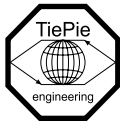


Multi Channel software

User manual



TiePie engineering

Copyright ©2024 TiePie engineering.
All rights reserved.

Revision 1.11, June 2024

Despite the care taken for the compilation of this user manual, TiePie engineering can not be held responsible for any damage resulting from errors that may appear in this manual.

Contents

1	Introduction	1
1.1	About the software	1
1.2	How to use the software	1
1.2.1	Controlling the software	2
1.2.2	Edit fields for numerical values	2
1.3	Software installation	3
1.3.1	Check for updates	3
1.3.2	Submit usage statistics	3
1.4	Hardware installation	3
2	Software basics	5
2.1	Main window parts	5
2.2	User presets	7
2.2.1	Creating a User preset	7
2.2.2	Loading a user preset	7
2.2.3	Clearing a user preset	8
2.2.4	Naming a user preset	8
2.3	User preset Worksets	8
2.3.1	Add a workset	9
2.3.2	Select a workset	9
2.3.3	Remove worksets	9
2.4	Quick Setups	10
2.4.1	Quick Setup ordering	10
2.4.2	Selection	10
2.4.3	Only show compatible Quick Setups	11
2.5	Basic (quick) functions	11
2.5.1	New graph	12
2.5.2	Oscilloscope in Yt mode	12
2.5.3	Oscilloscope in XY mode	13
2.5.4	Spectrum analyzer	13
2.5.5	Data logger	13
2.5.6	Voltmeter	14

2.5.7	CAN decoder	15
2.5.8	I ² C decoder	15
2.5.9	UART / Serial decoder	15
2.6	Printing	15
2.7	Settings	16
3	Objects	17
3.1	Object tree	17
3.2	Creating objects	18
3.3	Configuring objects	18
3.4	Cloning objects	19
3.5	Connecting objects	19
3.6	Disconnecting objects	20
3.7	Inserting objects between other objects	21
3.8	Deleting objects	21
3.9	Saving objects	21
3.10	Exporting object data	22
3.11	Using aliases	22
3.12	Example	22
4	Instruments	23
4.1	Selecting and opening instruments	23
4.1.1	Combining instruments	24
4.1.2	Network instruments	25
4.1.3	Demo instruments	27
4.2	Controlling instruments	27
4.2.1	Using the oscilloscope toolbar	28
4.2.2	Using the channel toolbar	29
4.2.3	Using popup menus and settings windows	29
4.3	Settings	30
4.3.1	Instrument settings	30
4.3.2	Trigger settings	34
4.3.3	Channel settings	38
5	Arbitrary Waveform Generator	41
5.1	Toolbars	41
5.2	Power	42

5.3	Output invert	42
5.4	Generator mode	43
5.5	Signal properties	44
5.5.1	Frequency mode	44
5.5.2	Sweep	44
5.6	Setfiles	45
5.7	Arbitrary data	45
5.7.1	Loading arbitrary data from an open source	46
5.7.2	Loading arbitrary data from a file	46
5.7.3	Data resampling	47
5.8	Hotkeys	47
5.9	Oscilloscope triggering	47
6	Sources	49
6.1	Software generator	49
7	I/Os	51
7.1	Math I/Os	51
7.1.1	Gain / Offset	51
7.1.2	Add/Subtract	52
7.1.3	Multiply/Divide	52
7.1.4	Sqrt	52
7.1.5	ABS	52
7.1.6	Differentiate	52
7.1.7	Integrate	53
7.1.8	Logarithm	53
7.2	Filter I/Os	53
7.2.1	Filter	53
7.2.2	Ideal Filter	54
7.2.3	Signal Cleaner	54
7.2.4	Average	54
7.3	Adjusting I/Os	54
7.3.1	Deskew	54
7.3.2	Slice	55
7.3.3	Limiter	55
7.3.4	Comparator	55
7.3.5	Resampler	55

7.3.6	Window	55
7.4	Miscellaneous I/Os	56
7.4.1	Min / Max detector	56
7.4.2	Data collector	56
7.4.3	Reference	56
7.5	Analyzing I/Os	56
7.5.1	Duty cycle	56
7.5.2	RMS	57
7.5.3	Maximum	57
7.5.4	Minimum	57
7.5.5	Maximum - Minimum	58
7.5.6	Phase difference	58
7.5.7	FFT	58
7.5.8	EMI	58
7.6	Automotive I/Os	59
7.6.1	Crankshaft angle	59
7.6.2	RPM	59
7.6.3	EV charger control pilot analyzer	59
7.7	Decoder I/Os	60
7.7.1	Pulse decoder	60
7.7.2	I ² C decoder	60
7.7.3	UART / Serial decoder	60
7.7.4	CAN decoder	60
7.7.5	SAE J1939 decoder	61
7.7.6	SPI decoder	61
7.7.7	Base section to index	61
8	Sinks	63
8.1	Meter	63
8.2	Table	63
8.3	Disk writer	64
8.4	Auto disk	64
8.5	Sound	64
8.6	Compression test	64
9	Displaying data in graphs	65
9.1	Creating new graphs	65

9.2	Active graph	66
9.3	Axes	66
9.4	Vertical axes	66
9.4.1	Tabbing axes	67
9.4.2	Adjusting displayed signals using an axis	68
9.4.3	Show or hide a line	68
9.4.4	Automatically arranging axes	68
9.4.5	Merging axes	69
9.4.6	Axis type	70
9.4.7	Axis range	70
9.4.8	Visible range	71
9.4.9	Clipping detection	71
9.4.10	Show peaks and harmonics	72
9.5	Horizontal Axis	73
9.5.1	Axis range	74
9.5.2	Visible range	75
9.5.3	Follow source	75
9.5.4	Axis type	75
9.5.5	Time axis label style	76
9.6	Graph toolbar	76
9.7	Zooming	77
9.7.1	Zooming using keyboard controls	77
9.8	Automatic measurements using cursors	78
9.8.1	Measurements	79
9.8.2	Phase cursors	80
9.9	References	82
9.10	Using comments	83
9.11	Drawing options	84
9.11.1	Interpolation	84
9.11.2	Visual noise reduction	84
9.11.3	Legend	84
9.11.4	Markers	84
10	Displaying data in Meters	85
10.1	Clipping detection	87
10.2	Display properties and actions	87

10.2.1	Color	87
10.2.2	Digits	87
10.2.3	Decimals	88
10.2.4	Auto Prefix	88
10.2.5	Range	88
10.2.6	Measurement	88
10.2.7	Type	89
10.2.8	Delete	89
10.3	Column actions	89
10.3.1	Add Measurement	89
10.3.2	Disconnect source	89
10.4	Meter properties and actions	89
10.4.1	Single column	90
10.4.2	Measurement data range	90
10.4.3	Log to disk	90
10.4.4	Stop disk logging	91
10.4.5	Copy to clipboard	91
10.4.6	Always on top	91
11	Saving and logging measurements	93
11.1	Saving a single scope/spectrum analyzer measurement	93
11.1.1	Saving to a TPS file	93
11.1.2	Saving objects to a TPO file	93
11.2	Saving multiple scope/spectrum analyzer measurements	95
11.3	Save continuous data logger measurements	96
11.4	Exporting single measurements	98
11.5	Exporting multiple measurements	99
11.6	Saving a measurement to an image	101
11.6.1	Saving an image from the command line	101
11.6.2	Supported image file formats	102
11.6.3	Image dimensions	102
11.6.4	Image scheme	102
12	Loading saved measurements	103
12.1	Loading a whole file	103
12.2	Loading just a part of a file	103
12.3	Loading into selected objects	104

A	Number input prefixes, constants and functions	105
A.1	Prefixes	105
A.2	Constants	105
A.3	Functions	106
B	Standard measurements	107
B.1	Short description of the measurements	107
B.2	Reference resistor	113
C	Hotkeys	115
D	Files types	117
D.1	Multi Channel TPS files	117
D.2	Multi Channel TPO files	117
D.3	Multi Channel TPIDX files	118
D.4	WinSoft files	118
D.5	Exporting to other file types	118
E	Command line parameters	119
E.1	Batch converting multiple saved measurements	120
	Index	121

The Multi Channel software package is the measuring software for the TiePie engineering measuring instruments and function generators. This document explains the basic functionality in the application. It is intended to get you started and to teach you how to do basic and more advanced measurements.

A basic knowledge of controlling windows based applications and computer based measuring instruments is assumed. For a detailed and up to date description of functionality and objects that are not described in this document, refer to the help file that comes with the Multi Channel software or to the library section of the TiePie engineering website www.tiepie.com.

1.1 About the software

The Multi Channel software can control an unlimited amount of measuring *instruments*, each with an unlimited amount of *channels*. The Multi Channel software can display this data in an unlimited amount of different displays. Measured data can be displayed directly, but it can also be processed first.

In addition to measured data from the instrument input channels, the software can also work with other data *sources*. For example data from internal software signal generators can be combined with the measured data.

Besides for measuring and displaying data, the Multi Channel software can be used to control *Arbitrary Waveform Generators* (AWG). An AWG can be used to generate standard signals, like sine, block and triangle or for generating arbitrary signals, for example signals that have previously been measured. Refer to chapter 5 for information about the AWG.

1.2 How to use the software

The Multi Channel software is suitable both for experienced and inexperienced users. For performing most common measurements in an easy way, you can use the **Quick Setups**, which are explained in section 2.4. The Quick Setups automatically setup the hardware, create graphs and other objects that are needed to perform the measurement.

To easily perform more complex measurements, you can load predefined setfiles and start measuring right away.

To get more control and flexibility, you can create objects yourself and connect them to each other the way you want. You can find information about using objects in chapter 3 and the online help. When you have created your test setup, you can store it in a setfile for later use. See chapter D for more information about using files.

1.2.1 Controlling the software

There are many different ways to control the Multi Channel software. A few of them are treated in this section. This will be enough to get you started. Not all of the functionality is mentioned in this manual. Once you are used to how the software is controlled, you will discover it yourself.

Hotkeys The instruments and graphs in the Multi Channel software can be controlled with *hotkeys*. You can find a complete list of the hotkeys in appendix C. Once you know the most important hotkeys by heart you will be able to change settings very quickly.

Popup menus Almost all settings and options in the Multi Channel software are available via popup menus. When you right-click an object, a popup menu will appear which contains actions that affect the object you clicked. The best way to find all the possibilities is to try.

Drag and drop Besides the popup menus, *drag and drop* is very important. You can drag different *objects* onto each other or onto graphs to make connections and you can drag graph *axes* and *trigger symbols*. You can find more information about connecting different objects in section 3.5. In section 9.3 you can read how to drag axes. Just like with the popup menus, the best way to find all the possibilities is to try. The mouse cursor will indicate where the object you are dragging can be dropped.

1.2.2 Edit fields for numerical values

In many locations in the software, it is possible to enter a numerical value, e.g. to set the gain value of a Gain / Offset I/O.

Entering numbers can be done in various ways:

as plain number	e.g. 1200
in scientific notation	e.g. 1.2e3
using prefixes	e.g. 1.2k or 1k2
as an expression	e.g. $10 * (100 + 20)$

Several prefixes can be used in edit fields. Prefixes are case sensitive!

Expressions can besides numbers also use parentheses (always in pairs), constants and mathematical functions. These are case insensitive.

An overview of all valid prefixes, constants and functions can be found in appendix A

Any expression that is entered, will be evaluated by the software and the resulting value is used. The expression will be kept, to make later changes easier. The expression is also saved when saving a setup to a .TPS file and reloaded when that file is later reloaded.

1.3 Software installation

If you have just recently purchased your instrument, you can download the installation program from www.tiepie.com/download to install the software.

The installation process is straightforward and is not explained in detail here. During the installation you will be prompted if you would like to associate the file extensions *.DAT* and *.SET* with the Multi Channel software. These files are used by the old WinSoft measurement software. If you associate the extensions with the Multi Channel software (default), you will be able to open WinSoft files by just double clicking them or dragging them on the main window of the Multi Channel software. If you don't have WinSoft files or have other programs that use these file extensions, you can uncheck the checkboxes.

1.3.1 Check for updates

The Multi Channel software can automatically check for updates when it is started. When a new software version is available, a message will be shown, including a link to a page showing the changes in the software, as well as a download link. You can then download and install the new version. This is not done automatically.

1.3.2 Submit usage statistics

In order to help us improve the Multi Channel software, the software can send usage statistics to TiePie engineering. The statistics give us insight in which features are used and how they are used. These usage statistics are anonymized and do not contain any personally identifiable information.

1.4 Hardware installation

Before the Multi Channel software can control your instrument(s), a driver needs to be installed. Please refer to the user manual of your instrument(s) for instructions for installing hardware and drivers.

The drivers for the TiePie engineering instruments are continuously improved. It is recommended to download the latest version of the driver for your instrument from www.tiepie.com/download.

This chapter will explain the basics of the Multi Channel software to get you started. It will show you the different parts of the main window and how to use them to perform basic measurements. A Handyscope HS5 is used in most of the examples, but of course other instruments supported by the Multi Channel software can be used just as well.

2.1 Main window parts

When you start the Multi Channel software it will look like figure 2.1. This section will explain the function of the different parts of the main window.

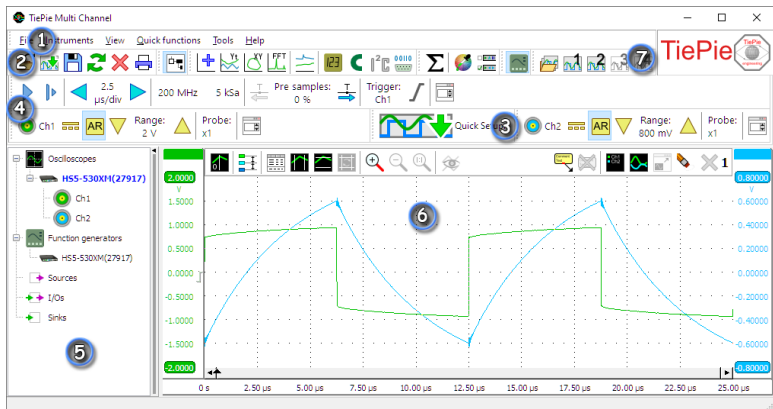


Figure 2.1: Main window

In the picture, the different parts are indicated with numbers:

- ① Main menu
- ② File toolbar
- ③ Quick Setups toolbar
- ④ Instrument toolbar
- ⑤ Object tree
- ⑥ Graph
- ⑦ User presets


2 File toolbar The file toolbar can be used for accessing frequently used items from the file menu, for example opening, saving or reloading files.

It also gives access to some quick functions that can be used to create a new scope, spectrum analyzer, data logger or multimeter, as well as some protocol analyzers. For more info on these quick functions see section 2.5.

Finally it holds buttons to create math channels, set the application color scheme and toolbar scheme and it gives access to the Arbitrary Waveform Generator window(s).

3 Quick Setups toolbar The Quick Setups toolbar gives access to the Quick Setups, a system to quickly setup the instrument and software for any measurement task. See section 2.4 for more information about the Quick Setups.

4 Instrument toolbar The instrument toolbar can be used to control basic instrument settings, like **Sample frequency**, **Record length** and trigger settings. For each channel, a channel toolbar is present to control channel settings like **Range**, **Auto ranging** and **Coupling**. Refer to chapter 4 for more information about controlling instruments.

5 Object tree The object tree is situated at the left side of the main window of the application. In figure 2.1 it is opened, but it can also be closed to maximize the available space for graphs. It can be revealed and hidden by clicking the  **Show object tree** button on the main toolbar or by clicking the ◀ or ▶ at the upper right corner of the object tree.

It contains the measuring instruments, function generators and other objects constructed in the application and shows how these are connected. These other objects are data **Sources**, **I/Os** and data **Sinks**, which all can be used in combination with the measured data. For detailed information on how to work with the objects in the object tree, refer to chapter 3.

6 Graph The Multi Channel software allows you to create and arrange multiple graphs for displaying measured or generated data. In chapter 9 you can read more about graphs.

7 User presets toolbar The Multi Channel software allows to create sets of user defined presets, to quickly load often used settings, for different applications. See section 2.2 for more information about the User Presets.

2.2 User presets


When you have several often used measurement setups, an easy and convenient way to handle these is to store them as user presets. A user preset contains instrument settings, software desktop settings like which I/Os are used and what is being shown in the graph(s), etc. and measured data.

A user preset is stored with the software and is then accessible via a User preset button on the preset toolbar.



Figure 2.2: User preset toolbar

Assigned User preset buttons are enabled and show a colored icon: 

Unassigned User preset buttons are grayed, showing a gray icon 

2.2.1 Creating a User preset

To create a user preset, you first have to setup the software and instrument for the specific measurement application. Once the setup is ready, you can create the user preset by pressing the **Ctrl** key and clicking one of the user preset buttons.

When a button is clicked that already has a preset assigned, the software will ask for a confirmation if the existing user preset can be overwritten.

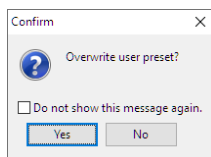


Figure 2.3: Confirm User preset overwrite

When an unassigned button is clicked, the current setup is assigned to that button. The button is now no longer grayed. When the highest unassigned button was clicked, a new, unassigned button with a higher number will be added. A maximum of 20 user presets can be assigned this way.

2.2.2 Loading a user preset

To load a specific user preset, all that needs to be done is clicking the corresponding button. All settings will be loaded from the user preset and measuring with that setting can immediately start.

2.2.3 Clearing a user preset

When a preset is no longer required, the preset location can be cleared. To clear a user preset, press the **Ctrl** and **Shift** key together and click the preset button of the user preset to clear. A confirmation dialog will be shown:

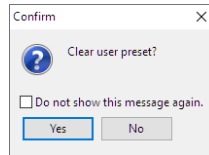


Figure 2.4: Confirm User preset clearing

When the user preset is cleared, the location becomes available again for a new user preset. When more preset locations are assigned, clearing one of the middle preset locations will unassign that location, leaving all other locations unchanged. When the cleared user preset was the highest assigned preset, all unassigned preset buttons higher than the now highest assigned preset button minus one will be removed from the toolbar.

2.2.4 Naming a user preset

When multiple presets are assigned, it may be difficult to find the proper preset for an application. When the user preset buttons are hovered with the mouse, a hint is shown, containing the title of the corresponding preset and an overview of the preset handling commands. When no title is given yet, the hint shows **User preset** and the corresponding number of the preset.

To give a user preset a title, press the **Alt** key and click the corresponding user preset button. A dialog to enter a name for the user preset is shown.

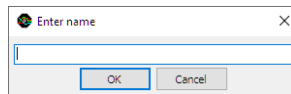


Figure 2.5: Enter User preset name


To clear an existing name for a user preset, empty the edit box and click the **OK** button.

2.3 User preset Worksets

The User presets can be ordered in Worksets, allowing to create one or more worksets for different applications, each with one or more user

presets. The Multi Channel software can handle an unlimited amount of worksets, with each up to 20 user presets. Initially, the Multi Channel software has only one default workset, where all user presets are stored.

2.3.1 Add a workset

To add a new workset, click the  **Worksets** button and select **Add workset ...** from the menu that pops up. A dialog is then shown in which the name for the workset can be entered. When an existing workset name is entered, a warning is displayed and a new name can be entered.

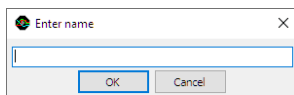



Figure 2.6: Enter workset name


Worksets are stored on disk, in separate directories for each workset. The name of the workset is used as directory name. When this directory for the workset can not be made, a warning is given.

After creating the new workset, it is also set as active workset. The new workset is empty, it can now filled with new user presets.

2.3.2 Select a workset

When more than one workset is available, a specific workset can be selected by clicking the  **Worksets** button and selecting the required workset from the menu that pops up. The user presets toolbar will now be filled with buttons for the presets in the workset. When a user preset will be created, modified or cleared, it will be done in the selected workset.

2.3.3 Remove worksets

When a workset is no longer required, it can be removed from the software. Click the  **Worksets** button and select **Remove worksets ...** from the menu that pops up. That opens a dialog where all worksets can be selected to be removed.

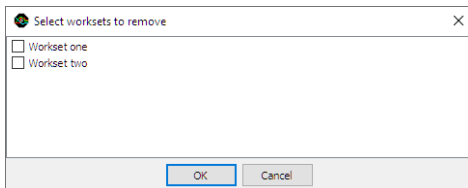



Figure 2.7: Remove worksets

All user presets in the workset as well as the workset itself will be removed. This is a permanent, non-reversible operation. The default workset can not be removed. After removing one or more worksets, the default workset is activated.

2.4 Quick Setups

To simplify setting up measurements, the Multi Channel software contains a large number of Quick Setups. A Quick Setup contains the basic settings for a specific measurement. After loading it, that specific measurement can be performed and if needed, small adjustments to the setup can be made. The Quick Setups are selected and loaded from a Quick Setup selection dialog, opened by clicking the  **Load Quick Setup** button, selecting the **Load Quick Setup** option in the File menu or by pressing hotkey **Shift** + **Ctrl** + **Q**.

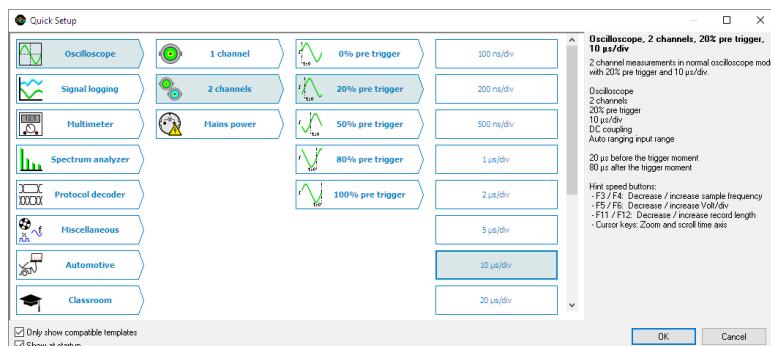


Figure 2.8: Quick Setup selection window

2.4.1 Quick Setup ordering

The Quick Setups are ordered in folders by measurement type, e.g. Oscilloscope, Protocol decoder or Spectrum analyzer. Each measurement type can have up to two levels of subfolders, indicating the discriminating properties of the measurement, e.g. number of channels or communication protocol. The final level contains one or more Quick Setups that can be loaded.

2.4.2 Selection

Folders and Quick Setups can be selected using the mouse or using the arrow keys on the keyboard. Selected folders and Quick Setups are indicated by their background color, the active selection is also indicated by a thick outline. The information pane at the right side of the dialog shows additional information regarding the selected folder or Quick Setup. It

can also show a link to a web page with additional measurement information.

Double-clicking the Quick Setup, pressing the **Enter** key or clicking the **OK** button will load the selected Quick Setup.

2.4.3 Only show compatible Quick Setups

Depending on the instrument that is used, certain Quick Setups may not be compatible with the used instrument, because it requires certain specifications that the instrument does not meet. These are e.g. a Quick Setup that requires more input channels than the instrument has, a Quick Setup that requires a higher sampling rate than the instrument supports, etc.

Quick Setups not compatible with the connected instrument are hidden from the selection dialog by default. Unchecking "**Only show compatible Quick Setups**" will add these Quick Setups to the dialog and allows to load these Quick Setups. Note that after loading the instrument setup will be different than was intended, because of the incompatibilities.














2.5 Basic (quick) functions

This section will show you how to use the *quick functions* to setup the displays to perform basic measurements. Besides setting up the display, in most situations instrument settings need to be changed to correctly measure a signal. You can read how to change instrument settings in chapter 4. The quick functions can be accessed through the main menu or the quick functions toolbar.




Figure 2.9: Quick functions toolbar


The quick functions toolbar contains quick functions to use the *active measurement instrument* as a standard virtual instrument:

-  Create a new, empty graph
-  Oscilloscope in Yt mode
-  Oscilloscope in XY mode
-  Spectrum analyzer
-  Data logger
-  Voltmeter
-  CAN decoder
-  I²C decoder
-  UART / Serial decoder
-  Create a Math channel
-  Select a color scheme
-  Select a toolbar scheme
-  Open the Arbitrary Waveform Generator window

2.5.1 New graph

To create a new, empty graph, click the  **New graph** button on the toolbar. This will create a new graph in the in the area of the largest available graph, dividing that area in two. When the width of the area is larger than $\sqrt{2}$ times its height, the new graph will be created next to the existing graph. Otherwise, it will be created below it.

