INTRODUCTION

Welcome to our user manual pages. If you have any questions, please email us at support@dewesoft.com

DEWESoft®
measurement innovation

OPTIONS

SETUP

MEASURE

ANALYSE

HOW TO SET UP DEWESOFT

Project

Settings

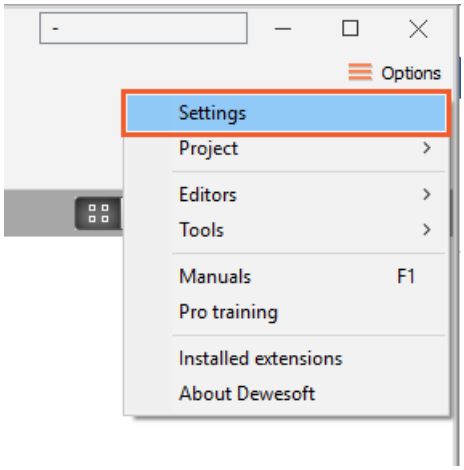
Editors

Tools

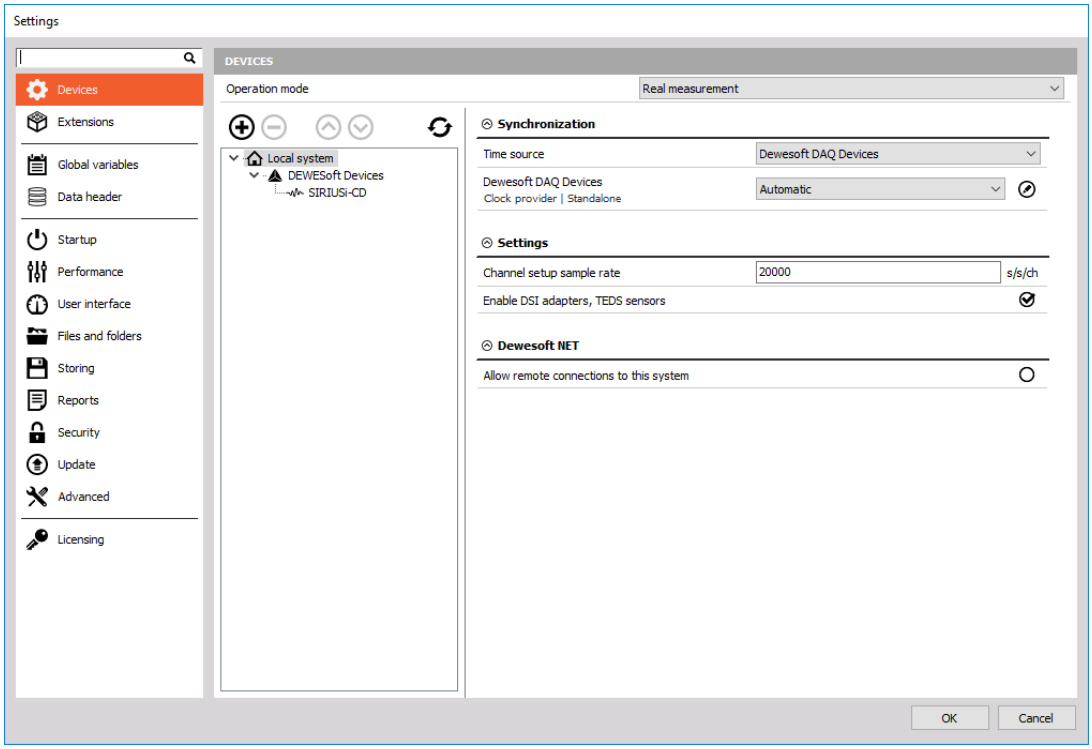
SETTINGS

After you have successfully downloaded and installed the Dewesoft software it is time to set it up according to your needs.

To access the settings in Dewesoft, go to the top right corner in the program.



When you open the Settings, the following screen will appear, which shows us the basic structure:



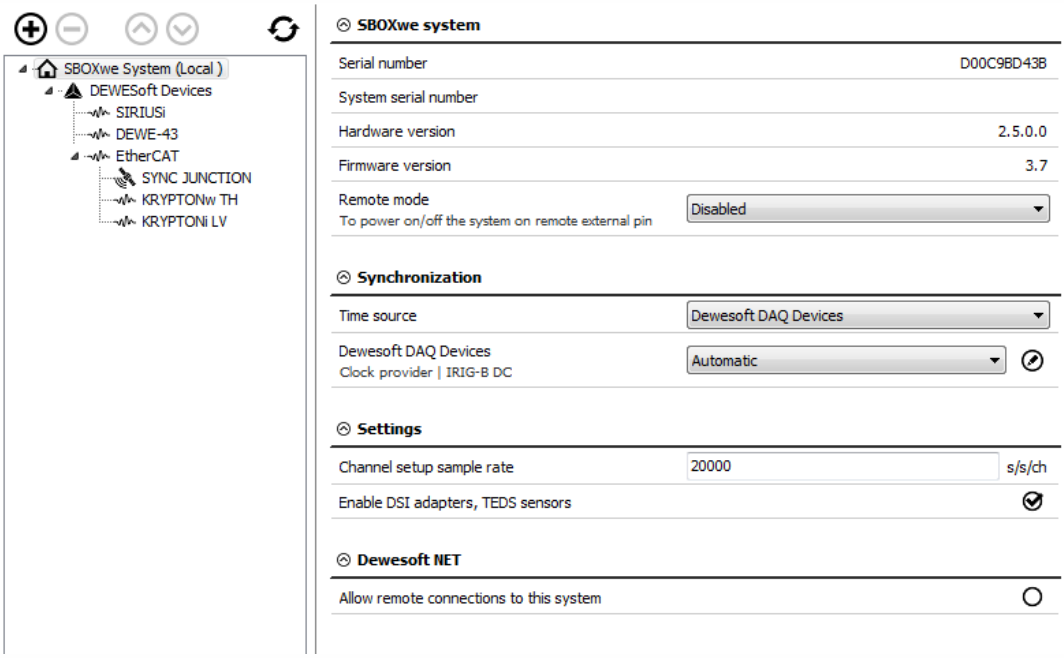
Devices - View the connected DAQ devices (Real measurement mode) or simuate channels (Simulation mode) to make an offline setup.

- Real measurement DAQ devices (Sirius, Dewe-43, Krypton) are used to acquire data
- Simulation mode Simulation mode can be used when we don't have any DAQ devices connected to our system and we want to simulate different types of channels to make an offline setup.
- Extensions Add a plugin or extension according to your needs.
- Global variables Define the channels, that can be used in a sequencer, math section or in displays.
- Data header Define the input fields for different operators.
- Startup Allow multiple instances of Dewesoft or load a setup or a sequence as soon as Dewesoft starts.
- Performance Select the application process priority, use multiple CPU cores and define other performance characteristics.
- User interface Adjust Dewesoft according to your wishes - change the language, font size, background color and many other settings.
- Files and folders Define the starting folder where the setups, data files, and other database files are stored.
- Storing Create a zipped data file to reduce the file size or use the video compression.
- Reports Use your own company logo in reports when printing the data file.
- Security Lock the settings with a password. A password can also be used to change the setup, editing sensors, stopping the measurement, changing the sequence or leaving the full screen.
- Update The firmware for Dewesoft hardware can now be upgraded directly in Dewesoft software.
- Advanced Set advanced options for different areas, like hardware, visuals, math, diagnostics, and analysis.
- Licensing Enter license number and register software. You can choose between online or offline registration. To receive a fully functional 30-day evaluation license for Dewesoft software fill and submit the form on our web page.

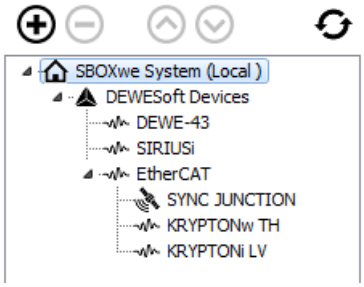
REAL MEASUREMENT

In the Real measurement mode, we can see all the devices that are connected to our system.

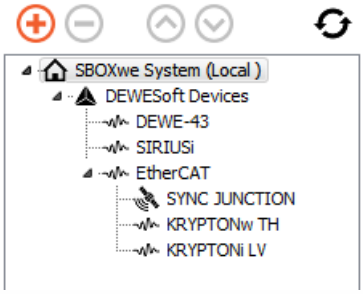
All Dewesoft devices (SIRIUS, DEWE-43, KRYPTON, SBOX, ECAT-SYNC JUNCTION) are automatically recognized, but there are some devices, that can't be connected automatically.



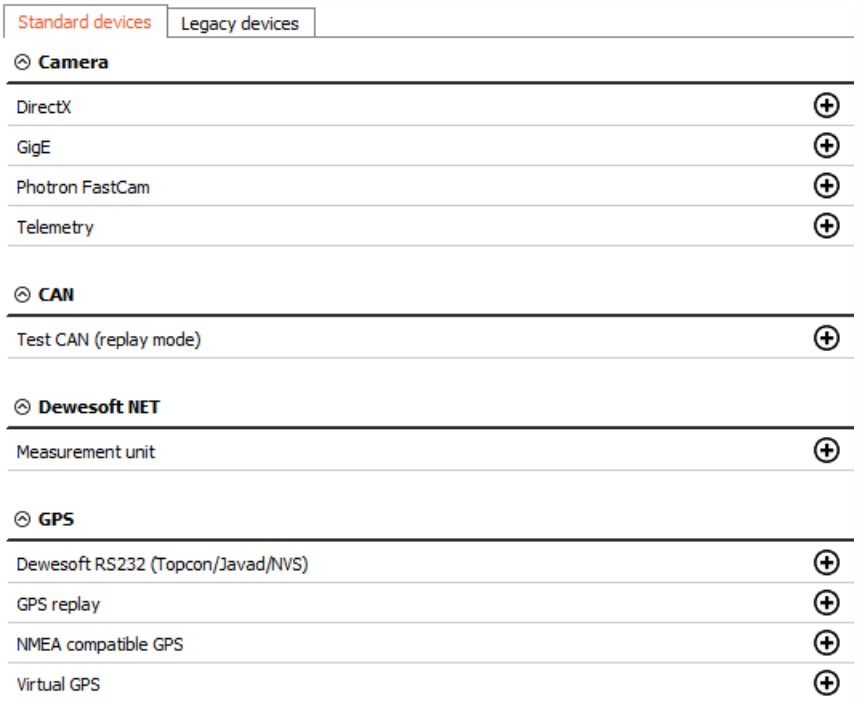
On the left side of the window, the structure of the system is shown. On the SBOX system, we connected USB devices (SIRIUS, DEWE-43) and EtherCAT devices (KRYPTON, ECAT-SYNC JUNCTION).



With the plus button, we add devices that are not recognized automatically and with the minus button, we can remove them.

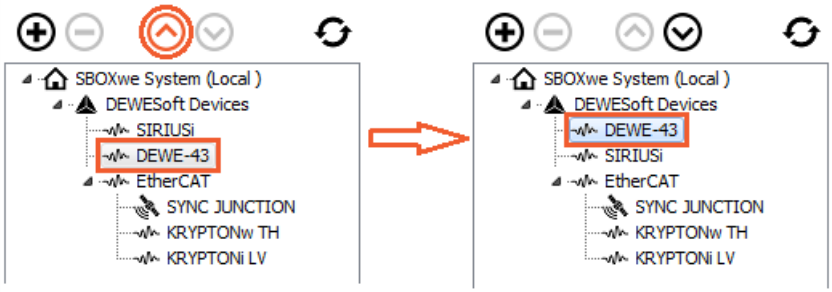


When adding a new device, we can select from Standard devices (Camera, CAN, Dewesoft NET, GPS)

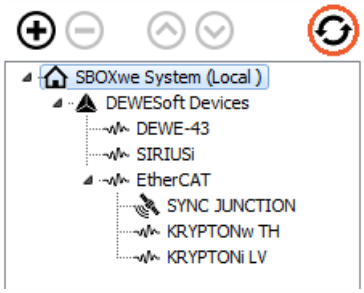


or from Legacy devices (CAN, GPS).

If we have multiple slices, we can rearrange their sort order with the up and down buttons.



With the refresh button, we scan the whole structure of a system again.

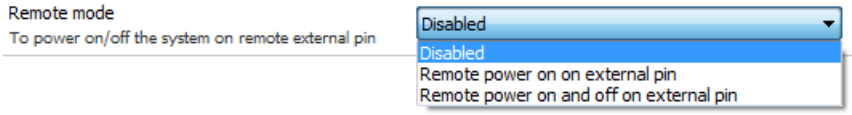


On the right side of the window, the properties of devices are shown and some settings can also be changed.

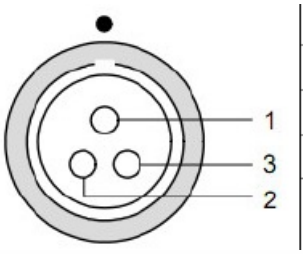
SBOX system

We can see the devices information (serial number, hardware, and firmware version).

If the SBOX system is used, the remote mode can be used to turn the SBOX on or off from the remote distance.



The connector for power supply on SBOX has three pins, one is for voltage, one for a ground connection and one is a remote signal.



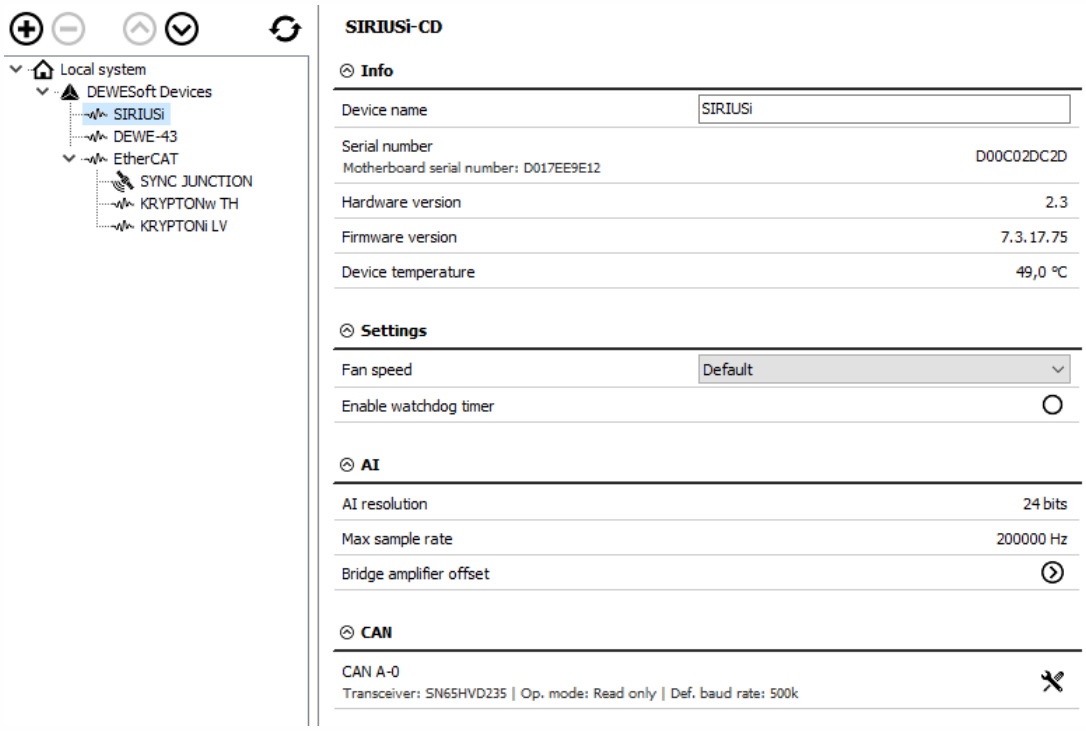
Pin	Name
1	V+
2	GND
3	Remote-On[2]

- Disabled - If the remote mode is disabled, SBOX can only be turned on or off by pressing the power button on the device.
- Remote power ON on external pin - The SBOX can be remotely turned on. When the voltage signal comes to pin for remote, the SBOX is turned on.
- Remote power ON and OFF on external pin - The SBOX can be remotely turned on or off. If the SBOX is off and the signal comes, it will be turned on. If the SBOX is on and the signal comes, it will be turned off.

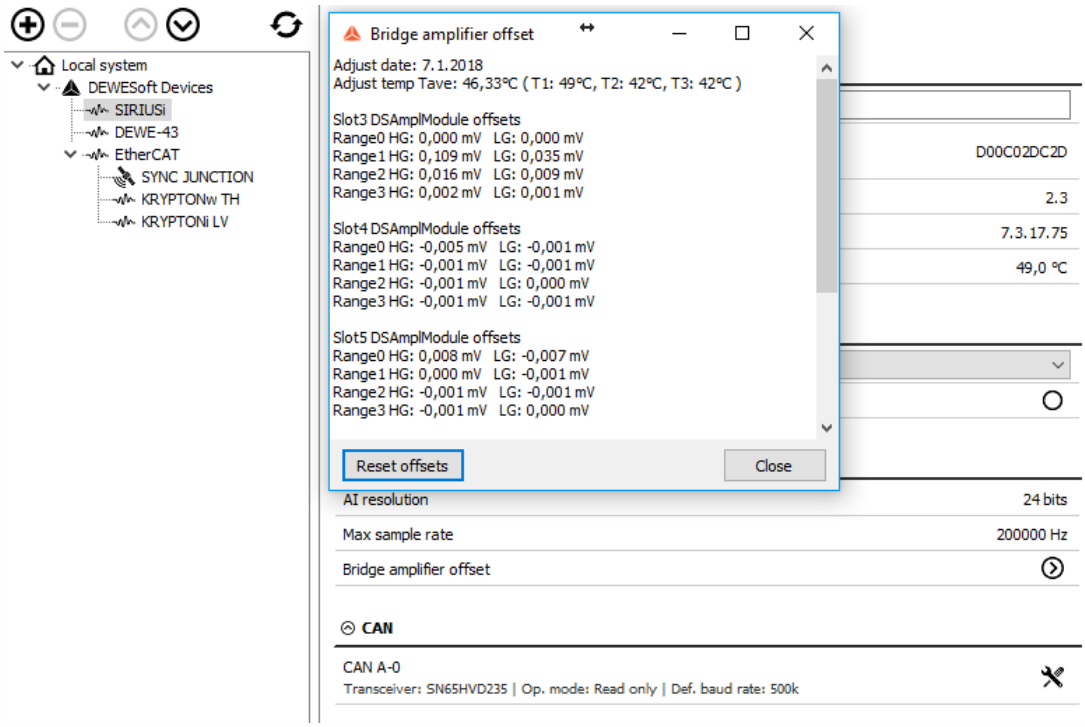
WARNING: After the Remote mode option is changed, a total reset of the device is required. The SBOX needs to be shut down and disconnected from the power supply. If the SBOX is also powered by batteries (e.g. R1DB, R8B,..) then the batteries need to be removed as well. The setting will become permanently enabled after a total reset.

SIRIUS

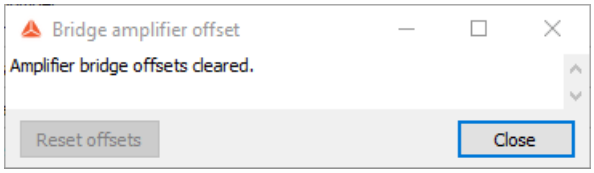
In this case, we have connected multiple Dewesoft DAQ devices. If we select the DAQ device and click on it, we see the devices information (device number, serial number, hardware and firmware version, temperature).



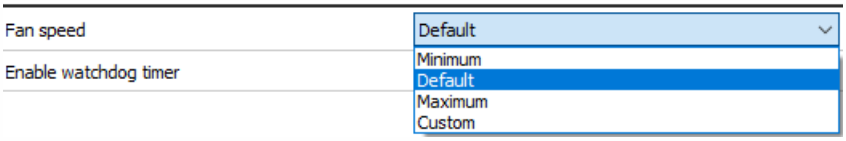
Bridge amplifier offset shows us the initial amplifier offset. Sirius connects the input short on and the offset is measured.



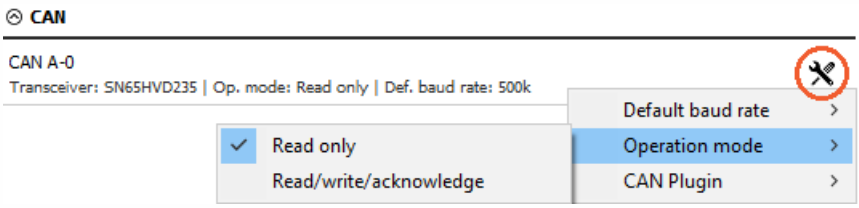
With the button Reset offsets, we clear the initial amplifier bridge offsets.



Fan speed can be set only to Sirius devices. We can choose from default, minimum or maximum. The custom speed is defined with the percentage of the maximum fan speed.



We can change the CAN baud rate and CAN operation mode (Read only or Read/Write/Acknowledge).

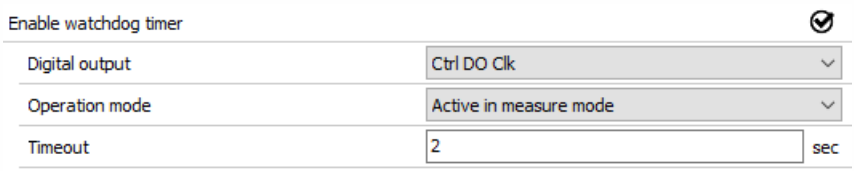


Watchdog timer

Dewesoft is quite often used in many critical applications where the control system depends on the data acquisition system or the data acquisition system outputs alarms used to warn the user or switch off the test. In such cases, it is important to know the operation state of the data

acquisition system. Watchdog is a safety feature to switch on the digital output when the system is in operation and switches off if the data acquisition system becomes unresponsive. A timeout can be set to delay switching off the watchdog. This feature can be used with any Sirius slice having Digital output. For SIRIUS 8xSTGM DB special watchdog hardware (DS-WDT) is available which has four relay outputs for directly switching on and off. Relays can be programmed as the Watchdog or standard alarm/manual outputs.

The Watchdog functionality enables an external way to monitor the behavior of the Dewesoft software. To be able to use the watchdog a Sirius slice with a Digital out is needed or when using a single slice, the Sync signals can also be used. The slice must be connected in order to set up the watchdog.



Once enabled, the output selection can be made. Default settings are:

- Digital output: Ctrl DO Clk
- Timeout: 2 sec
- Operation mode: Active in measure mode

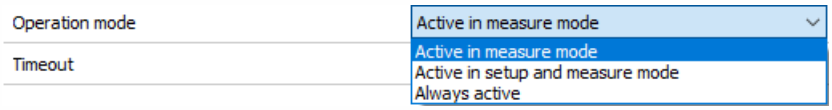
Digital output menu selects on which digital output the Watchdog functionality is enabled. When using the Watchdog module, which is connected to the Sirius slice via a 25 pin cable, this selector should be set to Ctrl DO 1.

NOTE: **The Watchdog can be only mapped to one digital output!**

The timeout value specifies the time in which the Dewesoft must reset the Watchdog.

Operation mode can be selected from:

- Active in measure mode - The Watchdog functionality is active while Dewesoft is in measure mode.
- Active in setup and measure mode - The Watchdog functionality is active while Dewesoft is in channel setup and while in measure mode.
- Always active - The Watchdog in always active.



Digital output menu selects on which digital output the Watchdog functionality is enabled. When using the Watchdog module, which is connected to the Sirius slice via a 25 pin cable, this selector should be set to Ctrl DO 1. The Watchdog can be only mapped to one digital output!

NOTE: **The Watchdog is not active while in Analysis mode!**

The Watchdog behavior is as follows:

- When the slice is powered up, the module is in the Alarm state (Watchdog LED is RED). The slice does not retain any information about the Watchdog after power loss!

- When Dewesoft is started and no Watchdog functionality was set for the slice in a previous session the LED light is RED.
- When the Watchdog functionality is set, the following occurs:
- Active in Ch. Setup is not set: The watchdog LED light is RED until the user switches to Measure mode; then the Watchdog LED is GREEN. When switching back to channel setup mode the LEDturns back to RED.
- Active in Ch. Setup is set: The Watchdog LED light is GREEN in both Channel setup and in Measure mode.
- When disconnecting the slice from USB or turning off Dewesoft software the Watchdog will trigger (after the present time in case of disconnect or computer/software freeze or immediately when exiting Dewesoft). The LED light will turn RED.

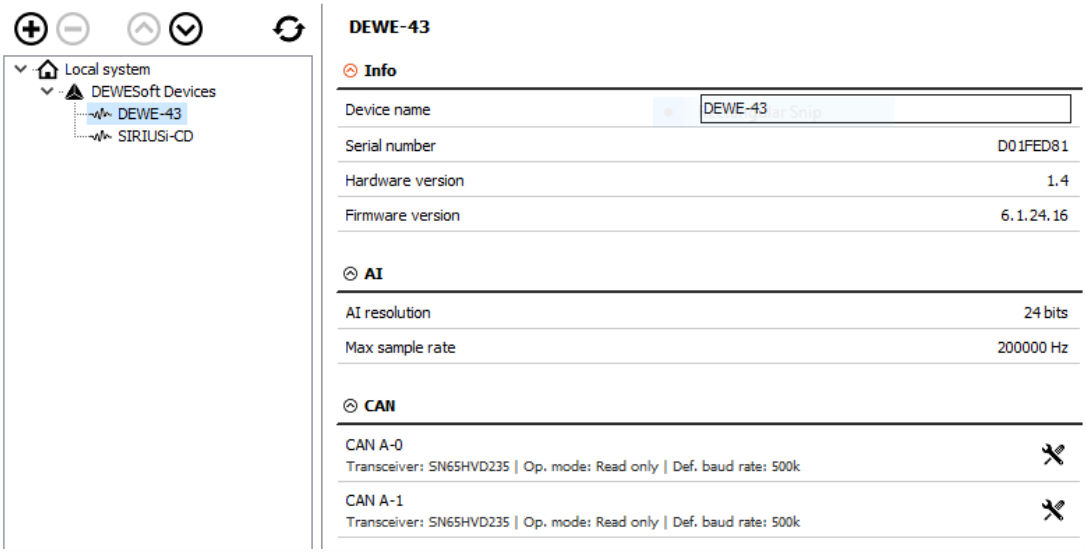
The Watchdog module itself has 3 additional outputs which are user controlled via the A/D module. The respective A/D outputs are CTRL DO 2, CTRL DO 3 and CTRL DO 4.

All four outputs are relay based with a possibility to connect with a NO (Normally open) or NC (Normally closed) position.

When the Watchdog has triggered or upon power up the Watchdog relay is not energized – meaning that the NO position is OPEN and the NC position is SHORTED.

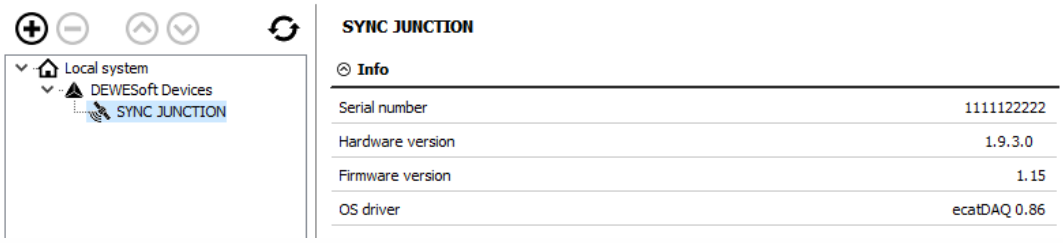
DEWE-43

In DEWE-43 settings, we can change the CAN baud rate and CAN operation mode (Read only or Read/Write/Acknowledge). We can also see its name of the device, serial number, hardware version, and firmware version.



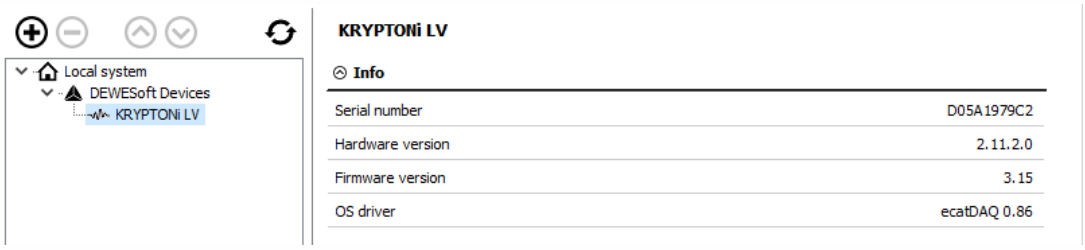
ECAT-SYNC JUNCTION

ECAT-SYNC JUNCTION works in the same way as other Dewesoft USB devices. It is automatically recognized within Dewesoft X software. By default, ECAT-SYNC JUNCTION will be set up to synchronize KRYPTON EtherCAT® and SIRIUS USB. The serial number and the firmware version can be seen and also updated.



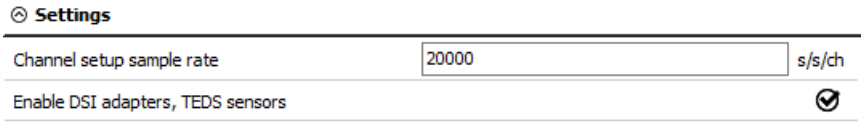
KRYPTON

Under KRYPTON section, serial number, hardware, and firmware versions are seen. Also, the firmware upgrade can be done.



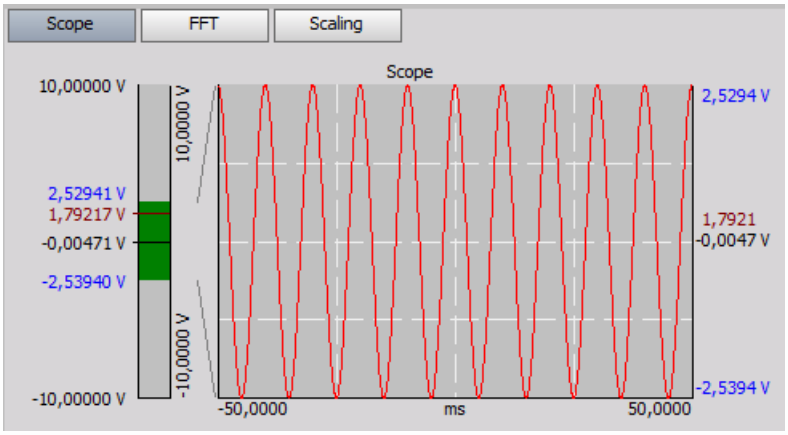
CHANNEL SETUP SAMPLE RATE

Channel setup sample rate defines the sample rate of channel setup. Channel setup sample rate does not run with the full acquisition sample rate but with the reduced one.

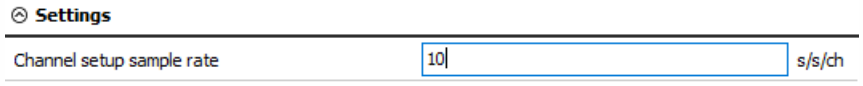


Let's take a look at a signal from a function generator (simple sine wave with 100 Hz). First, we let the channel setup sample rate at the default value, 20 s/s/ch.

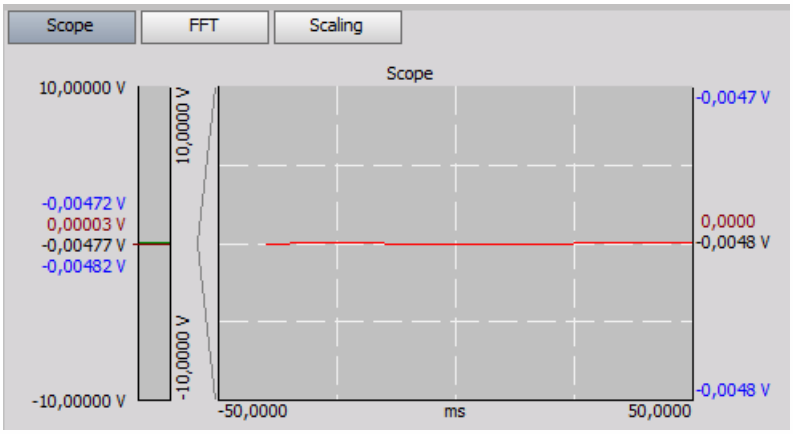
When we enter the setup of the selected channel, we see the preview of the scope nicely.



Then we change the channel setup sample rate to 10 s/s/ch.



Now, the preview of the channel does not show the proper form of the signal, because it is sampled with the lower frequency than it should be.



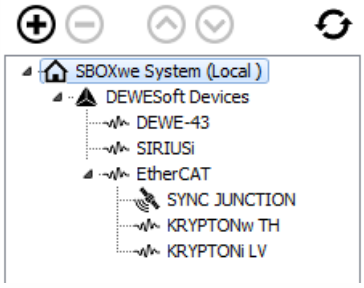
SYNCHRONIZATION

The basic idea of synchronization is to provide a clock signal from a time source. Clock slave receives the signal from the clock provider and the devices are synchronized.

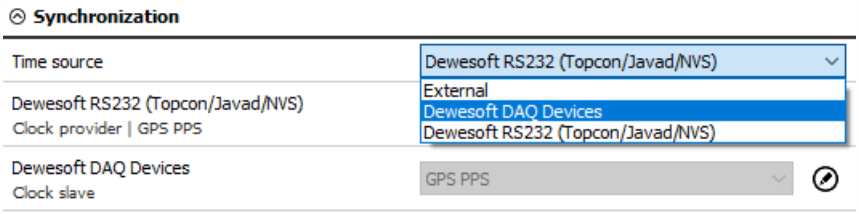
Devices can be synchronized in two different ways:

- Software synchronization - The software synchronization accuracy is around 2-10 ms, which is enough for a simple temperature measurement. This synchronization solution requires no additional hardware.
- Hardware synchronization - It is a hardware solution which can synchronize all USB devices (SIRIUS, DEWE-43, ...) and EtherCAT devices.

We have connected one Sirius, one DEWE-43 and one Krypton measurement unit to the SBOX.



Time source



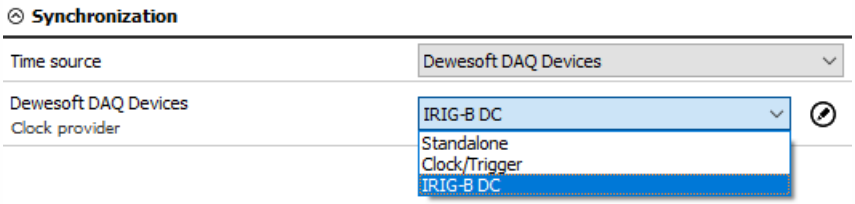
Time source provides a clock for synchronization. It can be selected from:

- Dewesoft DAQ devices (when we have a Dewesoft measurement device connected to our computer),
- External (Clock/Trigger, IRIG-B DC, NTP, GPS PPS),
- PC clock (when we have only a computer, without a measurement device),
- GPS devices (Dewesoft RS232 (Topcon/Javad/NVS), NMEA compatible GPS, ...).

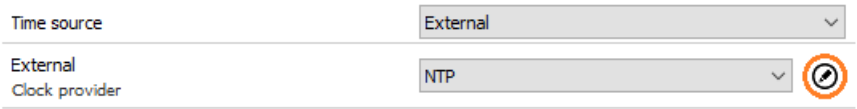
NOTE: **Synchronization types are dependent and automatically adjusted from the DAQ devices connected to our system.**

TYPE OF SYNCHRONIZATION

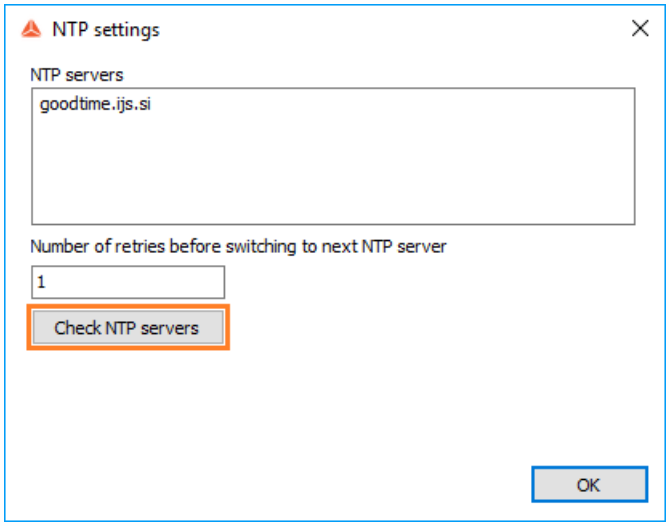
If Dewesoft DAQ device is selected as a time source, clock provider must be selected from:



- Automatic - This option automatically selects the best option for synchronization regarding the hardware connected to the system.
- Standalone - Only one device, there is no synchronization between the devices needed.
- SoftSync - This synchronization solution requires no additional hardware. The accuracy is > 10 ms.
- Clock/Trigger - The clock of the device gives a trigger signal.
- IRIG-B DC - It contains time-of-year and year information in a BCD format (it contains the information about the absolute time). This is the best way to synchronize the devices because it is the most exact one.
- GPS PPS - Since the satellites are transmitting exact absolute time and better receivers usually output this pulse with a high precision (below one microsecond), we can use this technology to synchronize remote systems - and there is no distance limit.
- NTP - Network Time Protocol (NTP) is a networking protocol for clock synchronization between computer systems. It is less accurate than other methods.

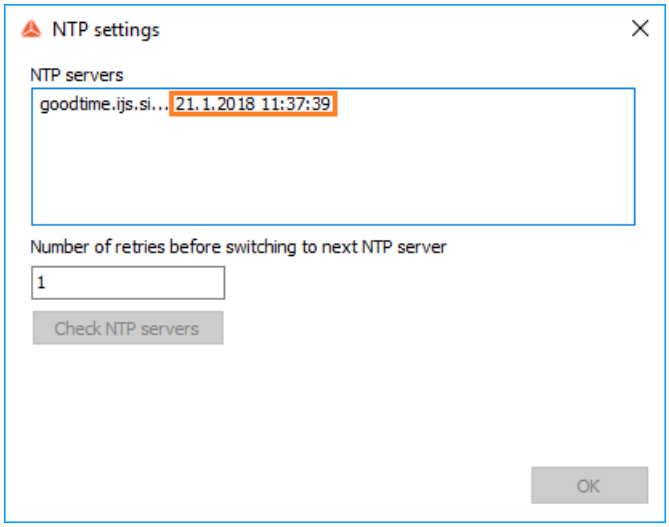


To enter the addresses of NTP server click the edit button near the drop-down menu. If you want to have more devices synchronized via NTP protocol, the same server address must be entered in all of them.

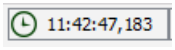


When you check NTP servers, time and date of the server will show. If the server address is wrong,

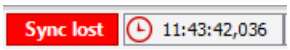
the check will fail.



If the synchronization via NTP is successful, the clock next to the time will be black:



If the sync is lost, the clock will become red and the "Sync lost" message will appear:



SYNCHRONIZATION BETWEEN DEWESOFT USB DEVICES

	Accuracy	When to use	Device
Clock/trigger	~ 1 us	stationary	Dewesoft, RoaDyn, ...
IRIG-B DC	~ 1 us	stationary	Dewesoft, Meinberg, ...
GPS PPS	~ 1 us	mobile	GPS receiver
NTP	+ 10 ms	Ethernet	NTP server
SoftSync	+ 10 ms	When there is no external time source	/

Any Dewesoft device can be precisely synchronized by hardware (Sirius, Dewe-43, Minitaur, DS-CAN2).

When using the NET system, there are several possibilities that can be used for synchronization:

- Clock/trigger - relative time
- IRIG-B DC Master / IRIG-B DC Slave - absolute time
- GPS - absolute time
- NTP - absolute time

Clock/trigger connection example

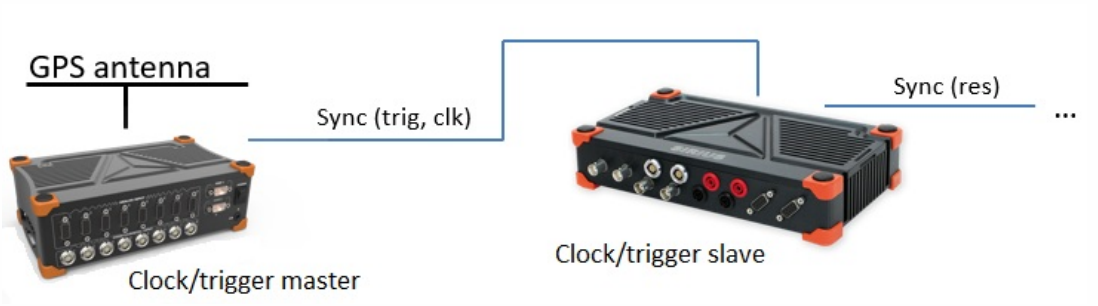
IRIG-B DC connection example of one unit

- Dewesoft device as IRIG-B DC generator

- External IRIG-B DC provider

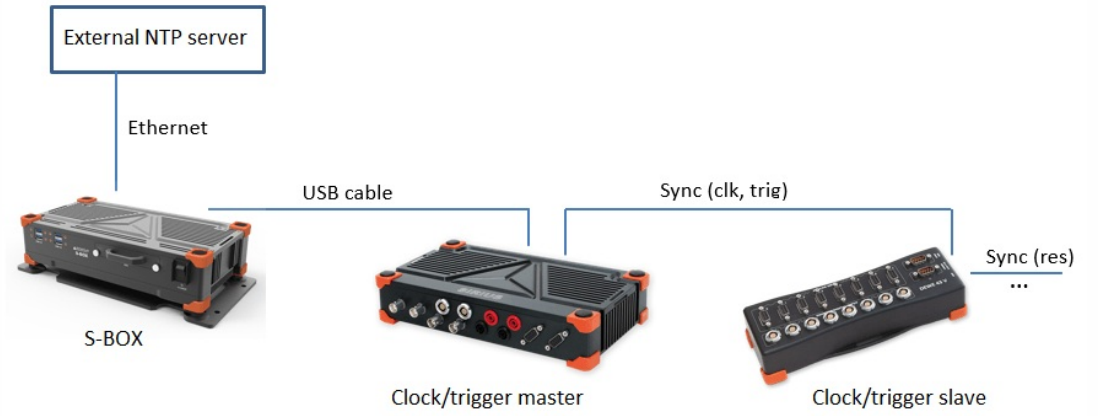
GPS PPS connection example of one unit

- Dewesoft device has a GPS receiver



* External GPS receiver

NTP connection example

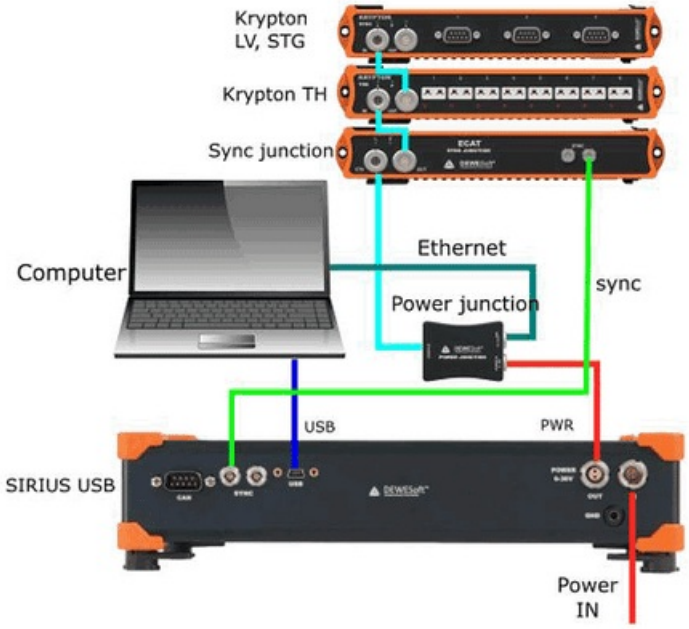


SYNCHRONIZATION WITH ECAT-SYNC JUNCTION

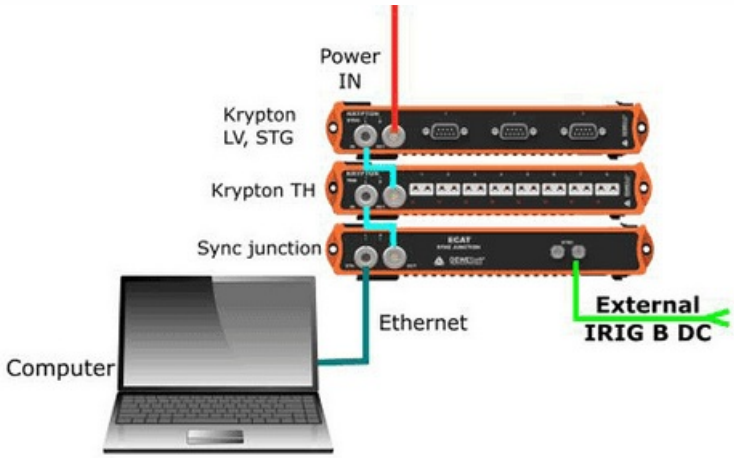
ECAT-SYNC JUNCTION works in the same way as other Dewesoft devices. It is automatically recognized within Dewesoft X software (supported in version X2 SP4). By default, ECAT-SYNC JUNCTION will be set up to synchronize between KRYPTON EtherCAT® and SIRIUS USB.

With ECAT-SYNC JUNCTION several connection options are possible:

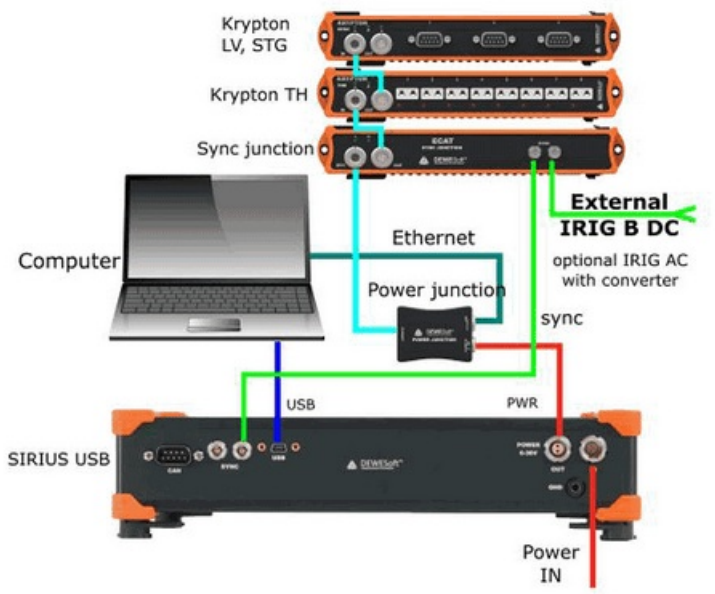
- Synchronization of SIRIUS/DEWE-43 USB with KRYPTON/SIRIUSiwe EtherCAT® devices, where the accuracy of synchronization is < microsecond.



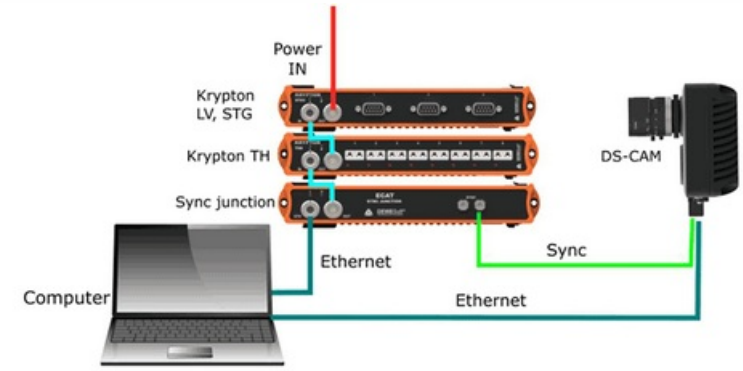
- Synchronization of KRYPTON module with external IRIG B DC triggering source.



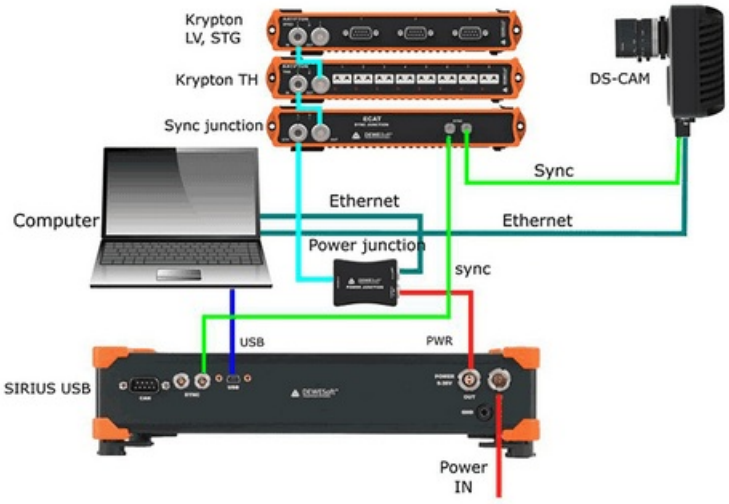
- Synchronization of KRYPTON and SIRIUS USB with external IRIG B DC triggering source.



- Synchronization of KRYPTON/SIRIUSiwe with triggered cameras.

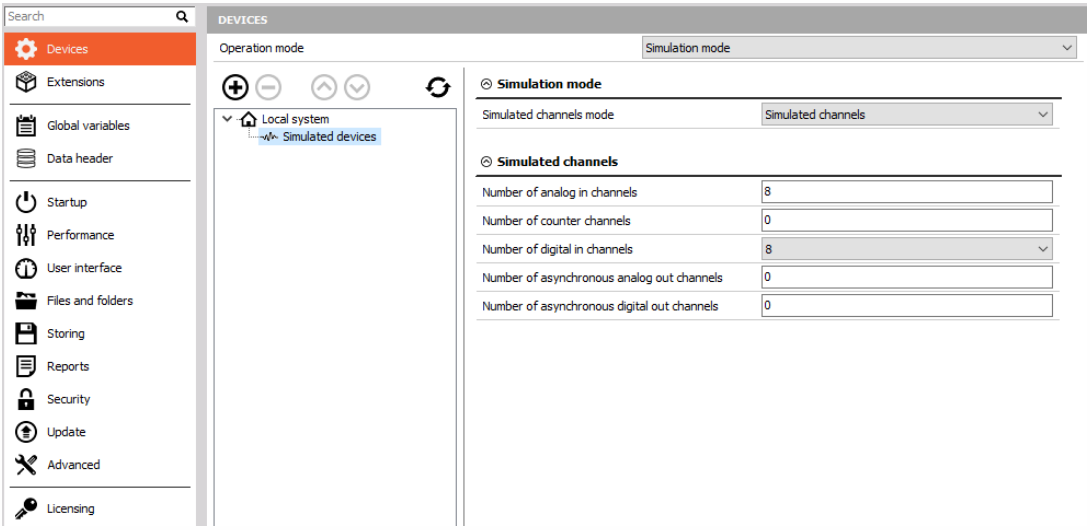


- Synchronization of SIRIUS/DEWE-43 USB with KRYPTON/SIRIUSiwe and triggered cameras.

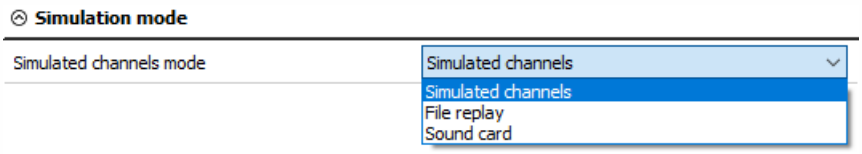


SIMULATION MODE

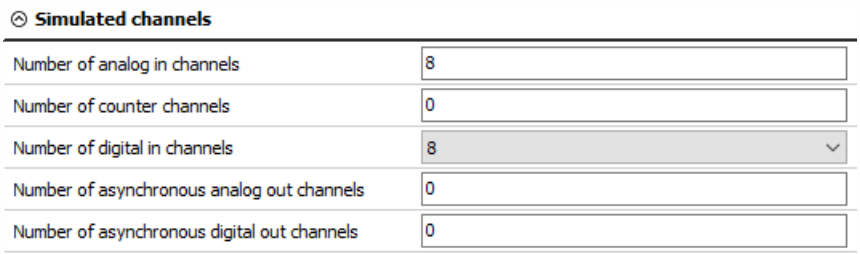
Simulation mode can be used when we don't have any DAQ devices connected to our system and we want to simulate different types of channels to make an offline setup.



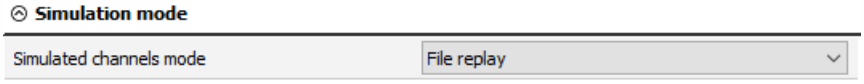
Simulation mode can be chosen from:

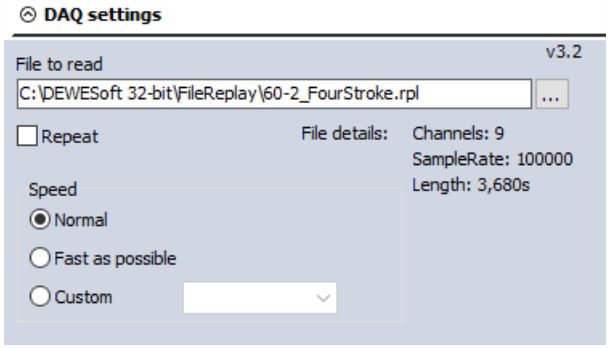


- Simulated channels - We define the number of analog in channels, counter channels, digital in channels, asynchronous analog out channels and asynchronous digital out channels.

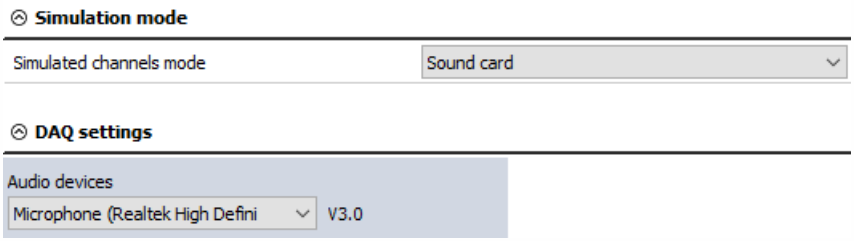


- File replay - We can replay files, that have already been recorded.





- Sound card - We can use the sound card in our computer to collect the data and make a measurement.



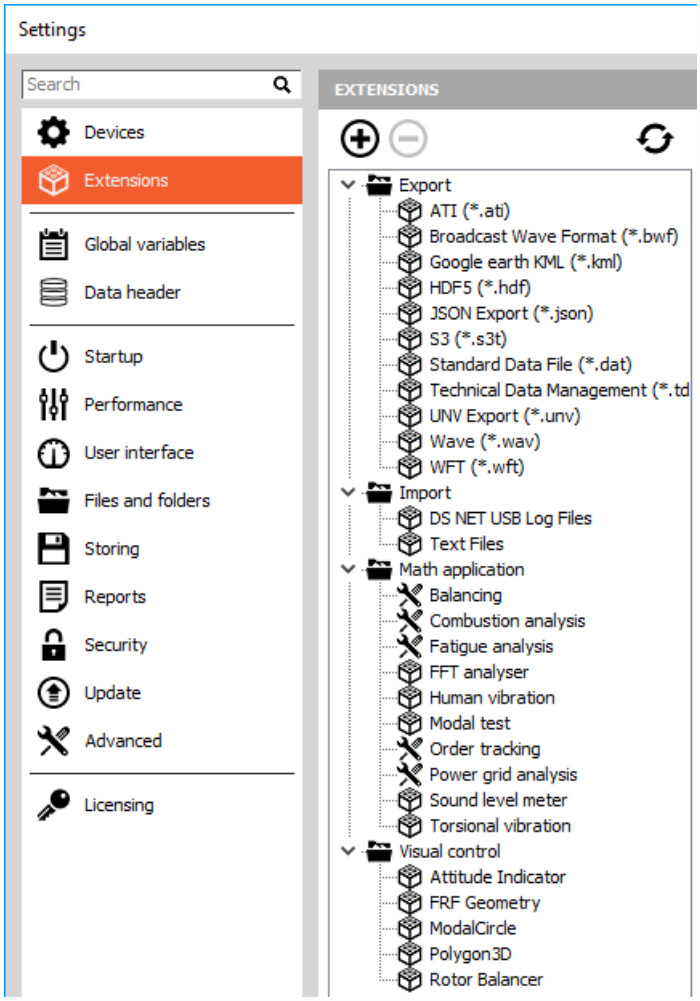
DEWESOFT NET

NET acquisition data

EXTENSIONS

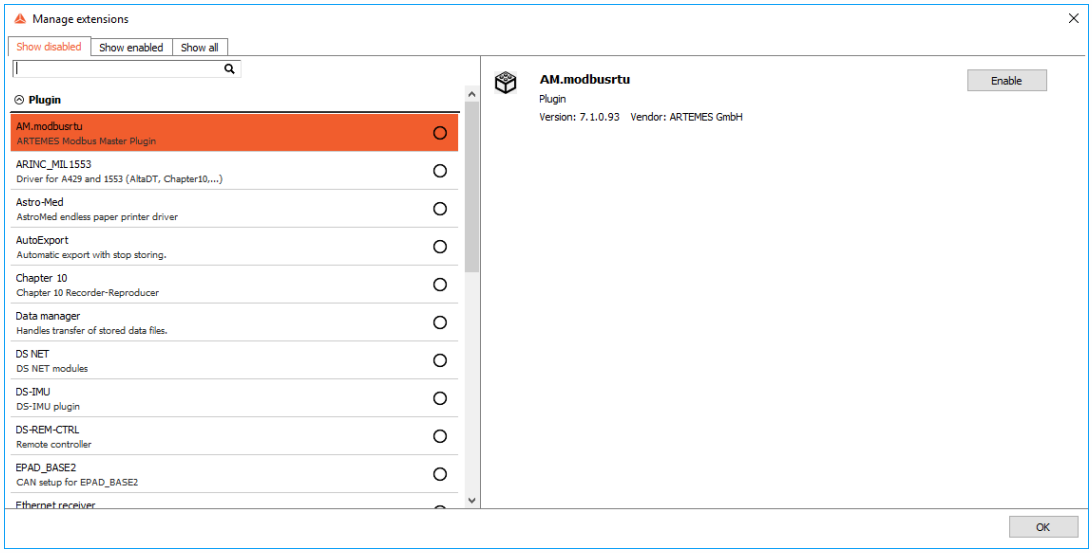
If you want to enable plugins or extensions, please go to Extensions section. Extensions are divided into five different sections:

- Export (export the datafiles to different formats, like ATI (.ati), Google Earth (.kml), Broadcast wave format (.bwf), Standard data file (.dat), Wave (.wav), Technical data management (.tdm), ...), Import (DS NET USB log files or text files.)
- Math application (set additional settings to several additional math applications like balancing, CA, fatigue analysis, FFT analyser, FRF, Power, ...)
- Plugin (enable several additional plugins like Polygon, Sound Power, Weather station, DS NET, Chapter 10, AutoExport, Control channels, Ethernet receiver, ...)
- Visual controls (information about different visual controls inside Dewesoft like FRF geometry, Modal circle, Polygon, Rotor balancer, Altitude indicator.)

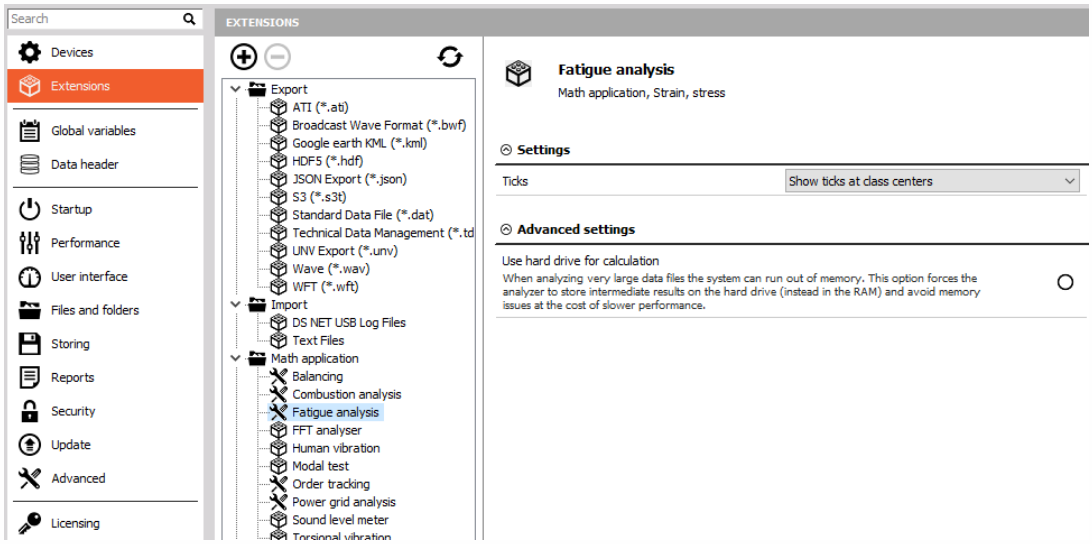


To add a new extension, click the plus button.

Here you can see plugins, that are disabled. To enable them, select the plugin and click on Enable button.



When selecting all the needed Extensions, we can see some specific settings related to the selected plugin. To learn more about their meaning, please go to courses that are describing the specific section.

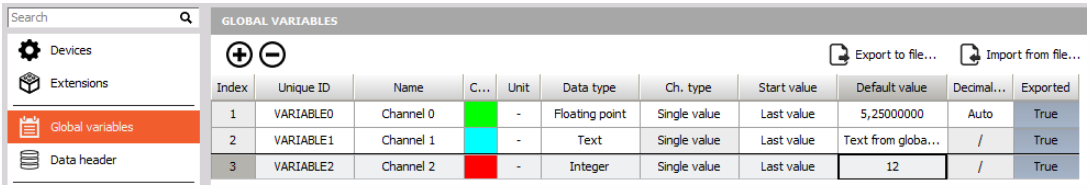


HINT: **If some of the plugins are missing on the list use the refresh extension button and restart Dewesoft.**

GLOBAL VARIABLES

Global variables define the channels, which can be used in a sequencer, math section and displays.

New variables can be added with a plus button.



Index	Unique ID	Name	C...	Unit	Data type	Ch. type	Start value	Default value	Decimal...	Exported
1	VARIABLE0	Channel 0		-	Floating point	Single value	Last value	5,25000000	Auto	True
2	VARIABLE1	Channel 1		-	Text	Single value	Last value	Text from globa...	/	True
3	VARIABLE2	Channel 2		-	Integer	Single value	Last value	12	/	True

The Unique ID must be defined, which is used as a reference for all others places where internal variables are used. Channel name can be freely defined, we can define the color and the unit.

