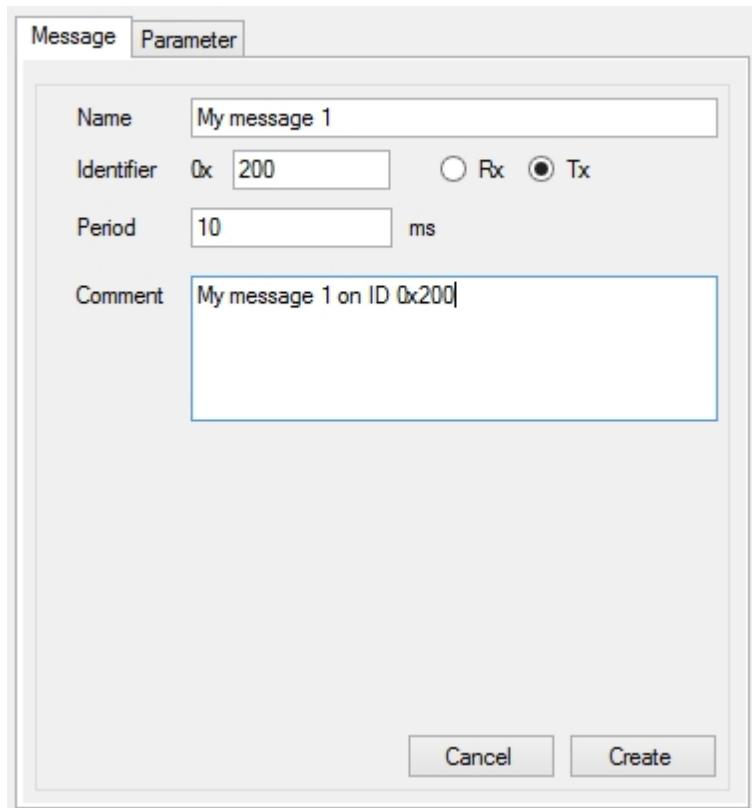
Bus properties

First, set the CAN bus properties in the 'BUS properties' panel. Most important properties here are bus rate and message length. By default those properties are set to 1000 kBit/s for the rate and 64 bits (8 bytes) for the message length. Optionally, you can give a name to your configuration.



CAN message properties

Click on the 'New message'  button of the tool bar to create a new CAN message.



Most important CAN message properties are its identifier (ID) which must be an hexadecimal value. The message direction, 'Rx' whether the message is received by CANStream or 'Tx' whether it is sent by CANStream. Period in millisecond to indicate the frequency of the message reception or transmission.

You may also give a name and add a comment to your message but those are not mandatory.

Then click the 'Create' button to add your new message into the current configuration.

Created with the Personal Edition of HelpNDoc: [Easily create PDF Help documents](#)

CAN signal properties

Once your message created, add some parameters (nodes) to it by clicking the 'New parameter' button



Message **Parameter**

Name	<input type="text" value="Parameter 1"/>		
Start bit	<input type="text" value="7"/>	Length	<input type="text" value="16"/>
Endianess	<input type="button" value="MSBFirst"/>	<input type="checkbox"/> Signed	
Gain	<input type="text" value="1"/>	Zero	<input type="text" value="0"/>
<i>Linearization from raw to engineering value Eng = Raw x Gain + Zero</i>			
Unit	<input type="text"/>		
Comment	<input type="text" value="First parameter of My message 1"/>		

Multiplexed parameter

Multiplexer parameter	<input style="width: 150px; height: 20px; border: none; background-color: #f0f0f0; border: 1px solid #ccc; padding: 2px 5px;" type="button"/>
Multiplexer Value	<input style="width: 150px; height: 20px; border: none; background-color: #f0f0f0; border: 1px solid #ccc; padding: 2px 5px;" type="button"/>

A CAN parameter has several important properties that need to be set with attention.

Start bit: The index of the bit in the frame where the parameter starts. Whether parameter endianess is MSB or LSB (see below for details) start bit will be different.

Length: The number of bits use by the parameter in the frame.

Endianess: How the parameter engineering (or physical) value is encoded inside its bits. There two different endianesses, MSB first (Most significant byte first) also known as Motorola format and LSB first (Least significant byte first) also known as Intel format.

There are tons of documentation available on the web describing in great details the difference between those two formats. Basically, for a given value, let's say 50 000, its encoded value will be c350 in MSB format, while it will be 50C3 in LSB format.

Signedness: Indicating whether parameter sign (positive or negative) is considered for the parameter value encoding.

Gain and Zero: Those are values used to scale the parameter value from the raw to the engineering format

Name, unit and comment: Those properties are not essential but it may ease parameters identification and

value exploitation.

Multiplexed parameter: Use those settings if you want multiplex your message. See the '[Multiplexed parameter](#)' section for more details.

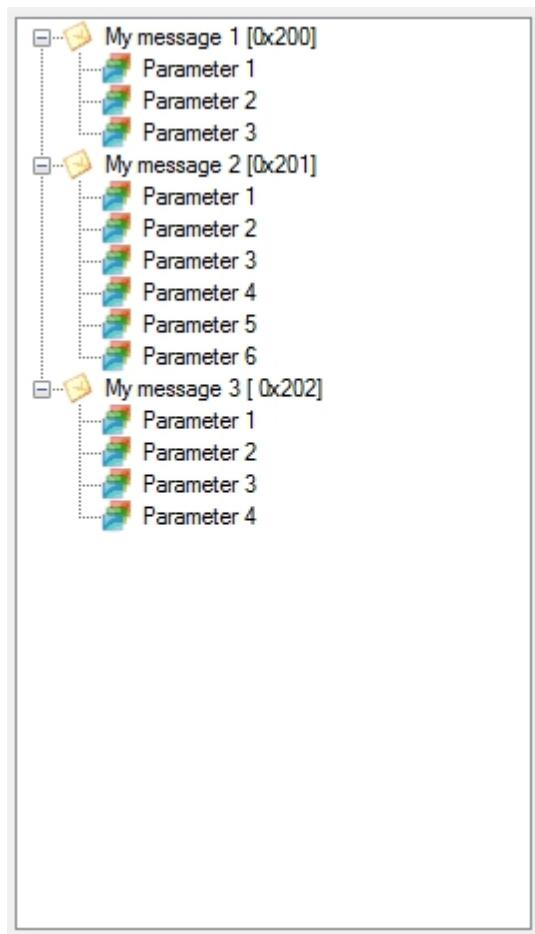
Once all properties set, click the 'Create' button to add the parameter to the current message.

CAN Configuration tree view

The CAN configuration tree view, on the left of the CAN configuration editor, shows all messages and all parameters of each message in a tree form.

Each message is represented by a tree node and all its parameters are a child node (or a branch) of this message node.

Click on a message or a parameter node to display its properties. Those properties can also be modified. Select the element that you want to modify, make all changes you need and then click on the 'Modify' button.



Using 'Cut' , 'Copy' and 'Paste' commands of the tool bar, you can duplicate or move messages and parameters inside the configuration.

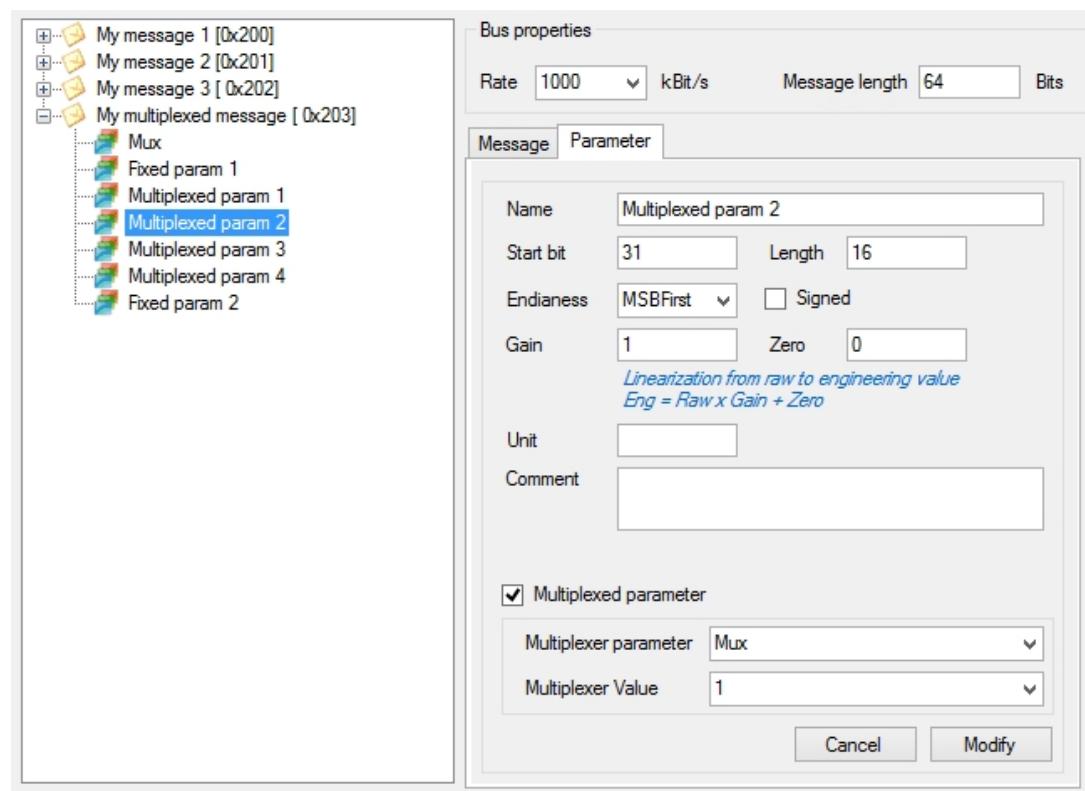
The 'Delete selection' button allows to delete a message or a parameter.

Multiplexed parameter

Once you have at least two parameters in a message, one of those parameters, the multiplexer, can be used to multiplex other parameters of the message.

As we saw in the '[CAN signal properties](#)' section, each parameter uses a range of bits inside the message frame. In a multiplexed message, the same bit range of the frame, or a part of it, can be shared between multiple parameters. In other words, a single message can have different parameters arrangements, current arrangement being defined by the current value of the multiplexer.

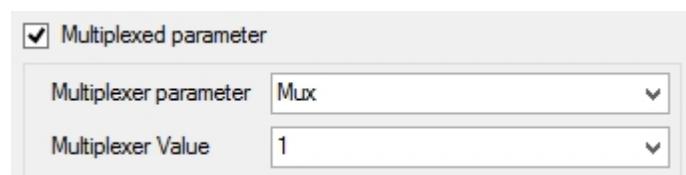
The multiplexer must be unique within a message while there could be bunches of multiplexed and fixed parameters.



On the example above, parameters '*Multiplexed param x*' are sharing the same range of bits within the message '*My multiplexed message*', multiplexer of this message being the '*Multiplexer*' parameter.

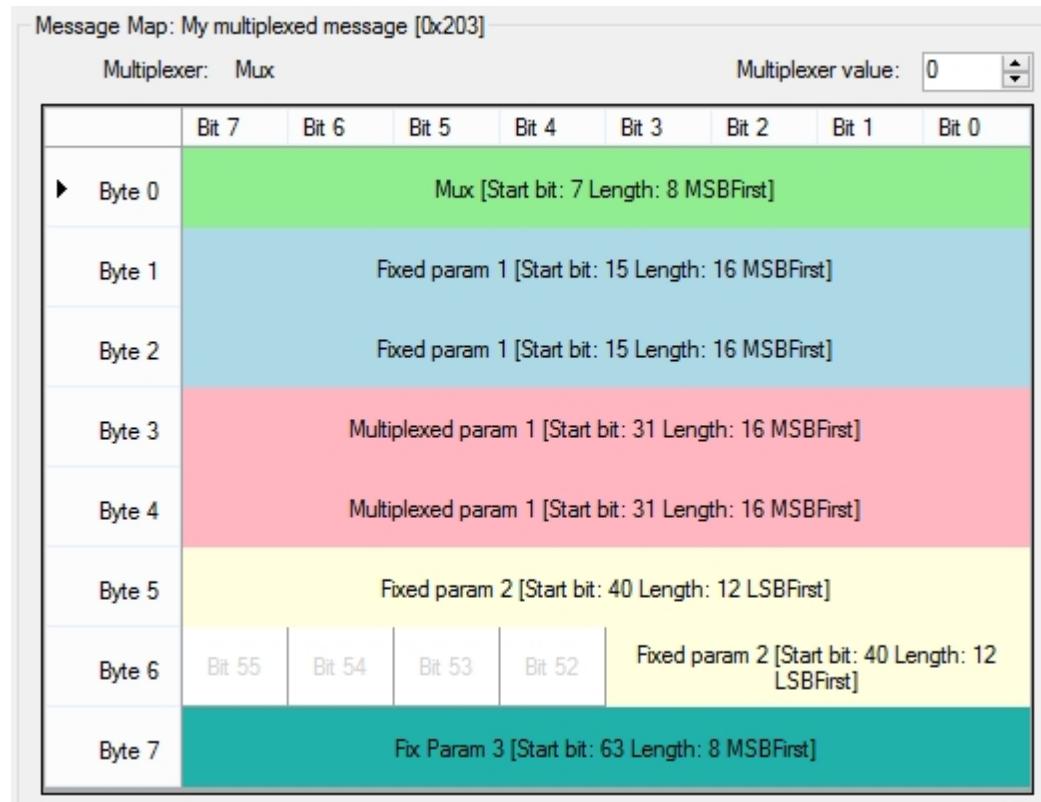
To create multiplexed parameters inside a message, in first create a fixed parameter (a parameter which is not multiplexed and which will be present in all arrangements of the message). Then create a second parameter. In the properties of your second parameter, tick the 'Multiplexed parameter' box. Lists 'Multiplexer parameter' and 'Multiplexer value' are now active.

Select the multiplexer parameter in the 'Multiplexer parameter' list and then select which of its value is making your parameter active in the 'Multiplexer value' list.



Message map

The message map is a grid that shows in graphical way the bit mapping of parameters of a given CAN message.



Each row of the grid represent a single byte of the message while each column is the bit number for each byte.

Inside this grid, parameters are represented by colored area. Most important properties of a parameter are written inside its area.

Click on a parameter to display or edit all its properties.

For multiplexed messages, change the multiplexer value to see the message map for any multiplexer value.

Virtual CAN signal

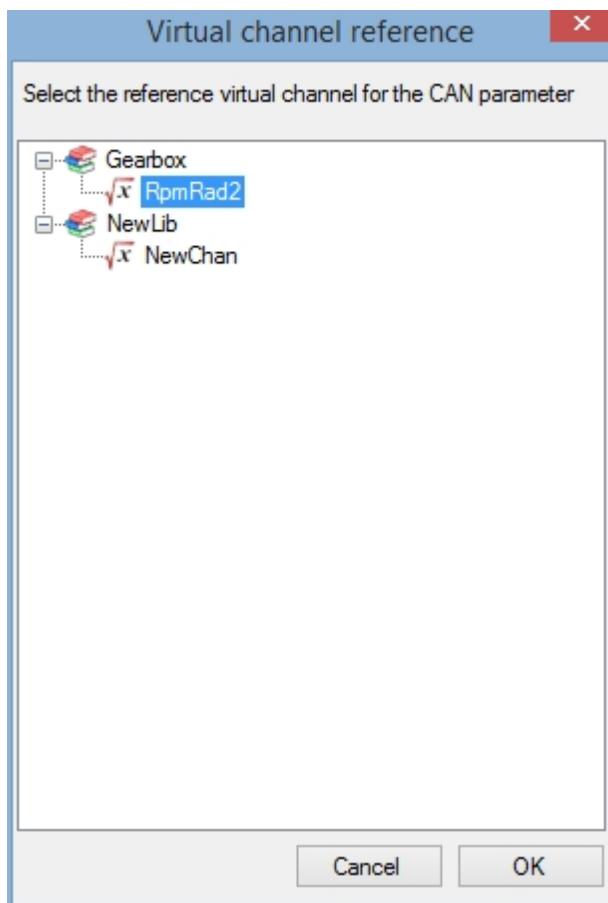
Basically, a virtual CAN signal is a virtual channel embedded into a [CAN message](#).

A standard [CAN signal](#) can be used either in manual TX or RX mode in a cycle. For transmission (TX) it uses a static value defined by the user. In reception (RX) it simply gets the value received.

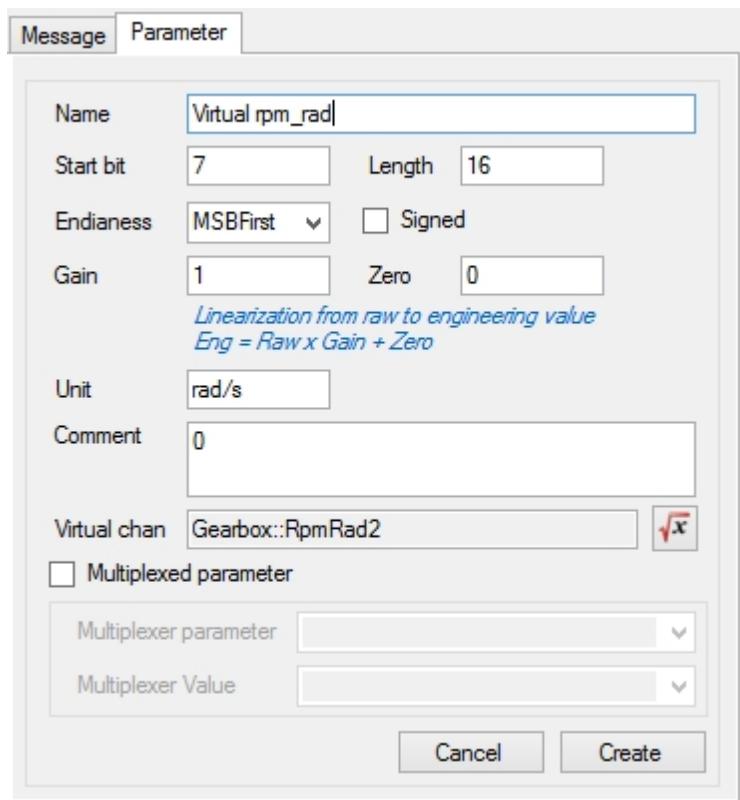
Value of a virtual CAN signal is neither defined by the user or received ! The value is computed real time using other parameter values. See the 'Virtual channel' section for more details.

Click the 'New virtual parameter' button  to create a virtual CAN signal.

The virtual channels selection window should appear.



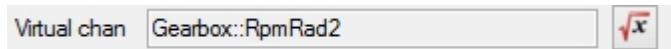
Pick up the virtual channel that you want to add in your message and click 'OK'.



Configure your virtual signal the same way you would do for a [standard parameter](#) and then click the 'Create' button.

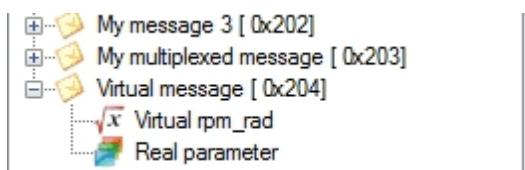
Note that compared to standard parameter, for virtual CAN signal the section 'Virtual chan' appeared. This section shows the virtual channel used as source of the virtual signal.

The source is indicated using the following format: 'XXX:YYY' where 'XXX' is the name of the library hosting your virtual channel and 'YYY' is the name of the virtual channel.



Click on the button to change the virtual source.

In order to be easily identified in the CAN configuration [tree view](#), virtual CAN signal are marked with the virtual channel icon .



Created with the Personal Edition of HelpNDoc: [Write eBooks for the Kindle](#)

Vector CAN data base (dbc) import

A Vector CAN data base file (extension *.dbc) is, at the end of day, a Vector CAN configuration. By the way, Vector is the editor of the famous tool CANalyser, one the most used (if not the most) tool in the CAN engineering world.

For more detail about CANalyser and DBC file, please visit http://vector.com/i_canalyzer_en.html

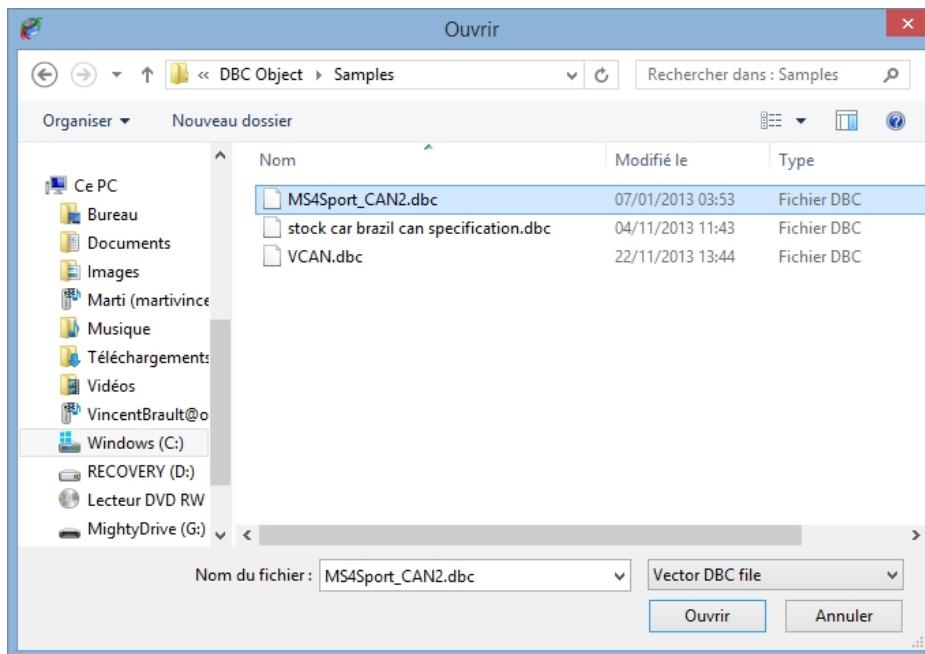
As per CANStream a DBC file contains CAN message and parameters description

To save the pain of creating a CANStream CAN configuration file from scratch if you have already a DBC, CANStream can import it and convert it to its own format.



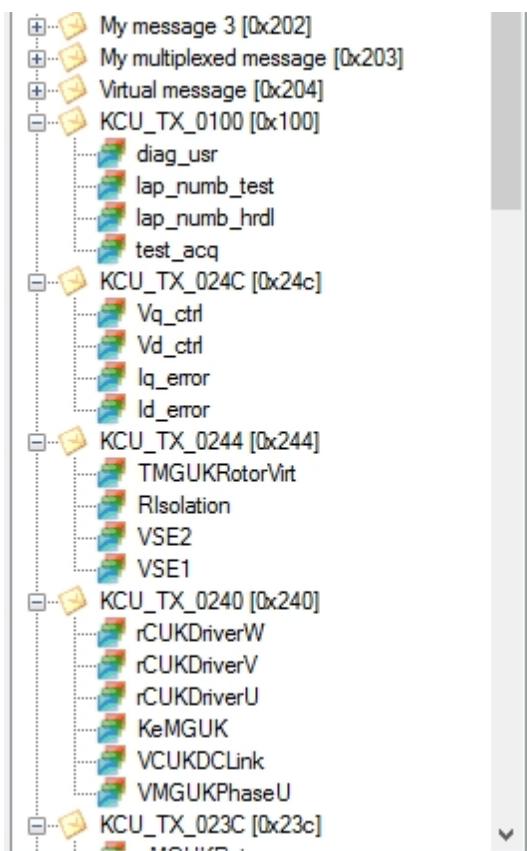
Do to so, click the 'Import DBC' button .

Select the DBC file to import and click 'Open'



All messages contained in the DBC as well as all their parameters are going to be added to the current CAN configuration.

Note that the 'DBC import' function doesn't reset the current CAN configuration, it simply add messages coming from the DBC.



Created with the Personal Edition of HelpNDoc: Write EPub books for the iPad

Multiple buses configuration

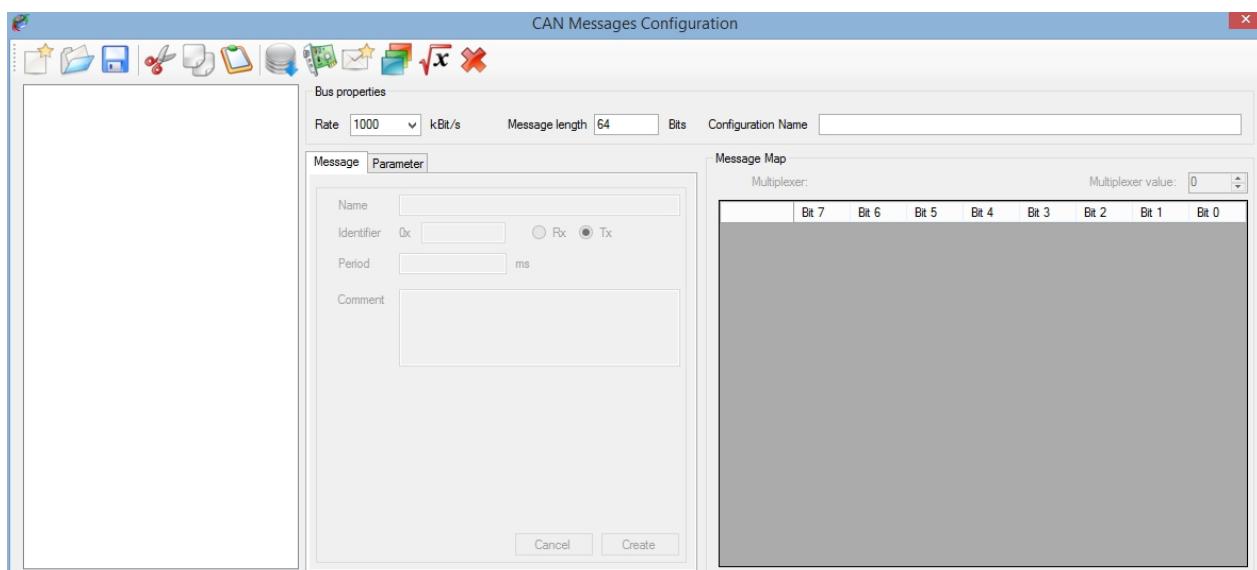
As we saw in the [previous section](#), a CAN configuration contains description of CAN messages and parameter for a single CAN bus.

In the CANStream world, a CAN bus is considered as a Peak PCAN-USB adapter. One CAN bus = one PCAN-USB.

Since CANStream can handle up to eight PCAN-USB adapters, it can run simultaneously up to eight CAN configurations. Those multiple configurations (or buses) can be grouped in a single file in order to have the user configuring its buses once and using and reusing them with a simple command.

To create a multiples buses CAN configuration, click on the 'CAN Configuration' menu of the menu strip and then click on 'New'.

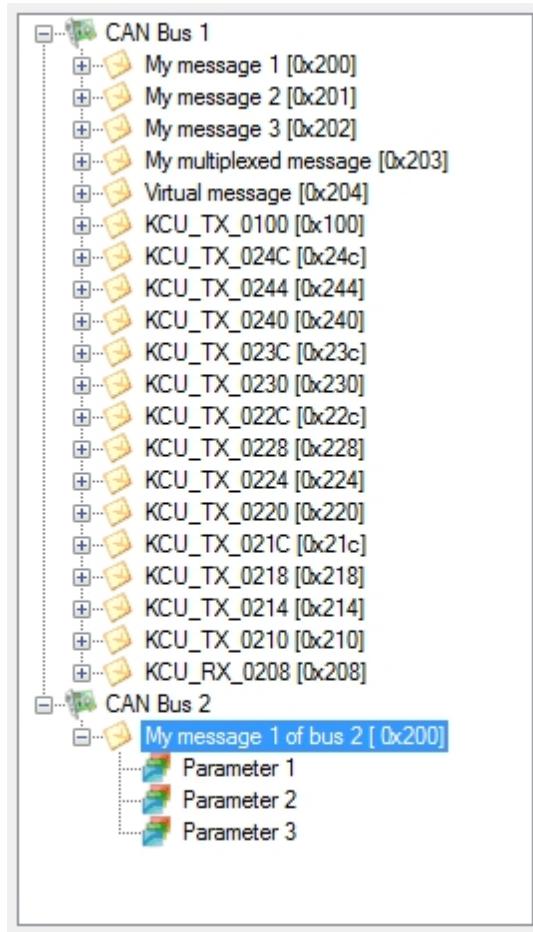
The CAN configuration editor should appear as bellow.



Click the 'New CAN bus controller' button  to create a new CAN Bus

Then, add [messages](#) and [parameters](#) using the same method you would use for a single bus configuration.

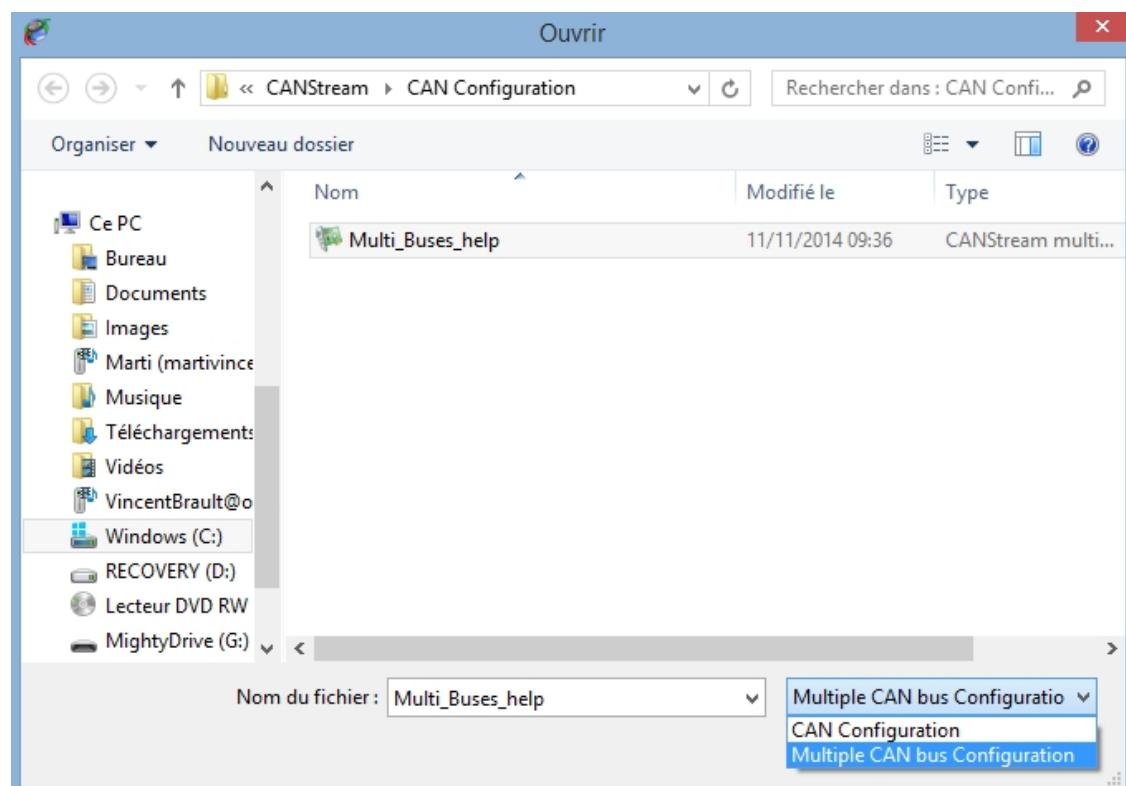
You don't need to create such configuration from scratch. You can open  an existing single bus configuration or [import a DBC file](#)  first and then create all buses on the top of it. On the 'New CAN bus controller' button  click, CANStream will automatically create a fist CAN bus for existing messages and then create a second CAN bus in which you can add messages and parameters.



In multiple CAN buses mode, each bus is represented in the configuration [tree view](#) by the bus controller icon . Then CAN messages of a bus become child nodes (or branches) of the CAN bus. Multiple Buses CAN configuration file has a specific extension (*.mcb). Use standard 'Save CAN

configuration' button to save your multiple buses configuration, CANStream will automatically switch to the *.mcb' extension.

When you are using the 'Open CAN configuration' command , the file opening dialog has two file extension filters: *.xcc' for single bus configuration and *.mcb' for multiple buses configuration. Just switch to the *.mcb' extension to open a multiple buses configuration and load it into the CAN configuration edition form.

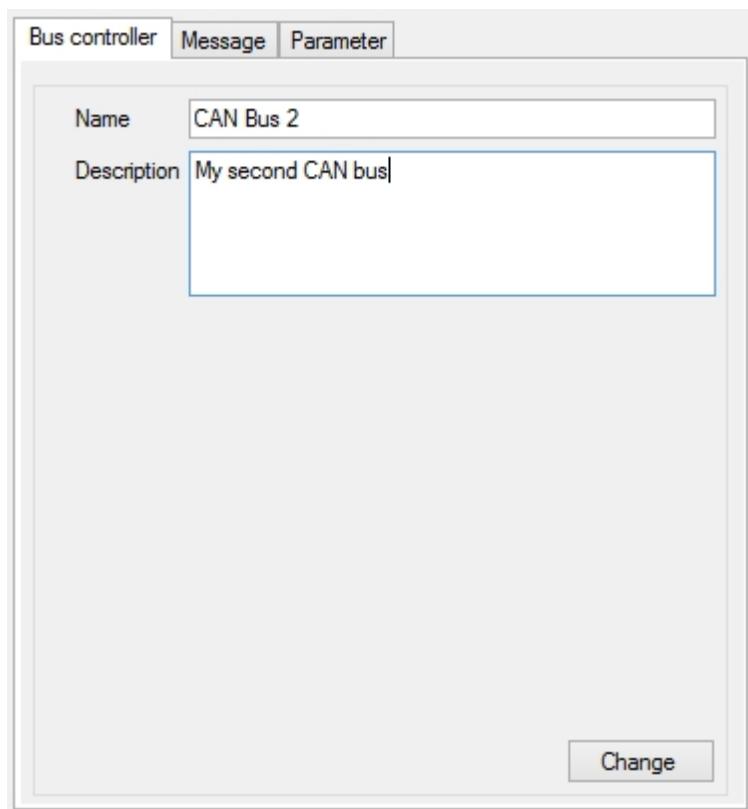


Created with the Personal Edition of HelpNDoc: [Full-featured EPub generator](#)

CAN Bus information

While working with multiple buses CAN configuration, a third tab 'Bus controller' appears in the edition form in addition of 'Message' and 'Parameter' tabs.

Click on a CAN bus node  of the CAN configuration [tree view](#) to edit its properties.



In addition of [standard CAN bus properties](#), in that tab, you can add few information about the current bus.

Name: To give a sensible name to your bus

Description: You can write a short description of the bus in order to ease usage of multiple buses configuration.

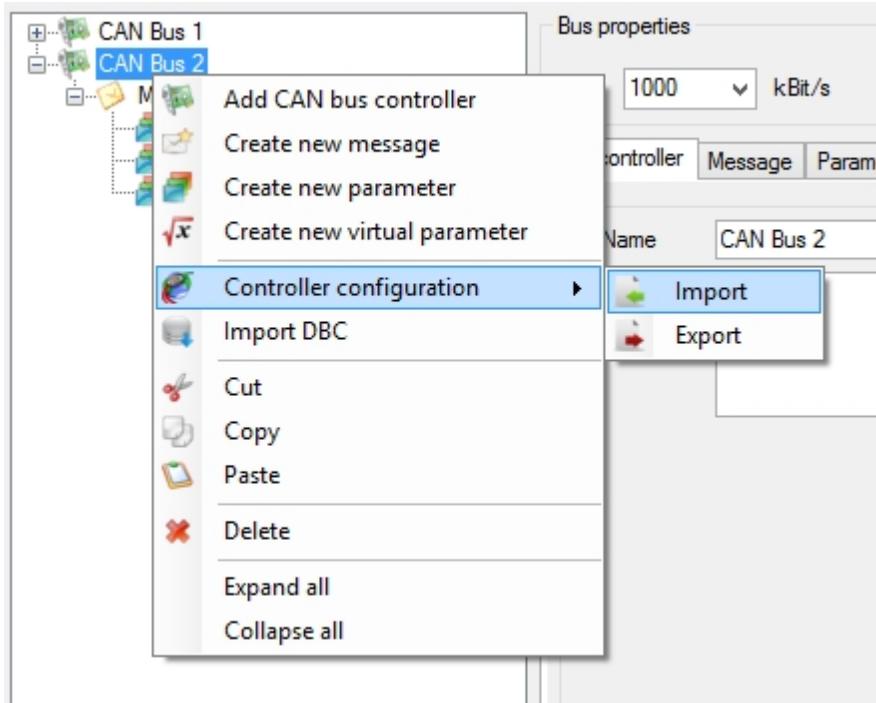
Then, click the 'Change' button to apply your changes.

Created with the Personal Edition of HelpNDoc: [Full-featured Help generator](#)

Configuration import & export

While creating a multiple buses CAN configuration, using the 'CAN Configuration\Import' command , CAN stream offers the possibility to use an existing single bus CAN configuration (*.xcc) for the current CAN bus.

Simply right click on the CAN bus node  of the [tree view](#) that you want to set, select the 'CAN Configuration' item  and then the 'Import' command  . A file opening dialog will pop up in which you can choose the CAN configuration file to import.



On the other way around, the 'Export' command  permits to export the CAN configuration of a particular bus in order to reuse it as a single CAN bus configuration.

Created with the Personal Edition of HelpNDoc: [Full-featured Help generator](#)

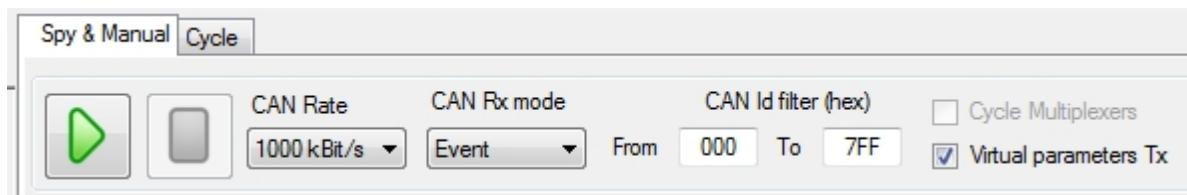
Manual mode

The 'Spy & Manual' mode is intended to be used to spy the content of frames (or messages) circulating on the bus.

Using this mode, it also possible to send messages containing values set by the user.

Both reception and transmission handle 'Engineering' and 'Raw' formats of data. 'Engineering' means that user can read and write actual physical values of CAN signals while the 'Raw' format means raw byte values of the CAN messages.

The 'Spy & Manual' mode is controlled through the tool bar of the 'Spy & Manual' panel.



This tool bar contains the following elements



Start button: Start data transmission and reception



Stop button: Stop data transmission and reception

CAN Rate

1000 kBit/s

CAN rate selection: Allows definition of the CAN communication speed.

CAN Rx mode

Event

Rx mode selection: Allows selection of the message reception mode (event, manual, periodic)

CAN Id filter (hex)

From

000

To

7FF

Rx IDs filter: Allows the definition of the range of CAN message IDs that have to be considered in the reception panel.