2.5.2 Oscilloscope in Yt mode

To use the active measuring instrument as an oscilloscope in Yt mode, click the  **Yt graph** button on the toolbar. The channels of the active instrument will be shown in an empty graph. A new graph will be created when no empty graph is present.

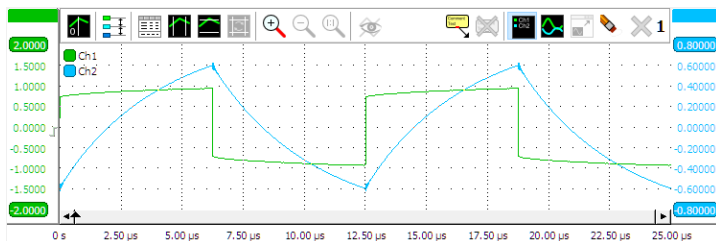



Figure 2.10: Yt graph.

2.5.3 Oscilloscope in XY mode

To use the active measuring instrument as an oscilloscope in XY mode, click the  **XY graph** button on the toolbar. Channels 1 and 2 of the active instrument will be shown in an empty graph. A new graph will be created when no empty graph is present. This function is disabled when the active instrument has less than two channels.

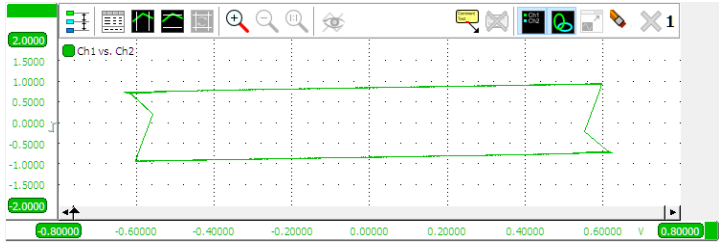



Figure 2.11: XY graph.

2.5.4 Spectrum analyzer

To use the active measuring instrument as a spectrum analyzer, click the  **Spectrum analyzer** button on the toolbar.

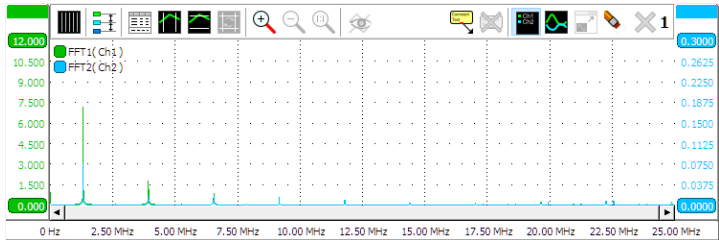



Figure 2.12: Spectrum graph.

An **FFT** I/O object will be created and connected to each channel of the active instrument. The FFT objects convert the measured time base signals to a spectrum by means of a *Fast Fourier Transform*, see section 7.5.7. The outputs of the newly created FFT objects will be shown in an empty graph. A new graph will be created when no empty graph is present.

2.5.5 Data logger

To use the active measuring instrument as a data logger, click the  **Data logger** button on the toolbar.

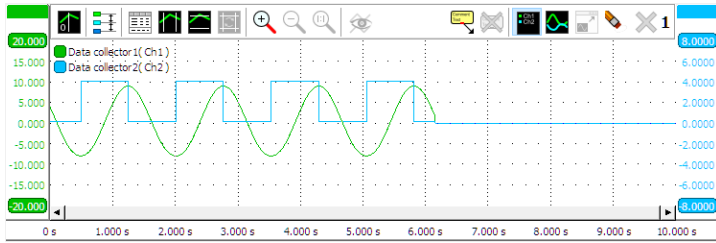


Figure 2.13: Data logger graph.


The active instrument will be set to **streaming mode** and a **Data collector** object will be created and connected to each channel of the active instrument. The outputs of the newly created data collector objects will be shown in an empty graph. A new graph will be created when no empty graph is present.

The Data logger is usually used for relatively slow signals. The instrument is set to streaming mode. The advantages over normal scope mode operation are:

- The measurement display is constantly updated during the measurement. You don't have to wait until the whole measurement is completed to see the result.
- Longer measurements are possible than would fit in the instrument's memory in normal scope operation.

Read more about the differences between scope mode and streaming mode in section 4.3.1.

2.5.6 Voltmeter

To use the active measuring instrument as a **voltmeter**, click the  **Voltmeter** button on the toolbar. The channels of the active instrument will be shown in a newly created meter object. Read more about meters in section 10.

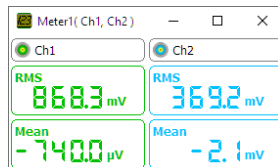



Figure 2.14: Voltmeter.

By default, the measurements **Mean** and **RMS** are enabled. Other measurements can be added for each channel. Examples are: minimum, max-

imum, top-bottom, variance, standard deviation, frequency and for frequency data: Total Harmonic Distortion. See appendix B for a list of the available measurements and a description.


2.5.7 CAN decoder

To use the active measuring instrument as a **CAN decoder**, click the  **CAN decoder** button on the toolbar. A new CAN decoder I/O will be created and connected to an newly created **Table** sink to display the decoded CAN data. Read more on the CAN decoder in section 7.7.4.


If the active instrument has two or more channels, the user is asked whether one or two channels should be used. When two channels are used, both the *CAN high* and *CAN low* signal should be measured. The difference signal *H-L* is calculated with a **Add/Subtract** I/O and fed into the analyzer.

If only one channel is used, this may measure *CAN high*, or the differential CAN signal *H-L*. The latter is only possible with a differential input.



2.5.8 I²C decoder


To use the active measuring instrument as a **I²C decoder**, click the  **I²C decoder** button on the toolbar. A new I²C decoder I/O will be created and connected to the first two channels of the active instrument. It is also connected to an newly created **Table** sink to display the decoded I²C data. The I²C decoder can only be used with instruments with two or more channels. The first channel will be used as I²C SCL (clock) and the second as I²C SDA (data). Read more on the I²C decoder in section 7.7.2.

2.5.9 UART / Serial decoder


To use the active measuring instrument as a **serial decoder**, click the  button on the toolbar. A new Serial decoder I/O will be created and connected to the first active channel of the active instrument. It is also connected to an newly created **Table** sink to display the decoded UART / serial data. It can be used to analyze RS232, RS485, MIDI, DMX or other compatible serial buses. Read more on the UART / Serial decoder in section 7.7.3.


2.6 Printing


You can print your measurements just like they are shown on the screen. Each graph is printed to a separate page. Choose  **Print...** in the **File** menu, press the  **Print** button on the file toolbar, or use the hotkey **Ctrl-P**.

The graphs are printed with the selected **graph scheme**. By default a black and white printing scheme is used, but you can also use a scheme with colors or define your own color scheme. You can select another scheme or change colors for printing in the application settings, in *Graph*→*Print*. To check how the graphs will look on paper without actually printing them, click  **Print preview...** in the **File** menu.


2.7 Settings

Many applications settings can be changed with the settings window. You can open the settings window by clicking the  **Settings...** item in the **Tools** menu.

Language The user interface of the Multi Channel software can be set to many different languages via the  **Select language...** item in the **Tools** menu. This item is always displayed in the currently selected language and in English to make it easily accessible. Changing the user interface language requires the Multi Channel software to be restarted.






Graph schemes You will notice the colors of the screen shots in this manual are different than the standard colors on your screen when you use the software. The screen shots were made using a different **graph scheme**. You can choose from several schemes or make your own schemes for on screen as well as for printing. Schemes can be altered or added via the application settings, in *Graph*→*Display*. Schemes can be selected using the  **Color scheme** button on the main toolbar.

Meter schemes Just as the colors of the graphs, the colors of the meters can also be changed with schemes. Schemes can be altered or added via the application settings, in *Meter*→*Display*.

Toolbars The instrument and channel toolbars (see sections 4.2.1 and 4.2.2) are fully customizable. Buttons and readouts for all settings can be dragged on or off the toolbar to meet specific needs. Also the icon size and text size can be adjusted. Toolbar configurations can be stored for easy reconfiguring different setups via the application settings, in *Instruments*→*Oscilloscopes*→*Toolbar*. Toolbar schemes can be selected using the  **Toolbar scheme** button on the main toolbar.


The Multi Channel software has a modular structure, with *measuring instruments*, *function generators* and other objects constructed in the application. Besides measuring data with the TiePie engineering measuring instruments and displaying it like on a conventional scope, it is possible to do different kinds of processing on the measured data. Data can be combined with other measured data or with software generated data.

The objects that are available, are divided into five groups:

-  **Oscilloscopes:** The detected and opened oscilloscopes with their channels. See chapter 4.
-  **Function generators:** The detected and opened Function generators. See chapter 5.
-  **Sources:** objects that generate data, like software generators. Sources only have output(s) (like the channels of an instrument). See chapter 6.
-  **I/Os:** objects that accept data (Input), process this data in a specific way, like e.g. adding, multiplying, filtering, etc. and then generate the processed data (Output). These objects have input(s) and output(s). See chapter 7.
-  **Sinks:** objects accepting data, like e.g. graphs and meters and tables. These objects only have input(s). See chapter 8.

Objects can be combined and connected to each other to create measurement applications that range from very simple to very complex.

3.1 Object tree

All objects are located in the object tree. The object tree is situated at the left side of the main window of the application. It can be resized by dragging the splitter bar at the right of the object tree. It can be revealed and hidden by clicking the  **Show object tree** button on the main toolbar or by clicking the ◀ or ▶ at the upper right corner of the object tree. In the application settings can be determined whether the object tree is default visible or hidden. The status of the object tree (visible or hidden) is also saved when a measurement is saved and restored when that measurement is loaded again.

3.2 Creating objects

A new source, I/O or sink is created by clicking on the root item in the object tree and selecting the object of your choice from the popup menu. In figure 3.1 the *I/Os* item is clicked and the category **Analyze** is selected in the popup menu.

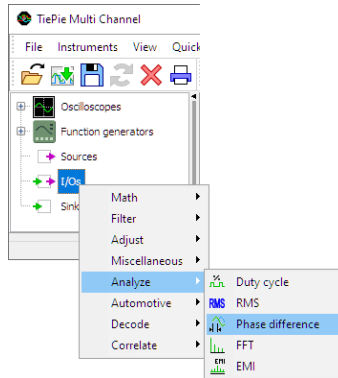


Figure 3.1: Creating an I/O object

3.3 Configuring objects

To control the behavior of an object, several settings and actions are available. These are divided in two categories: object specific settings and actions which only occur on that specific object and common settings and actions, which occur on each object.

The settings and actions can be accessed through a popup menu which is shown when the object is right clicked in the Object tree.

The settings of an object can also be accessed through its settings window which is shown when the object is double clicked in the Object tree. Each object gets its own settings window, affecting only the settings of that specific object.

The settings window shows the icon of the object and a small description of its function. If the object has a source, it shows the source(s) of the object and also gives access to the settings window of the source(s). It also shows the object specific settings of the object and some important common settings.

The settings windows will either show all settings of an object, or they are simplified, showing only the most important settings. Checking **'Advanced'** will then show all settings. The ability to select between simple and advanced is determined in the program settings of the Multi Channel software.

Changes to settings are activated immediately. The settings window can be left open to allow easy changing of object settings.

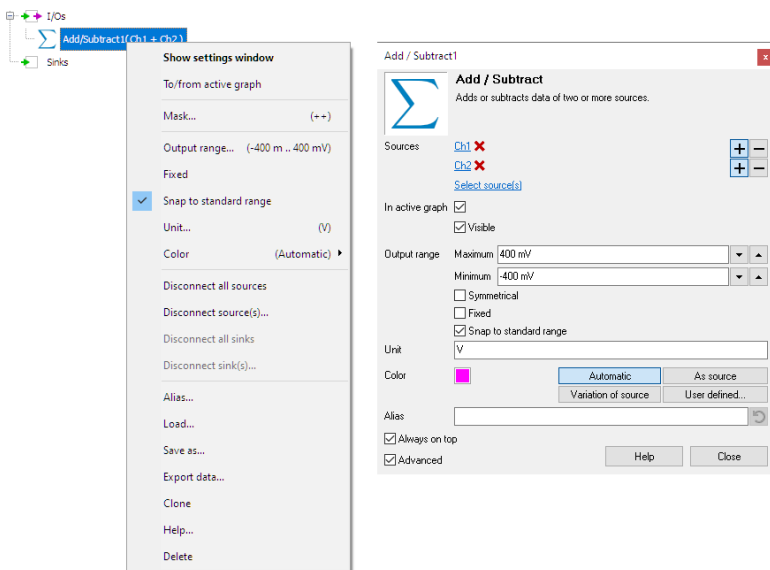


Figure 3.2: Object popup menu and settings window.

It is possible to change specific settings of multiple objects at the same time. Multiple objects can be selected by selecting each object individually by clicking them with the mouse and keeping the Ctrl key pressed. The popup menu that is then opened by right-clicking the selection will show all common settings and actions of the selected objects. A setting can then be changed and all selected objects will get the same value for that setting.

This can be used when e.g. 4 channels are measured and each channel is filtered using a Filter I/O and for all filters the cut off frequency needs to be adjusted.

3.4 Cloning objects

Another way to create objects, is to clone existing objects. A new object of the same type will be created with the same settings and data as the original. To clone an object, choose **Clone** in its popup menu.

3.5 Connecting objects

The different objects in the object tree can be connected to other objects by means of drag and drop. You can drag a source or I/O object on a sink

object or I/O block. The output of the source or I/O that is dragged will then connect to the input of the I/O or sink it is dropped on.

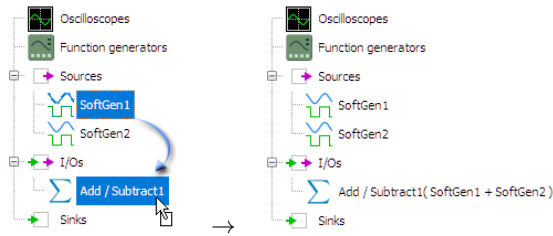


Figure 3.3: Connecting objects.

In the example in figure 3.3, two SoftGen sources are dropped onto the Add / Subtract I/O to add their signals together. The Add / Subtract I/O displays its sources between braces.

The output of the Add / Subtract I/O now contains the signals of the two SoftGens, added together. To display this result in e.g. a graph, drag the Add / Subtract I/O in a graph.

When a source with multiple outputs (e.g. an instrument) is dropped on an I/O or sink with one input, the I/O or sink is cloned multiple times and each output of the source is connected to such an I/O or sink.

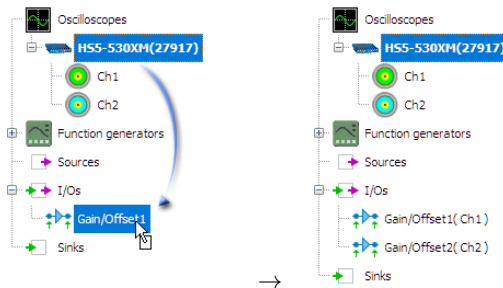


Figure 3.4: Connecting and cloning objects.

3.6 Disconnecting objects

There are different ways to disconnect objects. In the popup menu of each source you can find the items **Disconnect all sinks** and **Disconnect sink(s)...** When you click the latter, a dialog will popup which lets you choose which sinks to disconnect.

In the popup menu of sinks, you will find similar items to disconnect its sources. Since I/O objects are both sink and source, their menu contains items for disconnecting sources as well as sinks.

When the source of an object is disconnected, the data in the object remains present and can still be shown in a graph, meter or table. It can also still be used by other objects (I/Os or sinks) connected behind the object.

3.7 Inserting objects between other objects

To insert an object between two objects that are already connected to each other, first the objects have to be disconnected from each other. Then the new object can be placed between them.

Suppose a streaming measurement is performed where Channel 1 is connected to a Data Collector I/O and the data needs to be filtered before collecting it in the Data Collector I/O. The steps to do that are:

1. Right-click the Data Collector I/O and select Disconnect all sources from the popup menu.
2. Create a Filter I/O that has to be connected between Channel 1 and the Data Collector I/O.
3. Connect Channel 1 to the Filter I/O by dragging and dropping Channel 1 on the Filter I/O.
4. Drag and drop the Filter I/O on the Data Collector I/O.

The Filter is now placed between Channel 1 and the Data Collector.

3.8 Deleting objects

When an object is no longer needed, it can be deleted by right-clicking it in the Object tree and selecting **Delete** from its popup menu. If the object was connected to other objects, these connections are removed. These other objects stay intact.

3.9 Saving objects

An object can be saved to disk, into a TPO file (see D.2) to be used again in a later stage. Both all settings of the object as well as the data in the object are saved in the file. Right-click the object in the Object tree and select **Save...** from the popup menu to save the object.

When multiple objects are selected and the selection is saved to a TPO file, all objects, including their settings and data are saved. But also the connections between the objects are saved in the file. This makes it possible to create a complex calculation using multiple object and save that to disk. Later, this complete calculation can be loaded in the software

again and be used again. Opening this TPO file creates all saved objects and restores their interconnections. All that needs to be done then is to connect a source and a sink or graph. A TPO file can be read multiple times to load the calculation more than once.

3.10 Exporting object data

The data in an object can be exported to disk, in several common file formats. Right-click the object in the Object tree and select **Export data...** from the popup menu to export the object data. See also 11.4.

3.11 Using aliases

You can change the displayed name of the objects. This can be handy when you are dealing with a lot of objects and have problems keeping them apart. You can for example give channel one of your instrument the alias "Battery", when it is measuring a battery voltage. To give an object an alias, use the item **Alias...** in its menu or the **Alias** field in its settings window. To remove an alias and revert to the original name of the object, set the alias to an empty string or press the reset button of the **Alias** field.

3.12 Example

In this section you will find an example of how to use objects to add the signals of two channels together. To do this, create an **Add/Subtract** object and drag the sources you want to add on top of it in the Object tree. Figure 3.5 shows the result of summing the two channels of a Handy-scope HS5. To make comparison of the signals easier, put the signals on one axis by dragging the axes on top of each other, see section 9.3.

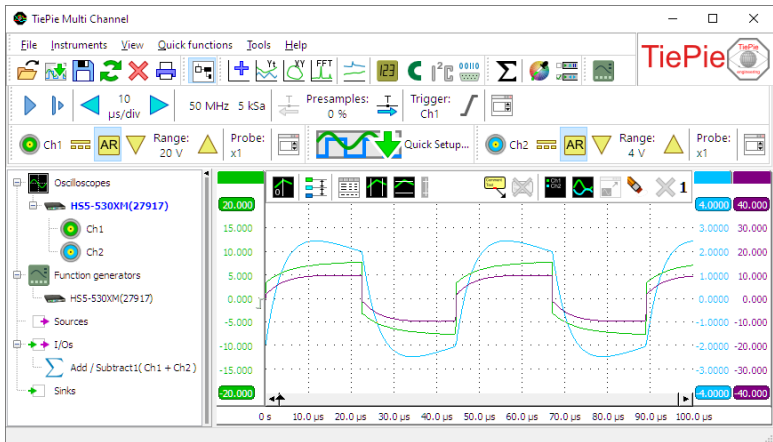


Figure 3.5: Summing two channels



Figure 4.1: Instruments

The TiePie engineering instruments, like the Handyscope HS5, consist of one or more input channels to acquire measurements. Some instruments also have a *function generator* or *Arbitrary Waveform Generator* output. In the software, an AWG is regarded as a separate independent device. Therefore, in the software and this manual, the word *instrument* refers only to the data acquisition part of the physical device.

The first part of this chapter shows how to select and open instruments in the software. The second part of this chapter shows how to control the measuring instruments and how to change the different settings. The sections that follow will show how to perform *streaming* measurements and how to *combine* instruments.



For instrument specific information, refer to the user manual of your instrument.

4.1 Selecting and opening instruments

When the Multi Channel software is started, all locally connected supported instruments not in use by other applications will be detected. When only one instrument is found, it will be opened automatically.

When more supported instruments are detected, the Open instruments dialog is shown with the detected instruments. To open an instrument, place a check in the checkbox for that instrument. The checked instrument(s) will be opened when the dialog is closed using the **OK** button. Removing a check will close an already opened instrument.

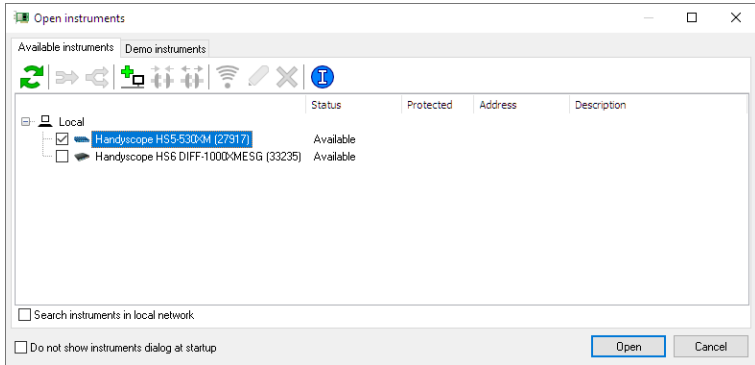




Figure 4.2: Open instruments dialog


The toolbar in the dialog gives access to additional functionality regarding selected instruments. Instruments can be selected by clicking them with the mouse, when more instruments need to be selected, keep the **Ctrl** pressed as well.


To update the list of detected instruments, the  **Refresh** button or function key **F5** can be used.

To show detailed information of the selected instrument, click the  **Info** button. This will show a dialog with instrument name, serial number, calibration date, driver version etc.

4.1.1 Combining instruments

The software supports combining multiple instruments to a large combined instrument. When the instruments have a **CMI interface**, this combining is done **automatically** when the instruments are coupled to each other via the CMI interface when the software is started. Only the combined instrument will be listed in the dialog.

When the instruments do not have a CMI interface, combining them must be done manually. To combine these instruments, select them by clicking them with the mouse having the **Ctrl** key pressed. Clicking the  **Combine** button will then combine the instruments. The instruments also need to be connected using a special cable.

When a manually combined instrument (consisting of instruments that do not have a CMI interface) is listed, it can be split up into individual instruments again by selecting the combined instrument and clicking the  **Separate** button. The special coupling cable then has to be removed.

4.1.2 Network instruments

WiFiScopes can be connected via a LAN cable or via WiFi. Refer to the WiFiScope Quick Start Guide that is delivered with the instrument for more information.

Selected USB Handyscopes can also be connected via the network, when using the **TPISS** Instrument Sharing Server, see www.tiepie.com/tpiss.

When a WiFiScope is connected to the network or a USB Handyscope is connected to a computer in the network running the Instrument Sharing Server, the Multi Channel software can access, open and use the instrument from a different computer in the network.

To open a network instrument, enable **Search instruments in the local network** in the Manage instruments dialog. That will scan the local network for WiFiScopes and for computers running the TPISS and will show the connected instruments on these computers. Connected instruments that are in use by another application will be listed as **Not available** and can not be opened.

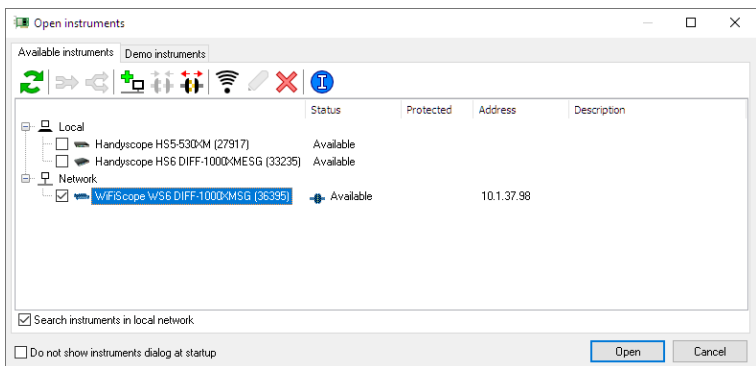


Figure 4.3: Network instruments selection dialog

To open an instrument, place a check in the checkbox for that instrument. The checked instrument(s) will be opened when the dialog is closed using the OK button. Removing a check will close an already opened instrument.