Data type can be:

- Floating point (floating decimal point, we can also define the number of decimal numbers)
- Integer (whole number)
- Text (text written by the user)

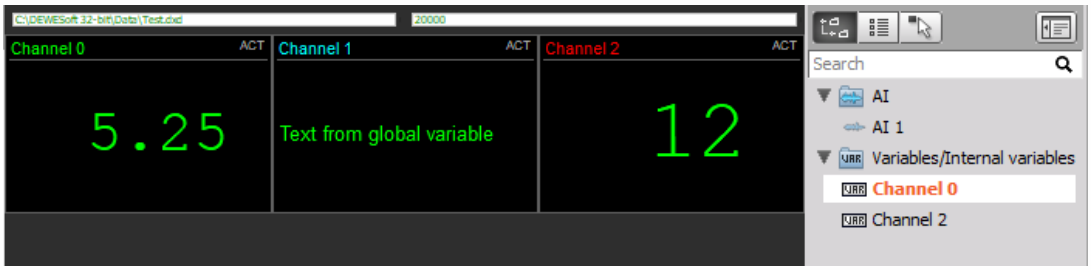
Channel Type can be:

- Single value (storing only one value per entire measurement)
- Async (adding a value each time the channel changes from sequencer or math)

Start value can be selected from:

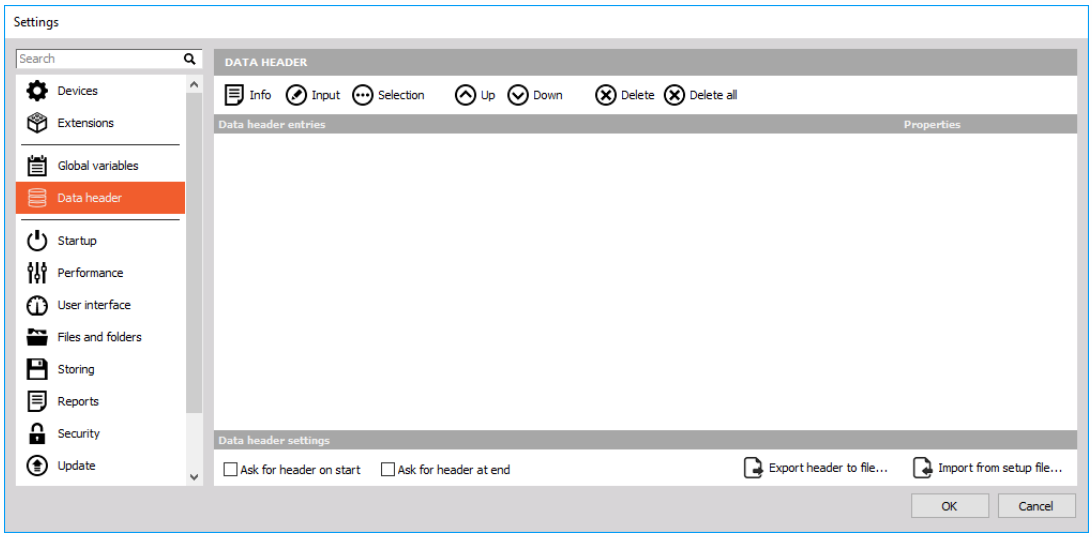
- Last value (variable will remember the last value)
- Default value (it will always start with the defined default number

Default value describes what value the channel has when starting up.



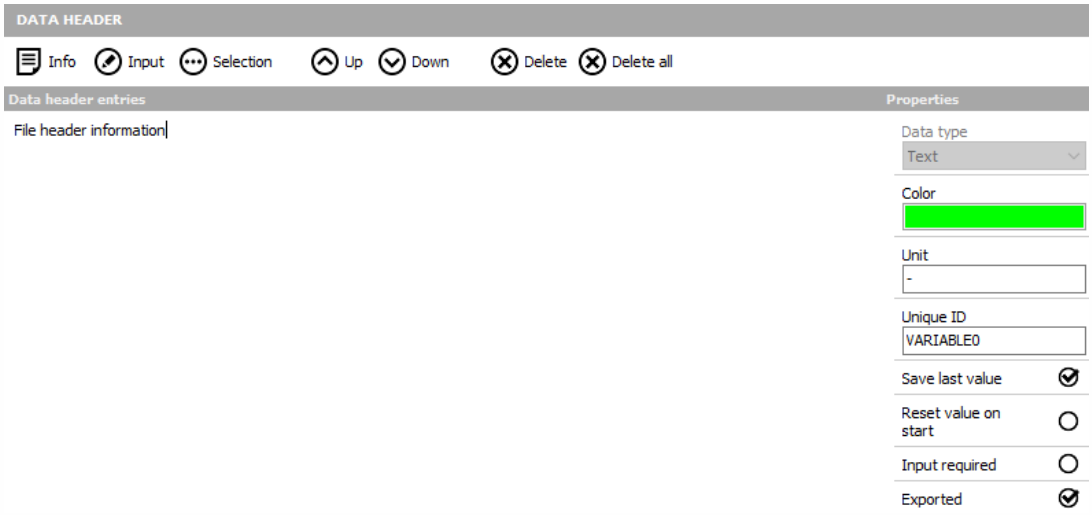
DATA HEADER

Data header allows defining input fields in which operators can enter additional not measured parameters at the start or end of the measurement.

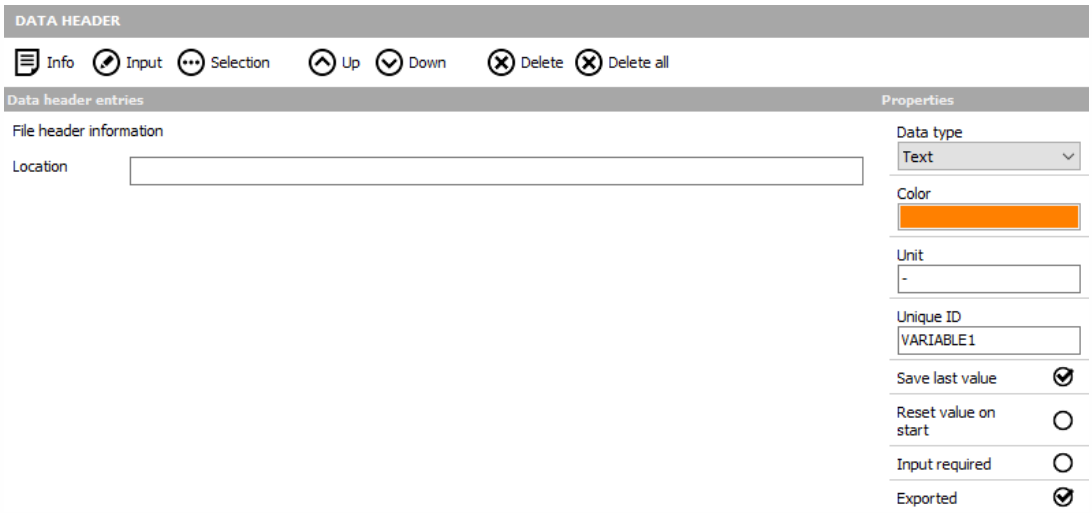


In data header, we can define:

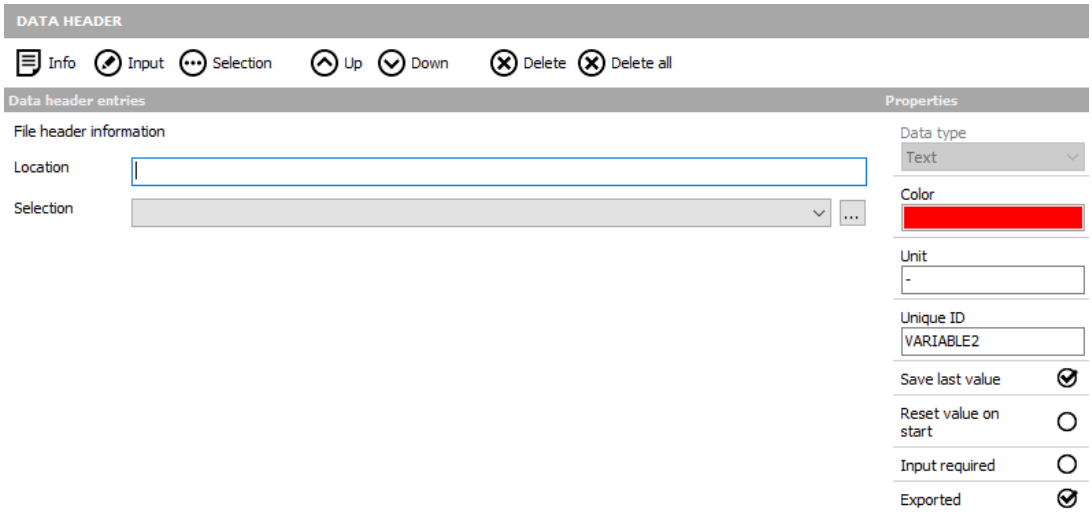
- Info field - The Info button adds a new line in the right section of the window. Just click on the new line and enter the desired text, for example: 'File header information'.



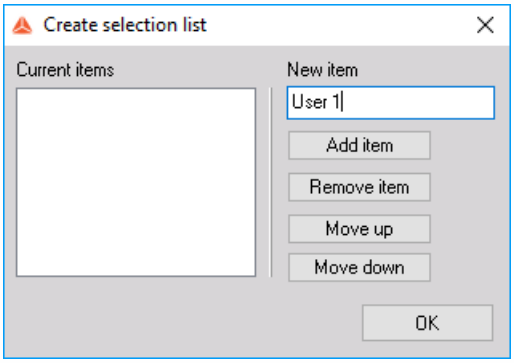
- Input field - The Input field can be used for any single line values or comments (you can use it for example for location information...). Let's create a new field with the press on the Input icon and rename it (for example to 'Location'). To rename the field, just click on the name and overwrite it.



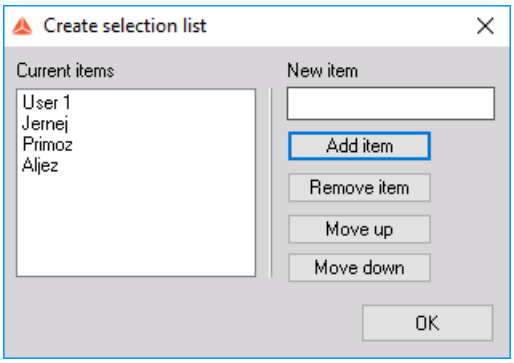
- Selection field - The Selection field can be used for creating a list of 'value' and is very useful when you have to use the same 'value' very often. This can be for example a list of usernames or also a list of departments within your company. Let's create a new list with the press on the Selection icon and label it (for example 'User' - click on 'Selection' and overwrite it).



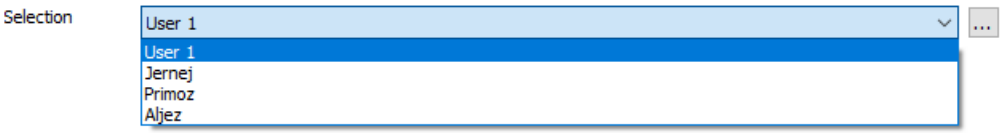
Now you have to define the content of your list. To do that, press the three dot button on the right side and create selection. List window appears. Enter the desired names in the New item field and press the Add item button.



The name will appear in the list on the left side, in our example labeled User 1.



Enter as many names as required. You can also edit values directly in the list by selecting an item. To remove an item, just select it from the list on the left side and press the Remove item button. When all names are entered, press the OK button or press Cancel to reject all entries. The result of upper entries is the following drop-down list in the global header window:



- Position up/down - With both buttons up and down, you can now change the position of the:
- Filename field
- Comments field
- Input field
- Selection field
- Info field (like created above)

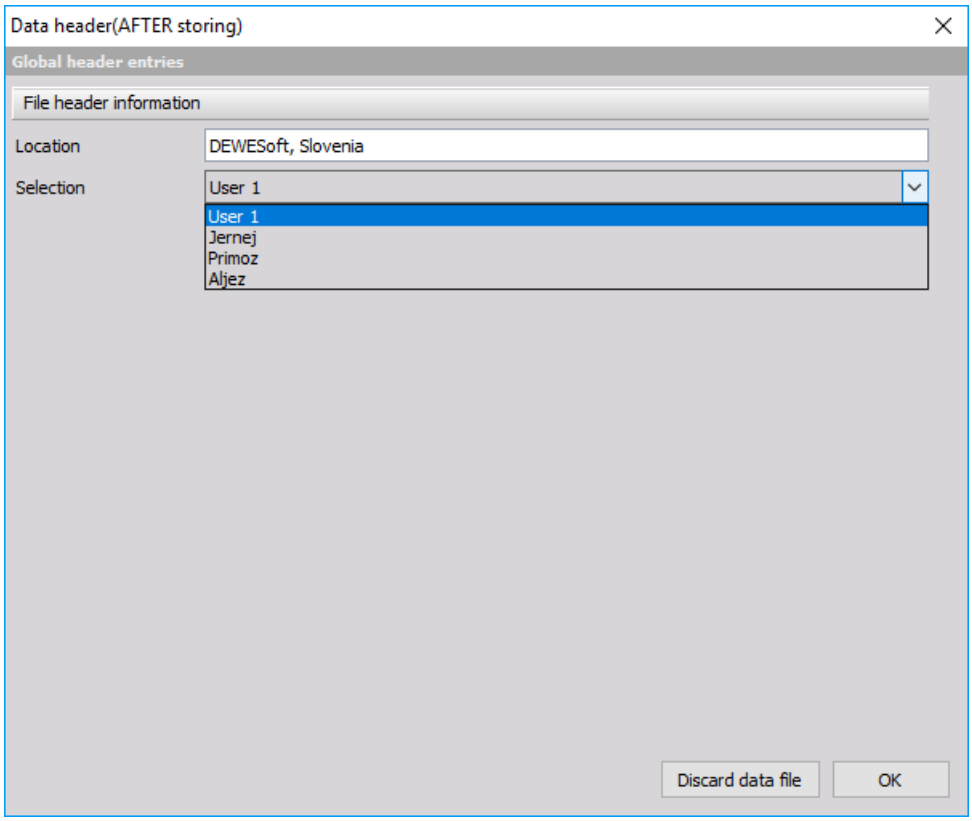
You can change the position of fields at any time. Just select it (the red box will appear) and move it up or down using the appropriate button.

- Delete field - To remove unused or non-required fields, select the field and press the Delete icon. The only field you can't delete is the File name entry field.

Under header settings, you can select whether to ask for header on start of the storing or at the end:

- Ask for header on start option will pop up a window with all entries before the start of storing.
- Ask for header at end will open the same window when the measurement is stopped.

If we have selected the option Ask for the header at end, we get the window like in the picture below after we stop storing the data. We can see the input sections defined earlier in Settings.

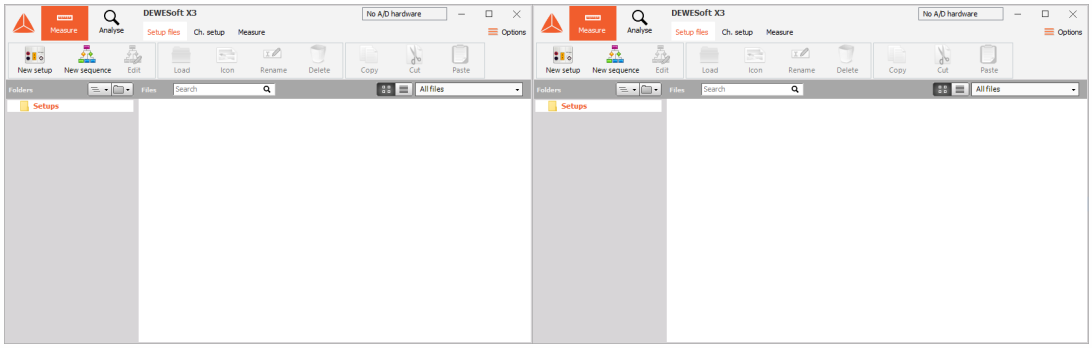

Data header(AFTER storing)
Global header entries
File header information
Location
DEWESoft, Slovenia
Selection
User 1
User 1
Jernej
Primoz
Aljez
Discard data file
OK

STARTUP

Startup option allows you to select what you want Dewesoft to do when you start it.

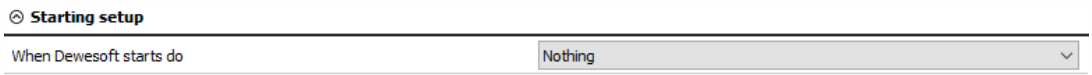
Allow multiple instances of Dewesoft can be selected to allow that Dewesoft is started multiple times on the same computer for special applications (running different devices, analyzing old data while measurement process is running...).

WARNING: **If you have a measurement device connected to the computer, you cannot have multiple instances of Dewesoft!**

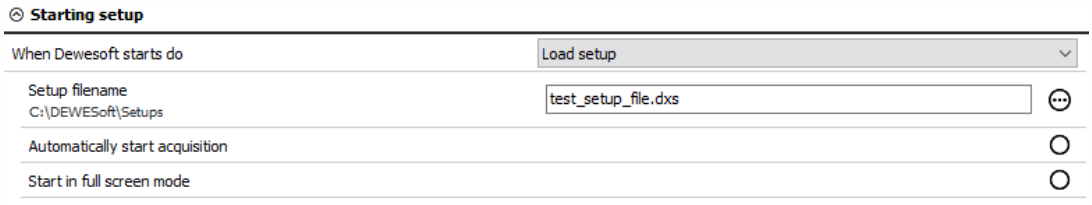


Under Starting setup options you select what to do when Dewesoft starts.

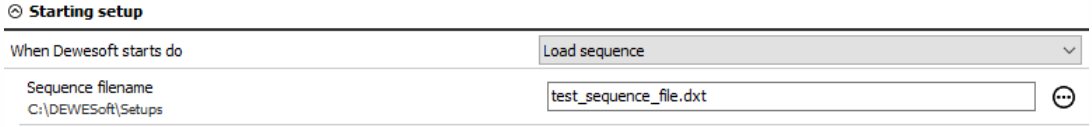
- Nothing (Dewesoft opens normally, no action is performed.)



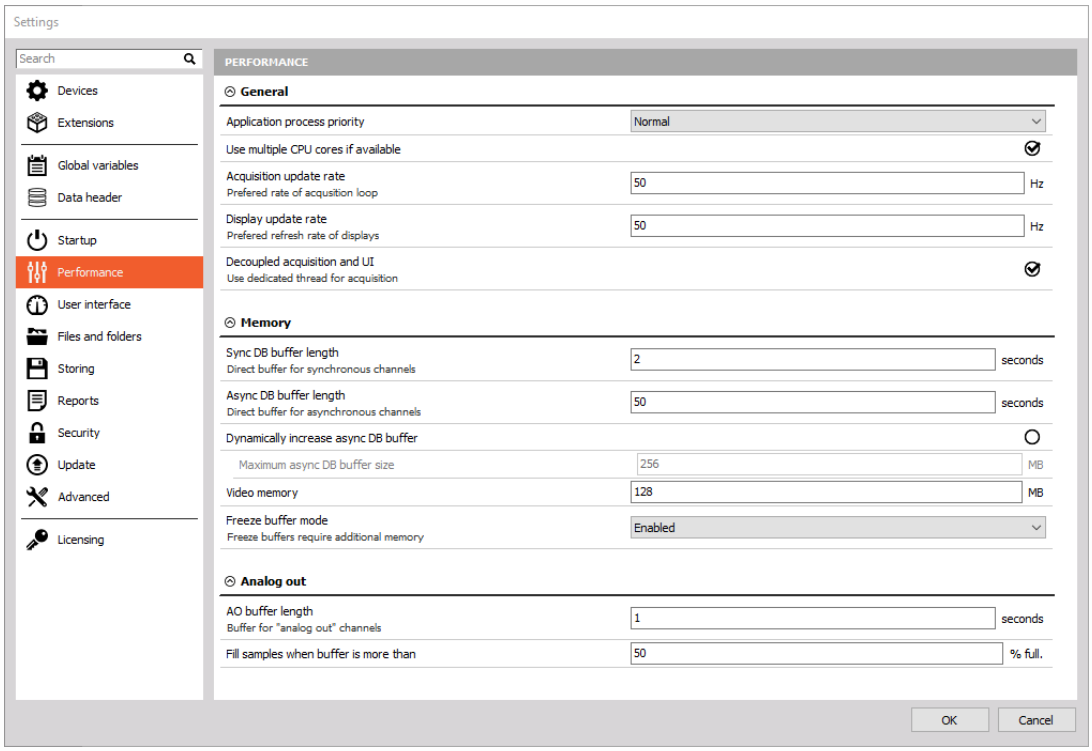
- Load setup (Dewesoft automatically loads a defined setup. We can also tell the program to go straight to Acquisition mode or to start Dewesoft in full-screen mode.)



- Load sequence (Load sequence at the start will load the sequence defined in the file name and run when Dewesoft starts.)



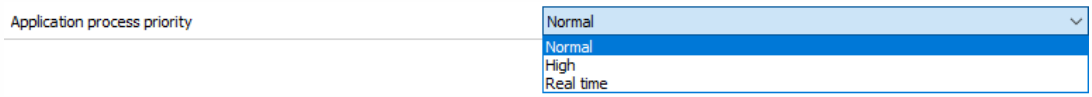
PERFORMANCE



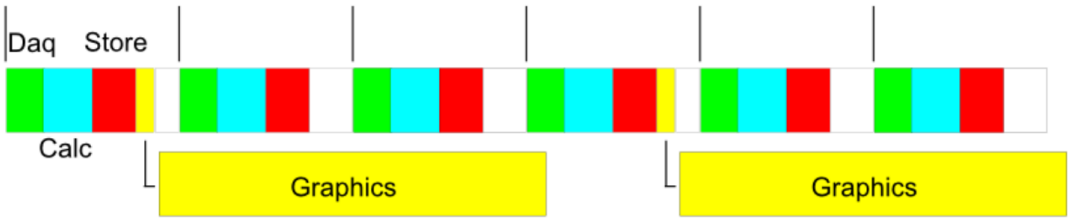
General

Application process priority defines the process priority of Dewesoft:

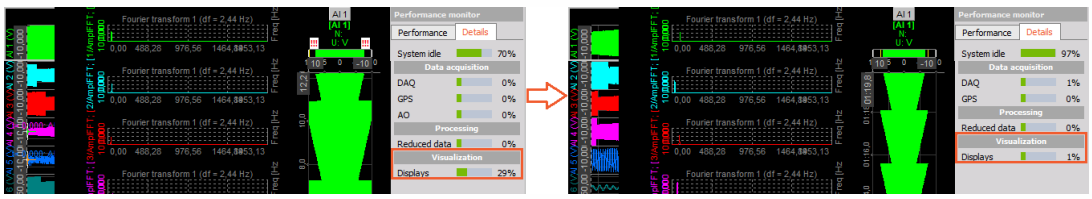
- Usually, Dewesoft should run on its own in the OS and, therefore Normal priority is enough.
- High priority allows Dewesoft to increase its performance if other processes are also running on the system.
- Real time is useful only in special applications. Even though thinking that real-time priority would be the best, but in reality, it might stall acquisition low-level drivers which must have enough CPU time. Setting Dewesoft to real-time would mean that tasks like displaying data on the screen would have to high priority.



- Use multiple cores is a very important function to split the data acquisition and math processing between different CPU cores.
- Acquisition update rate defines the preferred rate of acquisition loop. The graphics part was always a part of acquisition loop (data acquisition, calculation, storing, graphics). The graphics were drawn at the same time and that did not allow us to run a faster acquisition. Now, the graphics can be done in parallel. As the result, we could lower the priority of graphics part and have faster acquisition times. Average acquisition loop was running with about 50 Hz and now it can run up to 1000 Hz. Set the acquisition update rate higher if you want to have the faster reaction times.



- Display update rate option defines the preferred refresh rate of displays. With defining lower display update rate, we can reduce CPU load of Displays. If we change the update rate from 50 Hz to 1 Hz, the CPU load of displays dropped from 30 % to 1 %. This is very useful when we are on the limit with our computer CPU.



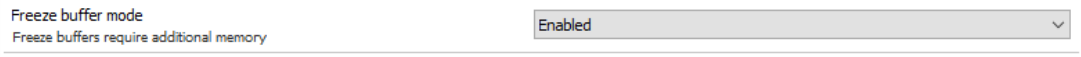
- Decoupled acquisition and UI option are enabled by default. This option will use the dedicated thread for acquisition. One core will be used for acquisition and one core will be used for the user interface.

Memory

Memory sizes are important to run the software correctly for a different application. There are sizes which need to be defined for:

- Sync DB - This is the memory size in seconds for all synchronous channels (analog, counters, ...). The value should be larger than the maximum refresh time - 2 seconds is the default.
- Async DB - This is the memory size for all asynchronous channels (CAN, GPS, and many others...). The default value is 50 seconds.
- Video Memory - Size of the video buffer; with lots of cameras the default 128 MB value should be reduced to half, for example not to run out of system memory.

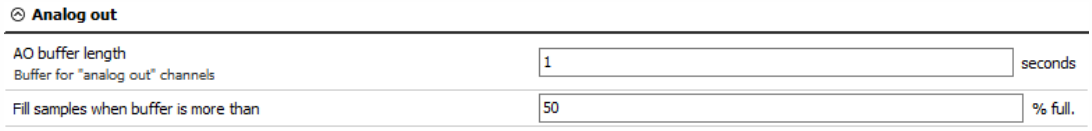
Enable Freeze buffers should be enabled if we plan to use Freeze mode (to see data during the measurement).



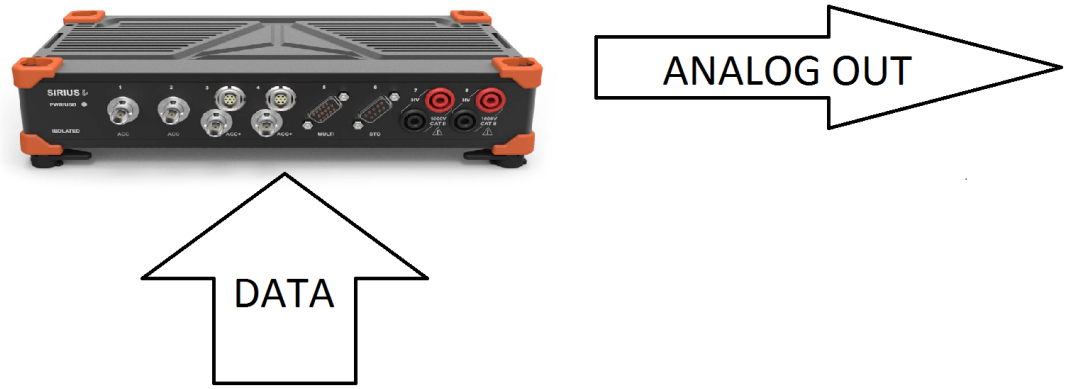
When we enable it, the Freeze button is seen during the Measure mode.

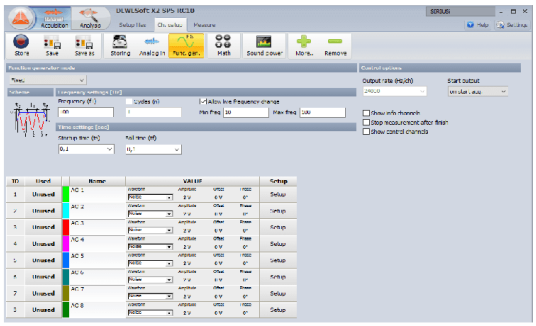
When storing the data, you will see a special Freeze button. This is a sign for Grandview, enhanced freeze mode. Grandview allows the user to review stored data from the start of a measurement without interrupting data acquisition and storing process. The user is able to zoom into any region of data already stored on disk during the measurement and review any type of signal including video, which makes (long-term) measurements easier to manage.

Analog out

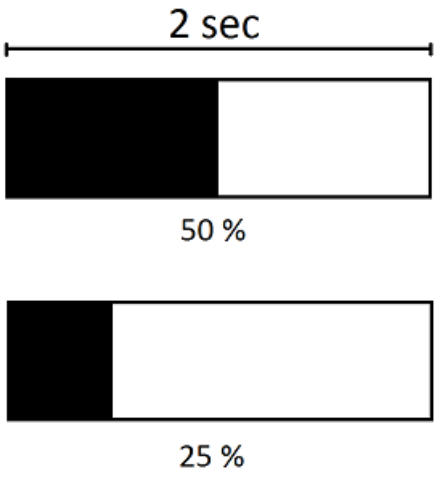


- AO buffer length - when you use the Function generator function in Dewesoft, the software sends the data to Sirius. Sirius waits for the amount of AO buffer length (in our case, this is 1 second) before it starts to output the signal. This has to be done, to prevent data loss. By default, the buffer length is set to 1 second.



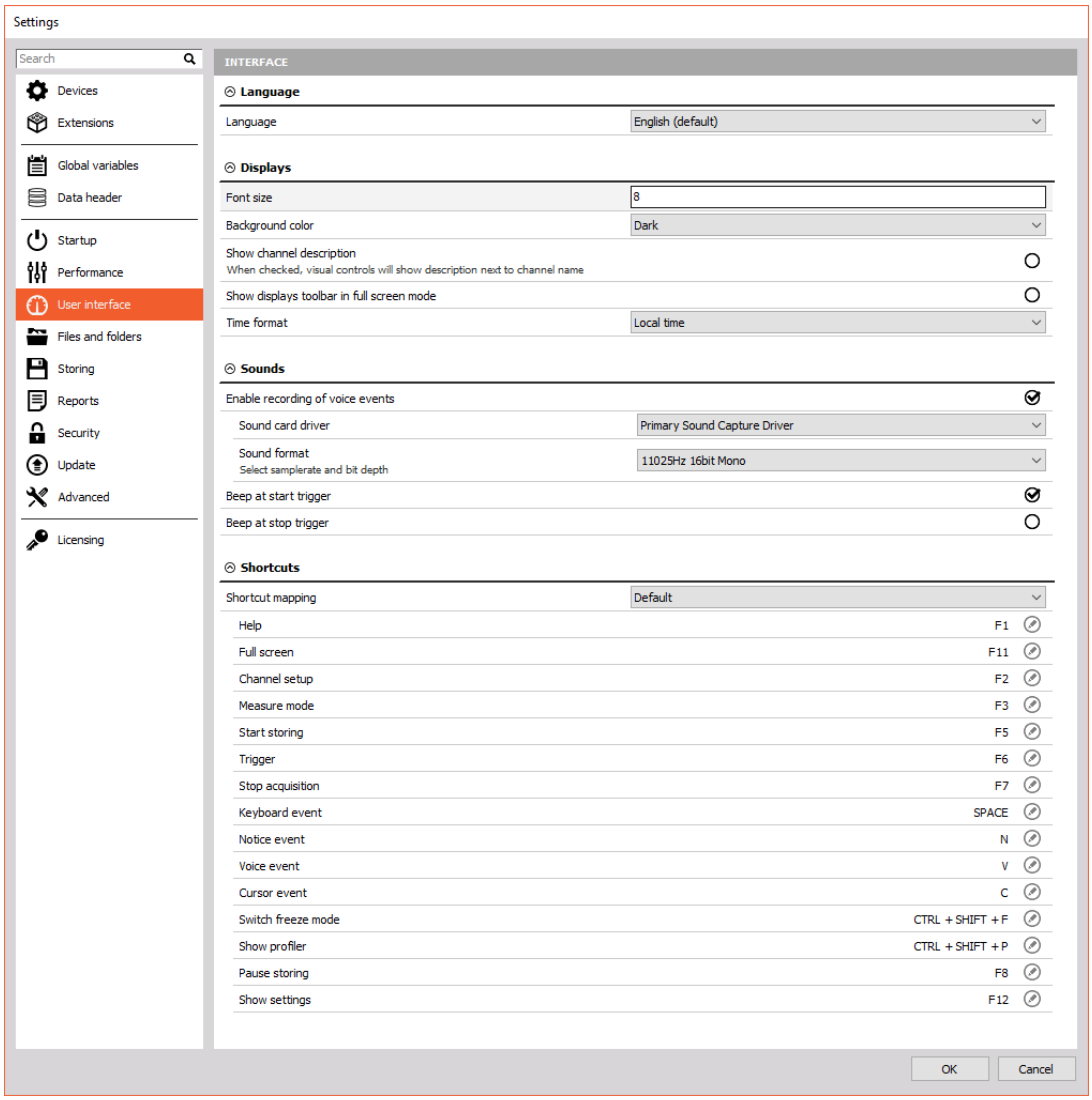


- Fill samples when the buffer is more than __ % full. Let's say that we have a buffer of 2 seconds. The samples will be sent when the buffer will be filled with the defined amount of percentage (at 2-second buffer, 50 % means, that it will wait for 1 second). By default, the value is set to 50 %.


2 sec
50 %
25 %

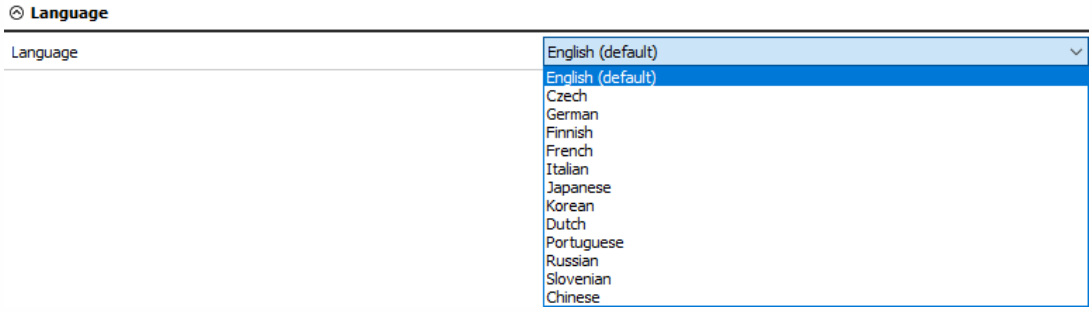
USER INTERFACE

Under User interface section, we define the language of the program, display characteristics, and sound card selection.

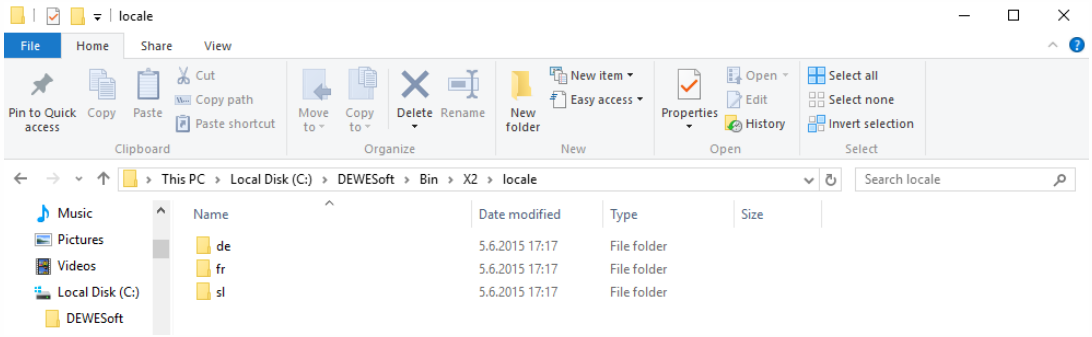


Language

Language file can be selected from the drop-down list.

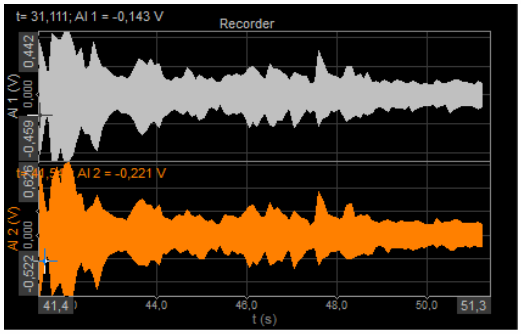


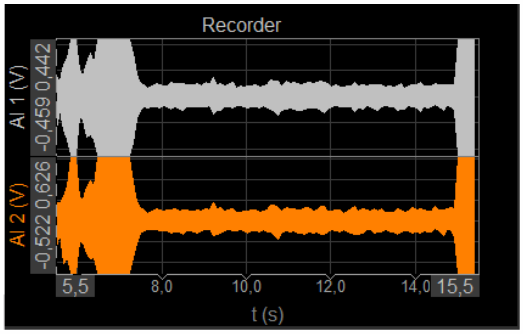
The language files can be found in a folder locale located in Dewesoft installation folder. Here, you can also add your own language files if they are written with the appropriate tool.



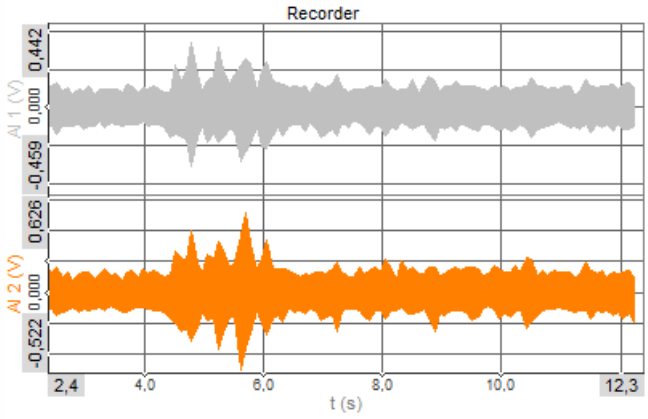
Displays

Font size defines the size of the font in measure mode (example: in the left picture, the font size is 8 and in the right picture the font size is 12).

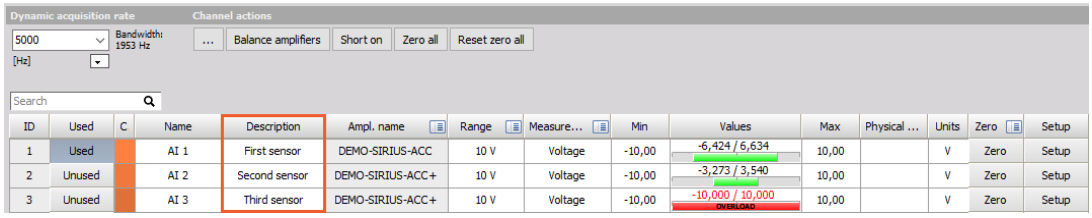


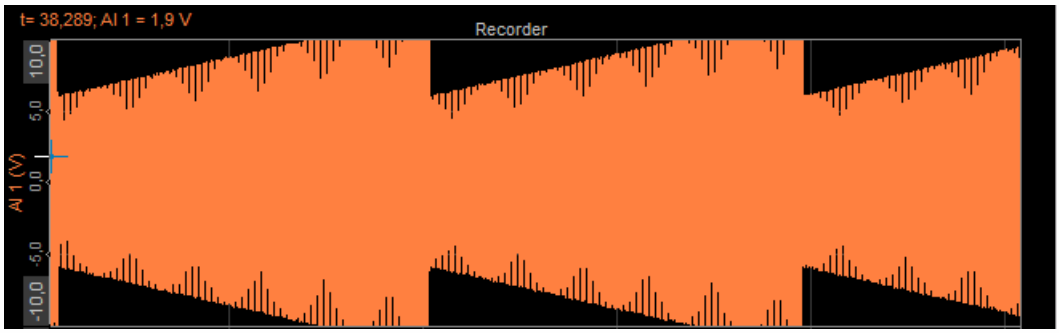


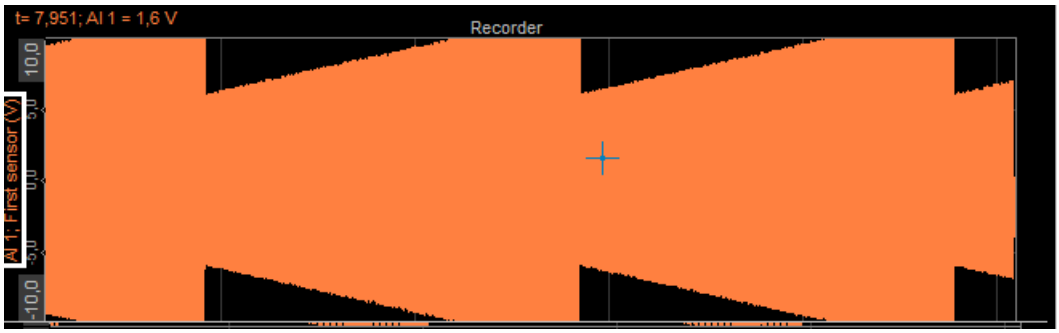
If we change the background color to White, the background of the measure mode will change.



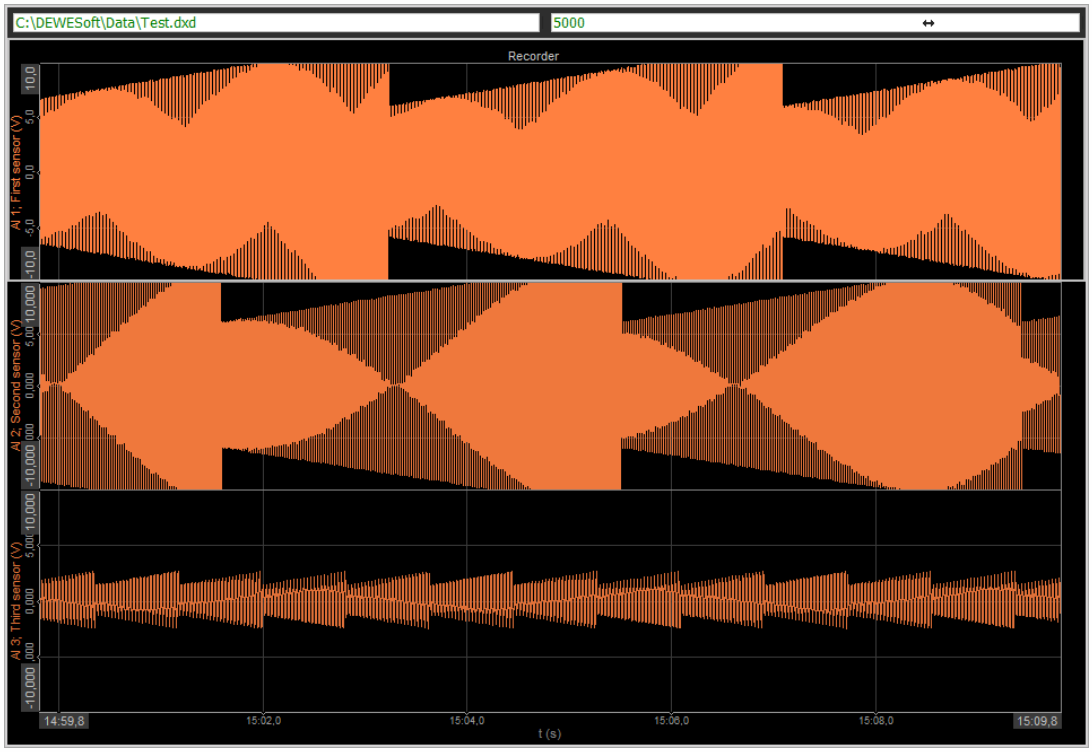
Show channel description sets that the channel description is shown in the name of the channels in visual controls (like the recorder, for example). When switched off, only names are shown.

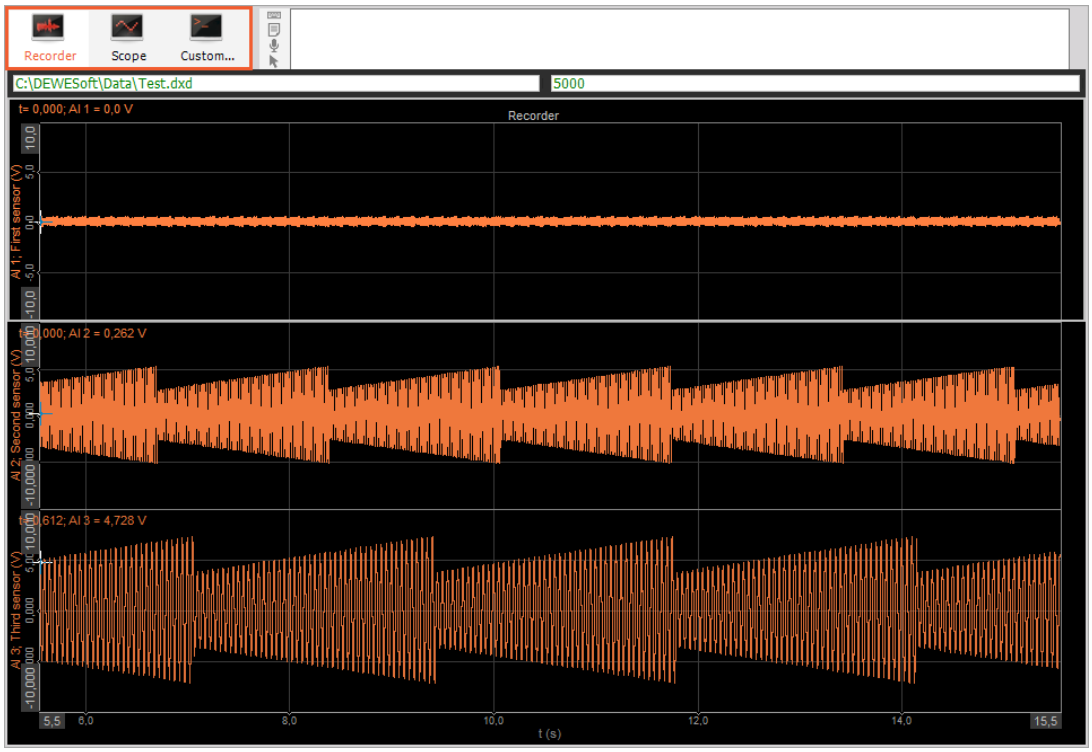




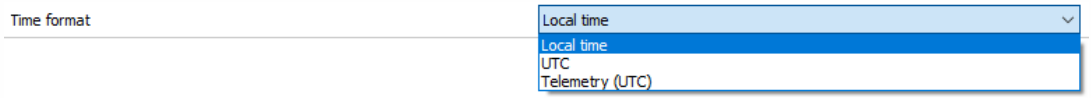


Show instrument toolbar in full-screen displays instruments button in full-screen mode; full screen (available with Ctrl-F) usually shows only instruments, but in this case, also a toolbar with standard displays are shown. In the above picture, this option is disabled and in the below picture, this option is enabled. We can see the toolbar with different screens.





For the Time axis display Local time, UTC and Telemetry (UTC) can be selected from the drop-down list:

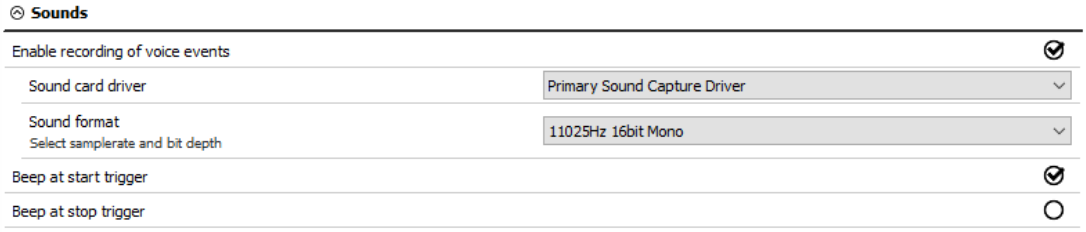


This selection will affect the absolute axis of recorder, time display in the multimeter and other displays showing absolute time. Dewesoft always stores the data in UTC format (without the local time bias).

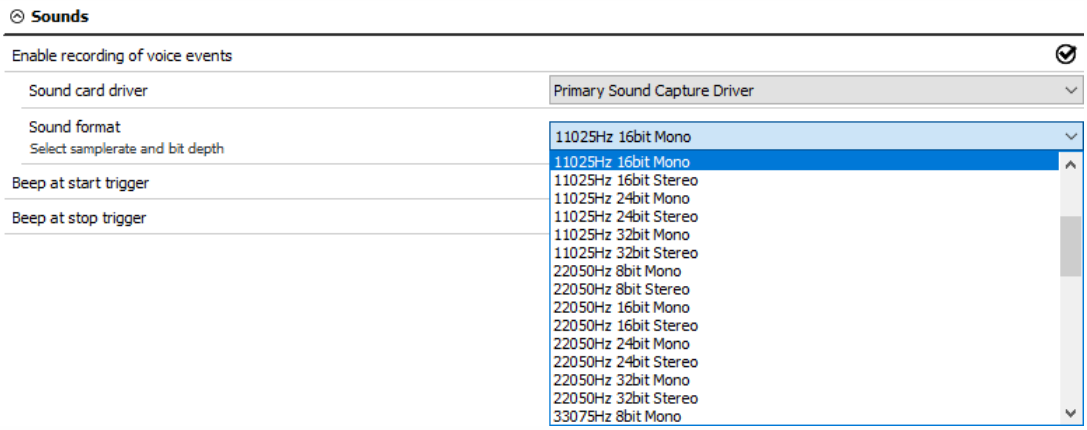
- Local time display will show with data in local time based on OS settings, therefore the same data file will have different absolute time if viewed in the USA than if it is opened in China.
- UTC will show the universal coordinated time (without bias) and will be displayed the same around the globe.
- Telemetry UTC will not display the day, month and the year, but will display the day of the year instead.

Sounds

Sound sections allow you to tell Dewesoft which capable sound card in this computer you want to use for making voice annotations when storing data.



The sound format can be chosen from the drop-down menu:

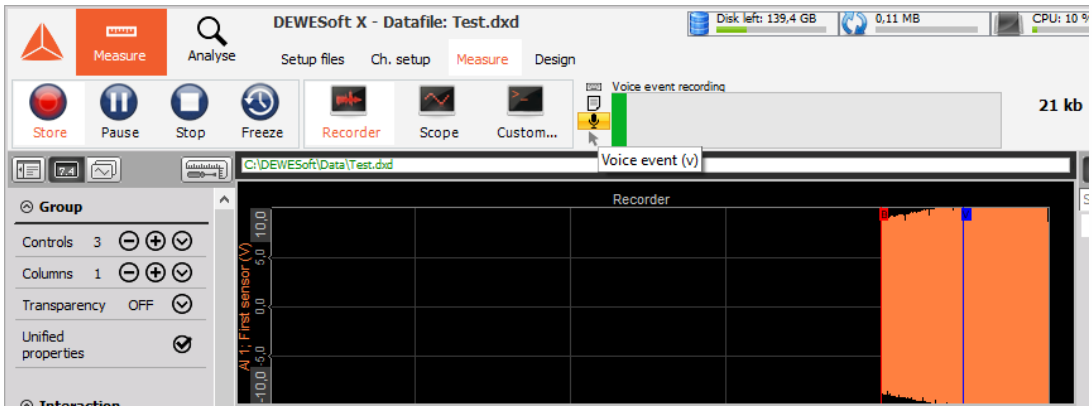


The voice events can be entered by pressing V during recording, but only if the sound card is defined. The sound at the start and stop triggers allows to have a sound indication for triggering.

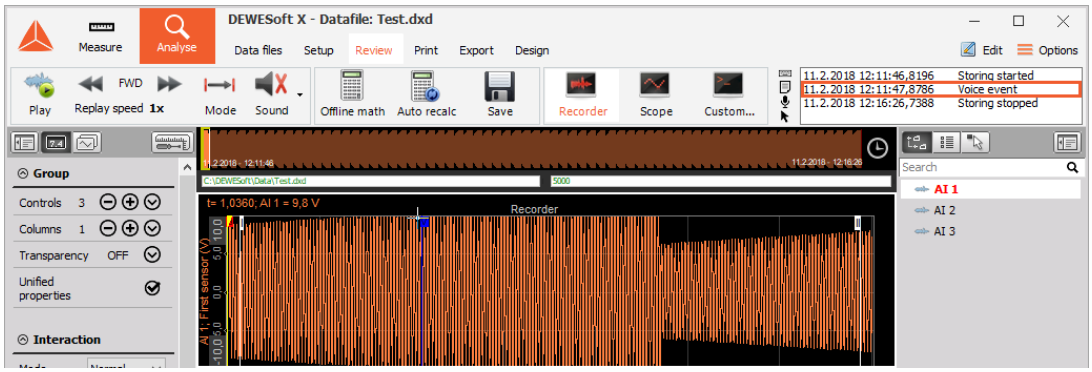
Let's make an example of voice recording. I have chosen my Microphone as a sound card driver.

When I go to Measure mode, click Store to start storing the data.

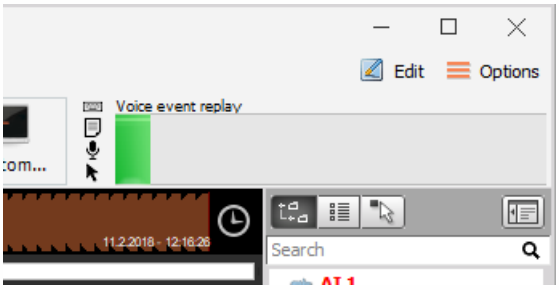
During the measurement, press the V button on the keyboard and the program will record the voice with the selected sound card.



In Analysis mode, when we open the data file, we can see a voice event under the events section.

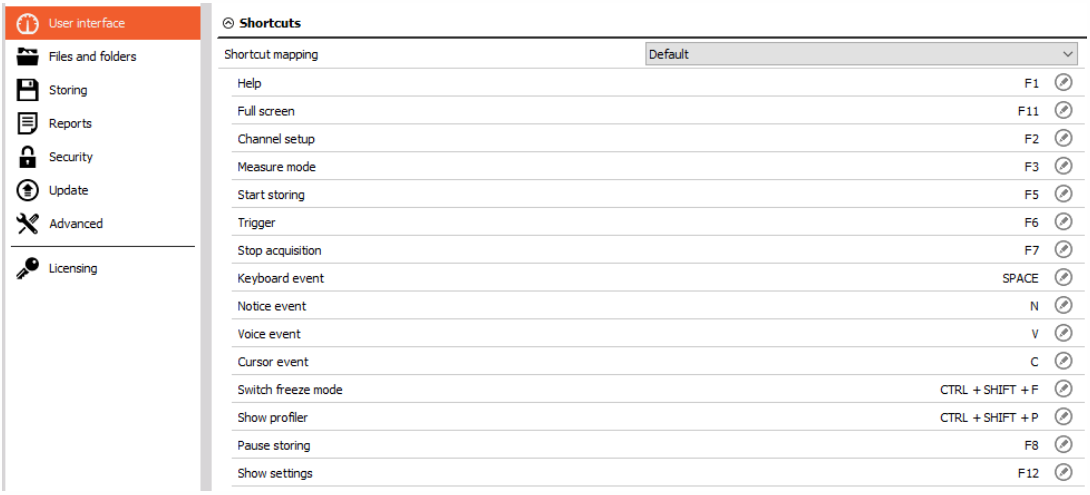


In the events window, you can see when we started and stopped the storing and when we made a voice event. If you want to replay the voice event, double-click on it. The following window will appear and the voice will be replayed.

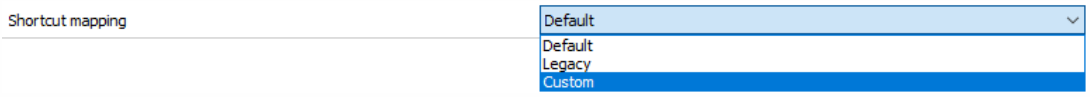


Shortcouts

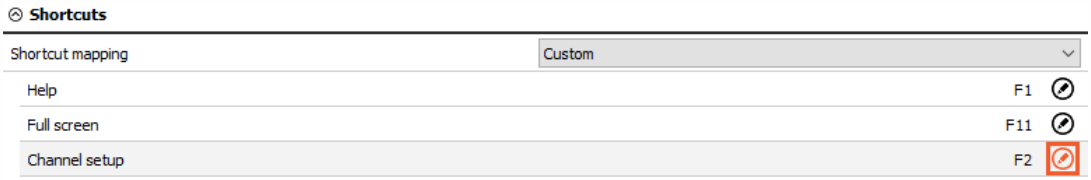
Under the shortcuts section, you can see all the available keyboard shortcuts in Dewesoft.



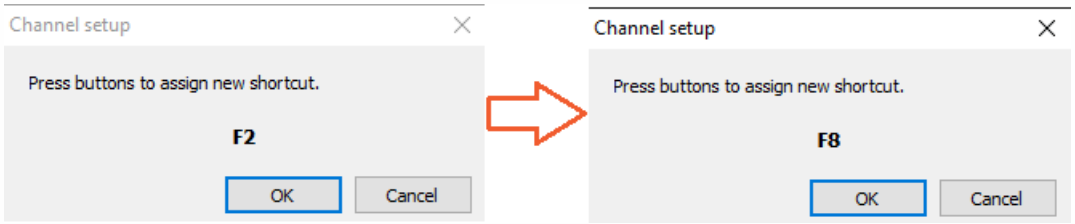
You can change the buttons that will lead you to the wanted shortcut. Select the Custom Shortcut mapping.



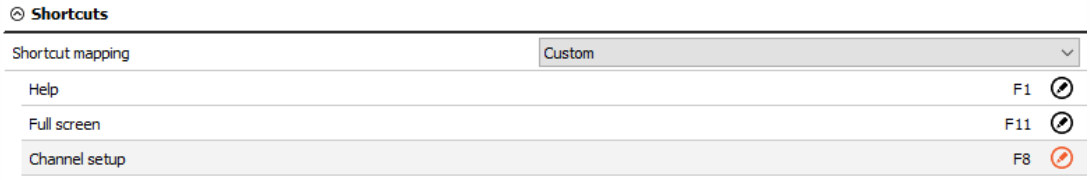
After that, a pen icon will show at every shortcut. To change the shortcut button, click on the pen icon.



The following window will appear. After assigning a new shortcut, click the OK button.

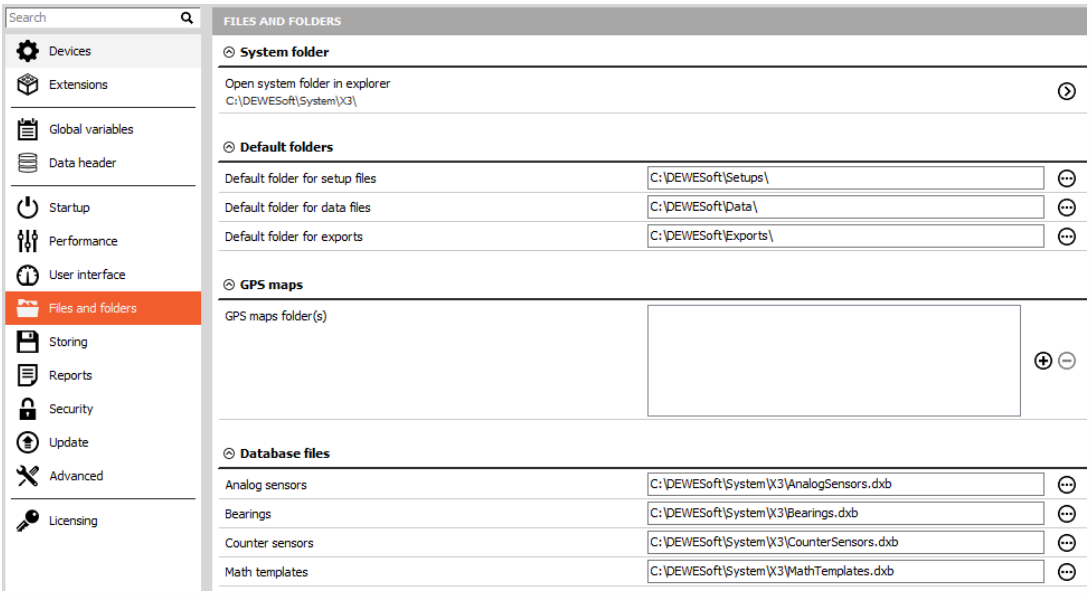


The new button for the shortcut is assigned.



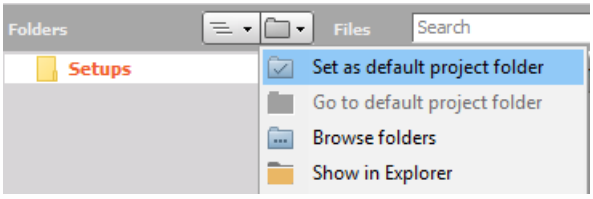
FILES AND FOLDERS

Section Files and folder defines a starting point within the folder structure.

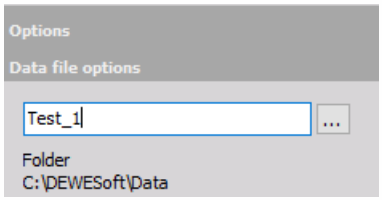


Default folders

- Default folder for setup files - Defines the starting folder where the setups will be loaded from. This default folder can be changed also from Folders menu in Measure - Setup files with setting Set as default project folder.



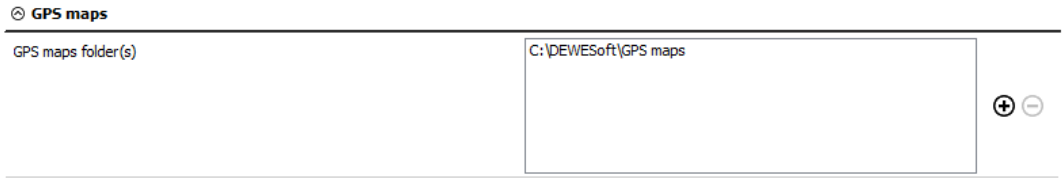
- Default folder for data files - Defines the folder where to store the data files by default. This can be also changed as above, but inside Measure - Ch. setup - Storing or Analysis - Data files.



- Default folder for exports - Defines the default place where the files are exported. If we select Remember last selected folder, then the folder which was last selected when exporting data will be set as default and remembered for next export.

GPS map

We have to define the path to GPS folder(s), that will be shown in GPS plugin in Dewesoft.

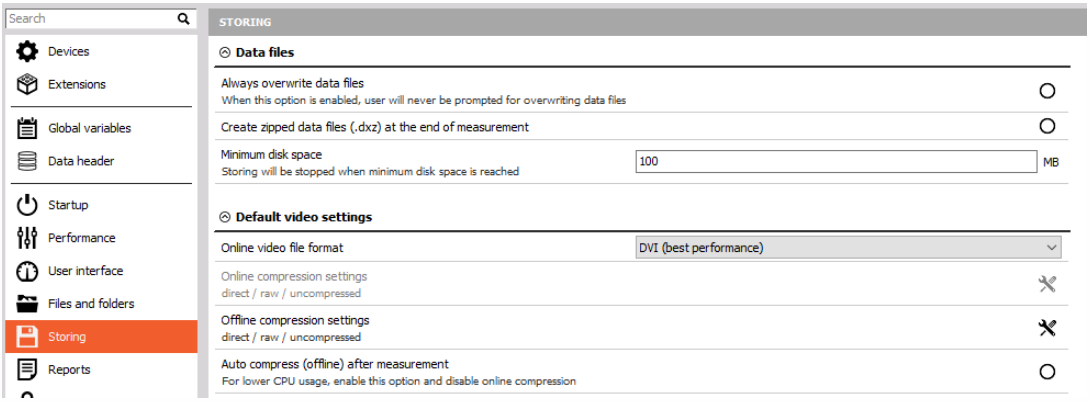


Database files

Under Database files, we define the path to different files for:

- Analog sensors
- Bearings
- Counter sensors
- Math templates.

STORING

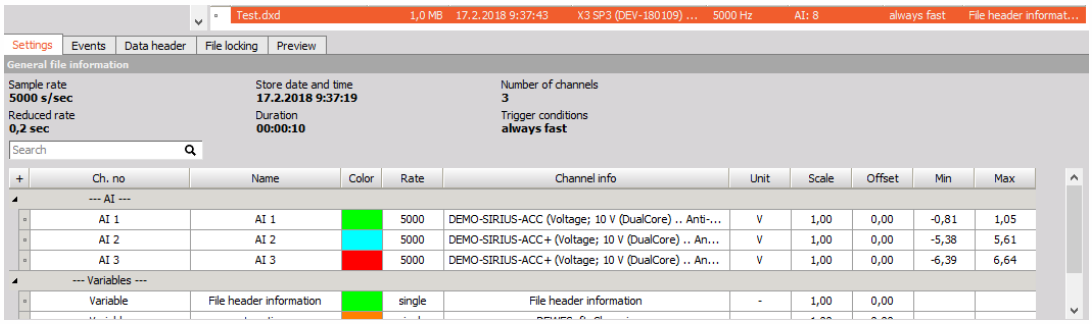

Search
Devices
Extensions
Global variables
Data header
Startup
Performance
User interface
Files and folders
Storing
Reports
STORING
Data files
Always overwrite data files
When this option is enabled, user will never be prompted for overwriting data files
Create zipped data files (.dxz) at the end of measurement
Minimum disk space
Storing will be stopped when minimum disk space is reached
100
MB
Default video settings
Online video file format
DVI (best performance)
Online compression settings
direct / raw / uncompressed
Offline compression settings
direct / raw / uncompressed
Auto compress (offline) after measurement
For lower CPU usage, enable this option and disable online compression

Data files

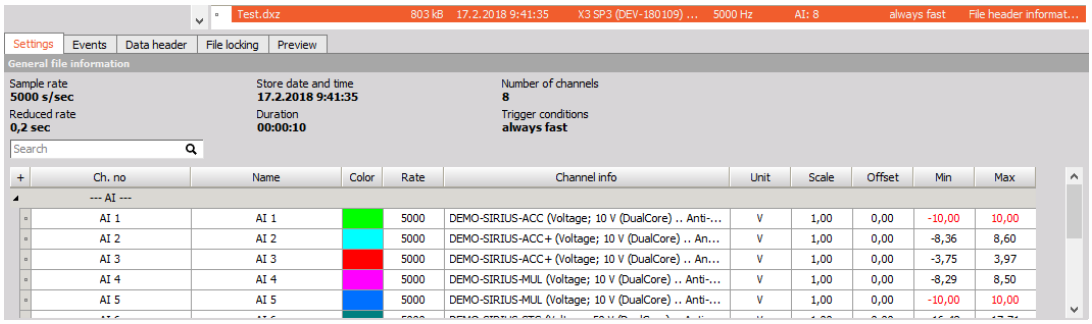
The always overwrite data files option will overwrite the data file with the same name without asking.