Virtual parameters Tx

[Virtual CAN signal](#) Transmission enabled flag: Enable and disable the transmission of virtual CAN signals

Cycle Multiplexers

[Multiplexer](#) auto-cycling flag: If the CAN configuration in use contains some multiplexed message, CANStream will automatically cycle the multiplexer value for data transmission.

Data Tx

The 'Start' button  of the tool bar will be enabled only if an operational PCAN-USB device has been selected.

Click that button to start the data transmission. While the manual mode is operating, 'Start' button  is disabled. Click the 'Stop' button  to stop the manual mode operation and consequently the data transmission.

Those commands control both transmission and reception parts of the 'Spy & Manual' mode.

Transmission panel of 'Spy & Manual' control is divided in two parts: 'Engineering' on the top and 'Raw' data transmission at the bottom.

Data Transmitted						
ID	RxTx	Period	Parameter	Value	Unit	Comment
200	Tx	10	Muxer	0		
200	Tx	10	Pemanent	0		
200	Tx	10	Mux_0_Val_1	0		
200	Tx	10	Mux_0_Val_2	0		
200	Tx	10	Mux_0_Val_3	0		
200	Tx	10	Mux_1_Val_1	0		
200	Tx	10	Mux_1_Val_2	0		
200	Tx	10	Mux_1_Val_3	0		
200	Tx	10	Mux_2_Val_1	0		
200	Tx	10	Mux_2_Val_2	0		
200	Tx	10	Mux_2_Val_3	0		
204	Tx	1000	Parameter 1	0		

ID	DLC	Period	Sent	B0	B1	B2	B3	B4	B5	B6	B7
400	8	0	<input checked="" type="checkbox"/>	00	01	02	03	04	05	06	07
4AC	8	0	<input checked="" type="checkbox"/>	AA	FA	D3	6F	BE	15	ED	CE

Created with the Personal Edition of HelpNDoc: [Single source CHM, PDF, DOC and HTML Help creation](#)

Engineering format data Tx

In order to send engineering data over the CAN bus a CAN configuration file has to be loaded. Once loaded, this CAN configuration is displayed in the 'Engineering data transmission' grid.

Simply change the content of the 'Value' cell to send a new value.

ID	RxTx	Period	Parameter	Value	Unit	Comment
200	Tx	10	Muxer	1		
200	Tx	10	Pemanent	5878		
200	Tx	10	Mux_0_Val_1	12		
200	Tx	10	Mux_0_Val_2	1		
200	Tx	10	Mux_0_Val_3	21		
200	Tx	10	Mux_1_Val_1	48		

Virtual data Tx

Virtual CAN signal appear with a green color into the grid.

ID	RxTx	Period	Parameter	Value	Unit	Comment
400	Tx	1000	VDC_Alarm_Tx	0		
400	Tx	1000	StdParam_1	0		
400	Tx	1000	Gen_Alarm_Tx	0		
400	Tx	1000	StdParam_2	0		
400	Tx	1000	Parameter 5	0	kW	

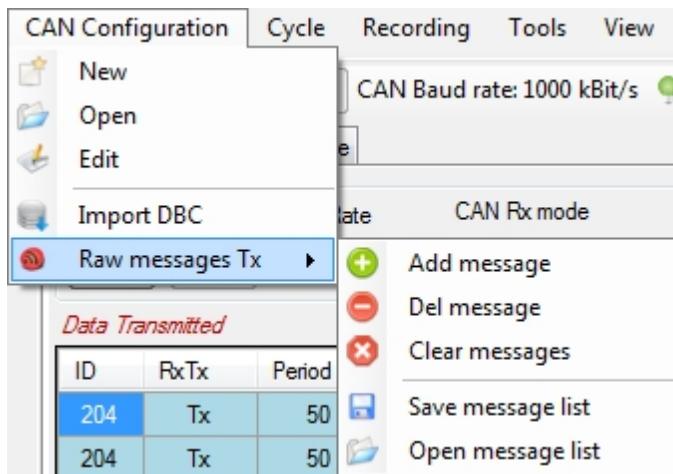
Since the value of a virtual CAN signal is calculated, its value cannot be set by the user.

Virtual CAN signal transmission can be turned on and off at any time, checking or unchecking the box

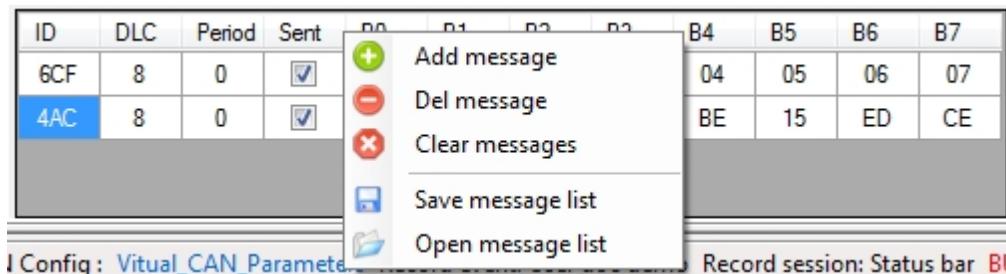
 Virtual parameters Tx

Raw format data Tx

To send 'Raw' CAN messages over the bus, you first need to define those messages. Click the 'CAN Configuration\ Raw messages Tx' menu to get the raw messages management functions.



Alternatively, right click in the raw message grid to open the creation menu.



This contextual menu contains a bunch of commands to manage the raw message list.

- ⊕ **Add message:** Add a message in the list
- ⊖ **Del message:** Remove a message from the list
- ✖ **Clear messages:** Remove all messages of the list
- 💾 **Save:** Save the message list in a file
- 📁 **Open:** Open a message list file.

Once messages have been loaded or created it is possible to set their properties in the different cells of the grid

ID	DLC	Period	Sent	B0	B1	B2	B3	B4	B5	B6	B7
6CF	8	0	<input checked="" type="checkbox"/>	00	01	02	03	04	05	06	07
4AC	8	0	<input checked="" type="checkbox"/>	AA	FA	D3	6F	BE	15	ED	CE

ID: Message identifier

DLC: Message byte length

Period: Message transmission period in millisecond

Sent: Define whether or not the message is sent

B0... B7: Message bytes value

If the property 'Period' is set to zero, message is not sent over.

Data Rx

The 'Start' button  of the tool bar will be enabled only if an operational PCAN-USB device has been selected.

Click that button to start the data reception. While the manual mode is operating, 'Start' button  is disabled. Click the 'Stop' button  to stop the manual mode operation and consequently the data reception.

Those commands control both transmission and reception parts of the 'Spy & Manual' mode.

Created with the Personal Edition of HelpNDoc: [Full-featured multi-format Help generator](#)

Engineering format data Rx

In order to decode data received over the CAN bus, a CAN configuration file has to be loaded. Then when CAN frames are received they are decoded and signals are shown in the 'Engineering data received' grid.

Engineering data received

Parameter	Value	Min	Max	Unit	Comment
Signal_B1	4560	1040	5000		
Signal_B2	1122	500	1122		
Signal_B3	3300	0	5000		
Virtual_Signal_B1	0	0	0		
Virtual_Signal_B2	0	0	0		
Virtual_Alarm_B1	1	0	1		
Virtual_Alarm_B2	0	0	0		
Virtual_A1	2841	867.5	2907.5		
Virtual_A2	1650.5	0.5	2500.5		
Virtual_Alarm_A2	1	0	1		
Virtual_Alarm_B2	1	0	1		

Virtual data Rx

If one or more virtual channels libraries are loaded and data received are containing signal used in those virtual channels, virtual channels are automatically computed and their values are shown into the grid with green background color.

Created with the Personal Edition of HelpNDoc: [iPhone web sites made easy](#)

Raw format data Rx

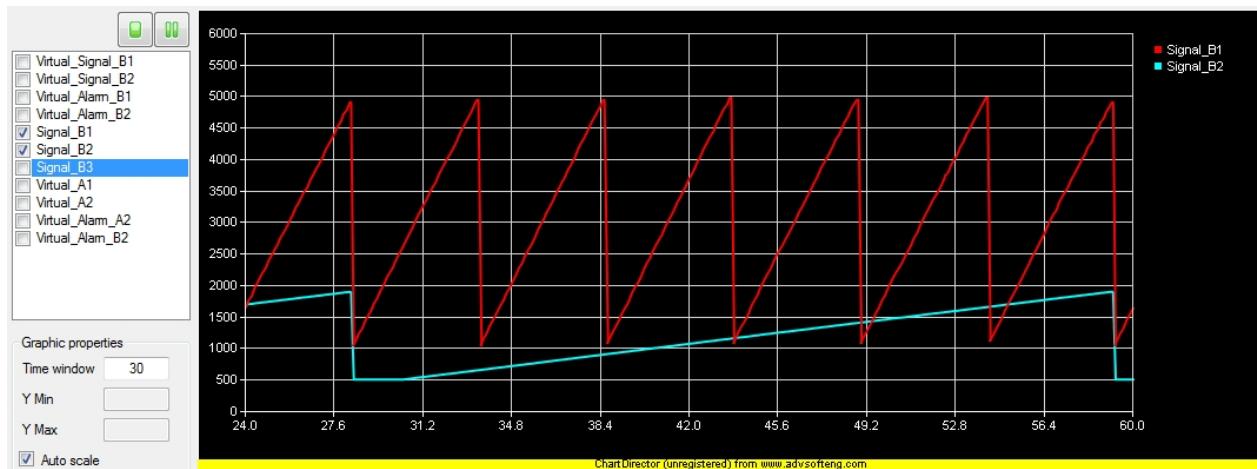
Received frames raw values (before decoding) are shown into the 'Raw data received' grid.

Raw data received			
ID	Data	Period	Count
204h	11 D0 04 62 21 00 00 00	49.6	290
208h	00 00 00 00 00 00 00 10	50.0	286

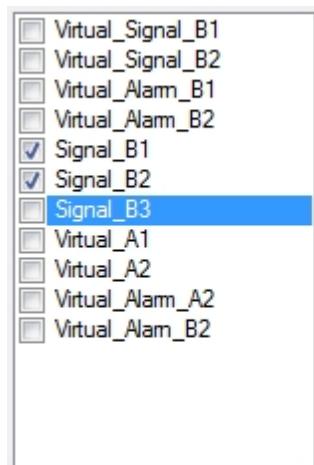
Created with the Personal Edition of HelpNDoc: [Easily create Web Help sites](#)

Rx data graphic

When CAN frames received contains data that can be decoded using the loaded CAN configuration, values of CAN signals received can be traced in real time into a graphic.



The graphic shows CAN parameters values as a function of the time. By default, the graphic is empty, simply select data to trace by checking channels in the channels list.



Using the small toolbar on the top of the channel list, it is possible to stop or freeze the graphic.



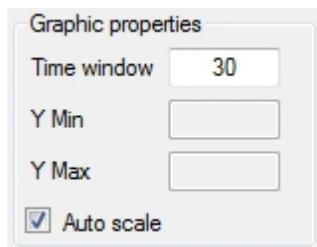
Button stops the real time graphic plotting. Once clicked this button becomes 'Start plot' button . Those two commands actually reset the graphic window so all data plotted will be lost after a 'Stop/Start' commands.

Button freezes the real time graphic plotting. Once clicked this button becomes 'Resume' button .



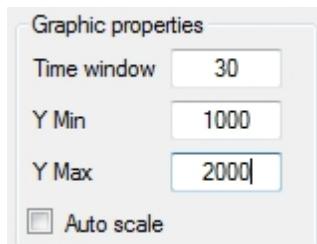
. Unlike the 'Stop' button, 'Freeze' command doesn't reset the graphic. On graphic resuming, graph will appear as if it never stopped before.

Graphic time window and Y axis scale can be changed through the 'Graphic properties' panel.



Change the 'Time window' field to enlarge or reduce timespan of the graphic.

By default Y axis of the graphic is set to have all data visible. Uncheck the 'Auto scale' box and set your own scale in 'Y Min' and 'Y Max' fields to have a fixed scale.



While modifying those parameters, you have to press the 'Enter' key in order to apply your modification.

Created with the Personal Edition of HelpNDoc: [Single source CHM, PDF, DOC and HTML Help creation](#)

Cycle mode

The 'Cycle' mode of CANStream sends a pre-defined sequence of message. This mode allows simulation of real usage conditions in which values have a very high dynamic that couldn't be replicated by a human.

The 'Cycle' mode is controlled through the tool bar of the 'Cycle' panel.



This tool bar contains the following elements



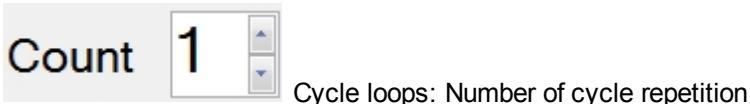
Start button: Starts cycle play



Pause button: Makes a pause in the cycle.



Stop button: Stop cycle play



Cycle loops: Number of cycle repetition

Infinite play: Infinite repetition: If that box is checked, cycle is in definitively repeated until the 'Stop' button is clicked.

Virtual parameters Tx Virtual CAN signal sent: If that box is checked, virtual CAN signal values are sent along cycle data.

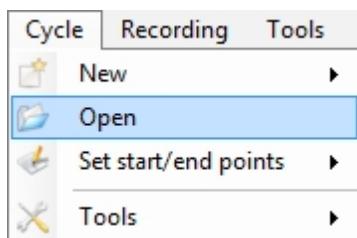
Start Cycle starting point: Defines the cycle play starting time for partial cycle playing.

End Cycle ending point: Defines the cycle play ending time for partial cycle playing.

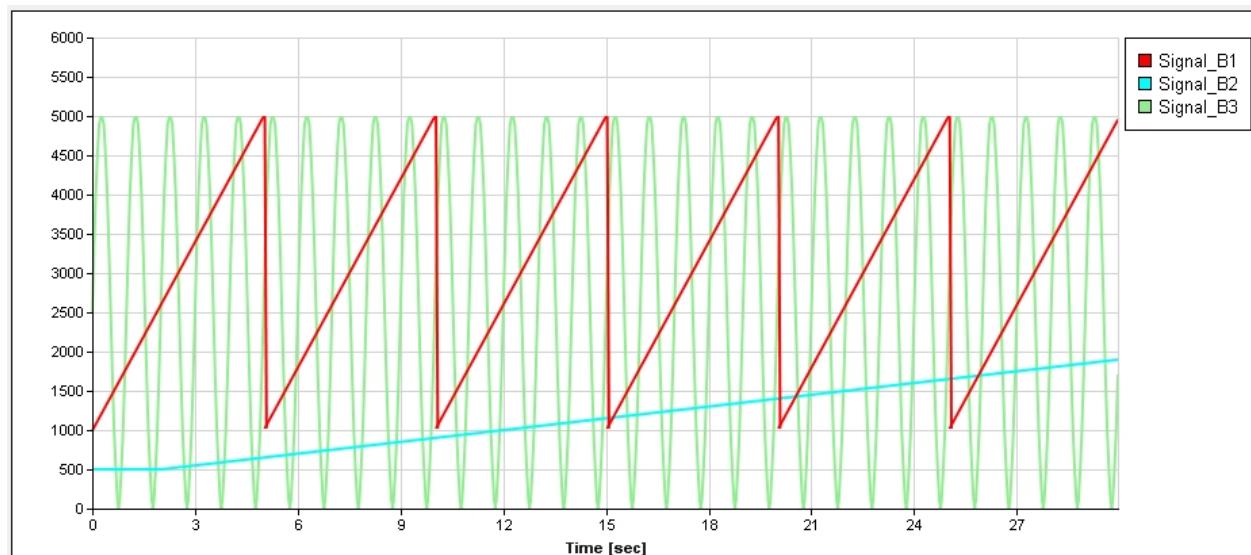
Created with the Personal Edition of HelpNDoc: [Easy to use tool to create HTML Help files and Help web sites](#)

Cycle playing

Open a cycle using the 'Cycle\Open' command of the menu strip.



Once loaded, cycle data are showed into the cycle graphic.



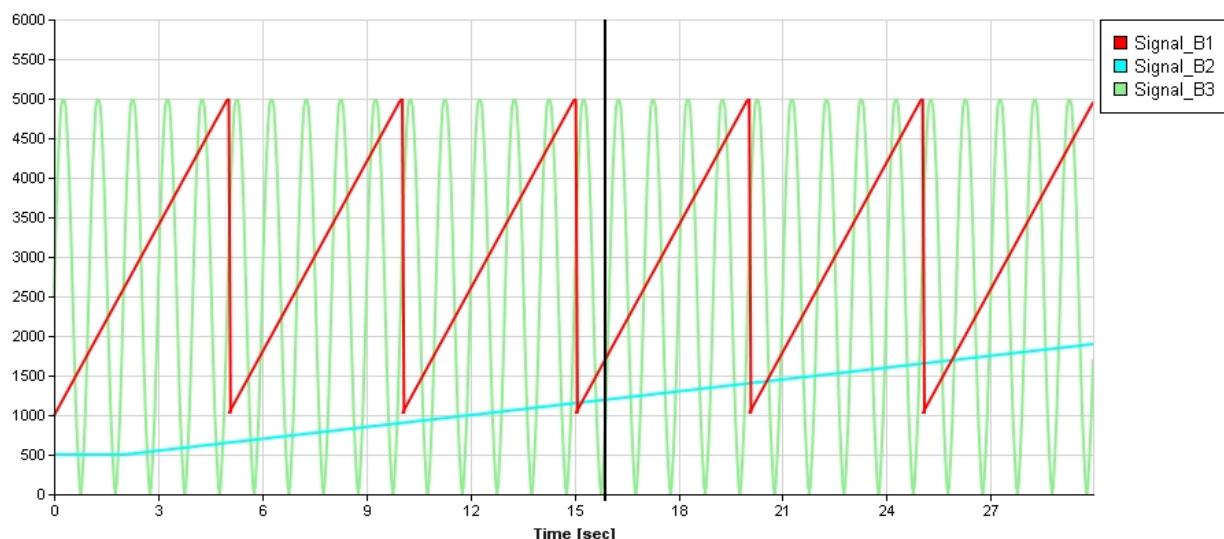
Prior to start the cycle; set the number of cycle repetitions in the 'Count' field. Then, click the 'Play' button



to start the cycle.

The cycle can be stopped at any time using the 'stop' button  or just held at the current position clicking the 'Pause'  button.

During the cycle playing, a cursor appears in the graphic showing the current position within the cycle.



In addition of this cursor, some more information shows up in the cycle control panel while the cycle is played.



Cycle progression bars.

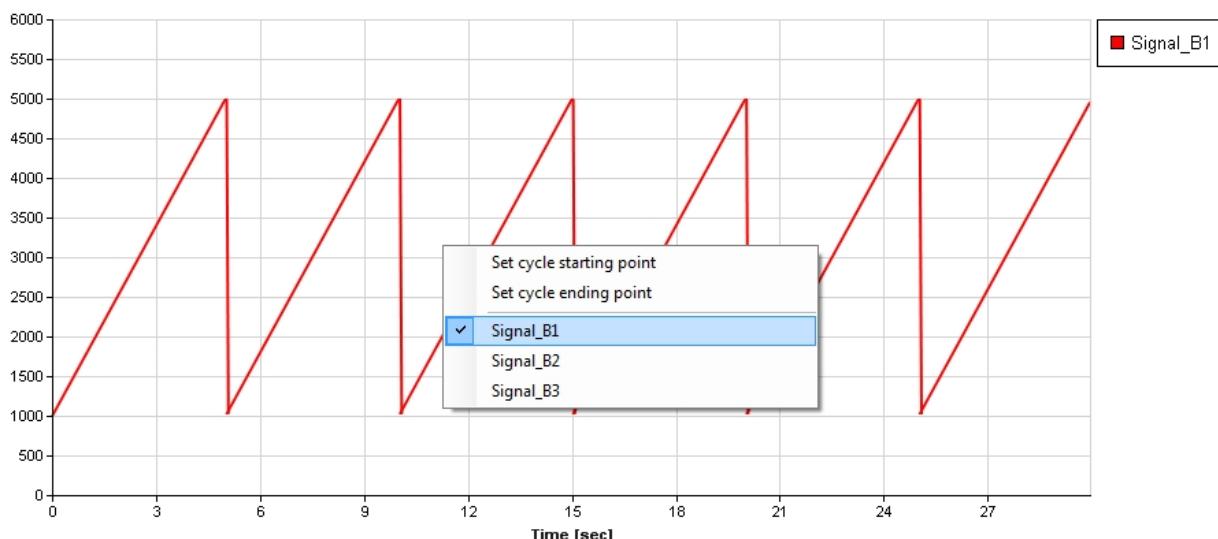


Those progress bars show the progression of the cycle.

Current cycle: Progression of the current cycle

Total: Progression of the whole sequence considering the number of repetitions. For instance, if repetition is set to two, the 'Total' bar will be filled up two times slower than the 'Current cycle' bar.

By default, all cycle data are plotted into the graphic. To hide one or more signal, right click in the graphic and uncheck signals that you don't want to see.

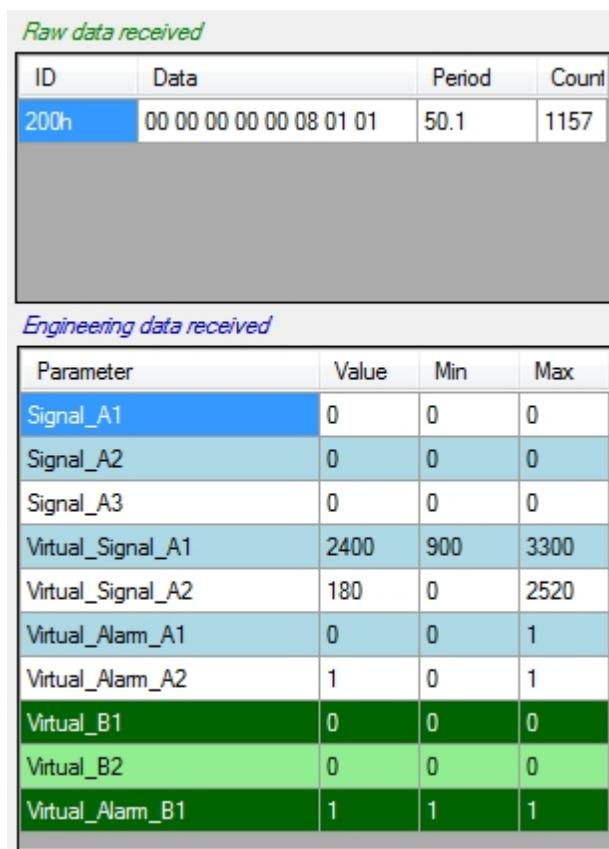


Created with the Personal Edition of HelpNDoc: [Free EPub and documentation generator](#)

Cycle RX data panel

During cycle playing, data received are shown and decoded in the 'Received data' panel. As per the [manual mode](#), this panel is split in two parts: [raw data](#) and [engineering data](#).

In the panel 'Raw data received' received CAN frames and values of their bytes are shown. While the 'Engineering data received' shows decoded signal values of received frames.



The screenshot shows two data grids side-by-side. The left grid, titled 'Raw data received', has columns for ID, Data, Period, and Count. It contains one row for a frame with ID 200h, data bytes 00 00 00 00 00 08 01 01, a period of 50.1, and a count of 1157. The right grid, titled 'Engineering data received', has columns for Parameter, Value, Min, and Max. It lists ten parameters: Signal_A1 (Value 0, Min 0, Max 0), Signal_A2 (Value 0, Min 0, Max 0), Signal_A3 (Value 0, Min 0, Max 0), Virtual_Signal_A1 (Value 2400, Min 900, Max 3300), Virtual_Signal_A2 (Value 180, Min 0, Max 2520), Virtual_Alarm_A1 (Value 0, Min 0, Max 1), Virtual_Alarm_A2 (Value 1, Min 0, Max 1), Virtual_B1 (Value 0, Min 0, Max 0), Virtual_B2 (Value 0, Min 0, Max 0), and Virtual_Alarm_B1 (Value 1, Min 1, Max 1). The rows alternate in color between light blue, light green, and dark green.

ID	Data	Period	Count
200h	00 00 00 00 00 08 01 01	50.1	1157

Parameter	Value	Min	Max
Signal_A1	0	0	0
Signal_A2	0	0	0
Signal_A3	0	0	0
Virtual_Signal_A1	2400	900	3300
Virtual_Signal_A2	180	0	2520
Virtual_Alarm_A1	0	0	1
Virtual_Alarm_A2	1	0	1
Virtual_B1	0	0	0
Virtual_B2	0	0	0
Virtual_Alarm_B1	1	1	1

As per the manual mode, columns and rows of those two grids can be customized using their contextual menus or the 'View' menu of the main menu strip.

Cycle starting and ending points setting

By default, a cycle is played from the beginning to the last sample value. This is not mandatory since user can define the part of the cycle that will be played. Then cycle will no longer be played from the beginning to the end but from the starting and/or ending points defined by the user.

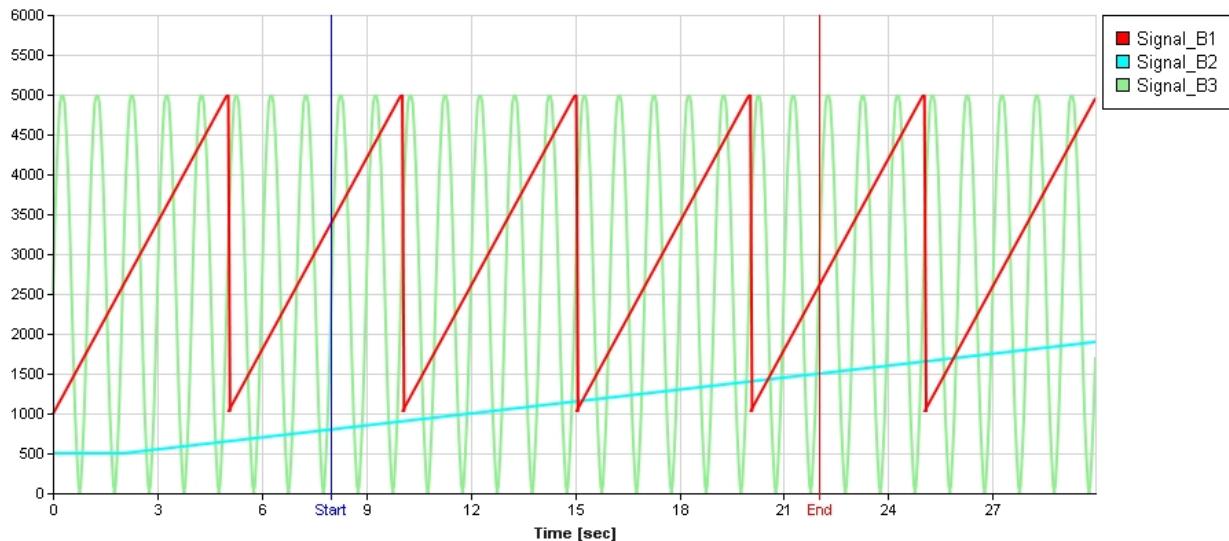
Even better, user doesn't necessarily need to define both starting and ending points. If only the starting point is defined, then cycle will be played from this point to the end. Alternatively, if only the ending point is set, cycle will be played from the beginning to this ending point.

There are two methods to define cycle starting and ending points

The numeric method: simply change the value of 'Start' and 'End' fields. Value of those fields must be time in second.

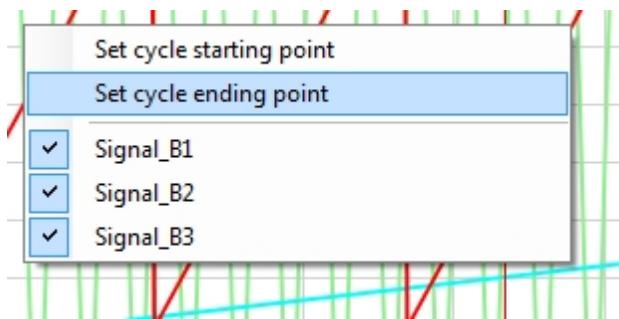
Start	8
End	22

If values set are different than actual cycle beginning and ending points cursors appear in the graphic showing current cycle starting and ending points.



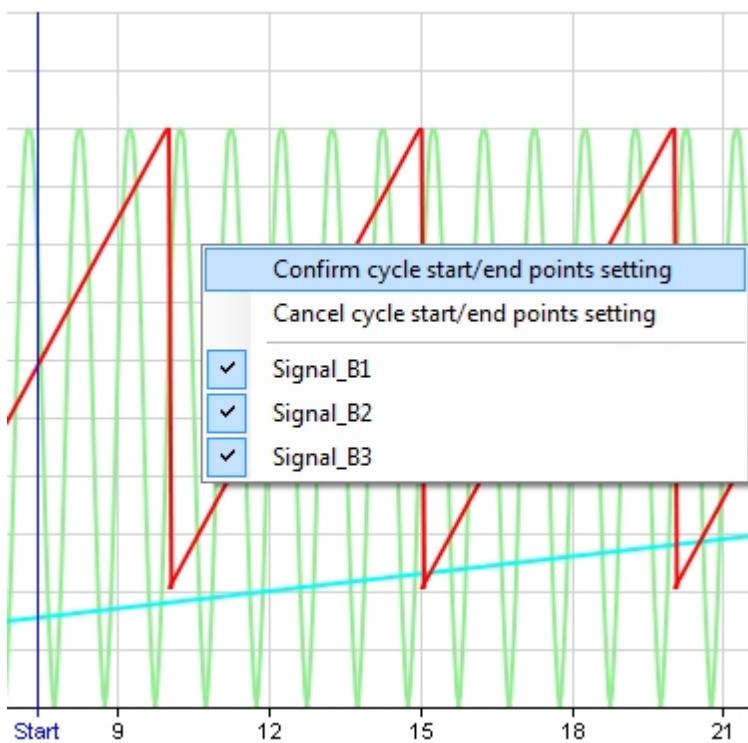
The second method consists in positioning those cursors directly into the graphic.

To do so, right click in the graphic and click 'Set cycle starting point' to set the starting point or 'Set cycle ending point' for the ending point.



Then place the cursor at the position you want the cycle to start or end by clicking this position into the graphic.

Once that position defined, right click again into the graphic to get the contextual menu and click 'Confirm cycle start/end points settings' to confirm starting or ending position.



If you click 'Cancel cycle start/end points settings', the cursor goes back to the previous position defined if any. Otherwise it simply disappears and starting or ending are either the beginning or the end of the cycle.

Created with the Personal Edition of HelpNDoc: [Easy to use tool to create HTML Help files and Help web sites](#)

Cycle loop number setting

A cycle can be played multiple times. To define the number of cycle repetitions, simply change the value 'Count' prior to launch the cycle.



Alternatively, a cycle can be played in definitively by checking the box 'Infinite play' [Infinite play](#). In that case, cycle will be played and repeated until the 'Stop' button is clicked.

Created with the Personal Edition of HelpNDoc: [Create HTML Help, DOC, PDF and print manuals from 1 single source](#)

Virtual CAN signals in cycle

As per manual mode, virtual channels can be computed based on values received on the CAN bus while a cycle is played. This can be particularly useful if virtual channels are alarms sent to a device to stop unit operation or ignore commands.

When a virtual channel is computed, it appears in the 'Engineering data received' with green background.

<i>Engineering data received</i>			
Parameter	Value	Min	Max
Signal_A1	0	0	0
Signal_A2	0	0	0
Signal_A3	0	0	0
Virtual_Signal_A1	2400	900	3300
Virtual_Signal_A2	180	0	2520
Virtual_Alarm_A1	0	0	1
Virtual_Alarm_A2	1	0	1
Virtual_B1	0	0	0
Virtual_B2	0	0	0
Virtual_Alarm_B1	1	1	1

Created with the Personal Edition of HelpNDoc: [Free iPhone documentation generator](#)

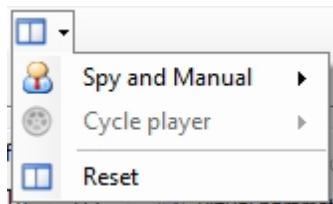
CAN controller appearance options

The appearance of a CAN controller can be customized in order to best fit with user needs.

Created with the Personal Edition of HelpNDoc: [Easy CHM and documentation editor](#)

Control layout

The 'View' command  of the tool bar contains functions to customize the appearance of the current CAN controller.

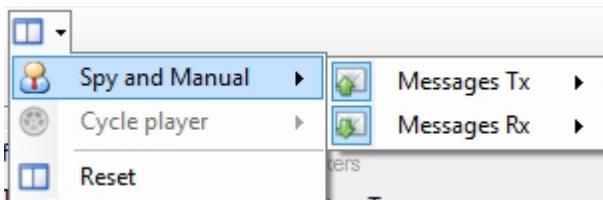


Whether the active panel is 'Spy and Manual' or 'Cycle' the corresponding commands group is enabled. 'Reset' button restores all default layout options for both 'Spy and Manual' and 'Cycle' panels.

Created with the Personal Edition of HelpNDoc: [Full-featured EPub generator](#)

Manual control layout

Click 'Spy and Manual' to get layout commands of this mode.



The menu is divided in two parts:

- 'Messages Tx' for the transmission part of the manual control
- 'Messages Rx' for the reception part of the manual control

The 'Message Tx' menu is divided in two parts:

Engineering messages: Showing or hiding the 'Engineering' section of the data transmission panel.

Raw messages: Showing or hiding the 'Raw' data section of the data transmission panel.



The 'Message Rx' menu is also divided in two parts:

Data: Showing or hiding the 'Data' section of the data reception panel.

Graph: Showing or hiding the 'Graphic trace' section of the data reception panel.



Finally, 'Data' menu is split in two parts

Engineering messages: Showing or hiding the 'Engineering' section of the data reception panel.

Raw messages: Showing or hiding the 'Raw data section of the data reception panel.



Created with the Personal Edition of HelpNDoc: [Free Kindle producer](#)

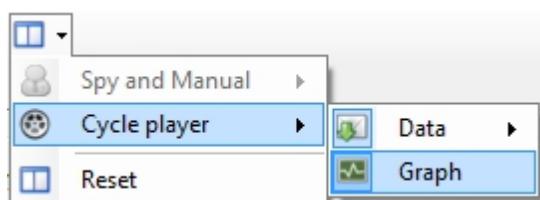
Cycle control layout

Click 'Cycle player' menu to get layout commands of this mode.

The 'Cycle player' menu is divided in two parts:

Data: Showing or hiding the 'Data' section of the cycle panel.

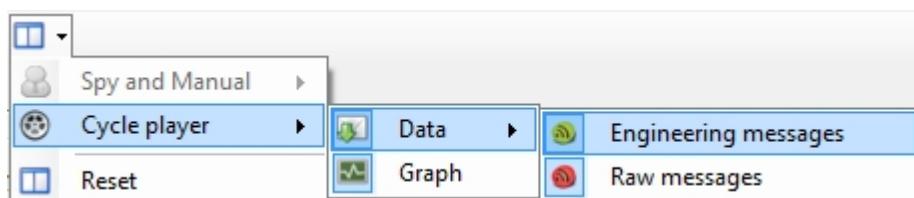
Graph: Showing or hiding the 'Graphic trace' section cycle.



Then, 'Data' sub-menu is split in two parts

Engineering messages: Showing or hiding the 'Engineering' section of the data reception panel.

Raw messages: Showing or hiding the 'Raw data' section of the data reception panel.

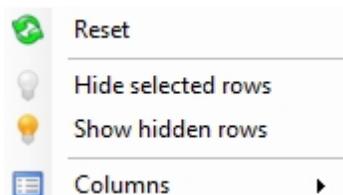


Created with the Personal Edition of HelpNDoc: [Free PDF documentation generator](#)

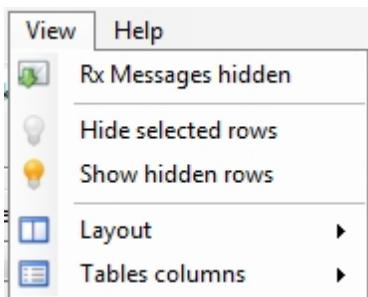
Grids contents control

The contents of all grids (columns and rows) of a CAN controller can be individually customized.

To customize the content of a grid, right click on it to get its contextual menu.



Grid customization commands are also available through the 'View' menu of the main menu strip.



Created with the Personal Edition of HelpNDoc: [Single source CHM, PDF, DOC and HTML Help creation](#)

Grid rows

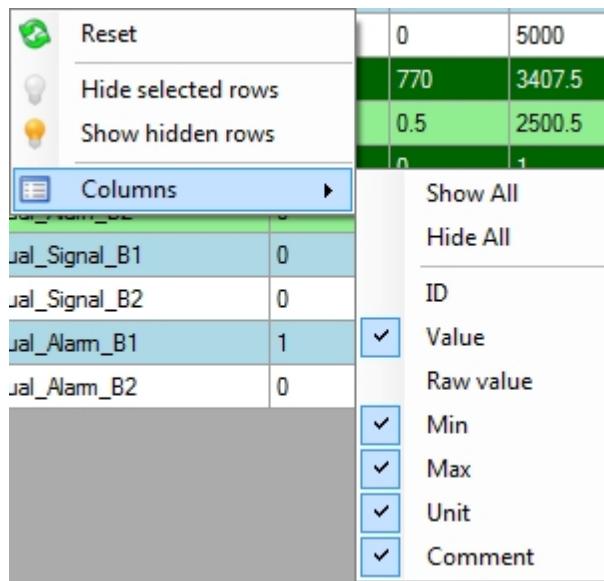
Each row of a grid can be hidden. Select rows that you want hide and click 'Hide selected rows' . Then click on 'Show hidden rows' to make hidden rows visible.

Created with the Personal Edition of HelpNDoc: [Easily create Web Help sites](#)

Grid columns

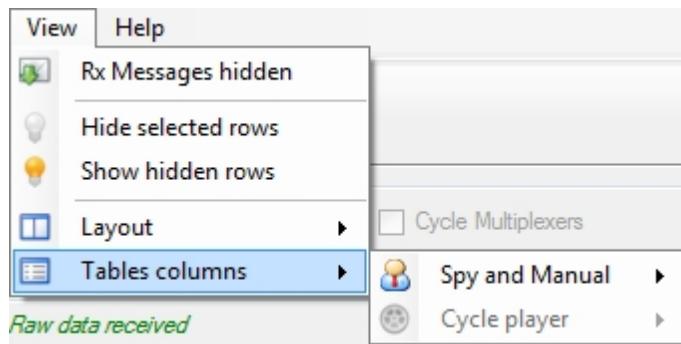
The width of the different columns of a grid cannot be changed but it is possible to selected columns that have to be shown and columns that have to be hidden.

Menu 'Columns' of the contextual menu of a grid contains columns title for the grid owning the contextual menu. Simply check or uncheck column titles to make columns visible or invisible.



Commands 'Show all' and 'Hide all' show and hide all columns at once. Then every single column can be individually set checking or unchecking its title.

The 'Table columns' menu of the 'View' menu contains columns control commands of all grid of a CAN controller.