The toolbar in the dialog gives access to additional functionality regarding the detected network instruments.

When a computer running the TPISS Instrument Sharing Server is not located in the local network, it will not show up in the dialog by default. Use the **Add network location** button to add its location to the list of searched network locations.

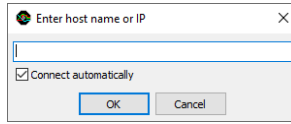


Figure 4.4: Add network location dialog

The network location can either be entered as IP address or as host name. When **Connect automatically** is checked, the Multi Channel software will automatically connect to the remote server and show its connected instruments. The location will also be added to the list of used locations. Editing this list can be done via the program settings.

To connect to a listed remote server, select the server by clicking it with the mouse and then click the **Connect** button. The Multi Channel software will connect to the server and list all connected instruments to that server. When network notifications are enabled, these will be shown for the connected server(s).

To disconnect from a connected remote server, select the server by clicking it with the mouse and then click the **Disconnect** button. The Multi Channel software will disconnect from the server and no longer list the connected instruments to that server. Notifications for the disconnected server are no longer shown.

When a remote server is connected to the network via a wire and it also has the possibility to connect to the network wireless using WiFi, the server can be instructed to connect to the WiFi network by selecting the server and clicking the **Setup WiFi** button. This will open a WiFi connection dialog in which the available networks are shown.

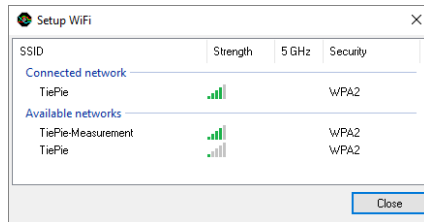


Figure 4.5: Setup WiFi dialog

To connect to a WiFi network, double click it, which will open a dialog to enter the WiFi network password. After that, the server will try to connect to the WiFi network, after which the cable can be removed. To disconnect from a WiFi network, double click the connected WiFi network, which will open a confirmation dialog for disconnecting from the WiFi network.

When the IP address or host name of a manually added network location has changed or was entered incorrect, it can be edited by selecting that

network location by clicking it with the mouse and then pressing the **Edit** button.

When a manually added network location is no longer required, it can be removed from the search list by selecting it with the mouse and clicking the **Delete** button.

4.1.3 Demo instruments

When no supported instrument is detected, the Open instruments dialog is shown with a demo instrument selection, showing all available demo instruments and their key specifications. The required demo instrument can then be selected and will be opened when the dialog is closed using the OK button.

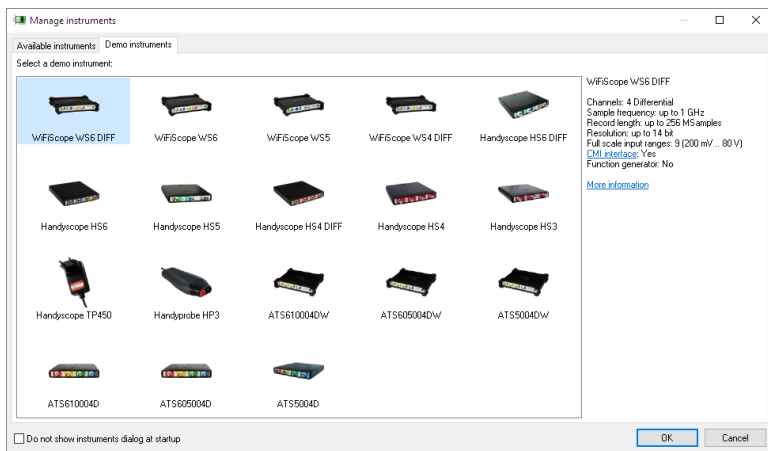


Figure 4.6: Demo instrument selection dialog

4.2 Controlling instruments

The WiFiScope(s) and/or Handyscope(s) opened by the Multi Channel software can be controlled in different ways. The most convenient way is to control the oscilloscopes and their channels by means of the toolbars, shown at the top of the main window, see figure 4.7.

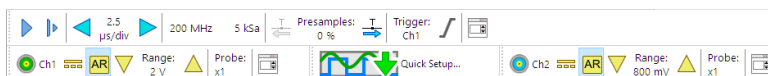


Figure 4.7: Instrument toolbar

An other way is to control the oscilloscopes by means of popup menus, opened by right clicking the oscilloscope. The oscilloscope properties can

also be accessed through its settings window which is shown when the scope is double clicked or the  **Settings window** button on the instrument toolbar is clicked.

The quickest way to change the settings is using the hotkeys. When you are using multiple instruments, the hotkeys affect the *active instrument*. This is the instrument highlighted in the object tree. You can make another instrument active by clicking on it in the object tree or using the hotkey **Ctrl** + **n**, with *n* the instrument number (1..0). See appendix C for a complete list of the available hotkeys.

4.2.1 Using the oscilloscope toolbar

An oscilloscope toolbar as shown in figure 4.8 is created for each opened oscilloscope. This convenient toolbar has large, clear, touch-screen friendly buttons for all oscilloscope settings. It shows the current settings of the oscilloscope and allows to change all settings.

When multiple instruments are detected and opened, each instrument gets its own oscilloscope toolbar and channel toolbars. To save screen space, these are tabbed. Clicking the tab of a specific instrument will show the oscilloscope toolbar of that instrument and will also show the channel toolbars of that instrument.

The oscilloscope toolbar is fully configurable through the program settings. You can set the button size, add or remove buttons and change the order of the buttons.

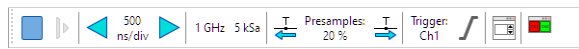





Figure 4.8: Oscilloscope toolbar

In the default toolbar setup, it shows the time base setting and buttons to adjust the time base, sampling rate and record length indicators, pre trigger setting controls and readout, the trigger settings and a button to open the instrument settings window. Clicking the text labels will open a popup menu to change the corresponding setting. Refer to the help for detailed information of all controls for the oscilloscope toolbar.

Measurements can be started and stopped by using the  **Start**,  **Stop** and  **One Shot** buttons on the instrument bar. When the start button or its hotkey **S** is pressed, the instrument will start measuring continuously until the measuring is stopped by pressing the button or hotkey again. When the one shot button or its hotkey **O** is pressed, one measurement is performed.

4.2.2 Using the channel toolbar

A channel toolbar as shown in figure 4.9 is created for each oscilloscope channel. This convenient toolbar has large, clear, touchscreen friendly buttons for all channel settings, such as input range and coupling. It shows the current settings of the channel and allows to change all settings by clicking on the buttons.

When a combined instrument is created and opened the channels of the combined instrument are grouped in a tab per sub instrument. This reduces the space occupied by the toolbars, leaving more space for the graph(s). It also simplifies adjusting a channel for a specific sub instrument in the combined instrument.

The channel toolbar is fully configurable through the program settings. You can set the button size, add or remove buttons and change the order of the buttons.



Figure 4.9: Channel toolbar

In the default toolbar setup, it shows a channel indicator, with its name and showing the used channel color, input coupling controls, input range controls, the probe setting and a button to open the channel settings window. Clicking the text labels will open a popup menu to change the corresponding setting. Refer to the help for detailed information of all controls for the channel toolbar.

4.2.3 Using popup menus and settings windows

When an oscilloscope or channel is clicked in the object tree, a popup menu is opened, showing all settings and actions which are available for that oscilloscope or channel.

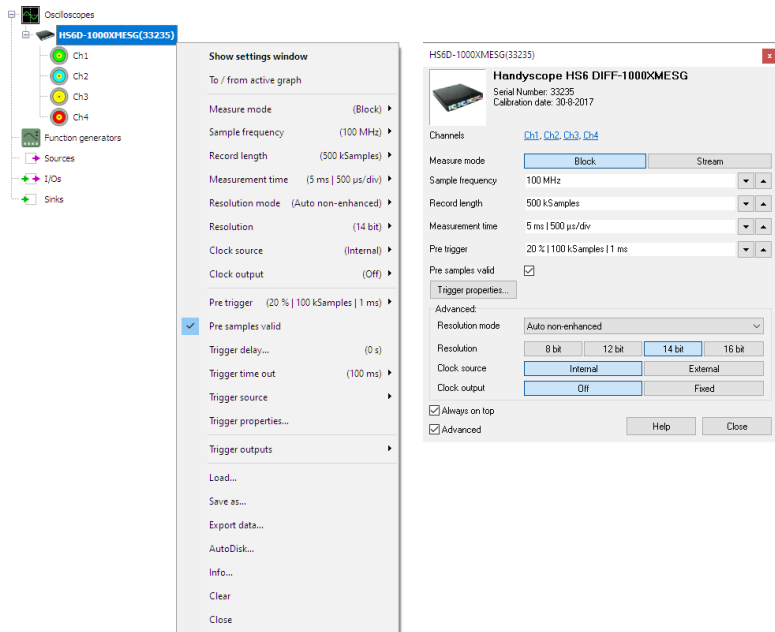


Figure 4.10: Oscilloscope popup and settings window

Using the option **Show settings window**, a settings window is opened, which allows to changes all available settings. The settings window is also opened when the oscilloscope or channel is double clicked or the **Settings window** button on the oscilloscope or channel toolbar is clicked. Only the most used settings and actions are available for an oscilloscope or channel.

4.3 Settings

The settings of an instrument can be divided in three groups: *instrument settings*, *trigger settings* and *channel settings*. Instrument settings, like *record length* and *sample frequency*, apply to **all** channels of an instrument. Channels settings, like *input range* and *signal coupling*, apply to individual channels.

Settings can be changed with the instrument bar and through popup menus of the instrument and its channels in the object tree. In the following sections you will find more information about the different settings.

4.3.1 Instrument settings

Instrument settings can be divided in a few groups: measure mode settings, time base settings and trigger settings. The hotkeys mentioned in

this section apply to the *active instrument*. This is the instrument highlighted in the object tree. You can make another instrument active by clicking on it in the object tree or using the hotkey **Ctrl** + **n**, with *n* an instrument number (1..0).

Measure mode

By default, most TiePie engineering measuring instruments work in *scope* or *block* mode. In this mode, the complete measurement is recorded in the instrument's memory. After the full record has been measured, the data is transferred to the computer. The next measurement is started after the data has been processed, therefore there are gaps between the measurements.

Besides working in block mode TiePie engineering instruments support working in streaming mode. In this mode the measured data is transferred directly to the computer, without using the internal memory of the instrument. This makes it possible to perform continuous measurements without gaps.

You can change the measure mode of an instrument via its settings window and via its popup menu in the object tree.



To change the measure mode of an instrument, the instrument must first be set to pause.

Streaming versus block measurements Both block and streaming measurements have their advantages and disadvantages. The key features of both modes are listed below. Block mode (/Scope mode):

- + Fast measurements are possible
- Record length is limited by the instrument's memory size

Stream mode:

- + Long measurements are possible
- Sample speed is limited by the data transfer rate to computer and the computer speed

In block mode, the next measurement is started after the previous data has been transferred to the computer. This means that there will always be a (small) gap in between the measurements. In streaming mode, no data is missed. All successive data chunks can be connected to form one big measurement.

A disadvantage of the streaming mode is that the maximum measurement speed depends on the data transfer rate from the instrument to

the computer and on the overall system performance. Also, in streaming mode, no triggering is available. Measurements are started and stopped manually.

Using streaming mode In streaming mode, the measured data will arrive in chunks. Each of those chunks contains *record length* samples which are sampled at *sample speed*. These two settings determine the time it takes for a chunk of data to be measured, which is also the time between the arrival of two consecutive chunks.

Each time when a chunk arrives, the computer will process it and, depending on what is connected to the channel(s) in the software, perform mathematical operations with it, show it in a graph and/or store it to disk.

When the time between two consecutive chunks is short (the update rate high), there is a risk that the computer is not able to process all data before the new chunk arrives. This may result in the streaming measurement being aborted.

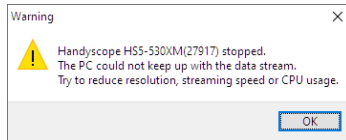


Figure 4.11: The PC could not keep up with the data stream.

It is recommended to keep the update rate lower than approximately 10 updates per second (a chunk measuring time larger than 0.1 second). Additionally it may help to reduce the post processing load by reducing the graph size by resizing the graphs and temporarily hiding lines in the graph(s) until the measurement is ready.

Collecting streaming data In streaming mode, successive measurements will arrive in chunks. Each of those chunks contains *record length* samples and can be connected seamlessly with previous and next data chunks.

The Data collector I/O can be used to collect the successive measurements and combine them into one big block of data of up to 50 million samples.

Resolution

The resolution of an instrument determines the smallest voltage step that can be detected. The resolution is indicated with a number of bits. Every extra bit doubles the accuracy. This means that the smallest detectable voltage step will be 16 times smaller when you use a 12 bit resolution instead of 8 bit. You can change the resolution of an instrument via its settings window or via its popup menu in the object tree.

Time base

The combination of sampling rate and record length forms the time base of an oscilloscope. To setup the time base properly, the total measurement duration and the required time resolution have to be taken in account.

With the required measurement duration and sampling rate, the required number of samples can be determined:

$$\text{record length} = \text{measurement duration} * \text{sampling rate}$$

where record length is defined in samples, measurement duration in seconds and sampling rate in Samples/second.

Time per div The Multi Channel software provides controls to change record length and sample frequency simultaneously to specific combinations to obtain certain time/div values.



Figure 4.12: Time per div controls

Because a certain time/div setting can be created from different combinations of record length and sampling frequency, the Multi Channel software must decide which combination is used. The software will try to use the highest sample frequency possible and will adjust the record length accordingly to obtain the required time/div setting.

To avoid the record length becoming this long that collecting the measured values takes longer resulting in the scope responding slower, the record length for time/div settings is limited to a value that can be set in the program settings. When the time/div setting is adjusted by one of the corresponding controls, the maximum record length is limited to this value. This limit does not apply to manually adjusting the record length.

Sample frequency The *sample frequency* is the rate at which the instrument takes its samples of the input signals. It can be set to predefined or user defined values via different menus. Use the hotkeys **F3** and **F4** to decrease or increase the sample frequency.

Record length The *record length* is the number of samples the instrument takes during each measurement or record. It can be set to predefined or user defined values via different menus. Use the hotkeys **F11** and **F12** to decrease or increase the record length.

Pre-trigger With digital storage oscilloscopes, the record length determines the number of samples that are measured. All these samples can be measured after the trigger has occurred. It is however possible to

measure (a part of) the record before the trigger occurs, by selecting *pre samples*.

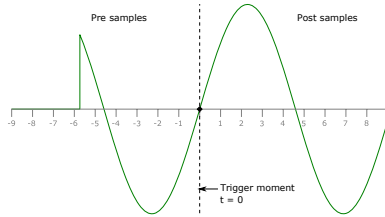


Figure 4.13: Pre trigger and post trigger

The total record will then be divided in a pre trigger part and a post trigger part, respectively containing pre samples and post samples. This way it is possible to “look back in time” since the pre samples were captured before the trigger moment.

With the TiePie engineering instruments it’s possible to define the trigger moment at any position in the record. It is set as a percentage of the total record.

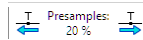


Figure 4.14: Pre trigger controls

You can change the pre/post-sample ratio with the buttons on the instrument bar and via different menus or with hotkeys **Shift** + \leftarrow and **Shift** + \rightarrow .



When setting the trigger moment through a popup menu, the option **User defined...** also allows to set the value as a number of samples or as a time.

4.3.2 Trigger settings

Most oscilloscopes are equipped with a trigger circuit to start measuring when a certain condition occurs in the input signal. Triggering is used both for capturing unique signal events and to stabilize the display of repetitive signals. Without triggering, signals are measured and displayed at random times. The trigger circuit has several settings which are divided into instrument trigger settings and channel trigger settings.

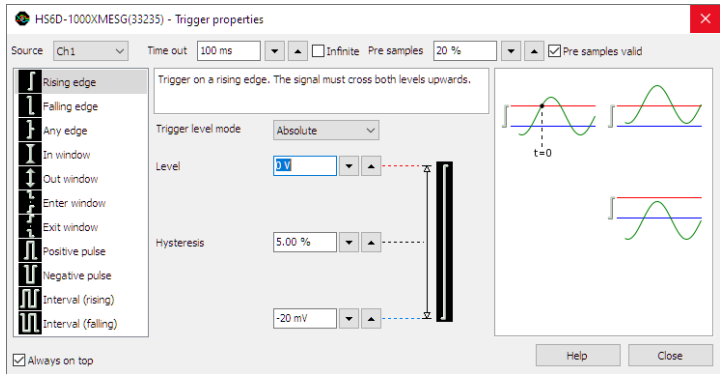



Figure 4.15: Trigger settings dialog

Most trigger settings are combined in the Trigger settings dialog. To open the trigger settings dialog, click the  **Trigger settings** button on the instrument toolbar. The trigger settings dialog allows to view and control instrument and channel trigger settings. Additionally, it gives an explanation on the selected trigger type and examples that do cause a trigger (left column) and do not cause a trigger (right column). There is a dialog available for each opened instrument.

Trigger source

The *trigger source* setting of the instrument determines which trigger signals are used to trigger the instrument.

The trigger source can be set to a single channel or to any combination of channels or other trigger sources like digital external inputs or generator triggers. The sources can be logically combined using an OR function via the windows that is shown when source **Advanced** is selected.

When no trigger source is selected, the trigger system is disabled and the instrument is free-running: it will start measuring the post samples directly.

Trigger time-out

Once the trigger conditions are set and the measurement is started, the instrument will wait until the trigger conditions are met before the post samples are measured and the measurement is finalized. If the trigger conditions are set in such a way that the input signal(s) will never meet the trigger settings, the instrument will wait forever. When no measurement is performed, no signals will be displayed.

To avoid that the system will wait infinitely, a **trigger time-out** is added to the trigger system. When after a user defined amount of time after start-

ing the measurement still no trigger has occurred, the trigger time-out will force a trigger. This will ensure a minimum number of measurements per second (On conventional desktop oscilloscopes, this is called Trigger mode AUTO).

The trigger time-out is entered as a number, representing the delay in seconds. There are two special values for the trigger time-out setting:

- **trigger time-out = 0**

Immediately after starting a measurement a trigger is forced. Basically this bypasses the trigger system and the instrument always measures immediately. No pre samples are recorded. The instrument is free-running, just like when no trigger source is selected.

- **trigger-time-out = infinite**

The system will wait infinitely for a trigger. The software will never force a trigger, only when the trigger conditions are met, a trigger will occur and a measurement will take place. This setting is particularly useful for single shot measurements. On conventional desktop oscilloscopes, this is called Trigger mode NORM.



When the instrument should measure **only** when a trigger event occurs the trigger time-out must be set to infinite!

Trigger type

Channel trigger circuits monitor the channels continuously and generate a trigger signal when the input signals meet some predefined trigger condition.

There are several different trigger types:

- **Edge trigger**
trigger on a rising, falling or any edge in the signal
- **Window trigger**
trigger when the signal enters or leaves a certain window or range, optionally shorter/longer than a specified time
- **Pulse width trigger**
trigger on a positive or negative pulse in the signal wider/narrower than a specified width, or inside/outside a specified time frame
- **Interval trigger**
trigger on a periodical signal with a period time shorter/longer than a specified length, or inside/outside a specified time frame

Changing the trigger type of a channel in the Multi Channel software can be done in the trigger settings window, by right-clicking the channel in the object tree or by right-clicking the trigger symbol in on the corresponding axis in a graph.

Trigger level

All oscilloscope channel trigger types use one or two trigger levels. Trigger levels can be set for each channel individually. The trigger level is set either in absolute values or in relative values, depending on the selected Trigger level mode.

Trigger levels can be set for each channel individually. Changing a trigger level of a channel in the Multi Channel software can be done in the trigger settings dialog. The level can also be adjusted by dragging the trigger symbol on the axis in a graph up or down, to the required level.

Trigger hysteresis

All oscilloscope channel trigger types use one or two trigger hystereses. The hysteresis defines the distance between the firing level and the arming level and determines the sensitivity of the trigger system. A small hysteresis means that the arming and firing level are close to each other and a small signal change will be enough to cause a trigger. A large hysteresis means that the signal change must be large before a trigger is generated. This makes the trigger system less sensitive to noise.

Trigger hysteresis can be set for each channel individually. Changing the trigger hysteresis of a channel in the Multi Channel software can be done in the trigger settings dialog. The hysteresis can also be adjusted by dragging one of the trigger symbol edges in a graph up or down, to the required position.

Trigger condition

Several oscilloscope channel trigger types can have an additional trigger condition and corresponding trigger condition time. Some trigger conditions have two trigger times, defining a trigger condition time frame. The following trigger conditions are available:

- **None**
there is no additional trigger condition.
- **Shorter than**
the signal requirements defined by the trigger type must last shorter than the specified trigger condition time to cause a trigger.
- **Longer than**
the signal requirements defined by the trigger type must last longer than the specified trigger condition time to cause a trigger.
- **Inside**
the length that the signal requirements defined by the trigger type last, must be inside the trigger condition time frame to cause a trigger.
- **Outside**
the length that the signal requirements defined by the trigger type last, must be shorter than or longer than the trigger condition time frame, in other words outside the trigger condition time frame to cause a trigger.

Setting the trigger condition can be done in the trigger settings window or via the popup menu that is shown when the channel is right-clicked in the object tree.

Trigger condition time The trigger condition time specifies the duration of a specific signal condition, in seconds. When two trigger condition times are available, the two times define a trigger condition time frame, in seconds.

4.3.3 Channel settings

Channels settings apply only to one channel. Channel settings can be adjusted by using the buttons and labels on the Channel toolbar, via the channel settings windows and popup menus and by hotkeys.

The hotkeys mentioned in this section apply to channels of the active instrument. The hotkeys on itself apply to channel 1. To change settings of the other channels add the key combinations shown in table 4.1.



Channel	Key(s)
Ch. 1	-
Ch. 2	Ctrl
Ch. 3	Shift
Ch. 4	Ctrl + Shift
Ch. 5	Alt
Ch. 6	Alt + Ctrl
Ch. 7	Alt + Shift
Ch. 8	Alt + Ctrl + Shift

Table 4.1: Selecting channel for hotkeys



Input type

Most oscilloscopes are equipped with standard, single ended inputs, which are referenced to ground. This means that one side of the input is always connected to ground and the other side to the point of interest in the circuit under test. Therefore the voltage that is measured with an oscilloscope with standard, single ended inputs is always measured between that specific point and ground.

Some oscilloscopes are equipped with a differential input. A differential input is not referenced to ground, but both sides of the input are "floating". It is therefore possible to connect one side of the input to one point in the circuit and the other side of the input to the other point in the circuit and measure the voltage difference directly.

When the oscilloscope is equipped with option **SafeGround**, the inputs can be switched between  **single ended** and  **differential**, using the dedicated button on the channel toolbar.

Input coupling

The channels of an instrument can be set to  **AC** or  **DC** coupling.

DC coupling allows for both the AC and the DC components of the input signal to pass and being measured. When AC coupling is used, the DC component of the input signal is rejected and only the AC component is measured. Use AC-coupling when you are investigating an AC signal on a rather large DC component.

Use the dedicated button on the channel toolbar or the hotkeys **A** (AC) and **D** (DC) to change the input coupling.

When the instrument is equipped with option **SureConnect**, the scope can also measure **resistance**. This is then accessed via the input coupling, which gets an extra setting Ω .

Input range

The *input range* of a channel determines how large signals can be measured. The input range is defined as full scale value, the highest voltage that can be measured, both in positive as in negative direction. When the input range is e.g. 8 V, signal values between -8 V and +8 V can be measured. Values outside that range will be clipped.

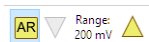


Figure 4.16: Input range controls

For oscilloscopes, another way of defining the input range is also used, in V/div. This is the full scale input range divided by the number of grid divisions that are used to display that range. A typical oscilloscope display has 8 division vertically, covering -full scale to +full scale. A full scale input range of e.g. 8 V equals 2 V/div.