Dewesoft can create a zipped data files (.dxz) at the end of the measurement. The advantage of this function is, that the data file is compressed and, therefore, takes less space.

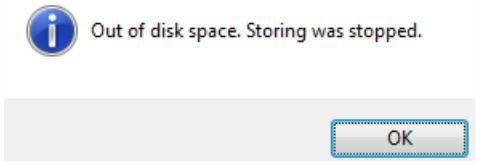
Let's make a 10-second data file with 8 channels and the sample rate of 5000 samples/second. With a normal storing, the size of the data file is 1 MB.



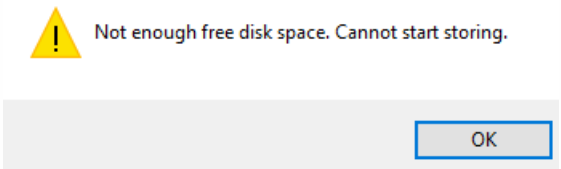
Then we select to have a zipped data file. The size of the data file got reduced.



We can also define the Minimum disk space, that is required to make a data file. If we reach this limit during the measurement, storing will be stopped and the data file will be saved.

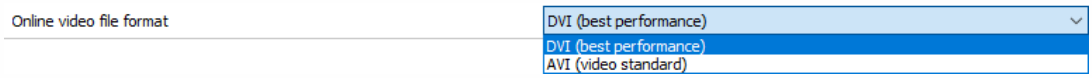


If there is not enough space from the beginning of the measurement, you will get a warning.

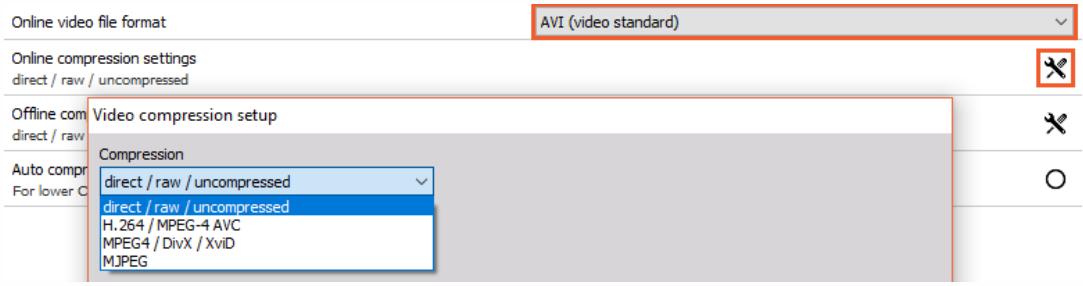


Default video settings

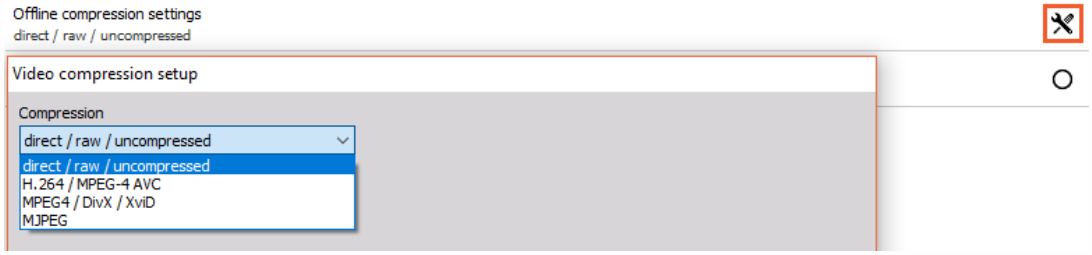
Online video file format can be selected from DVI or AVI format.



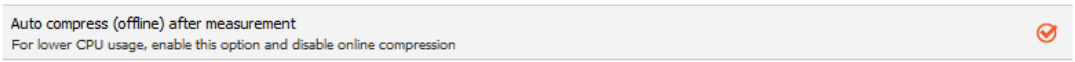
Online video compressions settings (We can compress the video during the measurement and make the data files smaller. Our computer has to have good performance characteristics to perform online video compression):



Offline video compression settings (with offline video compression we can reduce the size of the data file):

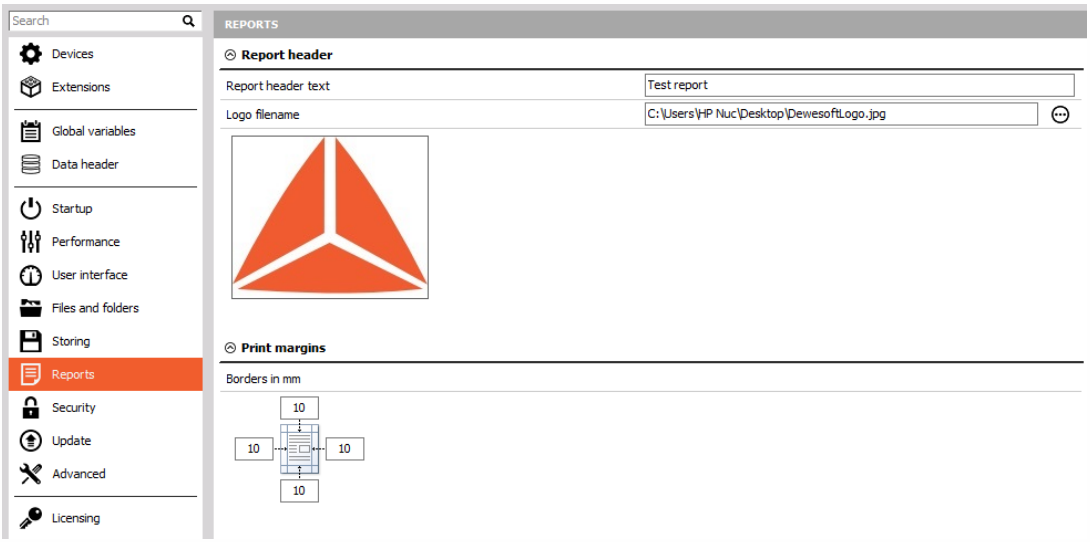


For lower CPU usage, we can enable the option Auto compress (offline) after measurement. By enabling this option, we also disable the online compression.

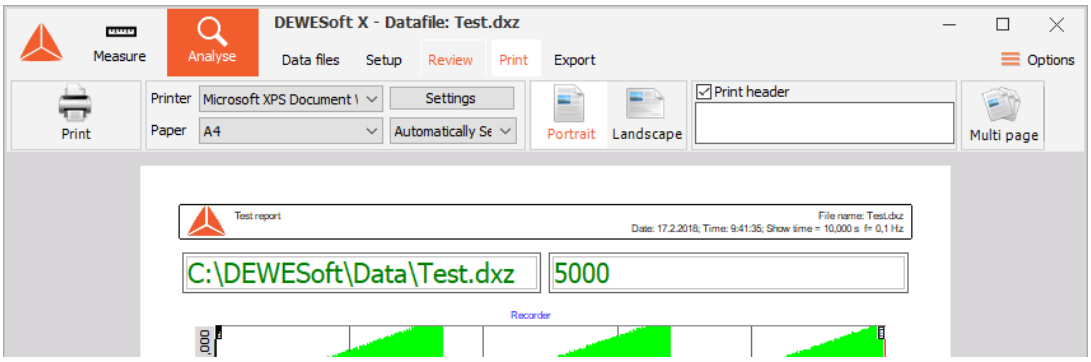


REPORTS

For the reports use your own company logo. After selecting the option to load the logo, Window explorer window appears where you simply select and image file (*.jpg file). Under Print margins, we define printer border in millimeters.



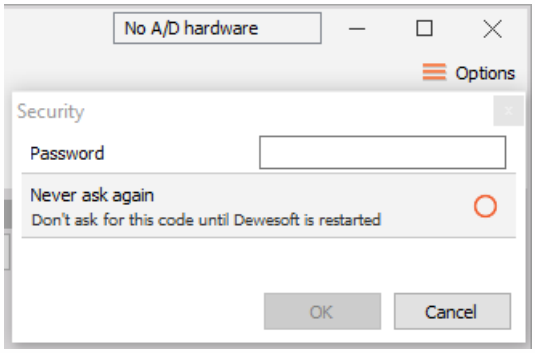
Report header is seen when we want to print the document. Go to Analysis mode, select print and on the top of the document is our report header (with text and logo).



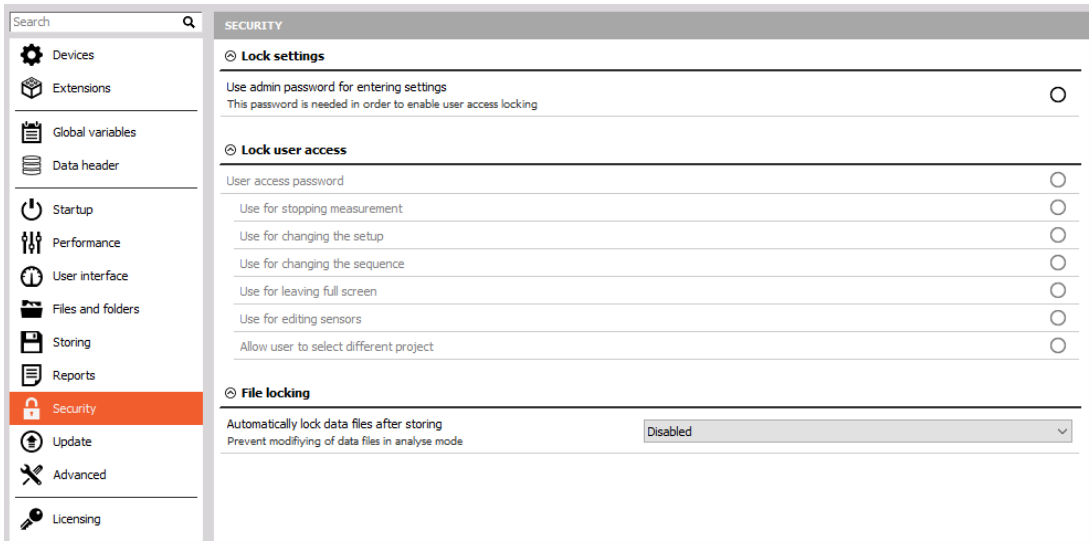
SECURITY

Under security options, you can lock the access to settings and prevent the access to Dewesoft procedures.

If we select this option, you will have to enter the password the next time you will enter the Settings.



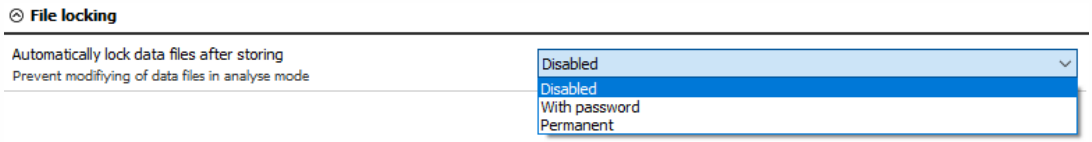
You can also select the option Never ask again. It will not ask you for a password unit Dewesoft is restarted.



If we select to use User access password, we can use it:

- for stopping measurement,
- for changing the setup,
- for changing the sequence,
- for leaving the full screen,
- for editing sensors and
- for allowing users to select different projects.

We can also define that the files being stored will be locked. We have three options:



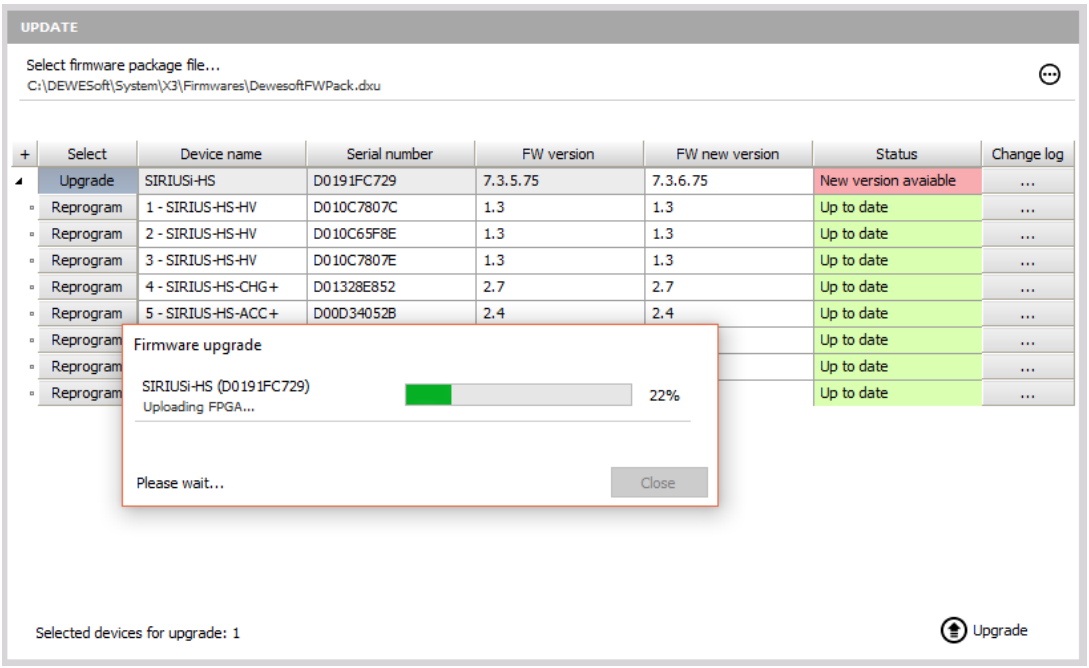
- Disabled - File locking will be disabled and the files can be post-processed.
- With password - File locking will be enabled and password will be defined to unlock the file in analyze.
- Permanent - File will be permanently locked and we can be sure it will not be modified in analyze.

UPDATE

The firmware for Dewesoft hardware can now be upgraded directly in Dewesoft software (firmware upgrade tool is no longer required).

Required equipment:

- Dewesoft instrument (SIRIUS, DEWE43, KRYPTON, ...)
- Dewesoft version X2 SP7 or above
- Dewesoft firmware upgrade package (.dxu file)



Upgrade process

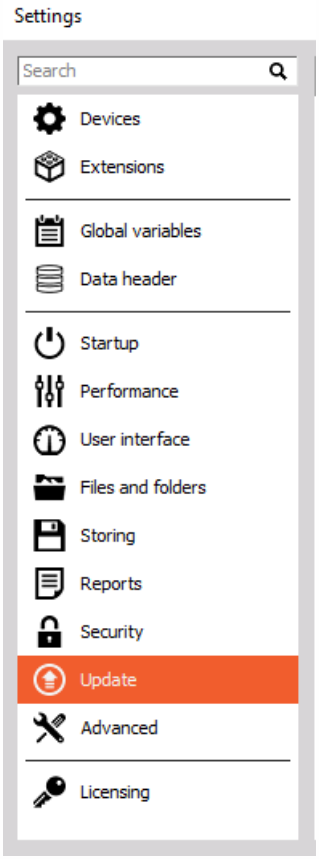
Download the Dewesoft upgrade package (.dxu file) from the Dewesoft downloads page under section drivers.

For Dewesoft manuals visit Dewesoft Web page -> Support -> Downloads

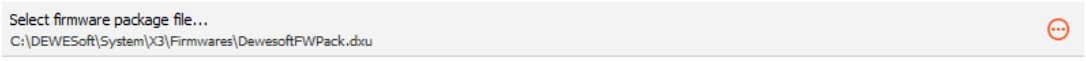
Copy the file into the Firmware folder of your Dewesoft installation (D:\DEWESoftX3\System\X3\Firmwares).

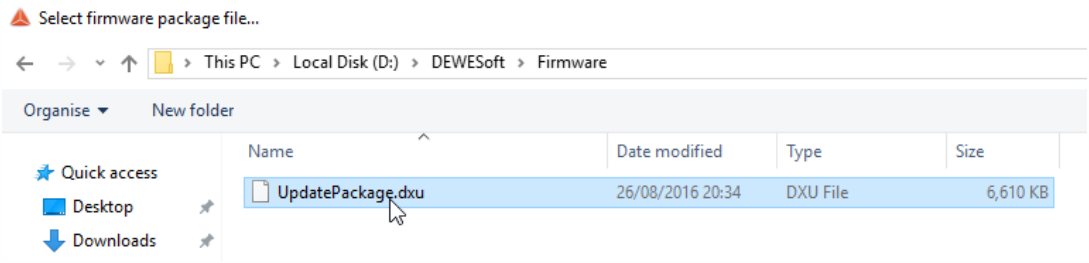
Connect the Dewesoft instrument to PC and run Dewesoft X3 software.

Go to Settings and the Update menu:



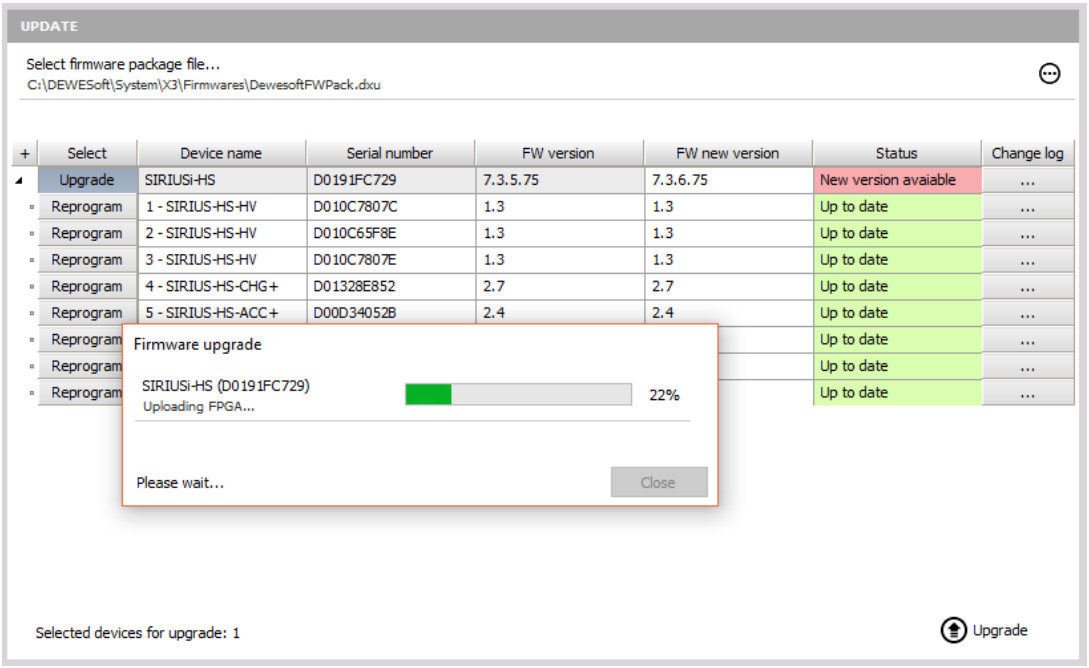
If the firmware package isn't selected, select it by pressing the three dot button and the folder with the firmware file in it.



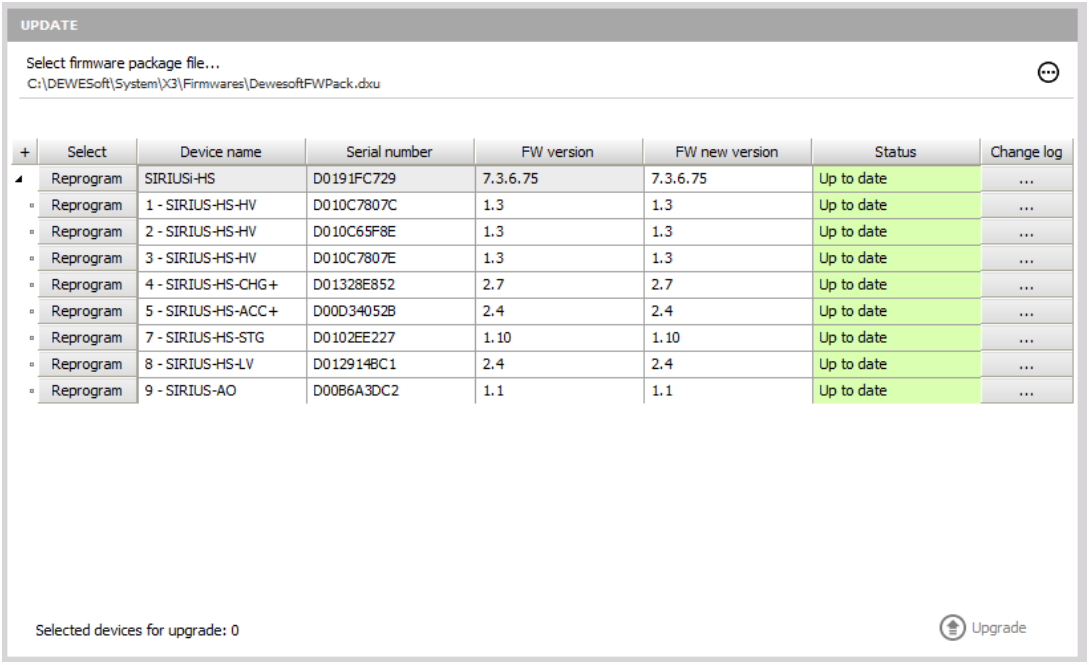


Select the device you want to upgrade and start the firmware upgrade by pressing the Upgrade button.

The window should appear indicating that the upgrade has begun:



Wait until the software finishes the upgrade and close the window. After the upgrade, the device status should turn green and say "Up to date".

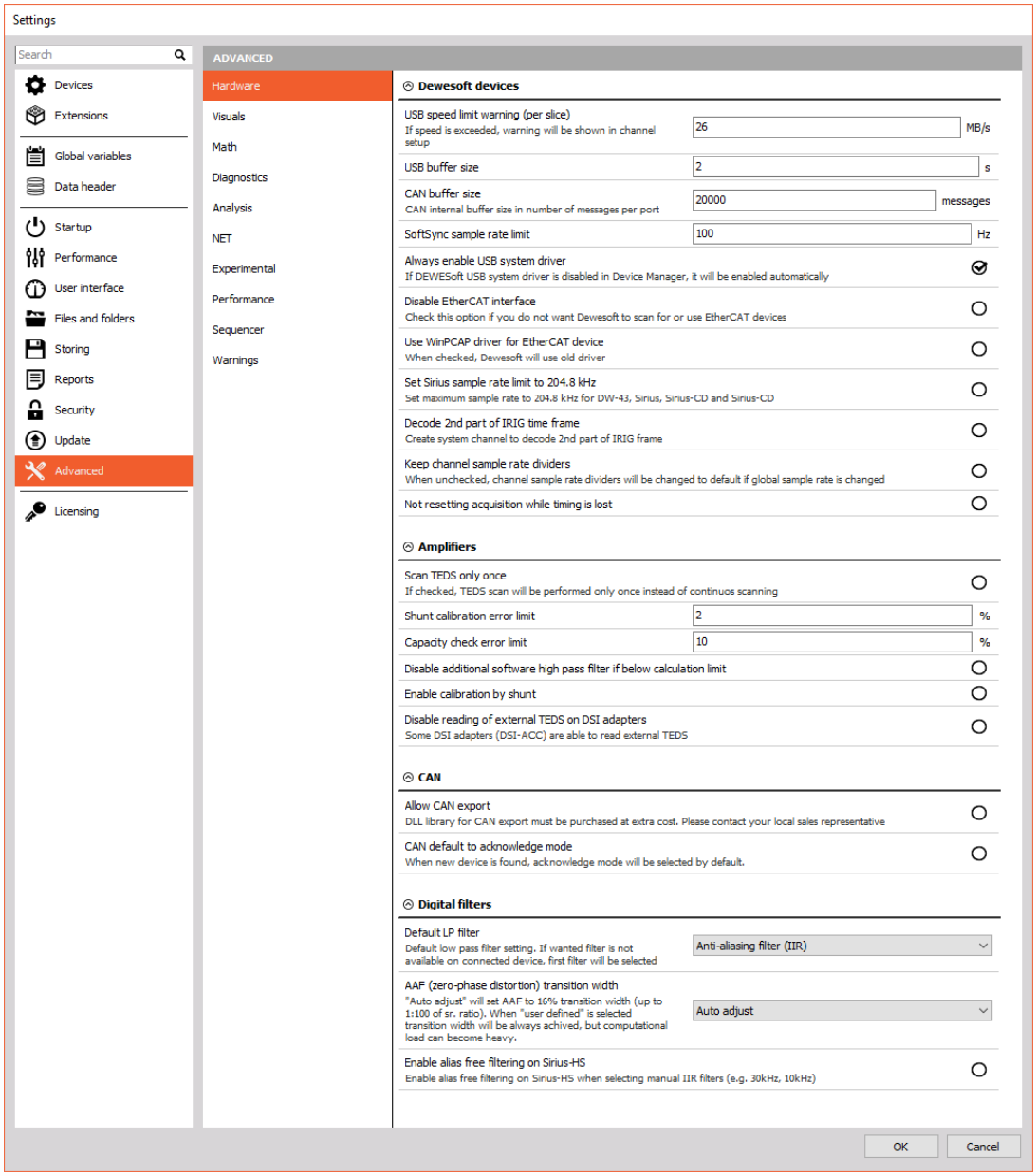


WARNING: **Do not disconnect the device during the firmware upgrade!**

ADVANCED

Here you can find advanced options for different areas, like hardware, visuals, math, diagnostics and analysis.

Hardware



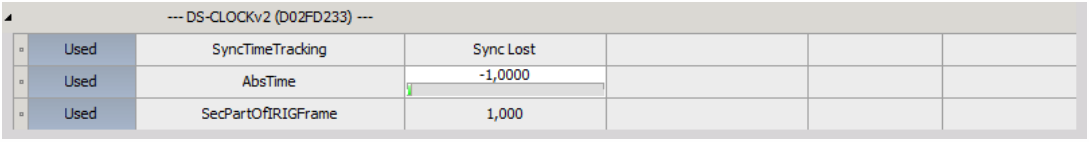
Dewesoft devices

- USB speed limit warning (per slice) - Here you can define the USB speed limit per slice. If the speed is exceeded, the warning will be shown in the channel setup.

- SoftSync sample rate limit - We can define the maximum sample rate limit if two devices are

synced with SoftSync (e.g. KRYPTON). The software synchronization accuracy is around 2-10 ms, which is enough for a simple temperature measurement. This synchronization solution requires no additional hardware.

- Always enable USB system driver - If Dewesoft USB system driver is disabled in the device manager, it will be enabled automatically.
- Disable EtherCAT interface - Check this option if you do not want Dewesoft to scan or use EtherCAT devices.
- Use WinPCAP driver for EtherCAT device - With this option enabled, Dewesoft will use older EtherCAT drivers.
- Set SIRIUS sample rate limit to 204.8 kHz - Set maximum sample rate to 204.8 kHz for DW-43, Sirius, Sirius-CD and Sirius-HD.
- Decode 2nd part of IRIG time frame - DS-Clock can now decode also 2nd part of IRIG signal. Once this option is enabled, a new channel is shown in the system monitor.



- Keep channel sample rate dividers - If the option is enabled the sample rate divider the sample rate dividers will stay applied.

Disabled:

Global sample rate[Hz]:	Channel sample rate[Hz]:
1000	500
500	500

Enabled:

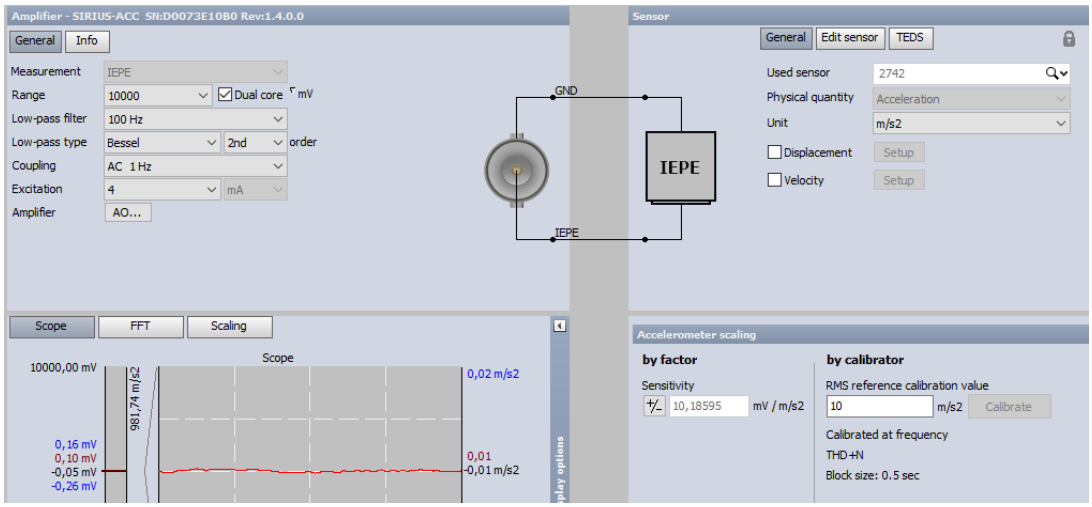
Global sample rate[Hz]:	Channel sample rate[Hz]:
1000	500
500	250

Not resetting acquisition while timing is lost - with this option enabled, Dewesoft will keep recording even if the timing is lost.

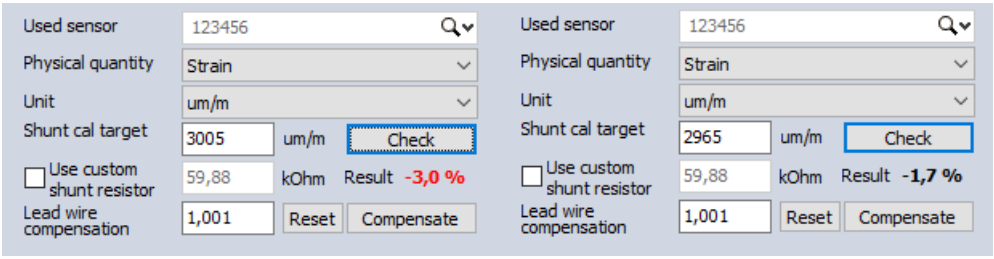
Amplifiers

When we have sensors with TEDS chip, we have to scan it. If we enable this option, we scan for TEDS chip only once or we can continuously scan for TEDS sensors and leave this option disabled.

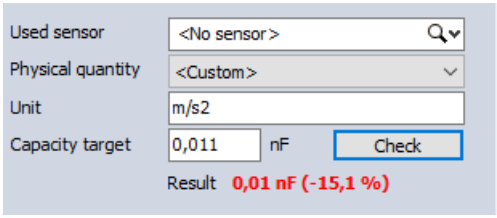
When the sensor is read, we can see the serial number, settings of the amplifier and the sensitivity of the sensor.



Shunt calibration error limit (works on STG, STGM modules) - The Shunt calibration can be used to measure and correct the resistance of strain gauges. Shunt calibration takes a while since it has to set several configurations and measure back the results. With Shunt calibration error limit, we define the maximum allowed deviation from the Shunt cal target value. If the value is more than 2%, the results are not OK and if the result is under 2%, the result is accepted.



Capacity check error limit (works on CHG modules) - If we have connected charge sensors, we can perform the Capacity check. We have to enter the known value for Capacity target. After the value is inputted, we press check. Dewesoft will generate a sine wave and check what is the actual capacity. After this procedure is finished, Dewesoft will give us the actual capacity and the error. If the value is bigger than our defined value, it will be written in red color.



Option Enable calibration by the shunt enables you to trust your shunt calculation result and use it for further measurement.

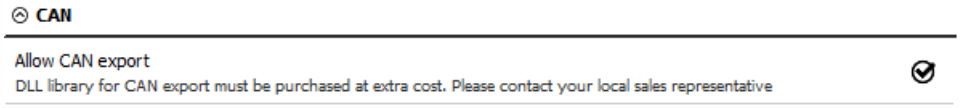
Disable additional software high pass filter if below calculation limit * If this check-box is checked and the limit of SW-HPFilter is reached, no additional SW-HPFilter is calculated. The SWHPFilter is needed for slow accelerometers to see also very low frequencies, and offset is ignored. If this check-box is not checked and the limit of SW-HPFilter is reached, the SW-HPFilter is calculated based on the limit possible offset can bother customers more than the very low frequencies missing). In the end, it depends on the sensor used and what the users want to see from it. This

option is special for some customers who have MSI-ACC (or DSI-ACC). Also, some accelerometers in IEPE mode might still have some offset even if the amplifier has an HW-HPFilter. To remove this offset, we calculate a HPFilter frequency based on this formula: Frequency(SW-HPFilter) = Frequency(HW-HPFilter) / 10. For most cases, this does not make any problems, but there are MSI-ACC which have an HW-HPFilter at 0.08 Hz. Depending on the sample rate, we can easily run into the limit of SW-HPFilter calculation where it is simply not possible to calculate it anymore.

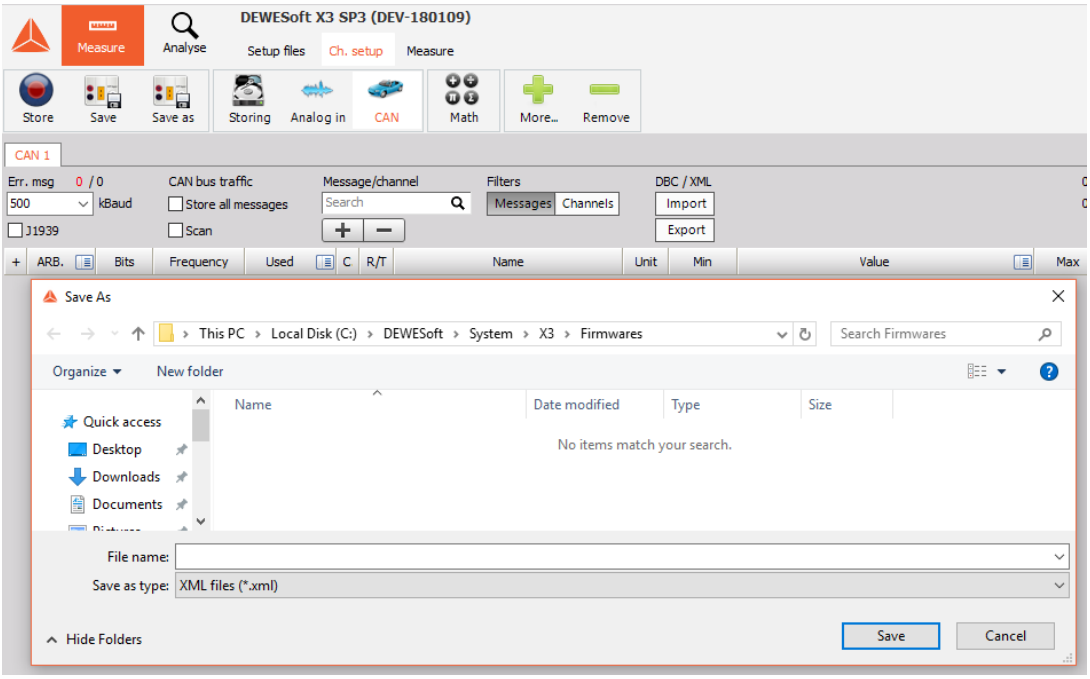
Disable reading of external TEDS on DSI adapters - the DSI ACC can read the TEDS also from the accelerometer. If this option is enabled, the TEDS from accelerometer won't be read.

CAN

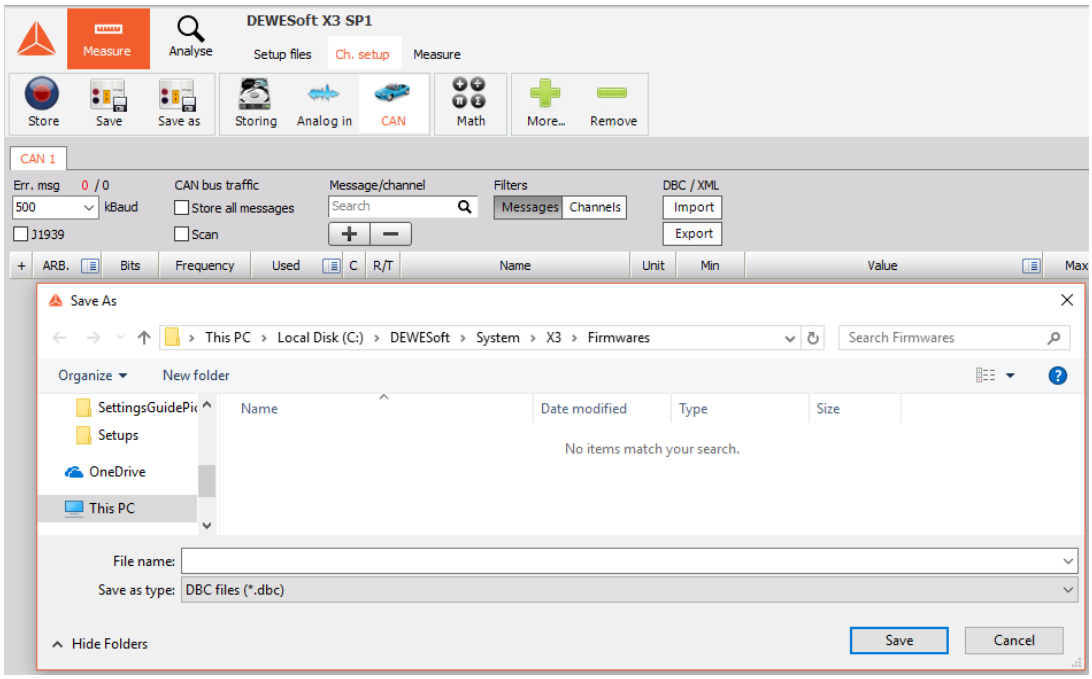
Allowing CAN export to DBC files must be purchased at extra cost. Please contact your local sales representative.



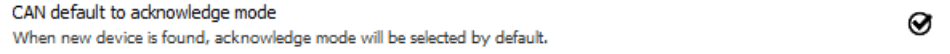
Without allowing the CAN export, you can export CAN data to the XML file.



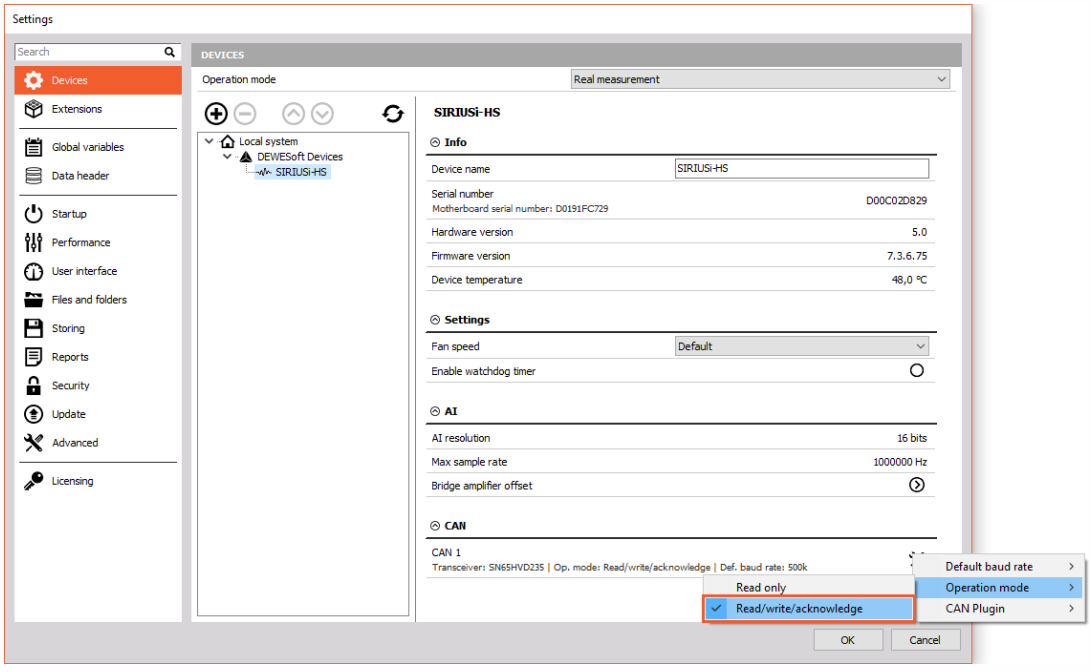
After purchasing the DLL library for CAN export, we can export data to DBC file format.



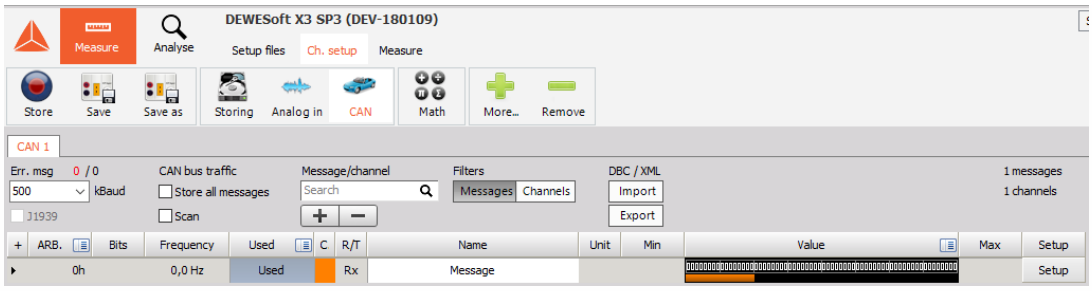
CAN can be set to acknowledge mode by default when a new device is found.



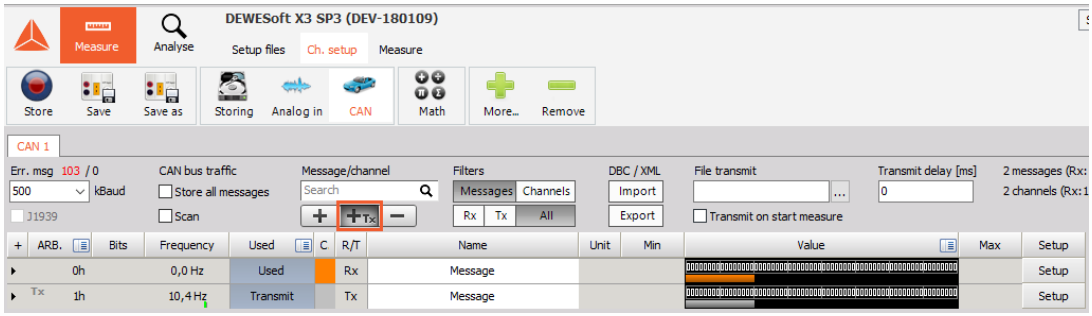
We can see the CAN operation mode when selecting the settings of a measurement device.



In the Read only mode, DAQ device can only read messages from CAN data.



In Read/write/acknowledge mode, DAQ device can read and send CAN data. To add a new transmit channel, click the icon in a red square.



Digital filters

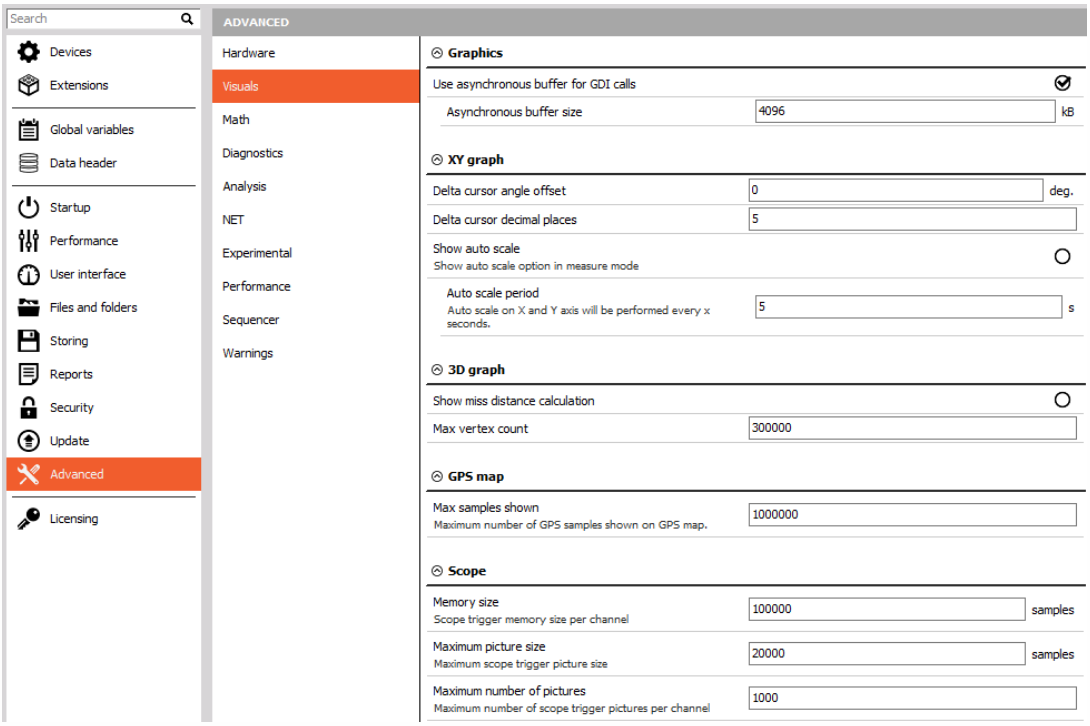
- Default LP filter setting - If the wanted filter is not available on the connected device, the first filter will be selected:
- OFF
- Anti-aliasing (IIR)
- AAF (zero-phase distortion)

- Ring free
- AAF (zero-phase distortion) transition width
- Auto adjust will set the AAF to 16% transition width (up to 1:100 of sample rate ratio).

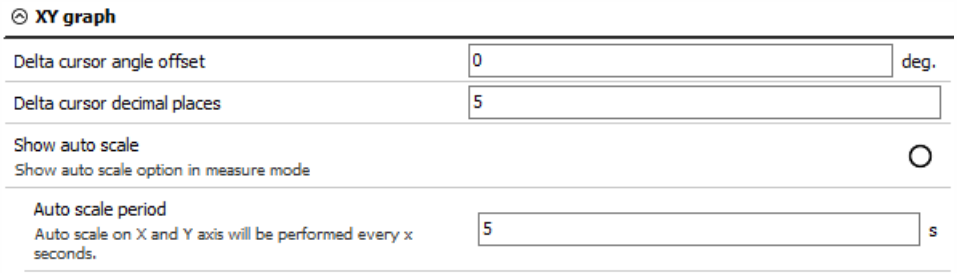
- User defined - The transition band will always be achieved, but computational load can become heavy.

- Enable alias free filtering on Sirius-HS - This option enables additional AAF has a cut-off frequency at 80 kHz, when the sample rate is equal or above 160kHz or a cut-off frequency of s/2 when the sample rate is under 160 kHz.

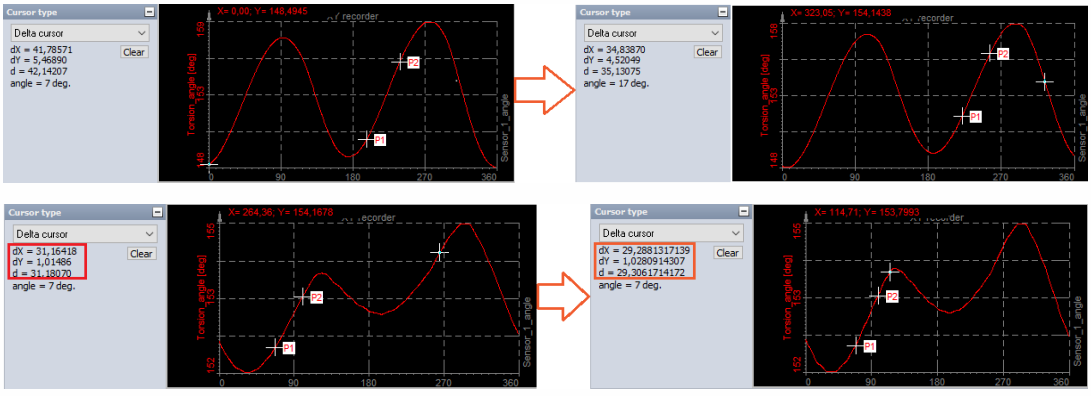
Visuals



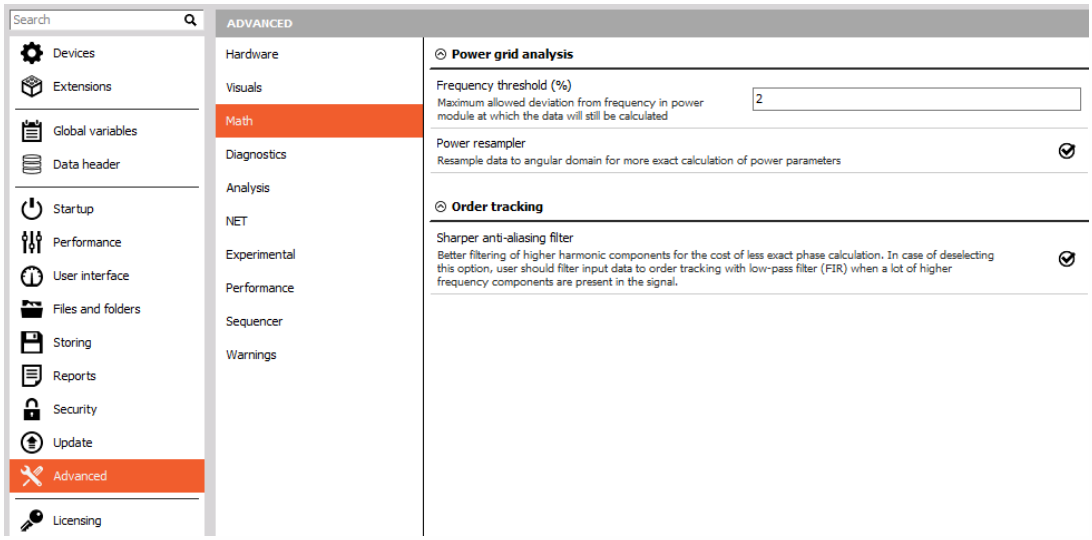
In the XY graph, we define delta cursor angle offset and delta cursor decimal places.



In the left picture, we see that the angle of the line between the two points is 7 degrees. Let's change the angle offset to 10 degrees. Now we can see, that the angle of the line between the same two points is 17 degrees.



Math

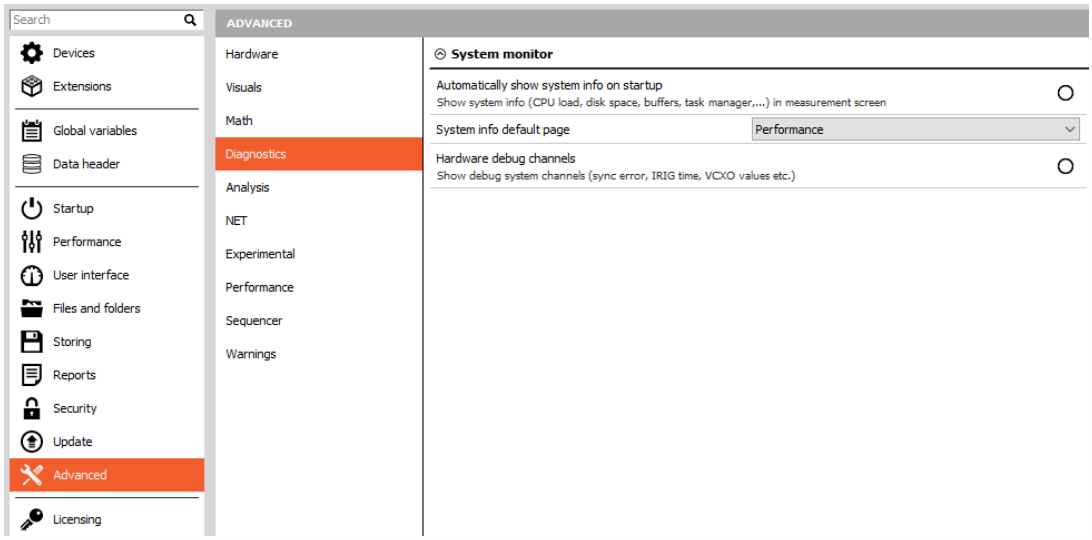


Under Math section, we define the maximum allowed deviation from the frequency in power module at which the data will still be calculated. For example, if our frequency is 50 Hz and the frequency threshold is set to 10%, the program will calculate in the range from 45 Hz to 55 Hz.

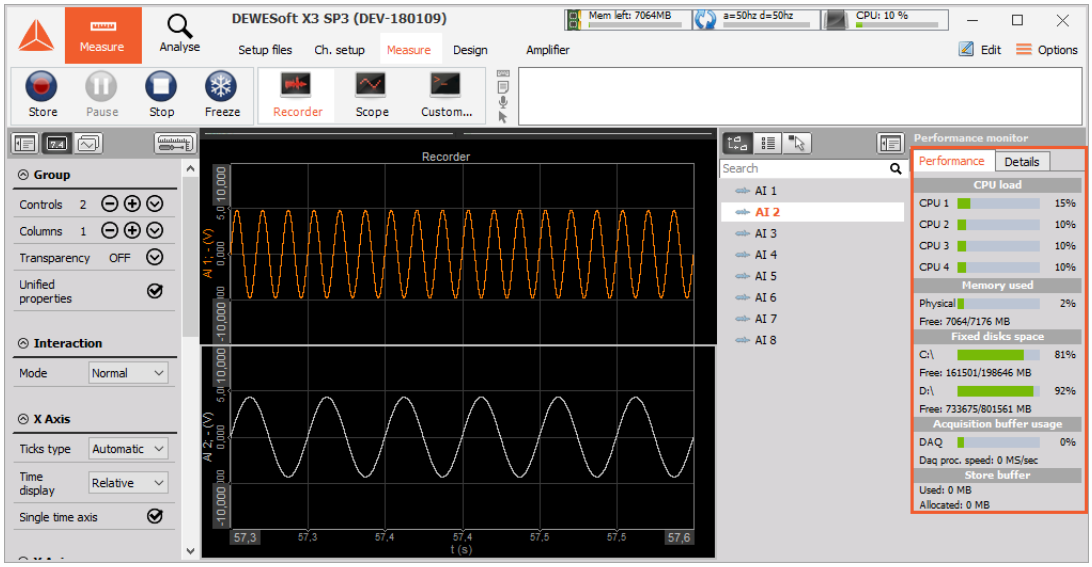
Power re-sampler resamples the data to the angular domain for more exact calculation of power parameters.

Sharper anti-aliasing filter is an option which enables better filtering for higher harmonic for the cost of less exact phase calculation.

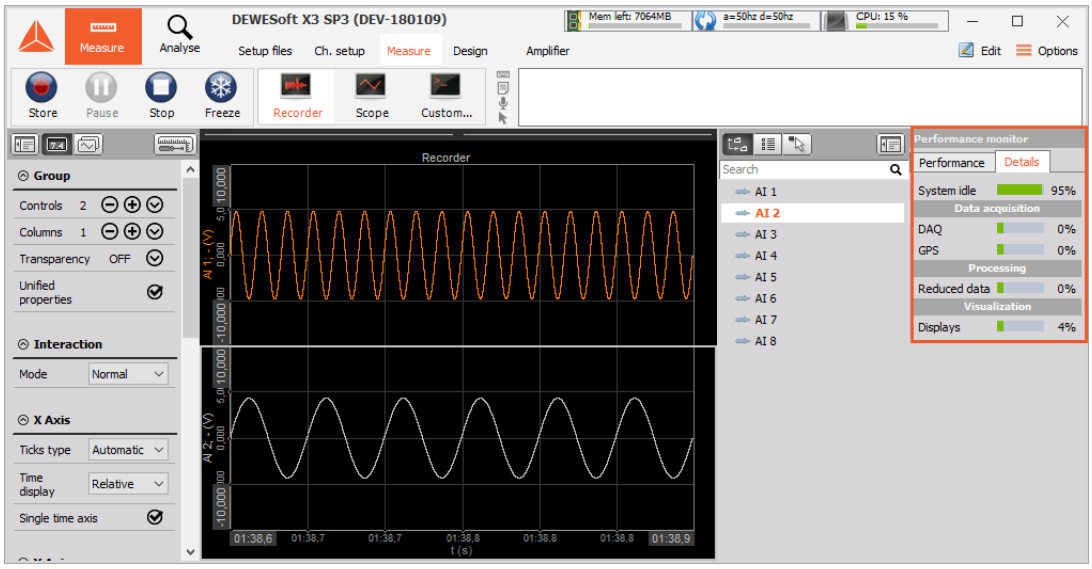
Diagnostics



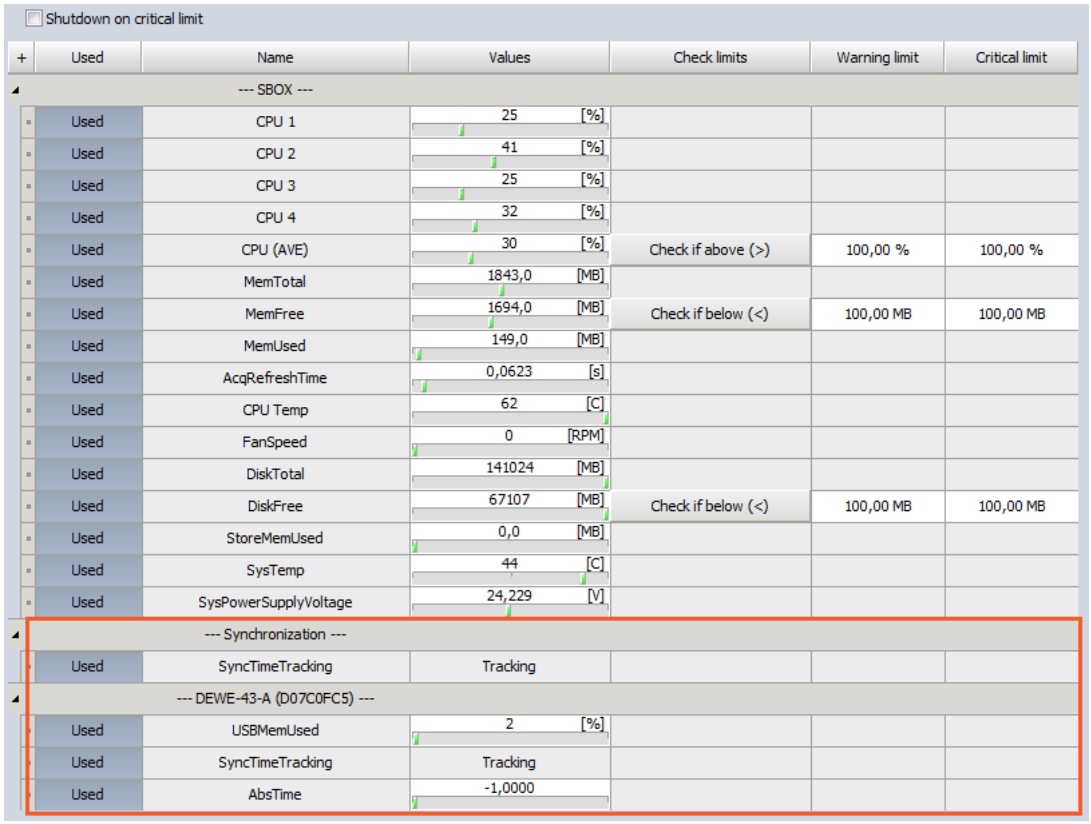
When we are in the measure screen we can access the system performances by pressing a shortcut CTRL+SHITF+P. Here, we can set that as the default part of the screen.



System info default page can be selected as Performance or Task manager



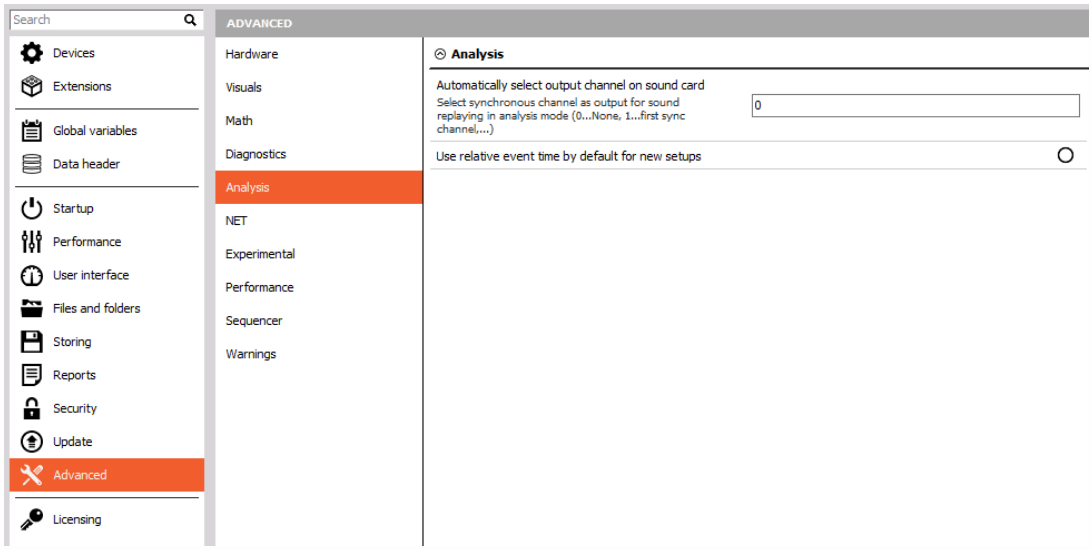
If we have measurement hardware connected to our computer, we can also show debug system channels (sync error, IRIG time, VCXO values, ...). These settings can also be seen under System monitor in Channel setup.