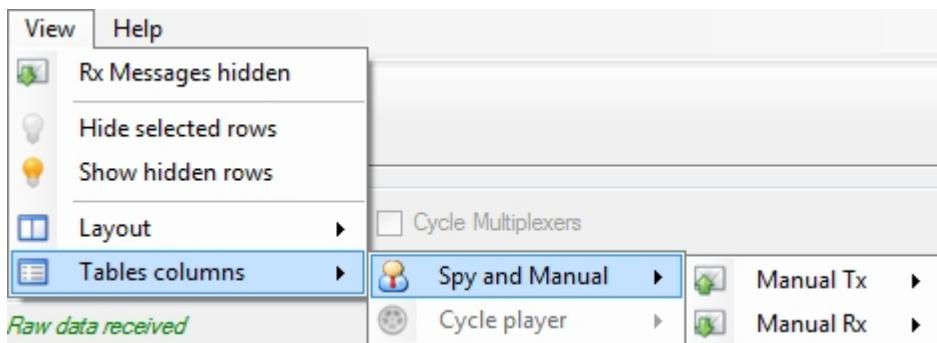


Whether the active panel is 'Spy and Manual' or 'Cycle' the corresponding commands group is enabled.

Created with the Personal Edition of HelpNDoc: [Free Web Help generator](#)

Manual control grid columns

Click 'Spy and Manual' to get grid columns commands of this mode.



The menu is divided in two parts:

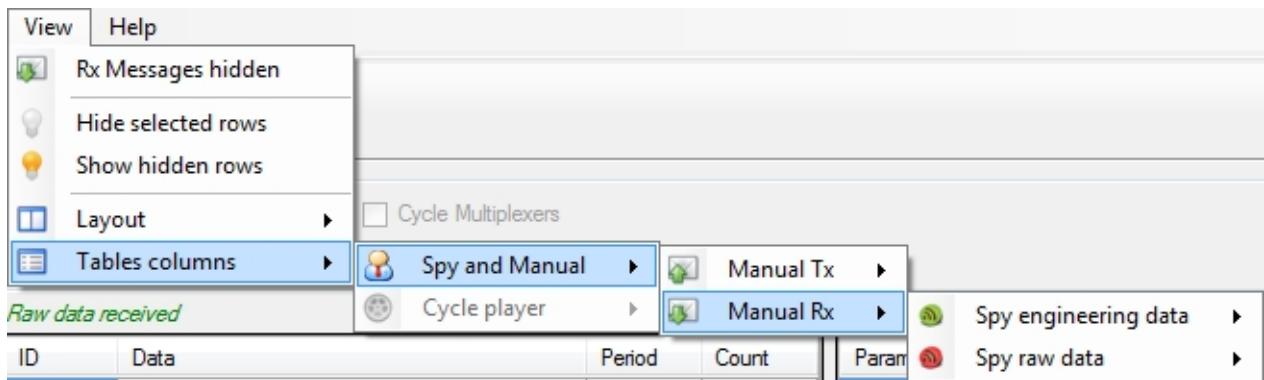
- 'Messages Tx' for the transmission part of the manual control
- 'Messages Rx' for the reception part of the manual control

The 'Manual Tx' contains columns control commands of the engineering data transmission grid.

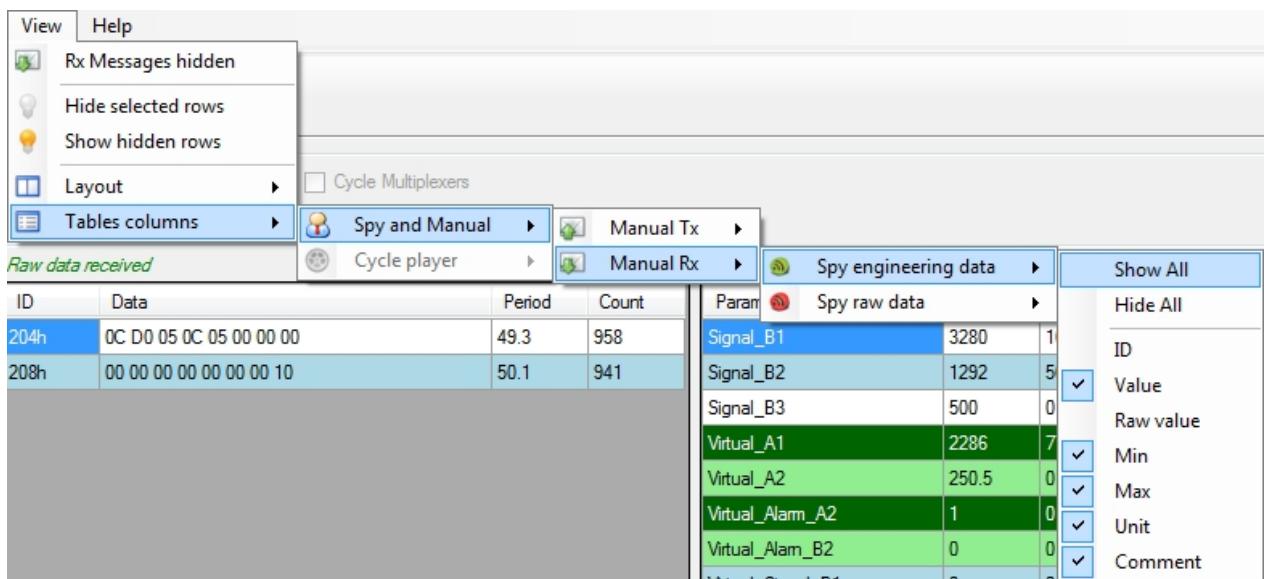
ID	Data	Period	Count	Param
204h	0C D0 05 0C 05 00 00 00	49.3	958	Signal
208h	00 00 00 00 00 00 00 10	50.1	941	Signal

Columns of the raw data transmission grid cannot be customized since all columns are need.

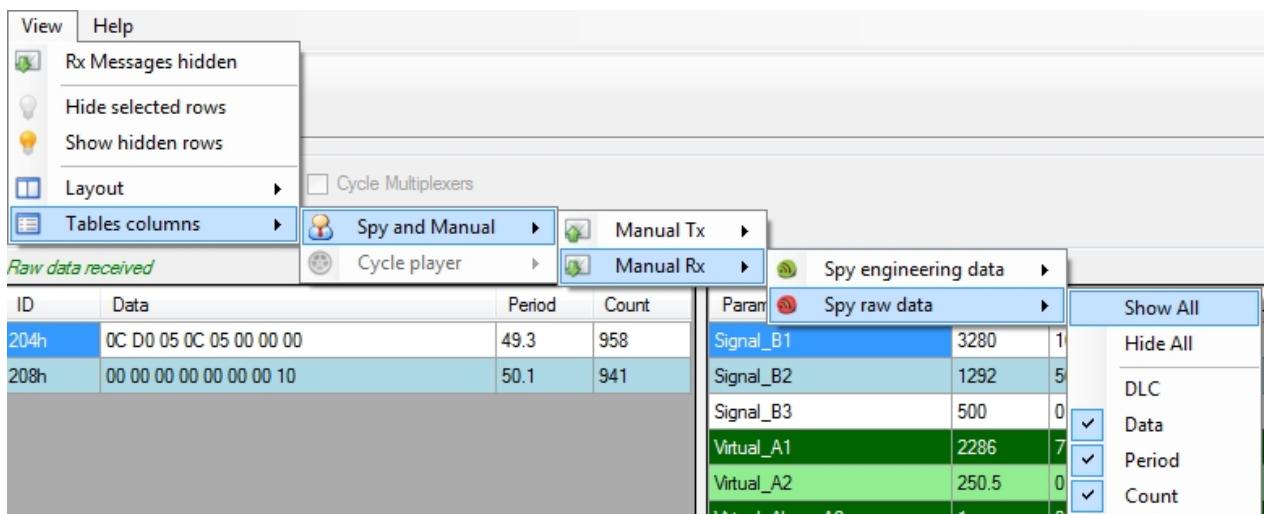
The 'Manual Rx' menu is divided in two parts:



Engineering messages: containing columns control commands of the engineering data reception grid.



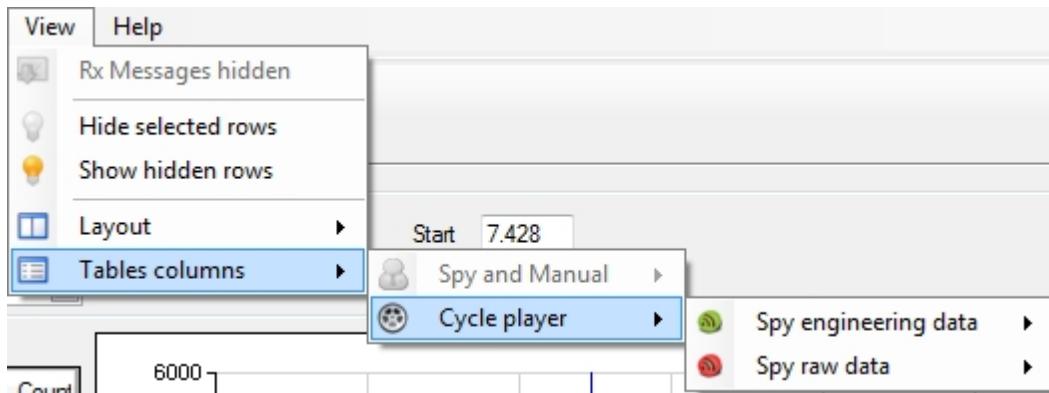
Raw messages: containing columns control commands of the raw data reception grid.



Created with the Personal Edition of HelpNDoc: [Easily create HTML Help documents](#)

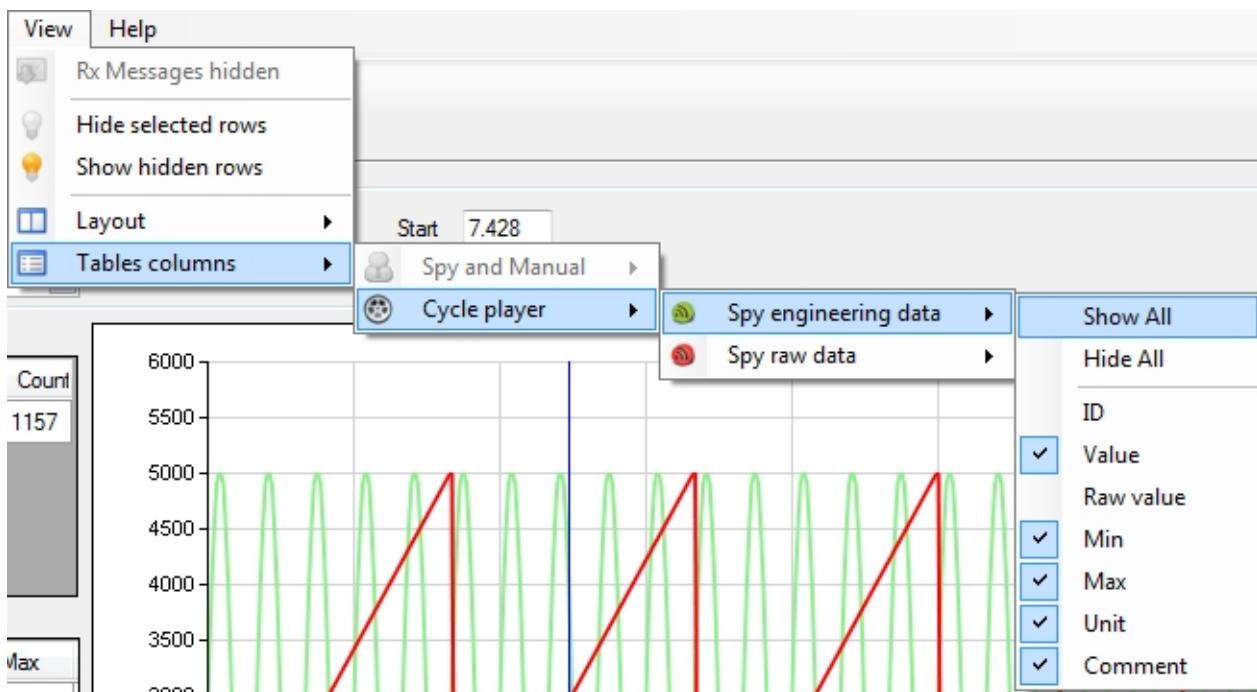
Cycle player grid control

Click 'Cycle player' to get grid columns commands of this mode.

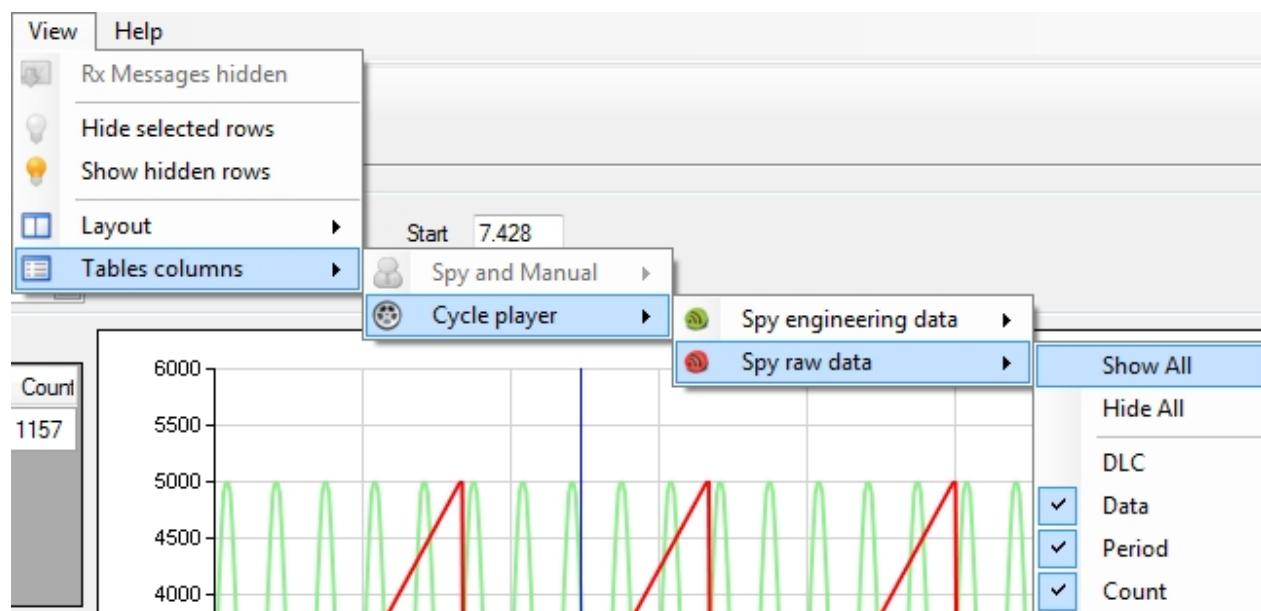


The 'Cycle player' menu is divided in two parts:

Engineering messages: containing columns control commands of the engineering data reception grid for the cycle panel.



Raw messages: containing columns control commands of the raw data reception grid for the cycle panel.



Created with the Personal Edition of HelpNDoc: [Write EPub books for the iPad](#)

Cycle creation

Prior to be loaded and played, a cycle has to be somehow created...

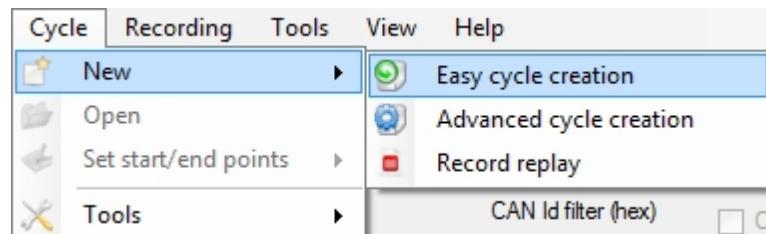
CANStream proposes three methods to create cycle files:

- Easy cycle creation
With this method the cycle is created based on a data file. This mode has very few options, that's why it is called 'easy'.
- Advanced cycle creation
Unlike the 'easy' mode, the 'advanced' mode has extended options in order to tweak the cycle.
- Record replay
This is actually the easiest cycle creation mode! Cycle is built using a record which is converted into a cycle file.

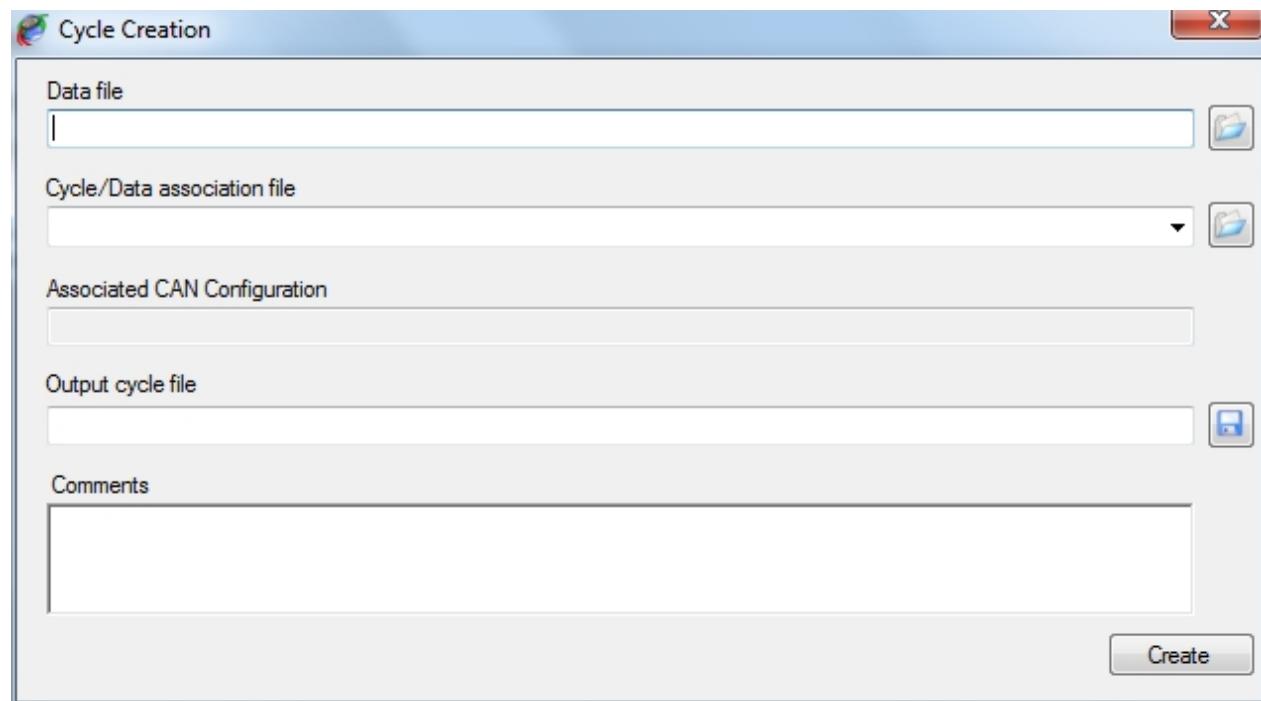
Easy cycle creation

The 'Easy cycle creation' method creates a cycle based on a data file. This mode has very few options, that's why it is called 'easy'.

Click the 'Cycle\New\Easy cycle creation' menu of the main menu strip to access the 'Easy cycle creation' mode.

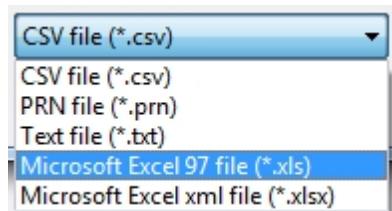


The 'Easy cycle creation' form shows up.



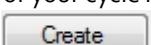
To create a cycle using this method, simply follow steps described below:

- Select a cycle source data file by clicking the 'Open' button on the right of the 'Data file' field. Several formats of data are supported for cycle creation. Change the extension by changing the extension filter of the open file dialog.



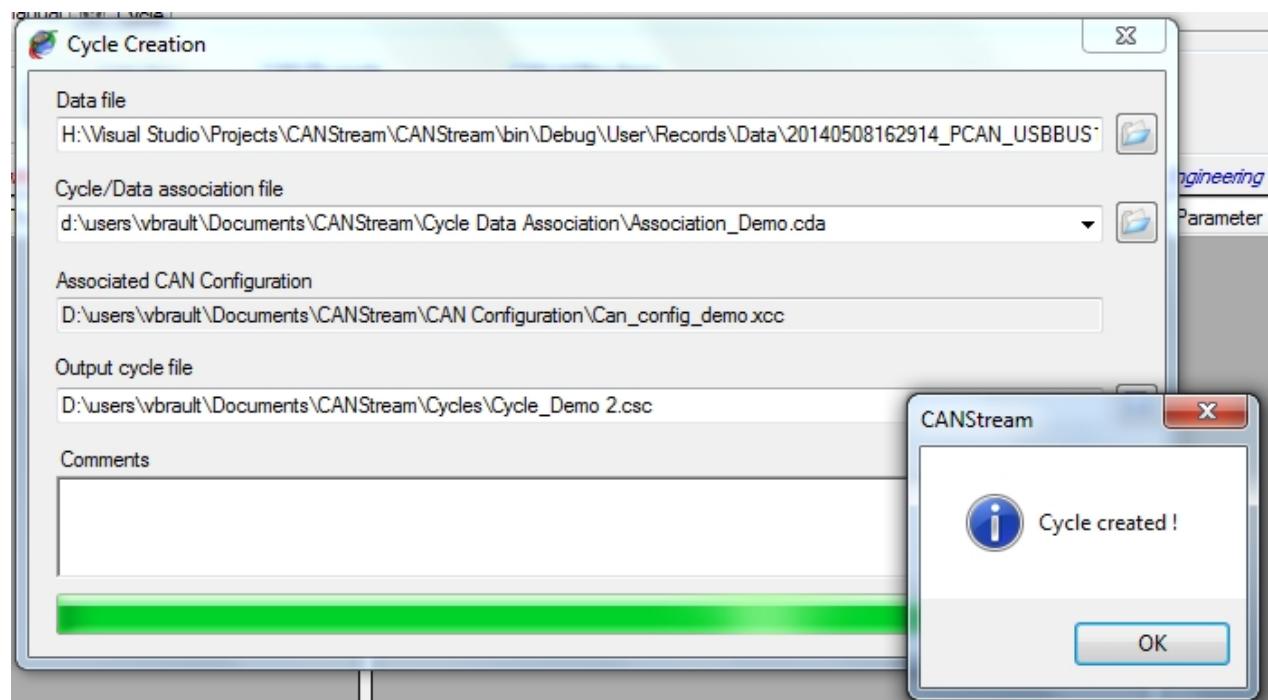
- Select a 'Cycle/Data association' file by clicking the 'Open'  button on the right of 'Cycle/Data association file' field. If your association file is located in the 'Cycle Data Association' folder of the 'CANStream' folder of 'My documents', it will present in the list. So you can drop down the list and pick up the association file that you want to use.

Once the association file selected, the CAN configuration for which the association has been made for is displayed (for information) in the 'Associated CAN configuration' text box. This text box cannot be modified, so select another association file (or make a new one) if you don't want use the CAN configuration showed.

- Set the name and path of the output cycle file clicking the 'Save'  button.
- Optionally write a description of your cycle in the 'Comments' area.
- Then click the 'Create' button  to launch the cycle creation.

Depending of the type of the source file (Excel or ASCII) and its size, it may take several minutes to create the cycle. This is particularly long with large Excel files since the Excel automation interface for .NET application is a kind of slow...

However, during the creation a progress bar appears indicating progression of the cycle creation process. At the end of the cycle creation, a message box pops up to let the user know that creation process has ended.



Created with the Personal Edition of HelpNDoc: [Write eBooks for the Kindle](#)

Cycle data association

A 'Cycle data association' file is a file containing association between CAN parameters and data source channels.

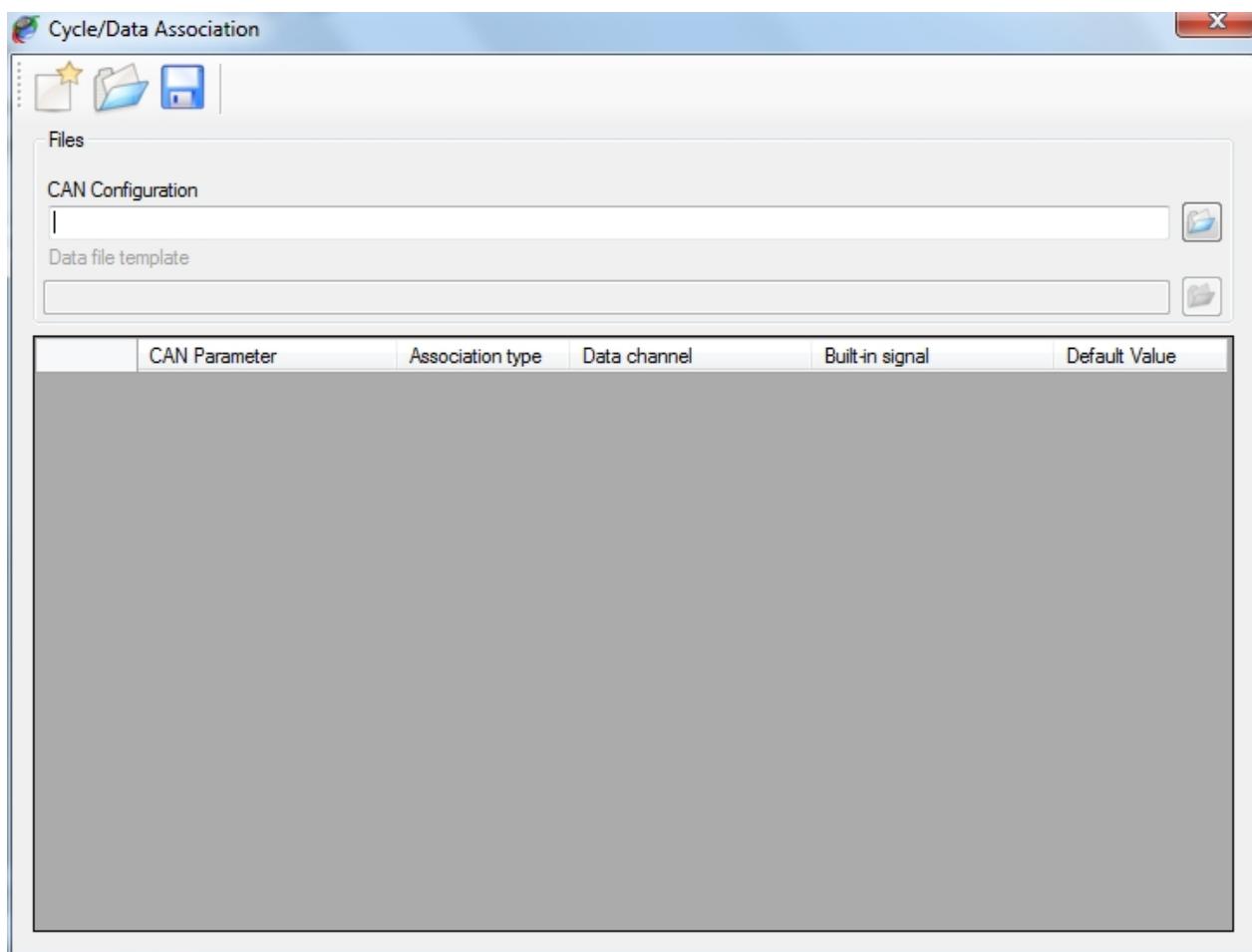
For example, let's say that you have a CAN configuration containing a CAN parameter called 'Speed' and that you want create a cycle using this configuration. You have a source data file to create your cycle but in this data file, the 'Speed' channel is called 'Velocity'.

A 'Cycle data association' file will fix that issue since it allow to associate CAN parameters with a data channel. So in our case we will simply states that the CAN parameter 'Speed' is using values of the data channel 'Velocity' as source of data.

To open the 'Cycle data association' edition form, click the 'Cycle\Tools\New Cycle\Data association' menu.



The 'Cycle data association' edition form appears



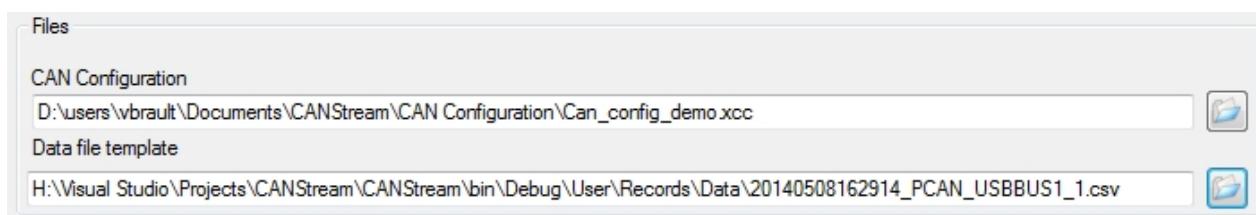
This form contains a tool bar on the top to access the 'Cycle data association' file management commands (new, open, save).

A control panel in which CAN configuration and data file are intended to be specified

A grid, containing actual CAN parameter/data channel associations.

Click the 'Open' button on the right of the 'CAN configuration' field.

Then select the data file that you want to use as template to set associations by clicking the the 'Open' button on the right of the 'Data file template' field.



As mentioned by its label, that file is a template. So it doesn't mean that association works only with this particular data file. It means association will work with any data file having the same structure as the template.

This is important here to underline that by 'structure' we consider both data channel name and data channel location. If channel 'Velocity' is placed on the third column of the data file, association will work with all data file having a channel 'Velocity' on the third column. If 'Velocity' moves to the fifth column for some reasons, association won't any longer work and association file will have to be modified accordingly.

Another important point is that CANStream consider data of the first column as time vector. If time isn't on the first column it may results to unpredictable cycle.

Once the CAN configuration loaded the association grid is filled, each row of the grid being a CAN parameter of the CAN configuration.

	CAN Parameter	Association type	Data channel	Built-in signal	Default Value
▶ 204	Signal_B1		▼	▼	▼
204	Signal_B2		▼	▼	▼
204	Signal_B3		▼	▼	▼

The association grid has six columns:

- Message ID: Identifier of the message containing the CAN parameter (for information only)
- CAN Parameter: Name of the CAN parameter in the CAN configuration
- Association type: Type of the association
- Data channel: Name of the data channel associated to the CAN parameter
- Built-in signal: Name of the built-in signal associated to the CAN parameter
- Default value: Fix value associated to the CAN parameter

Association type

Main purpose of an association file is to associate CAN parameters with data channel. Actually, a CAN parameter can be associated with something else than a data channel.

There are four types of association:

- None: CAN parameter isn't associated, its value will zero in the cycle
- AcqData: CAN parameter is associated to a data channel. CAN parameter values in the cycle will be values of the data channel.
- Built-in signal: CAN parameter is associated to a built-in signal. CAN parameter values in the cycle will be values of the built-in signal
- Fixed value: CAN parameter is associated to a fix value. CAN parameter values in the cycle will be the value defined all the time.

Click on 'Association type' cell of a CAN parameter and pick up the association that you want to use.

	CAN Parameter	Association type
▶ 204	Signal_B1	▼
204	Signal_B2	None
204	Signal_B3	AcqData

Once the association type defined, select the data channel or built-in signal in the corresponding cell.

	CAN Parameter	Association type	Data channel	Built-in signal
▶ 204	Signal_B1	AcqData	▼	▼
204	Signal_B2		▼	▼
204	Signal_B3		▼	▼

	CAN Parameter	Association type	Data channel	Built-in signal
204	Signal_B1	AcqData	Rpm	
204	Signal_B2	Built Signal		Signal_Lib_Demo:Slope_...
204	Signal_B3			Signal_Lib_Demo:Slope_100 Signal_Lib_Demo:Ramp_50 Signal_Lib_Demo:Sinus_1Hz

For a 'Fixed value' association simply type a value into the 'Default value' cell.

	CAN Parameter	Association type	Data channel	Built-in signal	Default Value
204	Signal_B1	AcqData	Rpm		
204	Signal_B2	Built Signal		Signal_Lib_Demo:Slo...	
204	Signal_B3	FixedValue			23

Several CAN parameters can be associated with the same data channel or built-in signal.



Once all associations are set click the 'Save' button of the tool bar to save the association file.

Created with the Personal Edition of HelpNDoc: [Create iPhone web-based documentation](#)

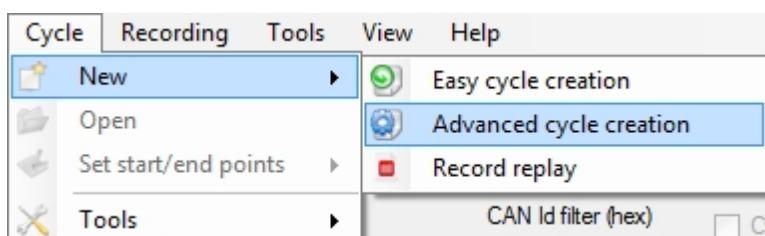
Advanced cycle creation

The 'Advanced cycle creation' method has extended options for cycle creation allowing tweaking the cycle in order to fit with particular needs.

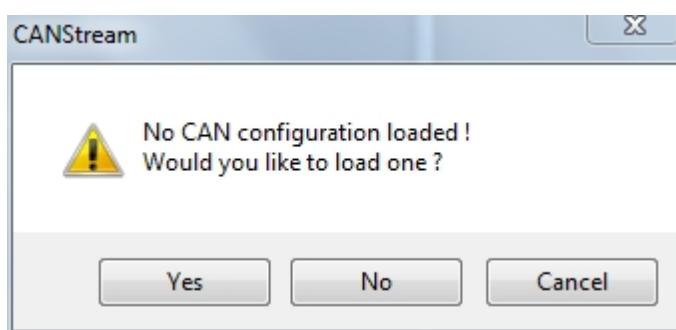
With the 'Advanced cycle creation' method, cycle creation configuration can be saved into a cycle creation configuration file (*.x3c) in order to have configuration re-useable. Such file contains all information needed to create the cycle (CAN configuration, built-in signals, virtual channels) so it can be easily distributed to others without having to give a whole bunch of files.

The biggest advantage of using the 'Advanced cycle creation' is that it can use virtual channels as source of data for the cycle. Thus, since virtual channels can be computed using multiple built-in signals, variables and functions, it allows the generation of complex profiles for the cycle.

Click the 'Cycle\New\Advanced cycle creation' menu of the main menu strip to access the 'Advanced cycle creation' mode.



If there is no CAN configuration currently loaded, a message box pops up proposing to load one.

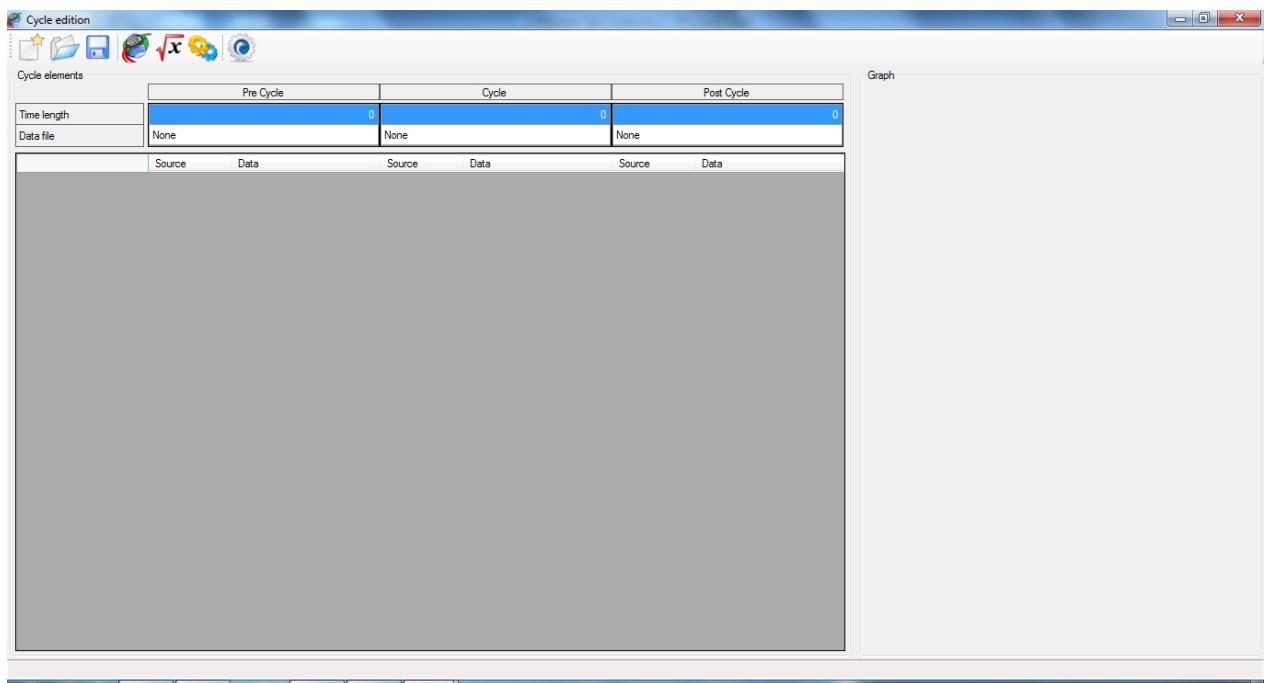


If you answer 'Yes' a 'File open' dialog will show up in which you can select the CAN configuration that has to be used to create the cycle.

If you answer 'No', the 'Advanced cycle creation' form will open but you still will have to open a CAN configuration file in order to create a cycle.

If you answer 'Cancel', it cancels the cycle creation and returns to the main form of CANStream.

Well, let's say that we have answered 'No' to this question... Form appears as below.

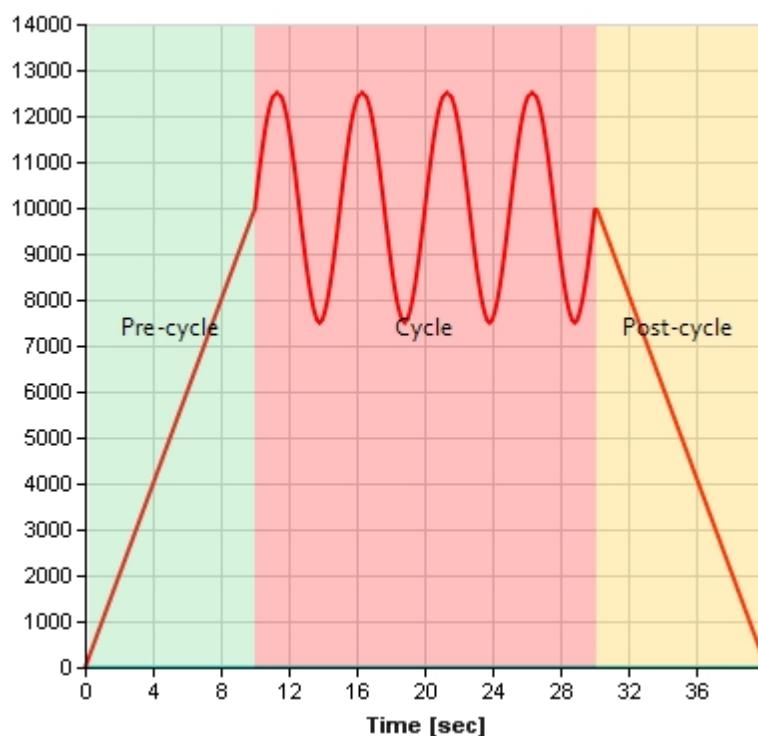


In the 'Advanced cycle creation' environment, a cycle is split in three parts: 'Pre-cycle', 'In cycle' and 'Post-cycle'.