Each channel has an *auto ranging* (hotkey **R**) setting, which will automatically adjust the input range to match the applied signal.


The input range of a channel can be set via the dedicated controls on the channel toolbar, as well as in the settings window of the channel and via the popup menu that is opened by right-clicking the channel in the object tree. You can decrease or increase the range with hotkeys **F5** and **F6**.

Probe

When using an attenuating oscilloscope probe or e.g. a current clamp or other special probe, the software needs to compensate the measured values for the attenuation (or gain) of the used probe. The probe readout shows the current probe setting of a channel. Clicking it opens a menu with available probes, attenuators and current clamps.

Selecting a certain probe or current clamp will compensate the values along the axis of a graph, the readout in the cursor window and in meter displays.

An Arbitrary Waveform Generator can be used to generate standard as well as arbitrary signals. This chapter shows how use the arbitrary waveform generator with the standard signals and how to load arbitrary signals. An arbitrary signal can be any previously measured signal or software generated data.

The arbitrary waveform generator is controlled with the Arbitrary Waveform Generator control window, which is shown in figure 5.1. This window is opened by clicking the  **Arbitrary Waveform Generator** button on the File toolbar in the main window of the Multi Channel software. It can also be opened by double clicking the Generator in the object tree.

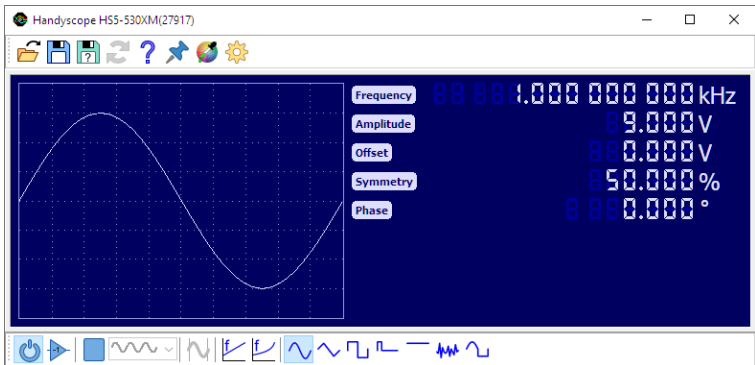


Figure 5.1: Arbitrary Waveform Generator control window.

5.1 Toolbars

The extended AWG control window has two fully configurable toolbars which are populated with buttons for easy access to frequently used functions and signal settings.

Both toolbars can be fully configured via the program settings of the software. Buttons can be added, removed or reordered.

In the default setup, the top toolbar contains buttons for general settings. The bottom toolbar contains buttons to control the generator and adjust signal settings.

























Icon	Action	Short description
	Open	Open settings/arbitrary data
	Save	Save settings/arbitrary data
	Save as	Save settings/arbitrary data in a new file
	Reload	Reload previously opened or saved settings/arbitrary data
	Help	Display help about the AWG
	Always on top	Make sure the AWG window is always visible
	Color scheme	Select a color scheme for the AWG window
	Settings	Display AWG settings
	Power	Switch AWG on or off
	Output invert	Switch AWG output invert on or off
	Start / Stop	Start / stop signal generation
	Generator mode	See 5.4
	Frequency mode	Signal or sample frequency, see 5.5.1
	Linear sweep	See 5.5.2
	Logarithmic sweep	See 5.5.2
	Sine	Generate a sine wave
	Triangle	Generate a triangular wave
	Square	Generate a square wave
	Pulse	Generate a pulse wave
	DC	Generate a DC level
	Noise	Generate noise
	Arbitrary	Generate an arbitrary signal, see 5.7

Table 5.1: Toolbar buttons

5.2 Power

The  **Power** button of the generator switches the generator hardware of the instrument on or off.

5.3 Output invert

The  **Output invert** button is used to invert the generated signal pattern. The signal is inverted around the selected offset level. Use it e.g. to create pulse signals with a negative pulse.

5.4 Generator mode

The following **Generator modes** are supported:









Mode	Description
 Continuous	When the start button is pressed, signal generation starts and continues until stopped by the user.
 Burst count	When the start button is pressed, signal generation starts. It stops automatically after 'burst count' periods. The output goes to the selected offset level.
 Gated	When the start button is pressed, signal generation starts, but the output remains at the selected offset level until external trigger input 2 becomes active. When external trigger input 2 becomes inactive again, the output goes to the selected offset level again.
 Gated periods	After the start button is pressed, signal generation starts at a new period, when external trigger input 2 becomes active. When external trigger input 2 becomes inactive again, the current period is finalized, signal generation stops and the output goes to the selected offset level.
 Gated period start	After the start button is pressed, signal generation starts at a new period, when external trigger input 2 becomes active. When external trigger input 2 becomes inactive again, signal generation stops and the output goes to the selected offset level.
 Gated period finish	When the start button is pressed, signal generation starts, but the output remains at the selected offset level until external trigger input 2 becomes active. When external trigger input 2 becomes inactive again, the current period is finalized and the output goes to the selected offset level.
 Gated run	After the start button is pressed, signal generation starts at a new period, when external trigger input 2 becomes active. When external trigger input 2 becomes inactive again, signal generation is paused and the output goes to the selected offset level.
 Gated run output	After the start button is pressed, signal generation starts at a new period, when external trigger input 2 becomes active. When external trigger input 2 becomes inactive again, signal generation pauses and the output remains at the last generated signal level.

Table 5.2: Generator modes

Note that the EXT2 input is active by default, unless pulled down. Generator mode can only be set when the generator is stopped.

5.5 Signal properties

Depending on the selected signal type and generator mode, various signal properties like frequency or amplitude can be set. To adjust a signal property, the property name label can be clicked. This opens a dialog in which the new value can be entered. This dialog accepts prefixes like u, m, k, M, etc.



Figure 5.2: Decreasing a digit

Properties can also be adjusted by clicking the digits or scrolling the mouse wheel when the mouse pointer is over a specific digit. Clicking the upper half of a digit will increase the value of that digit, while clicking the lower half of a digit decreases it. Digits that are currently off, can be enabled by clicking their upper half.

5.5.1 Frequency mode

When an arbitrary signal is selected, the **Frequency mode** of the Arbitrary waveform generator can be set. The following modes are supported:



Mode	Description
 Signal frequency	The frequency controls set the frequency at which the displayed signal will be repeated.
 Sample frequency	The frequency controls set the sample frequency at which the individual samples of the displayed signal will be generated.

Table 5.3: Frequency modes

With signal types sine, triangle, square and DC, the frequency mode is fixed to **Signal frequency**. With signal type noise the frequency mode is fixed to **Sample frequency**.

5.5.2 Sweep

The sweep function enables a linear or logarithmic continuous sweep with the selected signal type (sine, triangle or square). The sweep runs from **Frequency1** to **Frequency2**, where frequency1 is allowed to be

higher than frequency2. Optionally, the sweep can start at a different frequency by setting the **Start frequency** to a value between the two sweep frequencies. **Duration** determines the sweep duration.

The accuracy of the sweep is determined by sweep settings in the settings dialog. The minimum and maximum number of samples that are used for one period of the signal can be set, as well as the maximum amount of samples for the complete sweep. Higher values will give a more accurate sweep, but changes to sweep properties will take a bit longer to take effect.

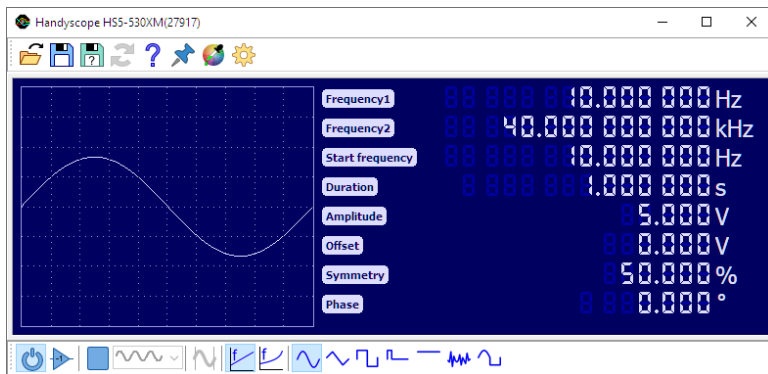


Figure 5.3: Linear sweep from 10 Hz to 40 kHz in 1 second.

Sweeping is only available on Handscope HS5 models and WiFiScope WS5 models with extended memory option XM.

5.6 Setfiles

All settings and arbitrary data of the AWG can be saved in setfiles with the **Save** and **Save as...** buttons on the toolbar. Setfiles can be loaded with the **Load...** toolbar button, or by dragging a setfile onto the AWG control window. See chapter D for more information about using files.

5.7 Arbitrary data

Besides some standard signals, the Arbitrary waveform generator can output arbitrary data. There are different ways of loading such data into the generator. Data can be loaded directly from an open source in the software, or from a file.

5.7.1 Loading arbitrary data from an open source

Data of every source in the Multi Channel software can be loaded directly into an AWG. This means that measured data, but also processed or generated data can be put into the AWG. There are **two ways** to do this.

- by dragging the source onto the AWG in the object tree.
- by dragging the source onto a AWG control window.

5.7.2 Loading arbitrary data from a file

Besides loading data from a source, it can also be loaded from a file. Currently, loading data from TiePie engineering TPS file, TiePie engineering WinSoft DAT file, Wave audio files and from specific ASCII files is supported. (WinSoft is old TiePie engineering scope software)

From a TiePie engineering TPS file, data can be read from each *AWG* or *Source* chunk in the file.

From a TiePie engineering WinSoft DAT file data will be read from the first channel in the file.

Data from Wave audio files can also be read into the AWG. If more than one channel (mono) is available in the file, only the first channel will be read. All uncompressed Wave audio files with a resolution of 8, 16, 32 or 64 bit are supported.

You can create an ASCII CSV file to be able to load your data into the function generator's memory. This file must contain a specific header line containing [SCOPE] and next various lines with waveform pattern data. Each waveform pattern line must start with 2 commas and then the sample value, its decimal separator must be a dot. The file name must have .DAT as extension.

The loaded pattern will be normalized. The highest absolute value in the pattern will be set to the selected amplitude. A value 0 will be equal to the selected offset value. All other values will be recalculated accordingly.

An example:

```
[SCOPE]
,,0
,,1
,,2.5
,,4
```

This file contains 4 lines of waveform pattern values, but the file may contain any number of data lines.

5.7.3 Data resampling

When loading data into an AWG, it is possible that the AWG does not support the data size of the loaded data. This can happen for example when the data is too big to fit into the memory of the AWG. The Handyscope HS3 has another limitation: the data size must be a power of two (2, 4, 8, 16, ..., 262144).

When it is not possible to set the data size of the AWG to the data's size, the data will be stretched or shrunk to fit exactly into the possible data size that is closest to the requested data size.

5.8 Hotkeys

The AWG can be controlled with several hotkeys, see appendix C for a complete list.

5.9 Oscilloscope triggering

The Arbitrary waveform generator in the WiFIScope WS5, Handyscope HS5 and Handyscope HS3 has three trigger output signals which are internally connected to the trigger system of the oscilloscope part of the instrument. In the oscilloscope they can be used as trigger source, to provide an exact synchronization between signal generation and measuring. The available trigger signals are:

Generator Start

This signal is generated when continuous generation or burst generation is started, either by the Start button or by an external trigger signal.


Generator New Period

This signal is generated when the whole buffer of the Arbitrary waveform generator has been processed and the generator starts at the beginning of the buffer again.

Generator Stop This signal is generated when continuous generation of burst generation is stopped by the the Stop button or burst generation is stopped because the required number of periods has been generated.



Sources are objects that generate data, like instrument channels and software generators.

To control the behavior of a Source, several properties and actions are available. These can be accessed through a popup menu which is shown when the Source is right clicked in the Object tree. The properties can also be accessed through its settings window which is shown when the Source is double clicked in the Object tree. To open the Object tree, click the  **Show Object tree** button.

Instrument channels are described in section 4.3.3 of this manual. This chapter will give an brief overview of Source objects in the Multi Channel software. Refer to the help in the software or the online website for complete and up to date information on the Sinks and how to control them.

6.1 Software generator



The **Software generator source** generates fully configurable standard signals like sine, block and triangle. All properties of these standard signals can be adjusted. The output signal of the Software generator source can be displayed in a graph or a meter and it can be used as source for I/Os and other sinks.


Typical applications for the Software generator are to compare measured signals with well known software generated signals or to test complex mathematical operations with well known signals before using them with real world measured signals.



I/Os are data processing objects that accept data (Input), process this data in a specific way, like e.g. adding, multiplying, filtering, etc. and then generate the processed data (Output). The inputs and outputs of the I/O block can either be time domain or frequency domain data.

The I/Os are categorized in the following categories:

- Math I/Os
- Filter I/Os
- Adjusting I/Os
- Miscellaneous I/Os
- Automotive I/Os
- Decoder I/Os

To control the behavior of an I/O, several properties and actions are available. These can be accessed through a popup menu which is shown when the I/O is right clicked in the Object tree. The properties can also be accessed through its settings window which is shown when the I/O is double clicked in the Object tree. To open the Object tree, click the  **Show Object tree** button.

This chapter will give an brief overview of I/O objects in the Multi Channel software. Refer to the help in the software or the online website for complete and up to date information on the I/Os and how to control them.

7.1 Math I/Os

The Math I/Os perform a basic mathematical operation on one or more signals.

7.1.1 Gain / Offset



The **Gain / Offset I/O** multiplies a signal with a constant gain factor and adds a constant offset. Both an input offset and an output offset are available. An input offset can be automatically removed using the Neutralize input offset action.

A typical application of the Gain / Offset I/O is converting sensor output signals.

7.1.2 Add/Subtract



The **Add / Subtract I/O** adds or subtracts data of two or more sources. Up to 32 sources can be added or subtracted with a Add / Subtract I/O.

A typical application of the Add / Subtract I/O is performing a differential measurement using two channels. When the high voltage and the low voltage of the differential signal are measured, the differential signal can be calculated by subtracting the low signal from the high signal.

7.1.3 Multiply/Divide



The **Multiply / Divide I/O** multiplies or divides data of two or more sources. Up to 32 sources can be connected to a Multiply / Divide I/O.

A typical application of the Multiply / Divide object is power measurement. When you measure the voltage over a load and the current through it, you can calculate the power by multiplying both measurements.

7.1.4 Sqrt



The **SQRT I/O** calculates the square root of each sample of the source's data.

The typical application of the SQRT I/O is to use it as part of larger, more complex mathematical operations.

7.1.5 ABS



The **ABS I/O** takes the absolute value of each sample of the source's data. The ABS operation does for every sample:

```
if sample < 0 then
  sample := -sample
```

The typical application of the ABS I/O is to use it as part of larger, more complex mathematical operations.

7.1.6 Differentiate



The **Differentiate I/O** differentiates the source's data. The output is proportional to rate of change of the input. For example, if a source has the unit V, the output of the DIFF will have unit V/s. The output range can be changed and fixed to user defined values.

The typical application of the Differentiate I/O is to use it as part of larger, more complex mathematical operations.

7.1.7 Integrate



The **Integrate I/O** integrates the source's data, it is the inverse of the Differentiate operation.

A typical application of the Integrate I/O is when measuring acceleration sensor signals. Integrating the acceleration signal will give a signal representing the speed. Integration of the speed will give the relative position.

7.1.8 Logarithm



The **Logarithm I/O** calculates the logarithm of the input data. The base number and a gain can be set.

A typical application of the Logarithm I/O is to create a vertical axis in dB in the oscilloscope or spectrum analyzer.

7.2 Filter I/Os

The Filter I/Os perform a filtering operation on a signal or consecutive signals.

7.2.1 Filter



The **Filter I/O** filters the source's data using a user definable software filter.

A typical application of the Filter I/O is to remove unwanted frequencies from a measurement, which may negatively affect further operations on the measured signal.

The filter type can be set and for each filter type, the used algorithm and order can be set. Depending on the selected filter type, the cut-off frequency or the center frequency and bandwidth can be set. Depending on the selected filter algorithm, additional settings can be available.

The Filter settings window also shows a graphical representation of the filter characteristics. A graph with gain and phase is shown as well as a Zero-Pole plot. The Phase graph can be switched between degrees and radians by clicking the axis legend. The frequency axis be switched between linear and logarithmic by clicking the axis legend.

7.2.2 Ideal Filter



The **Ideal filter I/O** filters the source's data using an ideal user definable software filter. It fully passes all wanted frequencies in the frequency spectrum and fully blocks all unwanted frequencies in the frequency spectrum. The edges of the filter are infinitely steep.

A typical application of the Ideal filter I/O is to remove unwanted frequencies from a measurement, which may negatively affect further operations on the measured signal.

7.2.3 Signal Cleaner



The **Signal cleaner I/O** cleans up a periodical signal by removing noise and other unwanted distortions from the signal.

A typical application of the Signal cleaner I/O is to remove noise from a measurement at a periodical signal.

7.2.4 Average



The **Average I/O** averages successive measurements. The average of these measurements is then placed on its output. The Average I/O object can work in two different modes: **Running average** (default) and **Average-of-n**.

Averaging is useful in situations when the signal of interest is periodic and (random) noise is present on top of it. By taking multiple measurements of the signal and averaging them, the signal to noise ratio will increase.

7.3 Adjusting I/Os

The Adjusting I/Os perform an adjusting operation on a signal or consecutive signals

7.3.1 Deskew



The **Deskew I/O** shifts the source's data in time.

A typical application of the Deskew I/O is to place two signals with a phase shift on top of each other in the scope screen.

An other typical application of the Deskew I/O is to remove a phase shift from a current signal, caused by a current probe, in order to perform correct power calculations.

7.3.2 Slice



The **Slice I/O** cuts a slice out of its input signal and removes the parts before and after the slice.

A typical application of the Slice I/O is to remove unwanted parts of a measurement. If a long measurement was performed to capture a specific phenomenon which is (much) shorter than the total measurement, the Slice I/O can be used to remove the parts that are not interesting. The original time info is kept. This makes processing and saving the interesting part much more efficient.

7.3.3 Limiter



The **Limiter I/O** limits or clips a signal to a certain range. The maximum and minimum level of the clip range can be set individually.

A typical application of the limiter is "cleaning up" digital signals. Set the clip range to a narrow window around the middle between the high level and the low level of the digital signal. Then use a Gain / Offset I/O to magnify the output signal of the limiter to the original magnitude.

7.3.4 Comparator



The **Comparator I/O** compares a signal to a certain reference value and indicates whether the signal is higher or lower than the reference value.

7.3.5 Resampler



The **Resampler I/O** resamples the source's data to decrease or increase the sample frequency (and record length with the same ratio) of a signal. This can be useful when several signals are sampled with a high sample frequency and record length but not all of them require this high speed and record length.

7.3.6 Window



The **Window I/O** applies a window function to its input signal. The window function changes the magnitude of the samples of in input signal, such that the ends smoothly taper to zero. Several different tapering functions are available.

A typical application of the Window I/O is to remove spectral leakage in a frequency spectrum using a Fourier Transform.

A second application is to shape signal bursts, e.g. to use in the Arbitrary Waveform Generator.

7.4 Miscellaneous I/Os

The Miscellaneous I/Os perform an other operation on a signal or consecutive signals:

7.4.1 Min / Max detector



The **Min / Max detector I/O** detects minimum or maximum values of the source's data. Every time the source of the Min / Max detector signals new data, the detector will compare each value in memory to the new source data and keep the smallest or largest value, depending on if minima or maxima are detected.

Typical applications of the Min / Max detector I/O are glitch capturing and capturing intermittent signal changes as well as creating envelope mode.

7.4.2 Data collector



The **Data collector I/O** is used when performing streaming measurements. During streaming measurements, the data arrives in consecutive chunks with a size equal to the instrument's record length. To form a continuous stream of data, these chunks must be appended to each other. The data collector does this job. It will fill its data with the arriving chunks of data.

7.4.3 Reference



The **Reference I/O** holds a reference copy of a signal. When the original signal changes, the reference copy remains unchanged.

Typical applications of the Reference I/O are to compare live signals with known-good signals or adjusting a system to a certain state.

7.5 Analyzing I/Os

The Analyzing I/Os analyze the input data and extract specific information from it.

7.5.1 Duty cycle



The **Duty cycle I/O** determines the duty cycle of a signal. The duty cycle is defined as the ratio between the time that a signal is "active" and the period. It is usually expressed as a percentage.

When the source data contains multiple cycles of a signal, the duty cycle will be determined for each cycle of the input signal. The output of the duty cycle I/O will then contain the progress of the duty cycle in the input data.

A typical application of the Duty cycle I/O is to analyze control signals of actuators that are duty cycle driven.

7.5.2 RMS

RMS The **RMS I/O** determines the True Root Mean Square value of a signal. For alternating electric current, RMS is equal to the value of the direct current that would produce the same average power dissipation in a resistive load.

Depending on the selected **Mode**, the RMS I/O returns one value over the total record, one value over all complete periods in the total record or one value per complete period in the record.

A typical application of the RMS I/O is logging RMS values to monitor e.g. mains voltage.

7.5.3 Maximum



The **Maximum I/O** determines the Maximum value of a signal. Depending on the selected **Mode**, the Maximum I/O returns one value over the total record, one value over all complete periods in the total record or one value per complete period in the record.

A typical application of the Maximum I/O is logging Maximum values to monitor e.g. mains voltage.

7.5.4 Minimum



The **Minimum I/O** determines the Minimum value of a signal. Depending on the selected **Mode**, the Minimum I/O returns one value over the total record, one value over all complete periods in the total record or one value per complete period in the record.

A typical application of the Minimum I/O is logging Minimum values to monitor e.g. mains voltage.

7.5.5 Maximum - Minimum



The **Maximum - Minimum I/O** determines the Maximum - Minimum value of a signal.

Depending on the selected **Mode**, the Maximum - Minimum I/O returns one value over the total record, one value over all complete periods in the total record or one value per complete period in the record.

A typical application of the Maximum - Minimum I/O is logging Maximum - Minimum values to monitor e.g. mains voltage.

7.5.6 Phase difference



The **Phase difference I/O** determines the phase difference between its two input signals.

The Phase difference I/O looks for the locations of all rising slopes in both input signals. It then determines the phase differences between corresponding rising slopes. The Phase difference I/O returns one value, which is the average phase difference between the measured signals.

7.5.7 FFT



The **FFT I/O** performs a spectral analysis of a signal. The FFT object converts a time domain signal to a frequency domain signal by means of a Fast Fourier Transform, an efficient algorithm to compute the Discrete Fourier Transform (DFT). The output of an FFT I/O can be connected to a graph with a frequency scale or an empty graph. The magnitude of each bin in the spectrum represents the amplitude of that frequency component.

Typical applications of the FFT I/O are to create a spectrum analyzer to examine the frequency response of a system or to examine the harmonic distortion in a system.

7.5.8 EMI



EMI The **EMI I/O** creates an environment that can be used for EMI pre-compliance testing. It sets up the instrument appropriately for performing the measurements and determines the frequency spectrum of the input signal, which is displayed in a graph.

The EMI I/O adds a new toolbar to the user interface of the software with all relevant settings. Additionally it gives access to two standard limit lines (EN61326-1:2006 / EN55011:2007) that can be added to the graph for easy comparison.

The EMI I/O can only be used in combination with an instrument with option E installed. And only channel one of an instrument with option E can be connected to an EMI I/O.

7.6 Automotive I/Os

The Automotive I/Os perform automotive specific operations on the data.

7.6.1 Crankshaft angle



In modern engines, usually a crankshaft sensor is present which generates a periodic signal with a number of cycles per revolution of the engine. The **Crankshaft Angle I/O** converts this signal into a crankshaft angle signal, showing the exact angle throughout the whole revolution.

This angle signal can be combined in a graph with an injector signal and / or an ignition signal. Relations between the crankshaft angle and the injection or ignition moments are then immediately visible.

7.6.2 RPM



In modern engines, usually a crankshaft sensor is present which generates a periodic signal with a number of cycles per revolution of the engine. The **RPM I/O** converts this signal into revolutions per minute. The engine speed can be calculated multiple times per revolution. Because of this, variations in the engine speed during a revolution can be seen.