Shutdown on critical limit

+	Used	Name	Values	Check limits	Warning limit	Critical limit
		--- SBOX ---				
	Used	CPU 1	25 [%]			
	Used	CPU 2	41 [%]			
	Used	CPU 3	25 [%]			
	Used	CPU 4	32 [%]			
	Used	CPU (AVE)	30 [%]	Check if above (>)	100,00 %	100,00 %
	Used	MemTotal	1843,0 [MB]			
	Used	MemFree	1694,0 [MB]	Check if below (<)	100,00 MB	100,00 MB
	Used	MemUsed	149,0 [MB]			
	Used	AcqRefreshTime	0,0623 [s]			
	Used	CPU Temp	62 [C]			
	Used	FanSpeed	0 [RPM]			
	Used	DiskTotal	141024 [MB]			
	Used	DiskFree	67107 [MB]	Check if below (<)	100,00 MB	100,00 MB
	Used	StoreMemUsed	0,0 [MB]			
	Used	SysTemp	44 [C]			
	Used	SysPowerSupplyVoltage	24,229 [V]			
		--- Synchronization ---				
	Used	SyncTimeTracking	Tracking			
		--- DEWE-43-A (D07C0FC5) ---				
	Used	USBMemUsed	2 [%]			
	Used	SyncTimeTracking	Tracking			
	Used	AbsTime	-1,0000			

Analysis



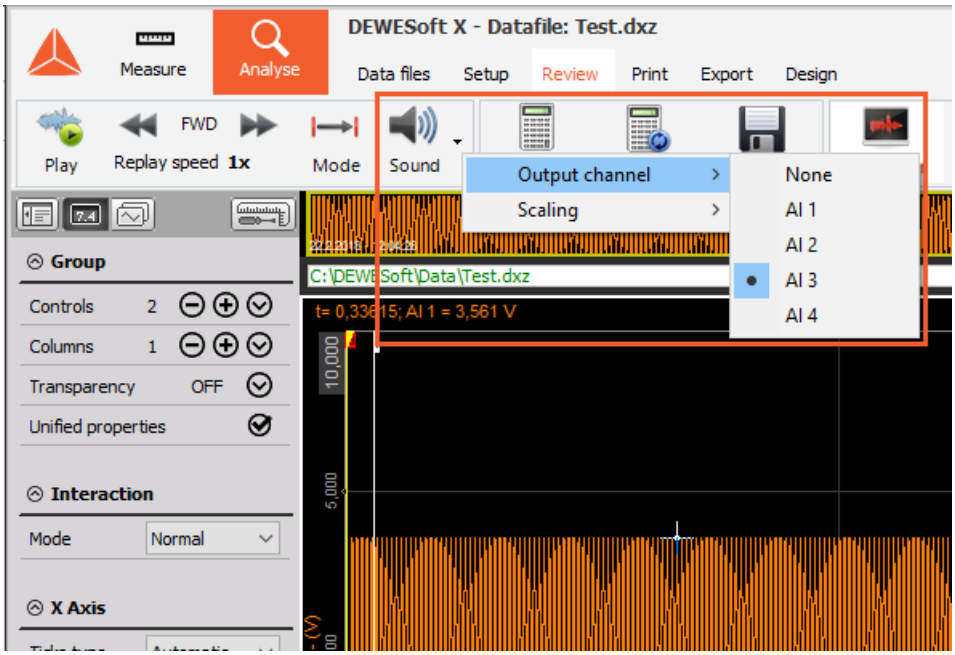
Here, you can select the synchronous channel as output for sound replaying in analysis mode (0 - none, 1 - first sync channel, ...).

Let's say we measure the signal from 5 channels, and we want to use the third one (AI 3) as a default channel for sound replaying.

ID	Used	C	Name	Ampl. name	Range	Measurement	Min	Values	Max	Physical quantity	Units	Zero	Setup
1	Used		AI 1	DEMO-SIRIUS-ACC	10 V	Voltage	-10,00	-7,602 / 7,851	10,00		V	Zero	Setup
2	Used		AI 2	DEMO-SIRIUS-ACC+	10 V	Voltage	-10,00	-3,662 / 3,909	10,00		V	Zero	Setup
3	Used		AI 3	DEMO-SIRIUS-ACC+	10 V	Voltage	-10,00	-2,701 / 2,922	10,00		V	Zero	Setup
4	Used		AI 4	DEMO-SIRIUS-MUL	10 V	Voltage	-10,00	-7,636 / 7,885	10,00		V	Zero	Setup
5	Used		AI 5	DEMO-SIRIUS-MUL	10 V	Voltage	-10,00	-7,630 / 7,868	10,00		V	Zero	Setup
6	Used		AI 6	DEMO-SIRIUS-STG	50 V	Voltage	-50,00	-38,23 / 39,34	50,00		V	Zero	Setup
7	Used		AI 7	DEMO-SIRIUS-STG	50 V	Voltage	-50,00	-38,14 / 39,32	50,00		V	Zero	Setup
8	Used		AI 8	DEMO-SIRIUS-HV	1000 V	Voltage	-1000,00	-189,7 / 214,6	1000,00		V	Zero	Setup

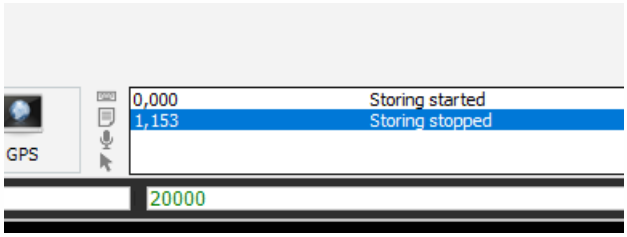
To set the third channel (AI 3) as the channel for sound replaying, just enter the number 3 in a check-box.

After we store and replay the data file we can see the third channel (AI 3) as the default one for sound replay output channel.



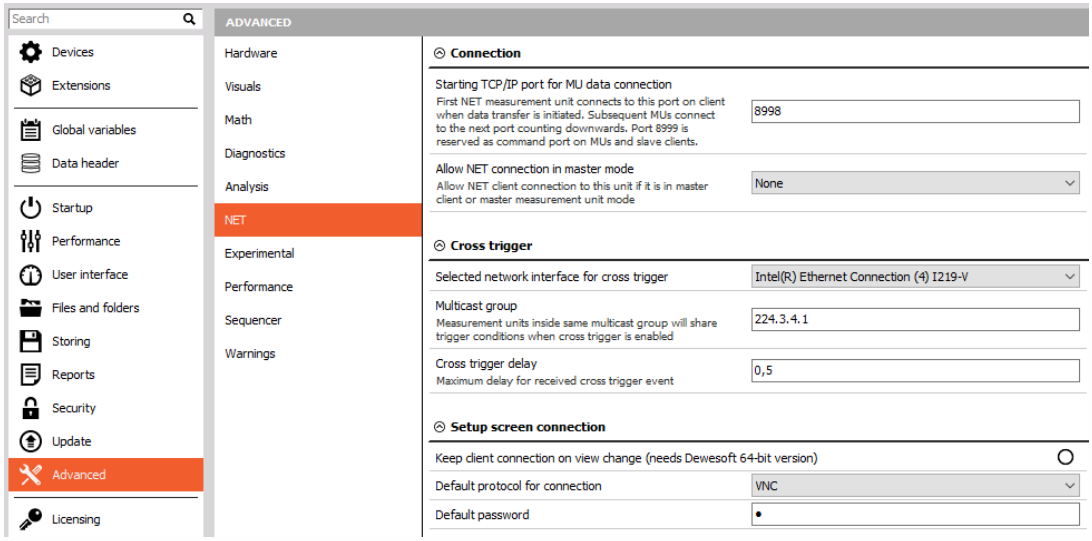
Use relative event time by default for new setups - with the new setup all the events will have the event time in relative time. If this option is disabled, the default time is absolute.

The time type can be also changed during measurement by right-clicking on the event and change time display.



NET

Starting TCP/IP port for MU data connection - the first NET measurement unit connects to this port on the client when data transfer is initiated. Subsequent Mus connect to the next port counting downwards. Port 8999 is reserved as command port on measurement units and slave clients.



Connection

Starting TCP/IP port for MU data connection.

It is possible to define the starting port on the master measurement unit.

For example, if the starting port is 8998, this port is used by the first measurement unit. The second measurement unit will use port 8997,...

The port 8999 is fixed and can't be changed. This port needs to be opened on master and slave units.

The port 1999 is used for Dewesoft launcher and also needs to be opened.

Allow NET connection in the master mode.

If Dewesoft is Master client or Master measurement unit, it is possible to define which clients are allowed to connect.

- None
- View client only
- Master or view client

Cross trigger

- Selected network interface for the cross trigger - Select the network card which is used for the NET system.
- Multicast group - Set-up the same IP multicast address on both master and slave units for the

cross trigger to work correctly.
- Cross trigger delay - Maximum allowed network delay for the cross trigger.

SETUP SCREEN CONNECTION

Keep client connection on view change (needs Dewesoft 64-bit connection)

- the master PC stays connected to the slave units even when they aren't selected. This allows a smoother transition between the slave units.

Default protocol for connection:

- VNC (ultra VNC)

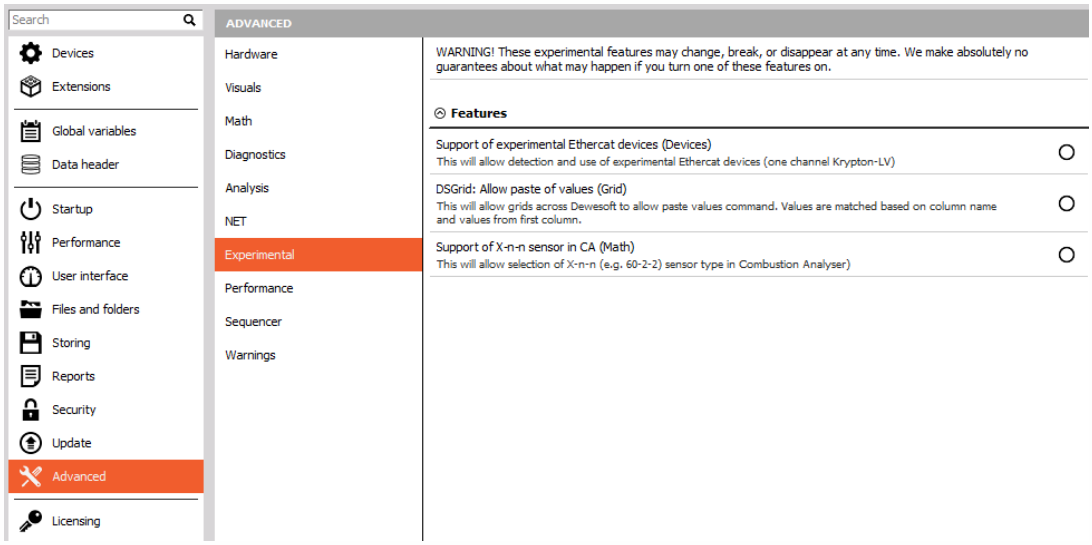
Default password - by default the password is set to "a" but for a better security Dewesoft allows you to change the password.

RDP (windows remote desktop)

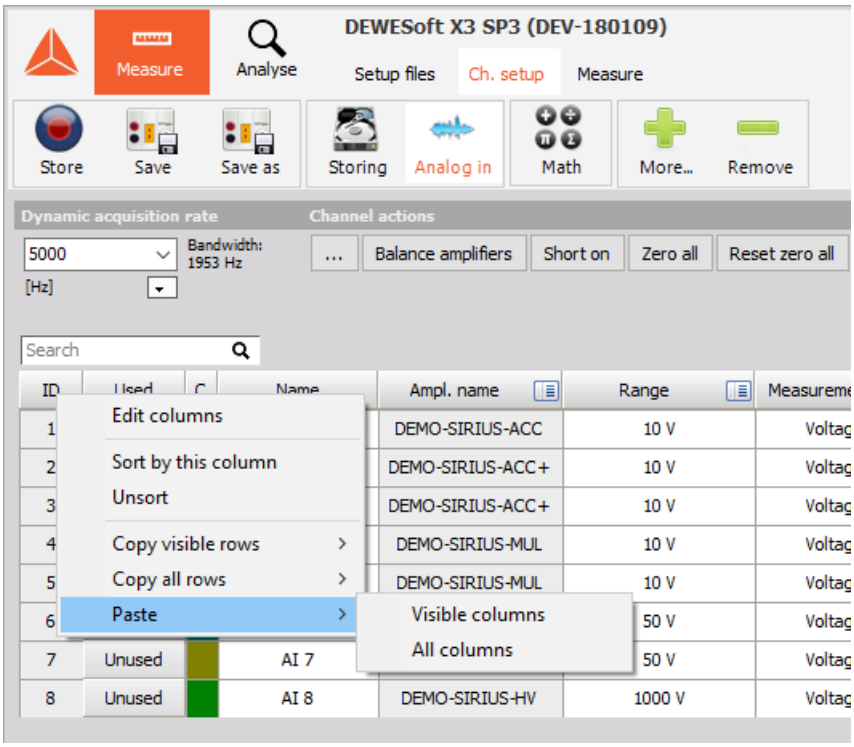
The remote desktop offers two types of authentication:

- Use Windows credentials - If the master unit was already connected to a slave unit via remote desktop, and if the connection is saved, this option can be used.
- User name and password - If the master unit is connecting to the slave unit for the first time.

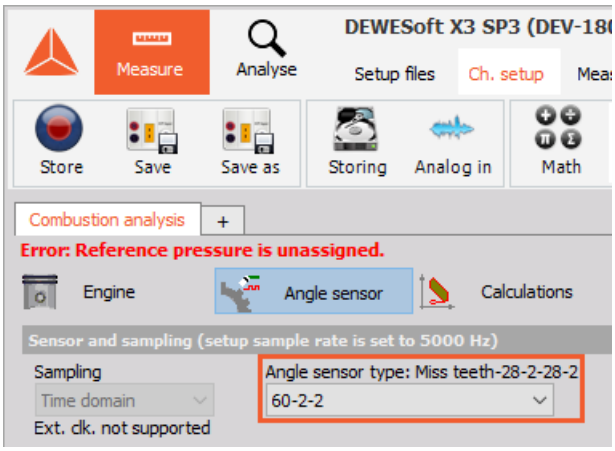
Experimental



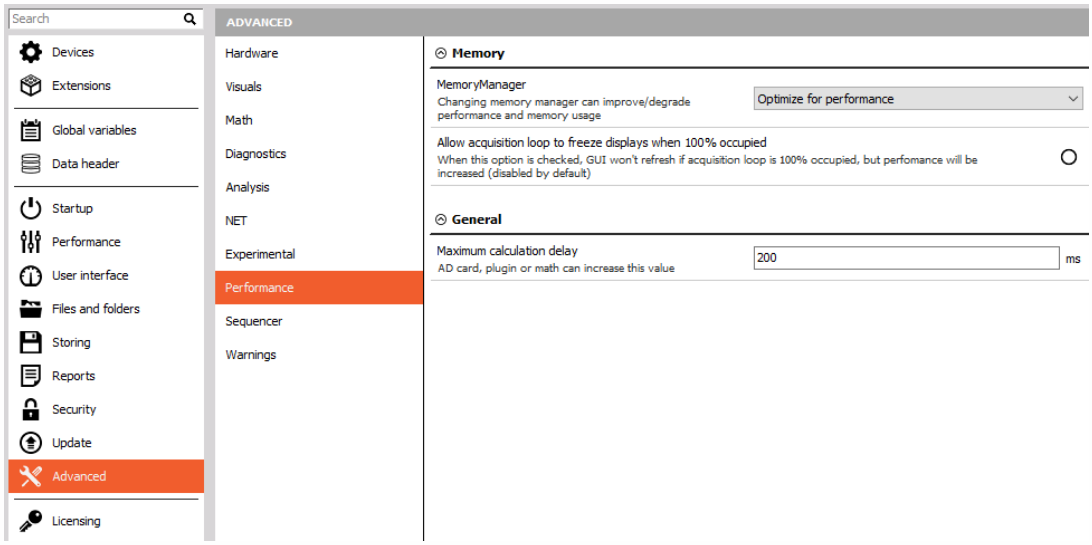
- Support for experimental EtherCAT devices - This option enables devices which are not yet released.
- DSGrid: Allow paste of values (Grid) - This option allows you to copy the grid as the text file, change it and copy it back in Dewesoft grid.



- Support of X-n-n sensor in CA (Math) - With this option enabled, combustion analysis allows you to choose the X-n-n sensor as shown on the picture below:


Measure
Analyse
DEWESoft X3 SP3 (DEV-18
Setup files
Ch. setup
Store
Save
Save as
Storing
Analog in
Math
Combustion analysis
+
Error: Reference pressure is unassigned.
Engine
Angle sensor
Calculations
Sensor and sampling (setup sample rate is set to 5000 Hz)
Sampling
Time domain
Ext. clk. not supported
Angle sensor type: Miss teeth-28-2-28-2
60-2-2

Performance

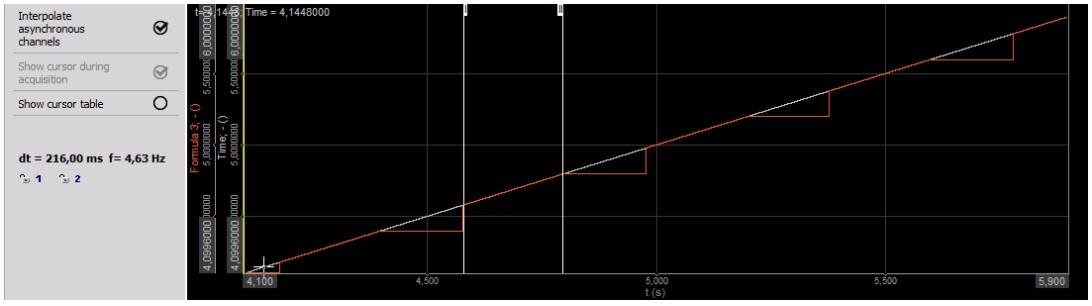


Memory

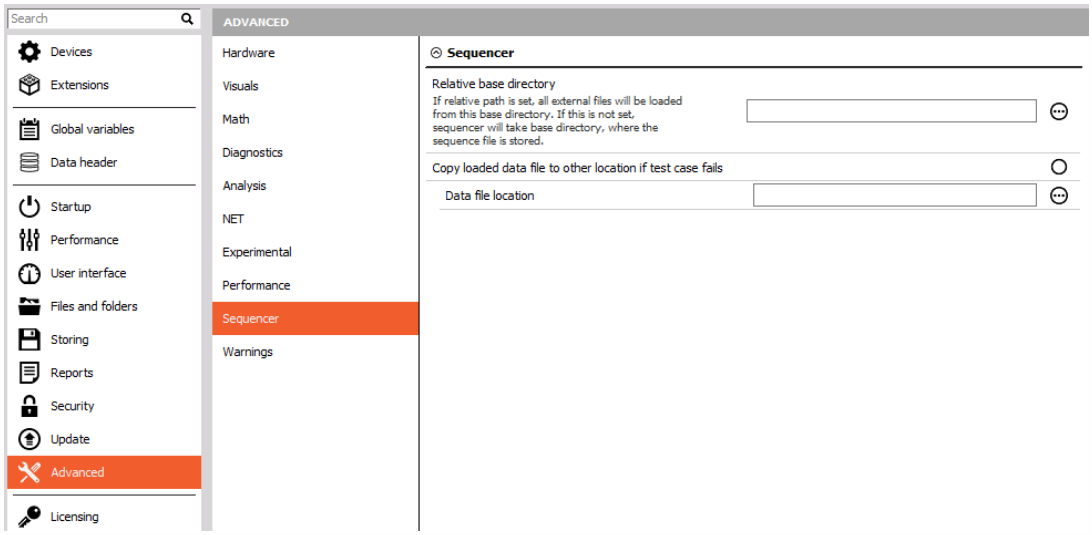
- Memory manager
- optimize for memory usage
- optimize for performance
- Allow acquisition loop to freeze displays when 100% occupied - If the setup is very complex, it needs more CPU. This option allows Dewesoft to freeze the graphics to save the CPU power for the calculations.

General

Maximum calculation delay - By default Dewesoft allows 200ms of calculation delay. If the asynchronous channels are up-sampled during measurement, the interpolation works only for the last 200ms.

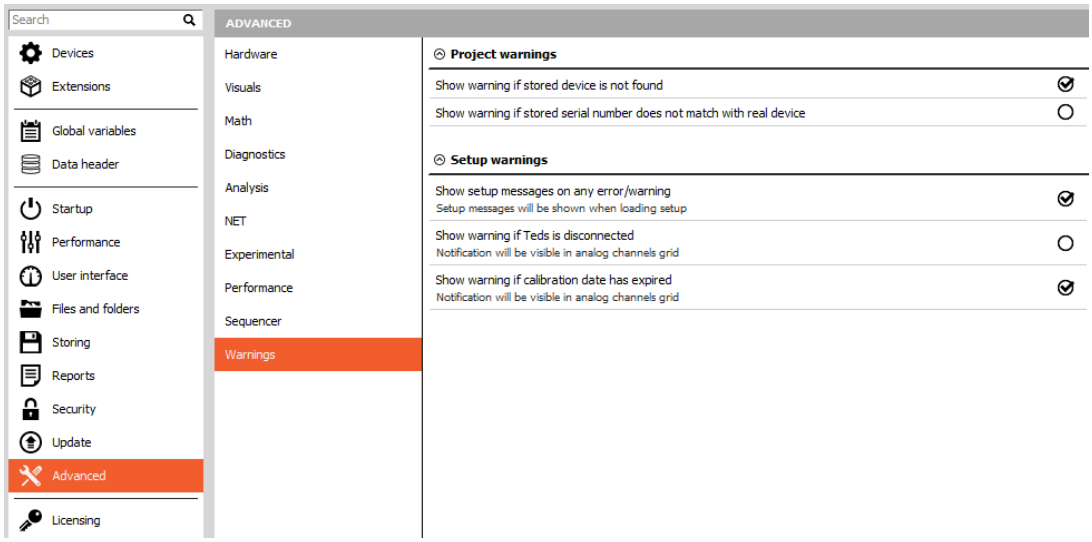


Sequencer



- Relative base directory - The sequencer will load or open files relative to the directory set here.
- Copy the loaded data file to other location if the test case fails - If the test case fails in analyze mode Dewesoft will copy that data file to the chosen location.
- Data file location

Warnings



Project warnings

Dewesoft allows you to enable the notifications which will be shown when the project is loaded:

- Show warning if the stored device is not found.
- Show warning if stored serial number does not match with the real device.

Setup warnings

Dewesoft allows you to enable the notifications which will be shown when the setup file is loaded:

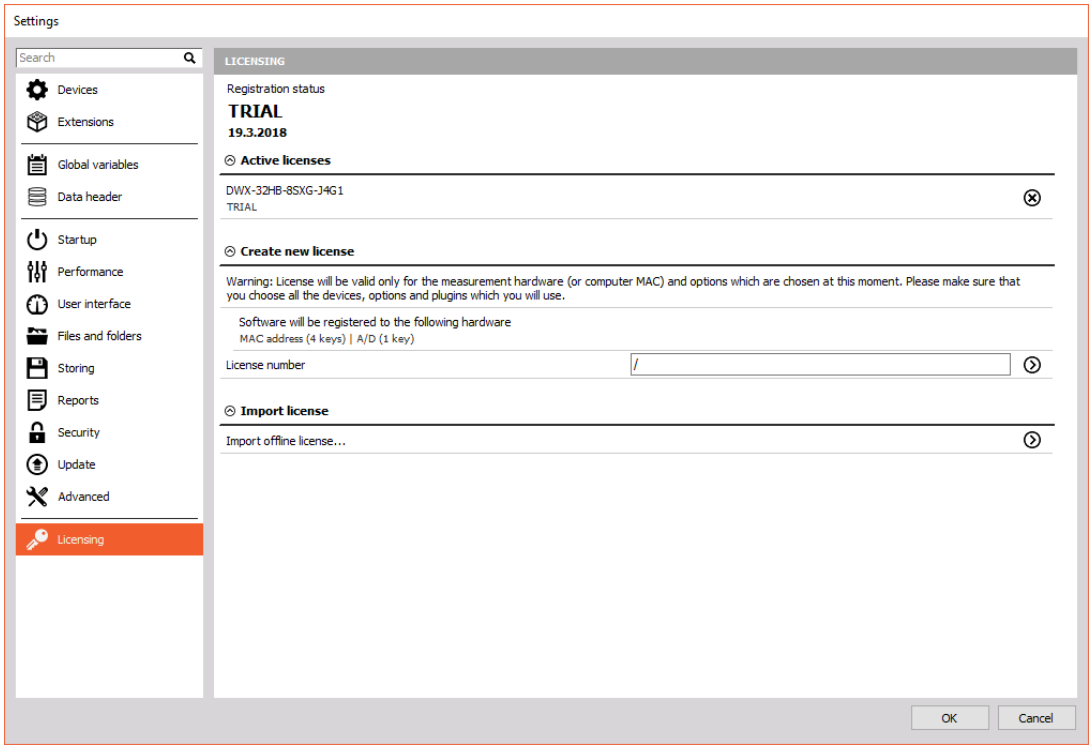
- Show setup messages on any error/warning.
- Show warning if TEDS is disconnected.
- Show warning if calibration date has expired.

LICENSING

The license for measuring with Dewesoft is included in the device (usually PROF version). Once it is connected to the USB port, it acts as a dongle.

The license for analyzing is free! Dewesoft can be installed on any computer and the stored data files can be opened, recalculated, and exported.

Additional licenses can be required for plugins, these can then also be written into the Dewesoft® device. To test plugins, you can request a 30-days-Evaluation license.



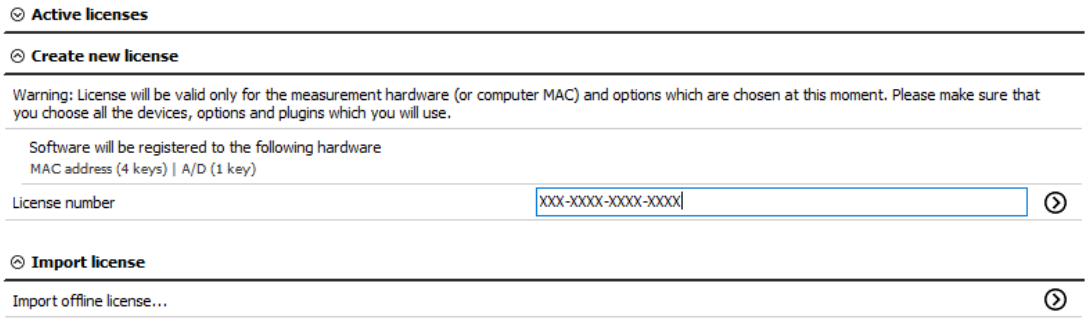
Under licensing section, you see your Active licenses, Create new license or Import offline license (*.lic file format).

Make sure that all the hardware which you intend to use is connected and switched on. Only in this case, the registration will be fully done for all measurement hardware being used. The license is then generated to hardware keys of the hardware OR to computer MAC address. So you can use the same hardware with the license file on any computer and you can also exchange the measurement hardware on the same computer with a need that MAC addresses are the same.

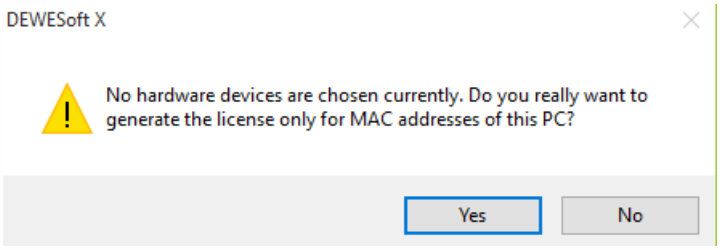
Offline registration

If your measurement PC has access to an internet connection you can register directly from inside Dewesoft software. If not Dewesoft will automatically offer offline registration.

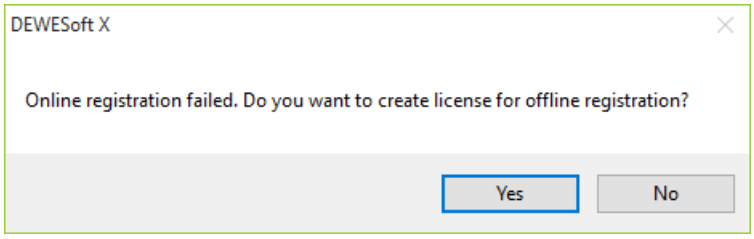
Enter your license key and press arrow next to the license number button.



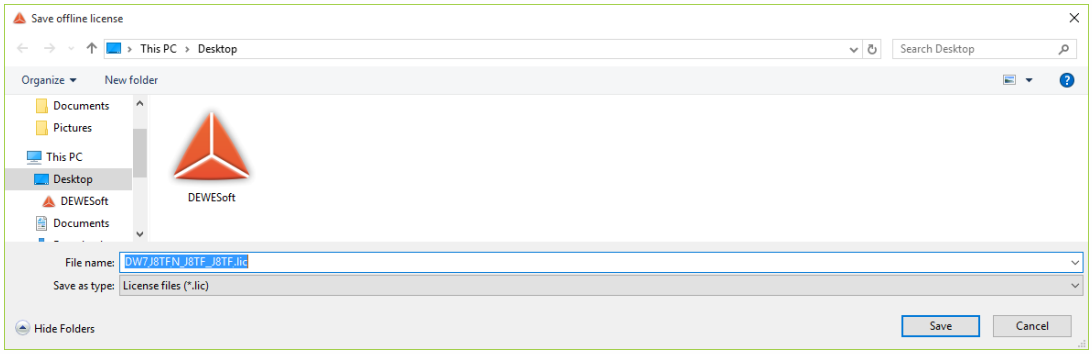
If there are no DAQ devices connected to the computer, the license is generated only for MAC addresses of the PC.



Dewesoft will connect to the internet and will register automatically (online). If there is no internet connection, Dewesoft will offer offline registration.



Dewesoft will create a *.lic file and you have to save it to the computer.



Copy the created license *.lic file from the measurement computer, and go to a computer with

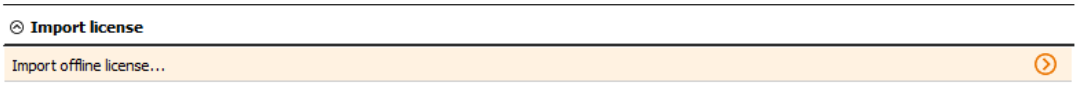
internet access. Visit Dewesoft home page, and select registration.

Drop the *.lic file to the window.


```
<?xml version="1.0" encoding="UTF-8"?>
<LicenseFile>
    <License ID="DW7-Y3CN-Y3CNY3CN'>
        <Hardwarekeys>
            <MACs>F4-B7-E2-F3-FC-C9;20-89-84-F7-F3-CA;F6-B7-E2-F3-FC-C9;00-FF-A1-F8-4B-18</MACs>
        <PluginGUIDs></PluginGUIDs><PluginNames></PluginNames></Hardwarekeys>
        <LicenseCodes><PluginGUIDs></PluginGUIDs><MACs>2091557;2591077;2147673;2247081</MACs></LicenseCodes>
    </License>
</LicenseFile>
```

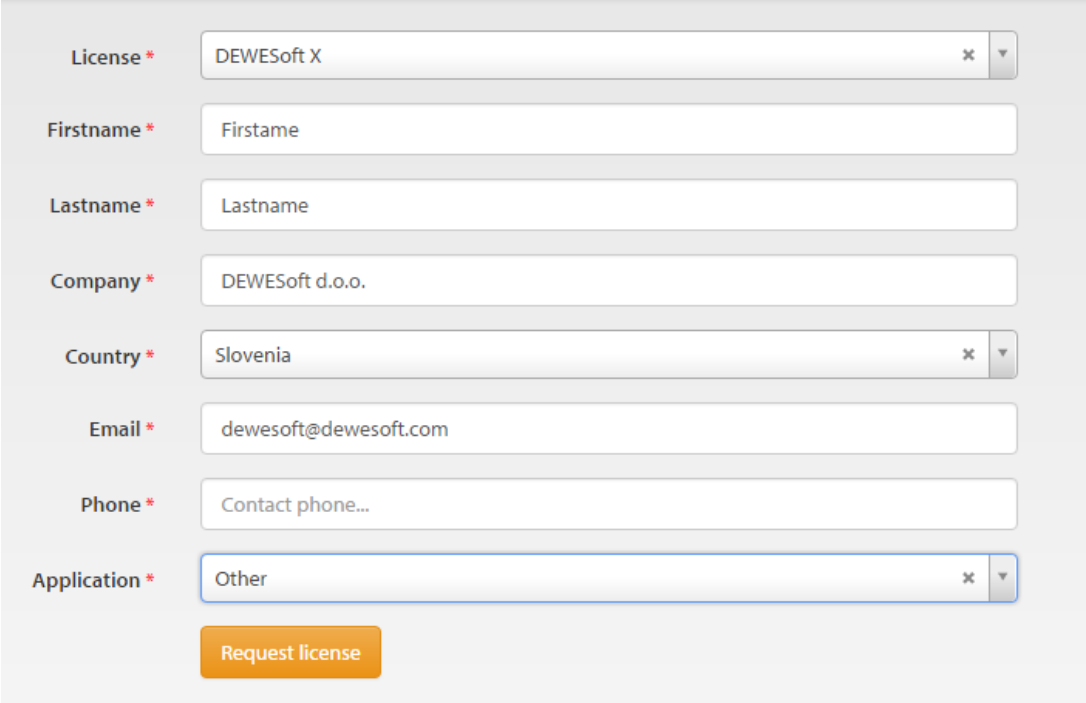

Download

Restart Dewesoft software and select the option Import license. Import the *.lic file that was downloaded from the web page.



Evaluation license

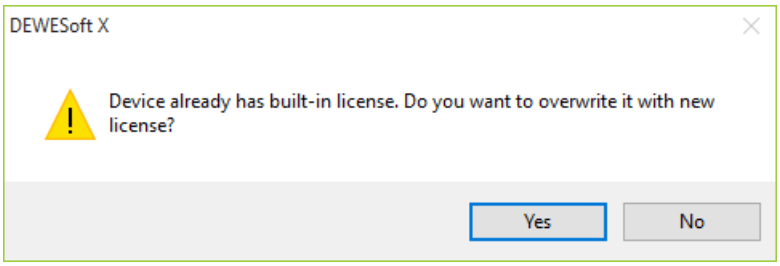
To receive a fully functional 30-day evaluation license for Dewesoft software fill and submit the form below. Data marked with * are required. Please provide a valid email address to which we can send the evaluation license.



Write license to Dewesoft devices

When we have a license for software on our PC, we can also write the license to Dewesoft measurement devices. This can be done, so the devices can be used also with another computer.

If the license is already written on the measurement device, we get a warning. We can overwrite the existing license with a new one.



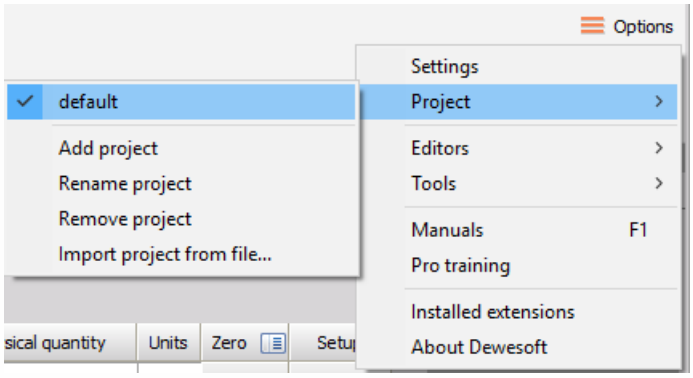
PROJECT

Project entry is the place to change the existing project. Here is the list of all available projects created so far. There is always a Default project. The project includes the Hardware setup and Project setup (Project folders, Starting setup, Security, Internal variables, Data header, and Memory).

To change between existing projects, select System settings - Project and then select a project from the list.

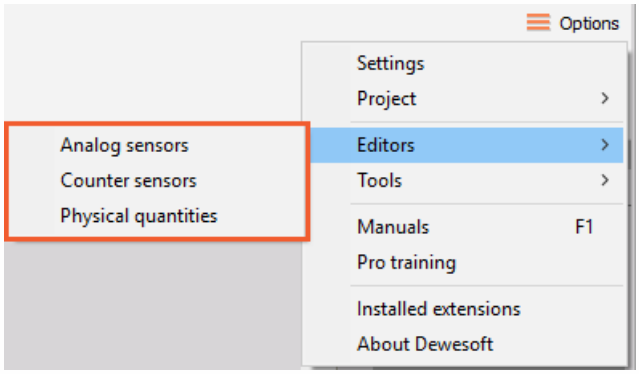
Projects are usually used in two ways: to change the hardware configuration (to use different devices with the same hardware) or to use different access rights and folder configuration.

There are several other options available in the menu:



- Add project - This will create a new project and open the dialog to name the project. The new project will not be created from scratch, but it will take all the settings of the currently selected project.
- Rename project - Rename project menu item will open the dialog box to rename the currently selected project.
- Remove project - Remove project menu item will delete the currently selected project after user confirmation. Please note that there is no chance to revert this operation.
- Import project from file - This menu item allows to import the project from setup file (*.dxs), data file (*.dxd) or XML setup (*.xml). In order to move a project from one system to another, a project file (found in the system folder) can simply be copied. This option goes one step further and allows the project (added to every setup and data file) to be extracted from any file from the source system.

EDITORS



Analog sensors

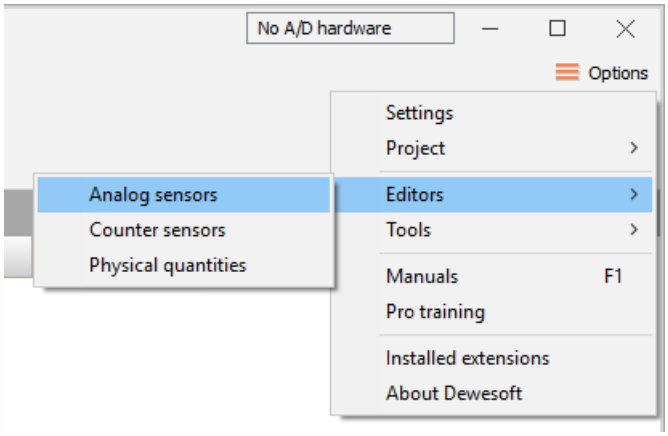
Counter sensors

Physical quantities

ANALOG SENSORS

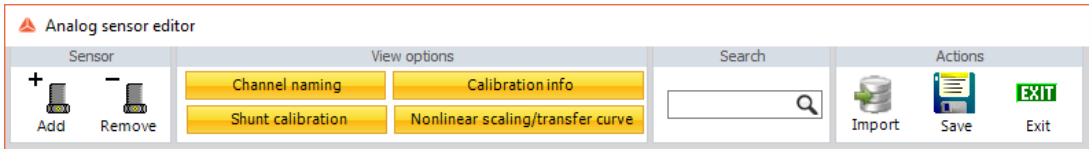
Dewesoft offers a sensor database which holds the list and the properties of all sensors which will be used. In addition to the linear scaling, which can also be done in the input channel setup, the sensor database also offers to scale by the table or by the polynomial and even transfer curves can be defined.

To enter the sensor editor go to Settings -> Sensor editor.



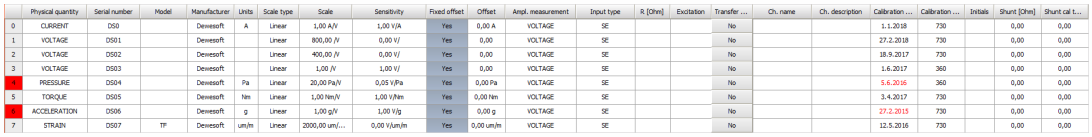
The sensor database can be created and adapted according to your requirements (used measurement hardware). The sensor (database) editor offers several functions for creation, editing and managing sensors:

- Add sensor (creates a new sensor in the sensor database)
- Remove sensor (removes the sensor from the sensor database)
- Import (import existing *.xml or *.dxd sensor databases)
- Save (store the sensor database to file)
- Exit (close the sensor database editor)



Each sensor is defined with the information in the database:

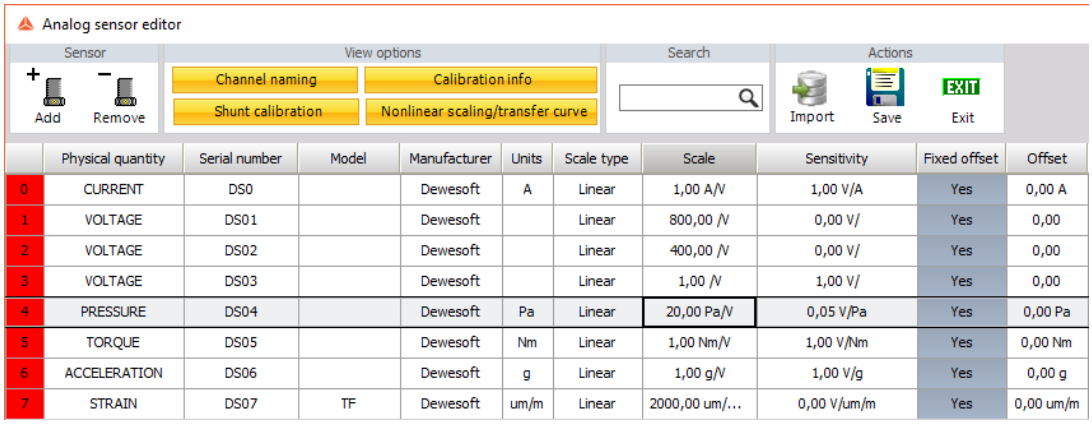
- Sensor type, serial number, calibration date, calibration period
- General sensor information
- Scaling type (linear, polynomial, table)
- Transfer curve (used for sensor correction)



	Physical quantity	Serial number	Model	Manufacturer	Units	Scale type	Scale	Sensitivity	Fixed offset	Offset	Ampl. measurement	Input type	R [Ohm]	Excitation	Transfer ...	Ch. name	Ch. description	Calibration ...	Calibration ...	Initials	Shunt [Ohm]	Shunt cal (...
0	CURRENT	DS0		Dewesoft	A	Linear	1,00 A/V	1,00 V/A	Yes	0,00 A	VOLTAGE	SE			No			1.1.2018	730		0,00	0,00
1	VOLTAGE	DS01		Dewesoft		Linear	800,00 /V	0,00 V/	Yes	0,00	VOLTAGE	SE			No			27.2.2018	730		0,00	0,00
2	VOLTAGE	DS02		Dewesoft		Linear	400,00 /V	0,00 V/	Yes	0,00	VOLTAGE	SE			No			18.9.2017	730		0,00	0,00
3	VOLTAGE	DS03		Dewesoft		Linear	1,00 /V	1,00 V/	Yes	0,00	VOLTAGE	SE			No			1.6.2017	360		0,00	0,00
4	PRESSURE	DS04		Dewesoft	Pa	Linear	20,00 Pa/V	0,05 V/Pa	Yes	0,00 Pa	VOLTAGE	SE			No			5.6.2016	360		0,00	0,00
5	TORQUE	DS05		Dewesoft	Nm	Linear	1,00 Nm/V	1,00 V/Nm	Yes	0,00 Nm	VOLTAGE	SE			No			3.4.2017	730		0,00	0,00
6	ACCELERATION	DS06		Dewesoft	g	Linear	1,00 g/V	1,00 V/g	Yes	0,00 g	VOLTAGE	SE			No			27.2.2015	730		0,00	0,00
7	STRAIN	DS07	TF	Dewesoft	um/m	Linear	2000,00 um/...	0,00 V/um/m	Yes	0,00 um/m	VOLTAGE	SE			No			12.5.2016	730		0,00	0,00

Edit sensor

After you enter the Sensor editor (Settings - Sensor editor) a list of all available sensors will open. Any already defined (and pre-defined) sensors and connected TEDS sensors will be listed automatically.



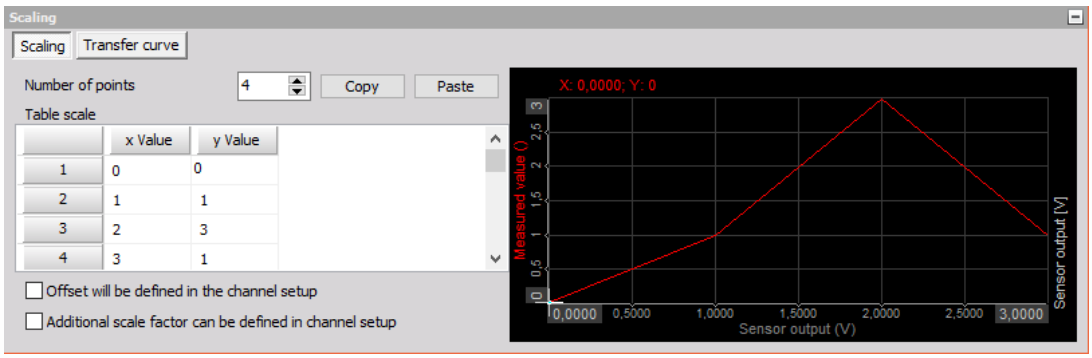
	Physical quantity	Serial number	Model	Manufacturer	Units	Scale type	Scale	Sensitivity	Fixed offset	Offset
0	CURRENT	DS0		Dewesoft	A	Linear	1,00 A/V	1,00 V/A	Yes	0,00 A
1	VOLTAGE	DS01		Dewesoft		Linear	800,00 /V	0,00 V/	Yes	0,00
2	VOLTAGE	DS02		Dewesoft		Linear	400,00 /V	0,00 V/	Yes	0,00
3	VOLTAGE	DS03		Dewesoft		Linear	1,00 /V	1,00 V/	Yes	0,00
4	PRESSURE	DS04		Dewesoft	Pa	Linear	20,00 Pa/V	0,05 V/Pa	Yes	0,00 Pa
5	TORQUE	DS05		Dewesoft	Nm	Linear	1,00 Nm/V	1,00 V/Nm	Yes	0,00 Nm
6	ACCELERATION	DS06		Dewesoft	g	Linear	1,00 g/V	1,00 V/g	Yes	0,00 g
7	STRAIN	DS07	TF	Dewesoft	um/m	Linear	2000,00 um/...	0,00 V/um/m	Yes	0,00 um/m

- Sensor type - The sensor type should contain the name or type of the sensor but can contain any text desired.
- Serial number - The serial number has to be unique! It is not allowed to use the same serial number because this information is used for sensor identification.
- Scale type - The sensors editor differ between linear, polynomial and table scaling.
- Transfer curve -Can be used for sensors with phase characteristics like current clamps, geophones, ...
- Recalibration data - Enter the date when the sensor has to be recalibrated. When the date expires, the Recall date will change to red color.

To edit a sensor, simply click on the desired field of the table.

	Physical quantity	Serial number	Model	Manufacturer
0	CURRENT	DS0		Dewesoft
1	VOLTAGE	DS01		Dewesoft
2	VOLTAGE	DS02		Dewesoft
3	VOLTAGE	DS03		Dewesoft

At the bottom of the sensor database editor there is additional information of the selected sensor:



- Scaling - contains the scaling information of the selected sensor.
- Transfer curve - is only available if the transfer curve is enabled.

General sensor information

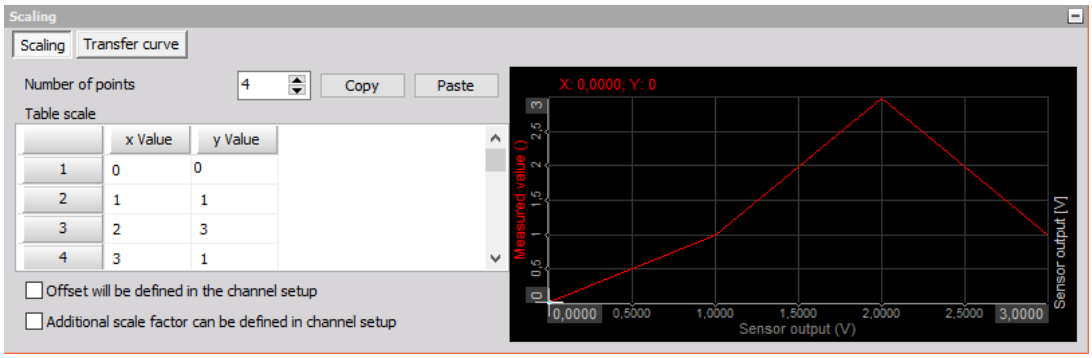
The general information about the sensor:

- Physical (input) unit - The physical unit of the sensor, e.g.V, A, °C, mm, psi, %,...
- Electrical (output) unit - The electrical output unit of the sensor, most times V or A.
- Channel name - Use this field to pre-define the channel name for the setup.
- Channel description - Additional information about the sensor, helps to select the right sensor in the input setup.

PRESSURE	psi	SE		Brake cylinder p...
TORQUE	Nm	SE		TE
VOLTAGE	V	SE		Prim Volts
STRAIN	u...	BRQUARTER3W		
STRAIN	u...	BRFULL		
STRAIN	u...	BRFULL		

Sensor scaling

Dewesoft supports different scaling types within the sensor database. When we select Scaling type field, the scaling type selection list appears in this field and the Scaling tab is selected automatically. After selecting the desired scaling types from the selection list in Scaling tab, scaling information appears automatically.



Dewesoft supports three different scaling types within the sensor database.

Linear

Linear scaling is used for linear sensors. It is calculated by the formula:

$$y = kx + d$$

- y = physical value
- k = scale
- x = measured value
- d= offset

The Scale and Offset factors are defined manually in the columns:

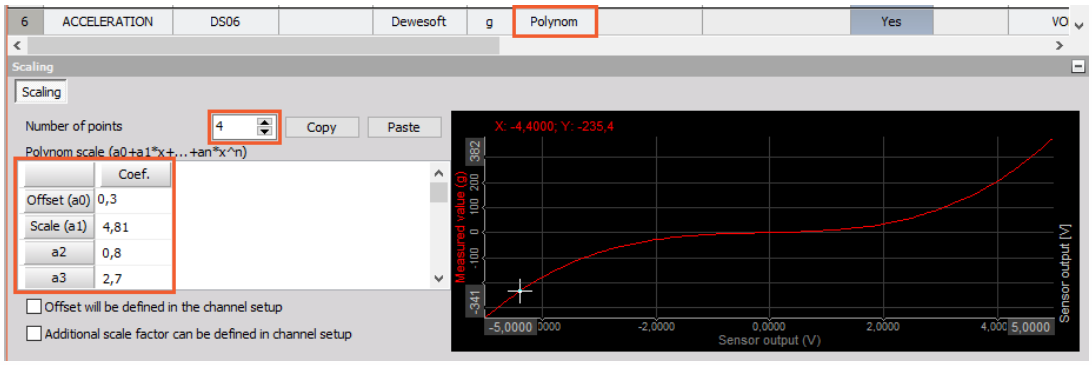
ACCELERATION	4231_Y	3093-M-34	Dytran	m/s2	Linear	92.32 m/s2/V	0.01 V/m/s2	Yes	0.00 m/s2	IEPE

Polynomial

Polynomial scaling is used for nonlinear sensors, calculated by the equation:

$$y = a_0 + a_1x + a_2x^2 + \ldots + a_nx^n$$

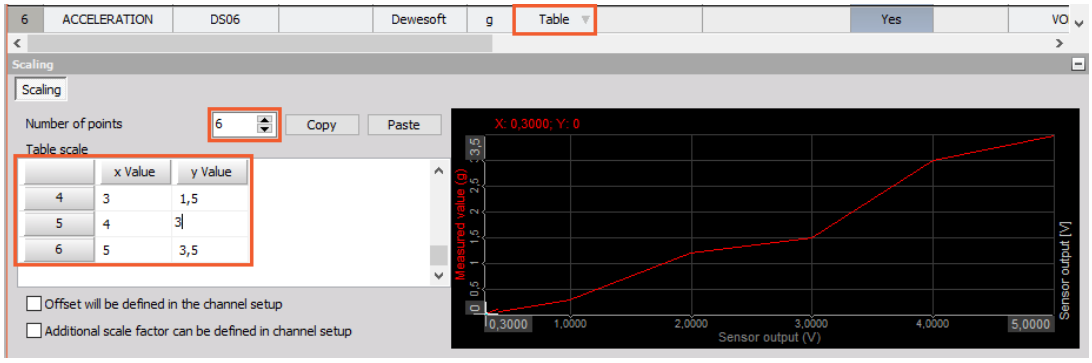
Example: Polynom scaling $a_0 + a_1x + \ldots + a_nx^n$ enter the n number and the coefficients in the Coef. column (coefficient a0 defines the offset).



Table

Table scaling is also used for nonlinear sensors, but it is normally easier to enter because most calibration information contains several calibration points.

Enter the number of points (rows of the table) and in the table below, enter the X and Y values.



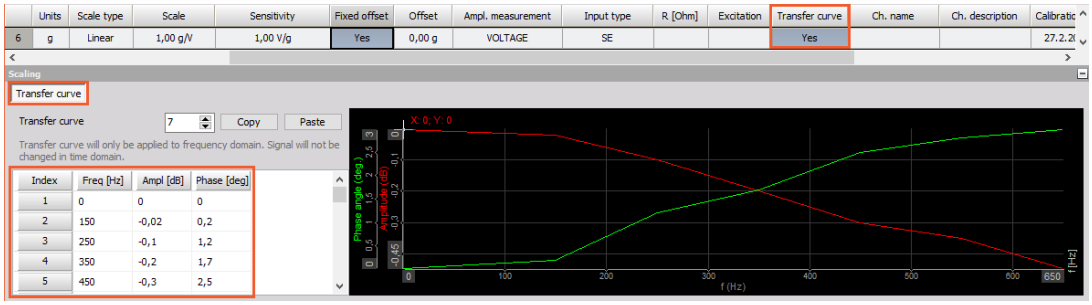
NOTE: **As these three scaling types can't compensate phase errors, they are used for time domain or angle based acquisitions. For frequency domain applications a transfer curve will deliver more accurate results.**

Transfer curve

The transfer curve calibration can be used when the frequency behavior of the sensor is known:

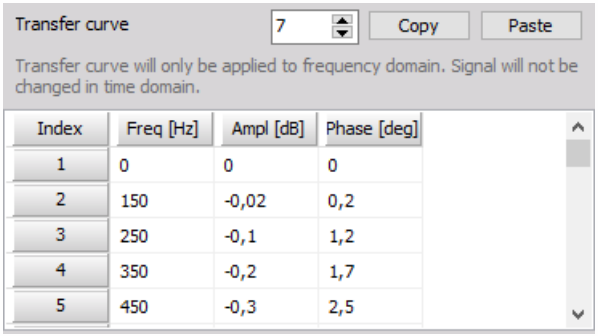
- transfer curves for most common sensors are already measured,
- copy it from the calibration sheet of the sensor (if the calibration sheet includes the transfer curve),
- the third option is to measure it with Dewesoft FRF modal test, but this requires some additional equipment.

Some companies offer calibration reports for sensors also in the frequency domain, for example for current clamps. The transfer curve compensates amplitude and phase, both in relation to the signal frequency. In the table under Transfer curve column, we need to enter the points of the curve.

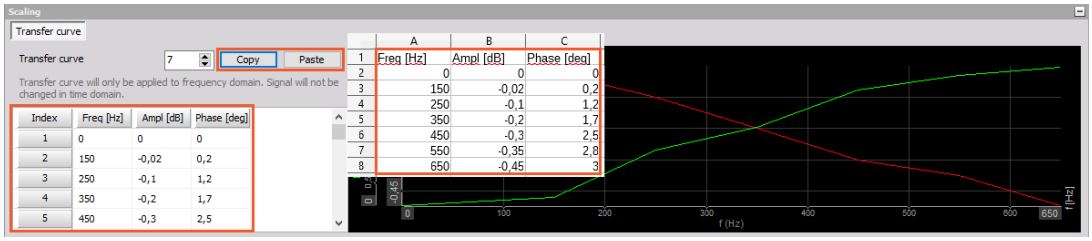


We can enter the sensors transfer curve in two ways:

- Manually enter the number of points (rows of the table) and in the numbers below the columns Freq [Hz] (signal frequency), Ampl [dB] (amplitude deviation) and Phase [deg] (phase angle).



- Using the Windows copy and paste the values from a table created in the external program (e.g. Excel, ...).



NOTE: **Please keep in mind that the transfer curve is only helpful in frequency domain application (FFT, harmonics, octave analysis, ...). You will not see the effect of sensors transfer curve in the time domain data - for that it is best to use a filter with similar characteristics like the transfer curve.**

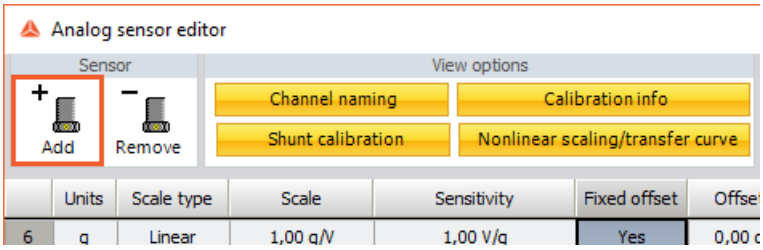
Save the sensors with the Save button and close the sensor editor with the Exit button.

Manage sensors

Add sensor

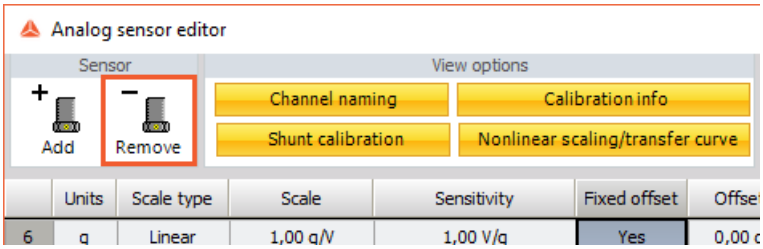
If you want to add new sensors, press the Add sensor button and new row.

- the sensor is added to the table:



Remove sensor

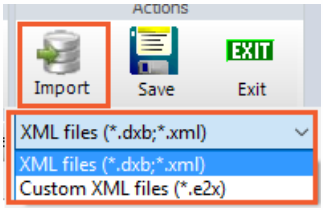
To remove a sensor just click on it - the whole line in the table will receive a gray background - and press the Remove sensor button.



WARNING: Removing a sensor will remove a sensor without a warning. It is not possible to restore deleted sensors.

Import sensor database

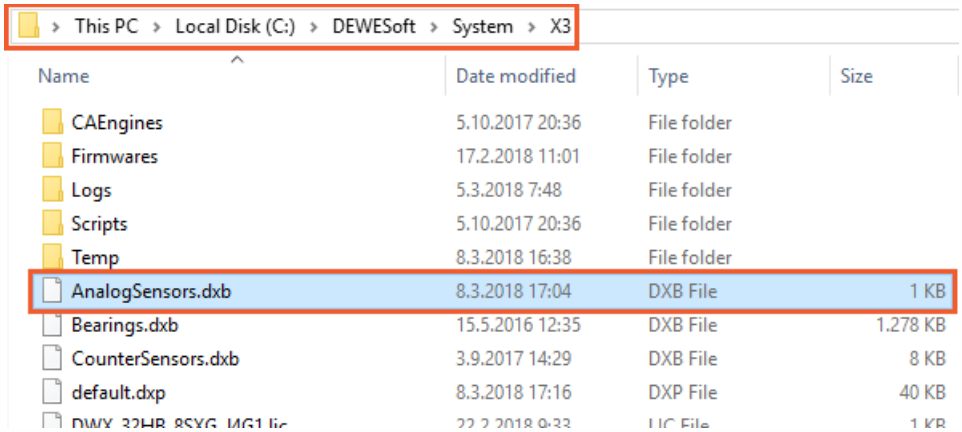
Import the existing sensor database. The supported formats are standard XML files (*.dxb, *.xml) and Custom XML files (*.e2x).



Save file

To save all changes in the sensors database just click on the Save file icon.

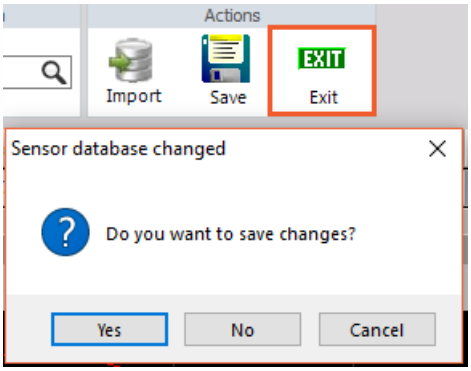
The data will be stored immediately in an XML styled file called AnalogSensors.dxb (the file AnalogSensors.dxb can be found in the Dewesoft installation folder).



WARNING: **Do not change anything directly in the XML file, this may cause errors and unusable files!**

Exit sensor database editor

To exit the sensors database editor simply press the Exit button. If you have not already stored changes, you will be asked if you want to store changes or leave without changes.

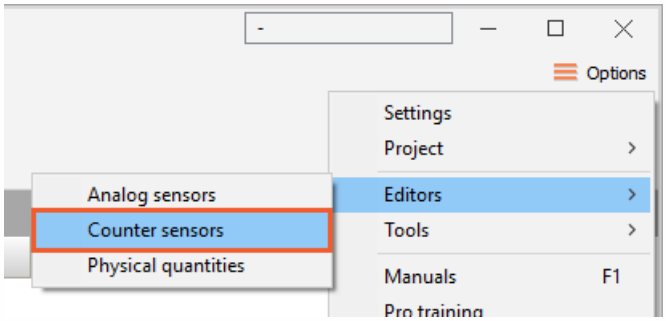


WARNING: **If you leave the sensor editor without storing changes, data will be lost!**

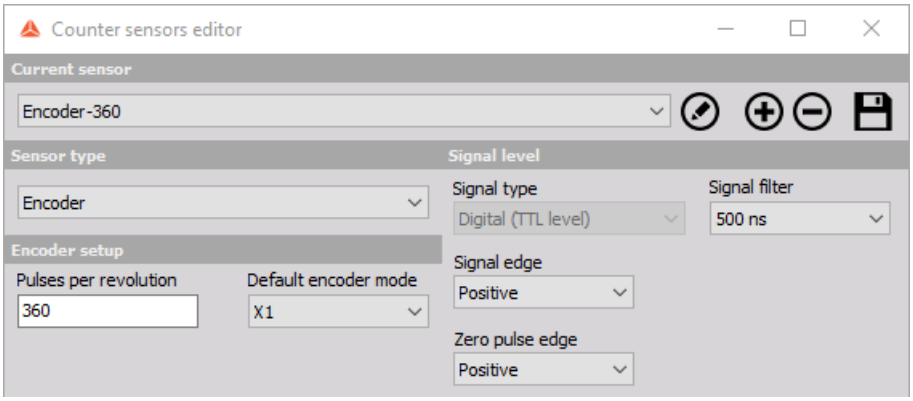
COUNTER SENSORS

Dewesoft offers counter sensor database which holds the list and the properties of the counter sensors which can be used in order tracking, combustion analysis, and angle sensor math. We can define encoders, Geartooth, and other angle sensors.

The Counter sensor editor can be accessed under Settings - Counter sensor editor.

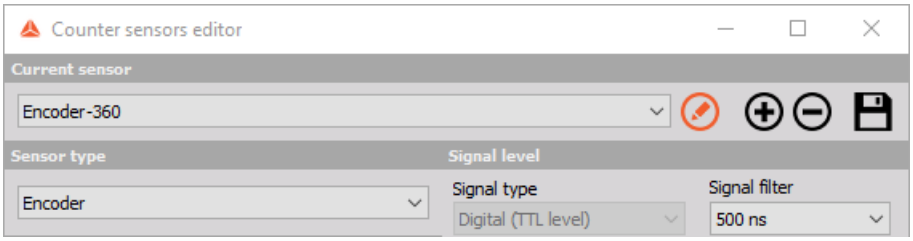


When we enter the editor, the following window appears where we define the counter sensors:

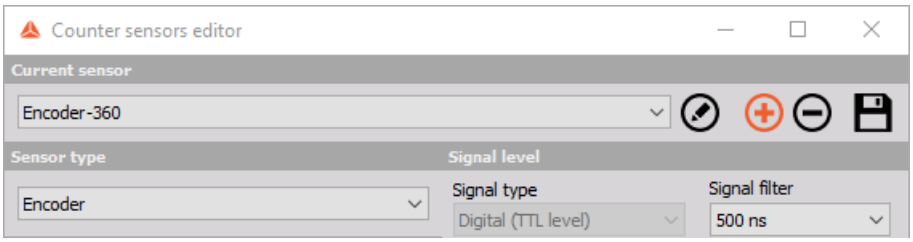


Several predefined sensors are already in the editor list; these sensors are installed with Dewesoft. We can always add new ones, modify existing ones or delete the sensors. On the upper right side of the Counter sensor editor, the window common command icons appear:

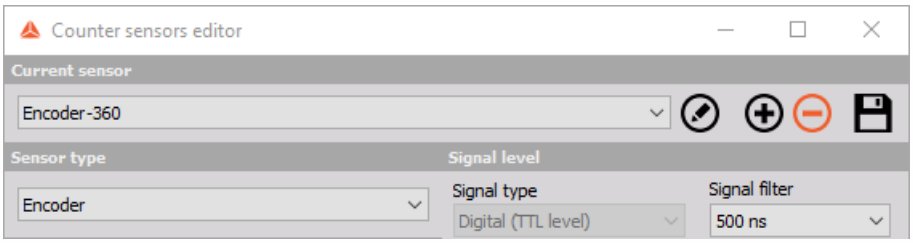
Rename sensor - rename the default or pre-defined sensor.