Those parts can be considered as sub-cycles, final whole cycle being the concatenation of the three parts. If we consider the 'In cycle' part as the actual cycle, the 'Pre-cycle' part is the sequence played before the cycle like an introduction. The 'Post-cycle' part is therefore the sequence played after the cycle like a conclusion.

'Pre' and 'Post' cycle concept is very useful when the system to test has to be gently started and stopped.

For instance, something looking like the profile bellow



The grid on the top regroups cycle parts properties.

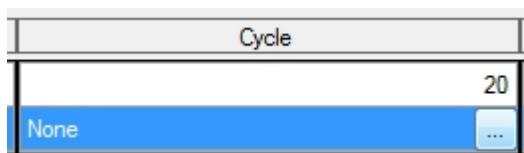
Cycle elements		Pre Cycle	Cycle	Post Cycle
Time length		10	20	10
Data file	None	None	None	None

There is one column for each cycle part.

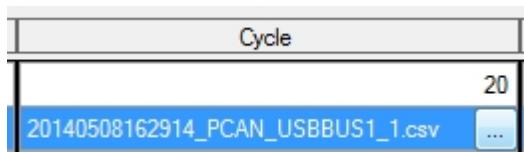
Row 'Time length' specifies the time length (in second) of each part, final cycle time length being the sum of cycle parts length. So in the example above, final cycle will be 40 seconds ($10 + 20 + 10 = 40$).

Row 'Data file' contains the file used as data source for each cycle part. All cycle parts can use the same or different files.

To set a data file for a cycle part, click in the 'Data file' cell of the part to make the 'Open file' button visible.



Click this button to open the open file dialog and select a file. Once the data file selected, its name is shown in the cell.



If the time length of the data file exceeds the time length of the cycle part, CANStream will propose you to change the cycle part length in order to match the length of the data file but this is not mandatory.

If the time length of a part exceeds the data file time length, the last sample value of the data file is used until the end of the cycle part.

The tool bar at the top of the form contains some commands:



New: Create a new cycle creation configuration file



Open: Open a cycle creation configuration file



Save: Save the current cycle creation configuration file



Load CAN Configuration: Load a CAN configuration file



Virtual channel: Reload the current virtual channels library collection (in case of virtual channel modification during the cycle edition).



Built-in Signals: Reload the current built-in signals library collection (in case of built-in signals

modification during the cycle edition).



Build cycle: Starts the cycle creation process.

If there was not CAN configuration loaded at the form start up, click the 'Load CAN Configuration' button  of the tool bar to load one.

Once loaded, all CAN parameters contained in CAN messages set with 'Transmission' (Tx) flag will be display in the grid.

Cycle elements		Pre Cycle		Cycle		Post Cycle	
Time length		10		20		10	
Data file	None		20140508162914_PCAN_USBBUS1_1.csv	...	None		
▶ Signal_B1	Source	Data	Source	Data	Source	Data	
Signal_B2	Default	▼ 0	Default	▼ 0	Default	▼ 0	
Signal_B3	Default	▼ 0	Default	▼ 0	Default	▼ 0	

Each row of the grid represents a CAN parameter and has three groups of two columns.

Those three groups are respectively, 'Pre-cycle', 'Cycle' and 'Post-cycle' parts of the cycle.

The two columns of each group are configuration of the CAN parameter for a given cycle part.

Source	Data
Default	▼ 0
Default	▼ 0
Default	▼ 0

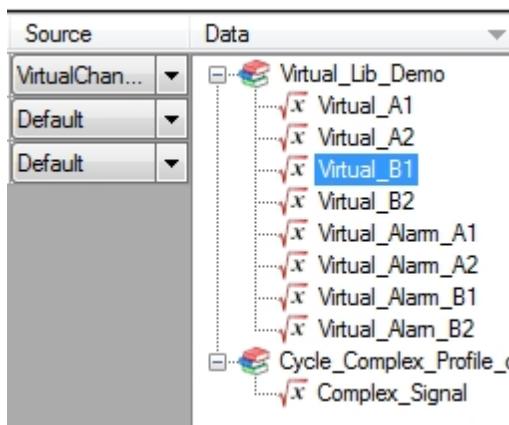
Column 'Source' defines the data source for the CAN parameter. Click the 'Source' cell to drop the source list down and pick up the desired source.

Source	Data
Default	▼ 0
None	0
Default	0
Constant	0
BuiltSignal	
VirtualChannel	
AcqData	

There are six possible sources:

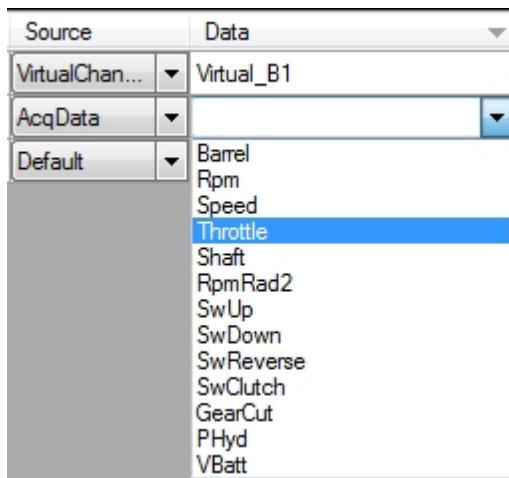
- None: CAN parameter value is set to zero
- Default: CAN parameter value is set with the default value which is zero
- Constant: CAN parameter value is set with the constant value set in the 'Data' cell
- BuiltSignal: CAN parameter value is set with value of the built-in signal defined in the 'Data' cell
- VirtualChannel: CAN parameter value is set with value of the virtual channel defined in the 'Data' cell
- AcqData: CAN parameter value is set with value of the acquisition data channel defined in the 'Data' cell

If the 'Source' property is set to 'BuiltSignal' or 'VirtualChannel', click the 'Data' cell to get the list of items available.



Available item list takes a form of a tree where libraries are roots of the tree and items (virtual channels or built-in signals are branches. Just double-click an item to select it as data source of the CAN parameter.

If the 'Source' property is set to 'AcqData', click the 'Data' cell to get the list of data channels available. Simply double-click a channel to select it as data source of the CAN parameter.



You don't need here to select a source for each CAN parameter and for all cycle parts. If a cycle part length is null, it won't generated so it can be ignored while selecting data sources of CAN parameters.

Once all CAN parameter of your interest are set, click the 'Build cycle' button to launch the cycle creation.

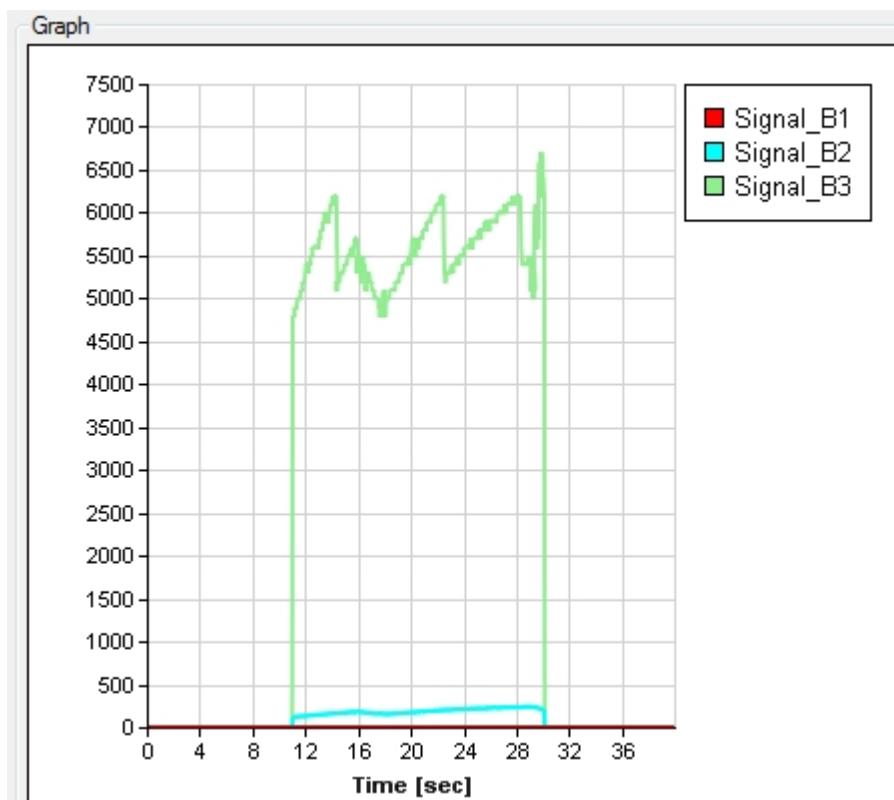
At the first cycle building, a 'file save' dialog pops up. Set the name and path of the output cycle file. The cycle can be built several times but the 'file save' dialog will show up only once.



During cycle building, the 'Abort' button will appear. Click this button at any time while CANStream is building the cycle to cancel the process.

At the end of the cycle creation process, a message box will pop up indicating that cycle has been successfully created.

In addition, a graphic preview of the cycle will be drawn on right side of the form.



As per the [cycle player](#) graph, right click in the graph to select traces that you want to see in the graphic.



Created with the Personal Edition of HelpNDoc: [Free iPhone documentation generator](#)

Record replay

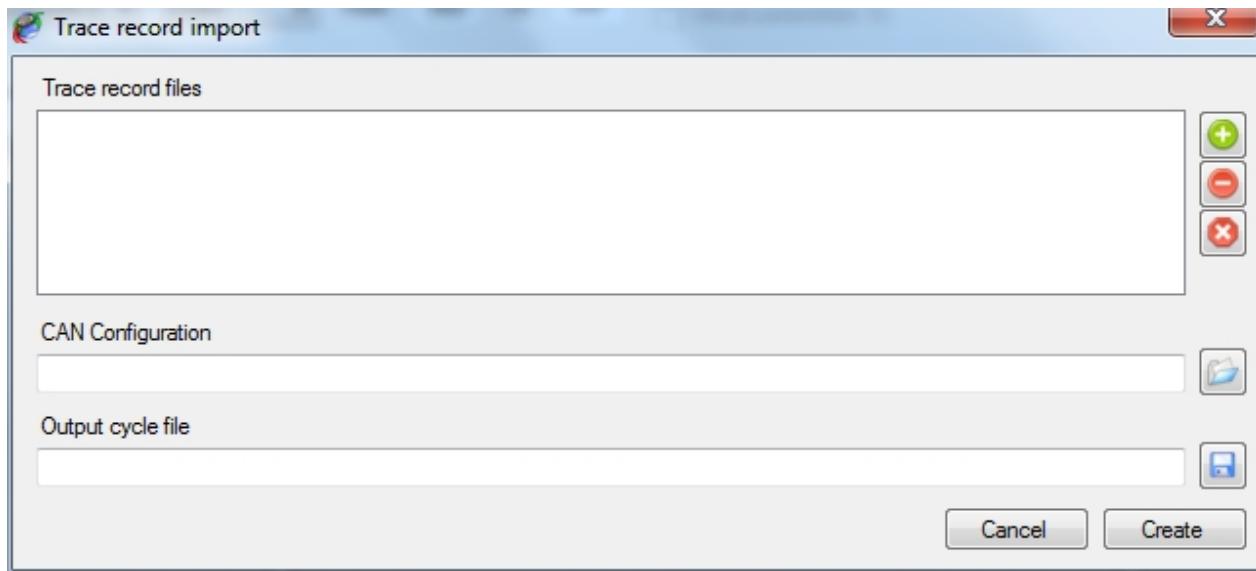
The 'Record replay' cycle creation method simply transform one or more PCAN Trace files (*.trc) into a cycle file. This method is particularly useful to quickly replicate a behavior that has been previously recorded.

With this method you don't need to associate anything, frames contained in the PCAN Trace file are simply reformatted to fit with the CANStream cycle file format.

Click the 'Cycle\New\Record replay' menu of the main menu strip to access the 'Record replay creation' mode.



The 'Record replay' form shows up.



To create a cycle using this method, simply follow steps described below:

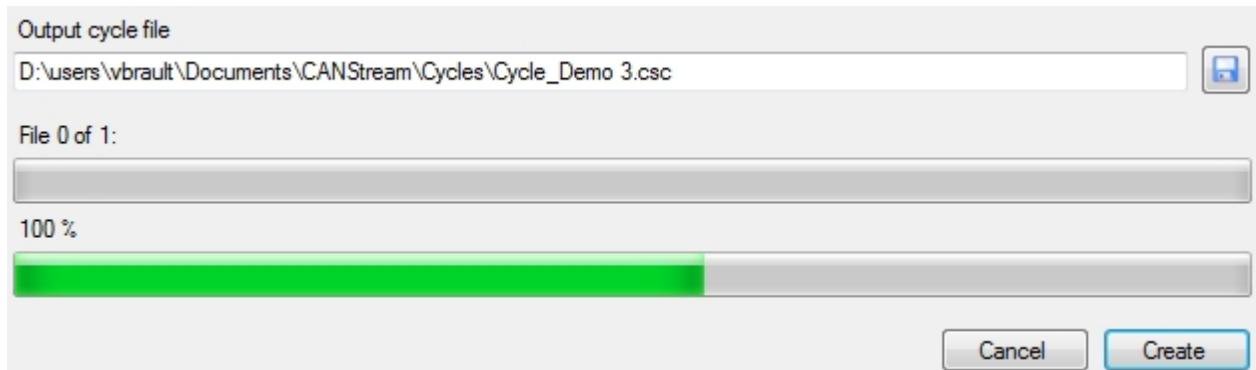
- Add one or more PCAN Trace file into the 'Trace record file'. Basically all trace files are concatenated to each other in the order they appear in the list in order to make the final cycle.

Trace files list control commands are located on the left of the list. There are three commands:

- Add: Add a PCAN Trace file into the list
- Del: Remove a PCAN Trace file from the list
- Clear: Clear all PCAN Trace file from the list

- Select a CAN configuration file by clicking the 'Open' button on the right of the 'CAN Configuration' field. This CAN configuration is only use to get the CAN bus speed (1000 kBit/s, 500 kBit/s) nothing else.
- Define name and file path of the output cycle by clicking the 'Save' button on the right of the 'Output cycle file' field.
- Click the 'Create' button to launch the cycle creation.

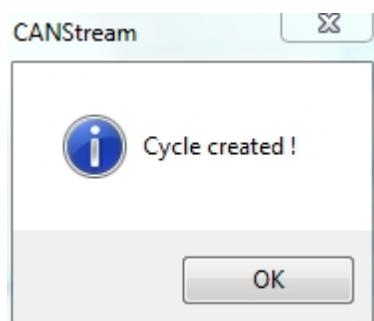
During the cycle creation process two progress bars appears.



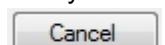
The first bar, on the top, indicates the process progression among PCAN Trace file to convert. The second bar, at the bottom, indicates the progression of the conversion for the current file.

At the end of the cycle creation, when all PCAN Trace file have been converted, a message box pops up

indicating that process has ended.



At any time in the process it is possible to cancel the cycle creation by clicking the 'Cancel' button



Created with the Personal Edition of HelpNDoc: [Create HTML Help, DOC, PDF and print manuals from 1 single source](#)

Data analysis

blabla

Created with the Personal Edition of HelpNDoc: [Free CHM Help documentation generator](#)

Data browser

blabla

Analysis window

The graphical data analysis window shows recorded data in a graphical form.

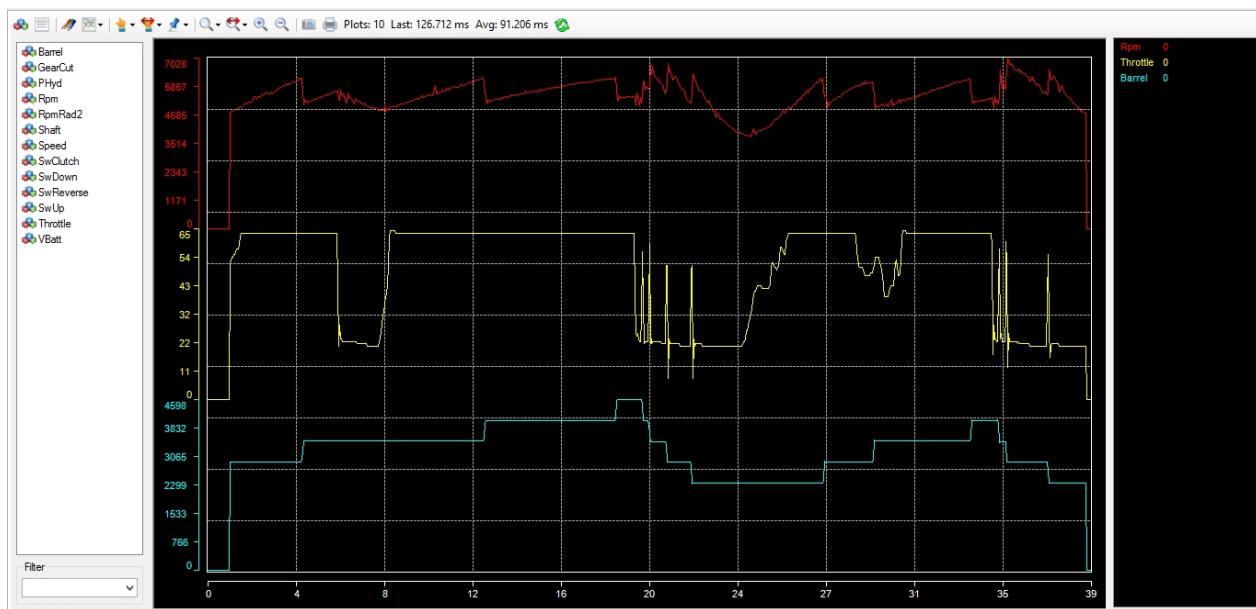
In addition of simply showing graphical data trace, the analysis window offers extensive tools such as zoom, cursors and statistics computation.

On the top of that, the graphical configuration of the analysis window is fully configurable run time. Through its intuitive configuration forms, it is possible for instance, to change the window back color or change the color of a data trace or even change its tracing mode just by simple clicks and without having to reload the data or reset the form.

Created with the Personal Edition of HelpNDoc: [Free help authoring environment](#)

Analysis window presentation

Screen shot below shows a typical analysis window graphic drawn using default properties.



The analysis window has four main areas:

- The graphic where traces are drawn
- The [channel list](#), on the left, where all channels available in the loaded data file are displayed
- The [legend](#), on the right, where all channel plotted as well as their values, units and statistics are shown.
- The [tool bar](#), on the top, where all principal command of the graphic window are available

Analysis window tool bar

The analysis window tool bar is shown on the top of the graphic window form. It regroups main analysis window commands.

There are actually a lot more commands for the analysis windows spread into configuration forms and contextual menus.



Channel list: Show or hide the [channel list](#) panel

Legend: Show or hide the graphic [legend](#) panel

Graphic configuration: Open the main graphic configuration form

Graphic layout: Change the current graphic [layout](#) (Parallel, Overlay, Custom)

Cursor type: Change the current type of the [main graph cursor](#)

Cursor step: Change the current [cursor step](#) for key board arrows keys

Reference cursor: [Reference cursor](#) command (Set, Clear)

Zoom mode: Change the current [mode of zoom](#)

Zoom factor: Change the current [zoom factor](#) of 'Zoom plus' and 'Zoom minus' functions

Zoom plus: [Zoom plus](#) command

Zoom minus: [Zoom minus](#) command

Snapshot: Graphic [snapshot](#) command

Print: Graphic [print](#) command

Created with the Personal Edition of HelpNDoc: [Full-featured multi-format Help generator](#)

Analysis window shortcut keys

Here is a summary of all shortcuts keys of the analysis window.

Key	Function
+	Zoom plus
-	Zoom minus
W	Zoom min (0% zoom)
N	Zoom max (100% zoom)
X	Zoom mode X
Y	Zoom mode Y
B	Zoom mode XY
V	Cursor vertical
H	Cursor horizontal
K	Cursor cross
Pg Up	Cursor step +
Pg Down	Cursor step -
R	Set reference cursor
ESC	Clear reference cursor
O	Graph layout overlay
P	Graph layout parallel
C	Graph layout custom
G	Graph properties edition
T	Show/Hide legend selected series
DEL	Remove legend selected series from the graph

Data file

The data file format used by the analysis window is CSV (Column Separated Value)

Columns must be separated by a semi-comma character [;].
Values decimal separator must be point [.] or comma [,].

First line of the file contains names of data channels.

First column must be abscisse value (X axis values)

```
Time;Barrel1;Rpm;Speed;Throttle;Shaft;RpmRad2;SwUp;SwDown;SwReverse;SwClutch;GearCut;PHyd;VBatt
1.008;2904;4759;118;51;-0.732;498.11;0;0;0;0;0;55;13
1.0095;2899;4769;120;52;-0.732;499.16;0;0;0;0;0;55;13
1.011;2899;4769;120;52;-0.732;499.16;0;0;0;0;0;55;13
1.0125;2899;4769;120;52;-0.732;499.16;0;0;0;0;0;55;13
1.014;2899;4769;120;52;-0.732;499.16;0;0;0;0;0;55;13
1.0155;2899;4754;120;52;-0.732;497.59;0;0;0;0;0;55;13
1.017;2899;4754;120;52;-0.732;497.59;0;0;0;0;0;55;13
1.0185;2899;4754;120;52;-0.732;497.59;0;0;0;0;0;55;13
1.02;2899;4754;120;53;-0.732;497.59;0;0;0;0;0;55;13
1.0215;2899;4754;120;53;-0.732;497.59;0;0;0;0;0;55;13
1.023;2899;4754;120;53;-0.732;497.59;0;0;0;0;0;55;13
1.0245;2899;4754;120;53;-0.732;497.59;0;0;0;0;0;55;13
1.026;2904;4754;120;53;-0.732;497.59;0;0;0;0;0;55;13
1.0275;2904;4754;120;53;-0.732;497.59;0;0;0;0;0;55;13
1.029;2904;4754;120;53;-0.732;497.59;0;0;0;0;0;55;13
```

Created with the Personal Edition of HelpNDoc: [Easily create Help documents](#)

Analysis functions

This section regroup analysis features documentation of the graphic window

Created with the Personal Edition of HelpNDoc: [Full-featured EBook editor](#)

Main cursor

The main graphic cursor is a graphical help for analysis. It permits to point any coordinates within the graphic area.

Just click anywhere in the graphic area to make the cursor visible. Press the mouse right button and move the mouse to have the cursor following your movements. Release the mouse right button and the cursor gets fixed.

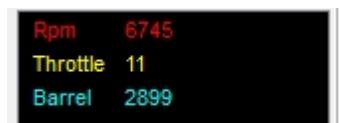
Default cursor type is 'Cross', which means that graphical cursor is taking a cross form.

There are actually seven cursor modes:

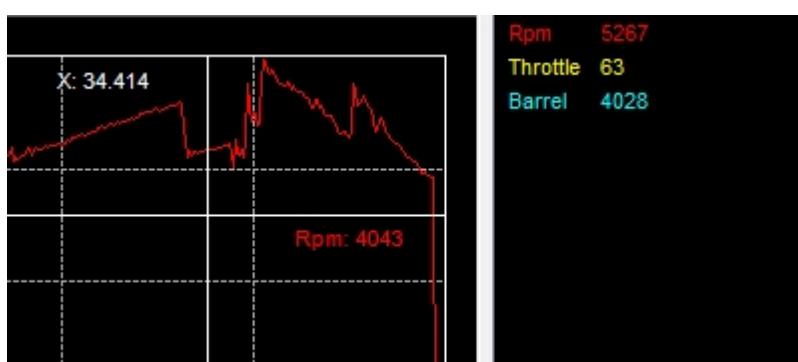
- None: Graphical cursor is disabled
- Vertical line: Cursor is a vertical line following the mouse along the X axis
- Horizontal line: Cursor is a horizontal line following the mouse along the Y axis
- Cross: Cursor is cross made by a vertical and a horizontal line, lines crossing each other at the mouse location
- Graticule: Cursor is a small cross following the mouse inside the graphic area
- Square: Cursor is a small square surrounding the actual mouse location
- Circle: Cursor is a small circle surrounding the actual mouse location

To change the cursor type, click on the 'Main graph cursor type' of the tool bar  and select the desired type. You can also use the 'Cursor' menu  of the graphic contextual menu (right click in the graphic area). 'Vertical line', 'Horizontal line' and 'Cross' are also available through keyboard shortcuts 'V', 'H' and 'K'.

While moving the graphic cursor, you can see in the [legend](#) that each trace value is updated with the actual value of the trace at the position of the cursor.

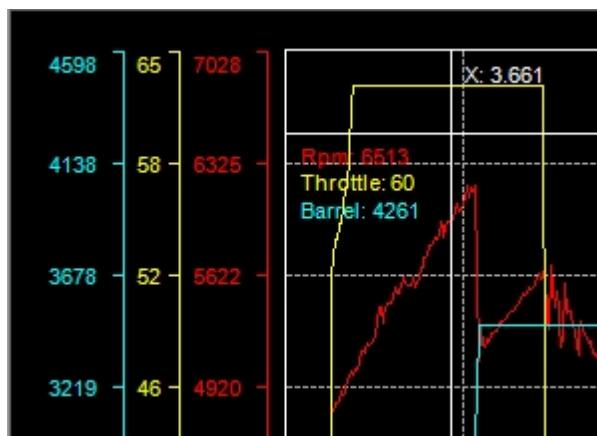


In addition of that, for 'Vertical', 'Horizontal' and 'Cross' cursor types, values of graphical coordinate are showed.



By coordinates, we mean X and Y axis values at the position of the cursor. For instance on the screen shot

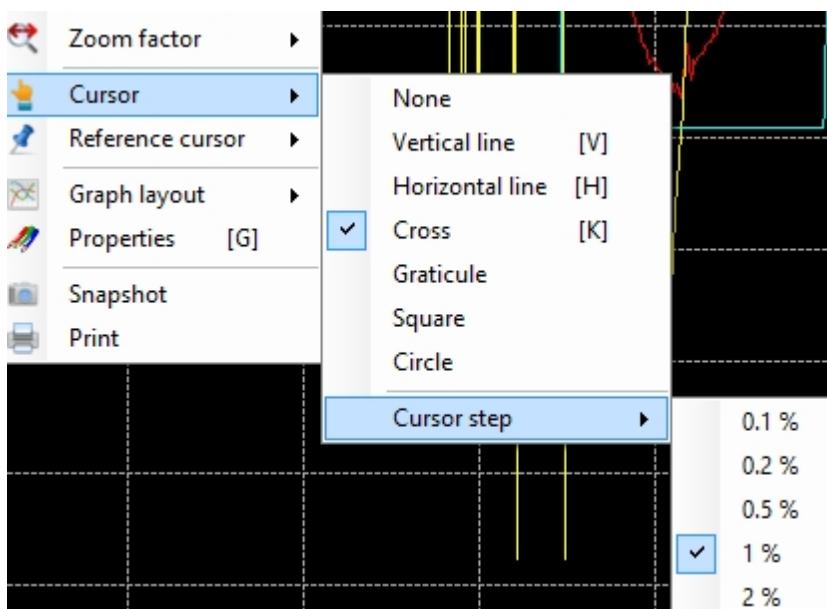
above, '34.414' is the value along the Xaxis and '4043' is the value along the Y axis which is named 'Rpm'. You may have more than one Y axis at the position of the cursor, in that case all axis values are showed.



Using 'left', 'right', 'up' and 'down' arrow keys of the keyboard you can move the cursor to the direction you want. Obviously, 'up' and 'down' movements are disabled for the 'Vertical line' cursor type, while 'left' and 'right' are disabled for the 'Horizontal line' type.

By setting the 'Cursor step' you can define how big (or small) cursor movement will be for each arrow key press event.

To change the cursor step, click the 'Main cursor step' button of the tool bar and select the step you want. This command is also available through the graphic contextual menu, under the 'Cursor \ Cursor step' menu. You may also use 'Page Up' and 'Page Down' keys to change the cursor step.



Cursor step values are percentage values... OK but percentage of what ? Good question !

Step values are percentage of the current Xaxis values span. For example, if 100 seconds of data are plotted and the cursor step is 1%, the cursor will move by step of 1 second.

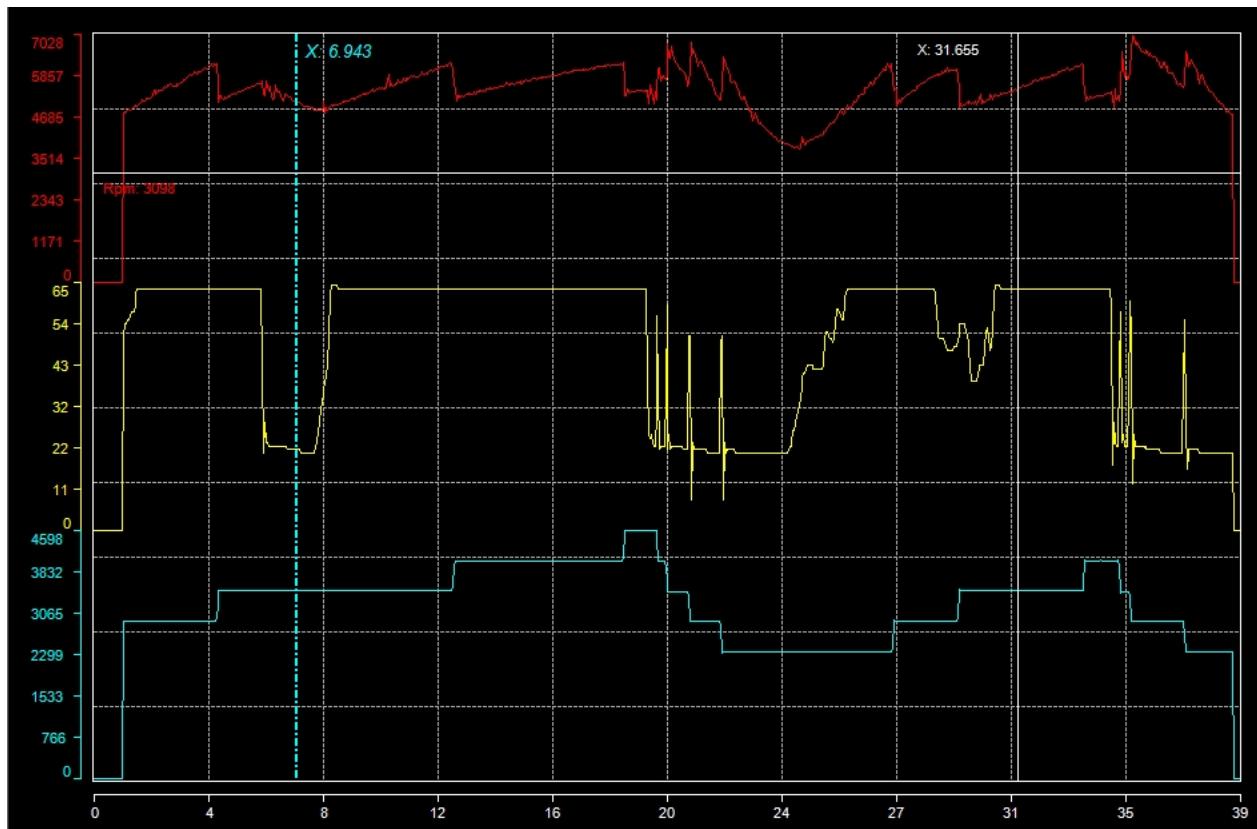
For Y axis, since it could be several different values here, screen size is used as reference value. In other words, with a cursor step of 1%, cursor will move up and down by steps representing 1% of the screen size.

Graphical properties of the graphic cursor (color, size) can be adjusted through the graphic configuration form. Please check the '['Cursors properties'](#)' section for more details.

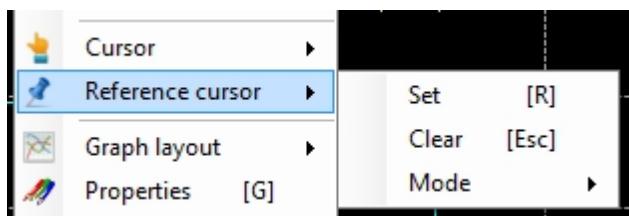
Created with the Personal Edition of HelpNDoc: [Create iPhone web-based documentation](#)

Reference cursor

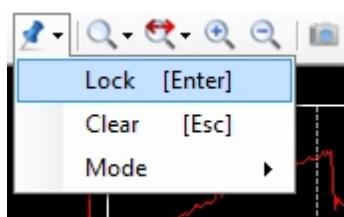
The reference cursor is an extension of the '[Main cursor](#)' function. It permits to compare values of all traces between two particular points in the graphic.



Click on the 'Reference cursor' \ Set' menu of tool bar to set the reference cursor position. This function is also accessible in the 'Reference Cursor\Set' command of the graphic window contextual menu, or by pressing the 'R' key of the keyboard.



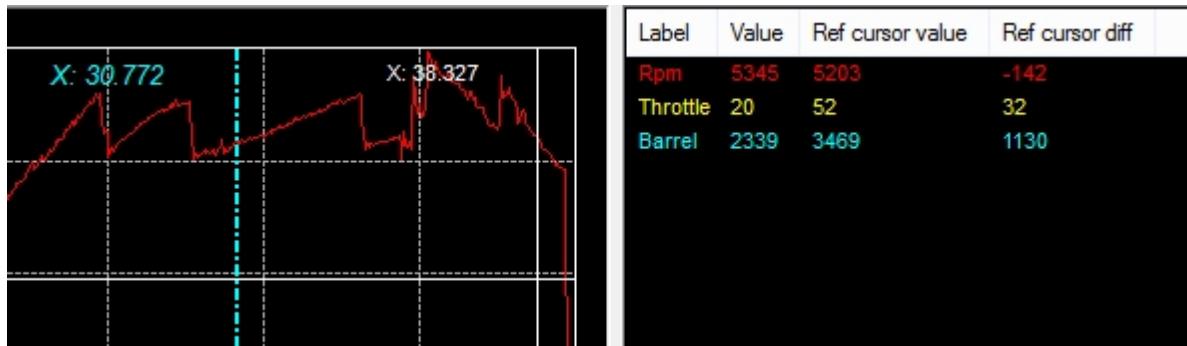
Place the reference at the position of your choice and then click the 'Reference cursor' \ Lock' command of the tool bar or the 'Reference Cursor\Lock' command of the graphic window contextual menu, or by press the 'Enter' to lock the reference cursor position and get the main cursor back on.



Main cursor being back, it can be place anywhere in the graph in order to compare the main cursor value and the reference cursor value.

Finally, once done with the comparison, click the 'Reference cursor \ Clear' command of the tool bar or the 'Reference Cursor\Clear' command of the graphic window contextual menu, or by press the 'Escape' to clear the reference cursor.

The biggest benefit of the reference cursor is that it shows both reference and main cursor values in the [legend](#), leading to have a direct cursors value comparison.



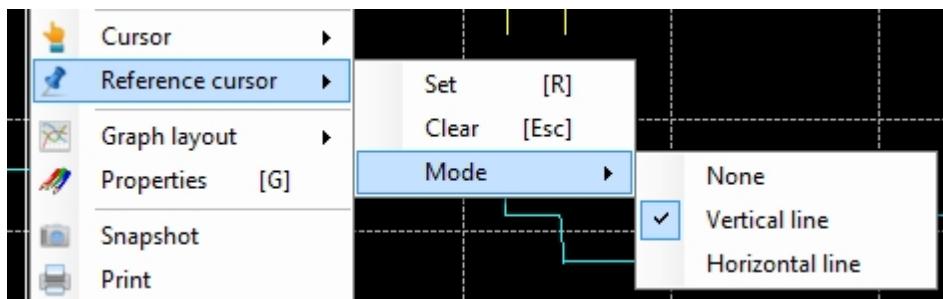
When the reference cursor is active, four statistics are added to the legend:

- Reference cursor value: The actual value of the trace at the reference cursor position
- Reference cursor value difference: The difference in value between the main and the reference cursor
- Reference cursor value difference percentage: The difference in percentage of the reference value between the main and the reference cursors values
- Reference cursor gradient: Value the gradient between the main and the reference cursor values. Gradient being defined as the difference in value over cursors X axis values difference.

There are three kinds of reference cursor:

- None: Reference cursor is disabled
- Vertical: Reference cursor is a vertical line. This mode is intended to be used with any main cursor type except the 'Horizontal line' main cursor type.
- Horizontal: Reference cursor is a horizontal line. This mode is intended to be used with any main cursor type except the 'Vertical line' main cursor type.

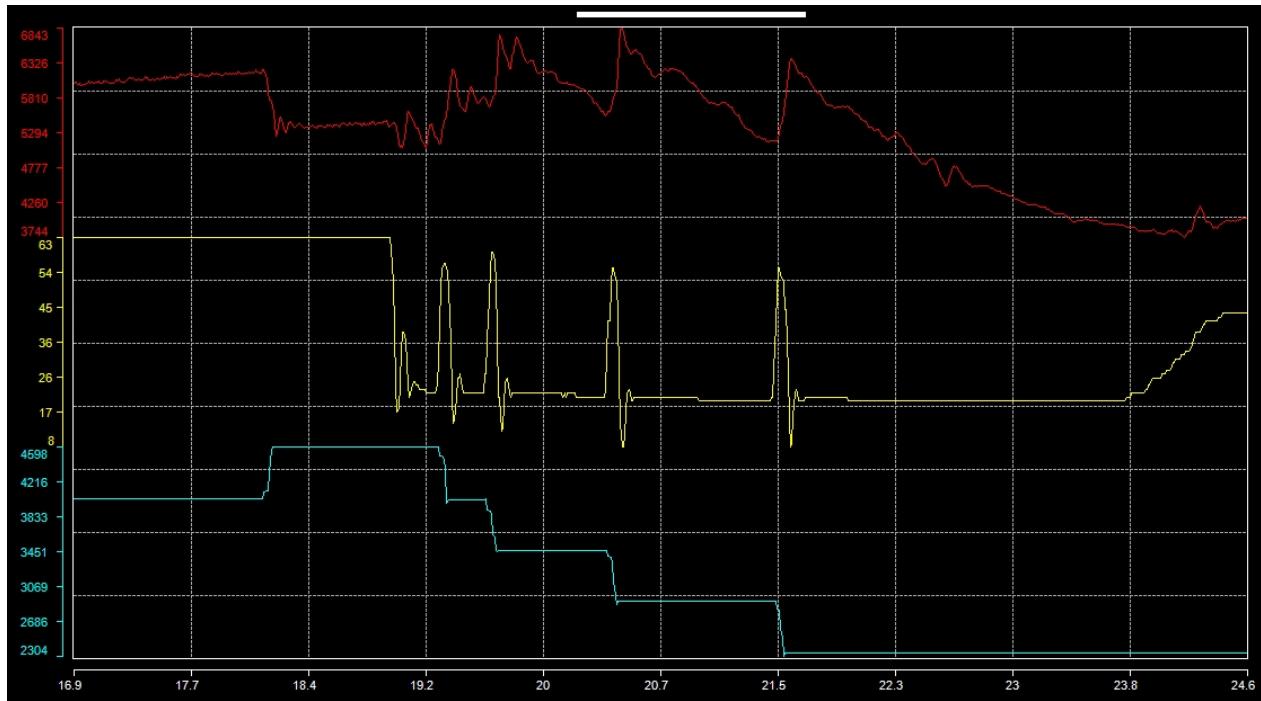
Use the 'Reference cursor \ Mode' command of the tool bar or the 'Reference Cursor\Mode' command of the graphic window contextual menu to change the current reference cursor type.