7.6.3 EV charger control pilot analyzer



The process of AC charging an Electrical Vehicle (EV) using a Type 2 coupling (IEC 62196-2) by Mode 2 Electric Vehicle Supply Equipment (EVSE) can be monitored by analyzing the information that is transferred by the Control Pilot (CP) signal. The **EV Charger Control Pilot Analyzer I/O** can be used to analyze this signal and show the available amount of current that can be provided by the EVSE and the status of the charging process.

7.7 Decoder I/Os

The Decoder I/Os decode serial communication data from the input data and present the decoded messages.

7.7.1 Pulse decoder



The **Pulse decoder I/O** decodes the two measured signals from a quadrature encoder to a pulse count/position. A quadrature encoder (or incremental rotary encoder or relative rotary encoder) has two outputs (A and B) which are 90 degrees out of phase. That makes it possible to determine both position and direction of movement. Additionally the Pulse decoder I/O can also decode the Z reference output of an encoder, which gives one pulse per revolution. The Pulse decoder I/O has auto level detection for all signal lines. The output of the Pulse decoder can be shown in a graph and in a table sink.

7.7.2 I²C decoder



The **I²C decoder I/O** decodes analog data on an I²C bus to I²C data. Just measure the Clock and Data lines of the I²C bus with a scope and connect the measuring channels to the I²C decoder I/O. The I²C decoder I/O has auto level detection and auto bus speed detection. The output of the I²C decoder I/O can be shown in a Table sink.

7.7.3 UART / Serial decoder




The **UART / Serial decoder I/O** decodes analog data on a serial bus to serial data. It can be used to monitor and analyze UART, RS232, RS485, MIDI, DMX, LIN or other compatible serial buses. The UART / Serial decoder I/O has auto level detection and auto baud rate detection. The output of the UART / Serial decoder I/O can be shown in a Table sink.

7.7.4 CAN decoder




The **CAN decoder I/O** decodes analog data to CAN data. The source of the decoder can either be the differential CAN signal (preferred) or the CAN-high signal. The CAN decoder I/O has auto level detection. The decoded fields can be shown in a table sink.

7.7.5 SAE J1939 decoder


 The **J1939 decoder I/O** extracts SAE J1939 SPN values from CAN messages. J1939 is a high-level protocol that defines the communication between nodes or electronic control units on a vehicle CAN bus. It was developed by the Society of Automotive Engineers (SAE) for car and heavy duty truck applications. Currently the J1939 standard is used worldwide by heavy duty truck and machinery companies. J1939 is commonly used for communications throughout a vehicle or machine.

The J1939 decoder I/O has one input that must be connected to a source of CAN messages, typically a CAN decoder I/O (see 7.7.4), and can have multiple outputs. Each output contains the values of a single SPN (Suspect Parameter Number), selected from a database or loaded from a custom INI file. The data from these outputs can be used in graphs, meter sinks, table sinks and by other I/Os.

7.7.6 SPI decoder


 The **SPI decoder I/O** decodes analog data on an SPI bus to SPI data. Just measure the Clock and Data (MOSI or MISO) lines of the SPI bus with a scope and connect the measuring channels to the SPI decoder I/O. The SPI decoder I/O has auto level detection and auto bus speed detection. The output of the SPI decoder I/O can be shown in a Table sink.

7.7.7 Base section to index

 The **Base section to index I/O** "fills gaps" in data with the last known value. The typical application of the Base section to index I/O is to "fill" gaps in data retrieved from CAN messages using a CAN decoder I/O and J1939 decoder I/O.



Sinks are data processing objects that accept data (Input) and make it visible to the user, like e.g. graphs, tables and meters. The inputs of the sink can either be time domain or frequency domain data.

To control the behavior of a Sink, several properties and actions are available. These can be accessed through a popup menu which is shown when the Sink is right clicked in the Object tree. The properties can also be accessed through its settings window which is shown when the Sink is double clicked in the Object tree. To open the Object tree, click the  **Show Object tree** button.

This chapter will give a brief overview of Sink objects in the Multi Channel software. Refer to the help in the software or the website for complete and up to date information on the Sinks and how to control them.

8.1 Meter



The **Meter sink** performs various automatic measurements on the source's data. The measurement values can be displayed in segment displays (default), gauge displays, as well as text displays.

Multiple sources can be connected to the meter and per source, multiple measurements can be displayed, in individual displays. Sources with time domain data and sources with frequency domain data can be connected to a meter. For each connected source, a column with displays is added to the meter window, a caption at the top of the column indicates the source by its icon and name or alias.

8.2 Table



The **Table sink** shows values in a table. The table can display both numerical and string values. The values contained in the source's data are displayed in multiple columns. Columns can be hidden or shown via the dedicated toolbutton or via the popup menu that can be accessed by right-clicking the column header. The rows in the table can be sorted by the values in a certain column, by clicking on the column header. Data in the table can be exported to various common file formats.

8.3 Disk writer



The **Disk writer sink** exports measured data directly to disk. Single oscilloscope measurements as well as multiple consecutive oscilloscope measurements and also streaming data logger measurements can be written to disk. Data from one or more sources can be stored. When streaming measurements are written to disk, the newly arriving data will be appended to the previous data, forming one large block of data per stream. Data can be written in several common file formats. The files written by the Disk writer sink can not be loaded into the Multi Channel software.

8.4 Auto disk



The **Auto disk sink** stores the measured data and instrument settings of each measurement of an instrument directly to disk in a .TPS file. Data from all active channels of the connected instrument is stored. Both block mode oscilloscope measurements as well as streaming data logger measurements can be stored. When streaming measurements are stored, the newly arriving data will be written in a new file.

8.5 Sound



The **Sound sink** makes data audible. This can be both block mode data and streaming data. It can play data through one of the installed sound cards. If necessary, the data is resampled to fit the sound card's sample frequency.

One or two sources may be connected to each sound sink. If one source is connected, the data will be played in mono, otherwise in stereo. When playing stereo, the first source connected will be at the left and the second source at the right.

8.6 Compression test



The **Compression test sink** performs a dynamic compression test on a combustion engine. This quick and easy test gives a good impression if an engine has a compression problem without having to apply pressure sensors to the cylinders.

The starter current is measured during cranking and then used to calculate the relative compression of each cylinder. When the deviation of one or more cylinders is too big, this is indicated with a red bar. The Compression test sink uses a special output window, with a bar graph for each cylinder.

The Multi Channel software allows you to create and arrange graphs the way you want. New graphs can be created on the main screen of the Multi Channel software and can be moved to a separate window outside the main screen.

9.1 Creating new graphs

Creating new graphs is very easy: Simply click the **+** *New graph* quick function button. This will create a new graph in the in the area of the largest available graph, dividing that area in two. When the width of the area is larger than $\sqrt{2}$ times its height, the new graph will be created next to the existing graph. Otherwise, it will be created below it.

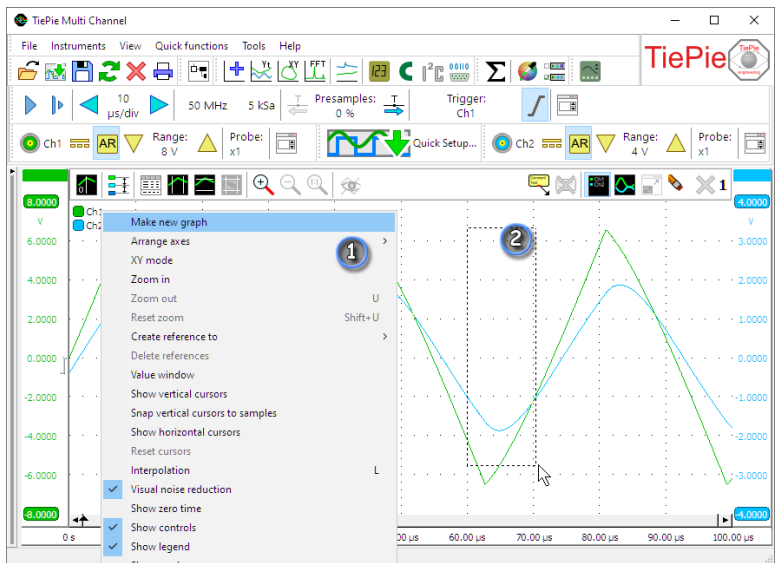


Figure 9.1: Create a new graph in two steps

It is also possible to create a graph at a self defined location. ① Right-click anywhere in a graph on the main screen and select **Make new graph** from the menu. ② After dragging a rectangle anywhere on the graph section, a new graph will be created on the selected position. See figure 9.1.

The size of the newly created graph can be adjusted by dragging the edges of the graph. When the mouse cursor is moved over the edges of a graph, the mouse cursor will change shape to indicate that the size of the graph can be changed.

9.2 Active graph

When more than one graph is available, one graph becomes the active graph, indicated by a blue border around the graph. Instruments, channels, sources and I/Os have actions that will add a line for that object in the active graph. The File menu contains an option to save the active graph as image.

Selecting a graph as active graph is done by clicking that graph with the mouse or by using hotkey **Shift** + **graph number** .

9.3 Axes

When a graph displays a signal, it has a horizontal axis or scale to indicate time or frequency information for that signal. It will also have a vertical axis or scale to indicate signal magnitude information. Each axis has text labels aligned with the grid lines of the graph, showing the corresponding magnitude, time or frequency value for that grid position. The range of the axis automatically adapts to the range of the signal data it belongs to.

When multiple signals are displayed, each signal will get its own vertical axis, the graph will have one horizontal axis that matches with the combined time or frequency ranges of all signals.

9.4 Vertical axes

A graph gets one vertical axis per displayed signal. This axis is divided in 8 divisions and gives information on the magnitude of the displayed signal and allows to control how and where the signal is displayed.

The axis can be located at the left hand side of the graph and at the right hand side of the graph. Axes of automatically added signals, e.g. when enabling a channel, will be placed at the left or right hand side, depending on the channel number of the instrument. Axes of manually added signals will be placed at the same side of the graph as where the signal source was dropped.

To change the location of an axis, use the mouse to grab it at its colored tab and drag it to the other side of the graph, **(B)** . In a similar way, an axis and its corresponding signal can also be moved to another graph, **(A)** . When keeping the **Ctrl** key pressed while moving an axis to another graph, the axis and its signal will be copied to the other graph.

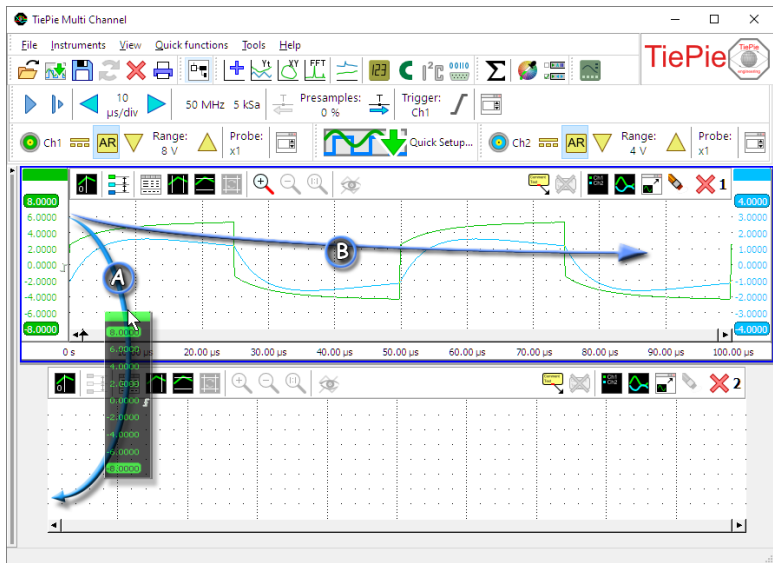


Figure 9.2: Axes can be moved to different targets.

9.4.1 Tabbing axes

When multiple signals are displayed, e.g. with large combined instruments, multiple axes will be added to the graph, taking up space, leaving less space for the actual signals. To reduce the space occupied by the axes, they can be tabbed. Right click the colored tab of one of the axes and check **Tabbed** in its popup menu. All axes at that side of the graph will now be placed on a tab. To untab them again, right click the colored tab of one of the axes and uncheck **Tabbed** in its popup menu.

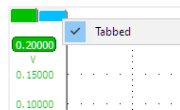


Figure 9.3: Putting axes on tabs

When the axes are tabbed, only the axis on the selected tab will be visible and can be used to position and resize the corresponding signal. To select another axis, click the tab of that axis, which will bring it to the front. That axis can now be controlled and its corresponding signal can be adjusted. Signals of tabbed axes can have different ranges, positions and sizes.

9.4.2 Adjusting displayed signals using an axis

The size and position of a displayed signal can be adjusted using the corresponding axis.

To change the position of a signal, simply grab the corresponding axis and drag it to the required position, see figure 9.4, the left axis. Use this to align signals to each other or to align them with grid lines.

To change the size of a shown signal, grab one of the end bars of the corresponding axis and drag it to the preferred position, see figure 9.4, the right axis. Use this to zoom in or out vertically on a signal.

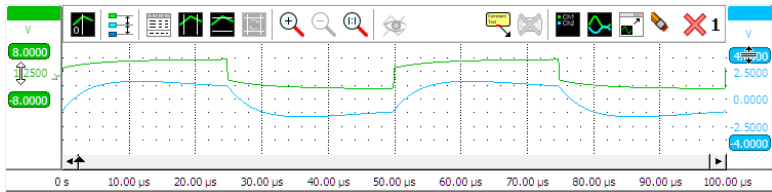


Figure 9.4: Adjusting vertical axes


It is also possible to adjust the position and size with the mouse wheel: when the mouse pointer is above the axis and the wheel is turned, the position is adjusted. When the **Ctrl** key is pressed during the turning of the wheel, the size is adjusted instead.

9.4.3 Show or hide a line

When a graph shows multiple signal lines, it may be convenient to temporarily hide one or more lines, for a better view on the other signal lines. When a signal is hidden from the graph, it remains being measured and all other settings remain unchanged. Right click the corresponding axis and select **Line visible** from the popup menu to toggle the visibility of the signal line.


When performing a streaming measurement with a high sample rate, hiding lines during the measurement will reduce the load on the computer. This may reduce the chance that the computer cannot keep up with the data stream, causing the stream to be aborted. When the measurement is ready, the line(s) can be made visible again, for signal analysis.

9.4.4 Automatically arranging axes

When multiple signals are displayed in a graph, they can be arranged such that signals do not overlap. The  **Arrange axes** button on the graph toolbar and the option **Arrange axes** in the graph's popup menu give the following modes:

Action	Description
Off	The Arrange mode is switched off. Signals are shown as how the user positions them. When a signal is added, it will be shown 1:1, potentially overlapping other signals.
1:1	All axes are shown unzoomed
Automatic	Each axis uses a whole number of divisions and axes are positioned without overlap. Axes can get different zoom factors.
Automatic with overlap	Each axis uses a whole number of divisions and axes are positioned with overlap. Axes can get different zoom factors.
1:N	All axes are evenly divided and they are positioned without overlap. All axes get the same zoom factor.
1:N with overlap	All axes are evenly divided and they are positioned with 50% overlap. All axes get the same zoom factor.
1:M	Each axis uses a whole number of divisions and axes are positioned without overlap. All axes get the same zoom factor.
1:M with overlap	Each axis uses a whole number of divisions and axes are positioned with overlap. All axes get the same zoom factor.

When a certain mode is selected and a line is added to the graph, all present signals will be repositioned according to the selected mode.

When a certain mode is selected the graph is zoomed in by dragging a rectangle with the mouse, pressing the  **Zoom reset** button will reset the original view corresponding to the selected mode.

When a mode is selected and the user then changes the position of one or more signals, the mode is switched to **Off**.

9.4.5 Merging axes

For easy comparison of two (or more) signals with the same unit, they can be placed on the same axis. When dragging one axis on another axis, the axes merge to a single axis. The colored tab of the axis will show the original colors of the merged axes. The range of the axis will be set to fit the ranges of the merged axes. All signals corresponding to the axis will be displayed using that axis. When changing position and/or scale, all corresponding signals will be adapted accordingly.

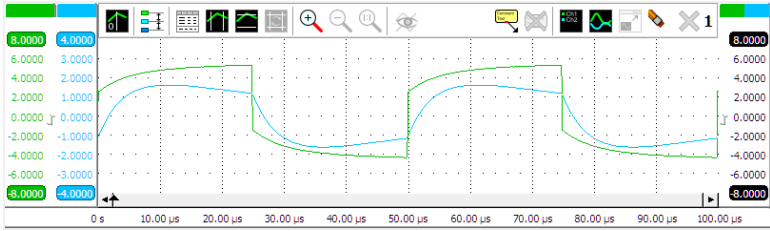


Figure 9.5: Two merged axes at the right

To extract a signal from a merged axis and give it its own axis again, right click the axis and select **Extract line** from the popup menu.

To delete a signal from a merged axis and no longer display it, right click the axis and select **Delete line** from the popup menu.

9.4.6 Axis type

A (vertical) axis can be set to two different types:

Linear: The data of the axis is displayed on a linear scale.

Logarithmic: The data of the axis is displayed on a logarithmic scale. 0 dB corresponds to 1 V.

Setting the axis type can be done by right clicking the axis and selecting **Axis type** from the menu that pops up.

9.4.7 Axis range

An axis has a range that follows the range of the data it is connected to. If the axis is connected to e.g. a channel that is set to an input range of -4 V to 4 V, the range of the axis will be -4 to 4 V too. When the range of the data changes (e.g. the channel is set to another input range), the axis range automatically changes accordingly.

In certain situations it may be useful to force the axis to keep a specified range, other than the range of the data. By right-clicking the axis and selecting **Set axis range...** a user defined axis range can be entered. A dialog appears in which a minimum and a maximum value can be entered. A **"Fixed"** checkbox, default checked, is also available to keep the axis fixed to the entered range. The axis will now remain in this fixed axis range, even if the range of the connected data changes.

To reset the axis range to follow the range of the data again, uncheck the **"Fixed"** check box. Now, when the range of the connected data changes, the axis range will follow the data range again.

9.4.8 Visible range

To adjust the size and position of a displayed signal, use the **Visible range** of an axis. The visible range of an axis has a maximum value and a minimum value that correspond to the top and the bottom of the axis. It defines which part of the axis range is visible in the graph. By setting the visible range smaller than the axis range, a signal is shown larger (zoom in vertically). By setting the visible range larger than the axis range, a signal is shown smaller (zoom out vertically).

When the visible range of an axis is adjusted and the range of the data changes, the visible range of the axis changes accordingly.

9.4.9 Clipping detection

When clipping detection is enabled in the settings, the software will place indicators in the graph and/or meter display when the signal is clipped due to being larger than the input range.

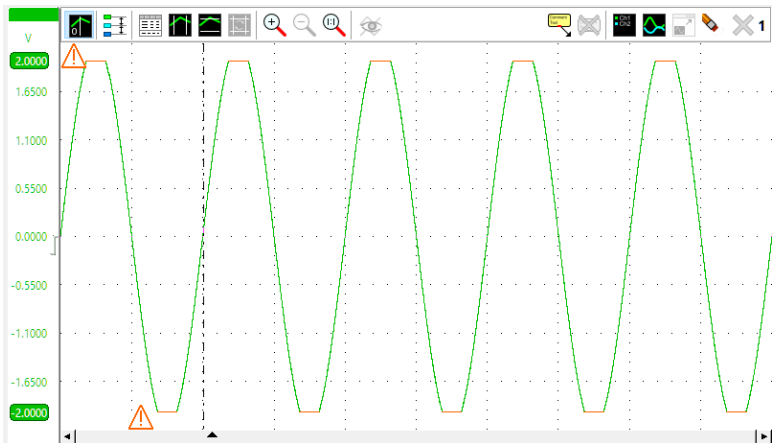


Figure 9.6: Clipping detected

When the clipping appears within the visible part of the signal, the clipping indicator will be placed next to the location of the first occurrence of the signal clipping. The clipped signal parts will be drawn in a different color. Clipping at both ends of the input range will be indicated separately, two indicators will then be placed.

When clipping occurs outside the visible part of the signal, the indicator(s) will be placed at the edge of the graph, closest to the location where the clipping occurs.

Hovering an indicator with the mouse will show a hint with more information.

When the displayed signal is processed by one or more I/Os, it may be no longer possible to detect whether the signal is clipped or not. In that case no indicators are shown.

The color of the clipping indicator and the clipped signal parts can be determined in the settings.

9.4.10 Show peaks and harmonics

Each signal line in a graph can show peaks in the signal by placing markers in the top part of the graph, with lines to the actual peak. The magnitude of the peak is shown in the marker, as well as a time stamp. When the graph displays frequency domain data, a frequency spectrum, it can also show the harmonics, with harmonics number, frequency and magnitude.

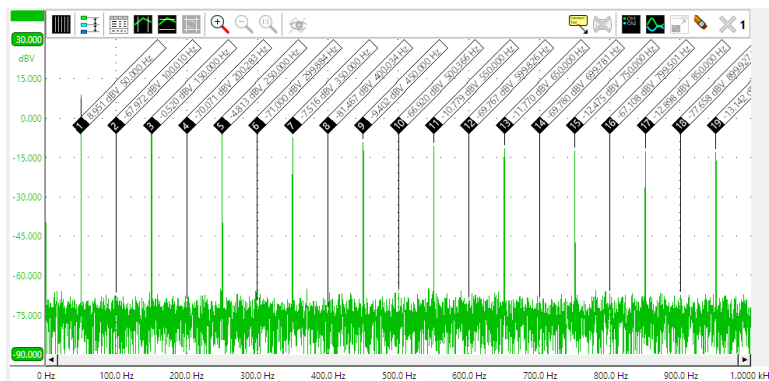


Figure 9.7: Show harmonics

Right-clicking a marker allows you to copy all relevant information of that marker to the clipboard. It also allows you to copy all relevant information of all markers to the clipboard.

Properties

To control the behavior of the graph markers, several properties are available. These can be accessed through a popup menu which is shown when the axis in the graph is right clicked. The properties can also be accessed through the line settings window which is shown via the popup menu.

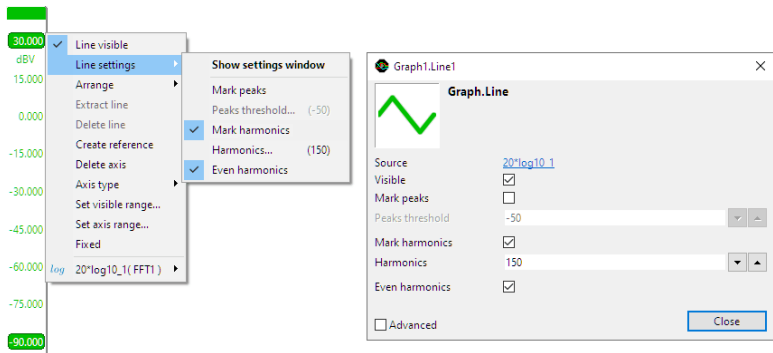


Figure 9.8: Line settings

Mark peaks

Enabling **“Mark peaks”** will place markers at the peaks in the signal. Any peak in the signal above **“Peaks threshold”** will be considered a peak.

Mark harmonics

When the graph contains frequency domain data, a frequency spectrum, the option to show harmonics becomes available. Enabling **“Mark harmonics”** will place markers at the harmonics in the frequency spectrum. The maximum amount of harmonics that is marked can be set using **“Harmonics”**. When Even harmonics is enabled, both uneven and even harmonics are marked, otherwise only uneven harmonics are marked.

9.5 Horizontal Axis

A graph gets one horizontal axis, independent of how many signals are displayed. This axis is divided in 10 divisions and gives information on the time or frequency of the displayed signal. When time domain data is connected (in an oscilloscope), the horizontal axis represents time. When frequency domain data is connected (in a spectrum analyzer), the horizontal axis represents frequency. The range of the horizontal axis automatically follows the range of the data it is connected to. When multiple signals are displayed, the axis range is set such that the ranges of all connected signals fit. The horizontal axis is always located at the bottom of the graph.



When the graph operates in XY mode it gets one or more horizontal axes that behave identically to the vertical axes. For more info on the horizontal axis in XY mode, refer to section 9.4.