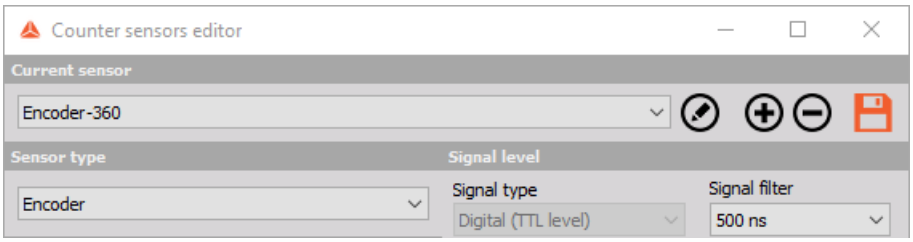
Add sensor - button will add anew sensor. The sensor will be named 'New sensor', but we can rename it.



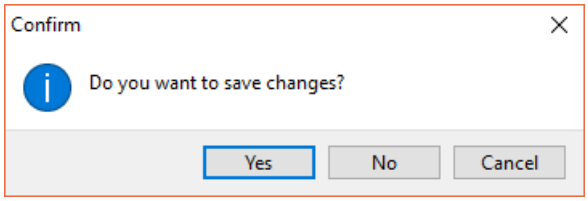
Remove sensor - button will remove the currently selected sensor.



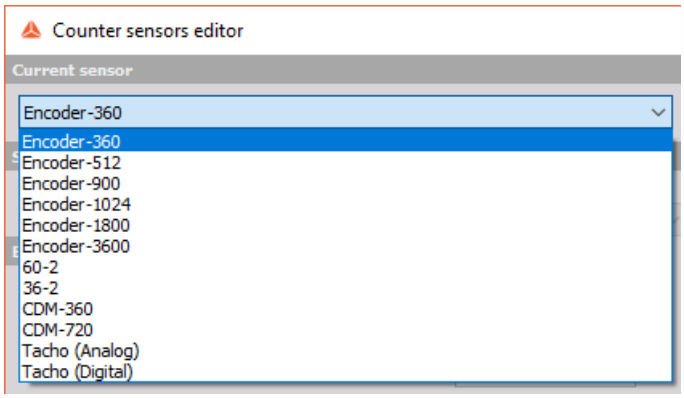
Save and exit - button will save the counter sensor database and close the editor.



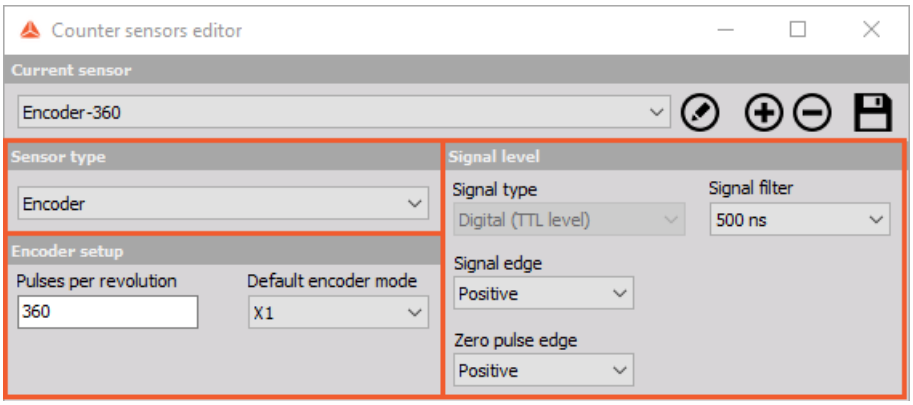
Exit - button will leave the editor without saving the data, so please be sure to use Save & Exit if you make any changes to the sensors or confirm you want to save the changes when you exit the editor.



In the counter sensor editor window, we can choose any sensor from the current sensor drop-down list for viewing and editing.



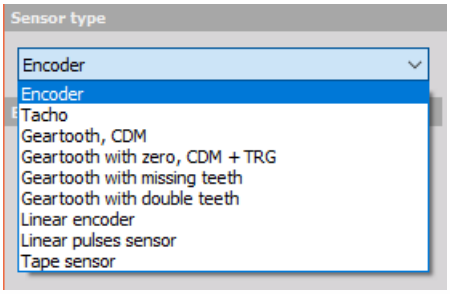
Settings and entered values for Counter sensor are divided into following sections:



- Sensor type
- Signal level
- Encoder setup or Geartooth setup (depend on selected sensor type)

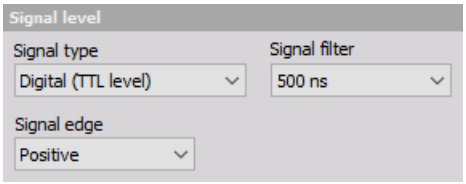
Counter sensor type

There are several basic sensor types available which can be selected from the Sensor type drop-down list:



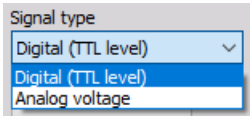
- Encoder - Classic angle encoder with A, B and Z signals. The signal can be only digital.
- Tacho - Sensor with one pulse per revolution. The signal can be either analog or digital.
- Geartooth, CDM - Sensor with the defined number of pulses per revolution, but without any zero pulses.
- Geartooth with zero, CDM + TRG - Sensor with the defined number of pulses per revolution with zero pulses.
- Geartooth with missing teeth - A classic in-vehicle sensor with any number of pulses where one tooth is missing for zero pulse recognition. A typical example is 36+1 sensor.
- Geartooth with double teeth - An in-vehicle sensor with any number of pulses per revolution with some double teeth missing. A typical example is the Geartooth with 60 teeth where two of them are missing, so in fact, there are 58 teeth and there is a gap between two teeth.
- Linear encoder - Sensor measuring displacement with any number of pulses per millimeter and pulses per revolution.
- Linear pulses sensor - A linear sensor, measuring displacement with any number of pulses per millimeter.
- Tape sensor - An angle sensor with white tape and black stripes attached to the rotating disc.

Counter signal level

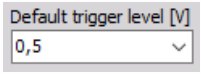


There are several signal level settings. From the drop-down list it can be selected:

Signal type

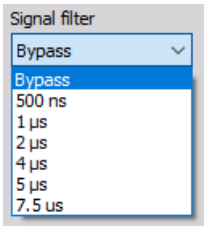


The encoder and Geartooth signal type can be only defined as digital (TTL level) and therefore used with counters while all other sensors can be also analog, which means that we need to define the trigger level.



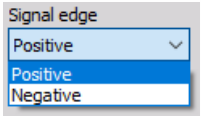
Signal filter

The signal filter is a debounce filter, used to prevent glitches in the signal. A signal must be present for the defined amount of time before the logis accepts it as a valid signal. This also inserts a delay for the signals!

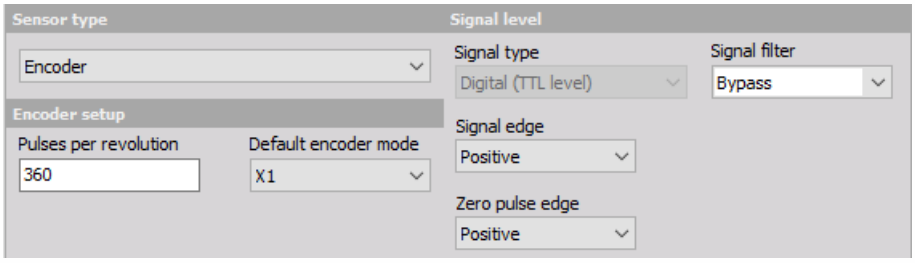


Signal edge

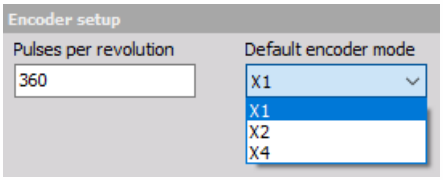
Signal edge can be either positive or negative.



Encoder setup



To prepare the encoder for the measurement, we have to define:



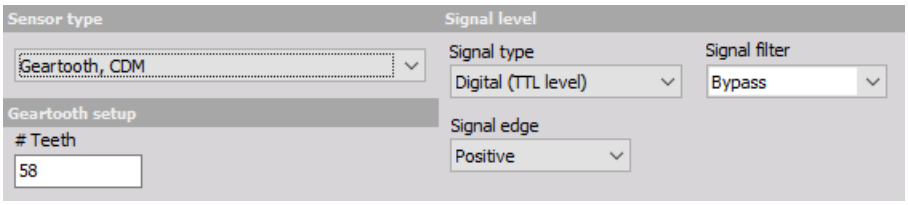
- Number of pulses per revolution - Standard values are multiples of 2 (256, 512,1024). These sensors are mainly used for external clocking so that we can have the frequency spectrum as a direct multiple of the number of revolutions to easily see the harmonic components. Other standard values are related to degrees (360, 720, 1800, 3600) where the reason is a nice angle resolution.
- Default encoder modes - Exact description of encoder modes can be found in the Dewesoft PRO training.

For additional help with visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training -> Counters.

Geartooth sensor setup

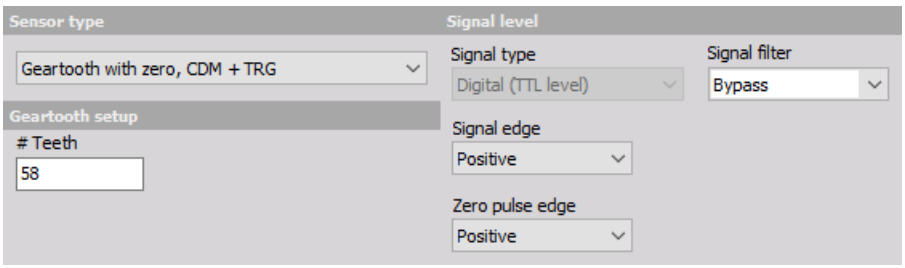
Geartooth sensor settings

For these sensors, we need to define the number of pulses per revolution (number of teeth).



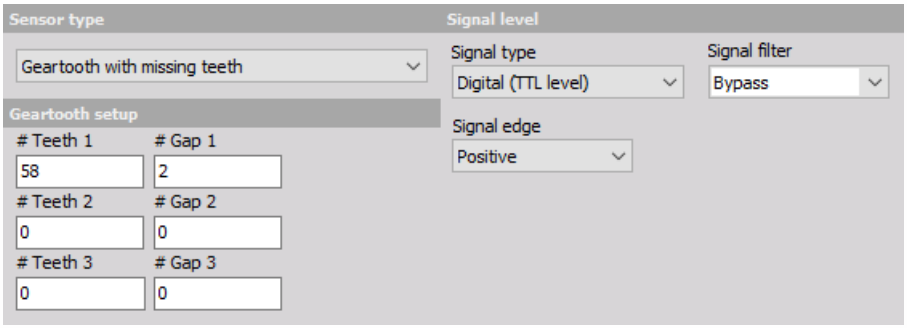
Geartooth with zero sensor settings

For these sensors, we need to define the number of pulses per revolution (number of teeth) and the direction of the zero pulse edge.

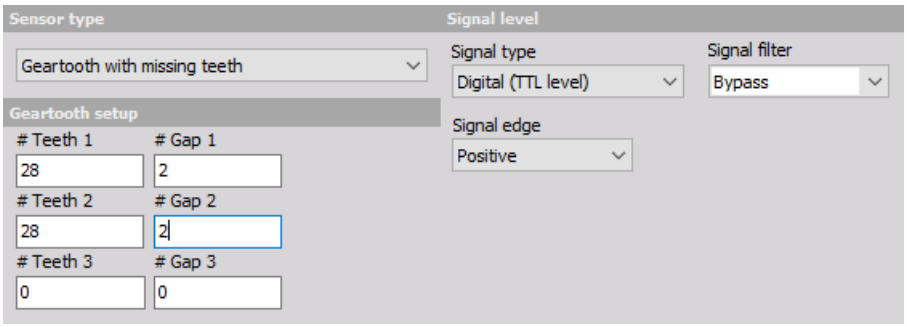


Geartooth with missing teeth sensor settings

For these sensors, we define the number of teeth (#Teeth) and the gap lengths (#Gap). For the number of teeth, we need to enter the number of teeth which exists. The example in the picture below shows the settings for 60-2 sensor. If the gap would not be there, there would be 60 teeth, but two of them are missing to create a gap, so there are only 58 teeth existing.

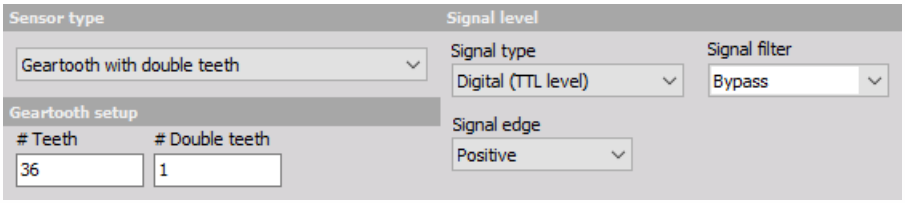


If we would like to enter the 60-2-2 sensor, we need to enter 28-2-28-2, so the whole sum of the numbers must always be the total number of teeth.



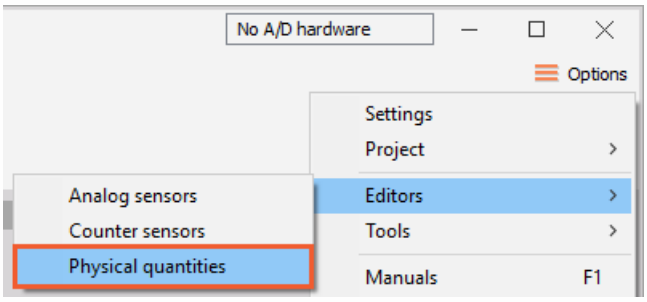
Geartooth with double teeth sensor settings

For these sensors, we define the number of teeth and the number of double teeth. The example below shows the settings for 36+1, a Geartooth with 36 teeth and one double tooth.



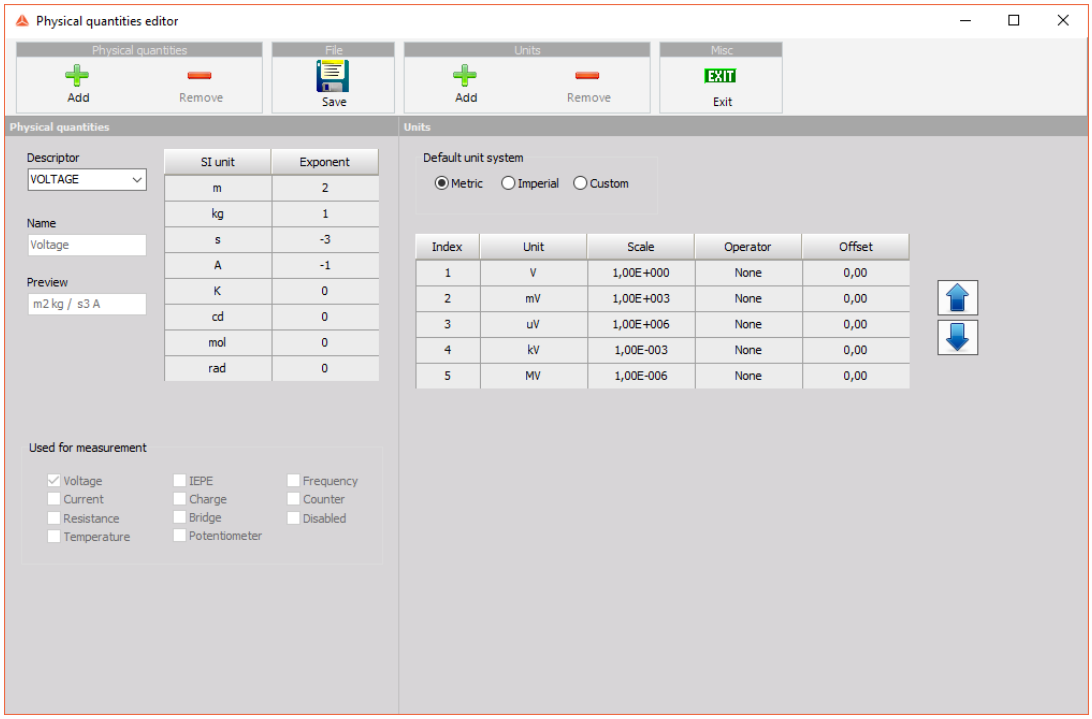
PHYSICAL QUANTITIES

Dewesoft offers editor which allow the user to add or modify physical quantities. The Physical quantities editor can be found in the editor list in the options.



When we enter the editor, the following window appears:

- Add physical quantity (if the physical quantity is missing, additional can be added here)
- Remove physical quantity (this option can remove additionally added physical quantities)
- Save (saves the changes to PhysicalQuantitiesX.xml)
- Add units (additional units can be added to any physical quantity)
- Remove units (additionally added units can be removed)
- Exit (close the physical quantities editor)

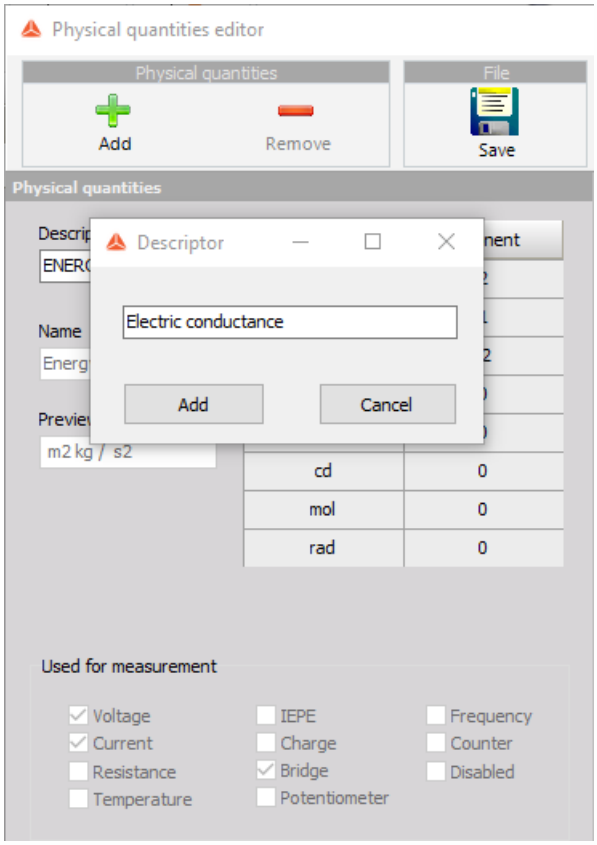


Add custom physical quantity

To add a new physical quantity click on the + button above physical quantities.

There are different fields which need to be filled to add the new quantity:

- Descriptor and name (write the name of the new physical quantity)



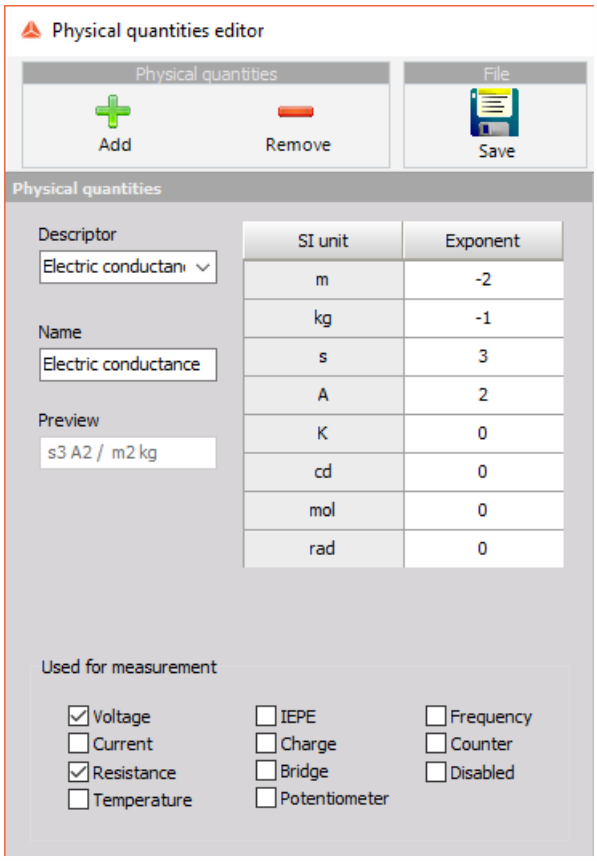
- SI units (write the correct exponent next to the SI unit)

The m/s is described with exponent 1 next to the "m" unit and exponent -1 next to the "s" unit.

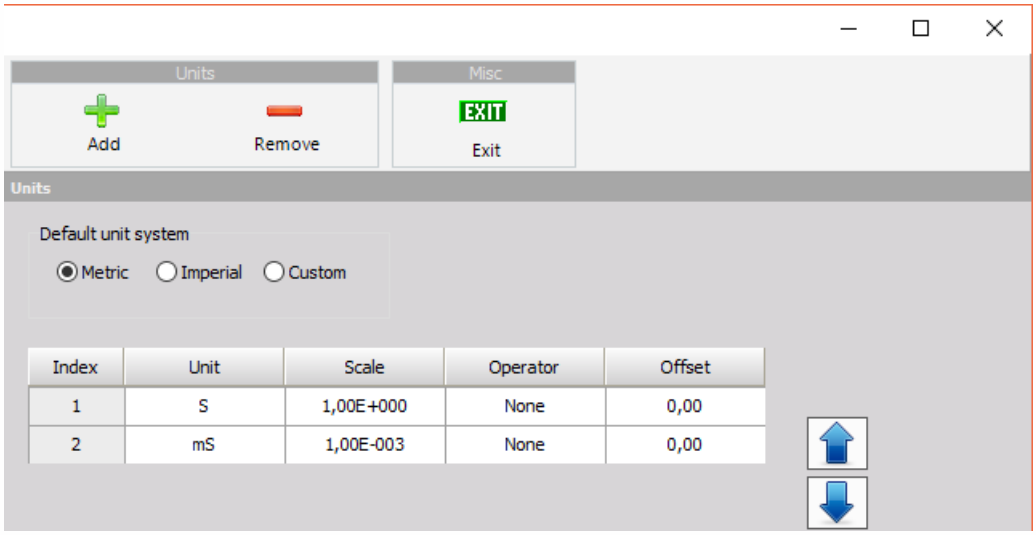
For our example we have used the electric conductance which has these SI exponents:

SI unit	Exponent
m	-2
kg	1
s	3
A	2

- Used for measurement (this option allows you to show the custom unit only in some of the measurement types in the channel setup)



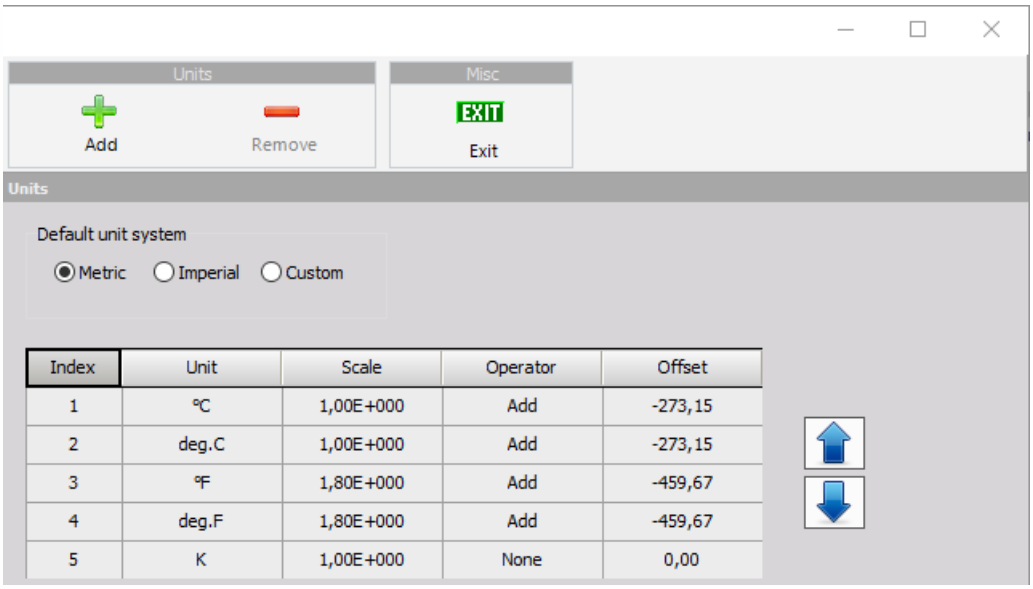
Every physical quantity needs also units which are defined on the right side of the editor.



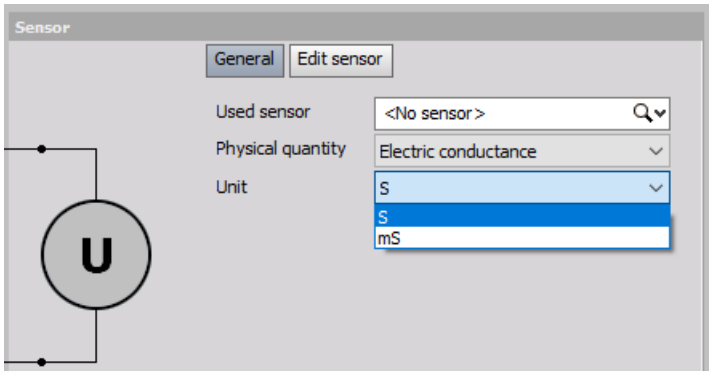
With every unit we need to define scale, operator and offset which allows Dewesoft to transfer between the units:

- Scale (defines the ratio new unit compared to t
- Operator (defines the offset type: add, ratio, Log. ref.)
- Offset (defines the offset between default and new unit)

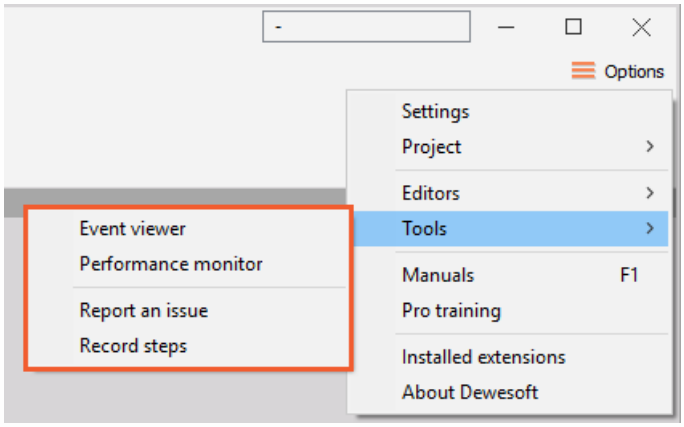
A great example where scale, operator and offset are used, its temperature:



As soon as the new physical quantity is added, we can also see it in the channel setup:



TOOLS



Event Viewer

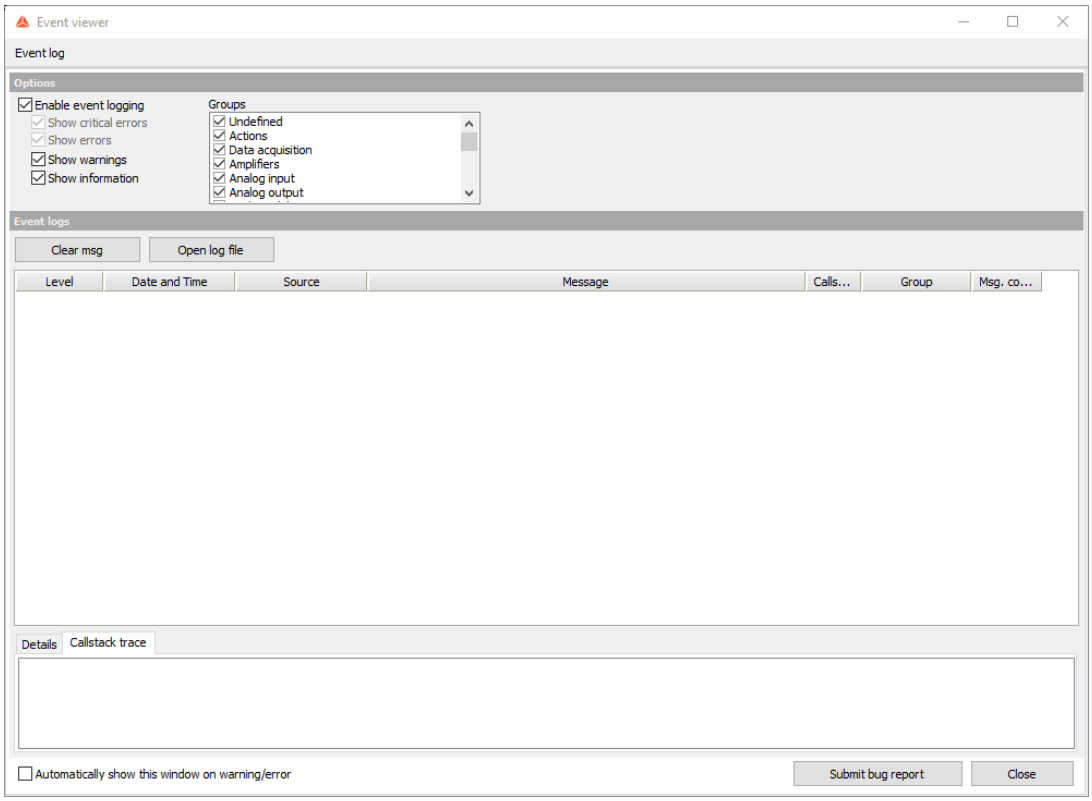
Performance monitor

Report and issue

Record steps

EVENT VIEWER

In the event viewer, it is possible to see the critical errors, errors, warnings, and information about the current session.

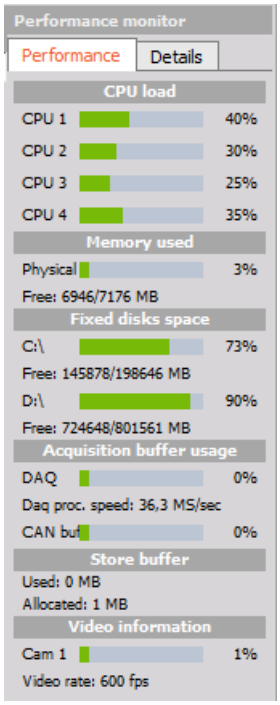


PERFORMANCE MONITOR

With the performance monitor, we can check how is the computer handling the current situation.

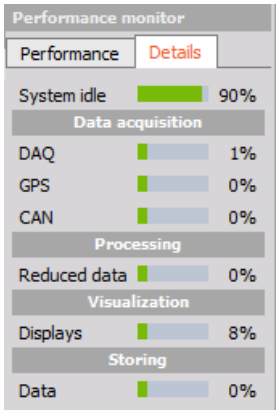
The performance monitor has two tabs:

- Performance
- CPU load - It shows the number of threads the computer is using and what is their usage.
- Memory used - It shows the amount of the memory the PC has and how much it is used. If the memory is close to 100%, it is possible to reduce the usage by decreasing the sync and async buffers or disabling the freeze option.
- Fixed disk space - It shows the amount of the free space on internal discs.
- Buffers - It shows the writing speed on disc and the amount of buffer used. If the buffer is increasing, the system is at its limit. It is possible that the USB speed limit or the write speed of the disc is exceeded.
- Acquisition buffer used
- Store buffer
- Video information

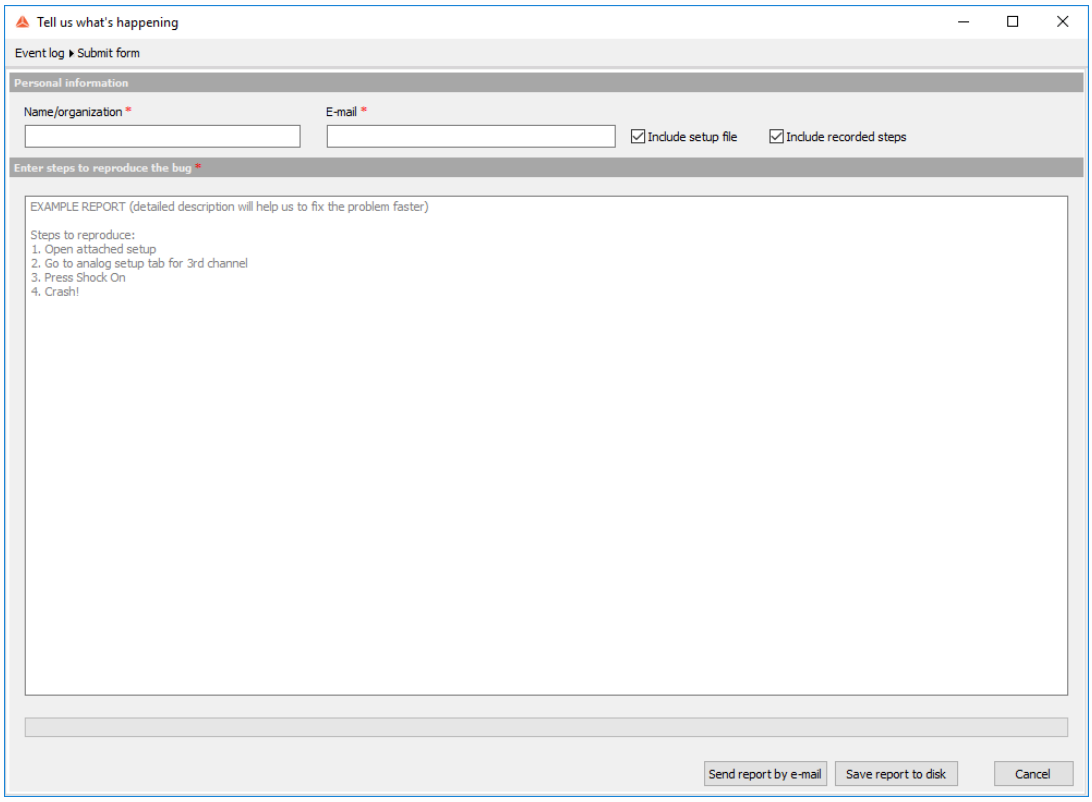


- Details
- Data acquisition
  - DAQ
  - GPS
  - CAN
- Processing
  - Reduced data
- Visualization

    - Displays
- Storing
    - Data



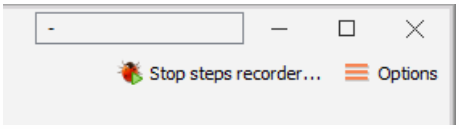
REPORT AN ISSUE



RECORD STEPS

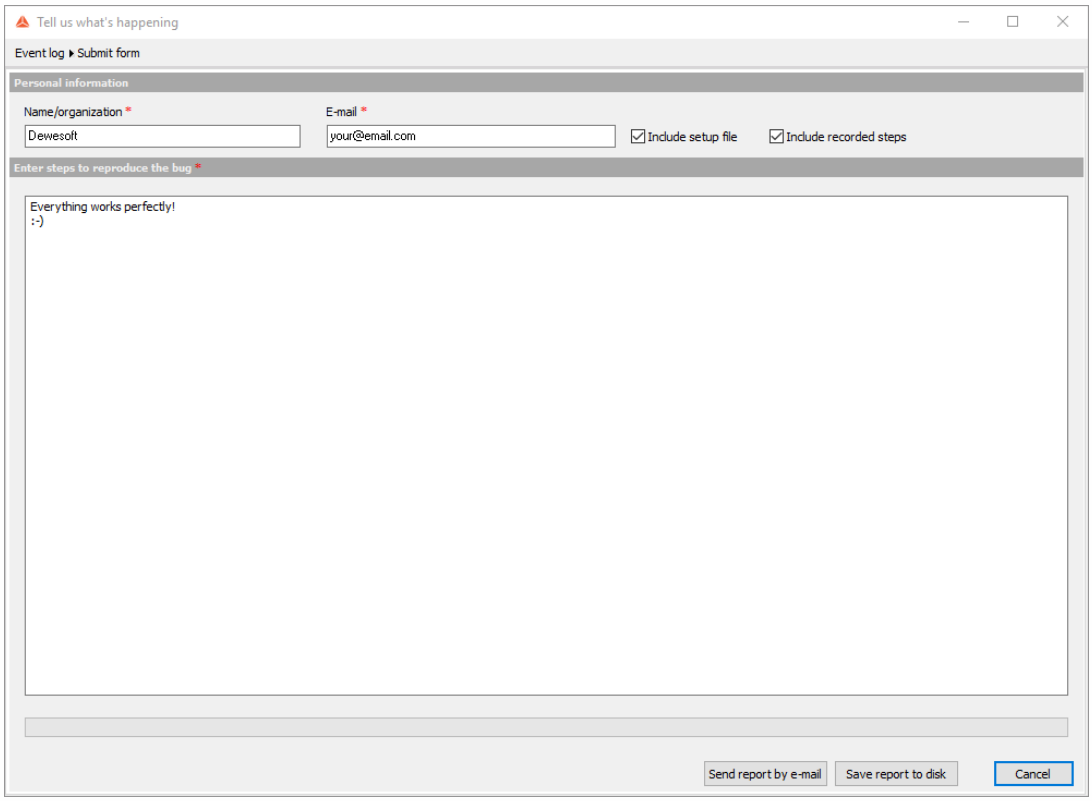
This option allows the customer to report the issue faster and easier.

Dewesoft will start making a screenshot for every click as soon as you click on the "Record steps" option. When this feature is enabled, you should see an additional "Stop steps recorder" option in the top right corner.



Once, you have recorded the all the steps which are needed to reproduce an issue click on "Stop steps recorder". The new window opens where it is possible to send the recorded steps, together with the latest setup file to us or save it to your PC.

Please write the correct email, so we can update you on the status of the issue.



SETUP

Add module

General

Freqeuncy domain analysis

Machinery diagnostics

Electrical measurements

Vehicle performance test

Strain, stress

Psophometer

Acoustics

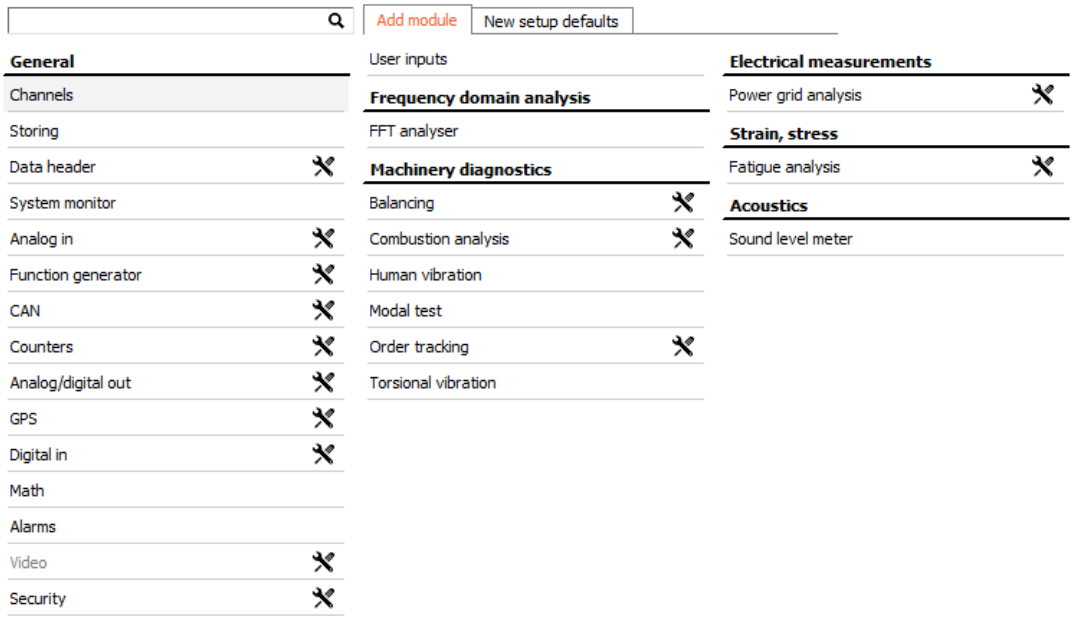
SRS

ADD MODULE

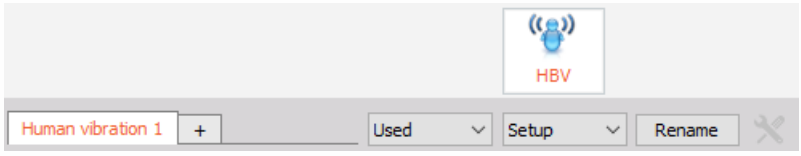
Adding new modules

To activate the desired module we simply go to Channel setup and click on Add module button.

A module menu like this will appear:



In this menu, we can see all possible modules that we can add to Channel setup. To select a module simply click on the desired name and a new module will be added and opened.



NOTE: **To add a second module simply repeat the procedure.**

Removal of modules

WARNING: **All settings of the selected module will be deleted if we remove it!**

To remove a module we open desired module and click the Remove module button.

DATA FILE OPTIONS

The folder view on the left side defines the folder where the data will be stored.

Basic recording data setup is available on the Dewesoft Setup screen and includes many settings for data acquisition and data storing.

Setting the filename

On the Dewesoft Setup screen, we predefine the filename to be used for data recording.

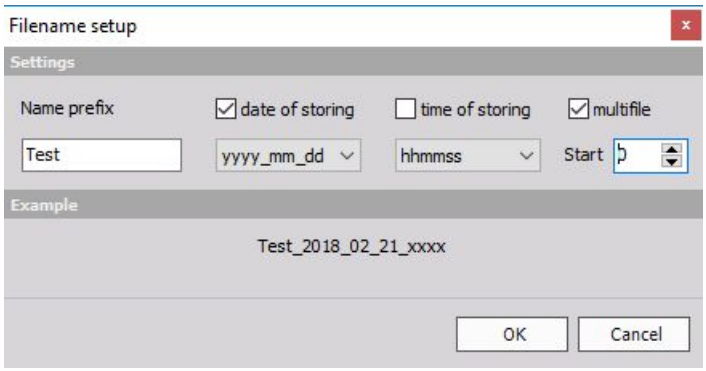
Create a multifile

For repetitive measurements, we can use multifile. Multifile automatically assigns a new file name for each start of storing. File naming can be either consecutive (like 0001, 0002, 0003) or by the date and time.

The Setup screen has a check-box called Create a multifile. If you check it, then Dewesoft will take your base filename, like 'Data' and just add the underscore and four digits to it each time you start storing (in the example bellow '_0003', for the first time this will be '_0000'). Right part of DATA FILE OPTIONS sections on Setup screen also change:

Now the Setup button appears and instead of Stop storing after check-box the Make new file after check-box also appears.

Press Setup button to change the default multifile settings and Filename setup window appear:



The multifile name can have different formats. First, you have to enter the main file name in the Start field. Then you can check the date of storing and/or time of storing (only if the multifile field is unchecked) and select the desired date and time format out of the drop-down list.

If the multifile field is checked, the time of storing will be disabled

- automatic unchecked and in Start field bellow multifile you can choose starting number of the filename.

Above the OK / Cancel buttons you see an example how the file name may look like. If the settings are done, confirm using the OK button. Cancel will discard changes.

WARNING: **The multifile storage is independent of the storage of triggered data. Even if the multifile function is enabled, all trigger events are stored into the same file unless you don't press the STOP button and the START / ARM button again. This will change the file name to the next file number.**

Save multifile option in setup

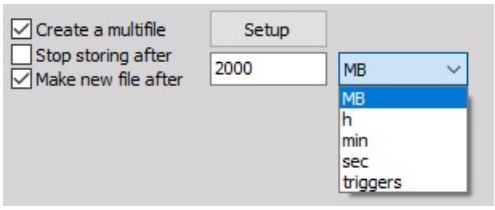
In Dewesoft all the file name settings are stored in setup file by default.

Make new file after

When the Create a multifile box is checked, the Setup screen has an additional check-box called Make a new file after. If you check it, you can enter a value and the unit. The criterion for switching the files is either the file size or time interval.

Please select between file size values like MB (Megabytes) and triggers (number of trigger events) and time-related values like h (hours), min (minutes), sec (seconds).

In this example Dewesoft will stop storing after 2000 MB:



If we select time-related values, we get additional Absolute time field. When this box is checked the absolute time is selected. This is very useful when acquiring data for longer time periods. If we choose to switch the file each hour with absolute time, then switching will be done exactly on the hour (01:00, 02:00, 03:00...). The time will be taken from absolute PC time (or other more exact timing sources, if available - defined in hardware setup). The file switching is done in the way that no data point is lost in between.

Stop storing after

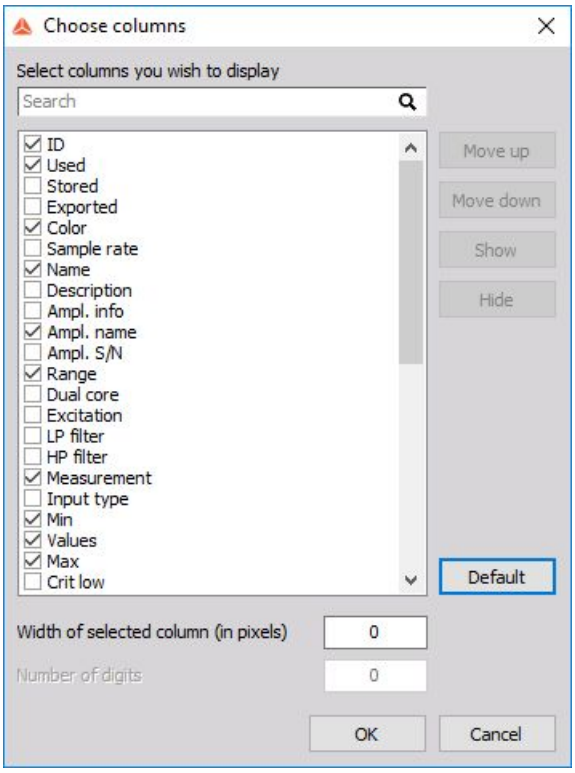
When the Create a multifile box is not checked, the Setup screen has also check-box called Stop storing after. If you check it, you can enter a value and the unit. Please select between file size values and time-related values - all setting are similar as for Make a new file after (see above).

CHANNEL GRID

Channel grid is a simplified overview of available input or output channels (depending on the module you select).

ID	Used	C	Name	Ampl. name	Range	Measurement	Min	Values	Max	Physical quantity	Units	Zero	Setup
1	Unused		AI 1	SIRIUS-ACCv2	10 V	Voltage	-10,00	0,002	10,00		V	Zero	Setup
2	Unused		AI 2	SIRIUS-ACCv2+	10 V	Voltage	-10,00	0,002	10,00		V	Zero	Setup
3	Unused		AI 3	SIRIUS-CHG+	10 V	Voltage	-10,00	0,000	10,00		V	Zero	Setup
4	Unused		AI 4	SIRIUS-HVv2	1200 V	Voltage	-1200,00	0,00	1200,00		V	Zero	Setup
5	Unused		AI 5	SIRIUS-STGv2	50 V	Voltage	-50,00	0,001	50,00		V	Zero	Setup
6	Unused		AI 6	SIRIUS-STGMv3	10 V	Voltage	-10,00	0,170	10,00		V	Zero	Setup
7	Unused		AI 7	SIRIUS-LVv2	200 V	Voltage	-200,00	0,00	200,00		V	Zero	Setup
8	Unused		AI 8	SIRIUS-MUL	10 V	Voltage	-10,00	-0,061	10,00		V	Zero	Setup

If a channel grid is displayed in Analog in, Function generator, Digital in, math, etc.. you can add function columns by right-clicking one of the top row fields or left clicking more options button. Then proceed to Edit columns and a window will appear:



Here you can select the desired columns you want to add to your grid.

Different possible functions through all setups:

- ID number - Identification number of channel, mostly used for reference.
- Used - Button is for activating the desired channel. If Channel is Unused it is not saved nor displayed and can't be used for calculations.
- Store - Defines if the acquired data is saved or not. If the store is not used, the data is acquired and can be calculated but is not stored.
- Exported - Is used to predefine the channels that will be exported if you chose so. The exported channels can also be selected in the Export module.

- Color - Select color of the desired channel.
- Sample rate - Enables us to acquire data with reduced sample rate in order to save data space. The maximum sample rate is defined with Dynamic acquisition rate.
- Name - Name of a channel can be changed to sooth individual needs. If we change the name of a channel, it is automatically changed in formulas or other used calculations. For own relief, refrain from using the same name for multiple channels.
- Description - Is used for personal description or notes.
- Amplifier info - Displays information about the physical amplifier. Typically voltage range, filters, Core count, etc..
- Amplifier name - Name of an amplifier is used to define the type of possible measurements.
- Amplfier S/N - Serial number of an amplifier.
- Range - Select the range of operation.
- Dual core - Activate or deactivate dual core mode. Only with dual-core amplifiers.
- Excitation - Define the required excitation.
- LP filter - Define Low pass filter
- HP filter - Define High pass filter
- Measurement - Select the type of Measurement, for example: Voltage, IEPE, Charge, etc..
- Input type - Select input type, for example: Single ended or Differential if required by the amplifier.
- Min - Defines a theoretical minimum value preset for displays.
- Values - Displayed values of an amplifier, math, output, etc..
- Max - Defines a theoretical maximum value preset for displays.
- Crit low and high - Defines critical warning Low an High level of the channel.
- Warn low and high - Defines Caution Low and High-level warning.
- Hysteresis - For low and high warning.
- Physical quantity - Select the physical quantity that you wish to measure. If the quantity is not present on the drop-down menu, you can select Custom and then define the Units.
- Units - Units of measured Physical quantity can be selected in reference to a selected Physical quantity or entered if Custom is selected.
- Scale - Set scale of the channel.
- Sensitivity - Set sensitivity of channel.
- Offset - Set software offset.
- Offset in amplifier unit - Set offset of the amplifier.
- Sensor model - Displays data of sensor model type if we have a sensor with TEDS.
- Sensor S/N - Displays the serial number of TEDS supported sensor.
- Sensor recall date - Displays the date of required sensor recalibration.
- Target-Shunt calibration check - Input value of Shunt calibration.
- Result-Shunt calibration check - Displays result of shunt calibration check.
- Error-Shunt calibration check - Displays error between Target and Actual value.
- Lead wire compensation - Input resistance value for compensation of lead wires.
- Sensor unbalance - You can input it manually or measure it in Channel setup.
- Self test error - Measured error.
- Zero - This button automatically zeroes out the channel.
- Group - Group Channels.
- Setup - Open setup menu.
- Measurement unit - Defines if the channel is Local or acquired from a remote measurement unit.
- Transferred - If a slave is connected this cell defines if the channel is transferred to the master

unit.

- Transfer rate - The rate at the which the slave is transferred to the master unit.
- Stored on client - Displays if the channel is being saved on a client.

ID NUMBER

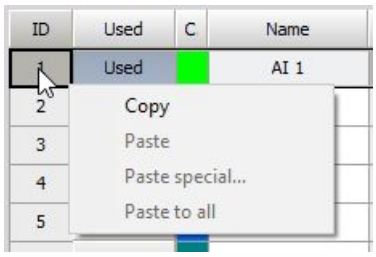
ID column indicates the number of the channel. It usually starts with 1 and counts up to the maximum number of available module outputs.

This column is a direct reference to the slots within your system.

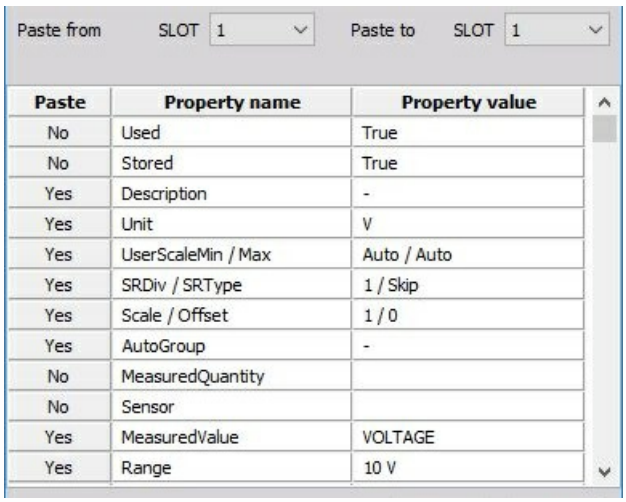
NOTE: **If you have a Dewesoft Sirius system, the first 8 modules are the ones on the mainframe itself.**

This field has also a copy/paste function. When you right-click on a slot number, a window will appear:

Example: several channels (e.g. 1, 2 and 6) contain the same module with the same sensor connected. If you click now on slot 0 and select Copy, the system will remember all the important settings from channel 0, like amplifier type, input and filter ranges, units, calibration and zero settings. Now click on slot 2 and select Paste - this will copy all settings from channel 0 to channel 2 and so forth with other same modules.



Paste special will be given additional options to select a region from where to where the selection will be pasted.



It will also show the properties which will be pasted and give the user a chance to paste only selected properties by clicking on Yes / No button left to the name of properties.

WARNING: **This function is working best with amplifiers of the same type. Properties**

which are not possible to set will be ignored. Channel names will not be copied.

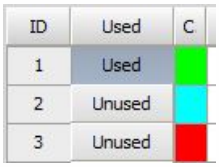
USED - ACTIVATING CHANNEL

With this setting, we can activate/deactivate input channels for online display and storage.

Select input channel for display

Input 1 is already active (marked Used beside it). Click on the input slot to activate them and the screen is displayed like this:

In On/Off column there is a button that you click to toggle this input on/off (Used / Unused). If it says Used, then it will be available in all instruments either as digital values or graph. Use the button to activate/deactivate used channels.



To activate/deactivate all or specific channels at the same time, mouse drag or Ctrl+click the channels.

Select input channel for storage

Inside the channel setup, Store button is available. It is on by default. To disable storing of the channel, depress it. Then this channel will be still displayed and can be used for mathematics, but it will not be stored.

This function has been added especially to avoid storage of not required data (for example you need the data only for mathematics and store the math result).

COLOR - CHANNEL COLOR

Simply click on the colored field of the desired channel to open the color selector:

Choose from any displayed color or create a user-defined one with selecting Define Custom Colors >> button.

Press OK to confirm the change or Cancel to keep current settings.

The color of the channel can also be changed on the Channel setup screen - General part of:

- Analog in channel -> see ->Analog in Channel setup or
- Analog out channel -> see -> Analog out Channel setup

This color will carry through the text and graph representations of this channel throughout all Dewesoft procedure screens.

NAME - CHANNEL NAME

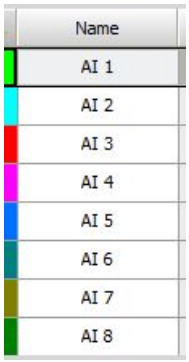
Free text field for naming this channel.

Just click into the field and you can enter the channel NAME as usual text.

Navigation keys are the same as in other Windows application.

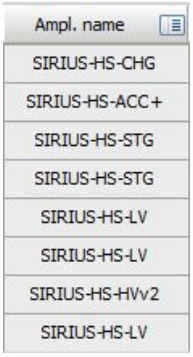
You can also enter this text on the Channel setup screen - General part of:

- Analog in channel -> see ->Analog in Channel setup or
- Analog out channel -> see -> Analog out Channel setup



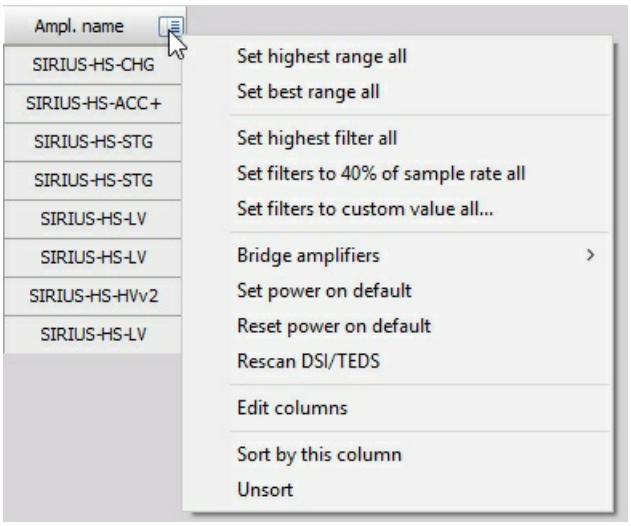
AMPLIFIER TYPE, RANGES AND SENSORS

Shows the name of the module and the range selected.



A small arrow on the right side of the field is running through all fields, indicating that all channels are scanned for new amplifiers or settings.

When you click on one of the Amplifier columns header line (with), a selection field will appear. In this field you can choose from different options which will have an effect on all available channels



For all modules, there is an option to Set PowerOnDefault option. This will keep the current settings of the modules when power is switched off. Reset PowerOnDefault (DAQP only) resets settings to factory default.

If the module is removed from the system during operation, it will turn to a red color. Clear not found modules option will remove all the modules from the list which are not found (all modules in red color).

- Set highest range - will set the range of all amplifiers to the highest possible input range.

- Set best range - will measure the real-time data and set the range of all amplifiers to the best possible fit.
- Set highest filter - will set the analog input filter to the highest possible values or switch it to off, if possible.
- Set filters to 40% of the sample rate - will set the analog input filter to the nearest value of the Nyquist frequency (40% of sample rate) to prevent aliasing.
- Set filters to custom value - will open the dialog and allow the user to enter the value to set the filter to it.
- Fill rack option - will enable an option to start adding modules to the system.

When there is at least one special module, there are more options available which help to work with these modules.

If anti-aliasing filters (AAF prints) are selected, the drop down will show also the option for setting these filters to either bypass or any available low pass filte range.

- Bridge zero - which zeroes the bridge by hardware.
- Amplifier zero - resets this zero value that it has no offset.
- Short on for 2s - shortens the modules for two seconds while
- Shunt on for 2s - switches on the shunt of the bridge module for two seconds.

These options help to quickly see that bridge modules are connected and working correctly.

The options for zero, shunt and short are available also from the Channel setup, short and shunt are available also during the Measurement to determine the start and end offset of amplifiers.

If any Charge modules are installed, the drop-down will give the option to reset the modules. If any FreqA modules are found, there is an option to find the correct analog trigger levels for all the modules.

- Clear not found modules option will remove all amplifiers which are marked in red in the channel list (which were found once, but couldn’t be found anymore) because the amplifier was removed from the system or from any additional reason.
- Disable amplifier option will disable the currently selected amplifier. It means that it will not search for the amplifier at this address.
- Rescan TEDS sensors will scan again the TEDS sensors for amplifiers where the SCAN interferes with data acquisition (IEPE amplifiers).

Sensors and TEDS sensors

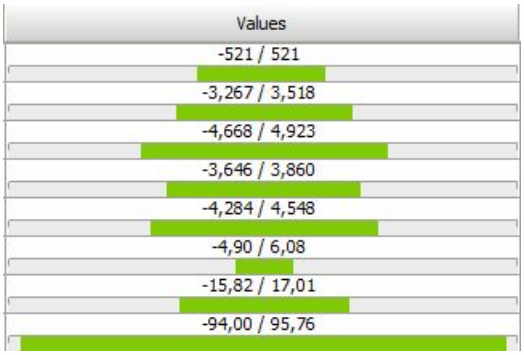
For each analog channel, a sensor can be predefined in the Sensor database and used in the Channel setup for an individual channel. To notify that the sensor is used, the sensor name and serial number are shown instead of the module range.

The same column shows the TEDS sensors if there are any attached to the amplifier. TEDS sensors are sensors with a built-in chip which automatically sends the information like scaling and serial number to the amplifier. This is shown as TEDS sensor in the same place as a user-defined sensor.

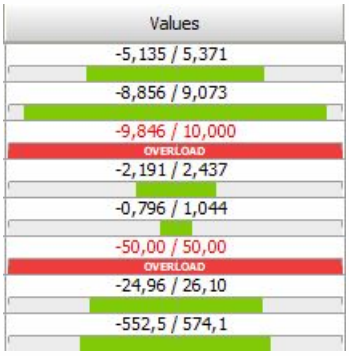
VALUES

Contains a dynamic representation of this input channel, as well as the units of measurement and description, and the scale.

All this can be set in Channel setup column, which can be reached by double click on channel cell in this column.



NOTE: **When the input signal exceeds the possible range, a red indicator Overload will be displayed.**



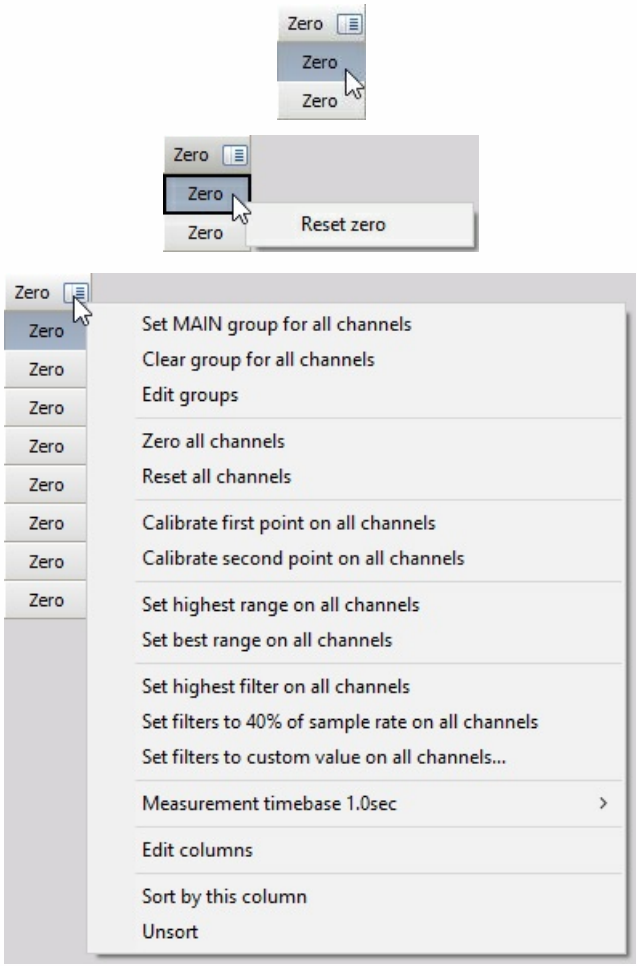
If this happens, check your sensor and/or select another input range (in Channel setup).

By clicking on the PHYSICAL VALUES caption, a drop down which values are shown is displayed. Normally Display measured values is selected. In the case when we have bridge amplifiers, there are additional option to display bridge balance or shunt calibration values.

INPUT ZEROING

This is a button that you can click to perform a mathematical zeroing of this input, to offset small variations in the zero position of the input.

Press the left mouse button to activate zeroing, and the right mouse button to deactivate it (reset to default input range).



- Zero all AUTO channels option will perform a channel zero for all channels set to auto.
- Reset all AUTO channels will set zero offsets to all channels.
- Calibrate the first point on all AUTO channels option allows calibrating all the channels at once.

Usually, this is useful when the CAL signal is available on all the inputs. If we have the option to put electrical 0% and for example 80% on all the channels, we can enter the 0 as the first point in the channel setup and 80% as the second point in the Channel setup for each channel, set all the channels to AUTO and then first apply 0%, press Calibrate first point, then apply 80% and press Calibrate Second point on all AUTO channels.