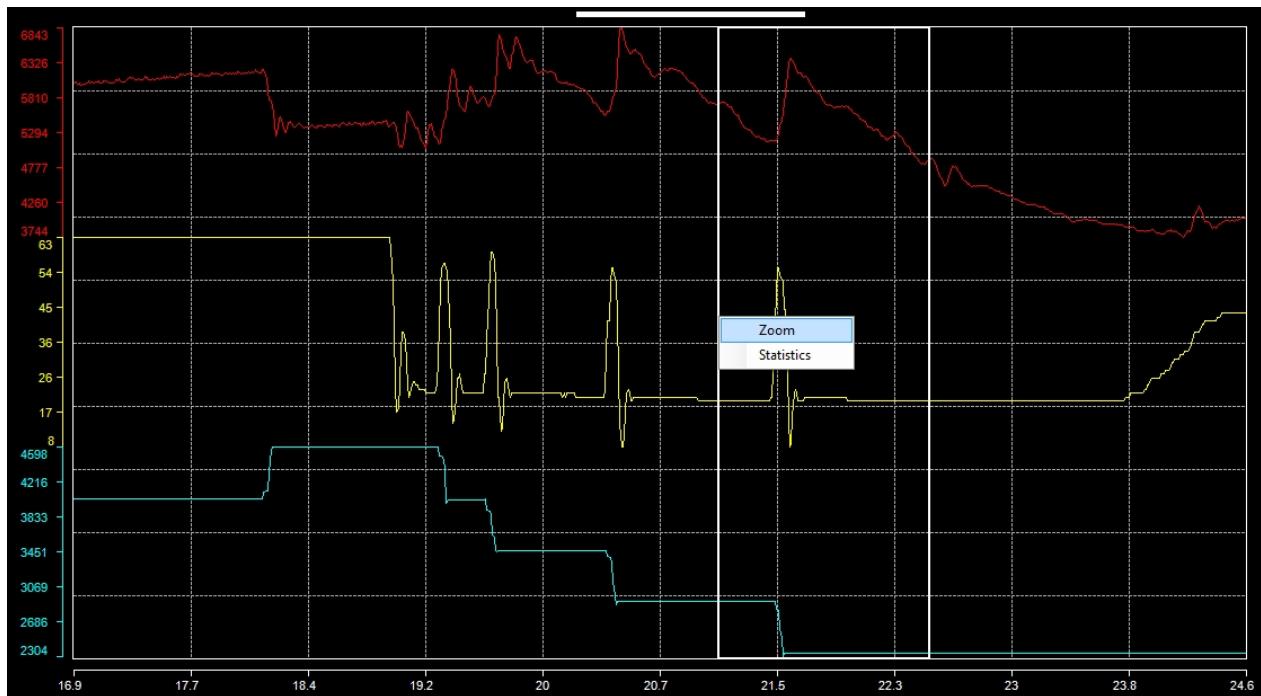
As per the main cursor, the reference cursor aspect can be customized in the graphic configuration form. Please check the ['Cursors properties'](#) section for more details.

Zoom

As per its name definition, zoom is a function permitting to zoom a particular graphic area and see traces of this area with greater details.



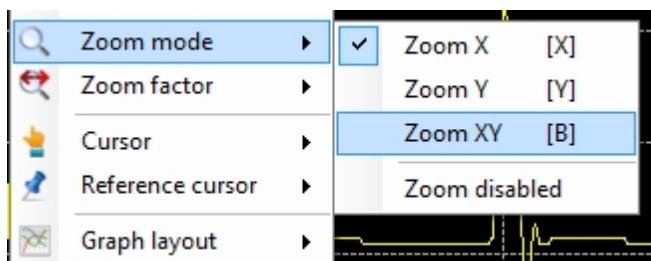
To zoom a particular area of the graphic, place the mouse cursor at the beginning, or at the end, of the area you want zoom in. Press the right mouse button and then drag the zoom box up the end of the zoom area. When the zoom area is defined click the 'Zoom' command of the contextual menu that pops up on the right mouse button release.



Four different zoom modes are available:

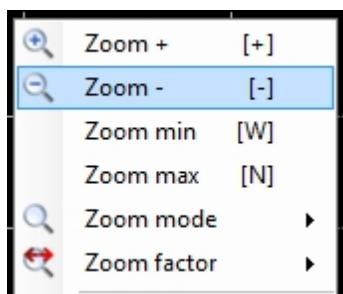
- Disabled: Zoom function is disabled
- Zoom X: Zoom is performed along the X axis only, Y axis being unchanged.
- Zoom Y: Zoom is performed along the Y axis only, X axis being unchanged.
- Zoom XY: Zoom is performed along both X and Y axis.

To switch to a different zoom mode, click the 'Zoom Mode'  command of the tool bar or the 'Zoom mode'  command of the graphic window contextual menu.



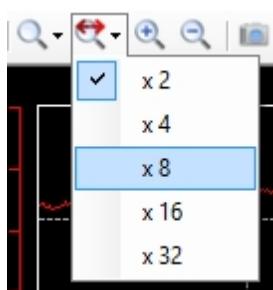
An alternative solution is to use zoom mode shortcut keys 'X', 'Y', 'B' for 'Zoom X', 'Zoom Y' and 'Zoom XY'

Another way to zoom a graphic in and out is to use 'Zoom +'  and 'Zoom -'  functions. Both of those functions are available either in the tool bar or in the graphic window contextual menu. They are also accessible by pressing '+' and '-' keys of the keyboard for 'Zoom +' and 'Zoom -'.



With this kind of zoom, the zoom area is defined by a factor applied around the [main graphic cursor](#) position. If the main graphic cursor is not set, the center of the screen is used as reference position.

The zoom factor can be defined either through the 'Zoom factor'  button of the tool bar or through the 'Zoom factor'  item of the graphic window contextual menu.



The zoom factor value represents the magnitude of zoom as a function of X axis values span (for zoom X and zoom XY) and the screen height (for zoom Y and zoom XY).

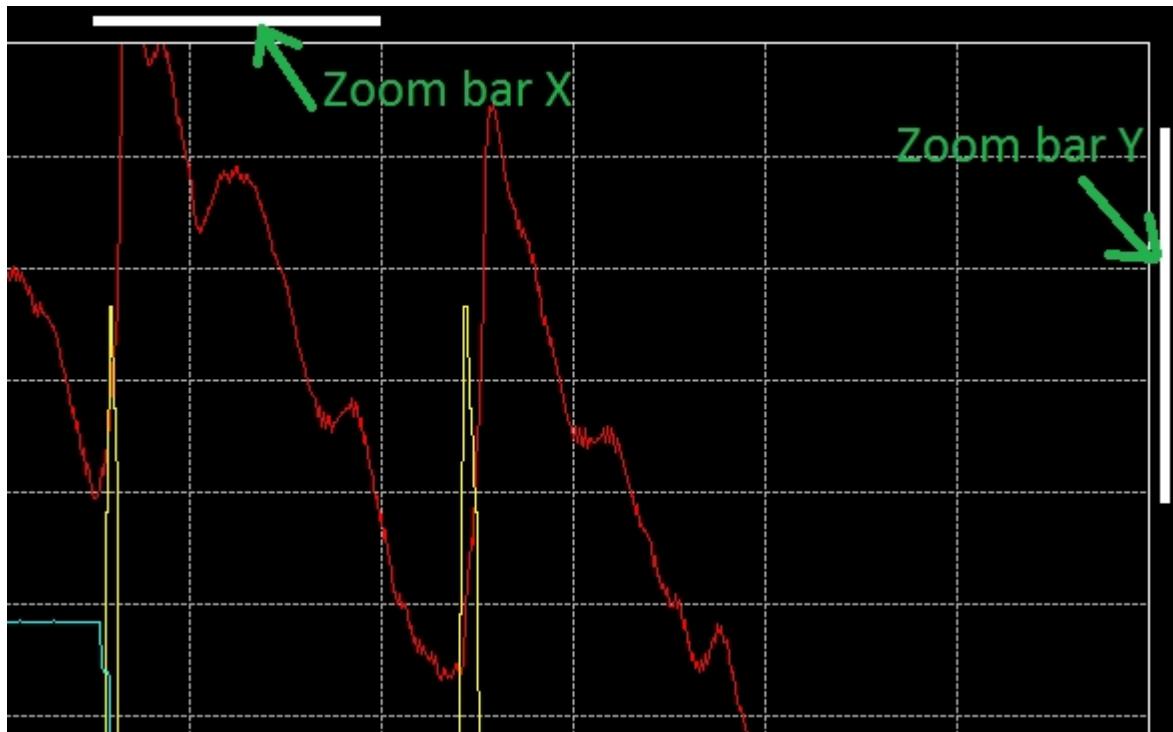
For example, if 80 seconds of data are plotted and the zoom factor is set to 'x 8', in the case of the 'Zoom X' mode, the Xaxis values span resulting of the zoom will be eight time smaller than the original span. So $80 / 8 = 10$ sec.

In the case of a zoom out, the X axis values span will be eight time bigger than the original one. So $10 \times 8 = 80$ sec.

The third and last method to zoom in and out is to use 'Zoom min' and 'Zoom max' functions. Those functions are not available in the tool bar (for clarity) but they are in the graphic window contextual menu and through the shortcut keys 'W' and 'N' for 'Zoom min' and 'Zoom max'.

The 'Zoom min' will revert to a zoom magnitude of 1, so the whole data will be shown as if there wasn't any zoom. While 'Zoom max' will do the exact opposite and apply the biggest zoom factor (x 32).

While the zoom function is operating, X and Y zoom bars are shown.



Obviously, the 'Zoom bar X' is shown only if the X axis is zoomed, so it won't be shown in 'Zoom Y' mode and the 'Zoom bar Y' is shown only if the Y axis is zoomed, so it won't be shown in 'Zoom X' mode.

Those zoom bars indicate two things:

- The magnitude of the zoom: bar width (for the zoom bar X) and bar height (for the zoom bar Y) are function of the zoom magnitude.
If the zoom factor is set to 'x 8', zoom bar X width will be eight time smaller than the whole graphic area width. The same logic is used for the zoom bar Y, with a zoom factor of 8, zoom bar Y height will be eight time smaller than the whole graphic area height.
- The position of zoomed area: bar left position (for zoom bar X) and bar top position (for the zoom bar Y) are function of the zoomed area position
if the whole data length is 60 seconds and the zoom area starts at the second 30, zoom bar X left position will right on the middle of the graphic area. The same logic applies as well for the zoom bar Y.

Zoom bar X and Y have also second purpose, they can be moved by user in order to move the zooming area.

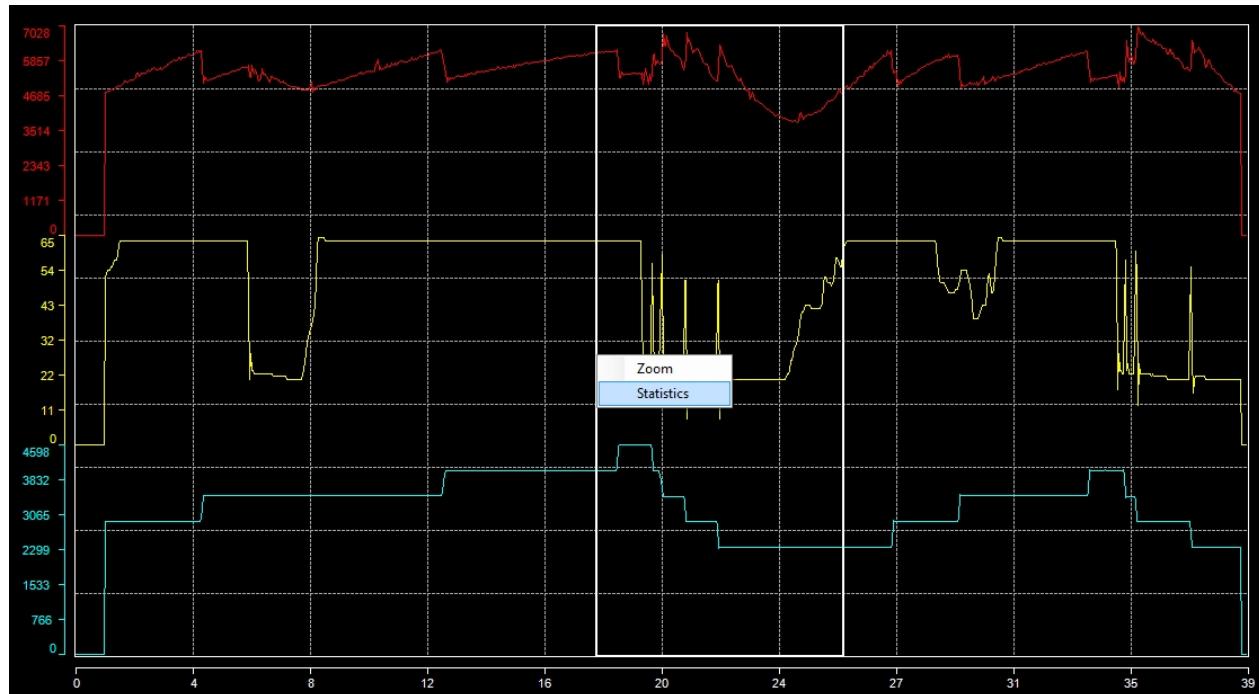
Place the mouse cursor over one zoom bar and the cursor will transform either in 'east-west arrows' cursor

for the zoom bar X or in 'north-south arrows' cursors for the zoom bar Y. Press the mouse right button and drag the zoom bar at the position you want to change the zooming area.

Statistics

Another useful feature of the analysis window is statistics computation.

As per the [zoom](#) function, right click is graphic area and drag the zoom box up to end of area in which you want compute statistics. Release the mouse right button and click the 'Statistics' command of the contextual menu that has popped up. Prior to do that, make sure the zoom mode is not 'Zoom disabled', otherwise the zoom box will not appear.



The statistics window appears

Series statistics								
Label	Min X	Min	Max	Max X	Avg	Avg Abs	Std dev	Samples
Rpm	24.172	3744	6843	20.486	5166.192	5166.192	844.217	5515
Throttle	20.486	8	63	17.451	34.737	34.737	17.472	5515
Barrel	21.546	2304	4598	18.506	3040.783	3040.783	850.912	5515

In this statistics windows we can find different statistics about plotted data

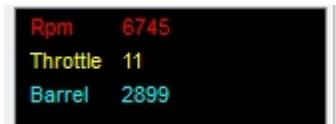
- Min X: X axis value at the minimum trace value
- Min: Minimum value of the trace inside the zoom box
- Max: Maximum value of the trace inside the zoom box
- Max X: X axis value at the maximum trace value
- Avg: Average of all trace values inside the zoom box
- Avg abs: Average of all trace absolute values inside the zoom box
- Std dev: Standard deviation of all trace values inside the zoom box
- Samples: Trace samples count used for statistics computation

Those statistics can even be copied into the Windows clipboard in order to be pasted into an external application such as Microsoft Excel.

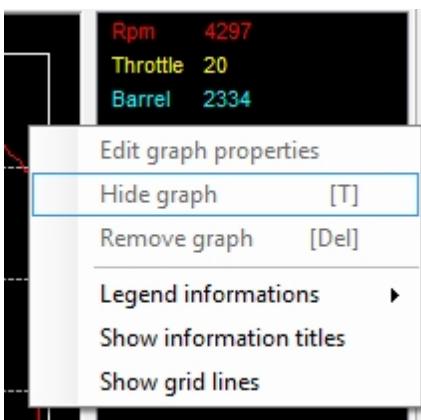
Select all cells of the statistics grid that are of your interest and press Ctrl+C on your keyboard. All selected cells values are now in the Windows clipboard and are ready to be used in any application accessing the clipboard.

Legend

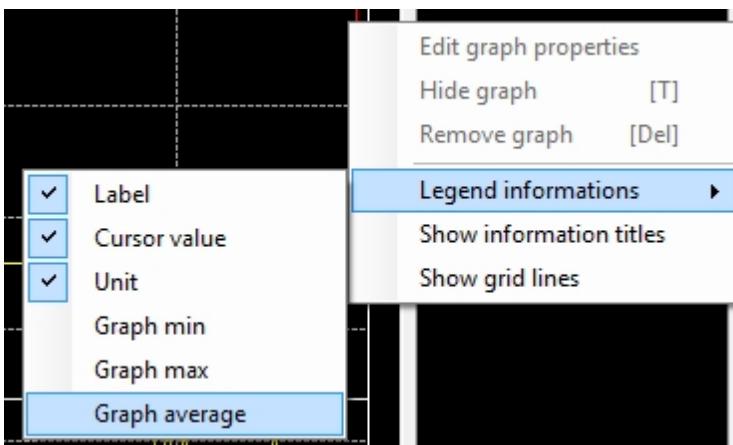
As briefly explained in the '[Main cursor](#)' and '[Reference cursor](#)' sections, legend shows plotted trace values at the position of the main [cursor](#).



If you make a right-click on the legend, the legend contextual menu will pop up and shows all available legend options



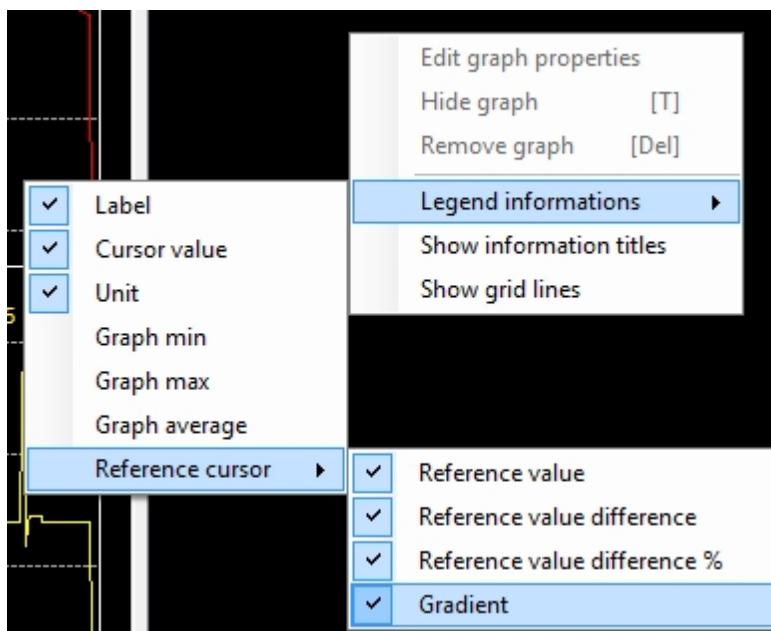
The 'Legend information' menu propose you to define what information you want to see in the legend.



There are six information that you can make visible or invisible. Just check infos that you want to see, unchecked info will be invisible.

- Label: Titles of the graphic traces set into the graphic configuration form
- Cursor value: Traces values at the position of the [main cursor](#)
- Unit: Trace units set into the graphic configuration form
- Graph min: Traces minimum value visible in the graphic area
- Graph max: Traces maximum value visible in the graphic area
- Graph average: Traces average value of all sample visible in the graphic area

In case of the [reference cursor](#) function usage, there are an extra bunch of infos that you can make visible.

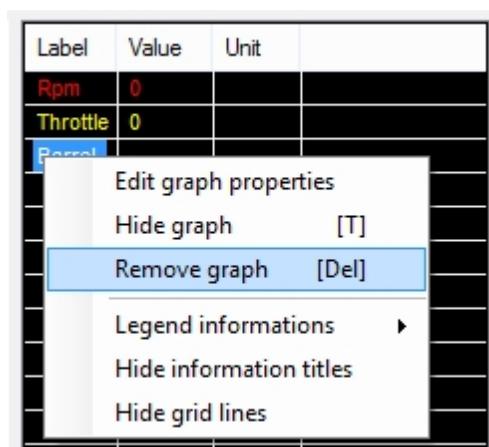


- Reference cursor value: The actual value of the trace at the reference cursor position
- Reference cursor value difference: The difference in value between the main and the reference cursor
- Reference cursor value difference percentage: The difference in percentage of the reference value between the main and the reference cursors values
- Reference cursor gradient: Value the gradient between the main and the reference cursor values. Gradient being defined as the difference in value over cursors X axis values difference.

Menus 'Show legend information titles' and 'Show grid lines' permit to show and hide the legend header and legend grid lines.

Label	Value	Unit	
Rpm	0		
Throttle	0		
Barrel	0		

A right-click on a legend item will enable trace related function of the legend



Menu 'Edit graph properties' will open the selected trace 'Detailed serie properties' windows in order to fine

tune graphical properties of a particular trace.

'Hide graph' command will hide the selected trace in the graphic area. Alternatively, once a trace hidden, this menu becomes 'Show graph' in order to re-enable a hidden trace. This command is also available by pressing the 'T' key of the keyboard.

'Remove graph' will definitively remove a trace from the graphic area. Be careful, this operation is not reversible ! Once a trace removed it is gone for ever and you will have to add it into the graph again.

The legend panel can be hidden at any time by clicking the 'Show/Hide legend' button  of the tool bar. Once hidden click again the 'Show/Hide legend' button  to re-open the legend panel.

Some more legend properties can be adjusted through the graphic configuration form. Please check the ['General properties'](#) section for more details.

Created with the Personal Edition of HelpNDoc: [Generate EPub eBooks with ease](#)

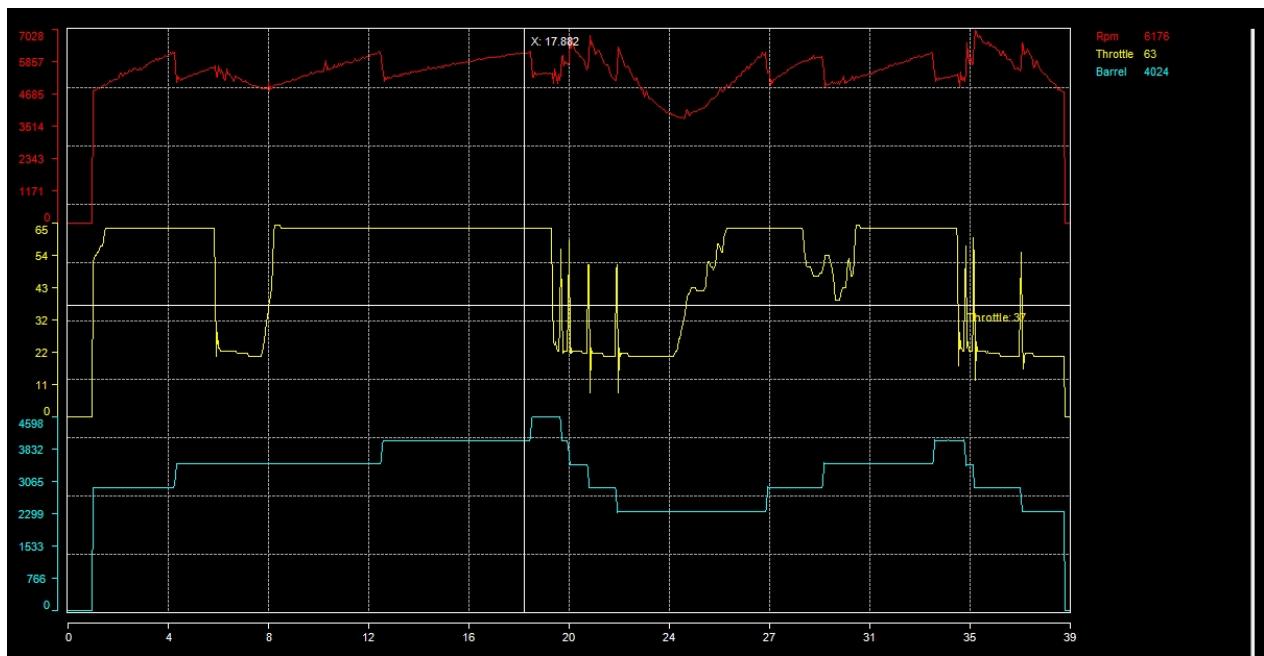
Snapshot

Snapshot function makes a picture of your graphic that can be reused in a presentation, or an email or anything else.

Snapshot outputs a bitmap picture file (*.bmp), this picture including legend but excluding the channel list.

To make a snapshot of your graphic, simply click the 'Make a snapshot image of the graphic' , set the name and path of the output picture in the file saving dialog and click 'Save'.

Example of a graphic snapshot.

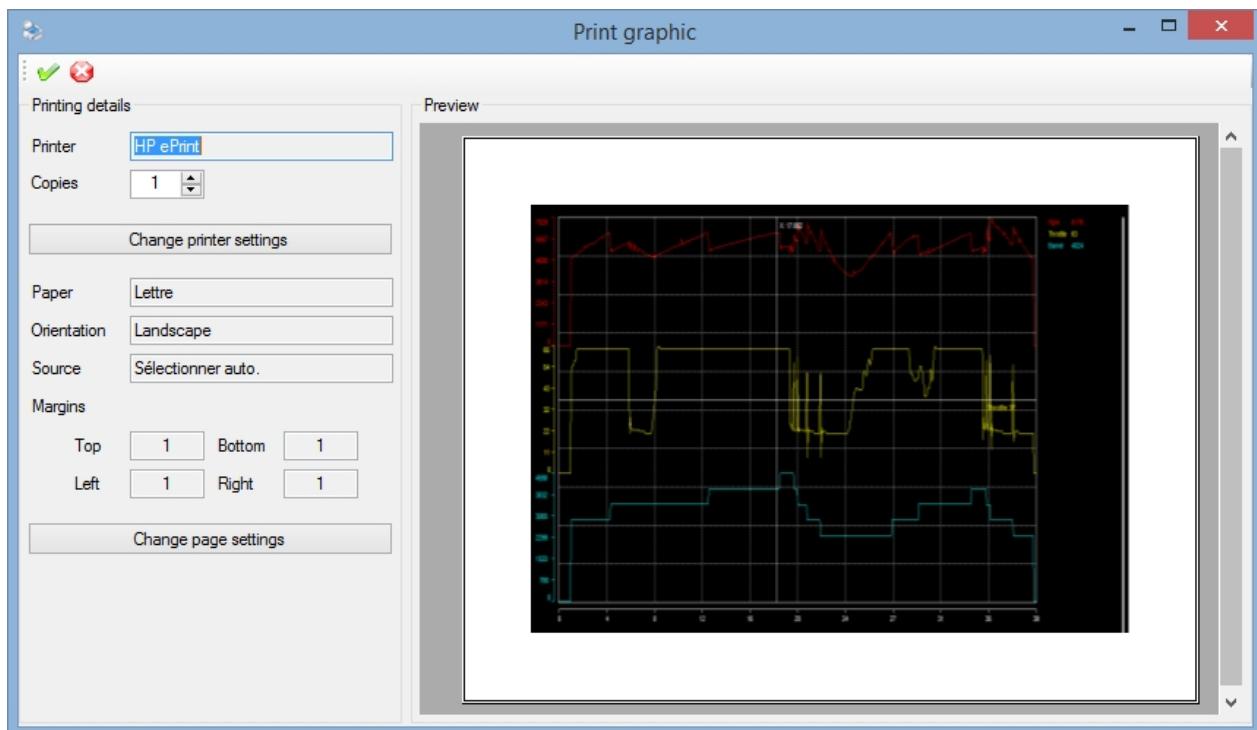


Created with the Personal Edition of HelpNDoc: [Easy CHM and documentation editor](#)

Print

The 'Print' function permits to print your graphic out.

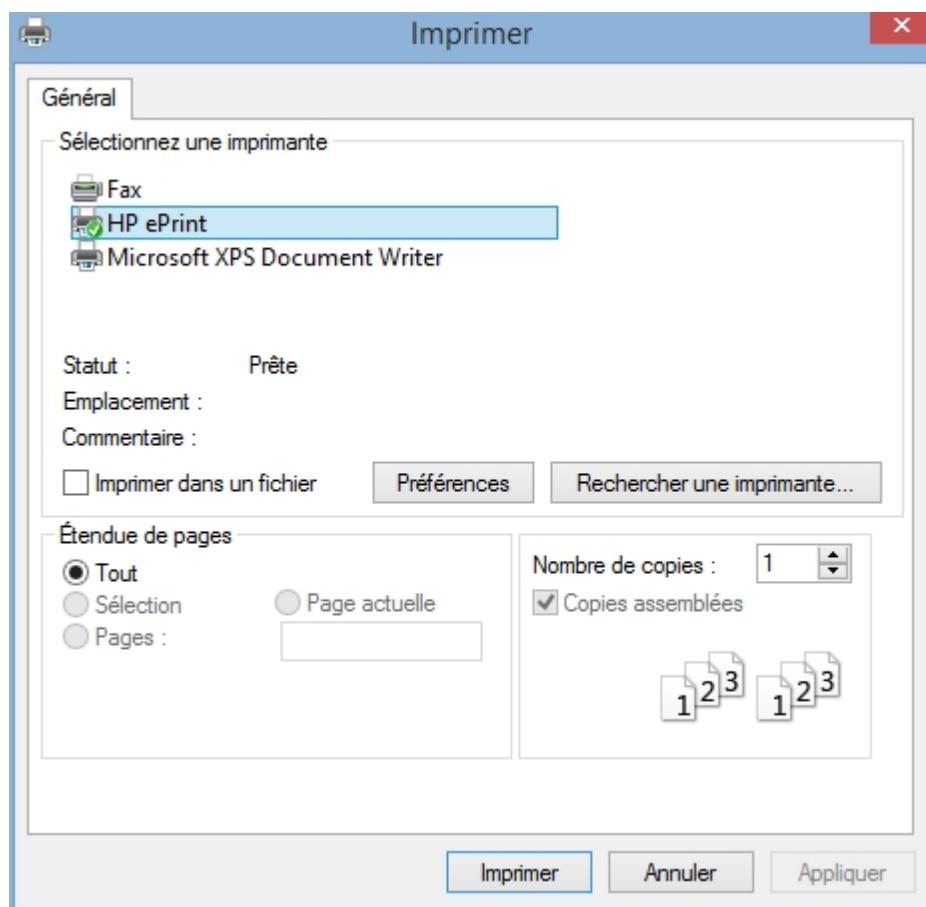
Click the 'Print graphic'  button to open the print preview window.



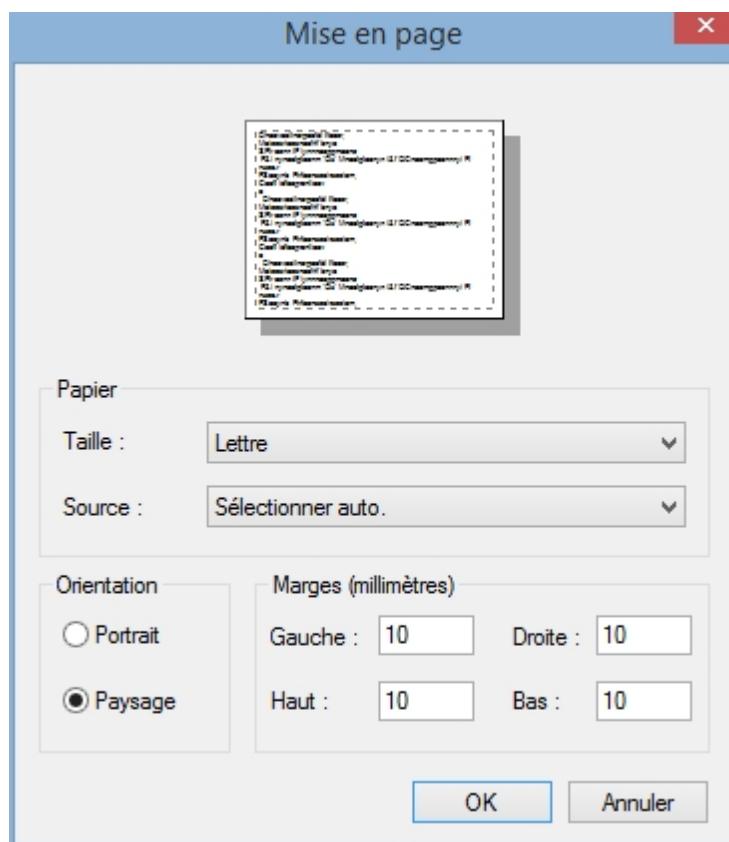
The right side panel of print preview form contains printing details such as printer name, number of copies, paper format and orientation.

The left side panel shows an actual preview of the document as it will be once printed out.

Set the number of copies to print in the field 'Copies'. To change the printer and its properties, click the 'Change printer settings' button.



Click the 'Change page settings' button to change paper orientation or adjust margins.



Once done with all settings, click the 'Print'  button to print the page.

The 'Cancel'  button, simply close the form. It doesn't cancel the printing request if such request has been sent to a printer.

Created with the Personal Edition of HelpNDoc: [Produce Kindle eBooks easily](#)

Graph window settings

This section regroup graphical properties configuration documentation of the graphic window

Created with the Personal Edition of HelpNDoc: [Easily create EPub books](#)

Channel list

The channel list shows the list of data channels contained in the [data file](#) loaded.

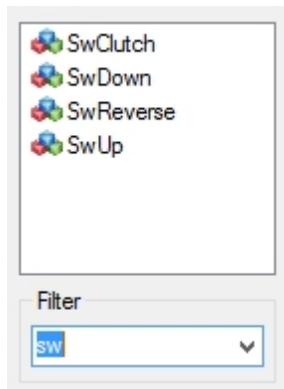
Channel list is located in the left side panel of the analysis window. It is visible by default but it can be hidden by clicking on the 'Show/Hide channel list'  button of the tool bar. Once hidden, click again on this button to get the channel list back.

Double click on a channel to add it in the graphic. This can also be done by a 'drag & drop' of a channel (or a bunch of channels) from the channel list to the graphic area.

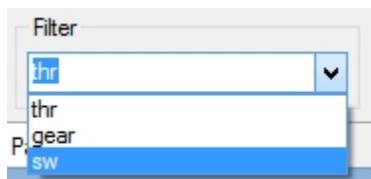
Alternatively, select channels that you want to add in the graphic and press 'Enter'.



At the bottom of the channel list there is the 'Filter' area. This area permits to filter channel names shown in the list. Just type a name or a part of a name that you are looking for and press enter. If the character string that you typed in is contained into one or more channels, only those channels will be shown in the list.



Filter is text box is actually a list that stores the last ten filters used. Just select that filter to reuse it.



Clear the filter text box to reset the filter and get back the whole channel list.

Graphic layout

Term 'graphic layout' means the way graphic series (or traces) are set into the graphic.

There are three possible layouts:

- Overlay: All series are overlaying each other on the full graphic area height.
- Parallel: Series are evenly arranged onto the graphic area.

If there are four series to plot, each serie will be plotted over 25% of the whole graphic area height. If there are five series, each serie will use 20% of the graphic height, 10% for ten series, so forth and so on...

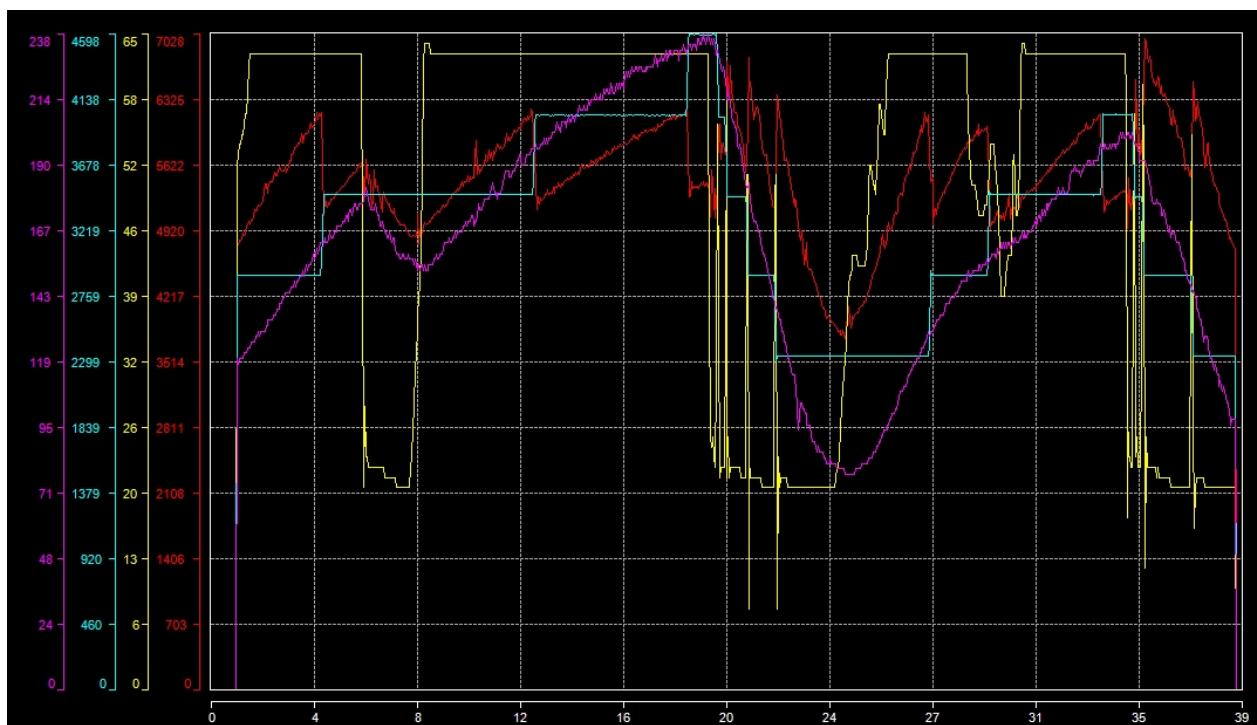
- Custom: Series top and bottom position are defined by the user.

Series are not necessarily evenly set, a serie can fully or partially overlap one or more series.