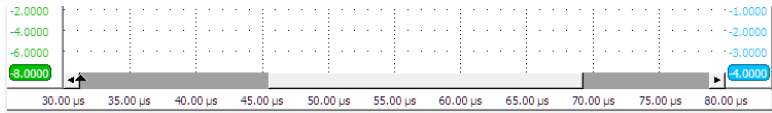


Figure 9.9: Horizontal axis and record view scrollbar

The horizontal axis is equipped with a scrollbar. The width of the scroll bar represents the total measured record, the width of the slider represents the visible part of the record. To change the horizontal size of a signal (magnify / stretch it), drag one of the edges of the slider either to the left or the right and watch the signal change corresponding to the slider width. To change the horizontal position of a signal, grab the slider and drag it to the preferred position or press the buttons at the sides of the scroll bar.

It is also possible to adjust the size and position with the mouse wheel: when the mouse pointer is above the scroll bar and the wheel is turned, the horizontal position is adjusted. When the **Ctrl** key is pressed during the turning of the wheel, the horizontal size is adjusted instead.

Double-clicking the slider, clicking the  **Reset zoom** button or pressing the hotkey **T** will return the horizontal axis to its default unzoomed state.

When zoomed in, the graph can also be panned horizontally by dragging the signal left or right, with the right mouse button pressed.

Changing the position or the size of the displayed signal(s) modifies the Visible range of an axis, see [9.5.2](#).

9.5.1 Axis range

An axis has a range that follows the range of the data it is connected to. If the axis is connected to e.g. a channel that is set to a time base of 1 ms/div, starting at $t = 0$, the range of the axis will be 0 to 10 ms. When the range of the data changes (e.g. the scope is set to another time base setting), the axis range automatically changes accordingly.

In certain situations it may be useful to force the axis to keep a specified range, other than the range of the data. By right-clicking the axis and selecting **Set axis range...** a user defined axis range can be entered. A dialog appears in which a minimum and a maximum value can be entered. A **"Fixed"** checkbox, default checked, is also available to keep the axis fixed to the entered range. The axis will now remain in this fixed axis range, even if the range of the connected data changes.

To reset the axis range to follow the range of the data again, uncheck the **"Fixed"** check box. Now, when the range of the connected data changes, the axis range will follow the data range again.

9.5.2 Visible range

To adjust the horizontal size and position of a displayed signal, use the **Visible range** of an axis. The visible range of an axis has a maximum value and a minimum value that correspond to the right and the left of the axis. It defines which part of the axis range is visible in the graph. By setting the visible range smaller than the axis range, a signal is shown larger (zoom in horizontally). It is not possible to set the visible range larger than the axis range.

When the visible range of an axis is adjusted and the range of the data changes, the visible range of the axis changes accordingly.

9.5.3 Follow source

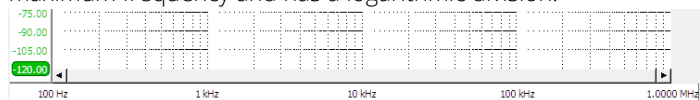
When a time domain graph contains a Data collector (see 7.4.2), it is possible, while the Data collector is being filled, to zoom in horizontally on the latest data and have the graph automatically keep the latest data visible. Choose the appropriate zoom factor in the graph and right-click the time base axis and select **Follow source** from the popup menu. This will show a sub menu with the possible sources to follow and an option not to follow a source.

9.5.4 Axis type

When a graph contains frequency domain data (spectrum analyzer), the horizontal frequency axis can be set to two different types:

Linear: The spectrum is shown on a linear frequency scale. The frequency axis runs from 0 to the maximum frequency and has a linear division.

Logarithmic: The spectrum is shown on a Logarithmic frequency scale. The frequency axis runs from the minimum frequency to the maximum frequency and has a logarithmic division.

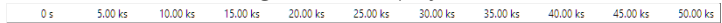


Setting the axis type can be done by clicking the **Linear axis** button or **Logarithmic axis** button or by right clicking the axis and selecting **Axis type** from the menu that pops up.

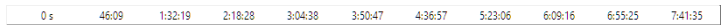
9.5.5 Time axis label style

The label style of a time axis can be set to three different styles:

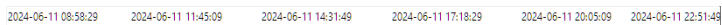
- **Seconds:** All labels along the axis display the time in seconds.



- **Days, hours, minutes and seconds:** The labels along the axis display the time in seconds. When the label time exceeds 1 minute, the time for that label will be displayed in minutes and seconds. When the label time exceeds 1 hour, the time for that label will be displayed in hours, minutes and seconds. When the label time exceeds 1 day, the time for that label will be displayed in days, hours, minutes and seconds.



- **Date and time:** The labels along the axis display the time in absolute date and time.



When the graph contains the output of a Crankshaft Angle I/O or phase cursors are enabled, the horizontal axis can also be set to a fourth style:

- **Degrees:** All labels show the angle in degrees or percent, depending on how the phase cursors are set.











To set the label style, right-click the horizontal axis and select the menu item *Label style*.














9.6 Graph toolbar

Each graph has its own toolbar to control how the graph shows signals.



Figure 9.10: Graph toolbar

-  Enable/disable a time=0 line in Yt mode
-  Toggle between linear and logarithmic frequency axis
-  Arrange the axes vertically
-  Enable/disable value window
-  Enable/disable vertical cursors
-  Enable/disable horizontal cursors
-  Reset horizontal and vertical cursors to 25% and 75% of screen
-  To zoom in, drag a rectangle in the graph

-  Undo the last zoom operation (**U**)
-  Zoom out to full scale, horizontally and vertically (**Shift** + **U**)
-  Show/hide reference signals
-  Update reference signal with live data
-  Add a comment to the graph
-  Remove all comments from the graph
-  Show legend
-  Yt or XY mode
-  Move graph to new window
-  Restore graph in main window
-  Delete all axes (**Ctrl** + **Del**)
-  Close this graph
-  1 Graph number

9.7 Zooming

If you want to view a small section of your measurement simply drag a rectangle using the mouse at the section you want to see in detail. Multiple mouse zooming operations are remembered and it is possible to unzoom with the Zoom out Undo zoom button or hotkey **U**. Fully resetting all horizontal and vertical axes is done with the Zoom reset Reset zoom button.







To zoom in **horizontal direction only** and leave the vertical zoom factor unchanged, press the **Ctrl** key and select a zooming area with the mouse. To zoom in **vertical direction only** and leave the horizontal zoom factor unchanged, press the **Shift** key and select a zooming area with the mouse.

Zooming in horizontal direction only is also possible using the mouse wheel. **Wheel up** causes the graph to zoom in horizontally, around the location where the mouse is positioned. **Wheel down** causes the graph to zoom out horizontally, around the location where the mouse is positioned.

When zoomed in, the graph can then be panned horizontally by dragging the signal left or right, with the right mouse button pressed.

9.7.1 Zooming using keyboard controls



Horizontal zooming and panning is also possible using hotkeys. The following hotkeys affect the horizontal zoom factor of the active graph:

Hotkey	Function
Ctrl + 	Zoom horizontally in from the left side of the graph, in fixed steps
Ctrl + 	Zoom horizontally out at the left side of the graph, in fixed steps
	Zoom horizontally in from the right side of the graph, in fixed steps
	Zoom horizontally out at the right side of the graph, in fixed steps
	Pan to the left, in fixed steps
	Pan to the right, in fixed steps
T	Reset to the total full record view


Zooming in and out in a graph affects the horizontal visible range and vertical visible range of the axes.

9.8 Automatic measurements using cursors

In each graph, vertical and horizontal cursors can be used to indicate certain parts of the graph. The cursors can be individually enabled by:

- using the  **Show vertical cursors** button on the toolbar
- using the  **Show horizontal cursors** button on the toolbar
- clicking **Show vertical cursors** in the graph's popup menu.
- clicking **Show horizontal cursors** in the graph's popup menu.

The cursors can also be used to measure values of interest indicated by the cursors. These measurements include e.g.: Minimum, Maximum, Top-Bottom, RMS, Mean, Variance, Standard deviation, Frequency.

Measurement results are shown in a special value window which is opened by clicking the  **Value window** button on the toolbar. When the value window is opened and the vertical and/or horizontal cursors are not switched on, the required cursors will be switched on, based on the default selected measurements.

Vertical cursors are waveform based, they can be positioned anywhere on the signal by dragging them with the mouse. Optionally, they can snap to sample positions, by selecting **Snap vertical cursors to samples** from the graph's popup menu. Two red lines in the time axis scroll bar represent the vertical cursor locations in the record. To position both vertical cursors simultaneously and keeping them at the same distance from each other, press the **Ctrl** key while dragging a vertical cursor.

Horizontal cursors are graph based, they can be positioned anywhere in the graph by dragging them with the mouse. To position both horizontal cursors simultaneously and keeping them at the same distance from each other, press the **Shift** key while dragging a horizontal cursor.

Due to zooming, cursors can become off screen. To position an off screen cursor, it can be grabbed at the edge of the screen. The mouse pointer will change shape to indicate that the cursor can be dragged. Off screen cursors can be positioned inside the screen again by clicking the **Cursor reset** button which will position all cursors at 25% and 75% of the visible screen.

A horizontal and vertical cursor can be positioned simultaneously by dragging the intersection of the two cursors. Using the **Ctrl** key and/or the **Shift** key, the other cursors can be positioned as well, maintaining their distance to each other.

9.8.1 Measurements

For clarity, not all measurements are enabled by default when the cursors are enabled. In the program settings (Settings) can be determined which measurements are default shown when the cursors are enabled. The cursor window will by default look like in the figure below.

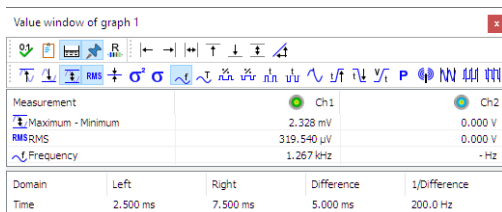


Figure 9.11: Cursor window


The measurements **Left**, **Right**, **Right-Left**, **Top**, **Bottom**, **Top-Bottom** and **Slope** require the vertical and/or horizontal cursors to be enabled. They use the cursor positions for their result.


All other measurements are calculated over a data range. When the vertical cursors are enabled, the data range is marked by the vertical cursors. When the vertical cursors are off, the data range equals the post samples.

Besides the calculated values in between the vertical cursors, the positions of the vertical cursors, as well as the difference between them, can be seen at the bottom of the cursor window. The **Show base info** button hides or shows this base info.

A user configurable virtual impedance is available that is used by the **P** Power measurement and the **dBm** measurement. Clicking the **Impedance button** will allow to set its value. Its default value is 600 Ω .

The measured values can be copied to the clipboard as text, by pressing the **Copy to clipboard** button.

When Always on top is switched on with the  **Always on top** button, the cursor window cannot be hidden under other windows of the Multi Channel software.

Enabling or disabling measurements can be done by clicking the corresponding measurement buttons on the toolbar, by clicking **Select measurements...** in the popup menu of the cursor window, or by clicking the  **Select measurements** button on the toolbar. A window will popup, in which the desired measurements can be selected.



For the frequency measurement, at least one cycle must be in between the vertical cursors. Furthermore, at least two rising slopes must be present in the signal.

9.8.2 Phase cursors

When examining phase shift between signals, or examining other periodic signals like e.g. crankshaft signals, special Phase cursors can be switched on in the graph. Right-click the graph and select **Phase cursors** from the popup menu to enable the phase cursors.

With these special cursors, the beginning and end of a cycle can be marked, which then is considered to be 0 to 360 degrees. It is also possible to mark that area as being 0 to 720 degrees, 0 to 100 % or some user defined range. The regular cursors can then be used to determine the angle of a specific position in the cycle. This angle is determined using linear interpolation between the two limits of the range.

Optionally, the horizontal axis can be set to show the angles instead of time. And overlays can be drawn over the graph to divide the selected range in e.g. 4 sections, for easy visual recognition.

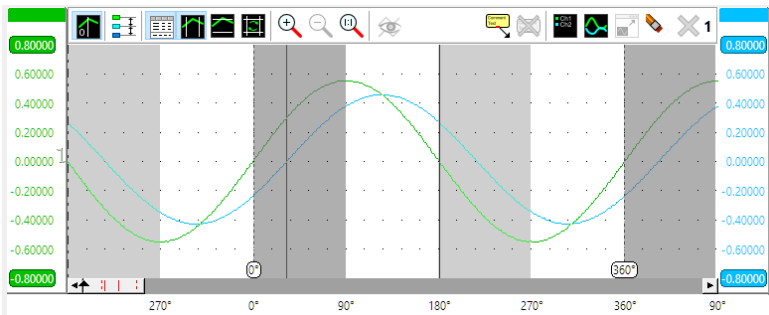


Figure 9.12: Phase cursors and background accenting

In figure 9.12, two sine waves with a phase shift are measured, where the phase cursors are placed on the beginning and end of one period of the Ch1 signal. The horizontal axis is set to show degrees. The left hand side normal cursor is placed at the zero crossing of the Ch2 signal.

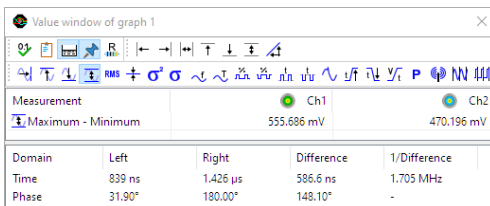


Figure 9.13: Phase cursors readout

The cursor readout shows the phase cursor information in the bottom row of the window. It shows that the phase angle for the left cursor, positioned at the start of a cycle on Ch2, is at 31.9 degrees, the phase angle between the two signals. The right cursor is placed half way the ch1 signal, at 180 degrees.

Phase cursor settings

Right-clicking the graph and selecting **Settings for the phase cursors...** will open the Phase cursor settings window.

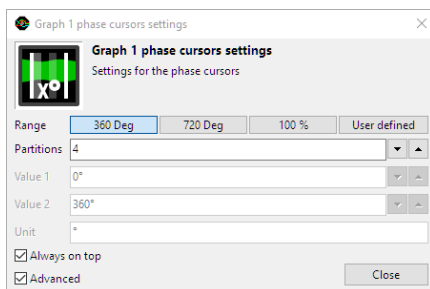


Figure 9.14: Phase cursors settings window


The following settings are available:


- **Range** determines the range of the section that is defined by the two special phase cursors:
 - **360 deg** the range covers 360 degrees, the left phase cursor marks 0° and the right phase cursor marks 360°.
 - **720 deg** the range covers 720 degrees, the left phase cursor marks 0° and the right phase cursor marks 720°.

- **100 %** the range covers 100 percent, the left phase cursor marks 0 % and the right phase cursor marks 100 %.
- **User defined** a user defined range can be set. This uses the settings Value 1, Value 2 and Unit.
- **Partitions** determines in how many partitions the range is divided. The time axis will get one label per partition and the partitions will be marked in the graph with a slightly different background color.
- **Value 1** sets the left hand side limit of the user defined range. This value can be higher than the right hand side limit.
- **Value 2** sets the right hand side limit of the user defined range. This value can be lower than the left hand side limit.
- **Unit** sets the unit for the user defined range.

9.9 References

You can create a *reference* to any signal that is displayed in a graph. A reference is a copy of a signal. By making such a copy and continuing the measurements, you will be able to see differences between the life signal and the reference. References can also be loaded with a "known good" signal of a device, to check life signals against this "known good" signal while troubleshooting a system.

You can create an automatic reference by choosing **Create reference to ►** in the popup menu of the graph or one of its axes. This creates a hidden Reference I/O containing the reference signal and which is added to the same axis as its original. When the reference is created, an  **Up-date reference** button is added to the graph's toolbar, which can be used to copy new data from the life signal into the reference.

Automatic references can be temporarily hidden from the graph. The  **Show references** button toggles the visibility of the available references.

When an automatic reference is deleted, the corresponding toolbar button(s) and the hidden Reference I/O are also removed.

It is also possible to create a manual Reference I/O via the object tree → I/Os → Miscellaneous → **Reference**. You have to connect a source to it manually, which automatically copies the data from the source to the Reference I/O. The reference I/O can then be shown in a graph or meter and can also function as source for other I/Os or sinks for further signal analysis. To update the data in the Reference I/O, right-click it in the object tree and select the **Copy** action from its popup menu.

Manual references do not get toolbar buttons and can not be hidden using the **Show references** button.

9.10 Using comments

Sometimes it is clarifying to add a text comment to a graph, to explain certain phenomena in the measured signals. An unlimited amount of comments can be added to each graph and an unlimited amount of arrows can be drawn from each comment to point at the signals.

A comment can be added by:

- using the  **Add comment** button on the toolbar
- clicking **Add comment** in the graph's popup menu.

The comments can be dragged to the desired position. Arrows can be created by "dragging" from the little square at the middle of the bottom of the comment to the desired position. You can move an existing arrow by dragging its point to a new position. To move a comment including its arrow, press the **Ctrl** key while dragging the comment. To delete an arrow, right-click on its point and click **Delete**.

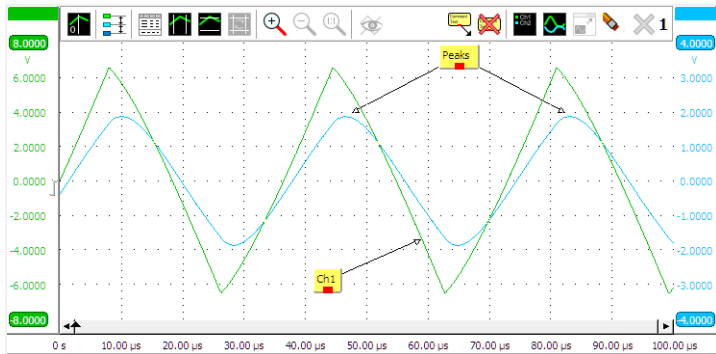



Figure 9.15: Text comments

To change the text in a comment, right-click the comment and select **Comment...** Comments can be resized by grabbing one of the edges with the mouse and dragging it to the required location.

To make a copy of a comment, right-click the comment and select **Clone** or **Clone with arrows** from its popup menu.

To delete a single comment, right-click the comment and select **Delete**. To delete all comments from a graph, click the  **Delete all comments** button in the graph toolbar.

When the scope is operating in streaming mode and a Data collector I/O is present in the graph, pressing the **space** bar will add a comment in the graph at the current position, to indicate a specific phenomenon that

happened at that time. The comment will contain the text **Trigger** and a serial number.

9.11 Drawing options

Several drawing options are available via the graph popup menu, which is shown by right-clicking a graph.

9.11.1 Interpolation

When the number of pixels in the graph is bigger than the number of displayed samples, two different ways of drawing the signal are available:

- **Interpolation** switched **on**: straight lines are drawn through the samples
- **Interpolation** switched **off**: a staircase line is drawn along the samples.

9.11.2 Visual noise reduction

The **Visual noise reduction** setting reduces some of the noise that may appear while drawing the measurements in the graphs. The result is smoother and thinner lines.

9.11.3 Legend

Using the option **Show legend** a legend of the shown signals is drawn in the upper left corner of the graph.

9.11.4 Markers

Using the option **Show markers**, each graph additionally gets small markers on the line. For each line a different symbol is used. These can be helpful in distinguishing signal lines.

The multimeter is an instrument that performs a measurement on a signal and then determines one or more specific properties of that signal and displays these as numeric values. Displaying can be done using a numerical display, with a gauge or with an text display.

The multimeter can be used to measure or monitor specific properties of a signal, like RMS value, frequency, maximum value etc. When using a gauge display, it can be very helpful when adjusting a circuit for a specific property, e.g. adjusting an offset to zero.

Appearance of the meter can be configured in the program settings.

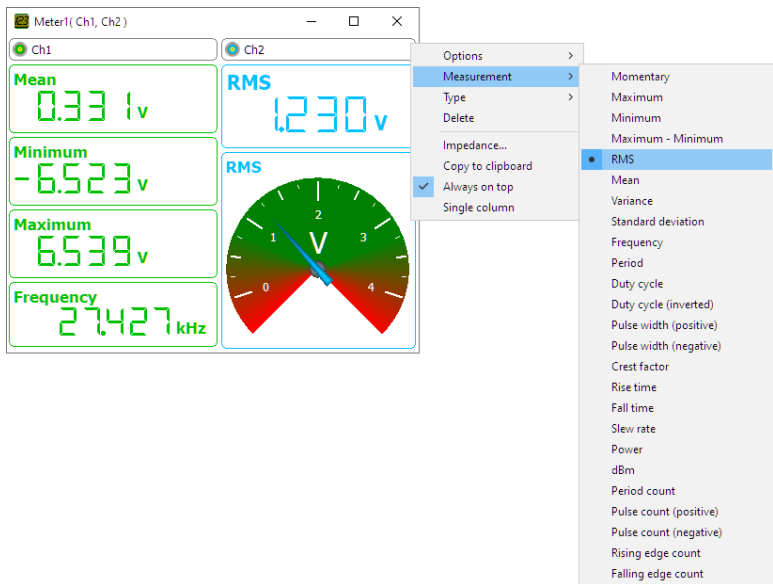


Figure 10.1: A meter example

The Multimeter uses the Meter sink (8.1).



























Multiple sources can be connected to the meter and per source, multiple measurements can be displayed, in individual displays. Sources with time domain data and sources with frequency domain data can be connected to a meter. For each connected source, a column with displays is added to the meter window, a caption at the top of the column indicates the

source by its icon and name or alias. Clicking that caption will open the settings window of that column's source. Right-clicking the caption will open a popup menu with additional actions and settings for the column.

Connecting a source can be done in two ways:

- drag the source onto the Meter in the object tree
- drag the source onto the window of the Meter

The following measurements are available:

 Momentary	 Pulse width (negative)
 Maximum	 Crest factor
 Minimum	 Rise time
 Maximum - Minimum	 Fall time
 Root Mean Square (RMS)	 Slew rate
 Mean	 Power
 Variance	 dBm
 Standard Deviation	 Period count
 Frequency	 Pulse count (positive)
 Period	 Pulse count (negative)
 Duty cycle	 Rising edge count
 Duty cycle (inverted)	 Falling edge count
 Pulse width (positive)	 Total Harmonic Distortion

More information on the measurements can be found in chapter [B](#).

The measurements are performed over a user specified data range of the associated source, see [B](#). In the program settings can be determined which measurements are default shown when creating a meter.

Settings for the meter are divided in three categories:

- Meter display properties and actions, see [10.2](#)
- Meter column actions, see [10.3](#)
- Meter properties and actions, see [10.4](#)

10.1 Clipping detection

When clipping detection is enabled in the settings, the software will place indicators in the graph and/or meter display when the signal is clipped due to being larger than the input range.

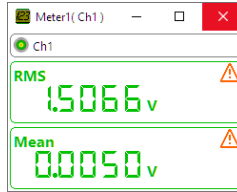


Figure 10.2: Clipping detected

When the displayed signal is processed by one or more I/Os, it may be no longer possible to detect whether the signal is clipped or not. In that case no indicators are shown.

The color of the clipping indicator can be determined in the settings.

10.2 Display properties and actions

To control the behavior of a display of the Meter, several properties are available. These can be accessed through a popup menu which is shown when the display is right clicked.

10.2.1 Color

When multiple sources are displayed in a meter, they can be distinguished from each other by their **Color**. Two color settings are available for a meter display.

- **As source**
The display will use the same color as its source.
- **User defined...**
The color of the display is user selected, using a color picker dialog. Any color can be used.

10.2.2 Digits

The **Digits** option determines how many digits are used to show the value on the segment display. Three different options are available:

- **Auto detect**
The segment display determines the optimal number of digits, based on the resolution of the value.

- **6**
The segment display uses six digits to display the value
- **User defined...**
The number of digits is user defined, using a number input dialog.

10.2.3 Decimals

The **Decimals** option determines how many decimal positions are used to show the value on the segment display. Three different options are available:

- **Auto detect**
The segment display determines the optimal number of decimals, based on the resolution of the value.
- **3**
The segment display uses three decimals to display the value
- **User defined...**
The number of decimals is user defined, using a number input dialog.