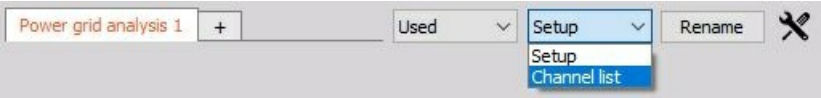
- Highest and best range selection works the same as for all channels (selectable from Amplifier section), but only on specific channels set to Auto.
- We can also choose the Calibration time base as 0.1 sec (Ave) or 1.0 sec (Ave) (same as in

Channel setup). 0.1 second will give faster response time while 1 second will provide more averaging time to perform calibration.

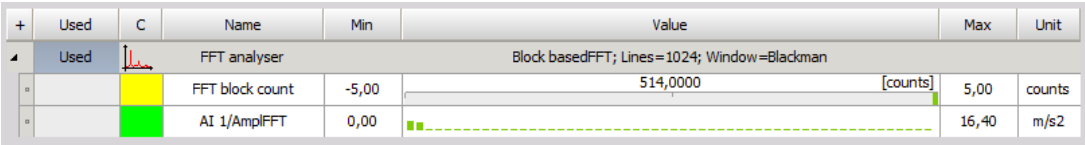
OUTPUT CHANNEL LIST

Output channel list contains a grid of all parameters calculated by the selected module.

To display all channels go to the second tab (Setup) in edit module section and select the Channel list from the drop-down menu.

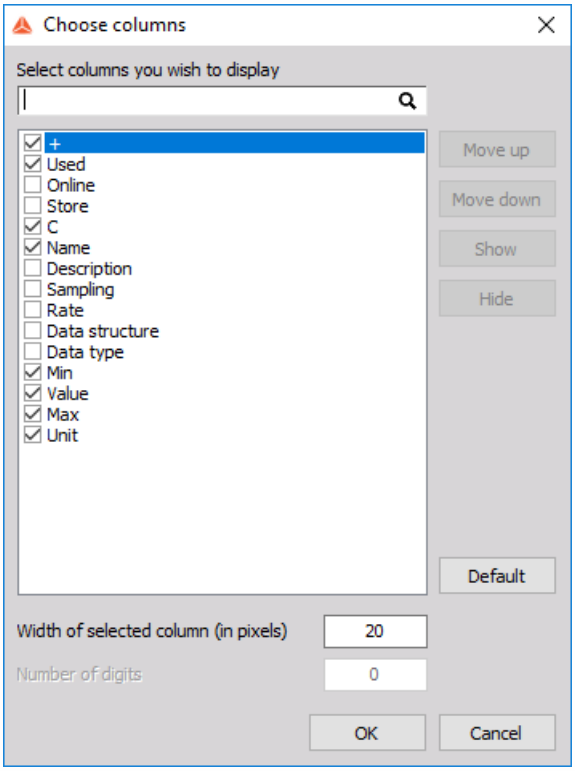


First, let's look at a sample of Channel list for FFT analyser module and it's columns.



If a channel grid is displayed in FFT analyser, Combustion analyser, Modal test, etc.. you can add function columns by right-clicking one of the top row fields or left clicking more options button.

Then proceed to Edit columns and a window will appear:



- + - Collapse or display all calculated output parameters under one module
- Used - Button is for activating the desired channel. If Channel is Unused it is not saved nor displayed and can't be used for calculations.
- Online - Defines if calculations are made in Measure "on the go" or in Analyse after
- Store - Defines if the acquired data is saved or not. If the store is not used, the data is

acquired and can be calculated but is not stored.

- Color - Select color of the desired channel.
- Name - Name of a channel can be changed to sooth individual needs. If we change the name of a channel, it is automatically changed in formulas or other used calculations. For own relief, refrain from using the same name for multiple channels.
- Description - Is used for personal description or notes.
- Sampling - Displays the type of sampling, Asynchronous, Synchronous or Single value.
- Rate - Displays the rate of sampling. If Asynchronous or Single value, in case of Synchronous it displays the Bandwidth.
- Data structure - Scalar or vector (and type of vector) output.
- Data type - Single-precision, Double precision, Integer or Complex used according to calculation requirements.
- Min - Defines a theoretical minimum value preset for displays.
- Values - Displayed values of the amplifier, math, output, etc..
- Max - Defines a theoretical maximum value preset for displays.
- Unit - Units of measured Physical quantity can be selected in reference to a selected Physical quantity or entered if Custom is selected.

GENERAL

Channels

Storing

Analog in

Functions generator

CAN bus

Counters

Analog/Digital out

GPS

Digital in

Math

Alarms

Video

Security

User inputs

Cursors

CHANNELS

Enter topic text here.

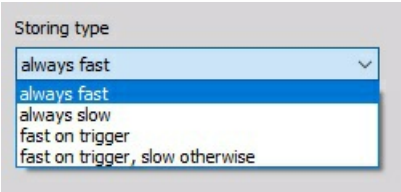
CHANNEL SETTINGS

Enter topic text here.

STORING

Storing strategies

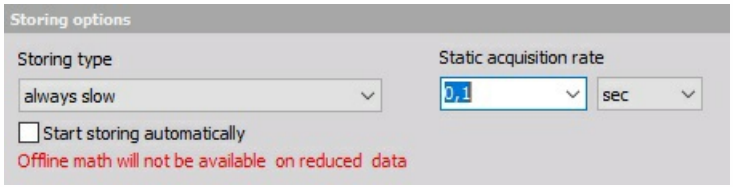
The Storing strategies are very important for the whole system. You can select four different types of storing from the STORING OPTIONS drop-down list:



- Always fast - The data will be stored all the time with the selected Dynamic acquisition rate. Now the data will be stored in the file at full speed.
- Always slow - This function stores data at intervals, set with the Static/reduced rate.

Even though storing is set to slow, Dewesoft will acquire the data at full speed, calculate minimum, maximum, average and RMS for this time interval and store only these values.

In example below it is set to 0,1 second, in this case much less disk space will be used for storing:



If our data consists of events which can be captured, we can choose to store with two options for triggers. The trigger event can be defined in the software and then Dewesoft will wait for this event and store only the portion of interest.

- Fast on trigger - The data will only be stored with the full acquisition rate when the trigger condition is true.
- Fast on trigger, slow otherwise - To be able to acquire data with two speeds: stores data with a reduced acquisition rate until the trigger condition became true - after then full speed is active.

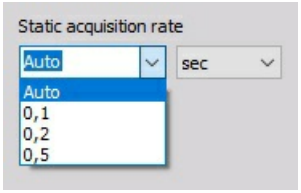
Static / Reduced rate

When the static mode is selected, the system will still run at the dynamic sample rate shown in the box DYNAMIC ACQUISITION RATE beside, however, every data point will NOT be stored. Instead, the system will reduce the data continuously according to the static/reduced rate selected here.

You can select the value from the drop-down list, like 0.1 seconds. This means that every 0.1 seconds the reduced data will be available.

If you select AUTO, the static/reduced rate will be adapted automatically depending on the setting of the dynamic acquisition rate.

You are also allowed to enter your own values, but be aware that not all values will be accepted - if so the real value will be mentioned below the drop-down list.



For example, if we have a dynamic rate of 1000 Hz, and a static rate of 5 seconds, and we check that we wish to store the static rate, the system will sample at 1000 Hz, but will store a set of min/max/ave/rms values for each input every five seconds.

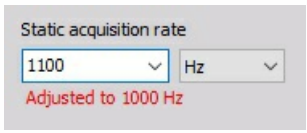
This means that 5000 samples are going by for each channel, every five seconds. Are they ignored? No - they are used to derive the min/max/ave/rms values which are saved for each active input, at the static rate!

So, each set of reduced samples are based on all 5000 samples that were seen for each channel between static intervals.

IMPORTANT: **When the system does this, it does not make a single column of data for each channel - instead, it creates multiple columns of useful data for each channel, including the min/max, RMS, and average values that were seen between each interval of the static/reduced rate.**

Change static / reduced rate scaling

Sometimes it is required to change the units for the stating / reduced rate (Hz, sec, min, samples). Simply click on the small arrow below the drop-down list to select another scaling. For each unit (scaling) Dewesoft adapt AUTO value automatically (and show in Adjusted to the field), depending on the setting of the dynamic acquisition rate. When 10000 Hz dynamic acquisition rate is selected, see left for examples of Adjusted to values.

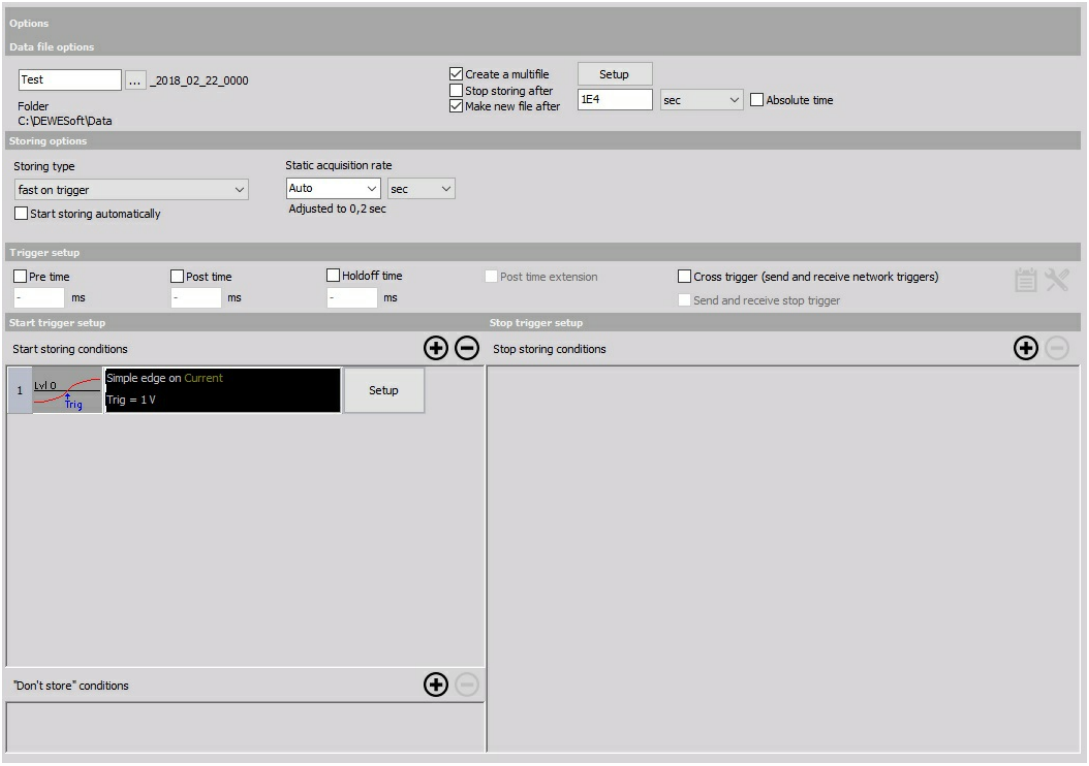


When you switch to the Analyse mode and load one of these files, you can export any set of values: min/max, average, or RMS.

Triggered storing

When you select one of trigger storing option, new Trigger tab will automatically appear on Dewesoft Setup screen.

NOTE: **With Dewesoft you can trigger from your signals by setting any channel(s) to start and stop recording according to levels.**



On this screen we can set start and stop trigger conditions:

- Trigger setup - to set the storage time before and after the trigger event
- Start trigger setup - defines the start and stop trigger condition; with Don't store setup
- Stop trigger setup - defines when the storage should be stopped

Timing setup

The Timing setup can be used to influence the storage before and after the trigger event. Four settings can be used to achieve the desired result:

- Pre-time - Pre trigger time, defined in milliseconds. This value defines the storage duration before the trigger event occurs -> Dewesoft will keep the data in the buffer until the trigger event occurs and then store also this data to the file. As a standard, this feature is not selected and the storage starts with the trigger event itself.
- Post time - Post trigger time, defined in milliseconds. This value defines the storage duration after the trigger event has been finished -> Dewesoft will continue to store until we stop it manually or stop condition occurs. As a standard, this feature is not selected and the storage stops immediately after the trigger event is over.
- Holdoff time - Gives you the possibility to suppress trigger events for a certain time after the last event had happened. This feature is not selected as a standard and will normally be used when you have plenty of events or very long storage times.
- Post time extension - The post time extension is checked automatically as long as the Post time is not selected. The acquisition duration will be prolonged when further trigger events

appear while the first one is still recorded.

The following example is set to 0,1 sec Pre and 0,2 sec Post time, so we will capture 300 ms of data in total per trigger event:

Start trigger setup

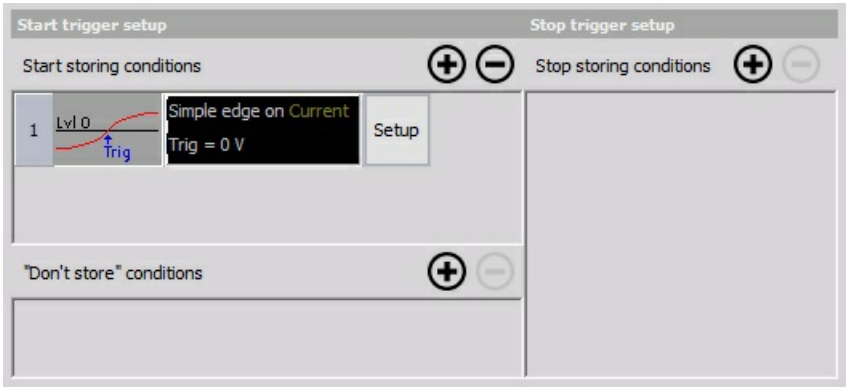
When you select the trigger setup for the first time, there is no start trigger condition defined:

When you press:

the Add button, a new trigger condition will appear immediately in the list

the (minus) button, the selected trigger condition is removed from the list

Use the Add button to add a new Start trigger condition:



After pressing Add button beside this new trigger Start storing condition also appear:

- empty "Don't store" conditions section on Start trigger setup partbof the screen
- empty Stop trigger condition in the new Stop trigger setup column

As a standard, the trigger condition is set to Simple edge trigger with the positive edge at the first active analog input channel.

When we select the Setup button on the trigger condition line of Dewesoft Setup screen, the Condition setup window appears so we can enter the trigger condition settings.

NOTE: **For detailed information about Trigger condition settings see -> User Guide ->**

Trigger Condition setup.

Store trigger settings

We can achieve the trigger condition settings also by simply pressing the Lock trigger (Link store trigger) button in the Store trigger settings section of the scope. it will take current pre and post time, trigger source and trigger level.

"Don't store" setup

The Don't store trigger condition and Setup works in exactly the same way that the Start trigger setup (see above). This function can be used to suppress data storage for the defined condition.

For example, a machine is producing a certain part and you measure the pressure. The system should trigger when the pressure grows above a certain limit. But you only want to store the data if there is really a part in the machine. Then you can connect a signal to another channel which reports if the unit is present or not and use this result as a Don’t store function.

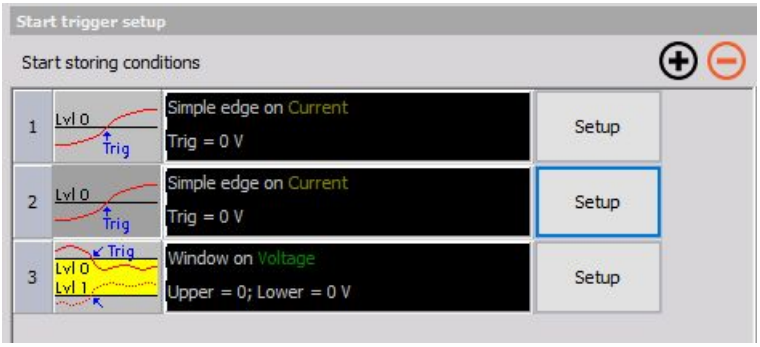
Stop trigger setup

The Stop trigger condition and Setup works in exactly the same way that the Start trigger setup(see above). The only difference is that this section defines when the storage should be stopped.

Use the Post time feature from the Timing setup to prolong the storage time if required.

Remove a trigger condition

To remove a trigger condition simply click on the trigger illustration or the description field to select the condition, the selected field becomes darker - and press minus button.



Trigger control elements

To activate the selected trigger condition, just switch into an online display (scope, recorder,...) and press the Arm button on main Dewesoft toolbar or F5 on the keyboard. As soon as the trigger condition appears on the input, the data will be stored into the file.

When the trigger condition is activated, we see additional Trig button which tells us that we are using triggered storing. We can also press this button to issue the manual trigger.

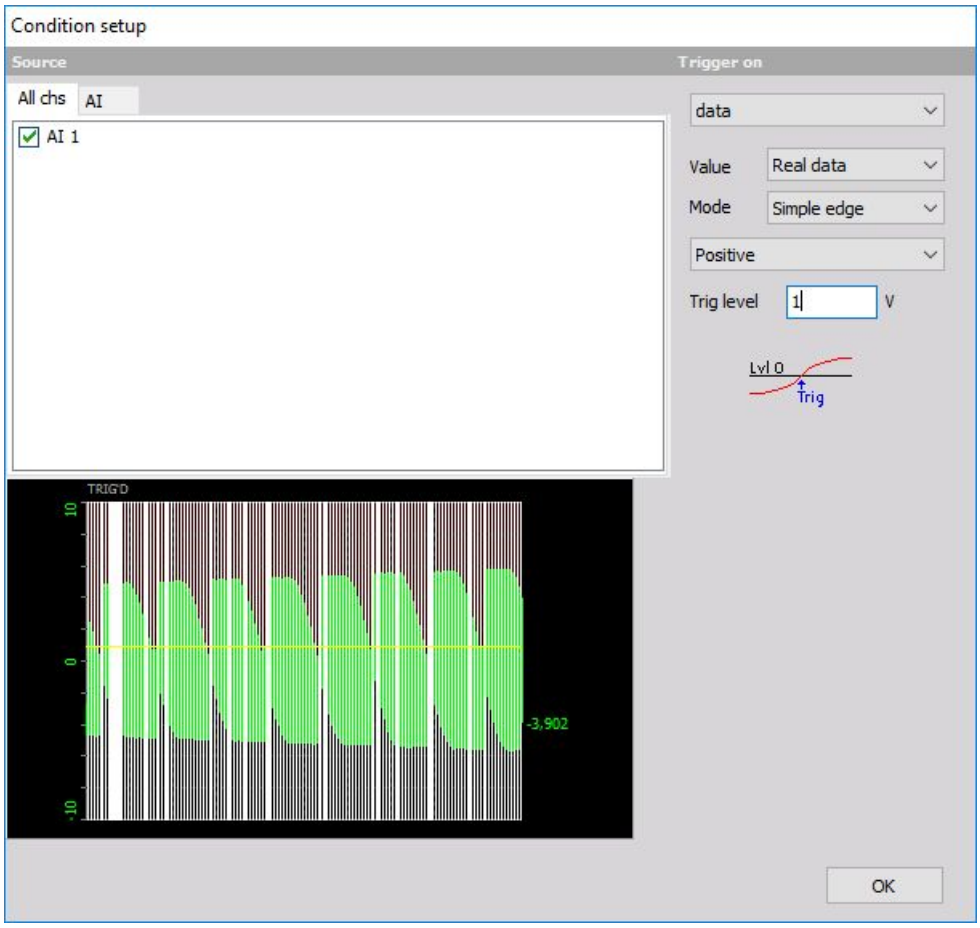
If you want to force a trigger event even when isn't available, press the Trig button or F6 on the keyboard. Trigger shots will appear in the scope and the Trig button flashes.

If several trigger events appear, all of them will be stored in the same file. Dewesoft offers a special analysis feature in the Analyse mode for an easy data analysis.

TRIGGER CONDITION SETUP

Dewesoft offers several different trigger conditions, which can be set on Condition setup window. Trigger conditions can be combined completely independent, that combined with an OR function. That means any defined trigger condition has to become true to activate the trigger.

When we select the Setup button on the Dewesoft Setup screen - trigger condition line, the following Condition setup window appears:

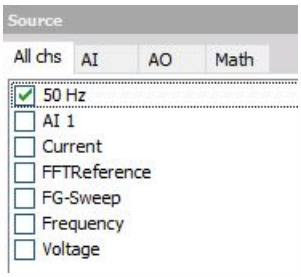


Choose the trigger condition according to your requirements and press the OK button to accept the trigger settings.

In Condition setup window we can enter all the trigger condition settings. Source section on screens left side and in upper part available Channels are displayed in two tabs:

- All chs - all channels
- AI - analog input channel

First of all, you have to select the desired channel out of the All chs / AI tab list. It displays all available channels. To select channel click on it. Selected SOURCE is surrounded with dashed white line and box before channels name is crossed.



In the lower part, the current signal value is displayed. The white lines displayed to indicate when the trigger condition became true.

Trigger on - on the screens right side are fields to set up trigger; this fields depends from selected trigger type; below trigger settings symbolic trigger curve is displayed.

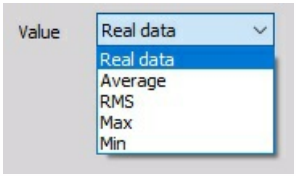
Trigger type

The following trigger types are supported:

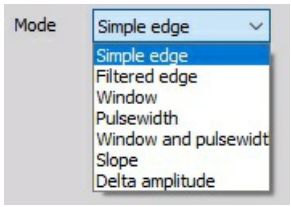
- Data
- Time
- FFT

Data trigger setup

1. Define the Value - When the data trigger is selected, you can also choose between Real data, Average or RMS from the drop-down list for your trigger condition.



2. Define the Mode - Select the trigger type Simple edge, Filtered edge, Window, Pulse-Width, Window and pulse-width, Slope or Delta amplitude from the drop-down list.

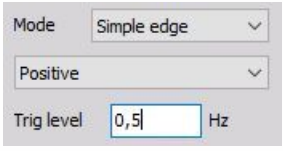


3. Setup other trigger condition - These settings (e.g. Slope, Trigger level, Rearm level, Pulse time...) depend on the selected trigger type in Mode field.

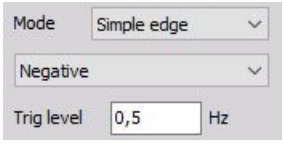
Trigger Mode and setting

- **Simple edge** - This is the most used trigger condition with data acquisition systems. The trigger event is a rising or falling edge, which crosses a defined level.

Positive slope:



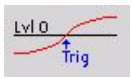
Negative slope:



Trigger on rising edge when the signal rises over defined Trig level.

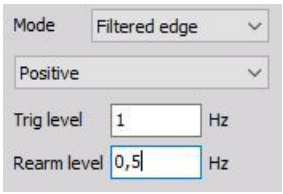
Trigger on falling edge when the signal drops bellow defined Trig level.

Symbolic display of trigger condition (below settings):

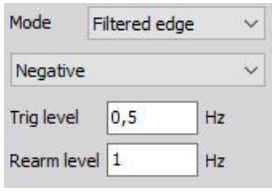


- **Filtered edge** - This is basically the same as the simple edge trigger, except for the rearming level. This level can be used to define a second level, which must be crossed before the trigger condition can become true again. This trigger type is mostly used with very noisy signals.

Positive slope:



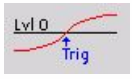
Negative slope:

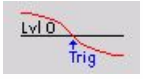


Trigger on rising edge when the signal rises over defined Trig level; re-triggers only when Rearm level has been crossed.

Trigger on falling edge when the signal drops bellow defined Trig level; re-triggers only when Rearm level has been crossed.

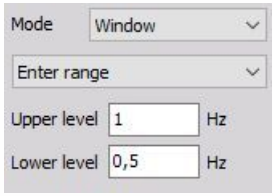
Symbolic display of trigger condition (below settings):





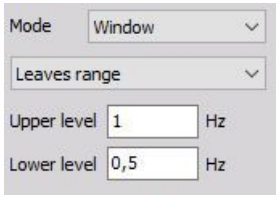
- **Window** - works with two independent levels, which build some kind of window. The trigger condition can become true when the signal enters or leaves the window.

Enter range:



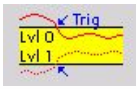
Trigger when the signal enters the window - signal falls below Upper level or rises above Lower level.

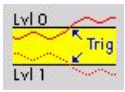
Leaves range:



Trigger when the signal leaves the window - signal rises above Upper level or falls below Lower level.

Symbolic display of trigger condition (below settings)

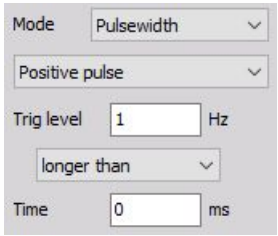




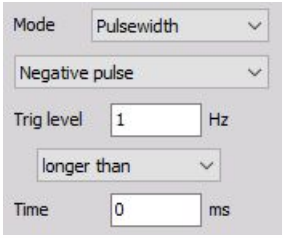
- **Pulse-Width** mode

Longer than Time - checks in addition to the level (like the simple edge trigger) the duration Time of the event and triggers only if the event is longer above the selected level

Positive pulse - Trigger on rising edge when the signal rises over defined Trig level and stays above this level longer than selected Time.

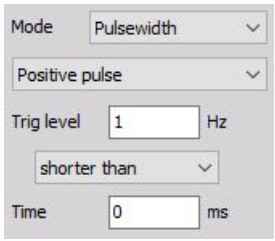


Negative pulse - Trigger on falling edge when signal drops bellow defined Trig level and stays below this level longer than selected Time.

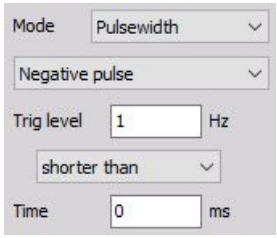


Shorter than Time - checks in addition to the level (like the simple edge trigger) the duration Time of the event and triggers only if the event is shorter above the selected level.

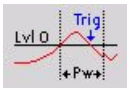
Positive pulse - Trigger on rising edge when signal rises over defined Trig level, but falls below this level earlier than selected Time.

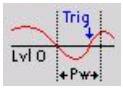


Negative pulse - Trigger on falling edge when signal drops bellow defined Trig level, but rises above this level earlier than selected Time.



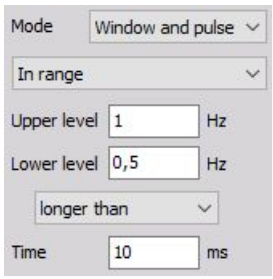
Symbolic display of trigger condition (below settings):



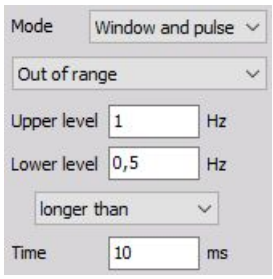


- **Window and Pulsewidth** condition combines the features of the window and the pulse-width trigger; it is very powerful, but you really have to know what you expect to trigger on.

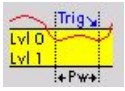
Longer than time (level in range) - Trigger when the signal enters the window (signal falls below Upper level or rises above Lower level) and stays inside for a longer than defined Time.

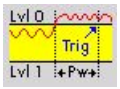


Longer than time (level out of range) - Trigger when the signal leaves the window (signal rises above Upper level or falls below Lower level) and stays outside for a longer than defined Time.

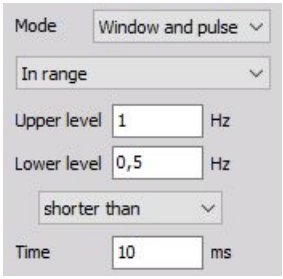


Symbolic display of trigger conditions:

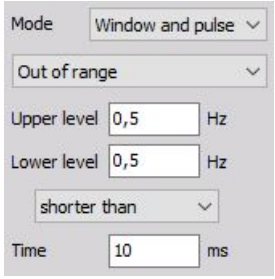




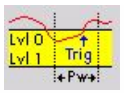
Shorter than time (level in range) - Trigger when the signal enters the window (signal falls below Upper level or rises above Lower level) but leaves before the defined Time is over.

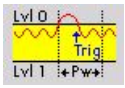


Shorter than time (level out of range) - Trigger when the signal leaves the window (signal rises above Upper level or falls below Lower level) but returns before the defined Time is over.



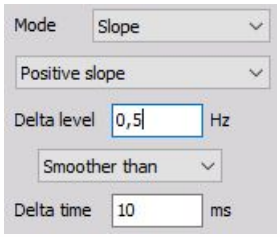
Symbolic display of trigger condition:



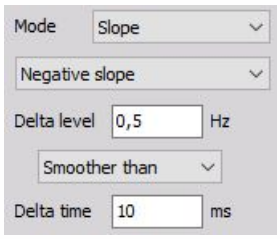


- **Slope** mode

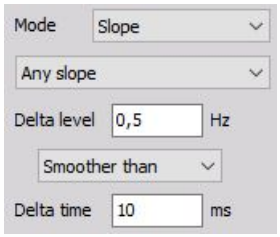
Smoother than Delta time (positive slope) - Triggers when signal rises over defined Delta level later than defined Delta time.



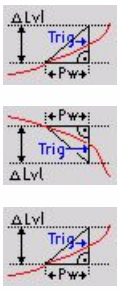
Smoother than Delta time (negative slope) - Triggers when signal drops below defined Delta level later than defined Delta time.



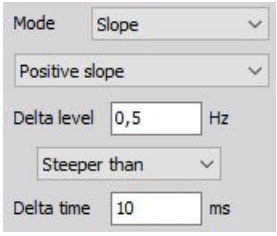
Smoother than Delta time (any slope) - Triggers when signal rises over or drops bellow defined Delta level later than defined Delta time.



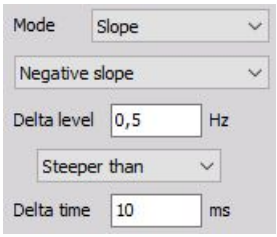
Symbolic display of trigger condition:



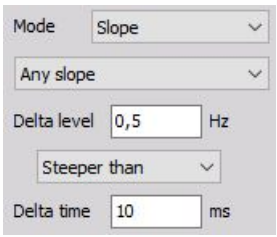
Steeper than Delta time (positive slope) - Triggers when signal rises over defined Delta level earlier than defined Delta time.



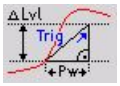
Steeper than Delta time (negative slope) - Triggers, when signal drops bellow, defined Delta level earlier than defined Delta time.

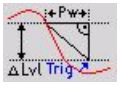


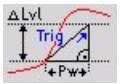
Steeper than Delta time (any slope) - Triggers, when signal rises over or drops bellow, defined Delta level earlier than defined Delta time.



Symbolic display of trigger condition:

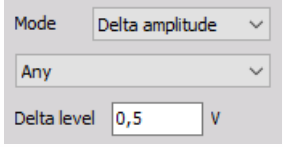




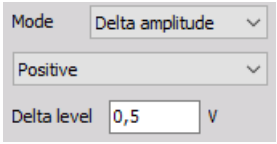


- **Delta amplitude** mode

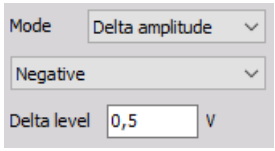
Any slope - the trigger event in this mode appears every time the amplitude of the trigger channel increases or decreases for the defined delta level



Positive slope - Triggers every time the signal increases or decreases over the defined Delta level.

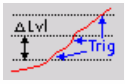


Negative slope - Triggers every time the signal increases over the defined Delta level.



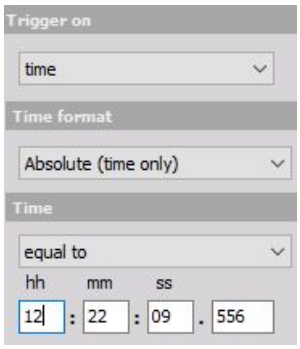
Triggers every time the signal decreases over the defined Delta level.

Symbolic display of trigger condition (below settings)

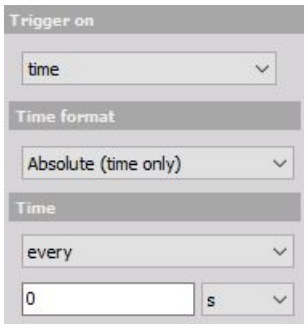


Time trigger setup

Absolute (time equal to) - The system triggers exactly at the defined time hh:mm:ss.xxx (also every day if the time matches).

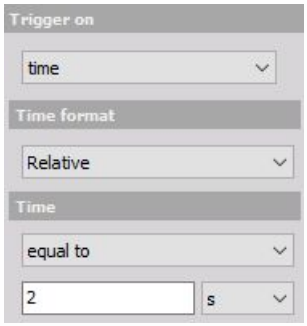


Absolute (every) - The system triggers every defined time x (unit), the time starts running from the beginning of the measurement.

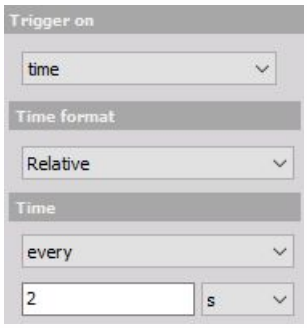


Displayed time:

Relative (time equal to) - The system triggers when the defined time x (unit) has been passed since the measurement has been started.

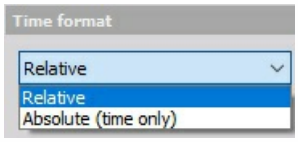


Relative (every) - The system triggers every time x (unit), the time starts running from the beginning of the measurement.

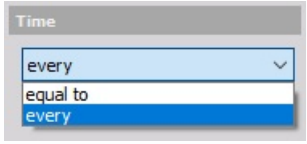


Displayed time:

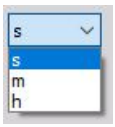
Time format selected from the drop-down list:



Time selected from the drop-down list:



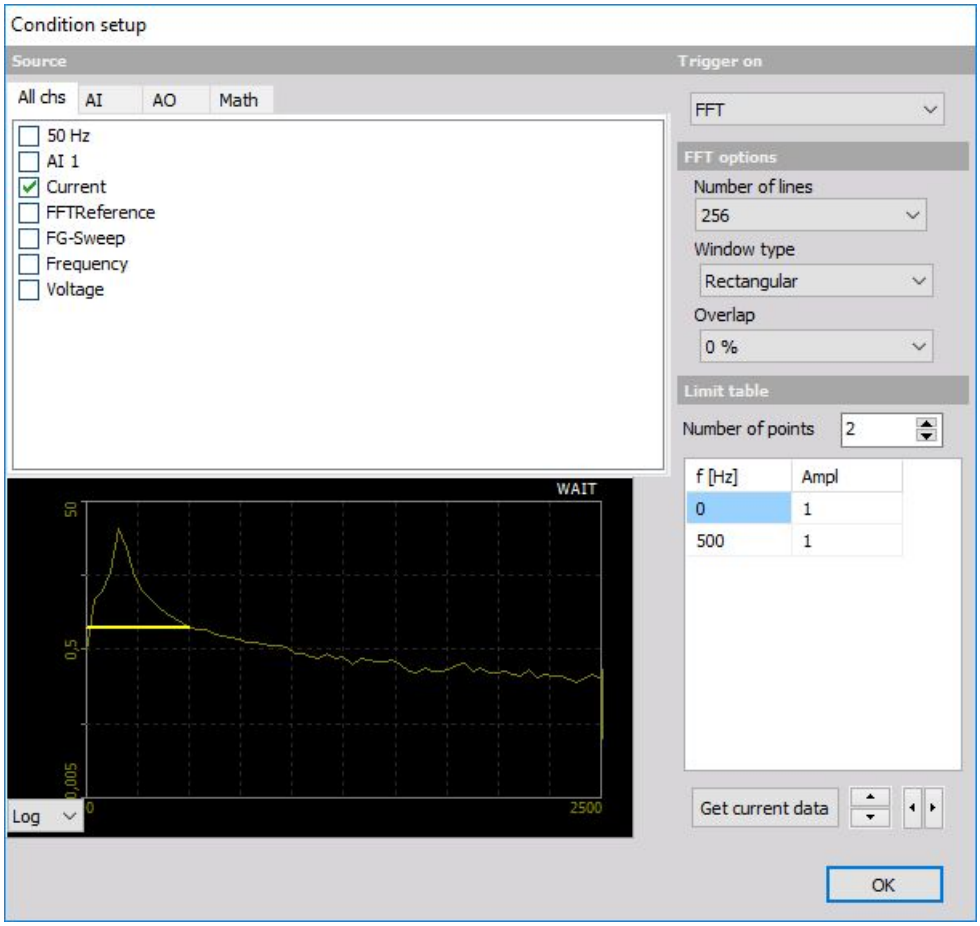
Units selected from the drop-down list:



FFT trigger setup

Up to now, we triggered the system only on amplitude values over the time and/or directly on time. The FFT trigger allows us to trigger on amplitude values in the frequency domain!

This type of trigger is very helpful in any kind of dynamic applications where you want to supervise the frequency behavior of the system under test.



FFT options

To trigger on frequency changes you have to define the FFT options to get a useable result to trigger on:

- Number of lines (256 to 64k),
- Window type (Rectangular, Hanning, Hamming, Flat Top, Triangle, Blackman and Exponent down) and
- Overlap (0, 25, 50, 66 and 75%)

For information about Number of lines, Window type and Overlap see FFT instrument settings

For background information about FFT analysis see Theory of frequency analysis

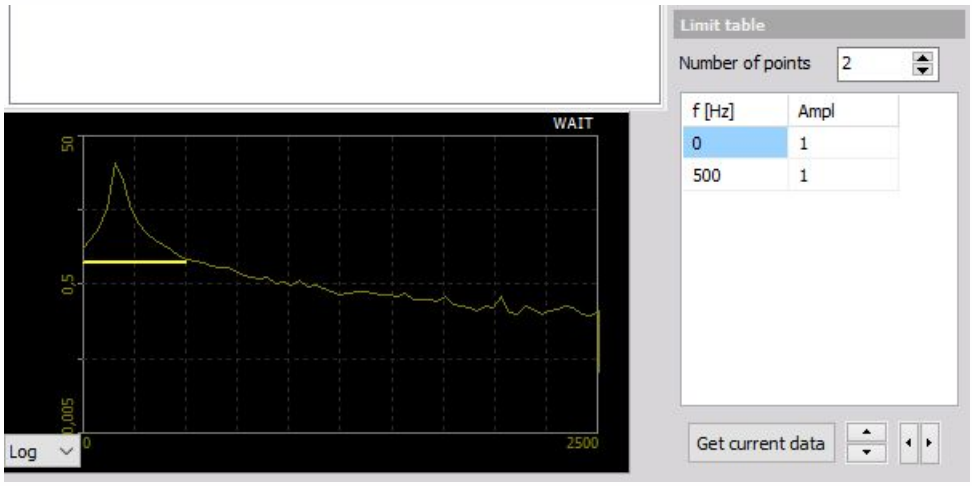
NOTE: **Preview at the left bottom area shows the change effects on the FFT immediately. On this display can be select beside Logarithm (see display above) also Linear display.**

Limit table

After you have done your FFT option settings you have to define:

- Number of points (limits on Limits table)
- The limits - Ampl. levels for f [Hz] on Limits table.

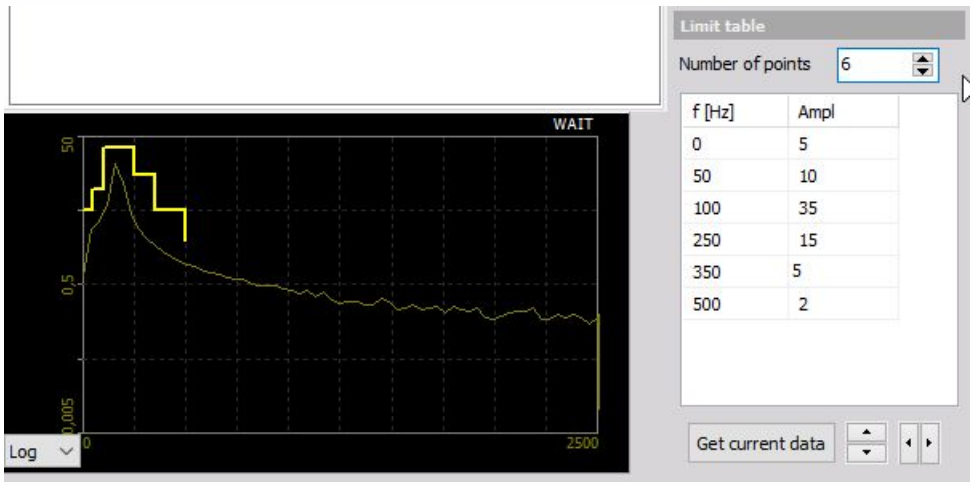
Default (standard) Number of points are 2. The standard Ampl. level is 1 for 0 and max. frequency. You can click on this field and change this value.



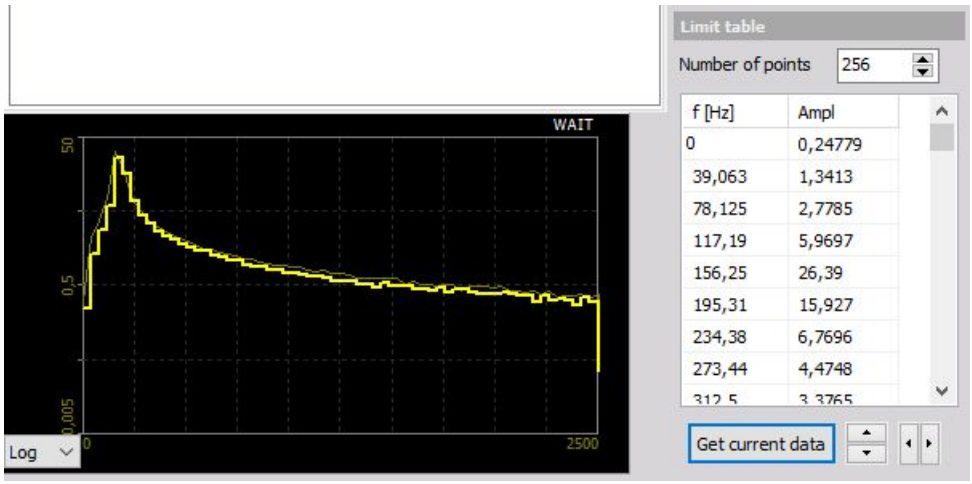
You can increase Number of points to expand the table:

You can now define the limits in two ways:

1. Enter the values manually - When you enter the values manually into the table, you normally take just a view points to define the frequency mask.



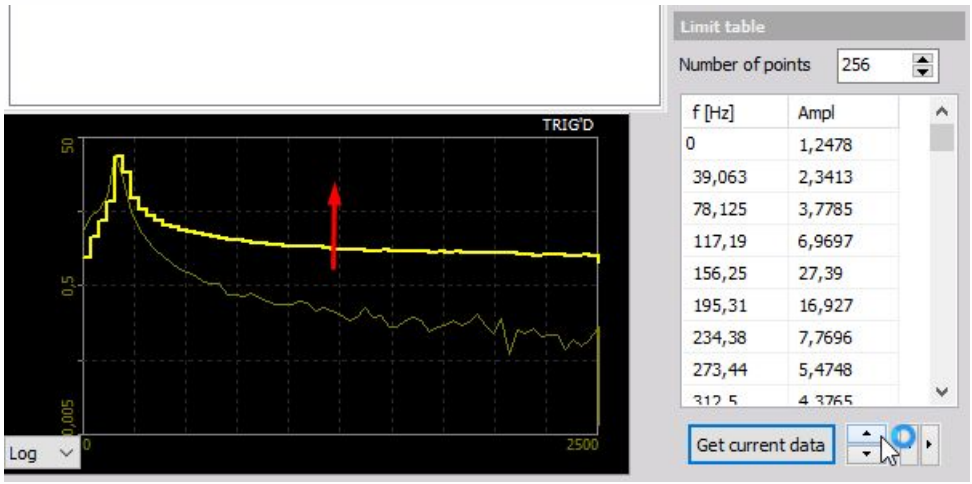
2. Take the current measurement from the system - The second way is to take a frequency mask out of the currently displayed signal. To do that simply presses the Get current data button. The currently calculated FFT will be stored as a mask and displayed both on the preview display and in the table at the left bottom.



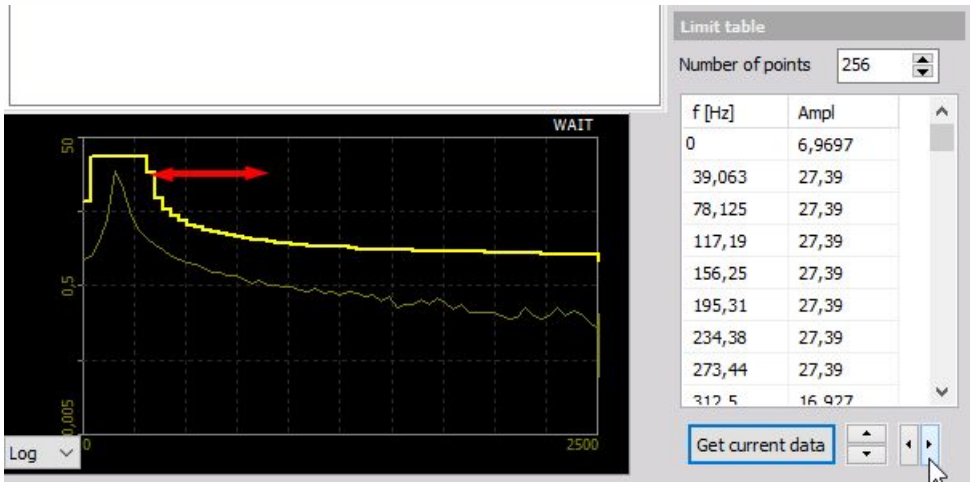
Now you can manipulate the mask by editing the table or - much faster -press the button to move the limit up/down or press the button to add/remove limit in the frequency domain.

Examples:

Frequency mask moved up:



Frequency limits widened:



DATA HEADER

For more information please see Data header and Header information chapters in a manual.

SYSTEM MONITOR

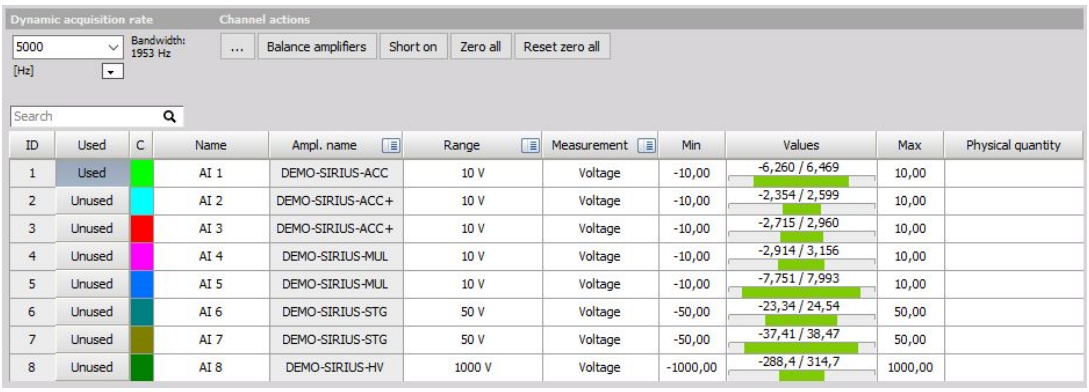
Enter topic text here.

ANALOG IN

In Dewesoft we can define and set all analog input channels which will be used during measure using installed measurement hardware.

After adding a new module (Analog in is a default module and must be added only if previously removed), the Analog in the tab on Dewesoft Setup screen can be selected to set all analog input channels and this screen displays two main parts:

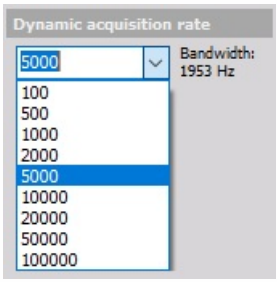
- Dynamic acquisition rate - defines the sample rate of analog, counter and digital channel
- Channel actions - shortcuts to amplifier commands
- Analog IN channel list - with information and settings in columns including open Channel setup button.



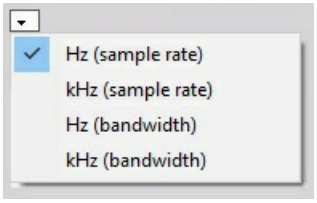
ID	Used	C	Name	Ampl. name	Range	Measurement	Min	Values	Max	Physical quantity
1	Used		AI 1	DEMO-SIRIUS-ACC	10 V	Voltage	-10,00	-6,260 / 6,469	10,00	
2	Unused		AI 2	DEMO-SIRIUS-ACC+	10 V	Voltage	-10,00	-2,354 / 2,599	10,00	
3	Unused		AI 3	DEMO-SIRIUS-ACC+	10 V	Voltage	-10,00	-2,715 / 2,960	10,00	
4	Unused		AI 4	DEMO-SIRIUS-MUL	10 V	Voltage	-10,00	-2,914 / 3,156	10,00	
5	Unused		AI 5	DEMO-SIRIUS-MUL	10 V	Voltage	-10,00	-7,751 / 7,993	10,00	
6	Unused		AI 6	DEMO-SIRIUS-STG	50 V	Voltage	-50,00	-23,34 / 24,54	50,00	
7	Unused		AI 7	DEMO-SIRIUS-STG	50 V	Voltage	-50,00	-37,41 / 38,47	50,00	
8	Unused		AI 8	DEMO-SIRIUS-HV	1000 V	Voltage	-1000,00	-288,4 / 314,7	1000,00	

Dynamic acquisition rate

Enter or drop-down and select a sample rate to be used for dynamic acquisition. The maximum sample rate allowable will vary according to which A/D board you have installed, and how many channels are activated for recording. It is important to understand the difference between dynamic and static/reduced rate acquisition. Note that there is a Storing option drop-down list to select different storage options.



When storing dynamically, every sample is stored. This means if you have ten channels activated, and you enter a dynamic acquisition rate of 5000 samples/sec/ch, the resulting data file will contain 5000 samples per second of acquisition for each channel. In other words, every sample point is recorded in the file.



Change sampling rate scaling

Sometimes it is required to change the units for the sampling rate (Hz/ch, kHz/ch, MHz/ch). Simply click on the small arrow below the drop-down list to select another scaling.

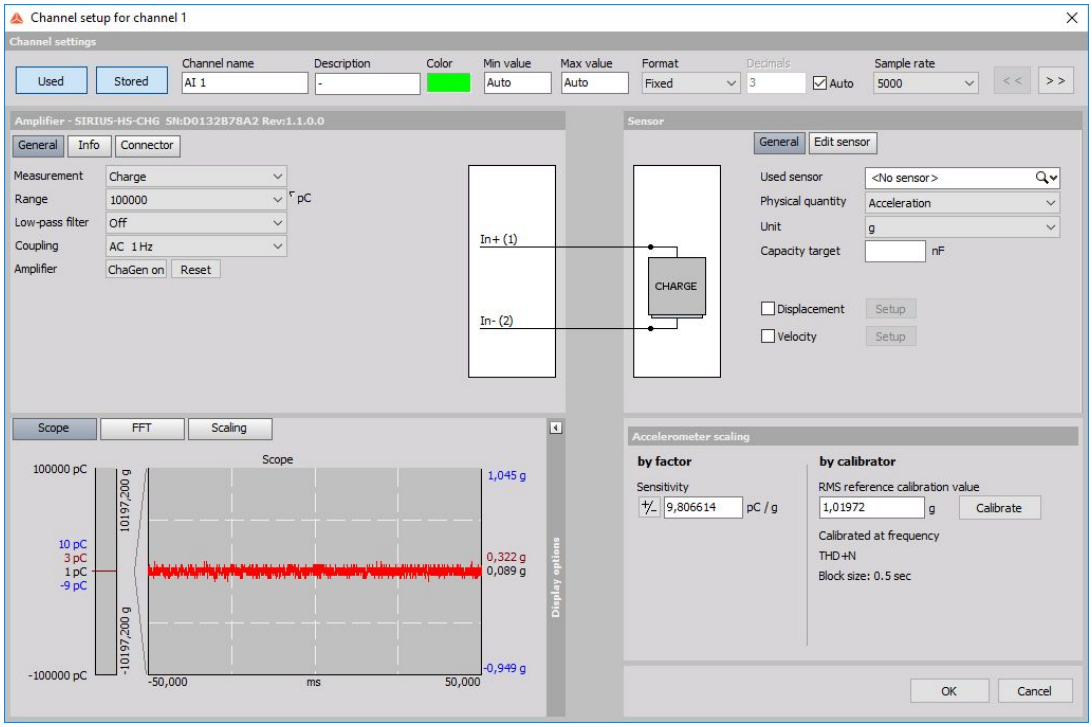
Only the synchronous channels are influenced by the dynamic acquisition rate, so raising the sample rate will increase the amount of stored data. The asynchronous channels (like CAN, GPS, ...) are not influenced by the sample rate, we only need to take care that the dynamic rate is faster than the rate of data coming from the asynchronous device.

NOTE: **Set the sampling rate before you do anything else; this setting will also be used for the setup. This is important to achieve a useful scaling!**

CHANNEL SETUP FORM

To set up and calibrate any input, just click the Setup button on SETUP column for that input, e.g. for set channel 1 press Setup button in the last column.

This is a button that calls up the dialog box Channel setup for channel. This setup window will appear:



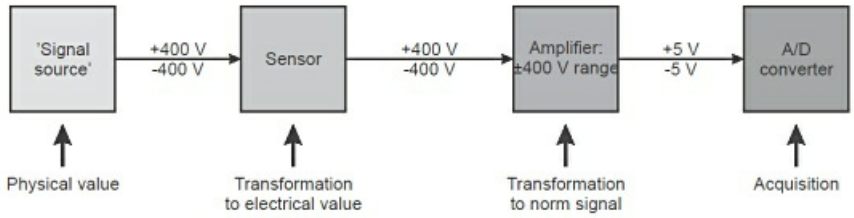
The channel setup window is divided into next segments:

- Channel settings - you can enter the Channel name, Description, Color, display range - Min value, Max value, format, and Sample rate divider
- Amplifier settings - define and set amplifier settings: Non-programmable, series and special programmable modules
- Sensor settings - define and set sensors information and data in the sensor database
- Scaling - perform scaling and manual or automated calibration; set a dynamic representation of your signal
- Calibrate - The ability to quickly calibrate directly to DC and AC signal sources is a great advantage of Dewesoft
- Live data preview - Real time display of the acquired signal.

After all settings are done we must confirm all changes with selecting OK bottom or discard changes with Cancel button and return to Analog input channel Setup screen.

Example: Principle module connection

A voltage up to ±400 V (= signal source) will be measured with cables (='sensor') connected to a high voltage module (=amplifier).



GENERAL CHANNEL SETTINGS

In this General part of Channel settings screen you can enter the:

- Channel name and Description
- Units of measurement
- Define Color (click on the color bar to change the color for this input)
- Display range - Min value, Max value
- Format of displayed results
- Change Sample rate of saved data for this channel
- Switch to next or previous channel setup

Description

Description of the channel can be used for personal reference and is not used in computing of data.

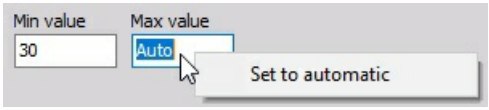
Min / Max value

The Min value and Max value fields represent a definable standard display range:

- As a standard, Dewesoft displays the whole input range of the channel in each display (eg the recorder). These fields are set to Automatic.
- If you now have a wide input range, but your signal is small you can define this field as narrow standard display range. You can enter range as the usual number; navigation keys are same as in other MS Windows application, Enter key also closes Channel setup window.

For example, a thermocouple amplifier has an input range from -30 to 370 °C. But your measurement requires just a range from 20 °C to 40 °C and then you can enter 20 °C for the Min value and 40 °C for the Max value. From now on, the recorder displays the 20 °C to 40 °C range as a standard without zooming in.

To set the values back to Automatic range detection, just right-click in the desired field and select Set to Automatic from the list:



NOTE: **This function is used only to define the standard range values for the display; it has no influence on the measured input range itself and can also be set to any other value in the displays.**

Format

Chose from different types of formats for result display. You can choose from Fixed, Scientific, and Hexadecimal.

Sample rate

The Sample rate can be used to 'reduce' the sample rate for each channel. The data is acquired with full set bandwidth but is reduced in the device. Dewesoft software is only computing and saving at the reduced (selected) sample rate.

Switch to

On the top right part of the screen are two buttons to switch between input channels without leaving this Channel setup window.

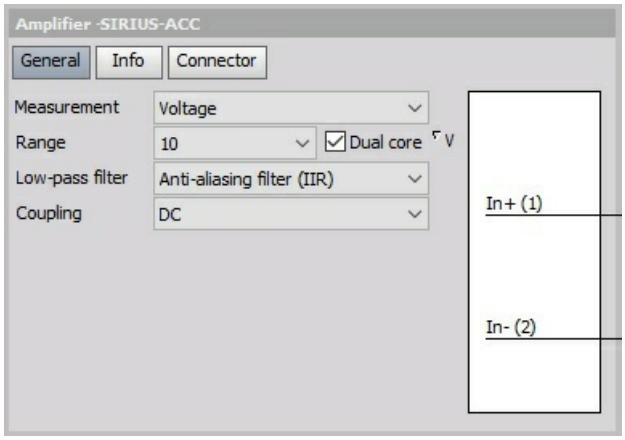
AMPLIFIER SETTINGS

On the left side of Dewesoft input screen Channel setup for channel n, we have Amplifier settings.

In the caption of Amplifier settings the Module type, Serial number and Revision number are displayed as the caption of this section.

For different modules, various settings are displayed on this section of Channel setup for channel n screen and you can control these modules settings directly from this screen.

Example: Amplifier settings part of Channel setup window, when DAQP-BRIDGE-A module is installed in this slot:



At Dewesoft we produce many series and special modules, e.g.:

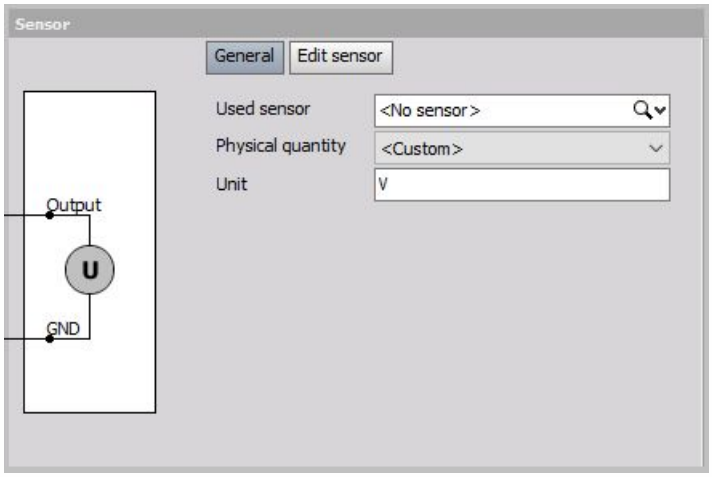
- SIRIUS® (ACC, ACC+, CHG, CHG+, HV, LV, LV+, MULTI, STG, STG+, STGM, STGM+, R2DB, R2D, R3, R8, R8D, R8DB, Mini, HD-LV, HD-STG, HS-ACC, HS-ACC+, HS-CHG, HS-CHG+, HS-HV, HS-LV, HS-LV+, MULTI, HS-STG, HS-STG+)
- DEWE-43)8 Analog inputs, 8 Super counters, 2 CAN bus port)
- DS-NET (DS-GATE, DS-NET CPU, ACC2, CFB2, BR4, V8, V8-200, V4, V4-HV, TH4, TH8, DIO8)
- KRYPTON (8xTH, 16xTH 4xLV, 8xLV, 3xSTG, 6xSTG, 8xRTD, 16xDI, 16xDO, 8xDI, 8xDO, KRYPTON CPU)
- CAN interfaces (DS-CAN2), SIRIUSim 4xCAN, SIRIUSif, 8xCAN

NOTE: **For details about Dewesoft Modules and all module configurations see -> Dewesoft online web page.**
NOTE: **For additional help with Module settings visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training.**

SENSOR SETTINGS

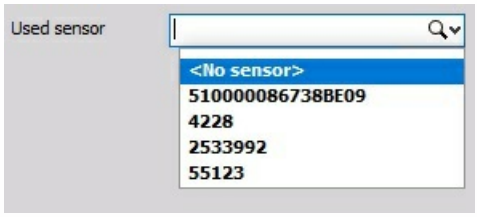
In this Sensor part of Dewesoft Channel settings screen, you can select from the General and Edit sensor tabs.

Sensor editor of Channel setup:

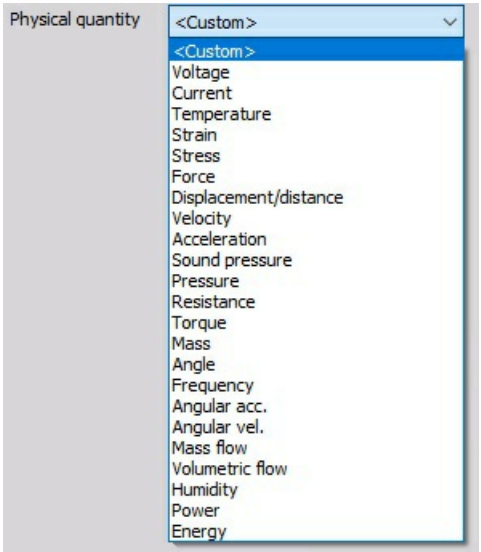


In General sensor tab we can choose from:

Used sensor drop-down menu enables us to select a previously used or created sensor.



In Physical quantity menu, we select the appropriate quantity of measurement.

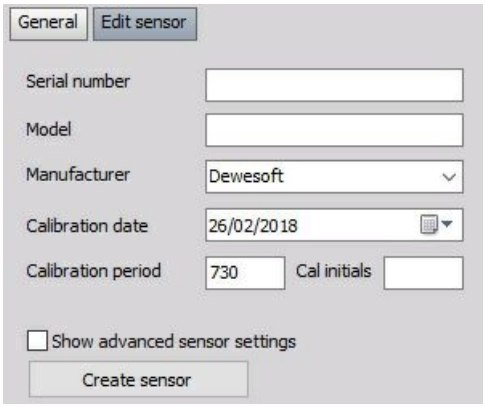


The unit drop-down menu will display the appropriate options depending on selected Physic quantity.

Units can be input by the user if **Custom** physical quantity is selected.

In the Edit sensor tab, we have options to define a new sensor

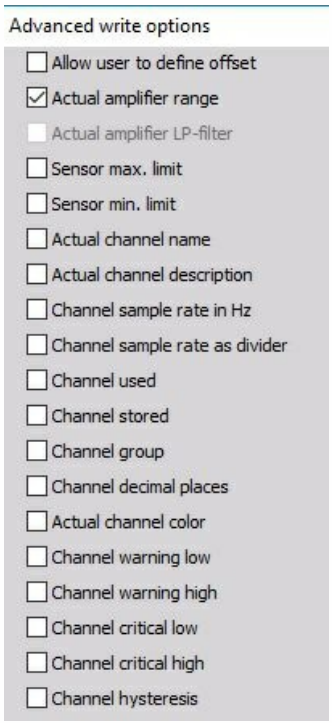
Enter the serial number of new sensor



Enter the model of the sensor and the manufacturer.

Define the calibration date, period and initials. If calibration is overdue, the date will be shown in red.

We can also Show advanced sensor settings:



Sensor database

Dewesoft offers an efficient, helpful sensor database. In addition to the linear scaling, which can be done also in the input Channel setup, the sensor database offers beside other information about sensor also scaling by table or polynomial, even transfer curves can be defined.

Dewesoft has some pre-defined sensors in his sensor database. But the pre-defined demo sensors are for sure not what you would need for your real application.

How to define your own sensors? To do that select in Options menu Editors / Analog sensors and the sensor editor will open.

SCALING

The Scaling procedures define the ratio between the electrical input value of the selected input channel (from the amplifier) and the scaled physical-engineering input value, which is used in Dewesoft procedures.