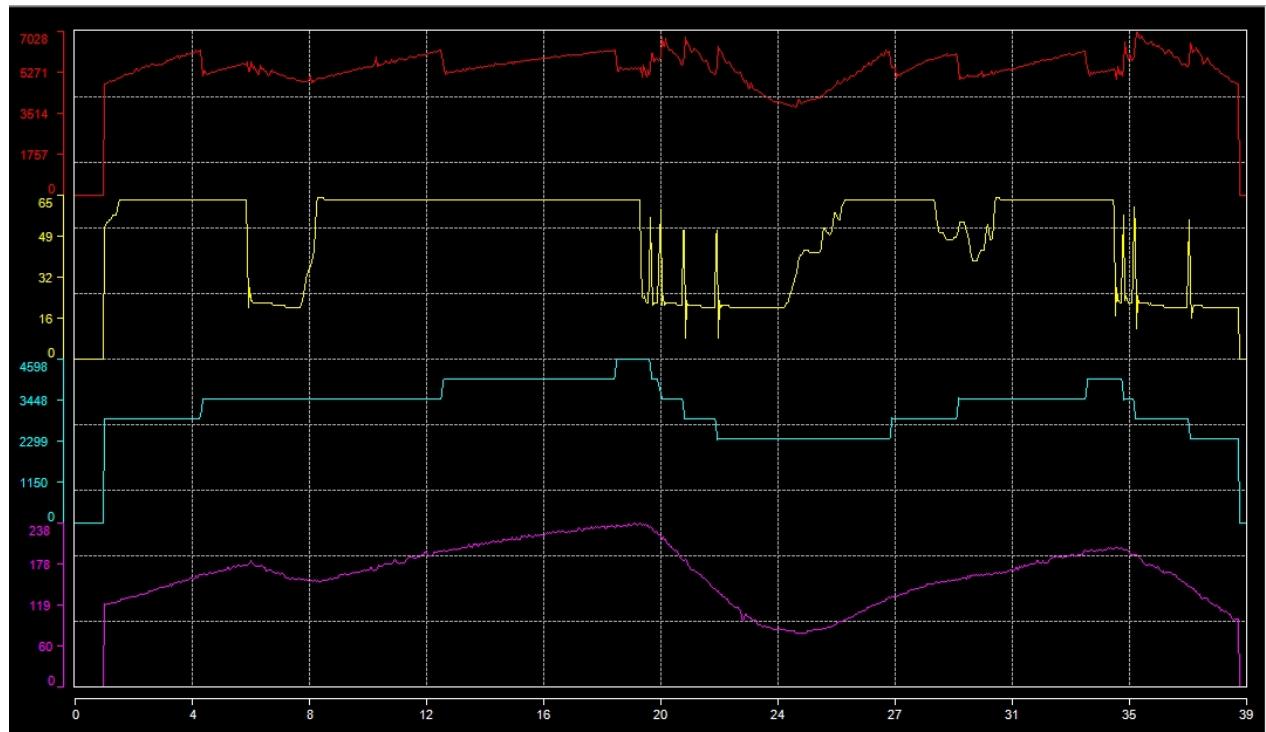
Click the 'Graphic layout mode'  button of the tool bar to change the current layout. This command is also available through the 'Graph layout'  item of the graphic contextual menu.

Layout mode can also be changed using 'O', 'P' and 'C' shortcut keys for 'Overlay', 'Parallel' and 'Custom' modes.

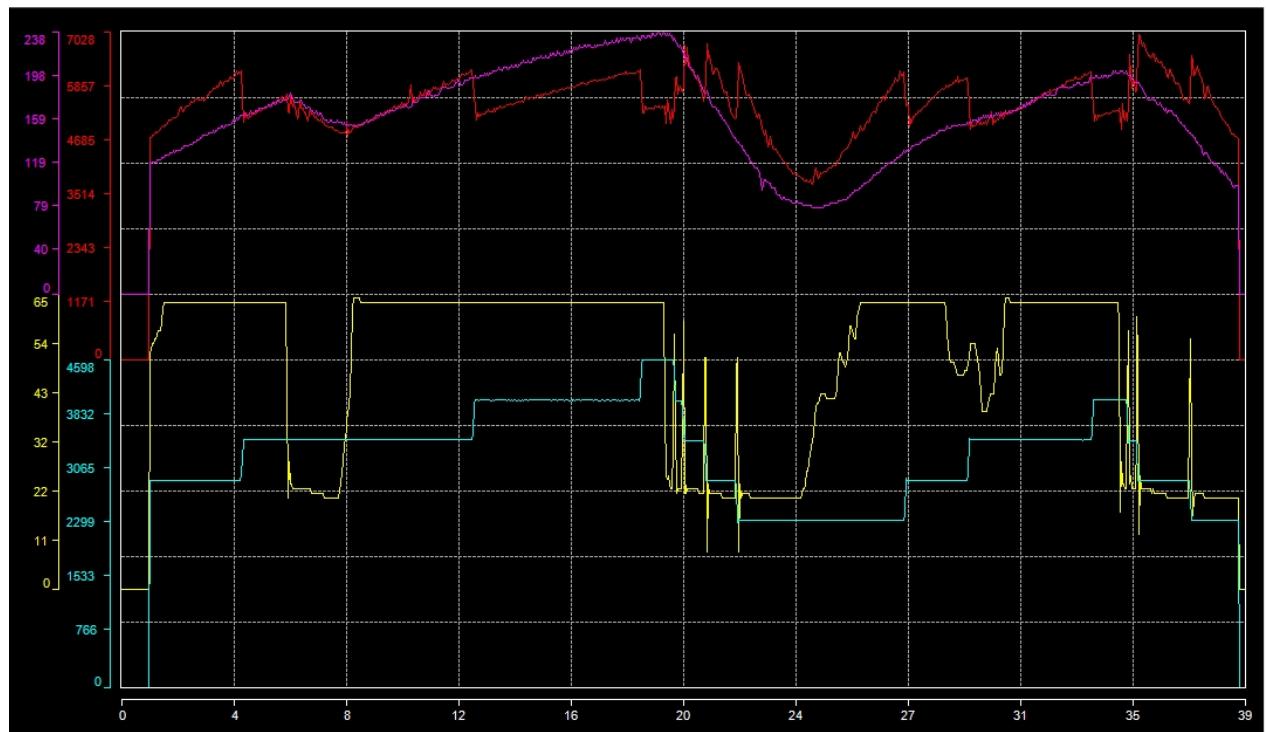
Graphic in 'Overlay' layout mode:



Graphic in 'Parallel' layout mode:



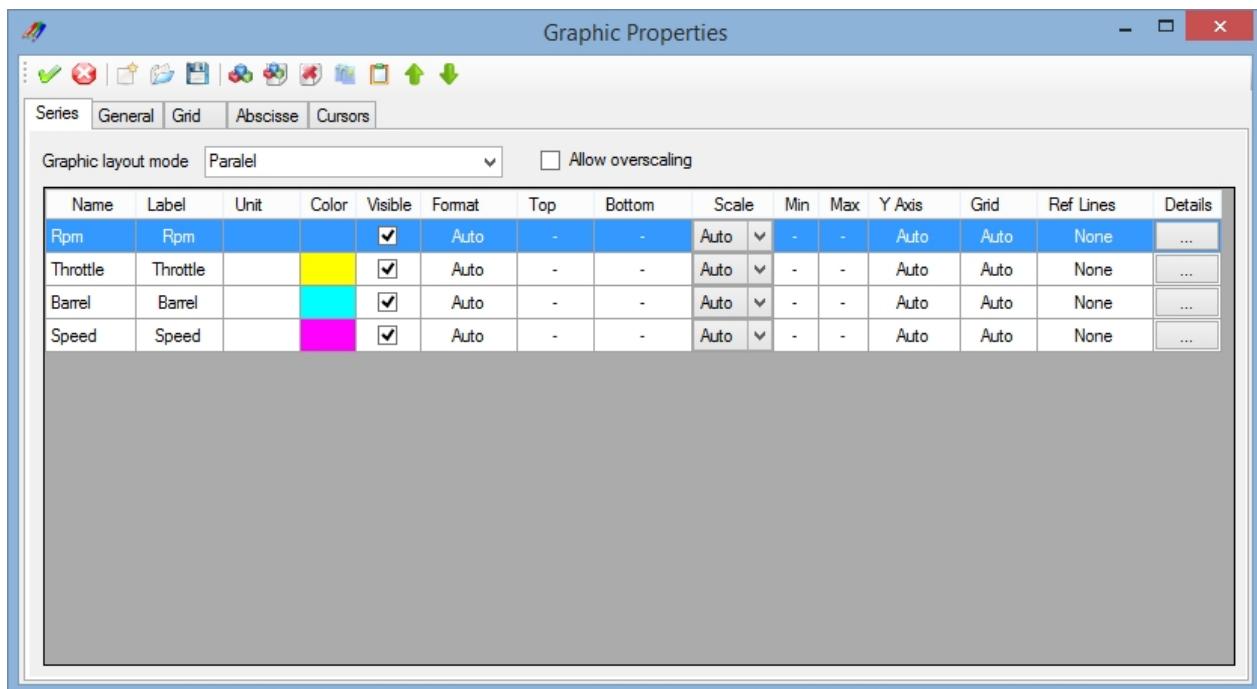
Graphic in 'Custom' layout mode:



Graphic window configuration

The graphic configuration window permits to edit all properties related to the the graphic window.

Click the 'Edit graph properties'  button of the tool bar to open the graphic configuration window. This command is also available through the 'Properties'  item of the graphic contextual menu. Alternatively, press the 'G' key of the keyboard to open the configuration window.



The graphic configuration window is composed by a tool bar and a multi-tabs window.

Each tab of the multi-tab is dedicated to a particular configuration section. There five different sections:

- Series: Generic properties of the graph series (or trace)
- General: General properties of the graphic window
- Grid: Graphic window grids properties
- Abscisse: Properties related to the graphic abscisse (X axis)
- Cursors: Graphic properties of main and reference cursors

Tool bar contains most common configuration commands.

 **Apply:** Apply graphic configuration changes

 **Cancel:** Cancel graphic configuration changes

 **New:** Create a new graphic configuration file

 **Open:** Open a graphic configuration file (*.xgw)

 **Save:** Save the graphic configuration file (*.xgw)

Check the '[Generic series properties](#)' section for details of the rest of tool bar commands

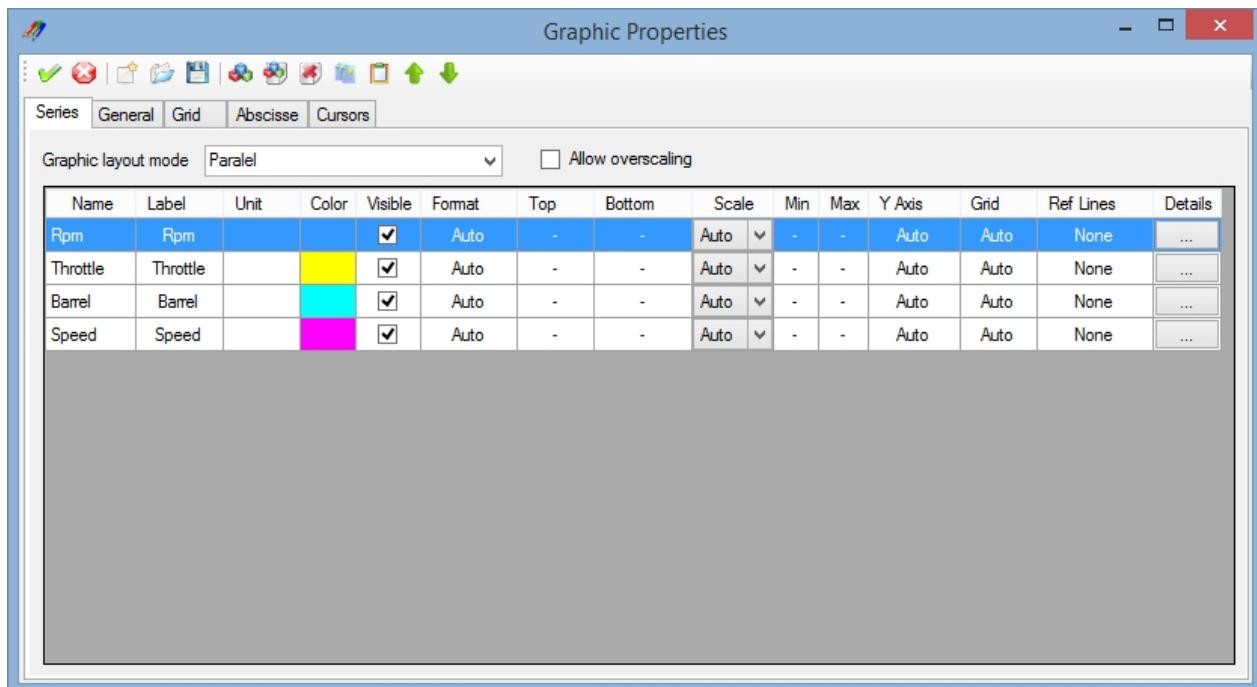
Created with the Personal Edition of HelpNDoc: [Easy EBook and documentation generator](#)

Generic series properties

The 'Series' tab of the graphic configuration window contains generic properties of graphic series (or trace)

List graphic layout mode permits to change the current [layout mode](#) of the graphic.

All graphic series (or traces) are shown in the series grid.



First cell apart, all cells value can be modified in order to set a particular property of a graphic serie.

- Name: Name of data channel as shown in the channel list used as data source of the serie.
- Label: Title of the serie in the graphic legend.
- Unit: Serie value unit shown in the graphic legend.
- Color: Trace color of the serie in the graphic and in the legend.
- Visible: Serie visible flag. Check the box to make the serie visible, uncheck it to hide the serie.
- Format: Double click on this cell to open the serie value format dialog.
- Top: Serie top position in the graphic area. This setting is only available with the 'Custom' layout mode.
- Bottom: Serie bottom position in the graphic area. This setting is only available with the 'Custom' layout mode.
- Scale: Serie scaling mode, either 'Auto' or 'Manual'. See 'General serie properties' section for more details.
- Y Axis: Double click on this cell to open the serie Y axis properties setting dialog.
- Grid: Double click on this cell to open the serie custom grid properties setting dialog.
- Ref lines: Double click on this cell to open the serie reference lines properties setting dialog.
- Details: Click on this cell to open the whole detailed serie properties dialog.

Some commands of tool bar are specific to the 'Series' tab

 **Channel list:** Open the channel list to add graphic series

 **Create serie:** Create new graphic serie

 **Delete serie:** Delete a graphic serie

 **Copy:** Copy a graph serie

 **Past:** Past a graph serie

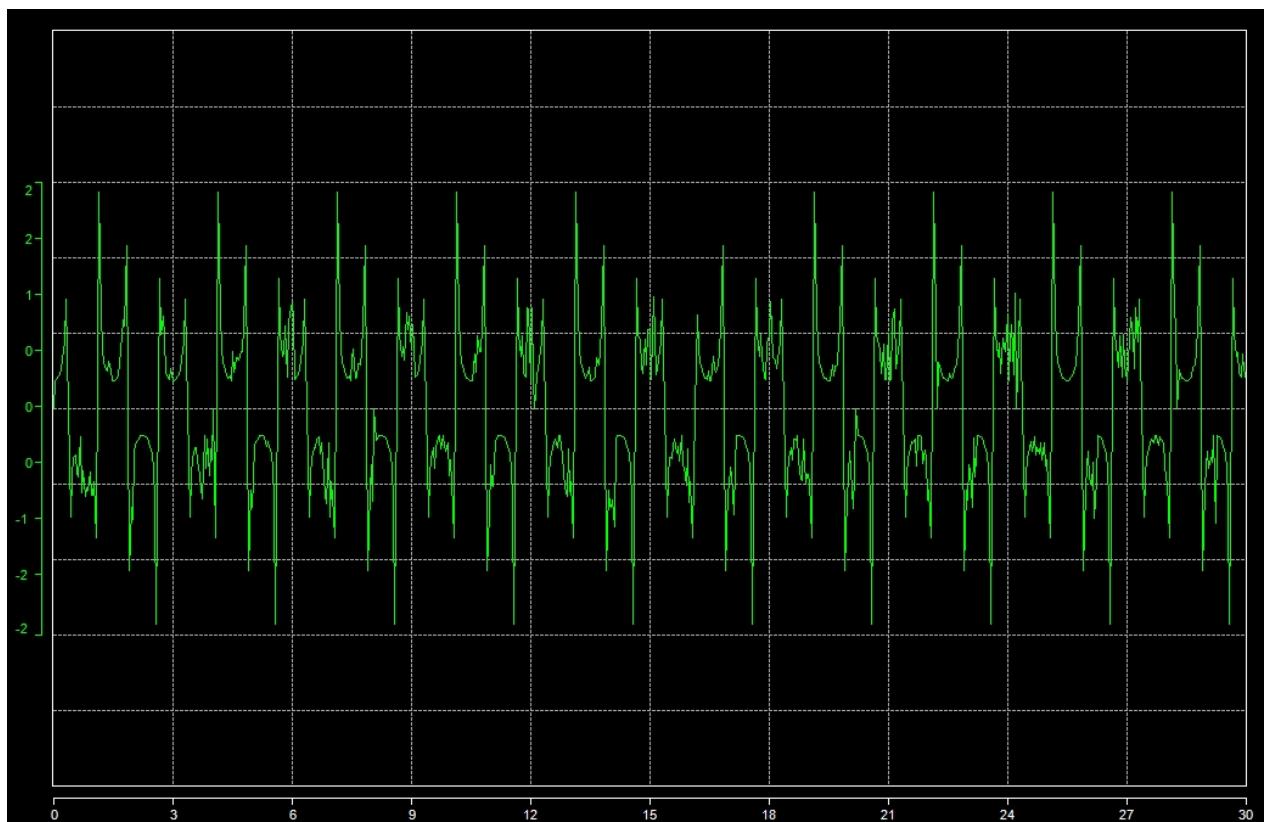
 **Move up:** Move a serie up in the grid

 **Move down:** Move a serie down in the grid

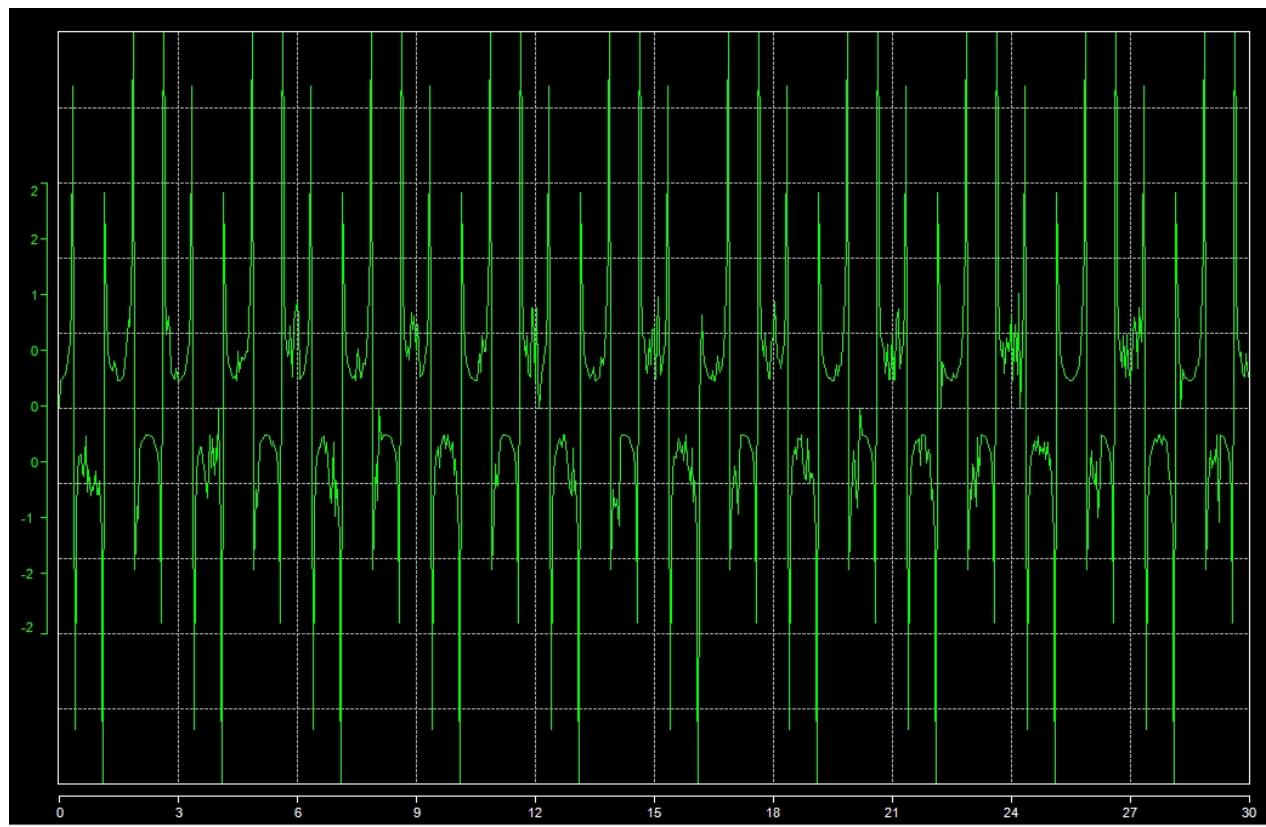
Check box 'Allow overscaling' enable over scaling for all graphic series.

Over scaling is applied on series only for the 'Manual' serie scaling mode. If the scale span set by the user is smaller than the actual serie values span, the 'Overscaling' flag will defined whether or not, out of scale sample should be drawn on the graphic.

Non over scaled graphic:

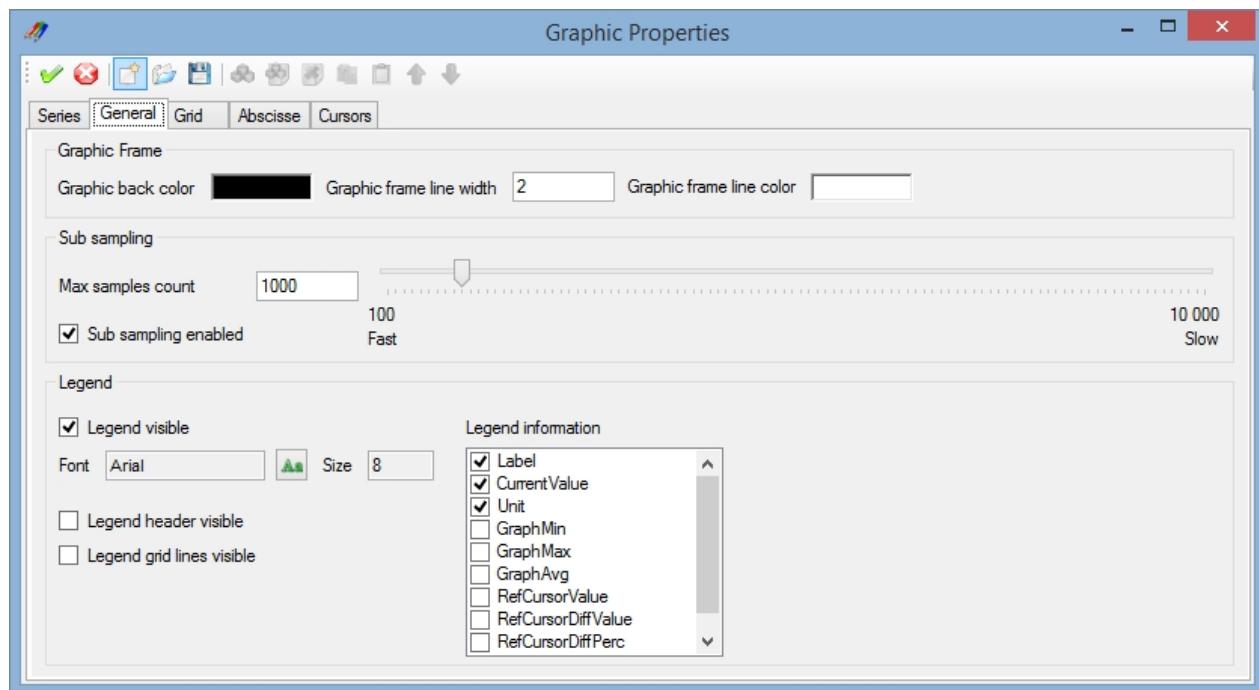


Over scaled graphic:



General properties

The 'General' tab contains general graph window properties



Graphic frame

This panel contains properties of the graphic frame

- Window back color: Double click on the colored square to open the color selection dialog and define the graph window back color.
- Graphic frame line width: Set the desired width of the graphic frame lines
- Graphic frame line color: Double click on the colored square to open the color selection dialog and define the graphic frame lines color.

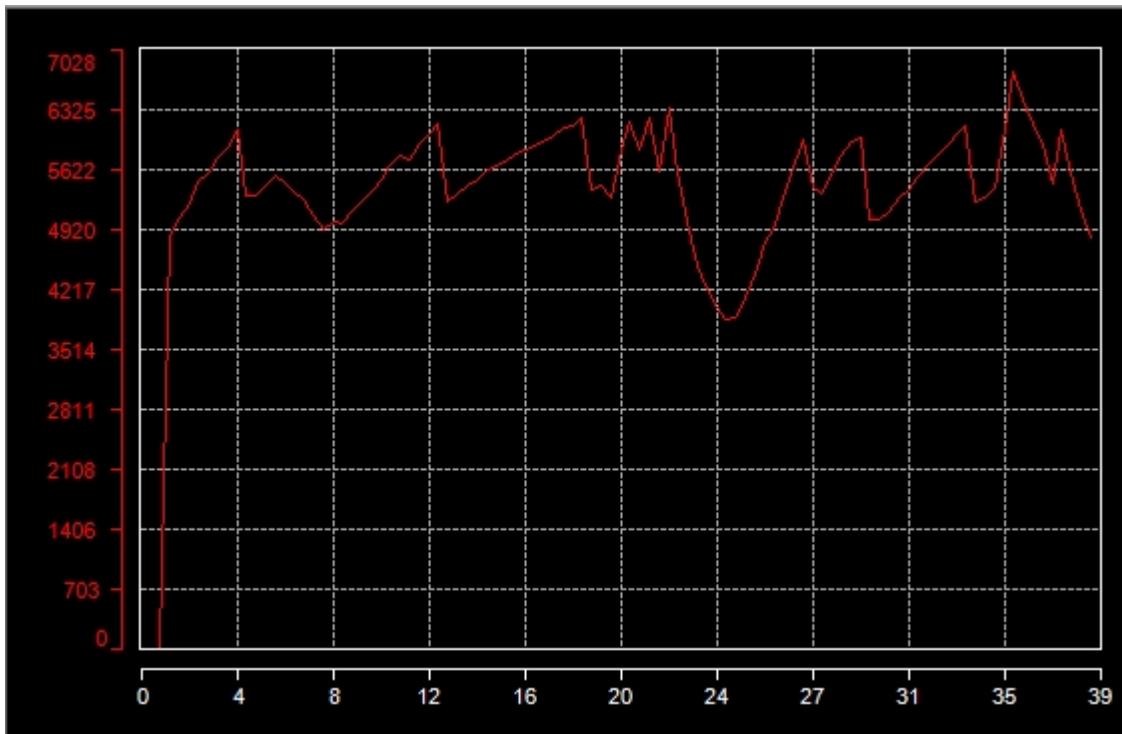
Sub sampling

Sub sampling property defines the maximum number of samples that will drawn in the graphic for each serie.

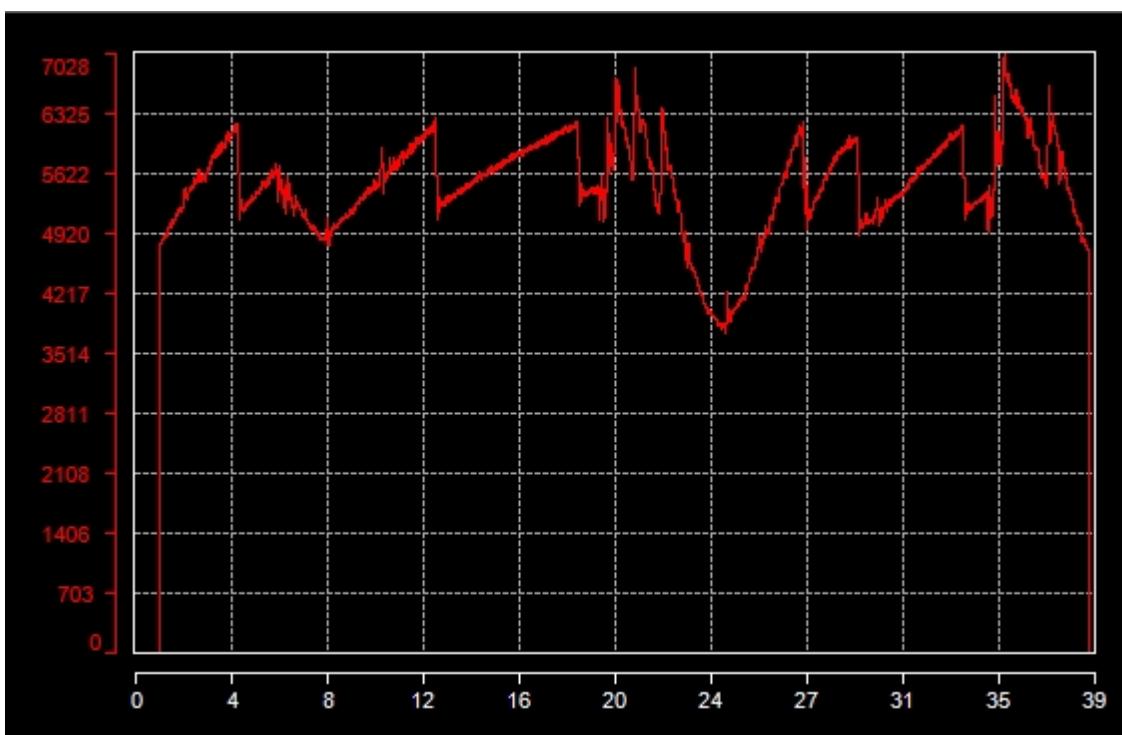
In order to accelerate graphic drawing, number of samples drawn can be limited to a certain value that keeps drawing time within a reasonable limit without altering the graphic accuracy. Bigger is the drawn samples count longer will be the graphic drawing.

Tick the 'Sub sampling enabled' box to enable the sub sampling and move the sub sampling cursor or type a value into the 'Max samples count' text box to define the sub sampling level.

Sub sampling set to 100 (100 samples drawn)



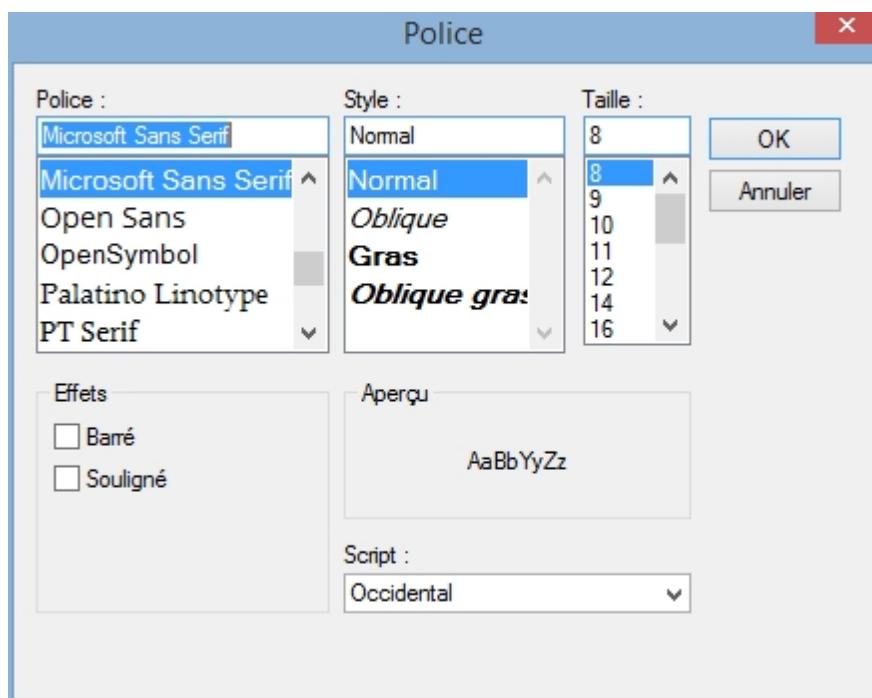
Sub sampling disabled (26 000 samples drawn)



Legend

In addition of [legend](#) properties that can be directly set in the graphic window (infos and header/grid lines visible), it is here possible to change the font used for legend items.

Click the 'Font'  button to open the font selection dialog, select a font and set its size and attributes and click 'OK'



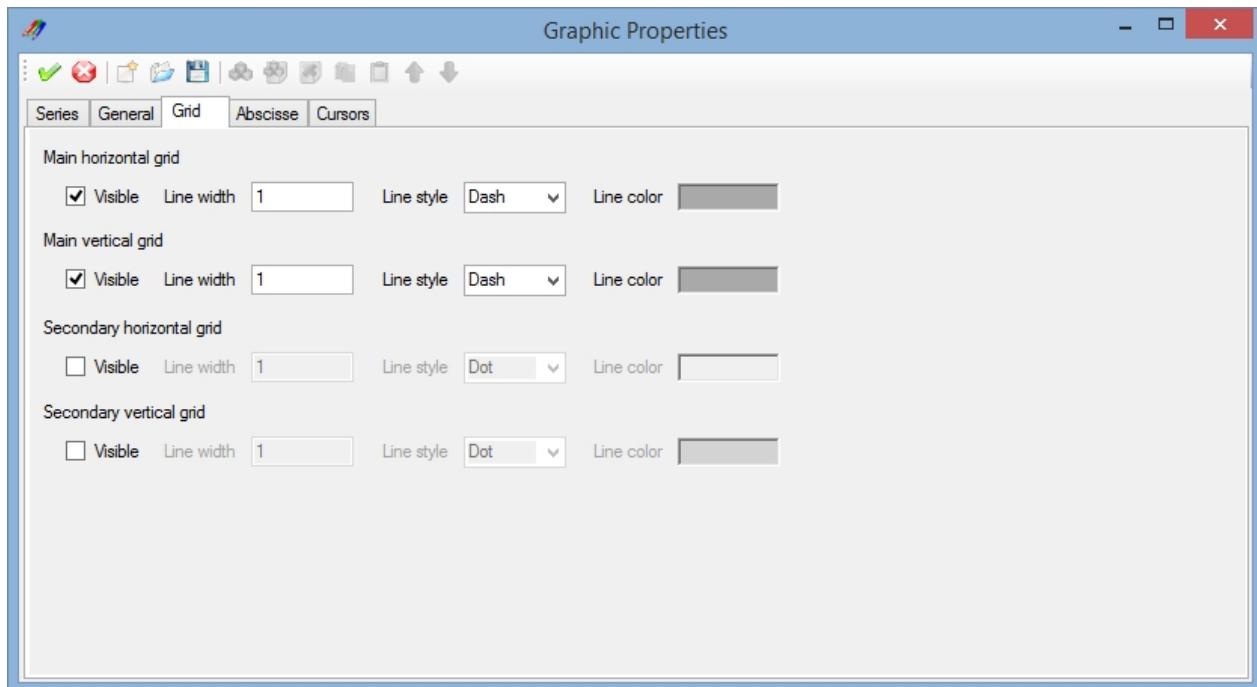
Created with the Personal Edition of HelpNDoc: [Full-featured multi-format Help generator](#)

Graphic grids properties

In order to enhance the analysis experience, vertical and horizontal grids are drawn in background of the graphic area.

There are two kinds of grid:

- Main grid: Main grid (horizontal and vertical) chops the graphic area in ten parts
- Secondary grid: Secondary grid (horizontal and vertical) chops the graphic area in twenty parts

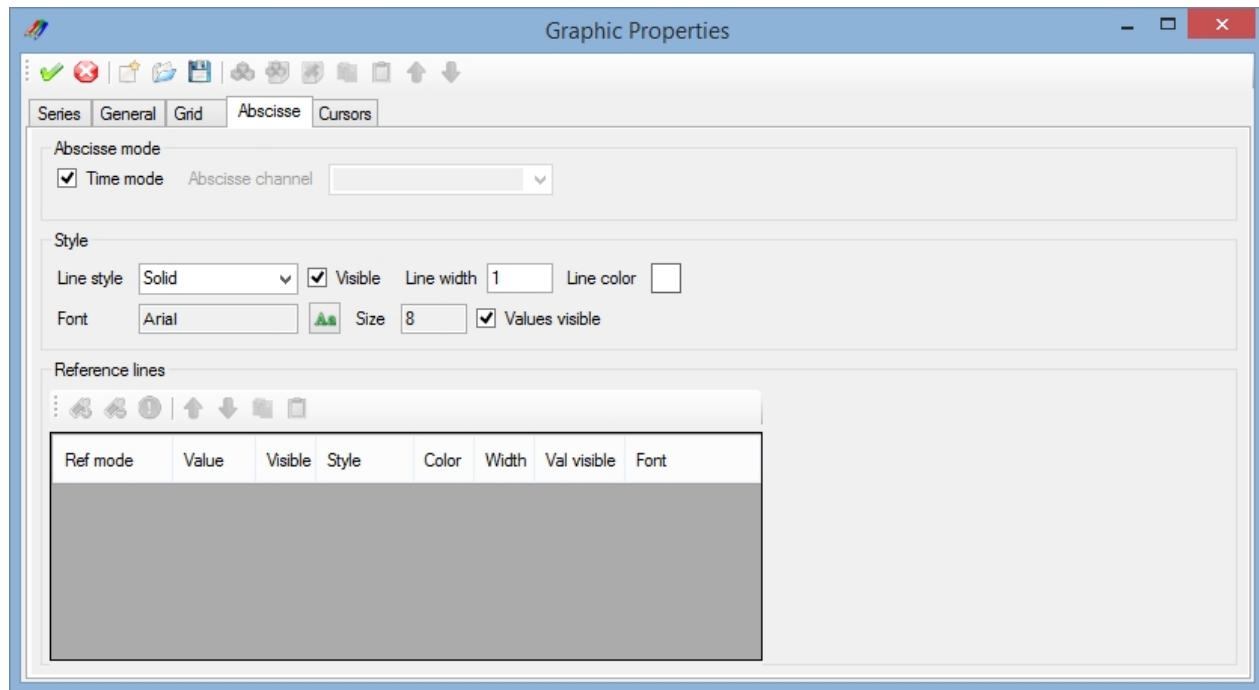


Each single grid (main/secondary, horizontal/vertical) can be individually set.

- Visible: Check the 'Visible' box to make the grid visible.
- Line width: Set the width of the grid line.
- Line style: Set the line style of the grid line. Check '[Line styles](#)' section for details.
- Line color: Double click the colored area to open the color selection dialog and select the grid line color.