10.2.4 Auto Prefix

The option **Auto prefix** determines whether a prefix (μ m k M G etc.) is added to the displayed unit on a segment display. The displayed value is shown accordingly. Auto prefix is default on. When Digits and Decimals are set to Auto detect, Auto prefix will be fixed to on.

10.2.5 Range

The **Range** option determines the full scale range of a gauge display. Two options are available:

- **Auto ranging**
The gauge display sets the full scale range according to the output range of the associated source of the display.
- **User defined...**
The upper and lower limit of the gauge display range are user defined, using an input dialog.

10.2.6 Measurement

The property **Measurement** determines which type of measurement is performed on the data of the associated source of the display. A sub-menu allows to select from the available measurements.

10.2.7 Type

The property **Type** determines whether a Segment display with numbers, a Gauge display with a pointer or a Text display is used to show the value. When a Text display is selected, the measurement for that display is automatically set to Momentary.

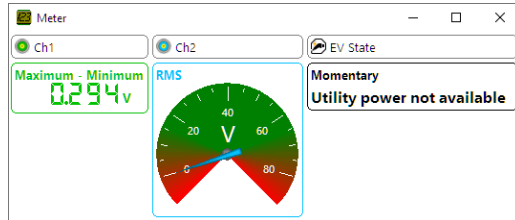


Figure 10.3: The different display types

10.2.8 Delete

The **Delete** action deletes the display.

10.3 Column actions

Some actions that are related to a column of a meter are available. These can be accessed through a popup menu which is shown when the column caption is right clicked.

10.3.1 Add Measurement

The action **Add measurement** adds a display with a specified measurement to the column of displays for the associated source. The measurement to add can be selected from a sub menu.

10.3.2 Disconnect source

To remove a source from a meter, the **Disconnect source** action can be used. This will remove all displays and caption for that source from the meter window.

10.4 Meter properties and actions

To control the behavior of the Meter sink, several properties are available. These can be accessed through a popup menu which is shown when the sink is right clicked in the object tree.

10.4.1 Single column

When measurements for multiple sources are displayed in the Meter sink, each source gets its own column with displays. When many sources are connected, the total Meter display can become very wide. In that case it can be more convenient to set the Meter to **Single column**, where all sources are shown in a single column, below each other.

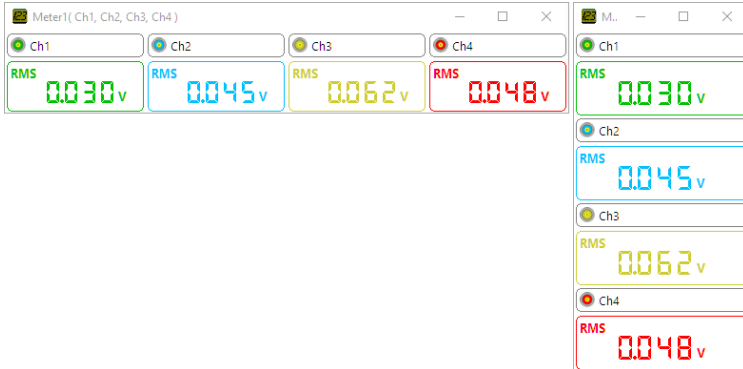


Figure 10.4: Multiple columns and single column

10.4.2 Measurement data range

The data range over which the measurements are calculated can be set to one of the following settings:

- **All samples:** Use both pre samples and post samples to perform the measurements calculations.
- **Post samples only:** Use only the post samples to perform the measurements calculations. (Default)
- **Pre samples only:** Use only the pre samples to perform the measurements calculations.

10.4.3 Log to disk

The meter can log its readings to a Comma Separated Values file to disk. When enabled, a dialog is shown in which the destination folder for the CSV files can be selected. A separate file for each connected source, with date and time stamp of the starting moment of the measurement as file name, as well as the source's name.

Each line in the file will contain a time stamp and the values of the different displays for that source, separated by commas. Each time the meter gets new data, a new line is added to the file(s).

10.4.4 Stop disk logging

When disk logging is no longer required, **Stop disk logging** is used to close the created file(s) and make them accessible to other applications.

10.4.5 Copy to clipboard

The **Copy to clipboard** action copies the values of all displays including layout to the Windows clipboard, from where it can be pasted in various applications. The Copy to clipboard action is available in the popup menu.

10.4.6 Always on top

When the **Always on top** property is switched on, the Meter window cannot be hidden under other windows of the Multi Channel software. The Always on top property is available in the popup menu.

In the Multi Channel software different ways to store measurements are available:

- Save a single scope / spectrum analyzer measurement, [11.1](#)
- Save multiple scope / spectrum analyzer measurements, [11.2](#)
- Save continuous data logger measurements, [11.3](#)
- Export single measurements to common, third party files, [11.4](#)
- Export multiple measurements to common, third party files, [11.5](#)
- Save a measurement to a graphical image, [11.6](#)

11.1 Saving a single scope/spectrum analyzer measurement

The Multi Channel software can save oscilloscope and spectrum analyzer measurements to TPS and TPO files. Refer to chapter [D.1](#) and [D.2](#) for more information on these file types.

11.1.1 Saving to a TPS file

There are different ways to save a measurement to a file:

- To save your complete setup in a **Desktop** type of file, create a TPS file with the **Save as...** item in the **File** menu. This creates a TPS file with all instrument settings, instrument data and all used objects in the setup.
- To save instrument settings and optionally data to an **Instrument** type of file, right-click the instrument in the object tree and click **Save as...** This creates a TPS file with just instrument settings and instrument data.
- To save function generator settings and optionally data to a **Function Generator** type of file, right-click the function generator in the object tree and click **Save as...** or use the save button on the function generator control window. This creates a TPS file with just generator settings and generator data.

11.1.2 Saving objects to a TPO file

In the object tree one or more sources, I/Os and sinks can be selected. You can save the selected object(s) to a TPO file by right-clicking on the

selection and choosing **Save as...** from the popup menu. The objects will be saved with all connections they have in common.

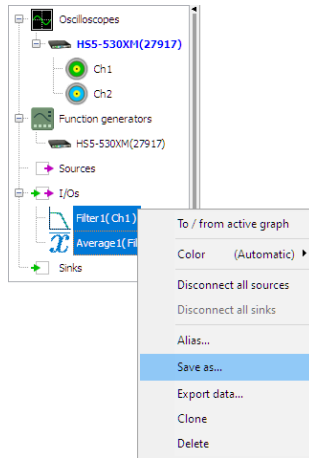


Figure 11.1: Saving two objects

Saving groups of objects can be handy when you want to use the group with the same settings in another setup, or just want to quickly create multiple equal groups of objects to perform the same kind of processing on several sources. After saving the file, you can open it multiple times to add the objects in the file to the setup.

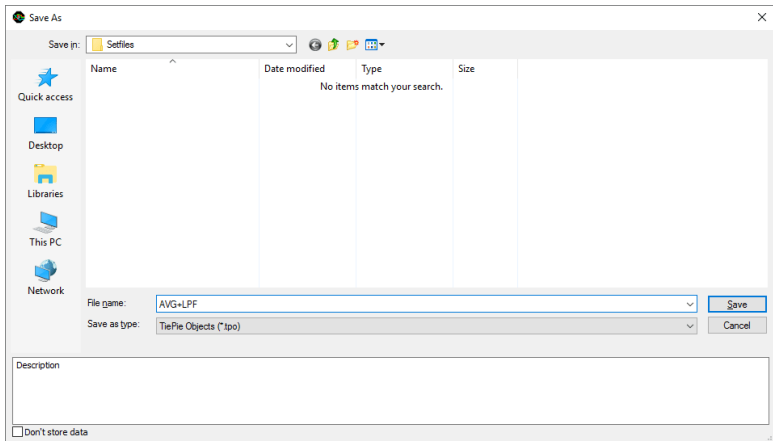


Figure 11.2: TPO Save as dialog.

You can enter a description and choose to only store settings and no data by checking the **"Don't store data"** checkbox.

11.2 Saving multiple scope/spectrum analyzer measurements

The **AutoDisk** function can be used to store all measurements of an oscilloscope to TiePie engineering TPS files. Besides the measured data, the TPS files contain the instrument settings. Each measurement is stored in a separate file.

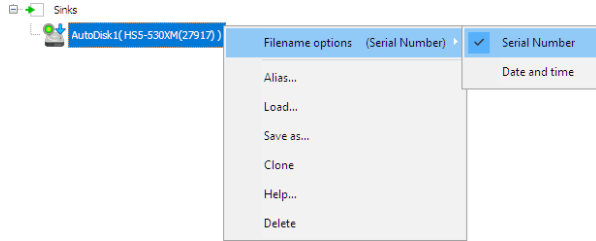


Figure 11.3: Setting up the Auto Disk

Logging measurements of an instrument using AutoDisk can be initiated in two ways:

- Right-click the instrument in the object tree and choose **AutoDisk...** from the popup menu. An AutoDisk sink is created and the instrument is connected to it.
- Click **Sinks** in the object tree, create an AutoDisk sink and drag the instrument on it.

A save dialog will appear in which you can enter a file name. After choosing the file name, the auto disk function is ready to store all measurements and a window as depicted below will appear, displaying the progress. A serial number and/or time stamp is appended to the file name for each measurement. Right-click the AutoDisk sink and select **Filename options** to determine which are added. The serial number starts at 000000 and is incremented after each measurement. All measurements of the instrument associated with the AutoDisk function will be stored until the **Stop** button is pressed.

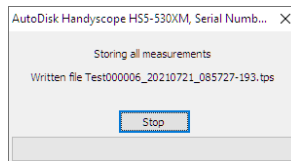


Figure 11.4: Auto disk progress dialog



The files saved with the AutoDisk only contain the instrument settings and data, no settings and/or data of other objects that were present during the measurements.

You can review the stored TPS files in different ways:

- Use **File** → **Load...** in the main menu
- Use **Load...** in the instrument's menu
- Drag the file from a Windows explorer window onto the Multi Channel software
- Double click the file in a Windows explorer window



When you load a TPS file with recorded data directly in the instrument, the data is processed by the processing blocks that are connected to the instrument. This can be very useful when you want to perform some kind of processing on auto disk measurements.

11.3 Save continuous data logger measurements

Continuous logging can be done using the **Signal logging** Quick Setups (see also 2.4). These Quick Setups setup the instrument and software for a specific logging operation with just a few mouse clicks. After selecting and loading the Quick Setup, all that needs to be done is starting the measurement. The measurement will then run for an unlimited amount of time, until it is manually stopped. All measured values are stored to disk.

These Quick Setups use **Data collector I/Os** (7.4.2). These can be set to save their content to disk in a TiePie engineering **TPO** file when they are full, then clear them self and start collecting again.

When performing continuous streaming measurements, Data collector I/Os can be used to collect and "glue" the streaming data and then display it and post process it.

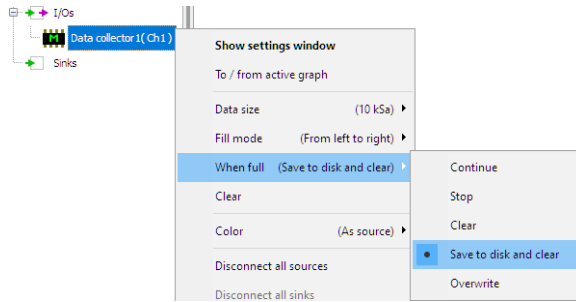


Figure 11.5: Setting up the Data collector

When for the Data collector property **When full** the option **Save to disk and clear** is selected, each time the Data collector is full, a .TPO file containing the Data collector is saved to disk. The Data collector is then cleared and starts collecting again. The file name for the .TPO files starts with the date and time of starting the measurement, followed by a serial number that increments each time the Data collector is full. Additionally an index file (.TPIDX) is stored keeping track of all saved .TPO files that belong to a certain measurement. The name of the index file contains the date and time of starting the measurement. The location where the files are stored can be set in the application settings. When the setup contains multiple Data collector I/Os, all saving to disk, they will all be included in the same index file. When the measurement is stopped, the final, incomplete chunk is saved as well.

When the measurement is started directly from a data logging Quick Setup and more than one chunk was saved, a dialog will appear when the measurement is stopped, asking to load and show all saved data.

When the measurement was setup manually without using a Quick Setup, the index file can be opened in the Multi Channel software, using **File** → **Load Data collector index file**. This will load a block of two, three or four of the saved Data collector .TPO files, depending on the data length of the saved Data collectors. The loaded Data collectors are shown together in a graph. When the original setup contained multiple Data collector I/Os, the .TPO files with matching times will be loaded and displayed together. Additionally, a special toolbar is shown that allows to navigate through the available .TPO files that belong to the specific measurement.

- ⏪ Go to the first part of the measurement
- ⏩ Go to the previous block of the measurement
- ⏴ Go to the previous part of the measurement
- ⏵ Go to the next part of the measurement
- ⏶ Go to the next block of the measurement
- ⏷ Go to the last part of the measurement
- ✖ Close the index file and remove all loaded Data collectors

Clicking the indicator label on the toolbar allows to jump to a specific part of the measurement. A dialog will be shown in which the number of the required part can be entered.

It is also possible to manually load all .TPO files into the Multi Channel software. This will create Data collectors with the saved data. When the Data collectors are then all shown in one graph, the various parts will be displayed after each other, giving an overview of the total measurement. Note that depending on the amount of Data collectors and their data size, this can be very memory consuming.

11.4 Exporting single measurements

Data of all sources can be exported. This means that data of all stand-alone sources like the instruments, and channels, but also outputs of other objects like I/Os can be saved to file.

The data displayed in a graph can also be saved to a file. All data of all visible lines will be exported. When cursors are switched on, only the data in the section between the two cursors will be exported.

Exporting data of sources, I/Os or graphs can be done by selecting the desired objects and choosing **Export data...** from its popup menu.

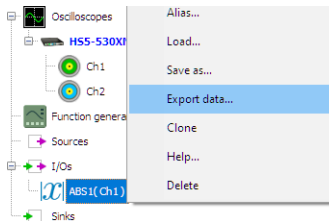


Figure 11.6: Exporting data.

A standard save dialog will popup, which is extended with options for the selected file format. Below, the save dialog is displayed with the options for saving binary files.

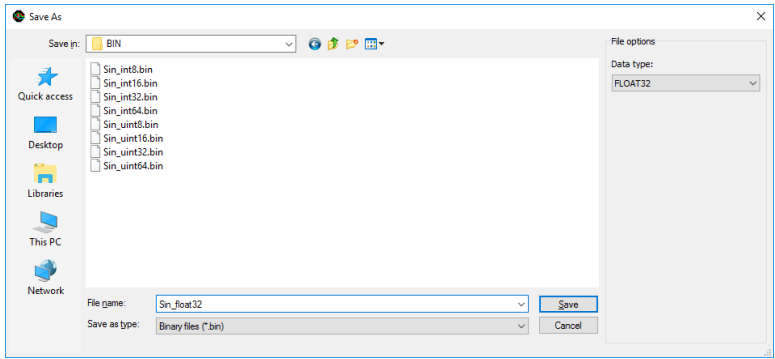


Figure 11.7: Exporting options for Binary files.

With the file type combo box, the desired file type can be selected.

- Binary (raw) (.bin)
- Comma Separated Values ASCII (.csv)
- JSON files (.json)
- Matlab (.mat) data (version 6)
- Wave audio (.wav)

The list of available file types depends on the selected sources that are to be saved. For example, most formats only support one time base. If sources with different time bases are selected, a separate file per source will be exported. The file name of the exported file will then also contain the name of the corresponding source.

11.5 Exporting multiple measurements

With the **Disk writer** sink object you can store the data of any source to several common, third party file formats. Supported file formats are:

- Binary (raw) (.bin)
- Comma Separated Values ASCII (.csv)
- JSON files (.json)
- Matlab (.mat) data (version 6)
- Wave audio (.wav)



These files can only be saved by the Multi Channel software, the software can not load these files.

Both block mode measurements (oscilloscope / spectrum analyzer) and streaming data logger measurements can be exported using the Disk writer. This way, very long measurements can be performed and store them to disk. Files with a size up to 4 GB can be written.

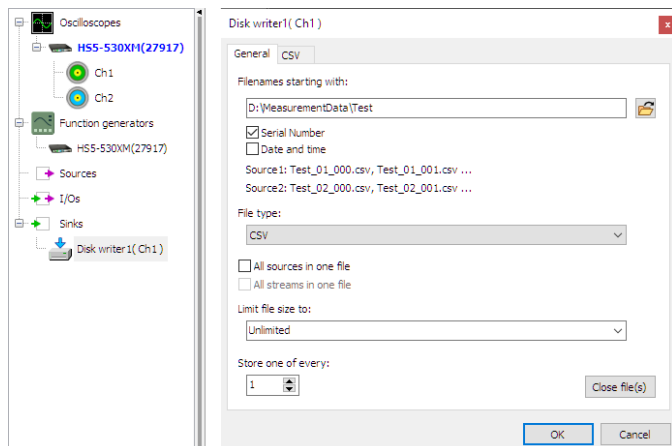


Figure 11.8: Setting up the Disk writer

To start exporting logging, first create a disk writer object by right clicking Sinks in the object tree and choosing the disk writer. A new disk writer object will now be created. By right-clicking the disk writer, a popup menu will show in which several settings can be changed. Entering a file name in the settings window of the writer is required.

When all settings are correct, you can connect the source(s) of which you want to store the data by dragging it/them onto the writer. If you want to store all channels of an instrument, you can drag the instrument instead of the single channels. After the objects are connected, all data will be recorded.



You can use several disk writer objects at the same time to store data in different file formats or locations simultaneously.

The Disk writer has several settings that change its behavior. They can be accessed via its settings window that is shown when the Disk writer is double clicked or via the popup menu of the Disk writer.

11.6 Saving a measurement to an image

To be able to use data displayed in a graph in reports or presentations or other applications, graphs can be saved as images. The image that is saved resembles the graph as it is shown on the screen, but is not an exact screen shot. The way the image is drawn is fully configurable. When cursors are used in a graph, the image will also contain the cursors and a table with the cursor readouts. Comment memos are saved to the image as well. The image can be saved in various common graphic file formats.

To save an image of a graph, it is recommended to pause the measurements first.

Then activate the desired graph by clicking in it with the mouse or by using the graph selection hotkey **Shift** + **graph number**. In case of multiple graphs, a blue border will be present around the active graph.

Once the required graph is selected, open the **File** menu and click **Save image....** This will open a save dialog with image saving options.

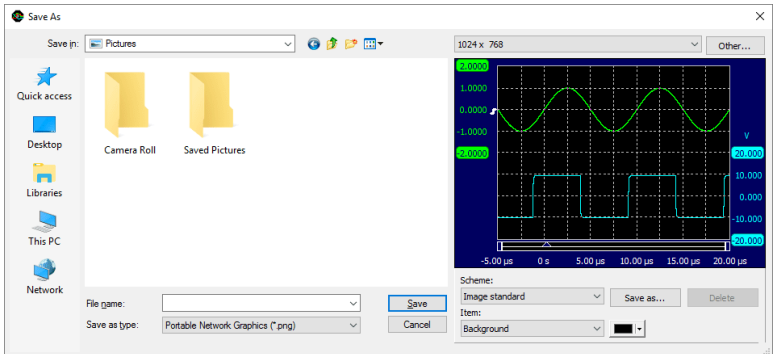


Figure 11.9: Saving an image



The example that is shown in the Save dialog is an example image, it is not the actual image that will be saved. When the image is actually saved using the **Save** button, the data of that moment is used. If the scope was still measuring, this will be different data than is shown in the example.

11.6.1 Saving an image from the command line

It is also possible to save an image file using command line parameters. Refer to the chapter **E** on command line options for more information.

11.6.2 Supported image file formats

The desired image file type can be selected in the file type combo box. Several different image file formats are available:

- Portable Network Graphics (*.PNG)
- Bitmap (*.BMP)
- Graphic Interchange Format (*.GIF)
- Enhanced Meta File (*.EMF)
- Joint Photographic Experts Group (*.JPG)

The PNG format is most suitable image format to save a graph as image. Using the JPG format will lead to loss of contrast and color information.

11.6.3 Image dimensions

When saving an image, several predefined dimensions are available:

- 320 x 200
- 640 x 480
- 800 x 600
- 1024 x 768
- 1280 x 1024
- 1600 x 1200

It is also possible to enter user defined dimensions, by clicking the **Other...** button. This will open the Image size dialog in which the required dimensions can be entered. The entered image dimensions will be added to the list of available image dimensions.

11.6.4 Image scheme

When saving an image, various parts of the image can be customized. These settings are combined in an image scheme. Several default schemes can be selected from. It is also possible to define own schemes, which can be saved for later use.

A TPS, TPO or WinSoft file can be loaded in several ways.

12.1 Loading a whole file

The common way to load a file is to use the **Load...** entry in the **File** menu. By default, the whole file is loaded when you press **“Open”**. Dragging and dropping a file on the main form of the application will also load the whole file.

12.2 Loading just a part of a file

The open dialog of the Multi Channel software is extended with a tree view. This tree view represents the contents of TPS and TPO files. The parts of the file that can be opened, are highlighted in the tree.

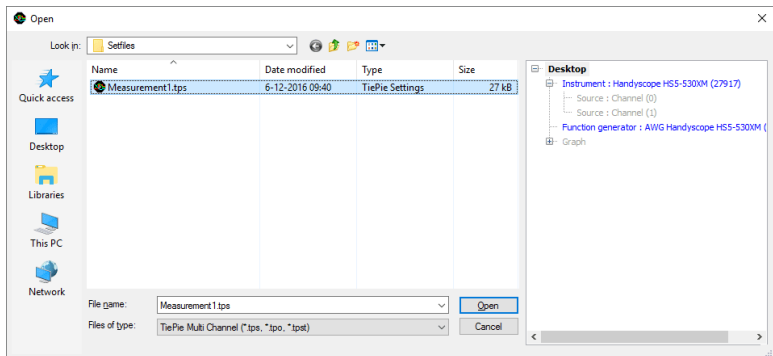


Figure 12.1: File Open dialog.

In the displayed open dialog, the selected file is a Desktop type TPS file, containing a complete setup. By default, the Desktop node in the tree is selected and the total file is loaded when you press **“Open”**.

As you can see in figure 12.1, the instrument and function generator are highlighted as well. They can be loaded separately, leaving the rest of your setup intact. To load the instrument settings from the file, click the Instrument node in the tree and press **“Open”**. If multiple instruments have been opened by the software, you will be asked to select which instrument(s) should load from the file. Loading a function generator part from a file works the same.

12.3 Loading into selected objects

In the object tree on the main form of the application, one or more objects of the same type can be selected. When you right-click on a selection of objects and choose **Load...** from the popup menu, the earlier mentioned open dialog will appear, but now only compatible parts from the file can be opened. The compatible items in the file are highlighted in the tree.

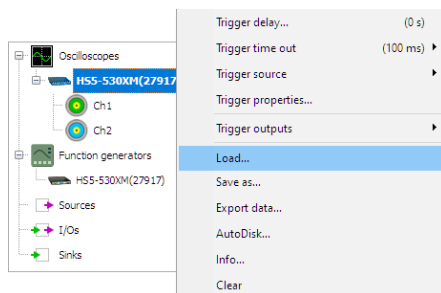


Figure 12.2: Instrument popup menu.

For example, when using the **Load...** function of an instrument on the same file used earlier, only the instrument part may be opened. It is selected automatically, because it is the first item in the file that can be opened by the instrument. Just press **“Open”** to load the instrument settings and data into the instrument. The rest of the setup is left intact.

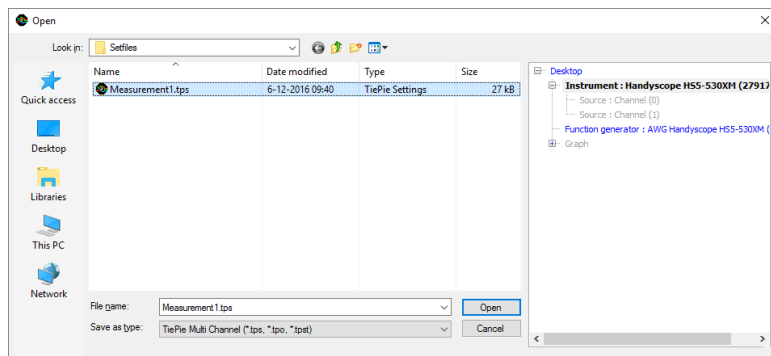


Figure 12.3: Load File into instrument.