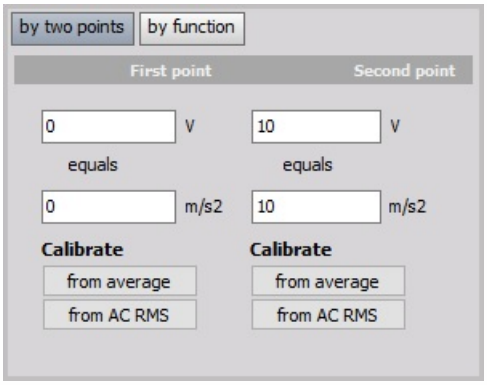
The scaled "physical-engineering" input value is needed for signal display or to have different units as the "electrical" input value.

The bottom-right section of the Channel setup for channel n screen is where you can perform:

- Scaling - calibrating on input: either on a 2-point or functional basis
- manual/automated calibration

The bottom-right section of the dialog contains a dynamic representation of your signal - the left side is the 'electrical' units, and the right side is the scaled 'physical-engineering' units, so you can directly see the effect of your calibration values.

Calibrating on input 'by two points'



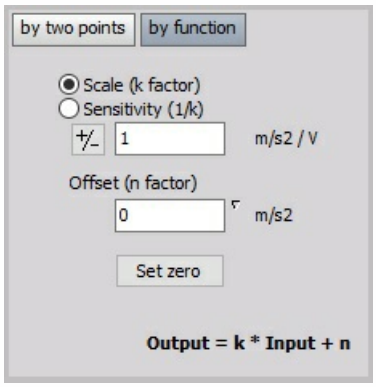
In this example we enter scaling factor so that the input channel on First point: 0 real volts equals 0 m/s2, and on Second point: 10 V (electrical units) equals 10 m/s2 (engineering units - Acceleration).

Please note that when changing one point, the scaling and offset changes as well. Therefore it is not good in the example above to 'calibrate from average' of the first point, because the scaling factor will change as well. Please look at the following section - Scaling by function and perform a sensor zero.

For additional help with scaling factor visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training.

Calibrating on input 'by function'

Another way to perform such a simple linear function would be to simply multiply the input by a factor. You can easily do this by clicking the by function button, which will change the two-point calibration portion of the dialog to look like this:



In the above example, we calibrate the system so that for channel this factor is 1.

Above shows your basic algebraic formula:

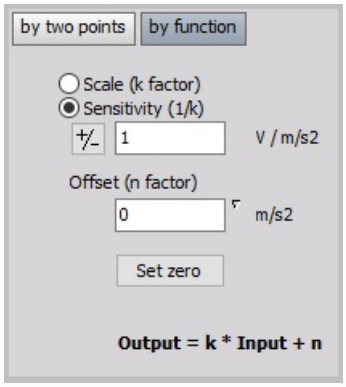
$$y = kx + n$$

- y - physical value
- k - scale
- x - measured value
- n - offset

for a general linear function, which makes it easy to input k factor and Offset (n-factor).

There are two ways of entering the value:

- Scale (k factor); example above: k = 1 m/s2 / V (Scale factor is the factor which defines how to scale the electrical signal to deduce the physical values).
- Sensitivity; example on below: k = 1 V / m/s2 (Sensitivity is the factor which is usually used in sensor calibration sheets and defines the electrical output per physical quantity measured by the sensor.



Set Zero button is used in the main setup to conveniently offset the signal to 'force' it to zero,

when it is not exactly at that value (but should be), due only to small sensor offsets and perhaps long cable lengths causing capacitive coupling and/or line loss.

Example:

An even easier way to convert from °C (Celsius) to °F (Fahrenheit) is to click by function, then enter the $y = kx + n$, to perform simple offset.

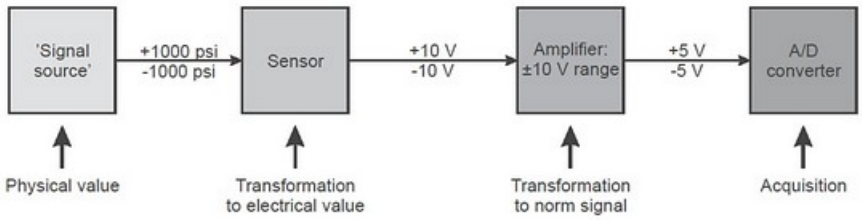
By multiplying °C by 1.8 as Scale and then adding 32 as an Offset, the same result is achieved.

Since modules that measure temperature output their values in the scientific standard Celsius scale, this is a handy formula to know when the temperature is desired in Fahrenheit.

For useful hints about scale (k factor) and Sensitivity by different Measurement applications see ->Dewesoft Web page -> PRO Training.

Calibrating on input example

Pressure up to ±1000 psi (= signal source) will be measured with a 100 psi / V sensor, connected to a high voltage module (= amplifier with 10V range).



In the example screen, you can see that the default values are entered - the real and engineering units are the same. But what if this was a pressure sensor that had a voltage output and each volt represented 100 psi of pressure?

Scaling 'by two point'

Here is how we would set it up - starting at the top; we enter our text values for this channel. Then in the bottom-left, we tell the system that 0 real Volt equals 0 psi, and 1 real Volt equals 100 psi. If you look at the graph on the bottom-right corner of the dialog, you can see that the real input signal has a min of 0.01466 V at this moment, which is being scaled to 1.466 psi, which is exactly what you would expect.

Scaling 'by function'

Another way to perform such a simple linear function would be to simply multiply the input by a factor of 100. You can easily do this by clicking the by function button, which will change the two-point calibration portion of the dialog.

There are two ways of entering the value (just a question of the value):

- Scale (k factor); example above: k = 100 psi / V
- Sensitivity; same example: k = 0.01 V / psi

Both methods can cause a small offset. Use the Set Zero button in the main setup to conveniently offset the signal to force it to zero.

CALIBRATE

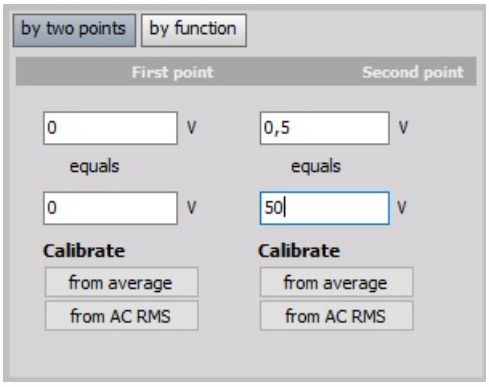
The ability to quickly calibrate directly to DC and AC signal sources is a great advantage of Dewesoft, and will save you much time, and ensure better and more reliable data recording long into the future.

NOTE: **This procedure is only available with ‘by two point’ scaling!**

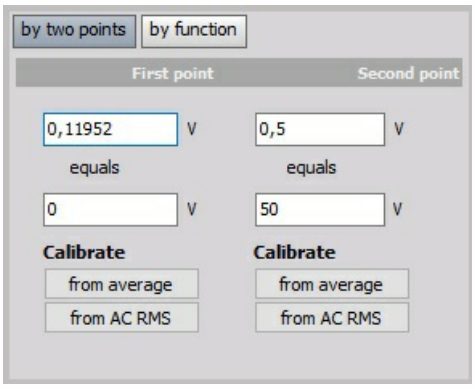
Calibrating to signal sources

In 2-point calibration, there are two other sets of buttons that will allow you to perform a more accurate calibration when you have access to a known signal source or load. Let's say that we have a load sensor that we want to calibrate to a known load. We want to enter two points - say zero (0 kg) as our first point, and twenty-five (50 kg) as our known second point.

The first thing to do is simply enter those two values in these boxes as shown:

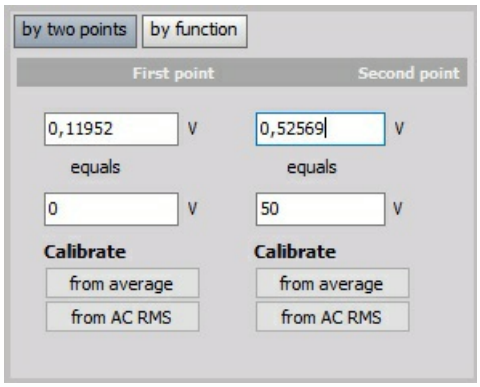


Now, we need to apply the known zero (0 kg) load. When it stabilizes, press the left-most from the average button, and the software will sample the input for a short time, average the values that it receives, and then calculate the required number for the upper left box - First point.

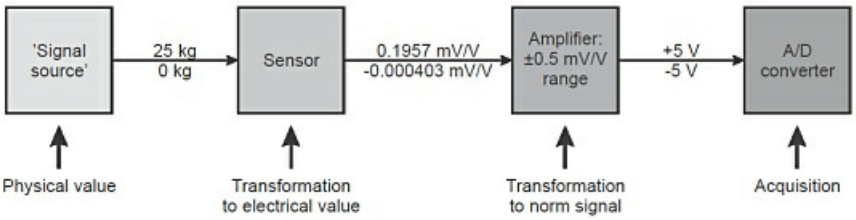


Now apply the known 50 kg load, let it stabilize, and press the right-most from the average button (see picture on right). Dewesoft is now perfectly calibrated to your signal based on the known loads that you input. Note in the screen picture above that Dewesoft has automatically input the two values in the upper two boxes that it needed in order to achieve this calibration.

If you have an AC signal source, you should not use the from average buttons - use the from RMS buttons instead! Also, press the red-labeled RMS button below the graph to add the RMS values to the dynamic display.



Background to this example: Defined weight from 0 to 50 kg (= signal source) will be measured with an uncalibrated strain gage sensor (= sensor) connected to a DAQP-BRIDGE module (= amplifier with 0.5 mV/V range).



For additional help with Calibration visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training.

LIVE DATA PREVIEW

The bottom-left portion of the dialog contains a live preview of your signal - the left side are the electrical units, and the right side are the scaled engineering units, so you can directly see the effect of your scaled (and calibration) values:

Depending on the **pressed button(s)** in this area, different **Input and Scaled values** are represented on the dynamic graph:

- on the left side the "electrical" units
- on the right side is the scaled "engineering" units

That way you can directly observe the effect of your calibration values.

You can also change **Block time** (defines time for calculation) for signal display.

If you have an AC signal source, you should press the red-labeled AC RMS button to add the RMS values to the dynamic display.

Calculation time (Block time)

On this section on the graph, you can change calculation time (Block time) and signal display frequencies of the selected values. Predefined choices are:

- 0.1 second
- 1 second.

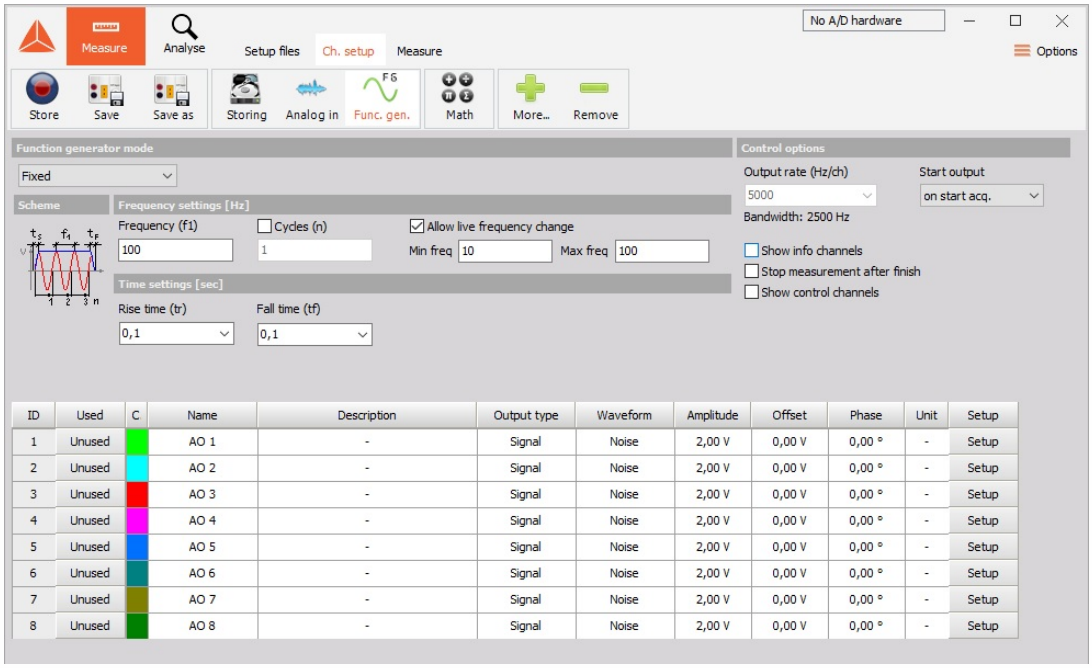
Dynamic representation of your signal

- The graph shows the current input signal. In addition, the min/max, AC RMS and average values are displayed. If the input signal is higher than the selected input range, you will see a message OVL in this screen.
- The values at the left side of the bar graph show the "electrical input" value, representing the input range of the amplifier. The right side shows the "physical input" value of the scaling, so you can directly see the effect of your calibration values. In this example, they have to be the same.
- Use the Average, AC RMS and Min/Max button to show or hide the values.
- Change the calculation time (Block size) between 0.1 and 1 sec to achieve the best view for your signal.

FUNCTION GENERATOR

Dewesoft offers a full-featured signal-Function generator with 16 or more analog signals. In the Dewesoft we can define and set all analog output channels which will be used during measure using installed measurement hardware.

After adding a new module the Function generator tab on Dewesoft Ch.setup screen must be selected. When we open a new Function generator module a setup screen will appear.

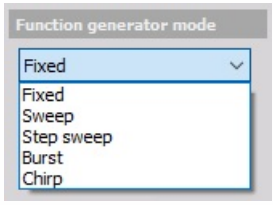


Example of use: If the shaker is used for excitation of structures, single or multiple shakers are driven by either analog output or by external shaker system. The force is induced in the structure and the response of the structure is measured by the accelerometers or another vibration transducers.

OPERATION MODES

In Func. gen. tab of Dewesoft Setup screen several operation mode settings can be chosen and adjustments to them done:

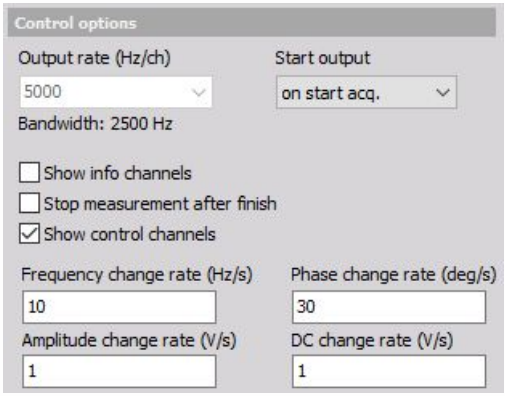
- Function generator mode



- Frequency setting [Hz]

- Time setting [sec]

- Control Options



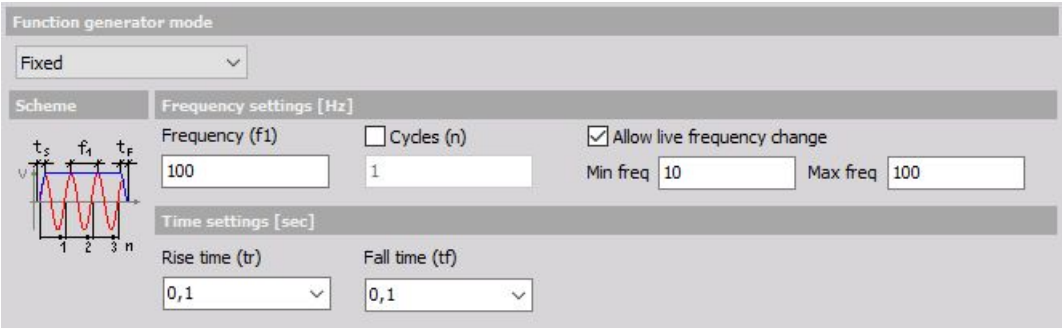
The function generator offers five different operation modes, which are set for all analog output channels:

- Fixed - signals with a constant frequency
- Sweep - sweep frequency from Start frequency to End frequency
- Step sweep - sweep frequency with certain fixed frequencies
- Burst - noise output
- Chirp - sweep frequency with is shorter time and repeated after a defined time

The amplitude will not rise from zero to full amplitude immediately but within a definable time. This ensures that there are no jumps or glitches on the generated signal, which is important to avoid impact shocks to the system under test.

Fixed

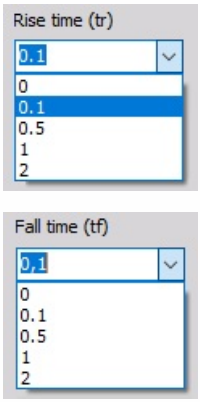
Fixed mode outputs the selected signals with a constant Frequency (f1). The output frequency is the same for all the channels and is defined in the setup.



It is also possible to define a number of output cycles if the Cycles (n) check-box is enabled, otherwise, the output is continuous.

The output frequency is the same for all selected channels. If Allow live frequency change is enabled, it is possible to vary the signal frequency during the output within the selected frequency range (Min frequency and Max frequency).

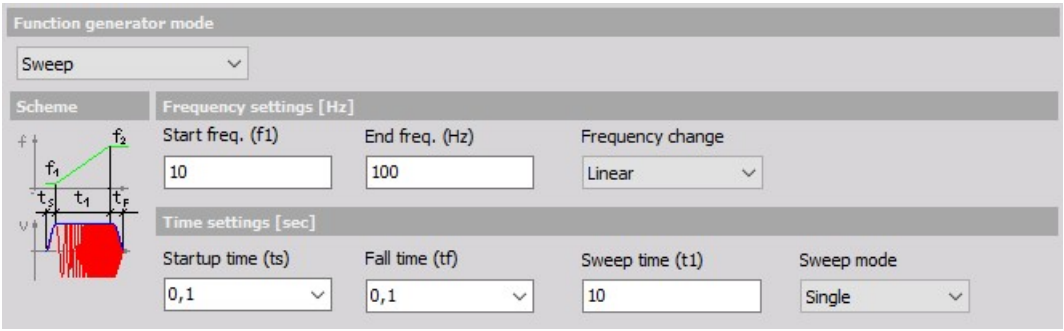
To avoid jumps in the signal on Rise time (tr) and Fall time (tf) at the beginning and the end, the signal output can be set with the selection from drop-down list.



APPLICATION: **Fixed mode is used if a defined waveform should be output without changing the frequency or amplitude automatically. Such a function generator is valuable in laboratory testing of electronic circuits, power amplifiers, filters and so on. Also, it provides a great base for calibration procedures. Multiple channels ensure a great chance to test delays and phase errors.**

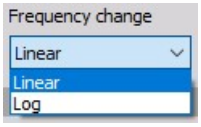
Sweep

A sweep is defined by a Start freq.(f1), an End freq. (Hz) - f2 and a Sweep time (t1).

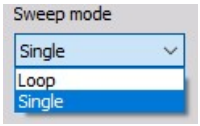


Startup time (ts) and Fall time (tf) define the ramp slope at the beginning and the end of the sweep.

Additionally, the Frequency change of the sweep can be either linear or logarithmic to provide more cycles at lower frequencies and can be selected from the drop-down list.



Two Sweep modes are selectable from the drop-down list, Single and Loop. In Single mode, you will get a single sweep output only, at loop mode the sweep is output continuously.

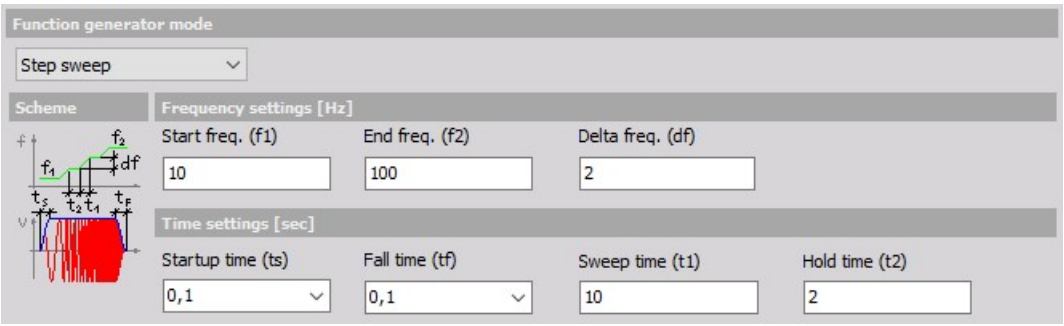


APPLICATION: **Sweeps are useful for testing electronic circuits like filters as well as in automated modal analysis.**

Step sweep

The step sweep mode is a special procedure for modal testing, where the signal remains at certain fixed frequencies to allow the structure to settle to that frequency. In addition to this more exact frequency resolution of the transfer function can be achieved with smaller steps. Even though this procedure is the slowest to finish the test, it provides the most accurate results.

In the middle, the frequency steps can be one channel defined as the trigger channel and this one can trigger the acquisition system to acquire the results of the modal test.



Start freq. (f1) and End freq. (f2) have a similar function to sweep mode, Delta frequency (df) defines the width of frequency change during Sweep time.

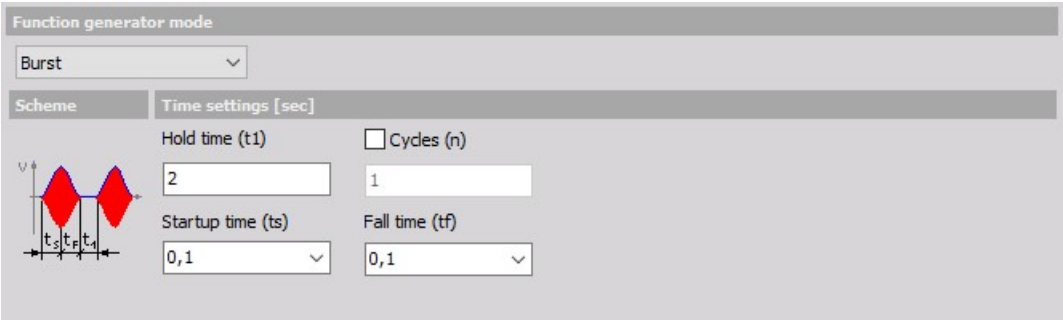
Sweep time (t1) defines the sweep duration to the next frequency step, Hold time (t2) sets the duration where the frequency keeps fixed.

Startup time (ts) and Fall time (tf) define the ramp slope at the beginning and the end of the function.

APPLICATION: **This mode is mainly used for modal structural tests.**

Burst

In burst mode Startup time (ts), Fall time (tf) and Hold time (t1) can be defined. Within the burst, we can choose to have any signal, but in practice, noise is usually used as the base generated signal.



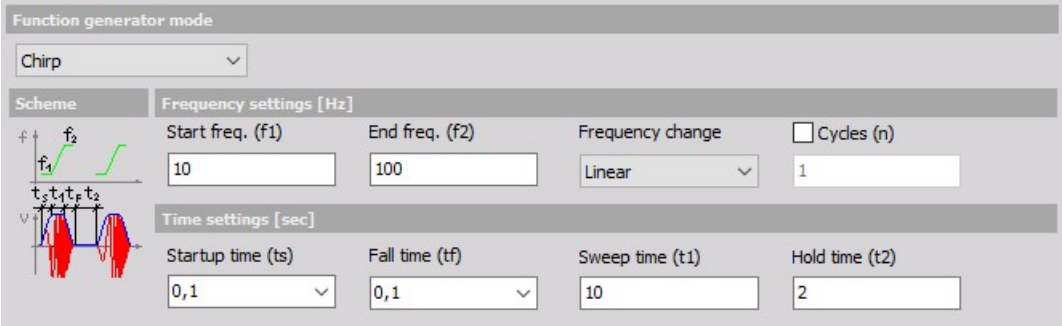
Within the startup and fall time, you get noise output.

If the Cycles (n) check-box is enabled, it is possible to set the number of burst to output.

APPLICATION: **Bursts are usually enhancements of the noise excitation for measuring transfer functions. With the burst mode, the need of using windows for frequency analysis (and therefore errors caused by windowing) is eliminated. They are useful at testing response characteristics of the systems.**

Chirp

The chirp mode generates a signal which changes its frequency within a certain time. It is similar to the sweep mode, but in chirp mode, the sweep time is shorter and the function is repeated after a defined time.



Start freq. (f1) and End freq. (f2) define the frequency range of the chirp. The Frequency change can be either linear or logarithmic. Additionally, it is possible to define the number of chirp Cycles (n) to output. The time settings also contain Startup time (ts) and Fall time (tf) which defines the ramp at the beginning and the end of the chirp, Sweep time (t1) - the chirp duration - and the Hold time (t2) between two chirps.

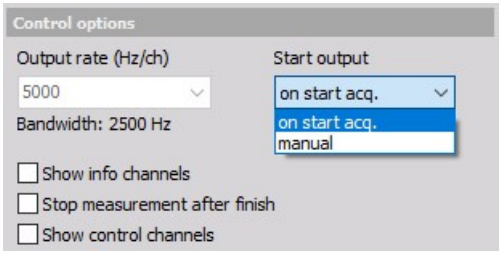
Chirp mode is similar to burst, except the short sine sweep instead of the noise. Chirp better defines the frequency range than the burst.

APPLICATION: **The chirp signal is used for spectral analysis of nonlinear systems.**

CONTROL OPTIONS

The condition for the analog output is set with the Output rate (Hz/ch) and Start output control on Control Options section.

The Start output condition can be set either to:



- on start acq. which means that the output starts with the data acquisition
- manual - the signal output start can be started manually during the data acquisition.

The Output rate of the function generator is not related to the sampling rate and should be more than 10 times higher than the highest output frequency to avoid harmonic distortion.

If Show info channels are enabled the values of the parameter channels (amplitude, phase angle, offset and frequency) are additionally available as dedicated channels in the instrument views and mathematics.

If Stop measurement after finish is enabled the data acquisition will be stopped after the signal output has been finished. This requires a defined Cycles (n)umber for output in the Operation mode settings.

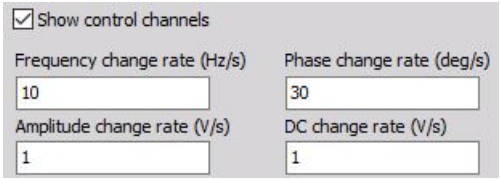
The start and stop of generation can be either on start acq.(acquisition) or manual. Each operation mode carries different manual control:

- at the fixed frequency the generation can be started and stopped
- at burst mode, the individual bursts are issued with manual control
- at stepped sweep, we can go manually to the next frequency

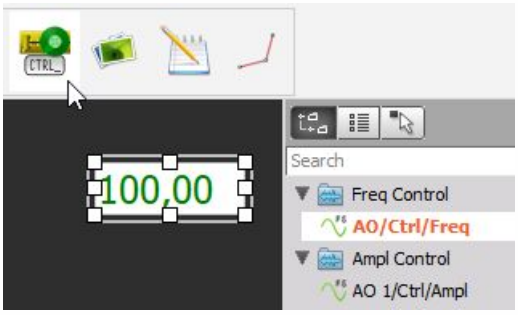
The start and stop of generation can be with a half sine window, so the amplitude will not rise from 0 to full amplitude immediately, but within a certain time which ensures that the systems under test don’t suffer from impact shock.

Show control channels option allows the user to change the output parameters via control channels. Measurement screens could have sliders and knobs to control function generator.

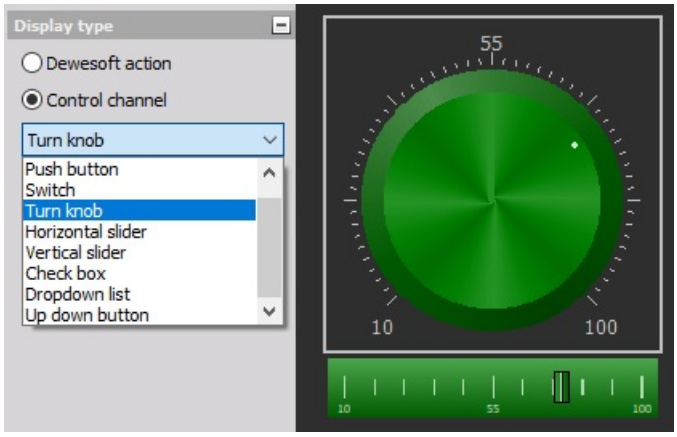
If this option is chosen, then the rates of change must be defined first in the channel setup screen to define maximum change when the slider is changed.



In the measurement screen, we can add control channels. First, we choose the control channel icon next, we assign what we want to control. Depending on the operation mode of the function generator, we can control frequency for all output channels and amplitude, phase and offset for each channel.



Now we can change the type of the control channel from text entry to horizontal, vertical slider or analog gage and create a perfect control layout for function generator.



CHANNEL VALUE

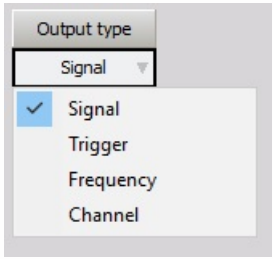
In Output channel setup you can set all analog output channels - signals individually for each channel:

ID	Used	C	Name	Description	Output type	Waveform	Amplitude	Offset	Phase	Unit	Setup
1	Used		AO 1	-	Signal	Sine	2,00 V	0,00 V	0,00 °	-	Setup
2	Unused		AO 2	-	Signal	Triangular	2,00 V	0,00 V	0,00 °	-	Setup
3	Unused		AO 3	-	Signal	Rectangular	2,00 V	0,00 V	0,00 °	-	Setup
4	Unused		AO 4	-	Signal	Saw	2,00 V	0,00 V	0,00 °	-	Setup
5	Unused		AO 5	-	Signal	Noise	2,00 V	0,00 V	0,00 °	-	Setup
6	Unused		AO 6	-	Signal	Arbitrary	2,00 V	0,00 V	0,00 °	-	Setup
7	Unused		AO 7	-	Trigger					-	Setup
8	Unused		AO 8	-	Channel					-	Setup

- **ID** - Identification number of output
- **Used / Unused** - Select Used /Unused to display or hide channel in measurement.
- **C** - Color of the channel
- **Name** - Name of the channel (displayed in Measure mode)
- **Description** - Description of the channel (personal description for own reference)

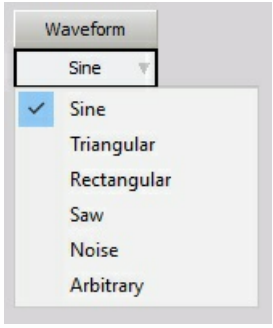
- **Output type** - Output type can be set individually for each channel to:

1. Signal
2. Trigger
3. Frequency
4. Channel

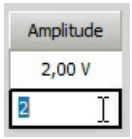


- **Waveform** - The Waveform type can be set individually for each channel:

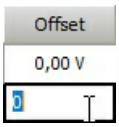
1. Sine
2. Triangular
3. Rectangular
4. Saw
5. Noise
6. Arbitrary



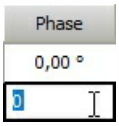
- **Amplitude** - Input desired amplitude of the signal (peak)



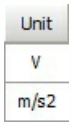
- **Offset** - Change offset of the signal



- **Phase** - Change phase offset



- **Unit** - Input desired unit, to be displayed in Measurement

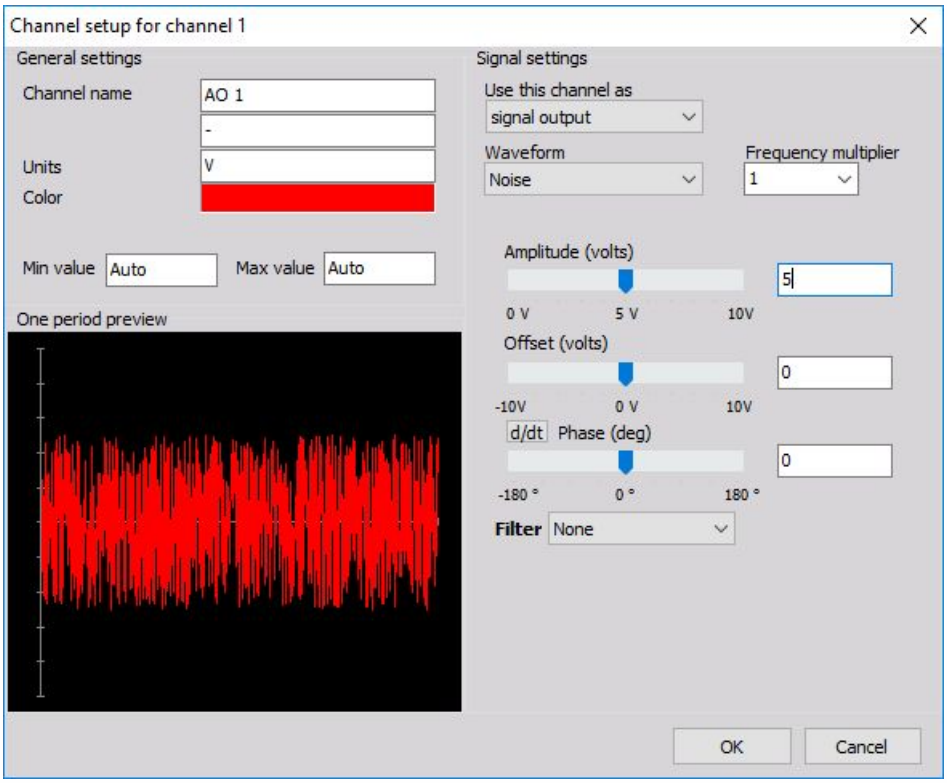


- **Setup** - Amplitude, offset and phase can be set with the slider or enter the desired value directly.

CHANNEL SETUP

Press the Setup button in Analog out Channel list SETUP column to enter the Channel setup screen for each channel.

- **General settings** - you can enter the base channel information on the General settings part of the screen: Channel name, the measurement value, units of measurement, define Color (click on the color bar to change the color for this input) and display range - Min value, Max value, to set a minimum and maximum display value; on the left lower side One period preview of signal is displayed
- **Signal settings** - on the upper right area of the screen to define signal usage and Waveform type, on the lower right area also sliders to change Amplitude, Offset and Phase and additional signal setting

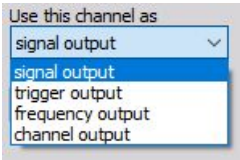


Signal settings

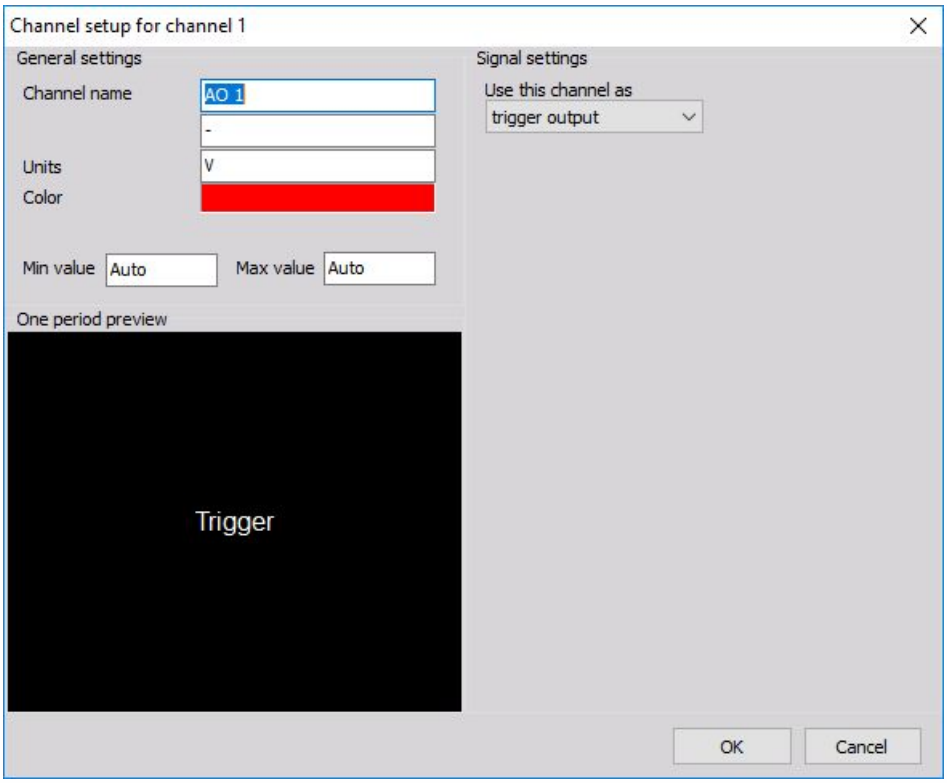
The upper right area contains two elements of Signal setting (signal usage selection in Use this channel output as).

The channels can be used as:

- Signal output for the selected waveform is the default.
- Trigger output each channel can be set to output a trigger (0 - 5V) instead of signal.
- Frequency output is a voltage representation (-5V to +5V) of minimum and maximum set frequency. This function is mostly used with sweep, step sweep and chirp.
- Channel output output any channel or math function.

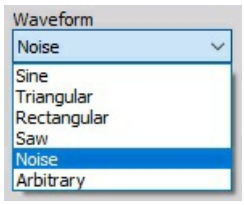


Often it is intended to trigger the acquisition on the measurement side when a certain criterion is met. When the sweep is active, the trigger value goes up. It is up at step sweep, at the fixed frequency and during the bursts.



Waveform type

The channels can be used as signal output for the selected Waveform. Waveform drop-down selection has a direct influence on the settings displayed in the lower right area.

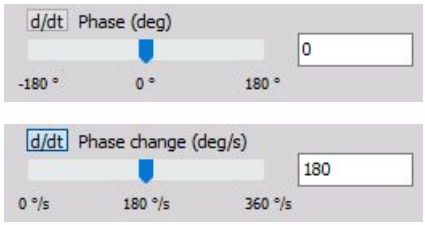


Amplitude, offset and phase angle

All generated channels themselves are available as Dewesoft virtual channels. Also amplitudes, offsets and phase angles are available as channels so they can be put in any visual control to monitor the frequency generator or they can be used in math channel for simulation or control purposes.

The right area bellow Waveform contains the sliders to change Amplitude, Offset and Phase. For input of precise values, an input field is next to the slider. Simply click numeric field -> and enter the desired value with your keyboard.

The phase setting has an additional option so you can not only define fixed phase but variable phase as well. This can be achieved by pressing d/dt button beside Phase caption (after pressing the button caption changes to Phase change (deg/s).



For example, variable phase can create stroboscope effect. One sine wave with fixed phase can drive a shaker, while the other is used for TTL rectangular signal with variable phase for camera triggering to create a stroboscopic video.

The lower left area shows a nice One period preview of the selected signal.

Additional signal setting

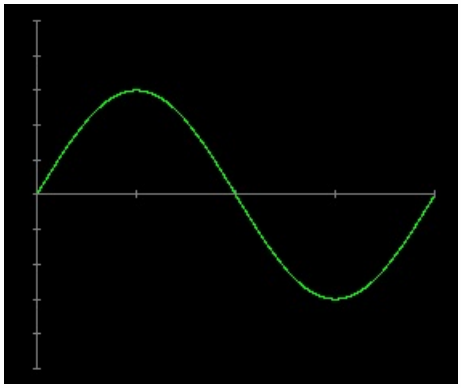
For Arbitrary and Noise type of waveform, beside sliders and numeric fields there is also additional signal setting:

- For Arbitrary type of waveform there are sliders and numeric fields in Signal tab, in addition to Points tab for this signal setting -> see -> Arbitrary - Additional settings
- For Noise type of waveform there is also an additional Filter signal setting.

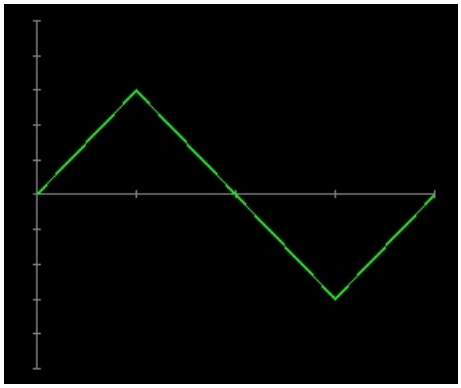
NOTE: **For additional help with Noise see -> Noise - Additional settings.**

WAVEFORM TYPE

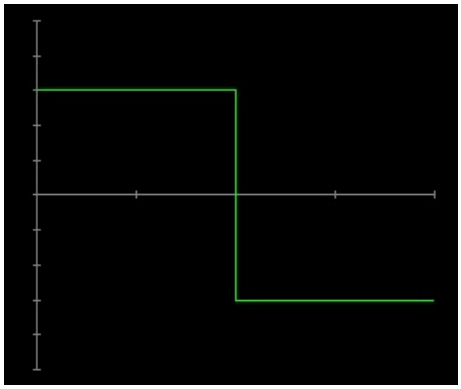
In Dewesoft software the waveform types are:



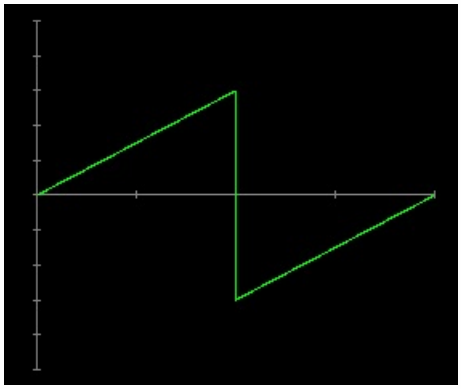
SINEWAVE



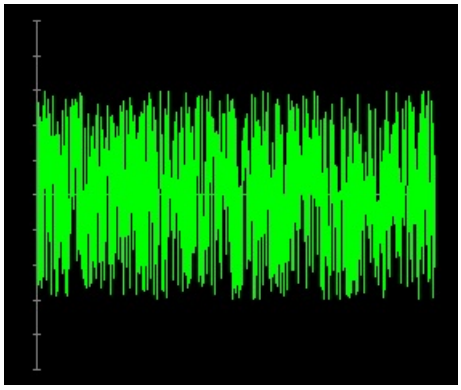
TRIANGLE



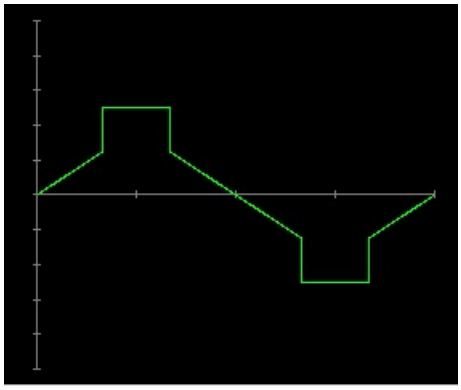
RECTANGLE



SAW



NOISE



ARBITRARY

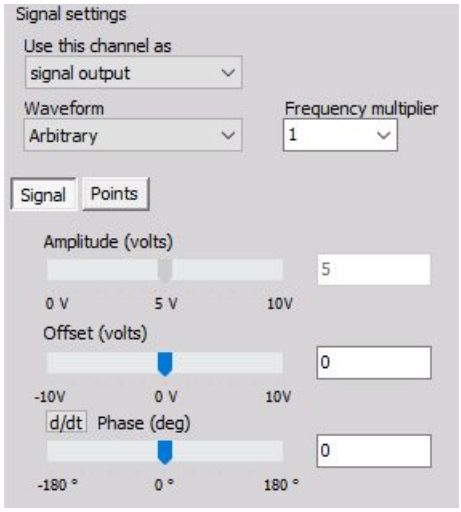
Waveform types can be set in:

- Waveform of Value column in Analog out tab,
- Signal setting of analog output Dewesoft Setup screen in Analog out tab

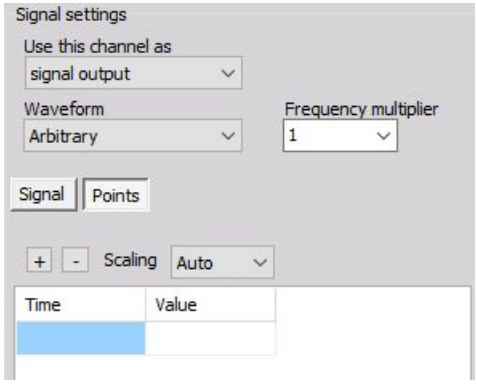
ARBITRARY - ADDITIONAL SETTINGS

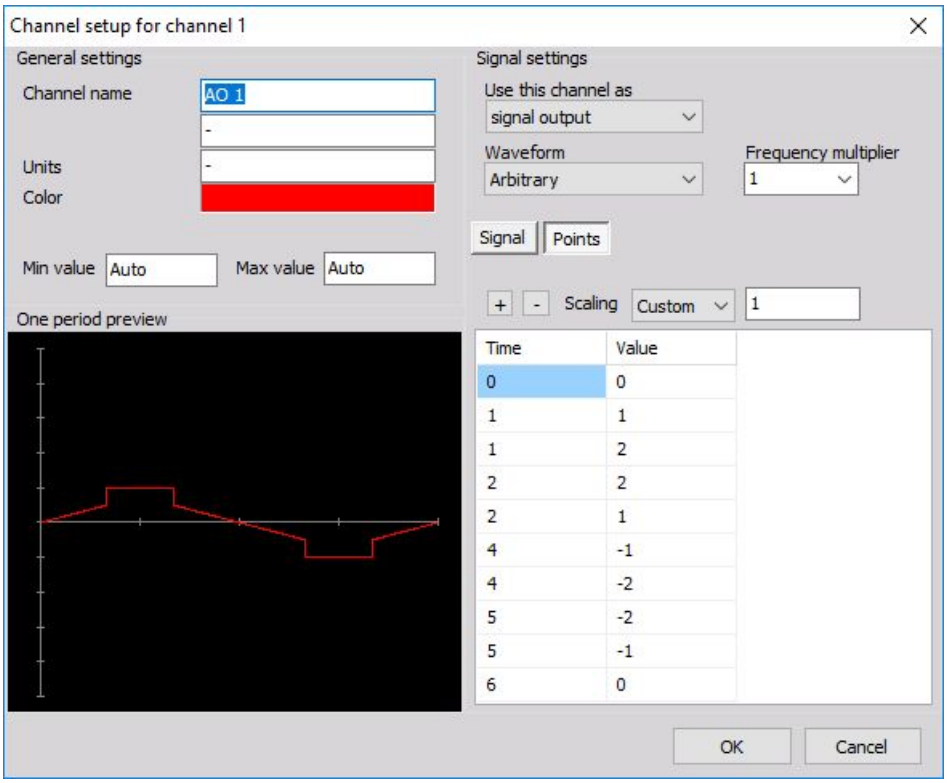
The arbitrary waveform makes the definition of special waveforms very easy through two tabs:

- Signal tab (parameters, like amplitude, offset and phase can be set up)



- Points tab (the waveform itself is defined in a table)





There are three ways to get a signal:

- The signal values can be entered directly in the table.

Press + or - buttons to add or remove rows in the table and enter values for Time and Value (amplitude)

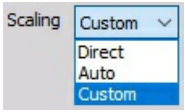
- Paste from clipboard

For example, you can copy from Excel to Dewesoft. If you copy from another program like Word or other Editors be aware to use the tabulator as the separator between the columns.

- Load from file - the data has to be provided as a standard text file (*.txt) with tabulator separated columns for Time and Value amplitude values. There is no header required.

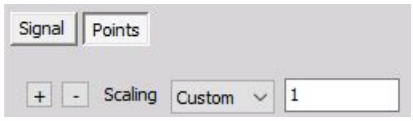
To paste values from clipboard or load values from a file select the Points tab and right-click. This function can also be used to Copy to clipboard or Save to file.

Amplitude Scaling of the waveform can be done in three ways:



- Direct - inserted values are taken directly and can never be changed
- Auto - inserted values are scaled with the amplitude value, like the other waveforms (even online)
- Custom - inserted values are scaled by a scaling factor. This factor is not changeable during the signal output!

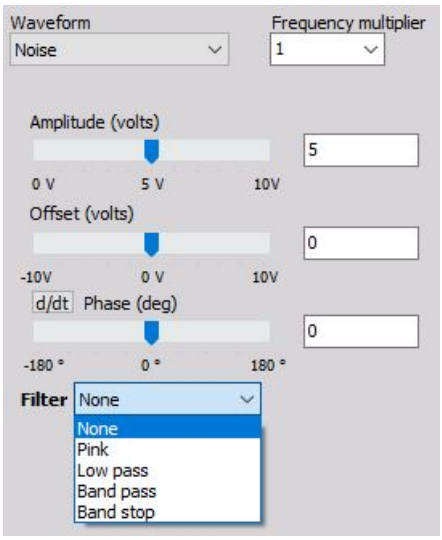
When this option is selected additional field appear to enter scaling factor:



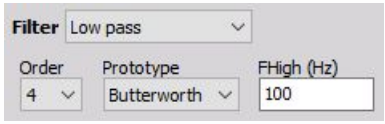
On One period preview, you can online see entered point - signal preview.

NOISE - ADDITIONAL SETTINGS

In addition to the base channel and signal settings, the Noise waveform type offers a Filter selection. Select between the filter types:

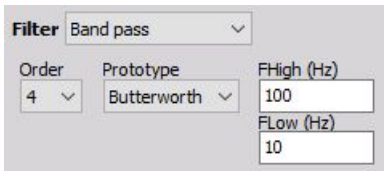


- None - no filter is used, the output signal is white noise
- Pink - a filter is used to obtain pink noise
- Low pass



Define a low pass filter:

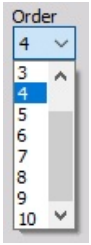
- Order (1 to 10)
- Prototype (Butterworth, Chebysev or Bessel)
- FHigh (Hz) - high frequency can be set
- Band pass

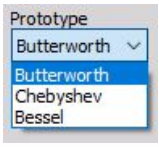


Define a band pass filter:

- Order (1 to 10)
- Prototype (Butterworth, Chebysev or Bessel)
- High- FHigh(Hz) and low frequency - Flow(Hz) can be set

Order and Prototype can be selected from the drop-down list:

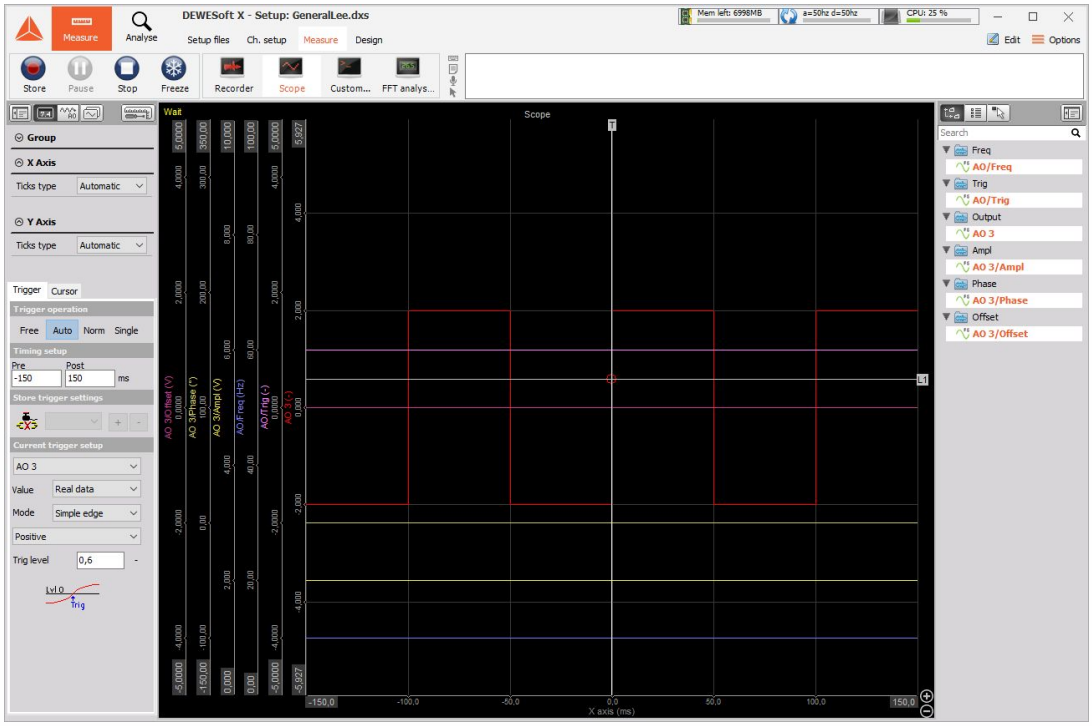




NOTE: **For information about Order and Prototype see ->IIR Standard filter.**
For noise excitation, it is common to define the upper and lower frequency limit. The lower limit prevents DC excitation and upper limit prevents too high-frequency noise which is higher than the bandwidth of the measurement.

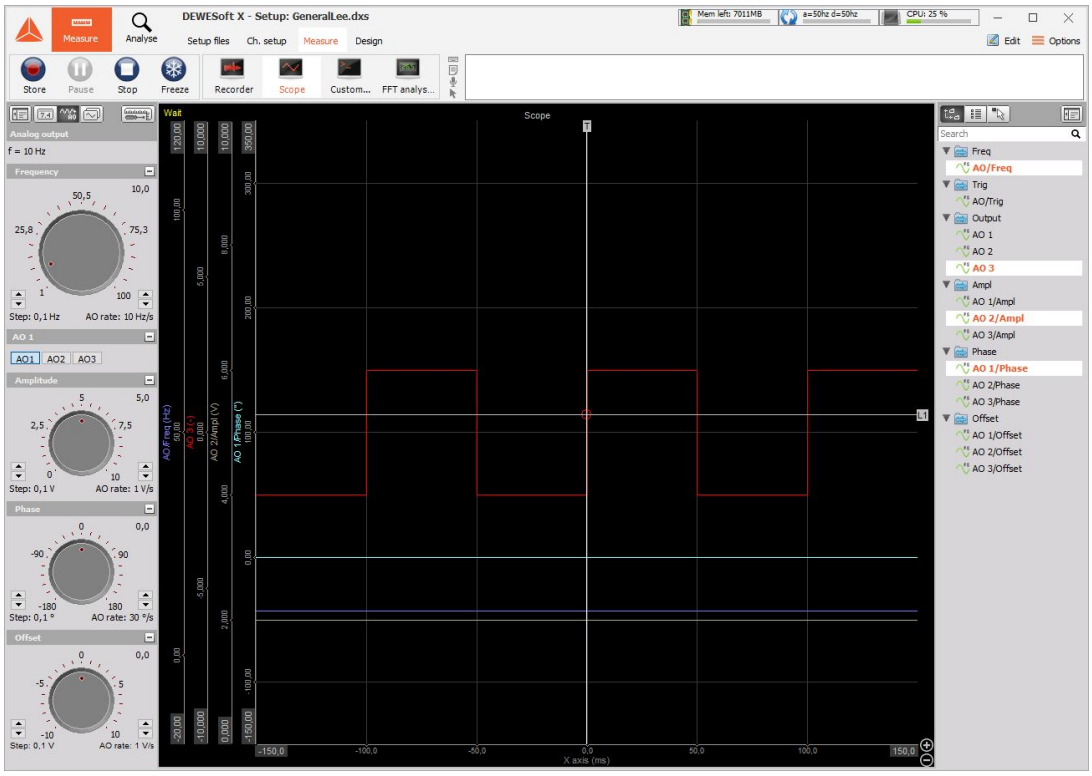
SETTING DURING MEASUREMENT

All analog output channels (including their parameters) are available in Dewesoft as internal channels (if Show info channels are selected in Control options of Function generator). Therefore you can display them together with your other signal sources.



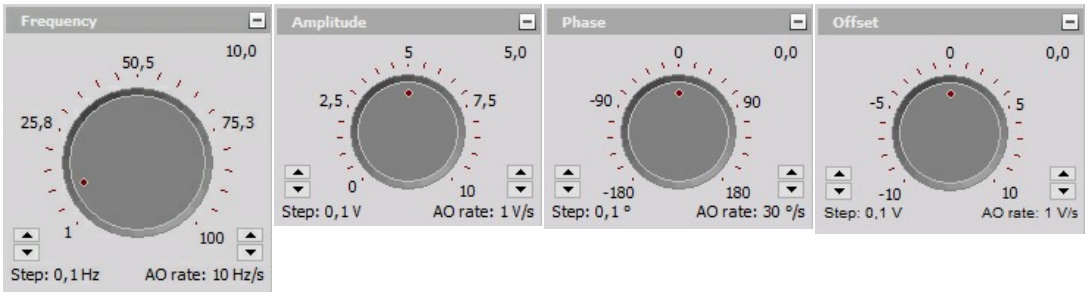
If analog outputs are activated in Dewesoft, you can see a channel list on the right side. The list contains all the channels that are currently in use and can be displayed or hidden simply by clicking on them.

When you press the button, the instrument will show the following controls:



Use the adjusting knobs to change the parameters Frequency, Amplitude, Phase and DC offset:

Frequency	Amplitude	Phase	DC offset



On the left corner of each element, the resolution can be set (Step). On the right corner, the AO rate can be chosen. High rate means fast changes to new values, with low alteration rate changes require more time. If we want to enter the value manually, we can click on the value of the upper right part of the display.

Each control except for Frequency have also - button in caption rows to minimize controls knob. With press on + button controls knob is displayed again - maximized.

Example: all controls knobs are minimized:

NOTE: **The frequency adjusting knob is only available when 'Allow live frequency change' has been enabled in the Analog out setup.**

Channel selector

The channel selector defines the active channel for the amplitude, phase and DC offset parameter settings:

As already described, all parameters are available also through control channels.

CAN BUS

Dewesoft CAN Acquisition module is able to listen to the traffic on CAN bus interfaces from different manufacturers and acquire CAN messages, which is CAN Data Acquisition basic unit and can contain several signals -channels. Depending on the used CAN system, different car data is available like temperature, revolution, acceleration, and many more.

- **Required hardware** - Dewesoft USB
- **Setup sample rate** - At least 1 kHz

CAN messages/channels must be defined and set up with:

- setting common CAN settings/options
- create new:
- Import from existing .dbc file
- Add manually with Can channel setup procedure for Message/Signal setup

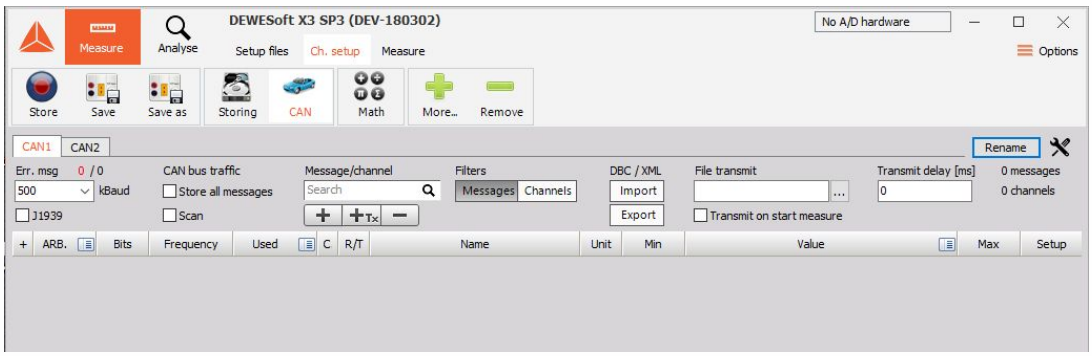
- Use scan mode to find messages automatically
- change existing:
- Edit

- Remove

For additional help with CAN bus Acquisition visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training.

When you select a CAN tab on Dewesoft Setup screen, on the upper part of screen CAN settings will appear:

- CAN settings/options with information about CAN message / channel settings; to import data from DBC library
- Message/channel list with information about CAN messages / channels and primary SETUP column with Setup button to call CAN channel setup window



As a standard, there are no messages and channels available (report about the number of messages and channels in the list is displayed on the right part of this area). There are three ways to create them:

- Import messages / channels from existing .dbc file
- Add messages / channels manually

- Use scan mode to find messages / channels automatically

All this work can be done with settings and options on this part of CAN Setup screen.

CAN settings / options

CAN port number

Before you add messages or channels, you have to select the CAN port. Up to 4 CAN ports are supported by Dewesoft.

To select the port simply click on appropriate CAN button, select from CAN0 to CAN3 (depending on available ports).

Numbers of buttons depend on selected ports in CAN HW setup.

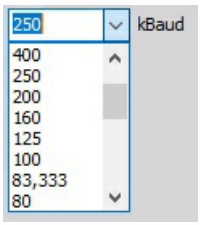
For further information about CAN visit ->Dewesoft Web page -> Support -> Downloads.

CAN speed - baud rate

The baud rate setting is very important. In fact, some vehicle operation can be interrupted if we connect to the bus with the wrong baud rate set.

Select the CAN speed from the kBaud field drop-down list or enter the value. Available are all common values from 5 to 1000 kBaud.

Dewesoft supports different CAN speeds for each CAN port. Under the baud rate edit box we have also a notification of how many messages came through the bus and how many of them were corrupted (red). This information shows if the baud rate is correct and also if the bus has any problems due to a bad connection or bus overload.



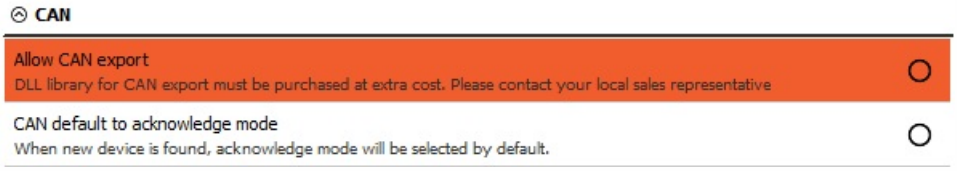
Import messages / channels from existing .dbc file

Press the DBC Library Import button to get a standard Open window. Now select the required *.dbc file and press the Open button to import the desired file and a list of all available messages and channels will appear.

(see also -> Messages/channel list)

You can edit the messages and channels manually.

Also export of CAN messages is possible. For that, please enable the Allow CAN export option in Dewesoft settings. The tuner can be found by opening System folder (go to Settings -> Advanced -> Hardware). The export can be enabled in the CAN section.

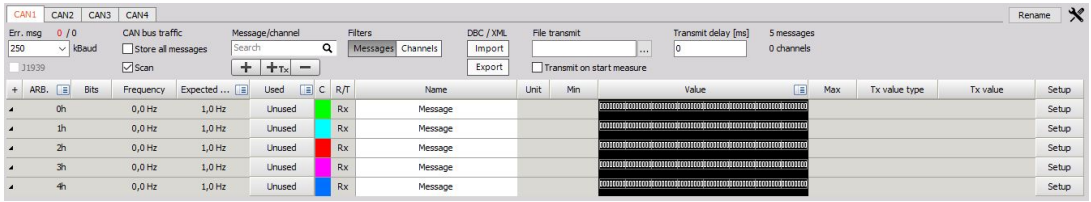


After restart of Dewesoft, export DBC option is also available in CAN channel setup.

Use scan mode to find messages / channels automatically.

When you have already connected a CAN bus system to the Dewesoft system, you can also click the Scan check-box in the upper left corner.

As soon as we check the Scan option, the messages which are coming from the bus will be displayed. So now we can see message IDs, the speed of messages and raw binary values coming from the bus.



The example above shows five new messages, which have been found automatically.

NOTE: **The scan function can also only find messages; it is not able to identify the signals (channels) within the messages.**

If we know, we can define the channels from the specification. We choose the Setup button and the empty message setup screen - Can channel setup window will appear (see below).

Selecting messages / channels

To select/deselect all message please click on the On/Off tab panel to get following options:

- **Select all** - this will select all messages and channels which are expanded in the current view. If you want to select all channels from all messages, it is easiest to use channel view and then Select all. All messages will be enabled as soon as there is a channel within that message set to Used.
- **Deselect all** - will set all channels to off
- **Remove all empty messages** - will erase all messages without channels
- **Remove all unused messages** - will erase all messages and channels within the messages which are set to off
- **Show all channels**- toggle will display all channels
- **Show used channels** - toggle will display only channels which are set to used

J1939 support

J1939 is used to enable special decoding of arbitration ID which includes the sender, receiver and the message ID itself. Arb ID is always extended in this case. This is most widely used on trucks. Please make sure that the bus type is really J1939 before enabling this option.