Abscisse properties

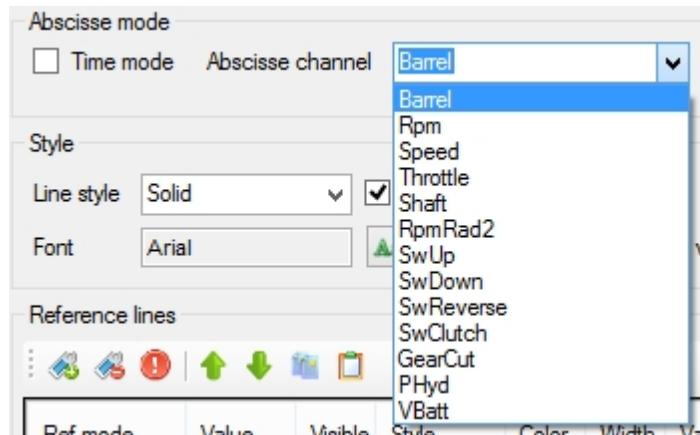
The 'Abscisse' tab contains graph window abscisse (X axis) properties



Abscisse mode

By default graphic uses the first channel of the data file (the time basically) as values for the Xaxis. It is however possible to choose another data channel as data source for the Xaxis.

Uncheck the 'Time mode' box and pick a channel up in the 'Abscisse channel' list to use it as Xaxis data source.

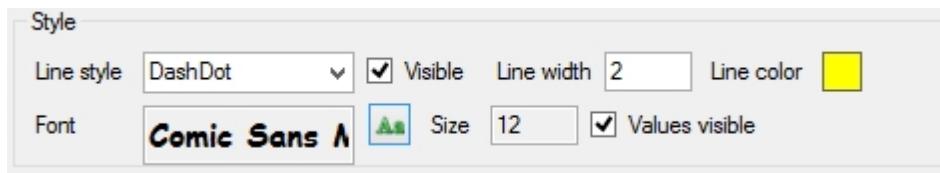


Style

This panels regroup Xaxis aspect properties

- Line style: Select the X axis line style. Check '[Line styles](#)' section for details.
- Visible: Check/uncheck this box to make the X axis visible or invisible.
- Line width: Set the width of the X axis line.
- Line color: Double click the colored square to open the color selection dialog and select the X axis line color.
- Font: Click the 'Font'  button to open the font selection dialog, select a font and set its size and attributes and click 'OK'
- Values visible: Check/uncheck this box to make the X axis values visible or invisible

Example of a customized X axis

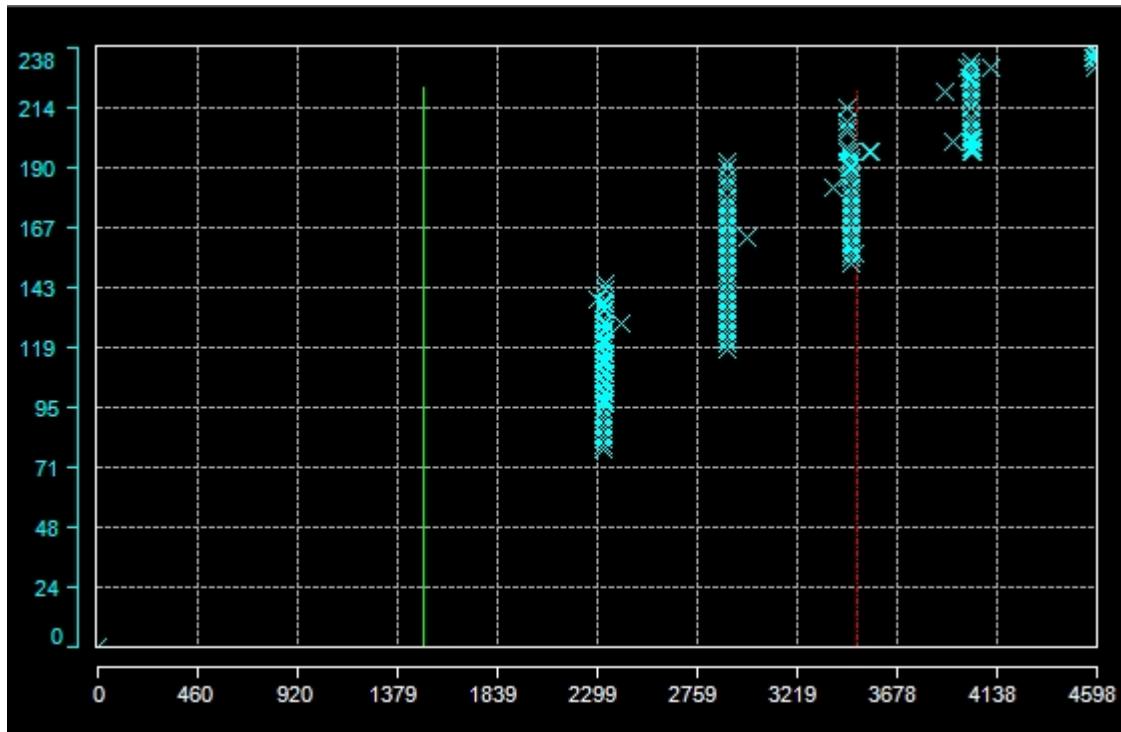


Reference lines

Abscisse reference lines are available only if the abscisse [time mode](#) is disabled.

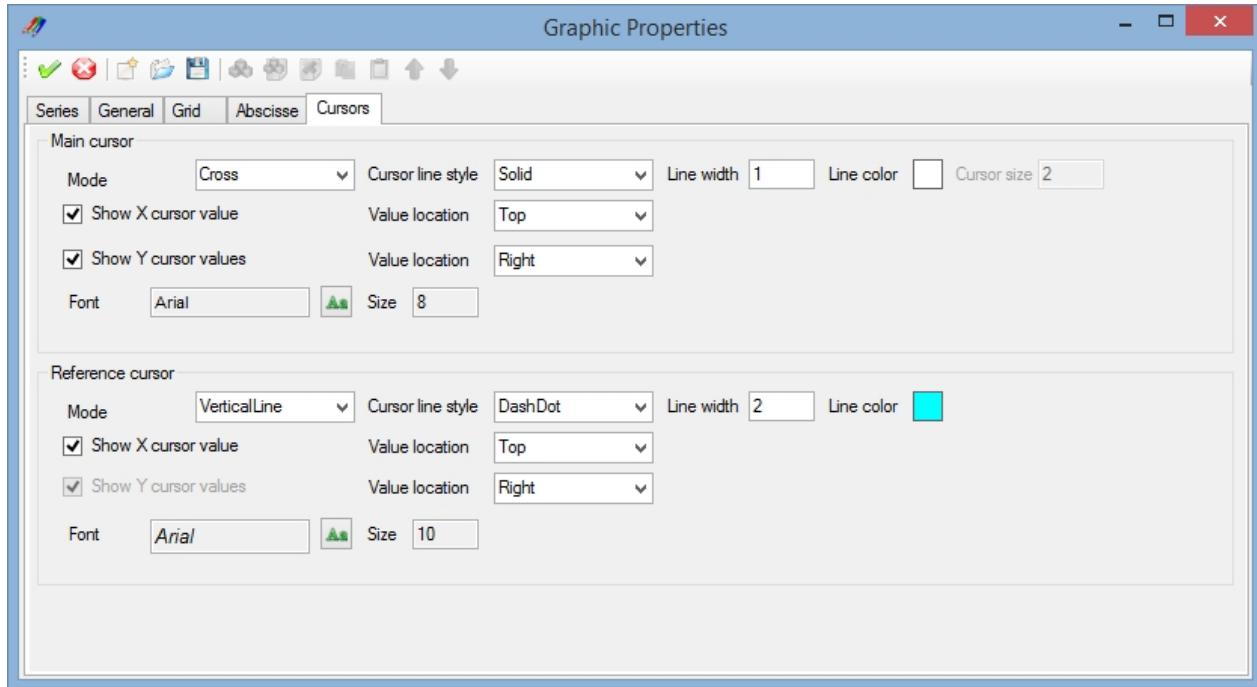
Manage X axis reference lines from this panel. Check '[Serie reference lines](#)' section for more details

Example of graphic using abscisse reference line



Cursors properties

The 'Abscisse' tab contains graph window abscisse (X axis) properties



Main cursor

In addition of main cursor properties that can be directly set in the graphic window, it is here possible to change the graphical aspect of the main cursor.

- Mode: Select in this list the [main cursor mode](#)
- Line style: Select the cursor line style. Check '[Line styles](#)' section for details.
- Line width: Set the width of the cursor line.
- Line color: Double click the colored square to open the color selection dialog and select the cursor line color.
- Show X value: Check/uncheck this box to make the [cursor X value](#) visible or invisible.
- Show Y values: Check/uncheck this box to make the [cursor Y values](#) visible or invisible.
- Xcursor value location: Set the location (left, right or center) of the [cursor X value](#).
- Y cursor values location: Set the location (top, bottom or center) of the [cursor X values](#).
- Font: Click the 'Font'  button to open the font selection dialog, select a font and set its size and attributes and click 'OK'

Reference cursor

In addition of reference cursor properties that can be directly set in the graphic window, it is here possible to change the graphical aspect of the reference cursor.

- Mode: Select in this list the [main cursor mode](#)
- Line style: Select the cursor line style. Check '[Line styles](#)' section for details.
- Line width: Set the width of the cursor line.

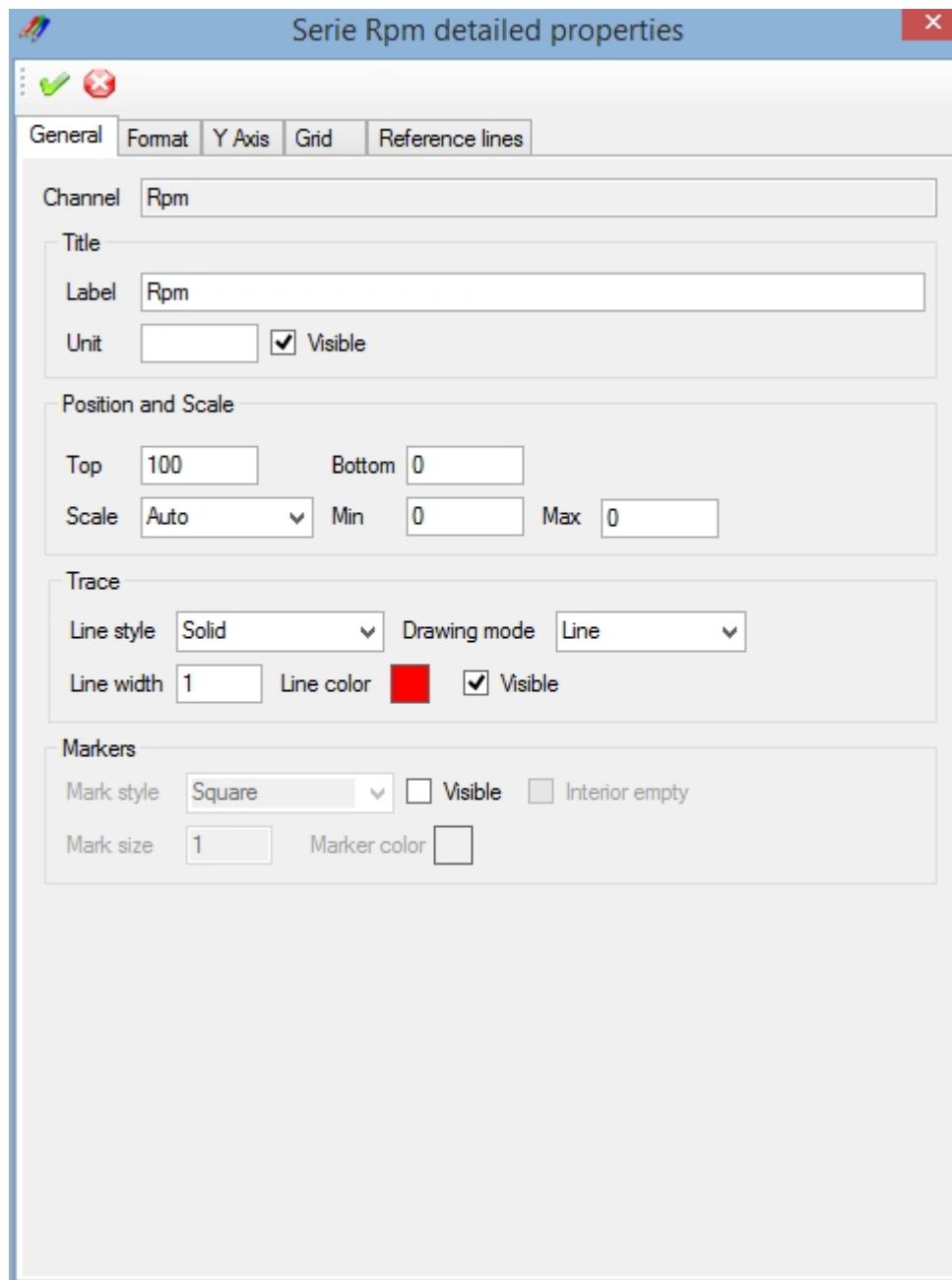
- Line color: Double click the colored square to open the color selection dialog and select the cursor line color.
- Cursor size: Only available for 'Graticule', 'Square' and 'Circle' [modes](#), set the size of the cursor.
- Show X value: Check/uncheck this box to make the [cursor X value](#) visible or invisible.
- Show Y values: Check/uncheck this box to make the [cursor Y values](#) visible or invisible.
- X cursor value location: Set the location (left, right or center) of the [cursor X value](#)
- Y cursor values location: Set the location (top, bottom or center) of the [cursor X values](#)
- Font: Click the 'Font'  button to open the font selection dialog, select a font and set its size and attributes and click 'OK'

Created with the Personal Edition of HelpNDoc: [Free EPub and documentation generator](#)

Graphic serie detailed configuration window

The graphic serie detailed configuration window permits to edit all properties related to a graphic serie.

To open this form, double click on a serie in the [legend](#), or open the '[Graphic window configuration](#)' form and click the 'Details' button of a serie in the '[Series](#)' configuration tab.



The graphic configuration window is composed by a tool bar and a multi-tabs window.

Each tab of the multi-tab is dedicated to a particular configuration section. There five different sections:

- General: [General properties](#) of the graphic serie
- Format: Serie value [format](#) properties

- Y Axis: Serie [Y axis](#) properties
- Grid : Serie [custom grid](#) properties
- Reference lines: Serie [reference lines](#) properties

Tool bar contains most common configuration commands.

 **Apply:** Apply graphic configuration changes

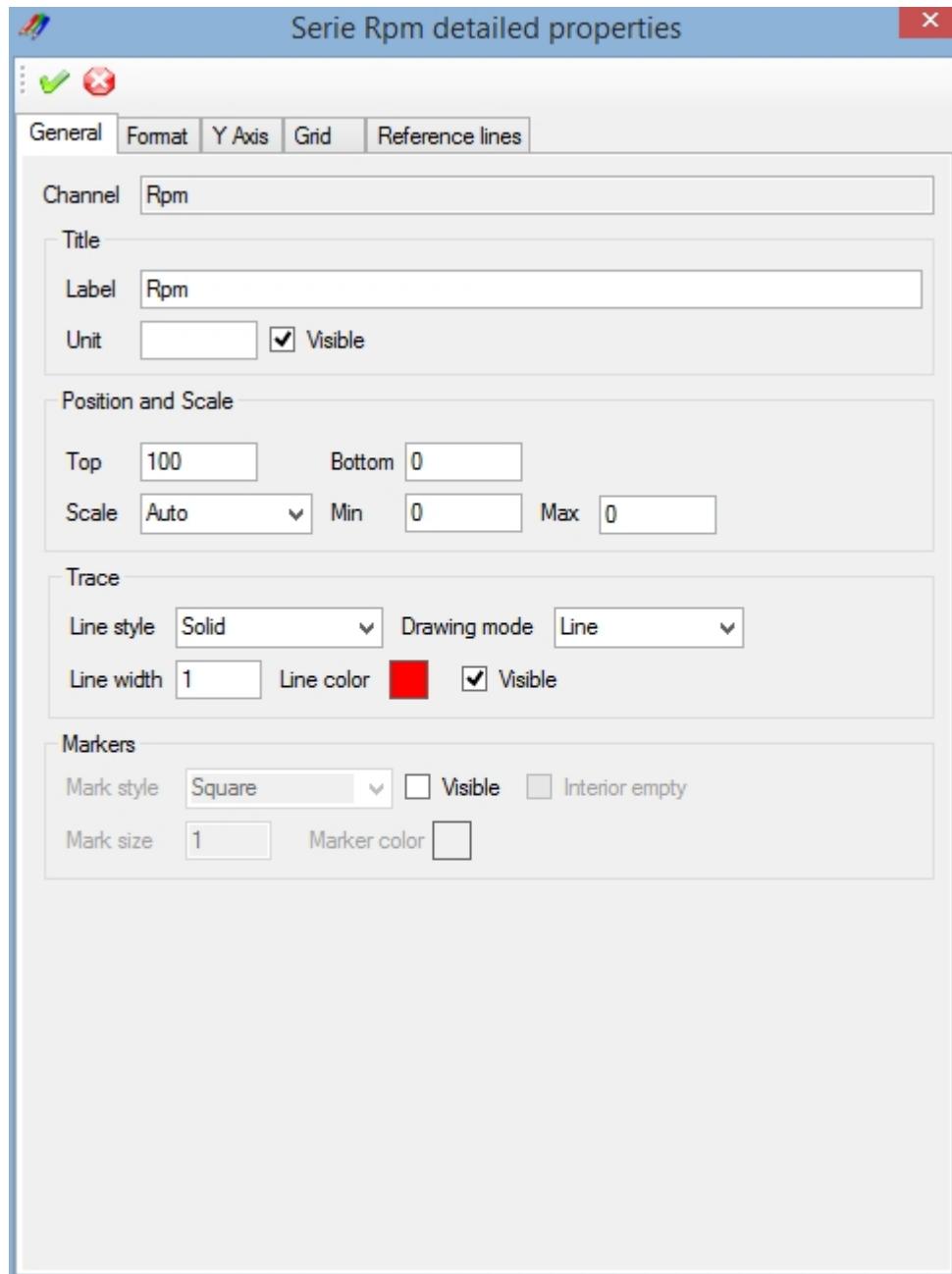
 **Cancel:** Cancel graphic configuration changes

Created with the Personal Edition of HelpNDoc: [Full-featured Kindle eBooks generator](#)

Serie general properties

As per the 'Generic series properties' panel of the 'Graphic window configuration' form, this regroups main serie properties, but not only.

Graphical trace aspect can be fully customized from this panel.



Channel

Content of this text box cannot be modified, it only shows the data channel source of the serie.

Title

This panel contains legend properties of the serie.

- Label: Title of the serie shown in the legend. Label can be different than the name of the data channel
- Unit: Serie value unit
- Visible: Master visibility switch, whatever 'Trace' and 'Marker' visibility states are if that box is not checked, serie will not be drawn in the graphic.

Position and scale

Set the serie position and scale properties

- Top: Serie's top position in the graphic area
- Bottom: Serie's bottom position in the graphic area
- Scale mode: Scaling mode of the serie, either 'Auto' or 'Custom'
- Min: Custom scale minimum value
- Max: Custom scale maximum value

It is important to note that 'Top' and 'Bottom' properties are applied only if the graphic layout is set to 'Custom'.

Regarding scaling mode, in 'Auto' mode, actual serie's minimum and maximum values are used to set the scale.

Trace

Set the serie trace aspect.

- Line style: Select the trace line style. Check '[Line styles](#)' section for details.
- Drawing mode: Serie's trace drawing mode, either 'Line' or 'Step'.
- Line width: Set the width of the trace line.
- Line color: Double click the colored square to open the color selection dialog and select the trace line color.
- Visible: Trace visibility flag. Check that box to make the trace visible

As far as the drawing mode, 'Line' means that two graphic samples will be joined by a simple line and 'Step' means that two sample will be joined with two lines in a shape of a step.

It is important to note, that assuming the the master visible switch of the title panel is set. If both 'Visible' checks of 'Trace' and 'Markers' are set, serie will be shown with trace and markers. If none of those are set, serie will not be drawn whatever value has the master visible switch.

Markers

Set the serie trace aspect.

Markers are special shapes placed on each graphical sample marking the real position of a sample in the graph.

There are 5 possible shapes for makers



Square

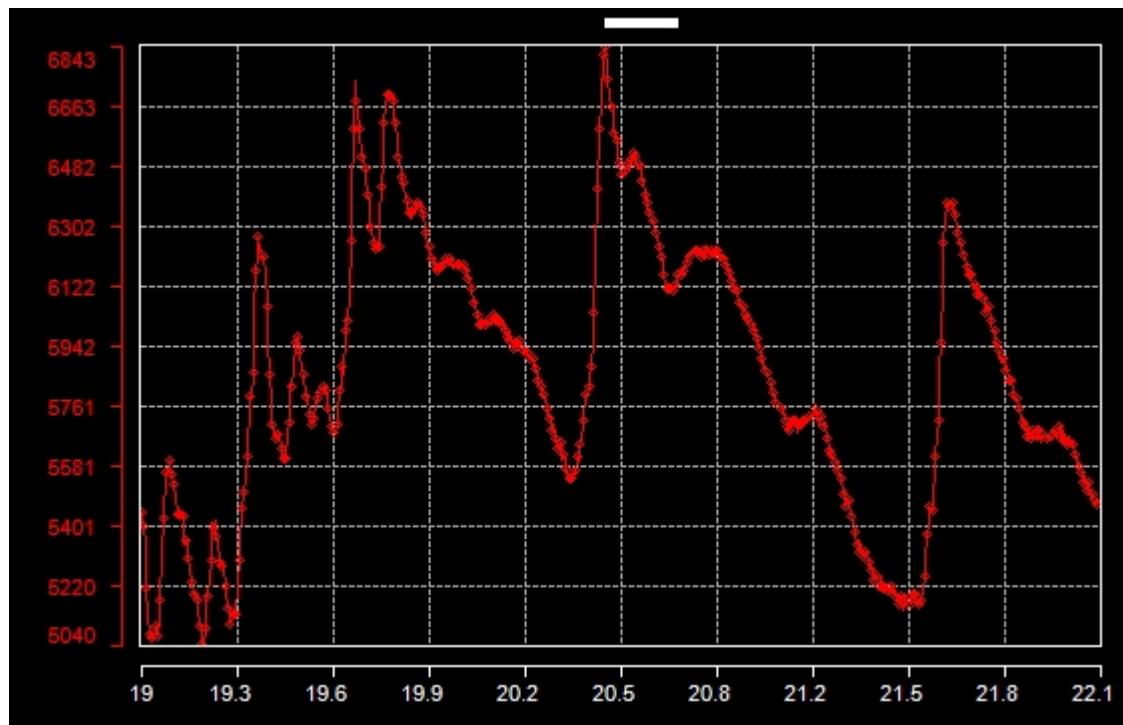


Markers have the following properties

- Mark style: Select the marker style. ('Square', 'Round', 'Diamond', 'Cross' or 'Triangle')
- Visible: Markers visibility flag. Check that box to make markers visible
- Interior empty: Set markers emptiness. If this box is not checked, markers will be filled with the marker color
- Mark size: Set the size of the marker
- Marker color: Double click the colored square to open the color selection dialog and select the mark color.

It is important to note, that assuming the the master visible switch of the title panel is set. If both 'Visible' checks of 'Trace' and 'Markers' are set, serie will be shown with trace and markers. If none of those are set, serie will not be drawn whatever value has the master visible switch.

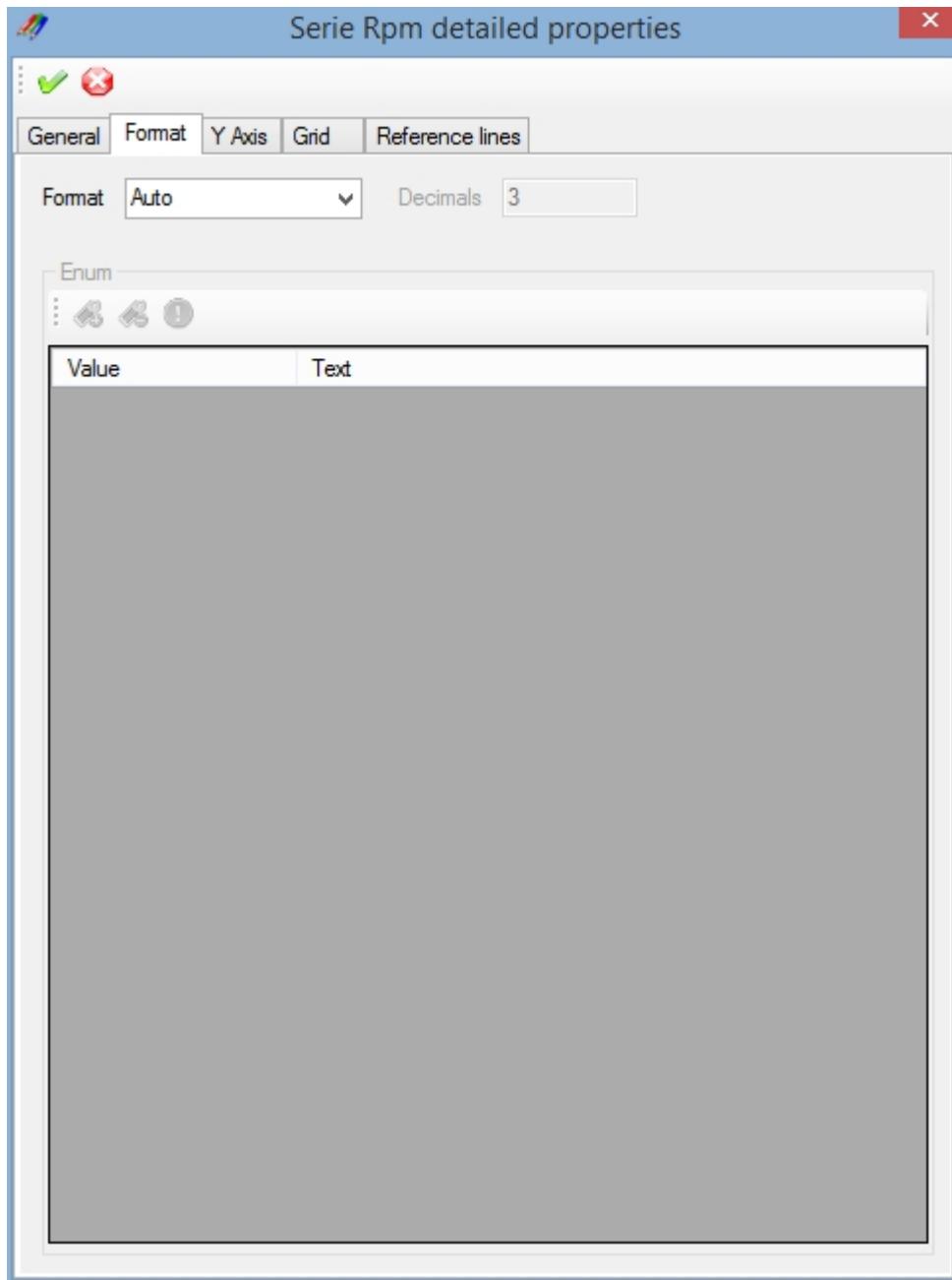
Example of graphic set with trace and markers



Created with the Personal Edition of HelpNDoc: [Create iPhone web-based documentation](#)

Serie format

The 'Format' tab contains serie value format properties



The serie's format describes how physical values of the serie are formated.

For example, if a the value of a serie is 21.47895526 in the data file, using the format property this value can be shown in the graphic as 21.5 or 21.48.

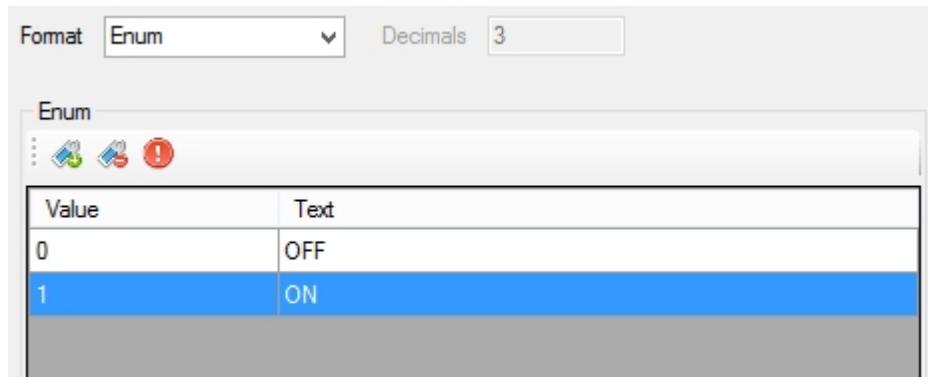
Five formats are available:

- Auto: Value is shown as a decimal value and number of decimal digits is a function of the span of plotted samples
- Decimal: Value is shown as a decimal value and number of decimal digits is set by the user

- Hexadecimal: Value is shown as an hexadecimal value, decimal part of the value being skipped.
- Binary: Value is shown as a binary value, decimal part of the value being skipped.
- Enum: A text is set for every possible value and this text is shown in the legend.

Enum format

To configure an enumeration for a serie, first select 'Enum' in the 'Format' list. Once 'Enum' selected, the enumeration configuration grid becomes active.



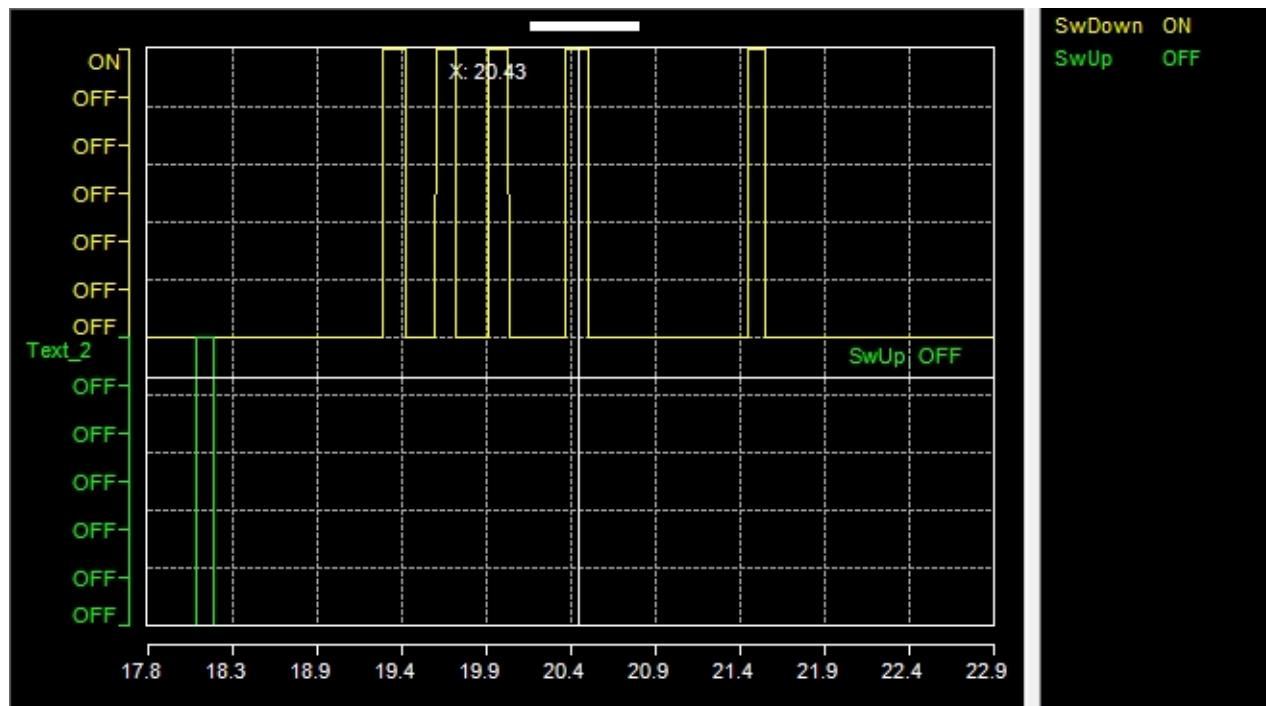
The 'Enum' panel is composed by a tool bar, containing enumeration control commands, and a grid containing every possible values of the serie and their corresponding text.

Click the 'Add enum' button to add a value into the enumeration. Set a value into the 'Value' cell and the corresponding text into the 'Text' cell.

Click the 'Delete enum' button to delete a value from the enumeration.

Click the 'Clear enums' button to remove all values of the enumeration.

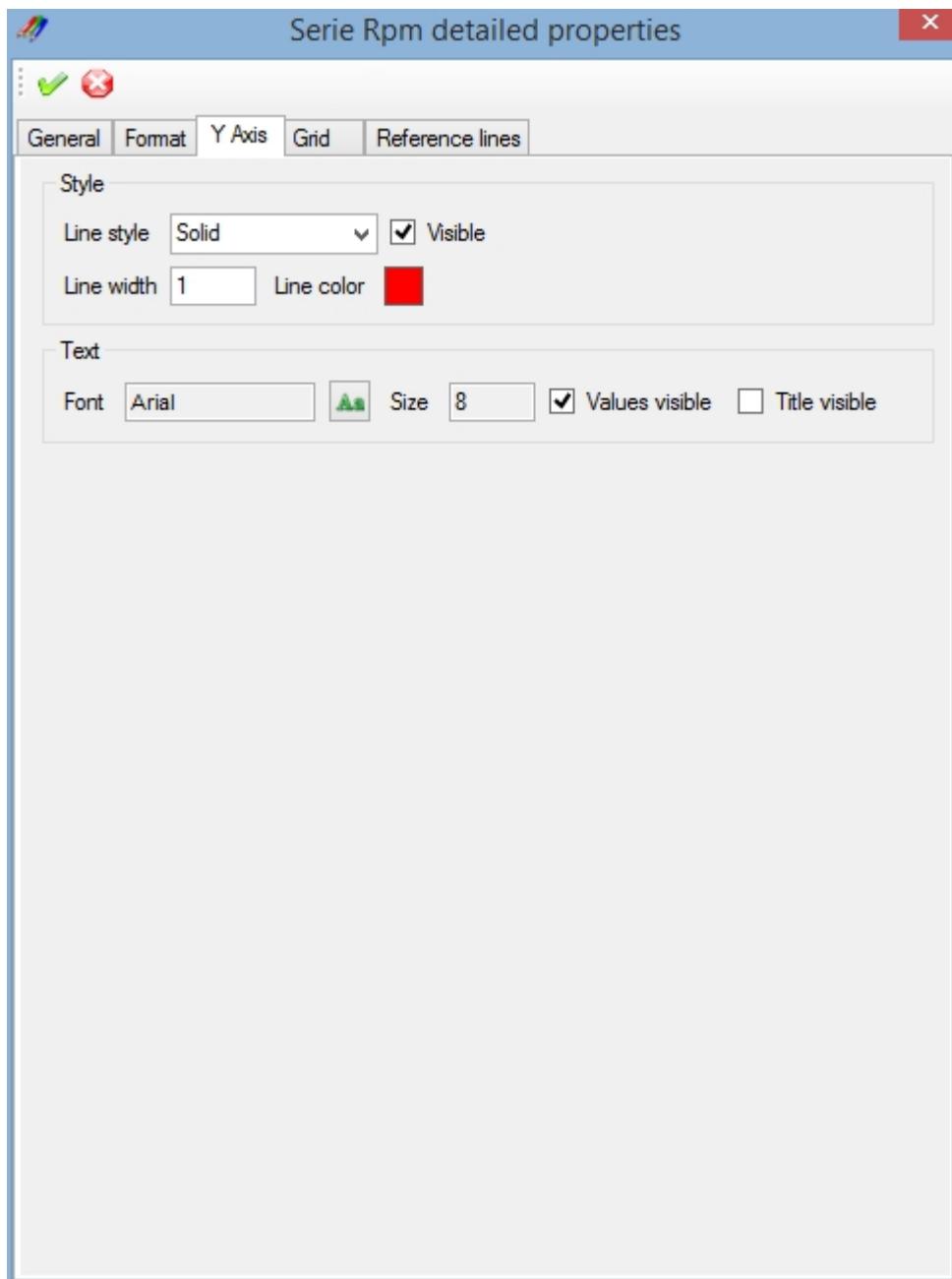
Example of graphic using enum format



Created with the Personal Edition of HelpNDoc: [Free PDF documentation generator](#)

Serie Y Axis

The 'Y Axis' tab contains serie Y axis properties



Style

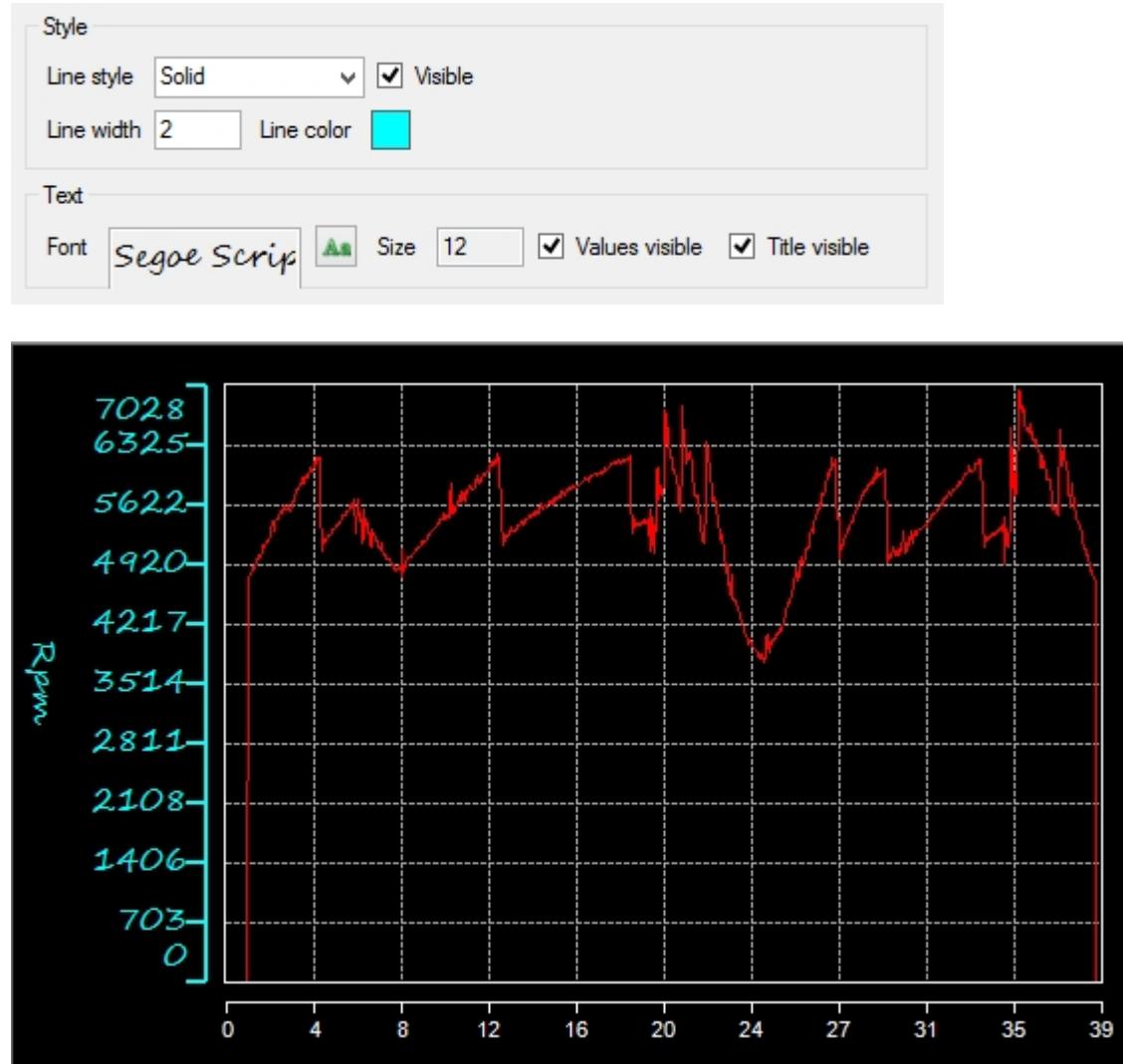
- Line style: Select the Y axis line style. Check '[Line styles](#)' section for details.
- Visible: Y axis visibility flag. Check that box to make the Y axis visible
- Line width: Set the width of the Y axis line.