Number input fields support the use of prefixes, constants and functions.

A.1 Prefixes

Valid prefixes that can be used in number input fields are:

Prefix	Name	Value
f	femto	10^{-15}
p	pico	10^{-12}
n	nano	10^{-9}
μ or u	micro	10^{-6}
m	milli	10^{-3}
k	kilo	10^3
M	mega	10^6
G	giga	10^9
T	tera	10^{12}
P	peta	10^{15}

Prefixes are case sensitive!

A.2 Constants

Supported constants are:

Constant	Value
pi	3.141592638
eulersnumber or en	2.718281828

Constants are not case sensitive.



























A.3 Functions

Supported functions are:

Function	Description
<code>sin(Arg1)</code>	returns the Sine of angle Arg1 in degrees
<code>cos(Arg1)</code>	returns the Cosine of angle Arg1 in degrees
<code>tan(Arg1)</code>	returns the Tangent of angle Arg1 in degrees
<code>arcsin(Arg1)</code>	returns the angle in degrees of the ArcSine of Arg1
<code>arccos(Arg1)</code>	returns the angle in degrees of the ArcCosine of Arg1
<code>arctan(Arg1)</code>	returns the angle in degrees of the ArcTangent of Arg1
<code>arctanxy(Arg1,Arg2)</code>	returns the angle in degrees of the ArcTangent of Arg2 / Arg1
<code>degtorad(Arg1)</code>	converts angle Arg1 in degrees to an angle in radians
<code>radtodeg(Arg1)</code>	converts angle Arg1 in radians to an angle in degrees
<code>sqr(Arg1)</code>	returns the square of Arg1
<code>sqrt(Arg1)</code>	returns the square root of Arg1
<code>ln(Arg1)</code>	returns the natural logarithm of Arg1
<code>log10(Arg1)</code>	returns the ten base logarithm of Arg1: $_{10}\log \text{Arg1}$
<code>log2(Arg1)</code>	returns the two base logarithm of Arg1: $_{2}\log \text{Arg1}$
<code>logn(Arg1,Arg2)</code>	returns the Arg1 base logarithm of Arg2: $_{\text{Arg1}}\log \text{Arg2}$
<code>exp(Arg1)</code>	returns e (euler's number) to the power Arg1: e^{Arg1}
<code>power(Arg1,Arg2)</code>	returns Arg1 to the power Arg2: $\text{Arg1}^{\text{Arg2}}$
<code>ceil(Arg1)</code>	returns the nearest integer equal to or greater than Arg1
<code>floor(Arg1)</code>	returns the nearest integer equal to or less than Arg1
<code>round(Arg1)</code>	rounds Arg1
<code>trunc(Arg1)</code>	returns the integer part of the floating point Arg1
<code>mod(Arg1,Arg2)</code>	returns the remainder of Arg1 divided by Arg2
<code>div(Arg1,Arg2)</code>	returns the whole number result of Arg1 divided by Arg2


Functions are not case sensitive.

In the cursor window of each graph and in meter objects, different measurements can be selected. These measurements include:

 Momentary	 Pulse width (negative)
 Maximum	 Crest factor
 Minimum	 Rise time
 Maximum - Minimum	 Fall time
 Root Mean Square (RMS)	 Slew rate
 Mean	 Power
 Variance	 dBm
 Standard Deviation	 Period count
 Frequency	 Pulse count (positive)
 Period	 Pulse count (negative)
 Duty cycle	 Rising edge count
 Duty cycle (inverted)	 Falling edge count
 Pulse width (positive)	 Total Harmonic Distortion


The measurements are calculated over a **sample range**. When using the cursors, the sample range is equal to the samples in between the left and right cursor. In a meter object, the sample range is user definable.


In the cursor window the following measurements are also possible:

 Left	 Top
 Right	 Bottom
 Right - Left	 Top - Bottom
 Slope	


B.1 Short description of the measurements

This section gives a short description of the standard measurements. Most measurements are very straight forward, but are explained for clarity. Some measurements are explained with a formula. In these formulas **m** corresponds to the index of the first sample in the sample range and **n** is the last index. **N** is the number of samples in the sample range and is equal to $n - m + 1$. The i^{th} sample is denoted as x_i .

 **Momentary** gives a momentary value of the measured signal. It is the last, or right-most value in the sample range.


 **Maximum** is the highest value in the sample range.

 **Minimum** is the lowest value in the sample range.

 **Maximum - Minimum**, also known as *peak-peak*, is the highest value in the sample range minus the lowest value.

RMS **RMS** is equal to the square root of the mean of the squares of all samples in the sample range.

$$RMS = \sqrt{\frac{1}{N} \sum_{i=m}^n x_i^2}$$

 **Mean** is the mean value of all samples in the sample range.


$$Mean = \bar{x} = \frac{1}{N} \sum_{i=m}^n x_i$$

σ^2 **Variance** is a measure of how values are distributed around the mean value.


$$Variance = \frac{1}{N} \sum_{i=m}^n (x_i - \bar{x})^2$$

σ **Standard deviation** (σ) is equal to the square root of the variance. The standard deviation is equal to the RMS value for signals with a zero mean value (AC signals).


$$\sigma = \sqrt{Variance}$$


 **Frequency** determines the frequency of a time based signal. The frequency is determined by searching rising slopes in a signal and measuring the time between them.


For a correct measurement, at least two rising slopes must be present in the sample range.

 **Period** determines the period time of a time based signal. The period time is determined by searching the rising slopes in a signal and measuring the time between them.


For a correct measurement, at least two rising slopes must be present in the sample range.

 **Duty cycle** is defined as the ratio between the time that a signal is **higher** than half the amplitude and the period. It is expressed as a percentage.


 **Duty cycle (inverted)** is defined as the ratio between the time that a signal is **lower** than half the amplitude and the period. It is expressed as a percentage.

 **Pulse width (positive)** determines the width of rising pulses in a signal. It uses the mid level crossings in the signal to determine the width of pulses.

For a correct measurement, at least two mid level crossings must be present in the sample range. When multiple pulses occur in the sample range, the average width of all detected pulses is shown.

 **Pulse width (negative)** determines the width of falling pulses in a signal. It uses the mid level crossings in the signal to determine the width of pulses.


For a correct measurement, at least two mid level crossings must be present in the sample range. When multiple pulses occur in the sample range, the average width of all detected pulses is shown.


 The **Crest factor** is equal to the peak amplitude of a waveform divided by the RMS value.


$$C = \frac{\max(|x|)}{x_{RMS}}$$

The Crest factor can be used to get an idea of the quality of a signal. A signal with more peaks will have a higher Crest factor. The following table lists Crest factors for some ideal standard signals.

Signal type	Crest factor
Sine	$\sqrt{2} \approx 1.41$
Triangle	$\sqrt{3} \approx 1.73$
Block	1
DC	1

 The **Rise time** is the time it takes for the signal to rise from 10% to 90% of its top-bottom value. The first rising slope in the sample range is used.

 The **Fall time** is the time it takes for the signal to fall from 90% to 10% of its top-bottom value. The first falling slope in the sample range is used.

 **Slew rate** is defined as the change of voltage per unit of time and is expressed in V/s. The first edge in the sample range is used.

P **Power** shows the amount of electrical power that is dissipated in a virtual reference resistor by the measured signal. The following formula is used, in which an adjustable virtual reference resistor R_{ref} is used to convert the measured voltage to power. The default value of R_{ref} is 600Ω

$$Power = \frac{x_{RMS}^2}{R_{ref}}$$



dBm is the power ratio in decibels of the measured power referenced to one milliwatt. The following formula is used, in which a reference resistor R_{ref} of 600Ω is used to convert the measured voltage to power.

$$dBm = 10 \cdot \log \frac{1000 \cdot x_{RMS}^2}{R_{ref}}$$



Period count uses the mid level crossings in the signal to determine the number of periods or pulses in a signal.

For a correct measurement, at least three mid level crossings must be present in the sample range.



Pulse width (positive) determines the width of rising pulses in a signal. It uses the mid level crossings in the signal to determine the width of pulses.

For a correct measurement, at least two mid level crossings must be present in the sample range. When multiple pulses occur in the sample range, the average width of all detected pulses is shown.



Pulse width (negative) determines the width of falling pulses in a signal. It uses the mid level crossings in the signal to determine the width of pulses.

For a correct measurement, at least two mid level crossings must be present in the sample range. When multiple pulses occur in the sample range, the average width of all detected pulses is shown.



Rising edge count determines the number of rising edges in a signal. It uses the mid level crossings in the signal to determine the amount of edges.

For a correct measurement, at least two mid level crossings must be present in the sample range.



Falling edge count determines the number of falling edges in a signal. It uses the mid level crossings in the signal to determine the amount of edges.

For a correct measurement, at least two mid level crossings must be present in the sample range.




Total Harmonic Distortion (THD) is defined as the ratio between the power of the harmonic frequencies above the base frequency and the power of the base frequency. This ratio is displayed in dB. It is a measure of the distortion in a signal.


The THD is calculated using the following formula:


$$THD = \frac{\sqrt{V_2^2 + V_3^2 + V_4^2 + \dots + V_n^2}}{V_1}$$


where V_1 is the RMS amplitude of the base frequency and V_2 to V_n are the RMS amplitudes of each higher harmonic.


The THD measurement can only be used on frequency spectrum signals.


 **Left** can be used in the cursor window. It is the value of the signal at the position of the left side cursor.


 **Right** can be used in the cursor window. It is the value of the signal at the position of the right side cursor.

 **Right - Left** can be used in the cursor window. It is the difference between the value of the signal at the position of the right and left cursor.

 **Top** can be used in the cursor window. It is the value of the signal at the position of the top cursor.


 **Bottom** can be used in the cursor window. It is the value of the signal at the position of the bottom cursor.

 **Top - Bottom** can be used in the cursor window. It is the difference between the value of the signal at the position of the top and bottom cursor.

 The **Slope** of a signal is the change of magnitude divided by the elapsed time. The measurement slope determines the slope in a signal between the left and the right cursor.

$$Slope = \frac{x_{right} - x_{left}}{t_{right} - t_{left}}$$

B.2 Reference resistor

A user configurable virtual impedance is available that is used by the **P** Power measurement and the  dBm measurement. Its default value is 600Ω.

Clicking the  **Impedance** button in the cursor readout window will allow to set its value.

Right-clicking a Meter display and selecting **Impedance...** from the popup menu allows to set the impedance value for this Meter. Any display in this Meter that will show Power or dBm, will use the selected impedance value.



For the most frequently used functions, hotkeys have been included to allow faster and easier control of the Multi Channel software.

Common

F1	Context sensitive help
Shift + (1 ... 0)	Select graph 1 through 10
Ctrl + (1 ... 0)	Select instrument 1 through 10
Alt + S	Search instruments

Per instrument

F3 / F4	Decrease / increase sample frequency
F11 / F12	Decrease / increase record length
S	Start / Stop
O	Start O neshot / Stop
0	Trigger timeout = 0
1	Trigger timeout = 1 sec.
W	Trigger timeout = infinite
Shift + ← / →	Change pre-/post samples ratio
space	Directly force a trigger
Q	Perform a q uick auto setup
Ctrl + Q	Perform a q uick auto setup and keep following the signal

Per channel

F5 / F6	Decrease / increase input range
R	Autorangeing on / off
F7 / F8	Decrease / increase Trigger level
[/]	Decrease / increase Trigger hysteresis
A	AC-coupling
D	DC-coupling

The mentioned channel hotkeys by default apply to *channel 1*. If the hotkeys are used with the keys mentioned in table C.1, the hotkey applies to the indicated *channel*, of the *active* instrument.

-	Ch. 1
Ctrl	Ch. 2
Shift	Ch. 3
Ctrl + Shift	Ch. 4
Alt	Ch. 5
Alt + Ctrl	Ch. 6
Alt + Shift	Ch. 7
Alt + Ctrl + Shift	Ch. 8

Table C.1: channel selection hotkeys

Per graph

L	Interpolation
U	Zoom out
Ctrl + DEL	Clear graph
Shift + DEL	Delete graph
← / →	Move scroll bar slider
Ctrl + ← / →	Move left edge of the scroll bar slider
↑ / ↓	Move right edge of the scroll bar slider
T	Full record view
X	XY-mode
Y	Yt-mode

Per Arbitrary Waveform Generator

F	Enter the frequency
A	Enter the amplitude
O	Enter the offset
S	Enter the symmetry
P	Enter the phase
B	Enter the burst count
D	Enter the sweep duration

The Multi Channel software uses various file types to load and store data and settings. This chapter gives a short description of the TiePie engineering file types used in the Multi Channel software. For information about other file types to which data can be exported, refer to section [D.5](#).

D.1 Multi Channel TPS files

TiePie Settings (TPS) files are used to store all kinds of settings as well as data. The files can contain instrument settings, graph settings, function generator settings, etc. They can be used to save a (complicated) measurement setup for later use. The files can also contain measured data, making it possible to examine the measured data at a different location or at a later time.

There are three kinds of TPS files which can all include data:

TPS type	Contains
Desktop	all settings of the instruments, function generators, objects, graphs, etc.
Instrument	settings of one instrument
Function Generator	settings of one function generator

When selecting a TPS file in the open dialog, you can see its type in the tree view at the right side of the dialog. The root node in the tree view displays the file type.

Desktop TPS files can be loaded as a whole, but you can also load just parts from a file. See section [12.1](#) about loading files for details.

D.2 Multi Channel TPO files

TiePie Objects (TPO) files are used to store one or more objects with or without their data. The objects can be sources, I/Os and sinks. When multiple objects are stored in a TPO file, the connections between these objects are stored as well. This makes it possible to save e.g. a mathematical operation using multiple objects to a file for later use or to load it multiple times if you want to perform the same operation on multiple sources.

The file structure of TPO files is very similar to that of TPS files. The objects contained in TPO files can also be found in TPS files and can be loaded from both file types the same way. See the sections about loading and saving objects for details.

D.3 Multi Channel TPIDX files

TiePie InDeX (TPIDX) files are created by a Data collector I/O when it is logging to disk.

When for the Data collector setting **When full** the option **Save to disk and clear** is selected, each time the Data collector is full, a .TPO file containing the Data collector is saved to disk. The Data collector is then cleared and starts collecting again. Additionally an index file is stored keeping track of all saved .TPO files that belong to a certain measurement. The name of the index file contains the date and time of starting the measurement.

The index file can be opened in the Multi Channel software, using **File** → **Load Data collector index file**. This loads a selection of the TPO files and displays them, as well as a navigation toolbar to view the other files of that measurement.

D.4 WinSoft files

For backward compatibility, the Multi Channel software can open files created by the WinSoft measuring software. Both settings and data can be loaded. The following file types are supported:

WinSoft type	Description
DAT	Binary or ASCII data file. Must be accompanied by a .GEG settings file.
SET	WinSoft settings. Can contain instrument and function generator settings.

When a WinSoft file is opened via the **file** menu or *dragged and dropped* onto the main form, the current setup is closed and replaced by the setup in the file. When a WinSoft file is opened via the **popup menu** of an instrument or function generator, the settings of that device are loaded, but the rest of the setup is left intact.

D.5 Exporting to other file types

It is possible to export data to the following file formats:

- Binary (raw) (.bin)
- Comma Separated Values ASCII (.csv)
- JSON files (.json)
- Matlab (.mat) data (version 6)
- Wave audio (.wav)

Refer to the online help file that comes with the software for a complete list of the file formats and a description of their options.

The Multi Channel software can be started with command line parameters. Currently the following options are available:

Loading setfiles

-d setfile[.tps]	Load desktop from setfile
-sa setfile[.tps]	Load first scope settings from setfile into all scopes
-s<n> setfile[.tps]	Load first scope settings from setfile into the n^{th} scope
-ga setfile[.tps]	Load first AWG settings from setfile into all AWGs
-g<n> setfile[.tps]	Load first AWG settings from setfile into the n^{th} AWG
-l setfile[.tps]	Load setfile. Auto detect file type

Saving files

-SaveAll setfile[.tps][,NoData]	Save desktop to setfile
-SaveScp setfile[.tps] [,Index=1][,NoData]	Save scope number index to setfile
-SaveGen setfile[.tps] [,Index=1][,NoData]	Save generator number index to setfile
-SaveImg filename (.png/.bmp/.gif/.emf/.jpg) [,Index=1][,w=800][,h=600] [,Scheme=Image standard]	Save screenshot of graph number index to filename

Window size and position

-MWS FULL	Maximize main window
-MWS w,h	Set main window w idth and h eight
-MWS l,t,w,h	Set main window at position l eft, t op with w idth and h eight

Size and position values may be specified in pixels or a percentage of the primary monitor size (add a % sign to the value).

When **Allow multiple instances** is switched **on** in the Multi Channel software, each launch of the Multi Channel software will open a new instance. Any command line parameter is used in that instance. When **Allow multiple instances** is switched **off** (default), each launch will activate the cur-

rent instance. Any command line parameter will be passed on to the current instance and executed there. Several command line options work best when **Allow multiple instances** is switched off.

The following applies to the command line parameters:

- command line options are case-insensitive
- values between () are mandatory
- values between [] are optional
- values after = are the defaults.
- if the option string contains spaces, it must be escaped with quotes
- multiple command line options can be used in a single call

E.1 Batch converting multiple saved measurements

When batch converting multiple saved measurements to other file formats or creating images from them, each time starting the Multi Channel software and after the conversion/creation closing the software again, can be time consuming. A command is added that will start the software without creating the user interface, perform the required task and then close the software again.

-console	run the Multi Channel software without showing its user interface
----------	---

Example To make the Multi Channel software appear full screen on the right monitor of a dual monitor system, use: `-MWS 100%,0,100%,100%`

Index

- I²C decoder, 60
- ABS, 52
- Active instrument, 31
- Add, 52
- Alias, 22
- Arbitrary data, 45
- ASCII files, 99, 118
- Auto disk, 64
- Auto-ranging, 40
- AutoDisk, 95
- Automatic measurements, 78
- Average, 54
- AWG, 41
- Base section to index, 61
- Basic measurements, 11
- Binary files, 99, 118
- CAN decoder, 15, 60
- Channel, 38
 - Coupling, 39
 - Input range, 40
 - Input type, 39
 - Probe, 40
 - Toolbar, 29
- Channel toolbar, 29
- Cloning objects, 19
- Comments, 83
- Comparator, 55
- Compression test, 64
- Configuring objects, 18
- Connecting objects, 19
- Control Pilot, 59
- Coupling, 39
 - AC, 39
 - DC, 39
- Crankshaft angle, 59
- Creating objects, 18
- Crest factor, 110
- CSV files, 99, 118
- Cursors, 78
- Data collector, 14, 56
- Data logger, 13
- dBm, 111
- Decoder
 - I²C, 15, 60
 - Base section to index, 61
 - CAN, 15, 60
 - J1939, 61
 - Pulse, 60
 - Serial, 15, 60
 - SPI, 61
 - UART, 15, 60
- Deleting objects, 21
- Deskew, 54
- Differentiate, 52
- Disconnecting objects, 20
- Disk writer, 64
- Displaying data
 - Graph, 65
 - Meters, 85
- Duty cycle, 56, 109
 - Inverted, 109
- Edge count
 - Falling, 111
 - Rising, 111
- EMI, 58
- EV charger, 59
- Exporting object data, 22
- Fall time, 110
- Falling edge count, 111
- FFT, 13, 58
- File
 - ASCII, 99, 118
 - AutoDisk, 95
 - Binary, 99, 118

- CSV, 99, 118
- JSON, 99, 118
- Loading, 103
- Matlab, 99, 118
- Saving, 93
- Setfile, 117
- TPIDX, 118
- TPO, 117
- TPS, 117
- Wave audio, 99, 118
- WinSoft, 118
- Filter, 53
 - Band pass, 53
 - Band stop, 53
 - High pass, 53
 - Low pass, 53
- Frequency, 109
- Gain/Offset, 51
- Graph, 65
 - Active, 66
 - Interpolation, 84
 - New, 65
 - Toolbar, 76
- Graph scheme, 16
- Grid, 63
- I²C decoder, 15
- Ideal filter, 54
- Input range, 40
- Input type, 39
- Inserting objects, 21
- Instrument toolbar, 28
- Instruments, 23
 - Combine, 24
 - Demo, 27
 - Network, 25
 - Open, 23
 - Toolbar, 28
- Integrate, 53
- Interpolation, 84
- J1939 decoder, 61
- JSON files, 99, 118
- Language, 16
- Limiter, 55
- Log, 53
- Math, 22, 52
- Matlab files, 99, 118
- Maximum, 57, 108
- Maximum - Minimum, 58, 108
- Mean, 108
- Measure mode, 31
 - Stream, 14
- Meter, 63
- Meter scheme, 16
- Meters, 85
- Min/Max detector, 56
- Minimum, 57, 108
- Momentary, 108
- Multiply/Divide, 52
- Object
 - Cloning, 19
 - Configuring, 18
 - Connecting, 19
 - Creating, 18
 - Delete, 21
 - Disconnecting, 20
 - Exporting, 22
 - Inserting, 21
 - Saving, 21
- Object tree, 17
- Oscilloscope in XY mode, 13
- Oscilloscope in Yt mode, 12
- Peak-peak, 108
- Period, 109
- Period count, 111
- Phase, 80
- Phase cursors, 80
- Phase difference, 58
- Power, 110
- Pre-trigger, 33
- Printing, 15
- Probe, 40
- Pulse count
 - Negative, 111
 - Positive, 111
- Pulse decoder, 60

- Pulse width
 - Negative, [109](#)
 - Positive, [109](#)
- Quick functions, [11](#)
- Quick Setups, [10](#)
- Range, [40](#)
- Record length, [33](#)
- Reference, [82](#)
- Reference I/O, [56](#)
- Reference resistor, [113](#)
- Resampler, [55](#)
- Resolution, [32](#)
- Rise time, [110](#)
- Rising edge count, [111](#)
- RMS, [57](#), [108](#)
- RPM, [59](#)
- Sample frequency, [33](#)
- Saving objects, [21](#)
- Sensors, [51](#)
- Serial decoder, [15](#), [60](#)
- Setfile, [117](#)
- Settings, [16](#)
- Signal Cleaner, [54](#)
- Slew rate, [110](#)
- Slice, [55](#)
- SoftGen, [49](#)
- Sound, [64](#)
- Source, [49](#)
- Spectrum analyzer, [13](#), [58](#)
- SPI decoder, [61](#)
- Sqrt, [52](#)
- Standard Deviation, [108](#)
- Start measuring, [28](#)
- Stop measuring, [28](#)
- Streaming mode, [14](#), [31](#)
- Sum, [22](#), [52](#)
- Table, [63](#)
- THD, [112](#)
- Time base, [33](#)
- Time per div, [33](#)
- Toolbar
 - Channel, [16](#), [29](#)
 - Generator, [41](#)
 - Graph, [76](#)
 - Instrument, [16](#), [28](#)
 - Quick functions, [11](#)
 - Scheme, [16](#)
 - User presets, [7](#)
- Trigger, [34](#)
 - Condition, [38](#)
 - Condition time, [38](#)
 - Hysteresis, [37](#)
 - Level, [37](#)
 - Source, [35](#)
 - Time-out, [35](#)
 - Type, [36](#)
- UART decoder, [15](#), [60](#)
- User preset worksets, [8](#)
- User presets, [7](#)
- Variance, [108](#)
- Voltmeter, [14](#), [63](#)
- Wave audio files, [99](#), [118](#)
- Window, [55](#)
- Zooming, [77](#)

If you have any suggestions and/or remarks regarding this manual, please contact:

TiePie engineering
Koperslagersstraat 37
8601 WL SNEEK
The Netherlands

Tel.: +31 515 415 416
Fax: +31 515 418 819
E-mail: support@tiepie.nl
Site: www.tiepie.com