- Line color: Double click the colored square to open the color selection dialog and select the Y axis line color.

Text

- Font: Click the 'Font'  button to open the font selection dialog, select a font and set its size and attributes and click 'OK'
- Value visible: Y axis values visibility flag. Check that box to make the Y axis values visible
- Title visible: Y axis title visibility flag. Check that box to make the Y axis title visible

Example of a graphic with a customized Y axis

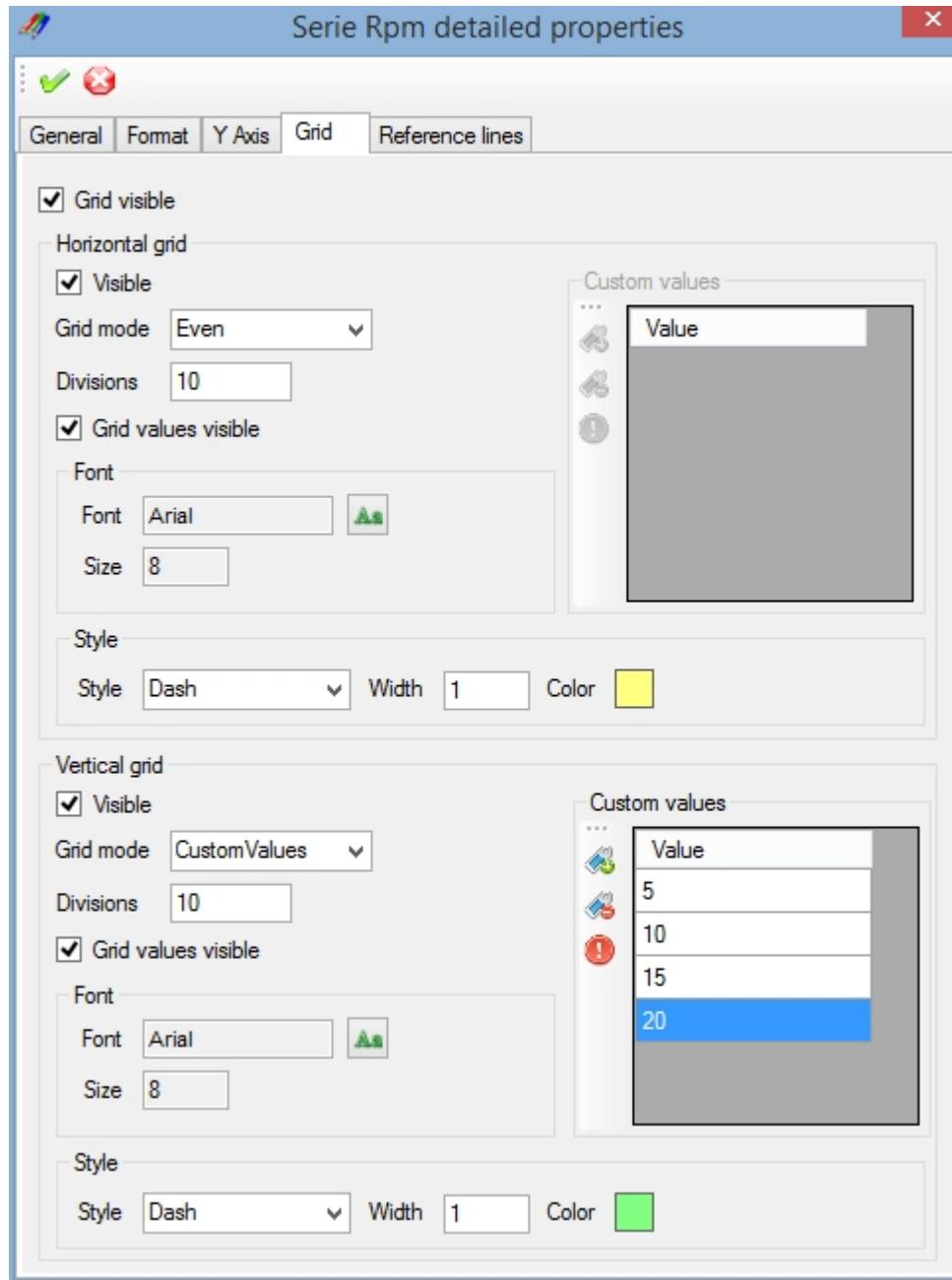


Created with the Personal Edition of HelpNDoc: [Full-featured Help generator](#)

Serie custom grid

In addition of standard [graphic grids](#), a graphic serie may have its own custom grid. Custom grid can be used as a complement of [serie reference lines](#).

The 'Grid' tab of the 'Graphic serie detailed properties' window permits to set up this custom grid.



The 'Grid' tab is split in two sections: 'Horizontal grid' and 'Vertical grid'. Both sections containing properties for horizontal grid lines and vertical grid lines.

Visible

The 'Grid visible' check box at the top of the tab Grid visible is the master grid visibility switch.

The serie's custom grid will be drawn only if this box is checked. This allows the user to decide whether or not the custom grid should be drawn by a simple command whatever the grid lines settings are.

Grid lines properties

Both 'Horizontal grid' and 'Vertical grid' panels contain the same properties for grid lines.

- Visible: Grid lines visibility flag. Check that box to make horizontal or vertical grid lines visible
- Grid values visible: Grid lines values visibility flag. Check that box to make horizontal or vertical grid lines values visible

Grid mode

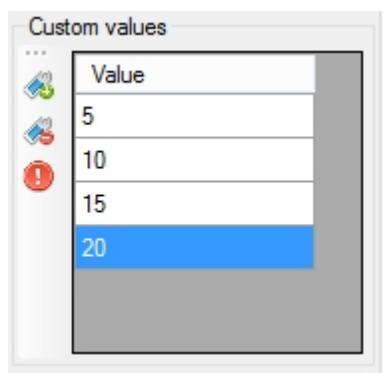
A serie custom grid may be drawn in different fashions:

- None: No grid lines (equivalent to uncheck the 'Grid visible' box)
- Even: Grid lines will be evenly set along graphic area of the serie
- MinMaxAvg: Three grid lines will be drawn at minimum, maximum and average values of the serie
- MinMaxZero: Three grid lines will be drawn at minimum, maximum and zero values of the serie
- CustomValues: Grid lines will be set at values defined by the user

For the 'Even' mode, number of grid divisions is defined by the 'Divisions' field. It is the actual number of divisions made by the grid, number of grid lines being the number of division +1.

Custom values

The 'Custom values' panel is used when grid mode is set to 'CustomValues'. This panel is composed by a grid containing the user values list and a tool bar regrouping list control commands.



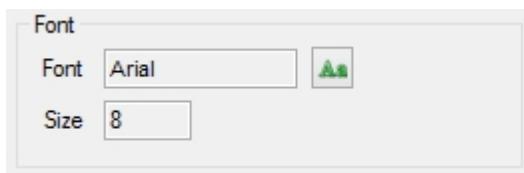
Click the 'Add value' button to add a value into the list. Set a value into the 'Value' cell.

Click the 'Delete value' button to delete a value from the enumeration.

Click the 'Clear values' button to remove all values of the enumeration.

Font

The 'Font' panel defines font properties for grid lines values if the box 'Grid values visible' has been checked.



Click the 'Font' button to open the font selection dialog, select a font and set its size and attributes and click 'OK'

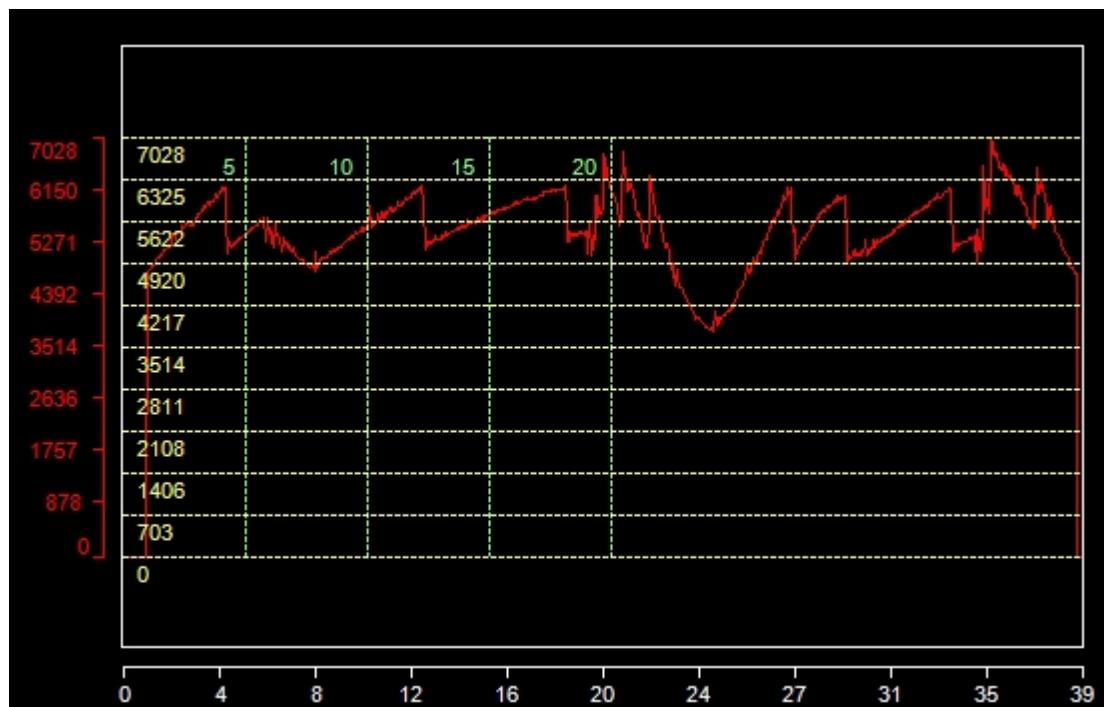
Style

The 'Style' panel defines graphical properties of grid lines.



- Style: Select grid lines style. Check ['Line styles'](#) section for details.
- Width: Set the width of grid lines.
- Color: Double click the colored square to open the color selection dialog and select grid lines color.

Example of a graphic using a serie custom grid:



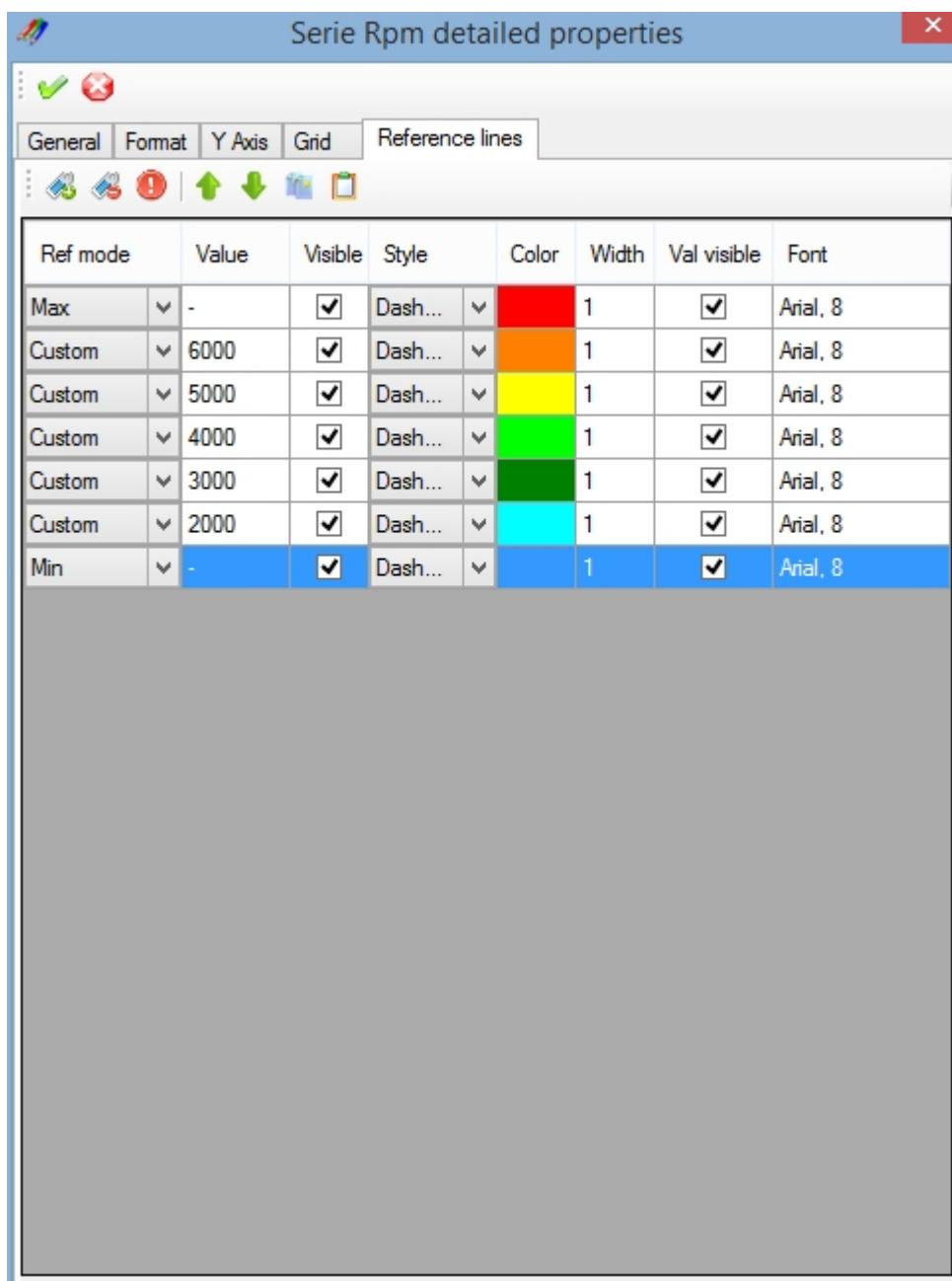
Created with the Personal Edition of HelpNDoc: [Free CHM Help documentation generator](#)

Serie reference lines

A reference line is a line drawn at a particular value of a graphic serie. A single serie can have multiple reference lines and reference lines can be used as a complement of the [serie custom grid](#).

Reference lines are usually horizontal except in the case of [abscisse](#) reference line where they are vertical.

The 'Reference lines' tab of the 'Graphic serie detailed properties' window permits to set up those reference lines.



The 'Reference lines' tab is composed by a tool bar, containing reference lines control commands, and by

grid showing properties of reference lines created by the user.

Tool bar

 **Add:** Create a new reference line

 **Delete:** Delete reference lines selected

 **Clear:** Delete all reference lines

 **Move up:** Move a reference line up in the list

 **Move down:** Move a reference line down in the list

 **Copy:** Copy a reference line

 **Past:** Past a reference line in the list

Reference lines grid

The reference lines grid has different columns for the different properties of a reference line.

Ref mode	Value	Visible	Style	Color	Width	Val visible	Font
----------	-------	---------	-------	-------	-------	-------------	------

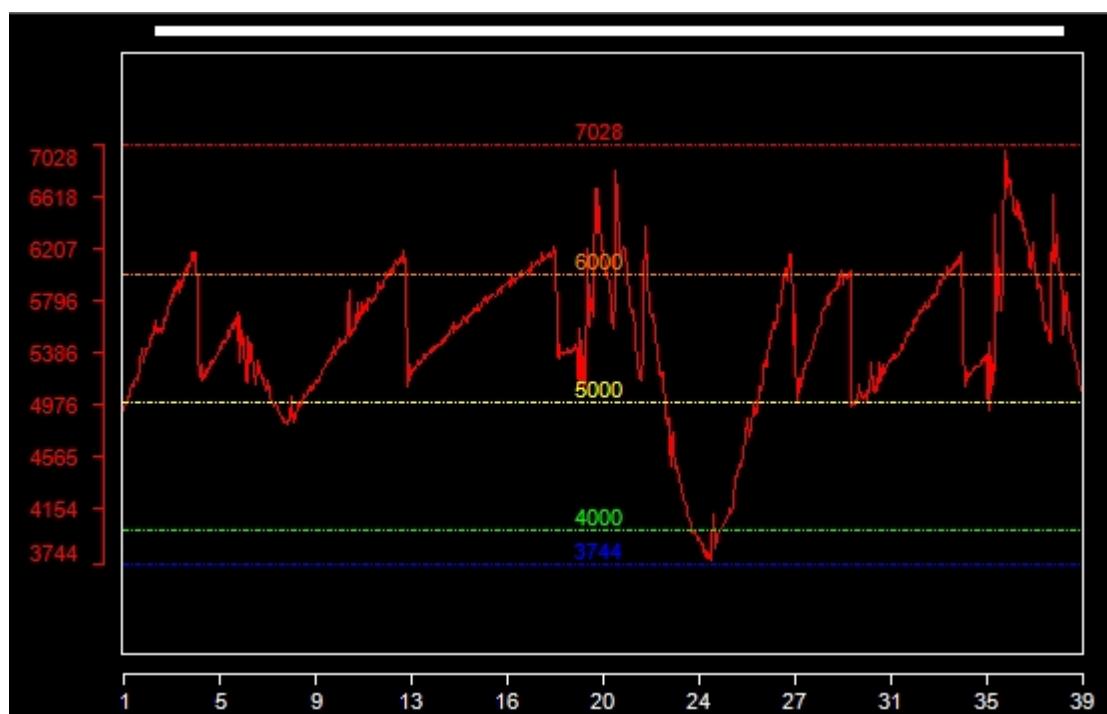
- Ref mode: Reference mode of the reference line
- Value: Value of the reference line in 'Custom' mode
- Visible: Reference line visibility flag. Check that box to make the reference line visible
- Style: Select reference line style. Check '[Line styles](#)' section for details.
- Color: Double click the colored cell to open the color selection dialog and select reference line color.
- Width: Set the width of the reference line.
- Val visible: Reference line value visibility flag. Check that box to make the reference line value visible
- Font: Double click the 'Font' cell to open the font selection dialog, select a font and set its size and attributes and click 'OK'

Reference mode

Value of a reference line may be defined by different methods.

- None: No reference line (equivalent to uncheck the 'Visible' box)
- Zero: Reference line is drawn at the zero value of the graphic serie
- Min: Reference line is drawn at the minimum value of the graphic serie
- Max: Reference line is drawn at the maximum value of the graphic serie
- Average: Reference line is drawn at the zero average of the graphic serie
- Custom: Reference line is drawn at the value defined by the user in the 'Value' cell

Example of a graphic using reference lines:



Created with the Personal Edition of HelpNDoc: [Easily create Web Help sites](#)

Appendix

Created with the Personal Edition of HelpNDoc: [Produce electronic books easily](#)

Line styles

Five line styles are available



Created with the Personal Edition of HelpNDoc: [Full-featured Help generator](#)

Virtual channels

Unlike a real CAN signal, a virtual channel is not contained within any CAN message, it is virtual! Value of such channel is therefore not static but dynamically calculated using a mathematical expression and values of one, two or more real CAN signals.

For example, let's consider CAN signals 'Signal_1', 'Signal_2' and a virtual channel 'Virtual' using 'Signal_1 + Signal_2' as expression.

On reception of signals 'Signal_1' and 'Signal_2', CANStream will evaluate expression of our virtual channel, then parameter 'Virtual' will take the sum of 'Signal_1' and 'Signal_2' as value.

Virtual channel expressions being evaluated each time a new value is received for 'Signal_1' or 'Signal_2', value of our virtual channel 'Virtual' is, at any time, equal to the sum of 'Signal_1' and 'Signal_2'.

Virtual channels calculation can be made using any CANStream object: CAN signal, built-in signal, logging data channel or another virtual channel as well.

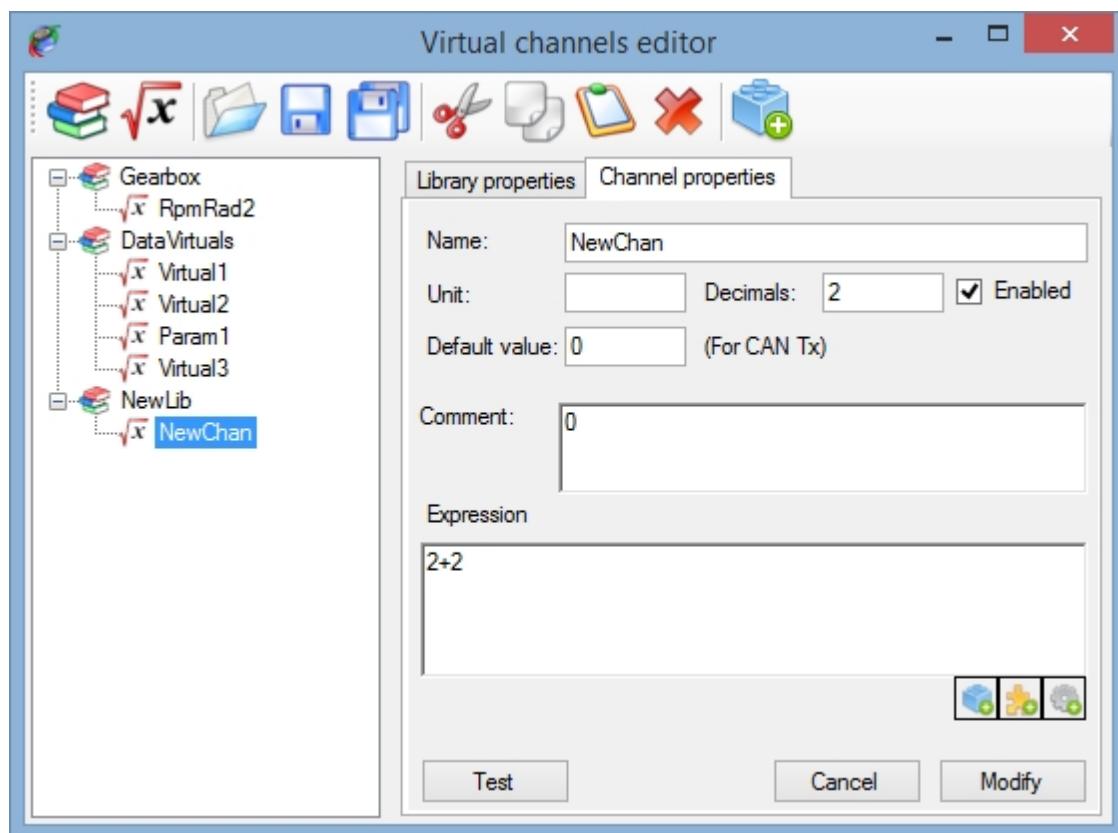
Virtual channels can also become real since they can be sent in CAN message as a 'Virtual CAN parameter'.

Virtual channel edition

Virtual channels are stored in library. Several virtual channels libraries can be loaded at the same time. If a virtual channels library is loaded once, CAN stream will reload it at each start up so you don't have to reload all your libraries all the time.

Simply remove a library from the active libraries list to not load it at start up.

Click the 'Tools\Virtual channels'  command of the menu strip to open the virtual channel editor.



The virtual channels editor is composed by:

- A tool bar, on the top, containing libraries and virtual channels control commands
- A tree view, on the left, showing all active libraries and virtual channels part of those libraries
- A multi-tab window, on the right, showing properties of either a library or a virtual channel

Tool bar



New library: Create a new virtual channels library



New virtual channel: Create a new virtual channel



Open library: Load an exiting virtual channels library



Save library: Save the active virtual channels library



Save all libraries: Save all active virtual channels libraries



Cut: Cut the selected item (library or channel)



Copy: Copy the selected item (library or channel)



Paste: Paste a cut/copied item (library or channel)



Delete: Delete selected item (library or channel)



Load object file: Load a virtual channel objects file

Tree view

The tree view shows all active libraries, each library is represented by a node

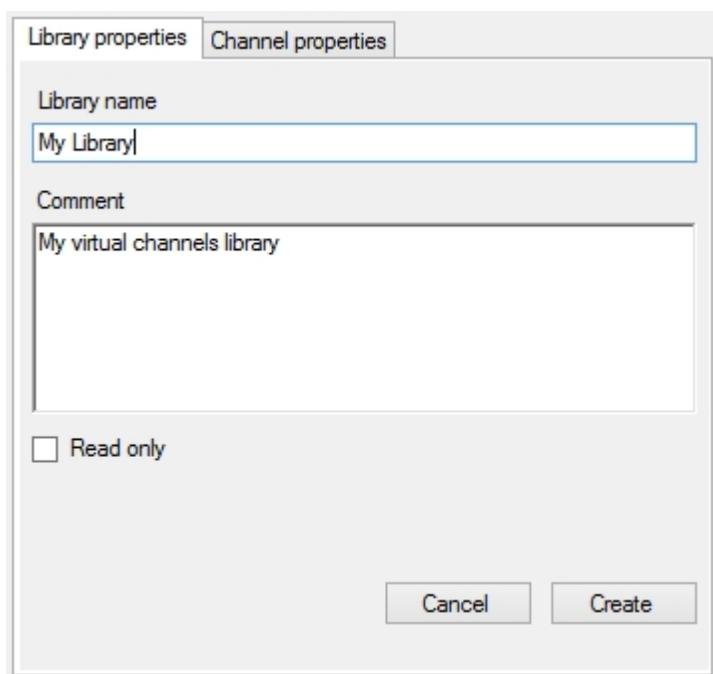
Virtual channels of a library are child nodes

Simply click on a library or a virtual channel to edit its properties.

Created with the Personal Edition of HelpNDoc: [Free PDF documentation generator](#)

Virtual channel library properties

Click on the 'New library'  button of the tool bar to create a new virtual channels library or click on a library node  of the tree view to edit its properties.



A library has actually very few properties:

- Library name: Name of the library
- Comment: Description of the library
- Read only: Check this box if you want protect your library against unwanted change

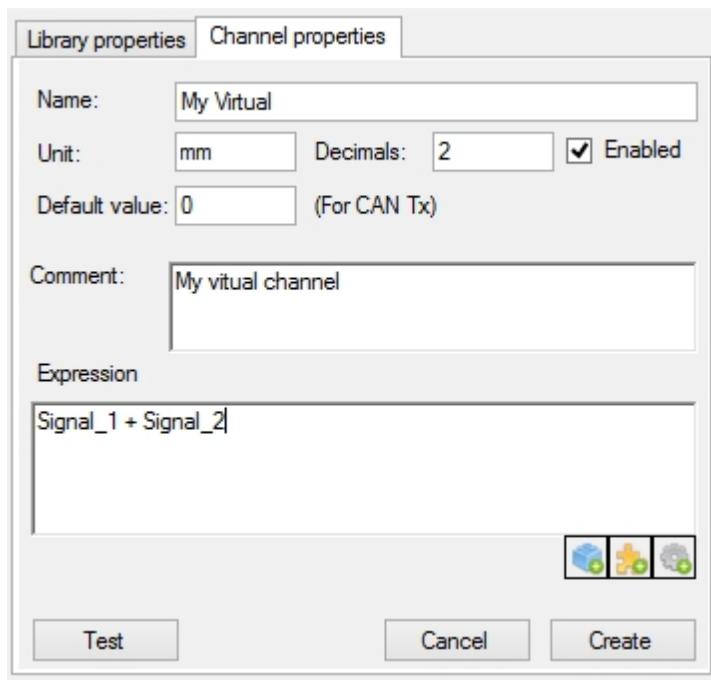
Once all properties set, just click the 'Create' button to add the library into the active libraries list.

In case of library edition, 'Create' button will become 'Modify', just click the 'Modify' button to apply changes.

Button 'Cancel' cancel the library creation or edition.

Virtual channel properties

Click on the 'New virtual channel'  button of the tool bar to create a new virtual channels or click on a virtual channel node  of the tree view to edit its properties.



A virtual channel has the following properties:

- Name: Name of the virtual channel
- Unit: Unit of the virtual channel value
- Decimal: Number of decimal
- Default value: Virtual channel default value (used only if the virtual channel is used as source of a virtual CAN parameter)
- Comment: Description of the virtual channel
- Expression: Mathematical expression of the virtual channel

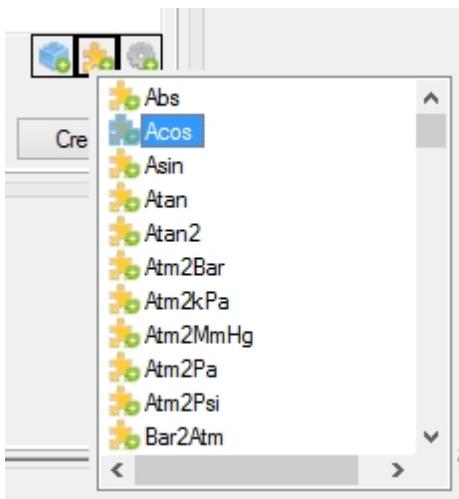
There are three buttons at the bottom of the mathematical expression text box

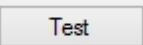


Those buttons permit to ease and accelerate edition of the virtual channel expression. Click on one of those buttons to open a list containing different items that can be used in the virtual channel expression. Double click an item of list to insert it into the expression.

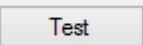
It is possible to insert three kinds of item

-  Virtual channel objects: List of variable (CAN signal, built-in signal, logging channel or another virtual channel)
-  Virtual channel functions: List of built-in virtual channel function (abs, cos, sin, ...)
-  Virtual channel operators: List of operators (+, -, *, /)



Regarding virtual channel objects, click the 'Load an object file'  button of the tool bar to load an object file and retrieve all names that can be used in a virtual channel expression.

While writing the virtual channel expression, it is possible to test it by simulating a computation. To do so,

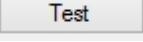
click on 'Test'  button.

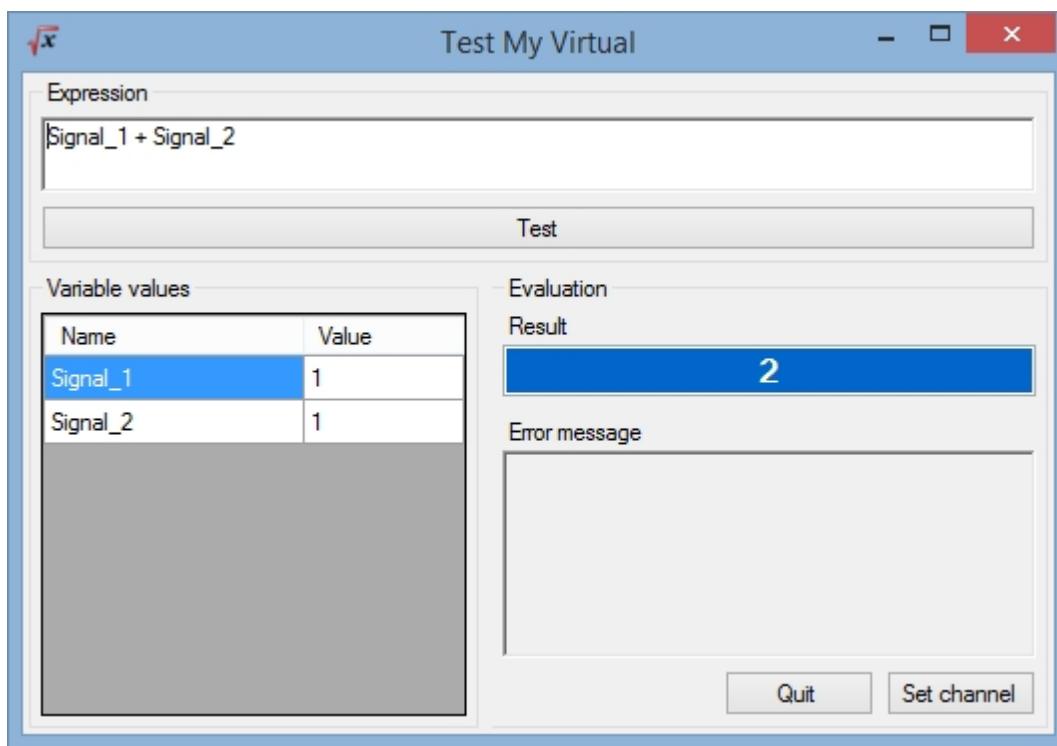
Check the Virtual channel testing section for more details.

Created with the Personal Edition of HelpNDoc: [Create HTML Help, DOC, PDF and print manuals from 1 single source](#)

Virtual channel testing

While writing the virtual channel expression, it is possible to test it by simulating a computation. To do so,

click on 'Test'  button of the virtual channel properties panel.

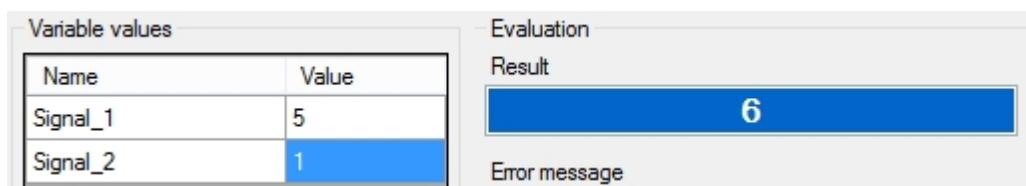


On the form opening, the virtual channel expression is evaluated using default values for expression variables.

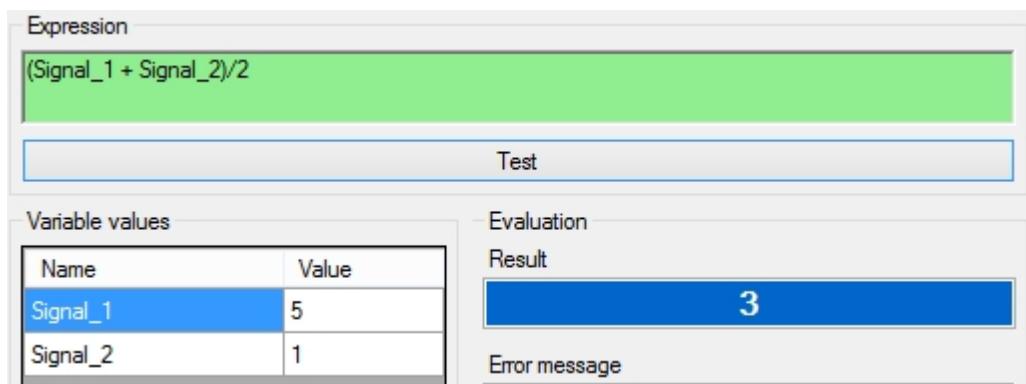
The evaluation result is shown the 'Result' text box of the 'Evaluation' panel.

The 'Variable values' grid, on the left, contains all variables of the virtual channel to test.

Change values in the grid to make a new calculation and verify the expression with different values.

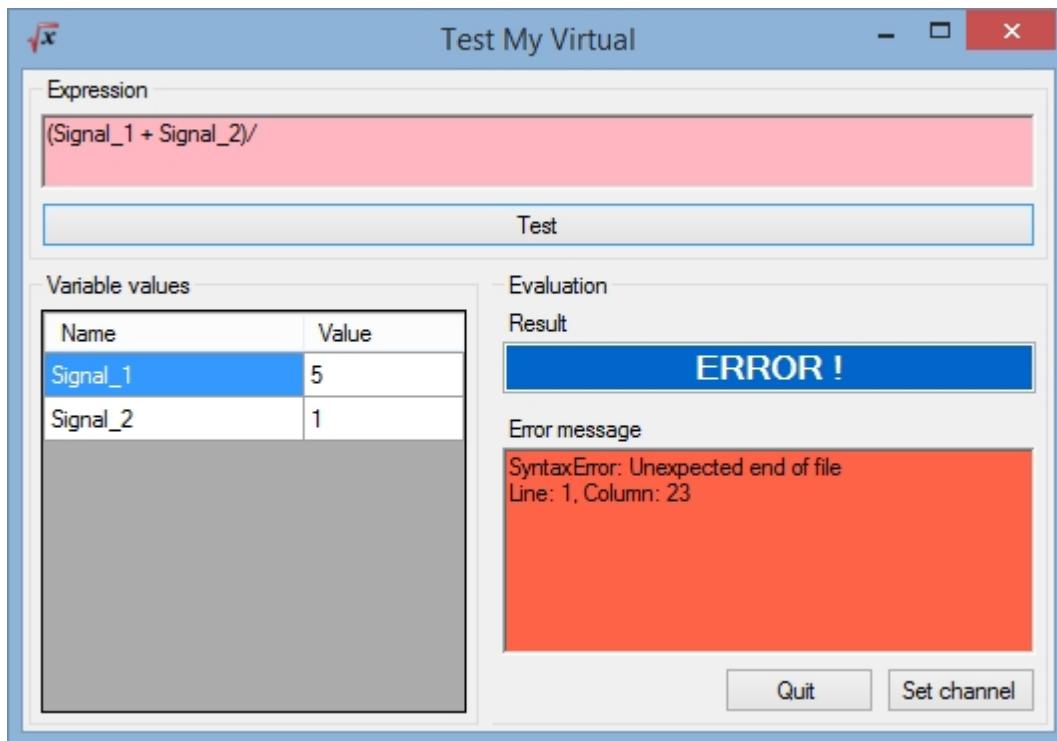


The mathematical expression itself can be modified. Simply change the expression written in the 'Expression' text box and click the 'Test' button to evaluate the modified expression.



If the modified expression is mathematically correct, 'Expression' text box goes green. If an error occurs during the evaluation, text box goes red.

In order to easily identify the error, the 'Error message' text box indicated what is the error and where it is.



Once the expression is working, click on the 'Set channel' button to set the test expression in the 'Expression' text box of the virtual channel properties panel.

Created with the Personal Edition of HelpNDoc: [Free EBook and documentation generator](#)

Virtual channel built-in functions

Here is the list of CANStream built-in function for virtual channel.

Created with the Personal Edition of HelpNDoc: [Easily create iPhone documentation](#)