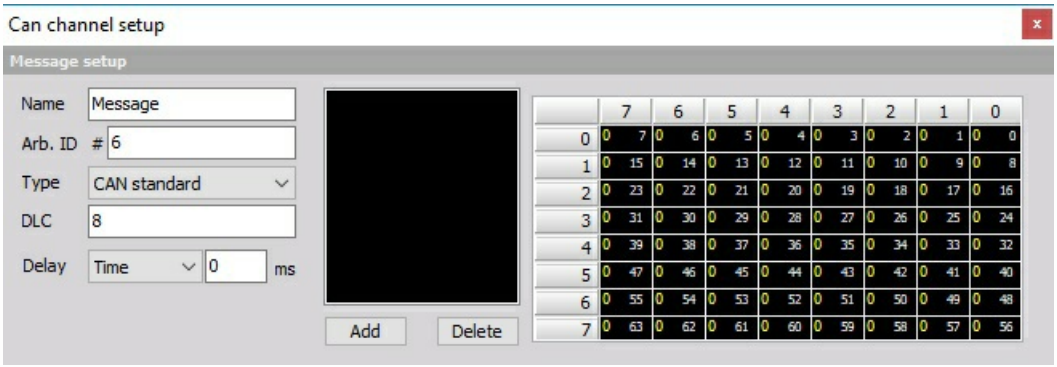
CHANNEL SETUP

Setup messages / channels

Add messages/channels manually - Can channel setup

To add a message manually simply click on the Message and then the Add button.

Setup window will automatically open for the new message:

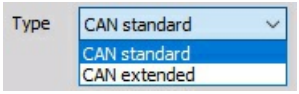


Message setup

The upper left area of the window is used to define the CAN message.

Before you add any channels you have to define the message itself:

- **Name** - The name of the superior message.
- **Arb. ID #** - ID number of your message on the CAN bus
- **Type** - Select between CAN standard and CAN extended from the drop-down list. Those two differs in identifier length - standard length is 11 bits and extended is 29 bits.

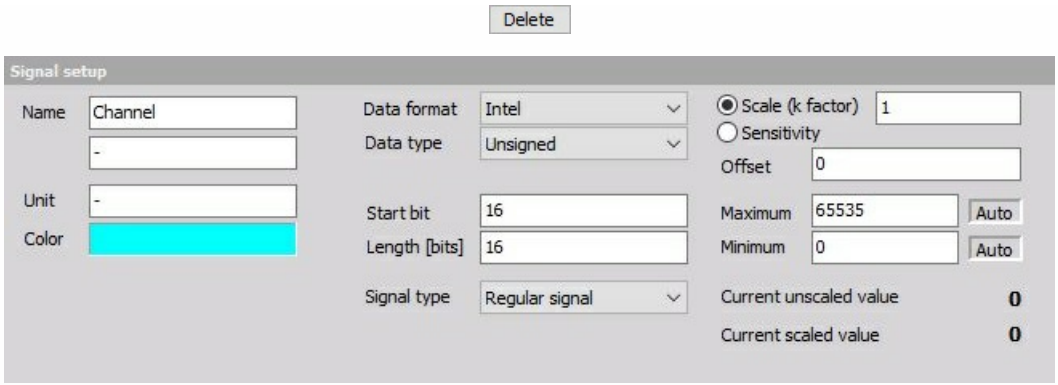


- **DLC** - DLC is the length of the message. It ranges from 1 to 8. As a standard, the DLC is set to 8.
- **Delay** - we can also enter the message delay in millisecond which shifts the time stamp of the message back in time. This can be used to perfectly synchronize the analog data the with CAN data with compensating the delays in digital data transmission.

Signal setup

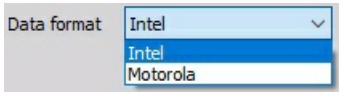
Now we are ready to define our CAN channels within the message. Press the Add button in the Can channel setup window to create a new channel.

Press the Delete button to remove the selected channel.

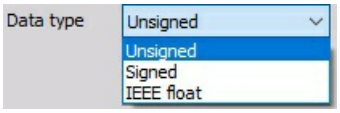


The Signal setup offers a lot of settings:

- **Name** - name of the channel (signal within the message)
- **Unit** - unit of the signal
- **Color** - color of the signal on displays and lists - color selector
- **Data format** - select between Intel (little endian) and Motorola (big endian) format from drop-down list:



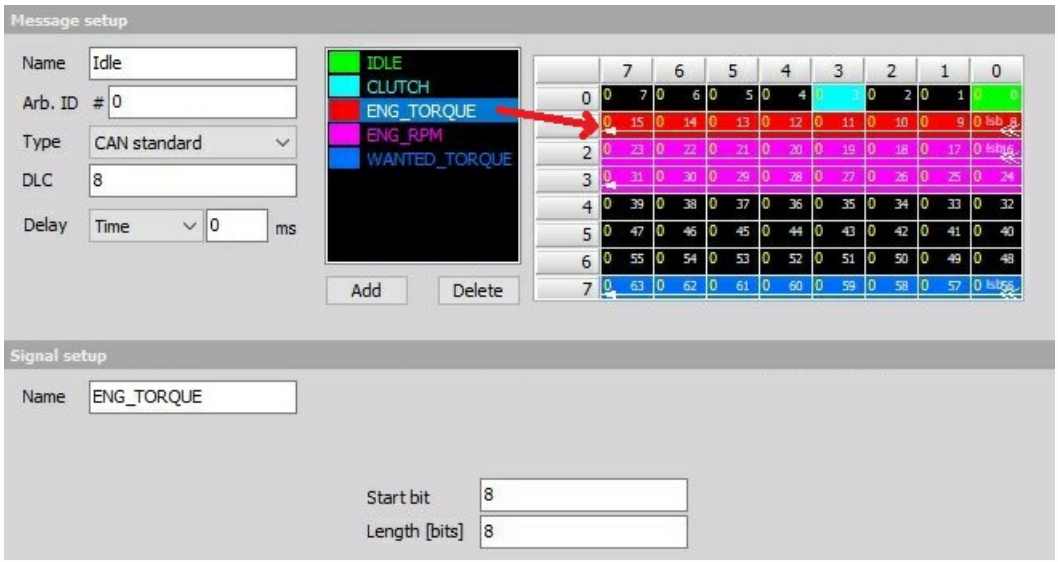
- **Data type** - select between Unsigned (only positive integer number), Signed (integer number where the first bit is the sign) and IEEE Float (floating point number) from the drop down list:



- **Start bit** - this value indicates the start of your signal within the message
- **Length (bits)** - this value defines the number of bits used for this channel
- **Factor, Offset** - like in the analog world, you can define offset and scale the CAN value to the physical value - scaling Factor, for example 0 to 100 % or 10 to 50 mm
- **Maximum/minimum** - depending on the settings above, the maximum and minimum possible values are displayed. With selecting the Auto button this values can be set automatically. When you use only a small range out of it, you can decrease the display range.

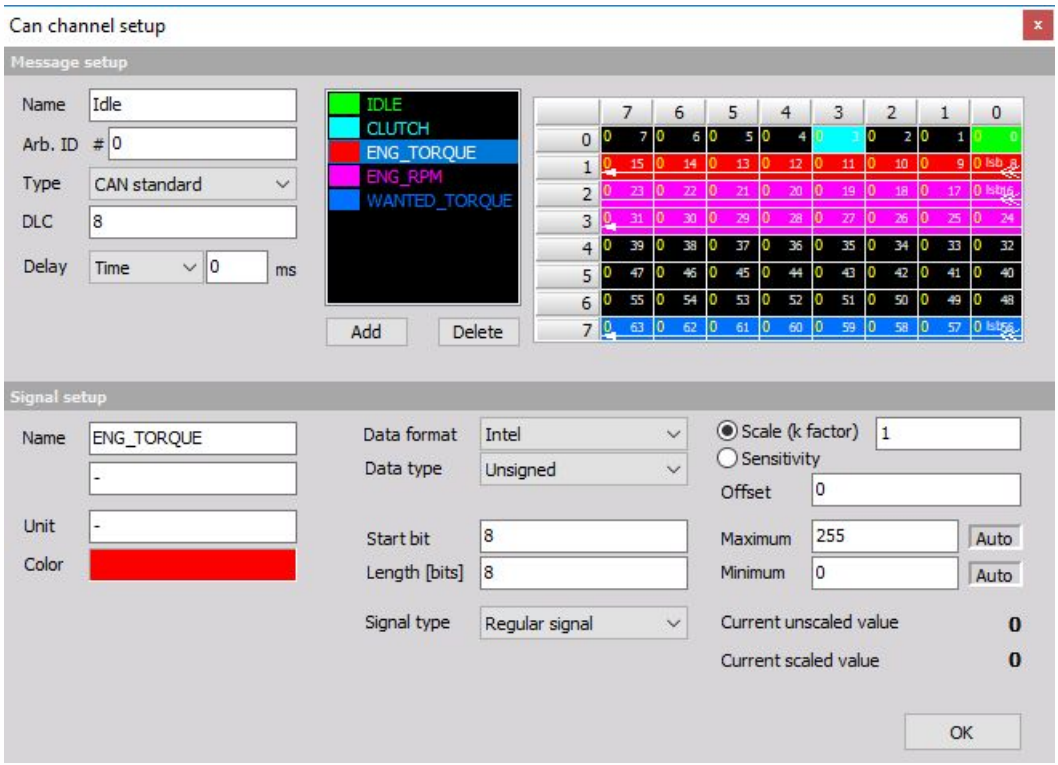
In addition to the signal settings, you have an online display of Current unscaled value and Current scaled value in lower right part of Signal setup area.

The currently available signals within the message are also displayed in the array on the upper area of the window to get a fast overview of your message:

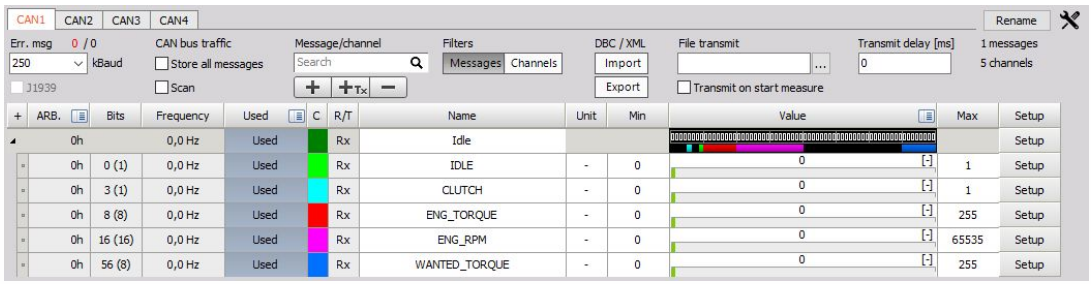


In this fields array, the running number of bit is displayed on right side of the field. The start bit is designated with double arrow, end bit with the simple arrow and both arrows are connected with a white line, which represents Length of the signal.

Example: CAN channel setup after defining several signals within one message



After you have defined all signals (channels) within your message, press the OK button to confirm and to leave the message setup. Two added channels will appear under the message on Message/channel list (if we expand the message to show also the channels).


CAN1
CAN2
CAN3
CAN4
Rename
Err. msg 0 / 0
250
kBaud
J1939
CAN bus traffic
Store all messages
Scan
Message/channel
Search
Tx
Filters
Messages
Channels
DBC / XML
Import
Export
File transmit
Transmit on start measure
Transmit delay [ms]
0
1 messages
5 channels
+ ARB. Bits Frequency Used C R/T Name Unit Min Value Max Setup
0h 0,0 Hz Used Rx Idle Setup
0h 0 (1) 0,0 Hz Used Rx IDLE - 0 0 [-] 1 Setup
0h 3 (1) 0,0 Hz Used Rx CLUTCH - 0 0 [-] 1 Setup
0h 8 (8) 0,0 Hz Used Rx ENG_TORQUE - 0 0 [-] 255 Setup
0h 16 (16) 0,0 Hz Used Rx ENG_RPM - 0 0 [-] 65535 Setup
0h 56 (8) 0,0 Hz Used Rx WANTED_TORQUE - 0 0 [-] 255 Setup

Edit messages/channels

To edit CAN messages or channels simply press the appropriate Setup button; you will receive the same CAN channel setup window that is already described above. Now you can do the required changes and confirm them pressing the OK button.

For information about message setup see above -> Message setup.

For information about channels setup see above -> Signal setup

Delete messages/channels

To delete CAN messages or channels, simply press the Remove button.

Message/channel list

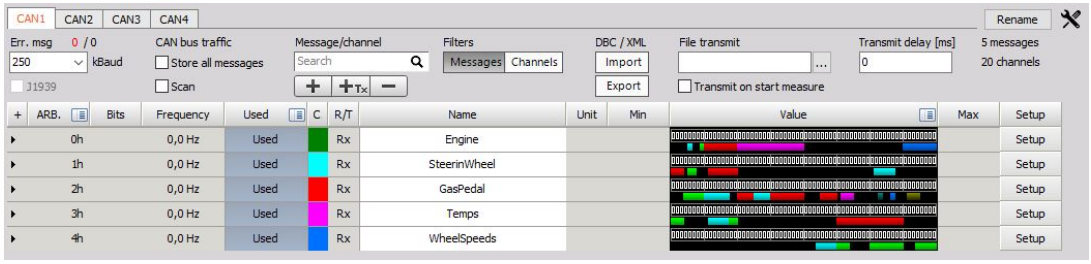
After you have defined all signals (channels) within all messages or you have import messages from the *.dbc file, all these messages appear on Message list or Channel list (channel overview) on the lower area of CAN setup window.

View options

These options enable two different views on message/channel list:

1. Messages list (view)

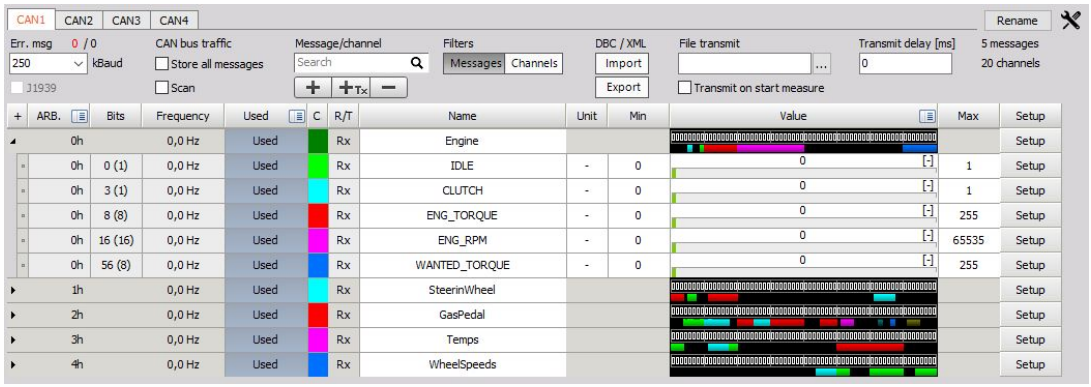
With selecting Messages View button list of defined CAN messages appears:



Columns that are shown in your CAN messages list:

EXP - to display/hide channels of particular message.

Use the button in this column to display all channels of this message and channel list will have the following appearance:



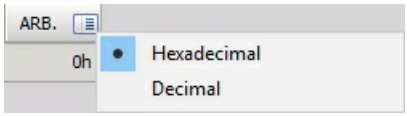
The button will change to a button and can be used to hide channels.

ARB.

- Shows the arbitration ID and refresh rate of the specific message.

Values can be displayed in Hexadecimal or Decimal form, use

icon and select the desired form from displayed the drop-down list.



ON/OFF

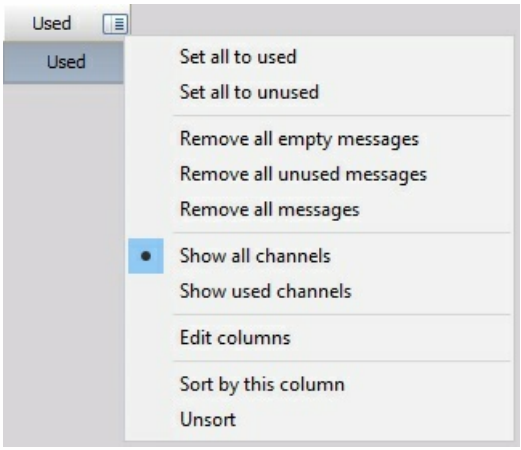
- To activate/deactivate CAN messages or channels for online display and storage.

Select messages/channels for online display.

Press the Unused button - it will change now to Used - to set the desired messages or channels active for CAN data acquisition and they are available in all instruments either as digital values or graph.

To activate all messages and channels, at the same time, click on the

icon or right-click into title On/off field at the top of this column and use Select all to activate all messages and channels or Deselect all to deactivate them.



NOTE: **Be aware that activating a message does not automatically activate the appropriate channels!**

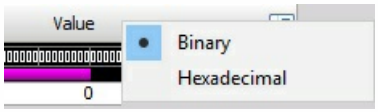
With that function, we can store whole messages without displaying or storing the channels itself.

After you have selected the required channels, you can hide the unused messages channels, automatically.

Simply click on the icon in On/off field at the top of the list again and select Show used channels. Or select Show all channels to display also the unused messages and channels again.

- **C** - channel color selector

- **Name** - CAN message (channels) name - just click on the NAME field and you can enter the new name as usual text.
- **Value** - current CAN messages / channels value. As a standard, the setup displays the current data transfer on the CAN bus in a binary format. Click on the CAN message menu Icon or right-click into title VALUE field at the top of this column to change the data format from Binary to Hexadecimal - simply select in the list.



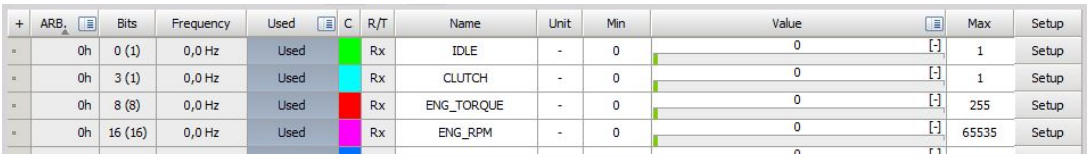
Example: displays the current data transfer in hexadecimal form

- **Setup** - CAN message (channels) settings. After selecting Setup button CAN channel setup window will automatically open. Now you can do the required message setting changes and confirm them pressing the OK button.

For information about Channel grid see -> Setup -> Channel grid.

2. Channels list (view)

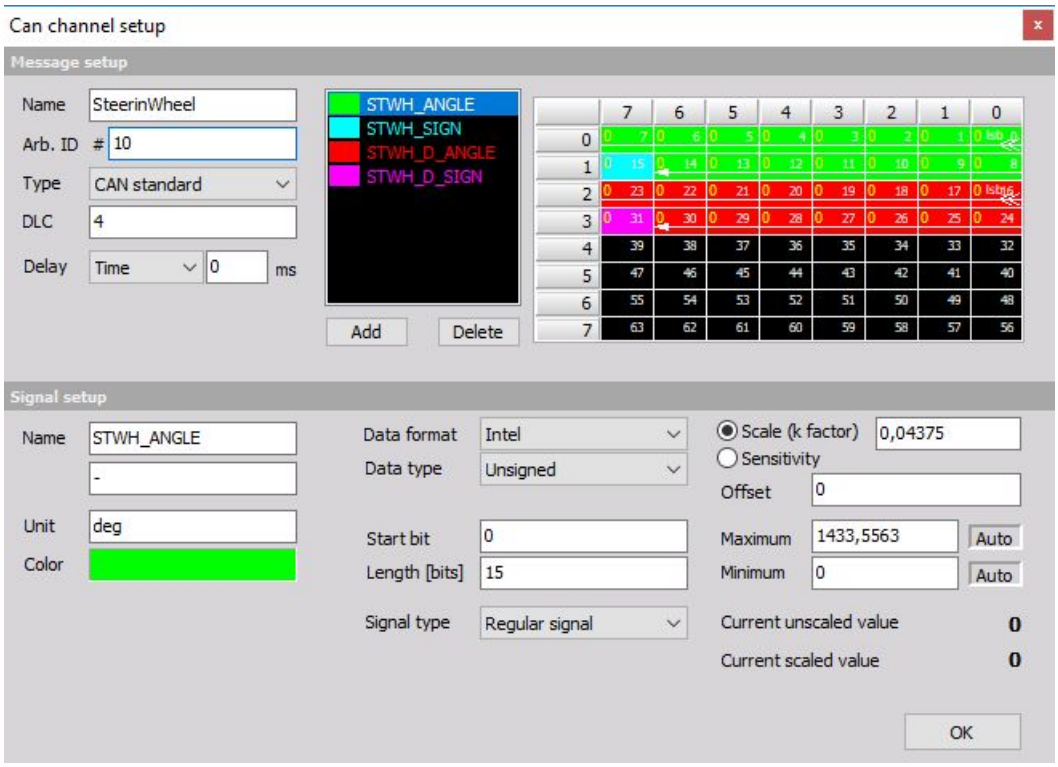
With selecting Channels view button list of defined CAN channels appear:



+	ARB.	Bits	Frequency	Used	C	R/T	Name	Unit	Min	Value	Max	Setup
▫	0h	0 (1)	0,0 Hz	Used		Rx	IDLE	-	0	0 [-]	1	Setup
▫	0h	3 (1)	0,0 Hz	Used		Rx	CLUTCH	-	0	0 [-]	1	Setup
▫	0h	8 (8)	0,0 Hz	Used		Rx	ENG_TORQUE	-	0	0 [-]	255	Setup
▫	0h	16 (16)	0,0 Hz	Used		Rx	ENG_RPM	-	0	0 [-]	65535	Setup

Columns that are shown in your CAN channels list are similar as in Messages view, except:

- **Name** - CAN channels name - just click into the NAME field and you can enter the new GPS channel name as usual text.
- **Value** - current CAN channels value and unit
- **Setup** - CAN channels (message) settings (see below). After selecting Setup button CAN channel setup window will automatically open for entering required channel setting changes:



This window is the same as for changing message setting (in Name field appear Message name), only it will automatically display the current selected channel. Here can be set also other channels in this Message.

GENERAL INFORMATION

Currently only the following Vector hardware is supported in Dewesoft and can be used for CAN FD:

- VN1610
- VN1611
- VN163
- VN1640
- VN7600
- VN7610

NOTE: **Minimum DEWESoft version: X3 SP4.**

Latest Vector drivers need to be installed and computer rebooted, before Vector HW can be used with Dewesoft.

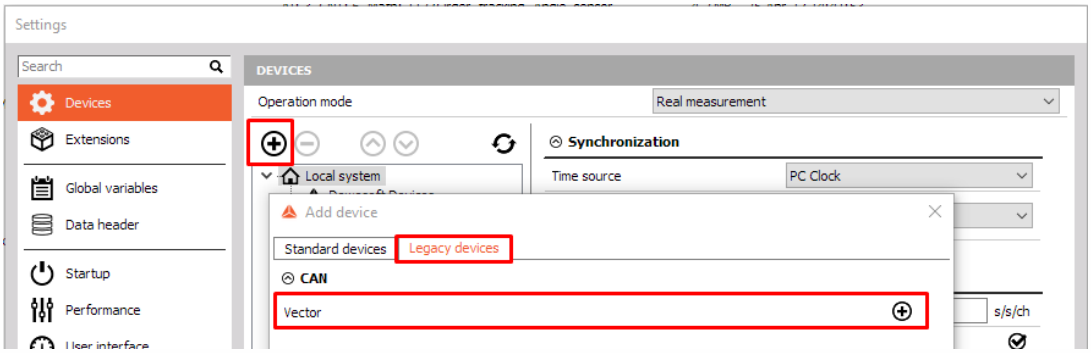
https://vector.com/vi_downloadcenter_en.html

LICENSING

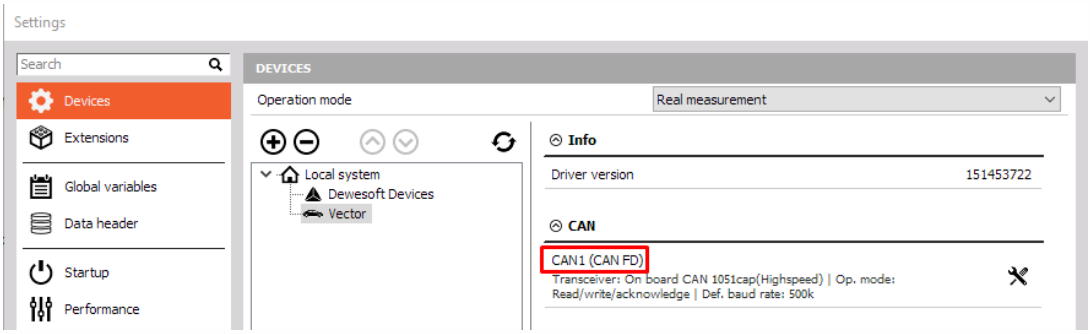
The plugin requires a valid DEWESoft® CAN FD license. To test the plugin, you can use an Evaluation license.

HARDWARE SETUP

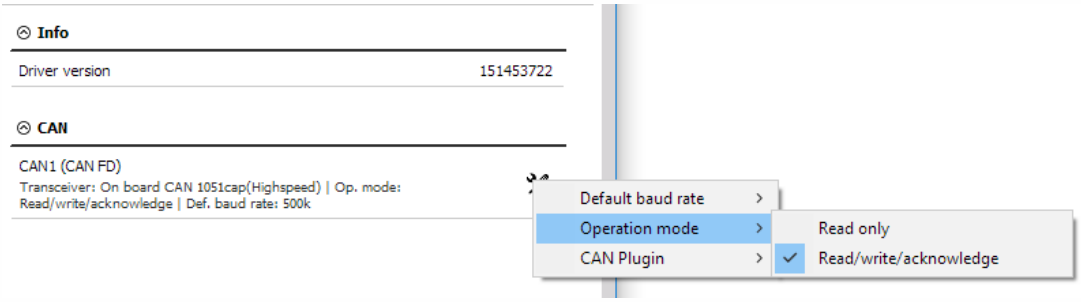
In hardware settings of Dewesoft, Vector hardware needs to be added:



If the driver is installed and correct version of Dewesoft used, the device will appear in the device list and it will display CAN FD next to it.

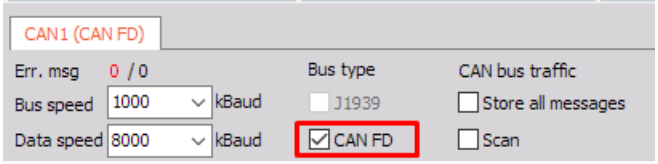


Operation mode can be changed in the hardware settings of Dewesoft from Read only to Read/write/acknowledge (**often required also for reading only**).



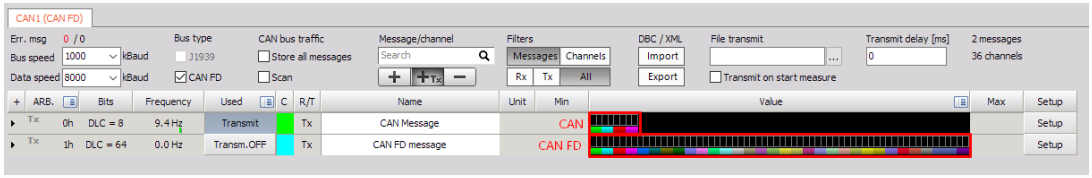
CHANNEL SETUP

Usage of CAN FD is exactly the same as CAN. To enable CAN FD, the checkbox next to it needs to be ticked.

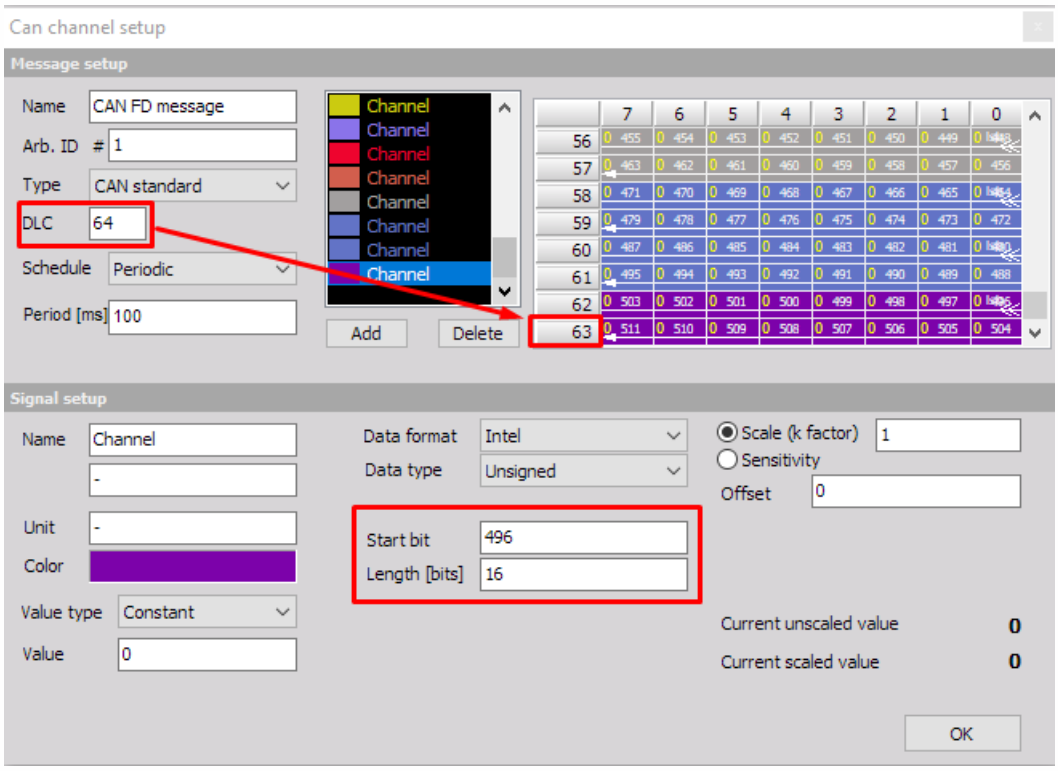


There are two main differences with CAN FD compared to CAN:

Message length is increased from 8 byte → 64 byte and data rate is increased up to 8 Mbit/s.



In the message settings dialogue, DLC can be set up to 64 instead of 8 with CAN. This means that the total length of the message can be up to 512bits.



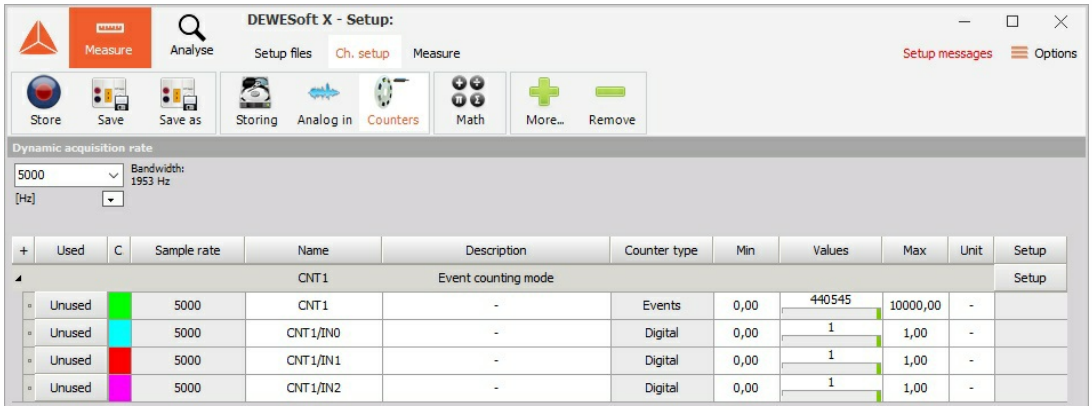
COUNTERS

Dewesoft Counter module is used to perform counting and frequency measurements. The counter setup depends on the used hardware platform.

Typical applications are: event, gated event and up/down counting, encoder measurements, period and pulsewidth measurements, two pulse edge separation and frequency/supercounter.

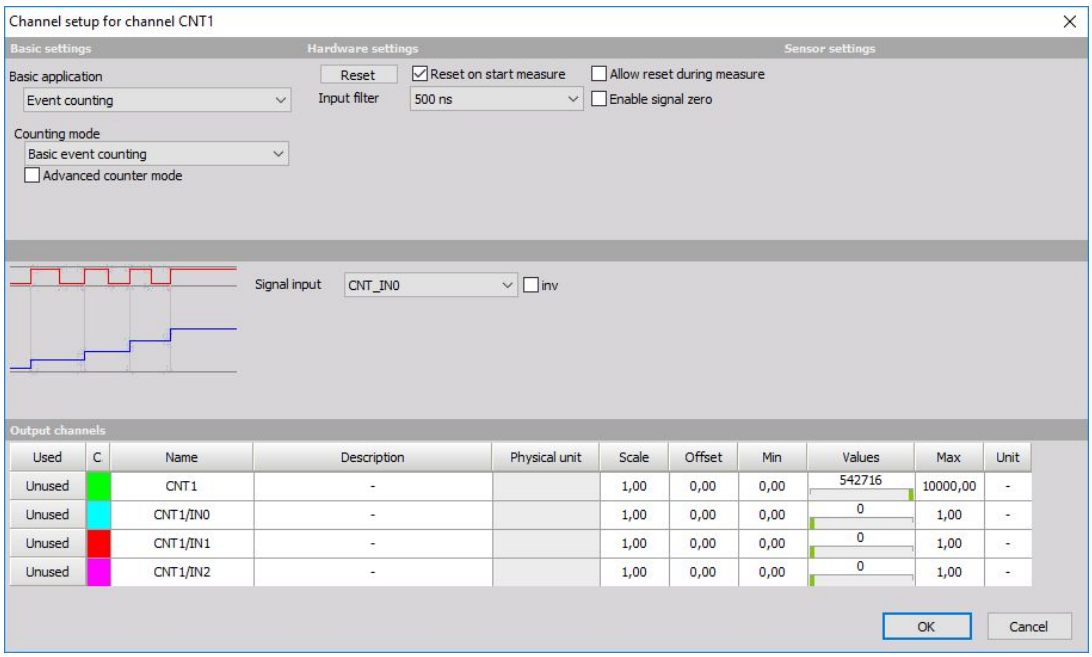
When you select a Counter tab on Dewesoft Setup screen, the Counter module will appear on the lower part of the screen:

For instructions on how to Add new module see -> Setup -> Addmodule.



For information about Slot, On/Off, C and Name column see - Channel grid.

Press Setup in SETUP column to do the base counter settings in Channel setup window:



General / Sensors parameters

- General parameters - are similar to the analog input channel settings and contain the Channel name, Units, Color and Min and

Max value -> see -> Analog (in) Channel Setting.

- Sensors parameters - are similar to the analog input settings and contain the Sensor groups, Sensors and Description: -> see -> Analog (in) Channel sensor setting.

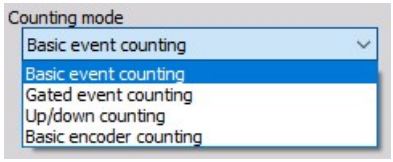
Hardware parameters

Set and number of parameters (settings fields) are dependent on the used hardware.

- Counter mode

Option of this field depends on the used hardware and applications respectively and can be:

- events, gated event and up/down counting
- encoder measurements
- period and pulsewidth measurements
- two pulse edge separation



For information about Hardware parameters setup click on caption above.

Some common counter Hardware parameter expansions are:

- Reset on start measure

Check this box to counter reset on start of measurement.

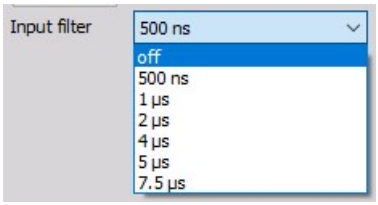
- Reset button

With a click on the button we can reset the actual counter to zero.

- Input filter

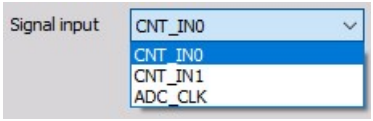
The filter is an important setting to prevent double counts. We need to choose the filter to react a bit faster than what we expect our events to be or, with a different logic; we need to set them a bit slower than what we expect to have glitches in the signal.

Except for frequency counter that can be selected in this field from the drop-down list value of input filter in ns or off for no filter. An option of this field depends on the used hardware (base clock).



- Signal input

Also, the Signal input depends on the used hardware, for example Source Pin (external clock), 100 kHz or 20 MHz (internal clocks) can be selected from the drop-down list in this field.



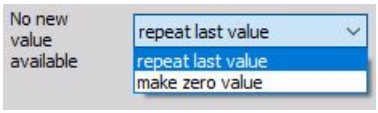
- Signal inverting

When the normal state is high, sometimes it is nice to invert the signal by choosing the inv checkbox beside the signal field. This has two effects: first is that the levels will change, so normal level will be low and consequentially the counter will count on falling edges of the input signal.

- No new value available

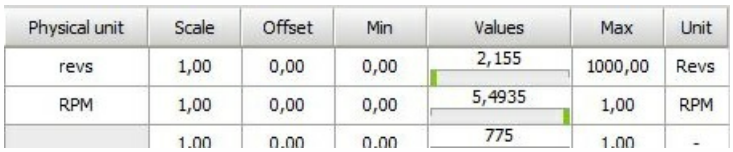
The field no new value available tells the software what to do when no new value is available. The new value is calculated only when a signal changes the value from low to high. Therefore the value can't be calculated most of the time.

If we choose to repeat the last value, then the same value will be added until a new transition is made. Alternately, we can select make zero value when no new value is available, so we will have only spikes at the points of new data and the rest of the data the value will be zero.



- Scaling - the scaling area can be used to adapt the sensor to your system by entering the desired value of:
- counts for Events and Encoder Counter mode
- ms (milliseconds) for Period Counter mode
- Hz for Frequency Counter mode

in upper field and value for 'equals' in channel Units (entered in General part of this setup window) in lower fields or by pressing Calibrate from the current button.



Physical unit	Scale	Offset	Min	Values	Max	Unit
revs	1,00	0,00	0,00	2,155	1000,00	Revs
RPM	1,00	0,00	0,00	5,4935	1,00	RPM
	1,00	0,00	0,00	775	1,00	-

for example, the connected sensor delivers 1800 pulses per 360 degrees

On right part Scaling section of Channel setup for channel CNTn window:

- Current unscaled value [cntUnit=counts, ms or Hz]
- Current scaled value [Units= -, m, Hz....]

and on the lower part, the actual equation for Output value is displayed.

EVENT, GATED AND UP/DOWN COUNTING

Event counting is one of the simplest counter operations.

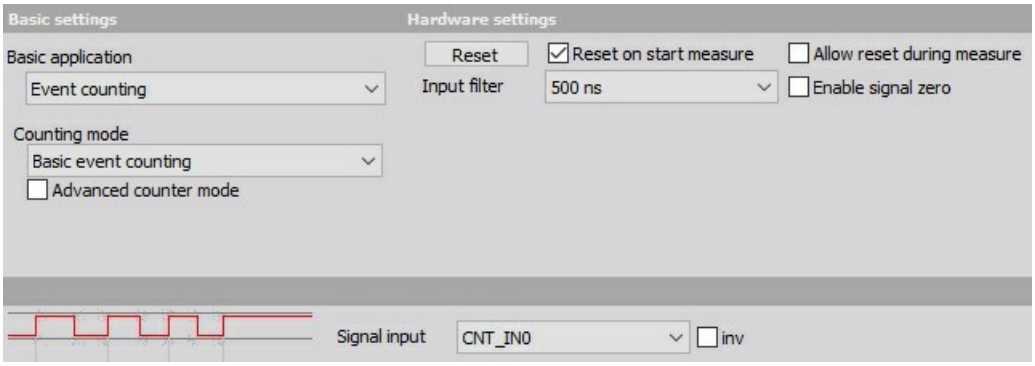
- **Required hardware** - Dewesoft USB
- **Setup sample rate** - at least 1 kHz

There are two special modes of event counting available:

1. Gated event counting, where events are counted only when a gate signal is high.
2. Up/down counting, where the events are counted up when the gate is high and down when the gate is low.

For additional help with Event counting setup and measurement visit Dewesoft PRO training.

Simple event counting

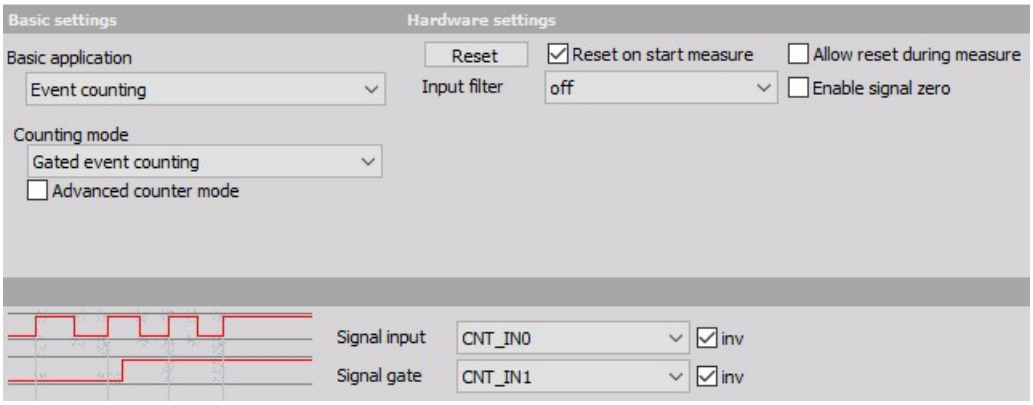


1. The Counter mode 'Event counting' is already selected by default.
2. Then we choose the Signal input. Usually, the signal input is Source Pin, but if we have connected other signal, we need to select that signal as the input (e.g. CNT_IN0 above).

When the normal state is high, sometimes is nice to invert the signal by choosing the inv check box>. This has two effects: first is that the levels will change, so normal level will be low and consequentially the counter will count on falling edges.

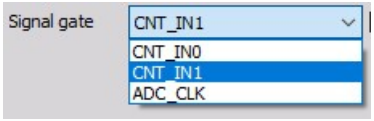
3. It is also reasonable to set the Input filter to prevent double counts. We can select Count direction either to count up or down. This value can be selected from the drop-down list in this field. ## Gated event counting

Gated event counting is the mode where the counter counts only when a gate signal is high. It is only available with Dewesoft counter module. This application is based on the previous 'Simple event counting' section (see above).



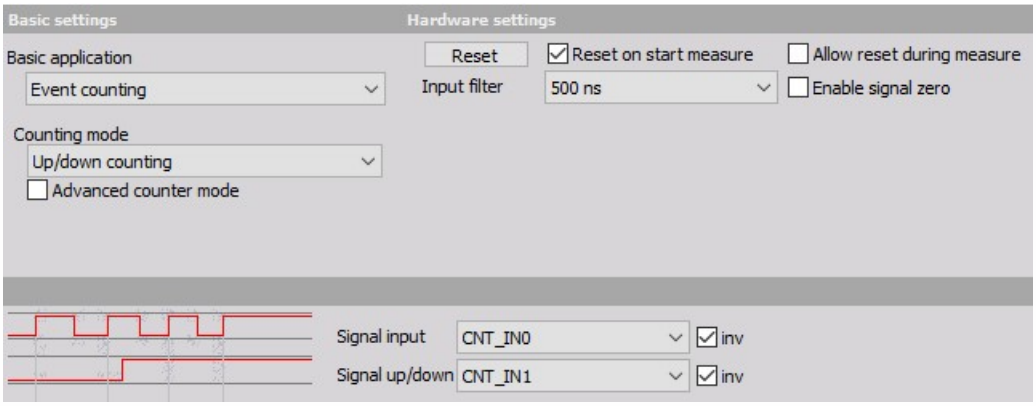
Additional to the above 'Simple event counting' settings (here we choose 'Gated event counting' as Counter mode we must set:

Signal gate (e.g. CNT_IN1 above). The counter will count the transitions from low to high only when a signal gate is high. Signal channel can be selected from the drop-down list. If we have the Signal gate inverted by choosing the inv checkbox (normally it is high), we choose to invert also the gate signal, then it will count only when a gate signal level is low.


Signal gate
CNT_IN1
CNT_IN0
CNT_IN1
ADC_CLK

Up / down counting

Up/down counting is counter operation which counts up when a gate is high and counts down when a gate is low. This application is based on the previous 'Gated event counting' section (see above).

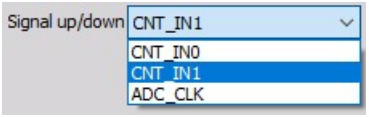


Additional to the above 'Gated event counting' settings:

- We choose 'Up/Down counting' as Counter mode

Select Signal input and Signal up/down (e.g. CNT_IN1 above). The counter will count the transitions from low to high - counts up when Signal up/down is high and counts down when this signal is low. Signals channel can be selected from the drop-down list.

If we have the Signal up/down inverted by choosing the inv checkbox (normally it is high), we choose to invert also the gate signal, so that it will count down only when gate signal level is high.



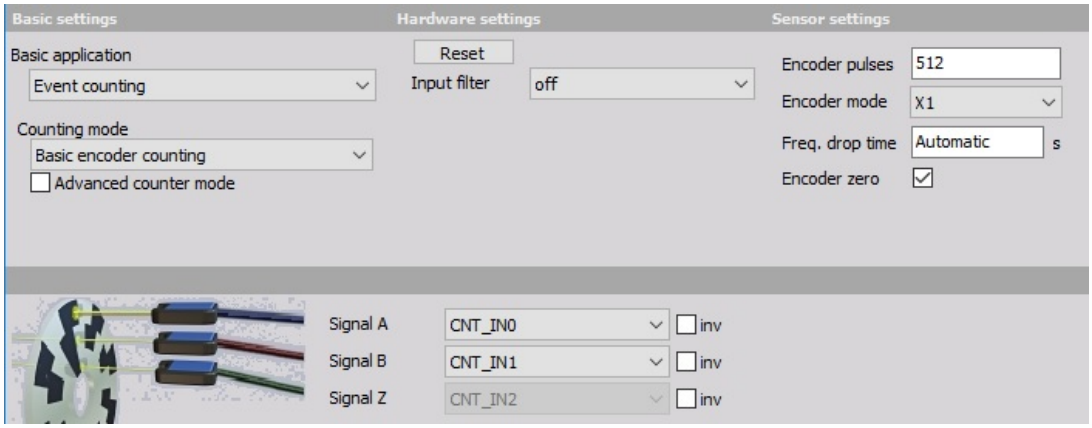
ENCODER

Encoder is a wheel (or linear bar) with marks on it. We usually have encoders with two marks (e.g. A and B signal) with a phase difference of 90 deg. one to another to determine the direction of movement and a zero pulse - one pulse per revolution which can tell us the absolute position of the encoder (e.g. 0° pulse).

- **Required hardware**- Dewesoft USB
- **Setup sample rate** - At least 1 kHz

For additional help with Encoder setup visit Dewesoft PRO training.

X1, X2, X4 modes



1. We choose 'Basic encoder counting' as the Counter mode.
2. Then we choose the Signal inputs: Signal A and Signal B (e.g. CNT_IN0 and CNT_IN1). Signal Z (e.g. CNT_IN2) is preset by Dewesoft. Signals can be selected from the drop down lists.
3. We set the Encoder mode to:

- **X1** - to measure just the rising edges of input A - the output of this counter is the counter which counts up when signal A leads signal B and counts down when signal B leads signal A; the positive edges of the signal A is used to make the counts
- **X2** - measures rising and falling edges of input A; therefore the resolution will be increased by a factor of 2 (also the scaling has to be changed)
- **X4** - mode measures rising and falling edges of both signals signal A as well as signal B, the resolution of the measurements is therefore increased by a factor of 4 (also the scaling has to be changed)

NOTE: **The X2 and X4 modes are extremely helpful if we have slow movement (for example with linear encoders) because it will actually increase the resolution of measurement by a factor of two or four. If we have fast dynamic measurement (like torsional vibration) it will sometimes introduce more errors if we use X2 and X4 mode, because those two modes assume that the gap ratio is exactly 0.5 and that the encoder electronics switches exactly with the same speed between dark and light areas. We can evaluate this error with Period and pulsewidth measurement.**

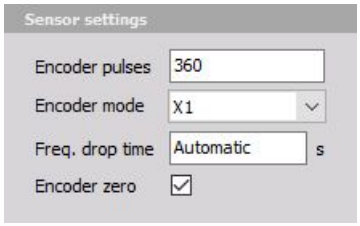
4. We set also the Input filter to match our highest frequency to prevent double counts.

For more information about Counters see -> Counter hardware parameters.

Zero pulse

The zero pulse is used to reset a measurement when a Z pulse is recognized.

The only change to the setup is to check the Encoder zero check box. This will reset the counter value to 0 when a zero pulse is passed. We also need to set the number of Encoder pulses for internal calculations.

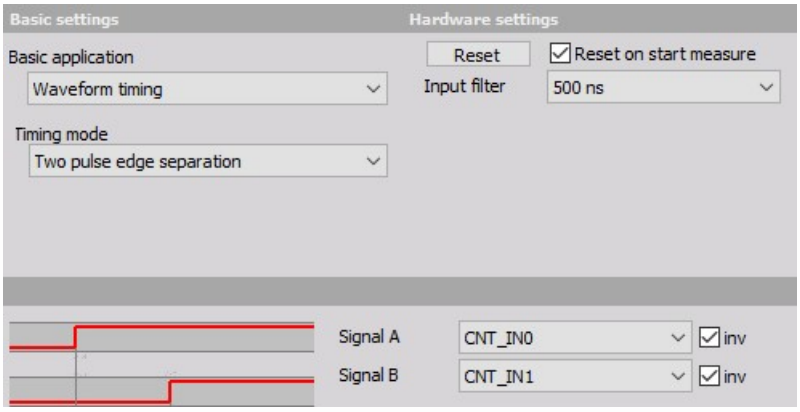


TWO PULSE EDGE SEPARATION

- **Required hardware** - Counter input
- **Setup sample rate** - At least 1 kHz

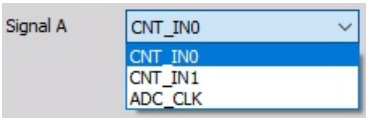
Two pulse edge separation is a special mode of counter operation. It measures the time between the rising edges of two signals. This is useful for measurement of the time between two events with a very high precision. The typical application is measured, for example, the velocity of an object passing by two sensors.

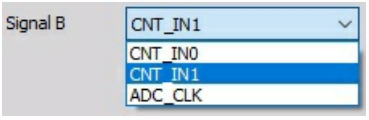
For additional help with Two pulse edge separation setup visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training.



1. Select the Counter mode as 'TwoPulseEdgeSep'.

2. Choose the Signal start and Signal stop input signal (e.g. CNT_IN0 and CNT_IN1 in above example). Both signals can be selected from the drop-down list:





When both of the signals are inverted, the real trigger condition will be falling edge of the input signal, because the values are normally high when events don't occur.

3. In the field no new value available tell the software what to do when no new value is available.

4. Set the signal Input filter to prevent glitches

For more information about no new value available and Input filter see -> Counter hardware parameters

We can use this value for example for speed measurements, if we are triggering (for example) from two light sensors. If we know an exact distance between them, the speed would be $v = s/t$,

where s is a constant and t is the result of the edge separation measurement. This can be done in the math channel.

ADVANCED COUNTER MODE

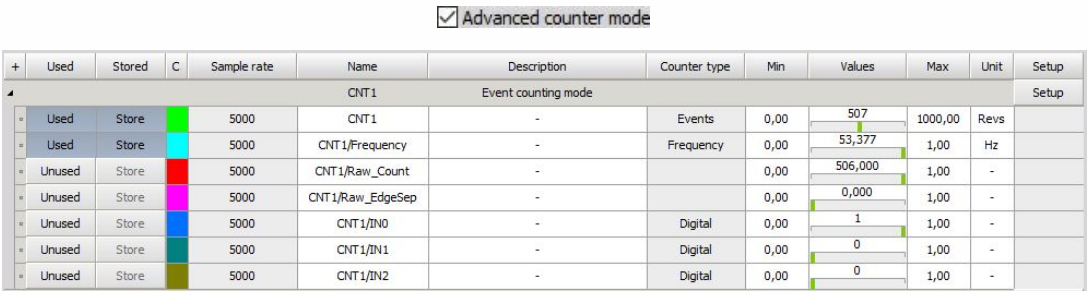
- **Required hardware** - Dewesoft USB
- **Setup sample rate** - At least 1 kHz

The problem with previous (traditional) counters is that the value of the counter is latched only at a sample rate interval. Therefore we have only discrete values on each sample.

But since the Dewesoft counters can measure where EXACTLY the position of the pulse is in between two samples, we can calculate two things out of this: the exact interpolated position of the counter at the sample point as well as the exact frequency of the pulses.

Dewesoft can perform these measurements on a single counter.

We have to select "Advanced counter mode" in setup:

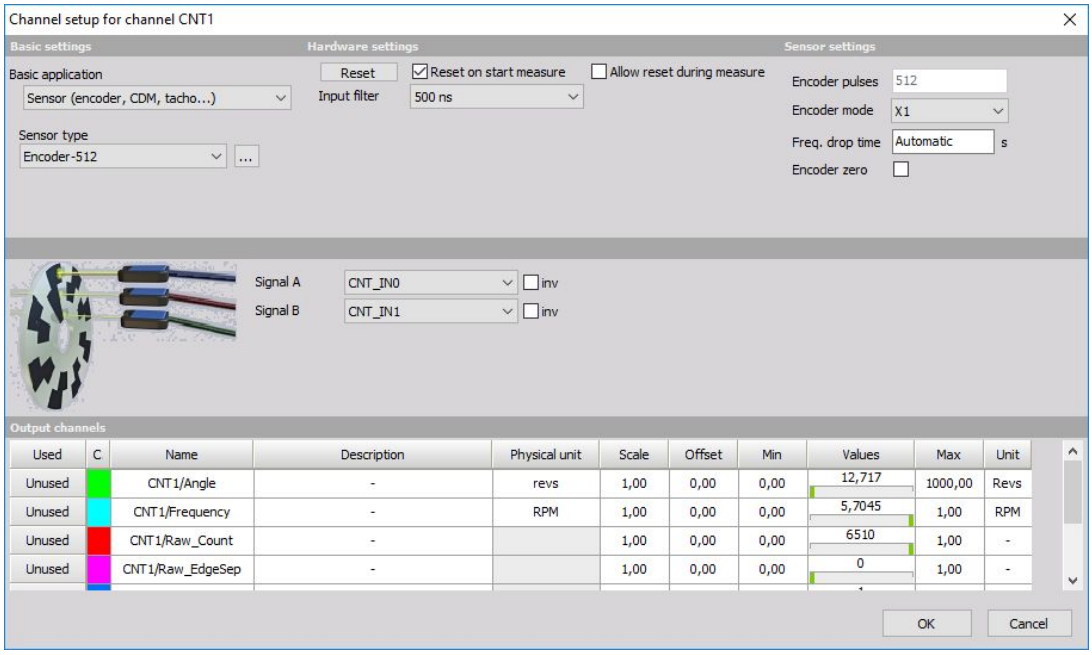


SENSOR MODE

- **Required hardware** - Dewesoft USB
- **Setup sample rate** - At least 1 kHz

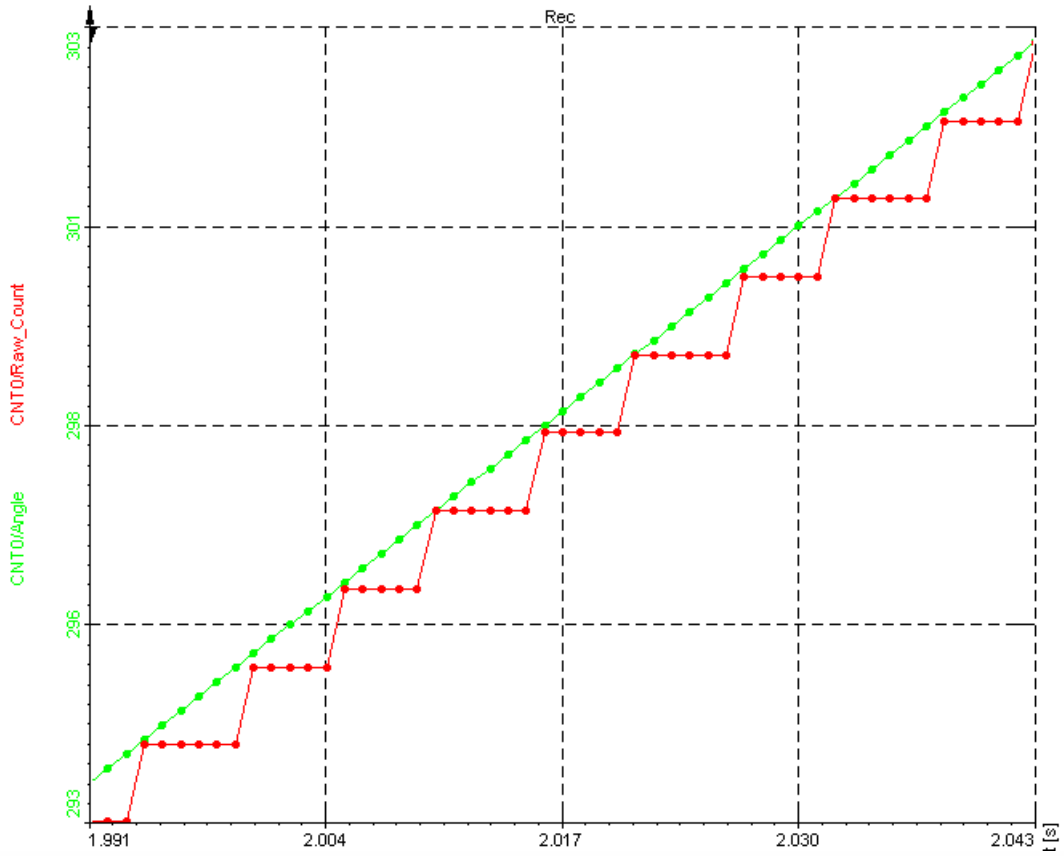
Sensor mode is very easy way to measure signals coming from the real sensors. By choosing the Sensor mode, Dewesoft offers a list of sensors, defined in: Settings -> Counter sensor editor.

Sensor modes includes rotary, linear encoders, CDM, geartooth, geartooth with missing or double teeth, tacho signals. The use of it is very simple - just select the sensor from the list. In our encoder-512 is selected. All the settings are done similar to the Basic encoder mode or counting mode, depending on defined sensor type.



Difference between normal and advanced counter mode can be easily seen by defining the counts as the output unit of the angle and comparing CNTx/Angle and CNTx/Raw_Count signal on the recorder.

Red signal is a normal counter value. Green signal calculates points between two counts and also takes into account WHERE between two analog samples the counting event happened. Therefore the green line out of the supercounter shows the real position of the counter as precise as possible, depending on the AD card with 80-100 MHz resolution independent of the sample rate.



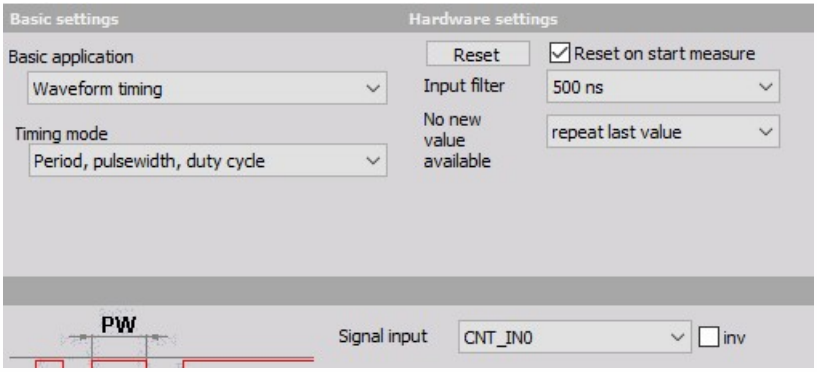
For additional help with Super-counter setup visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training.

TIMING MODES

Period and pulsewidth measurements are similar in function. The period measures the time between two consecutive low to high transitions while the pulsewidth measures the time that the signal is high.

- **Required hardware** - Dewesoft USB
- **Setup sample rate** - At least 1 kHz

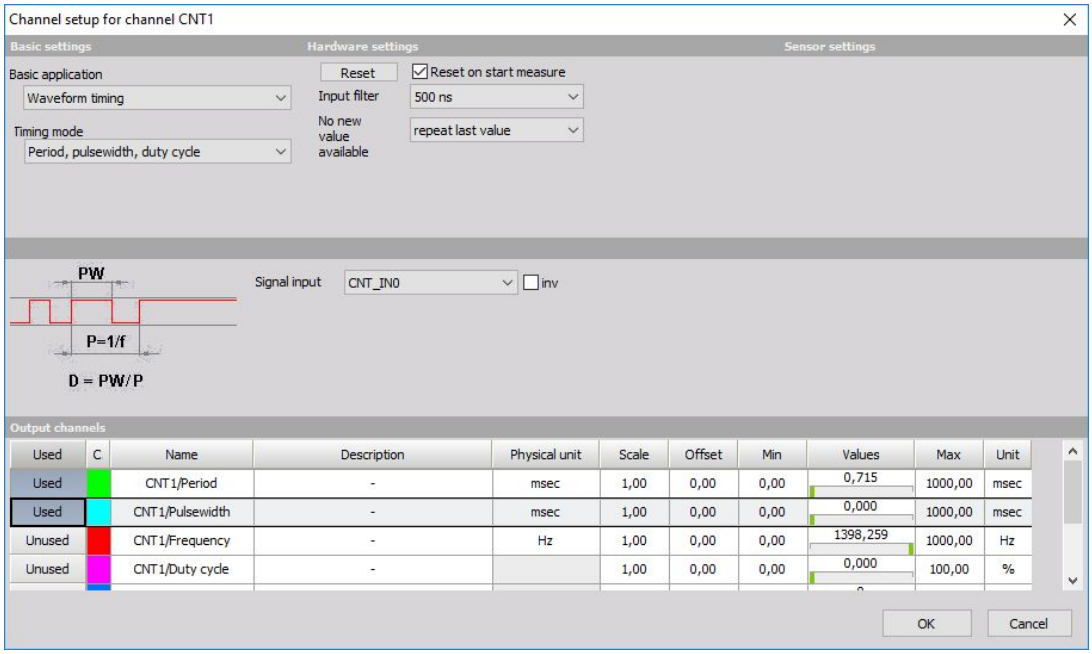
Period, Pulsewidth, Duty cycle measurement



1. Select the Counter mode as 'Period, pulsewidth, duty cycle'. 2. Choose the Signal input (e.g. CNT_IN0 in above example) and set the signal Input filter to prevent glitches. 3. In additional field no new value available tells the software what to do when no new value is available.

For more information about Counters see -> Counter hardware parameters.

We can also use the period and pulsewidth measurement combined to do the measurement of Duty cycle of the signal, where we measure the ratio between the high (or low) pulse of the signal and the period.

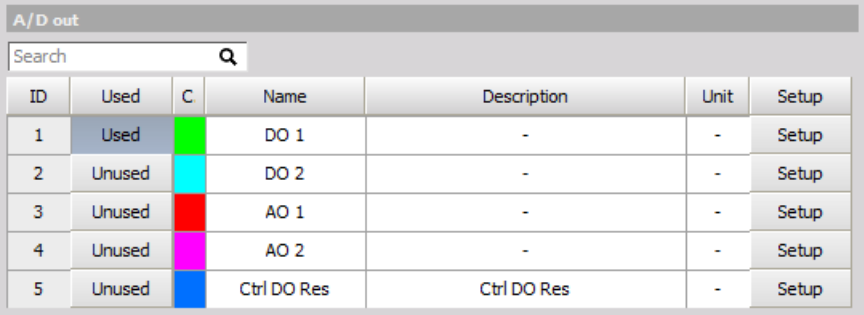


For this measurement, we need two counters: one is set to period and another one is set to pulsewidth as described above.

For hints about Duty cycle setup and measurement see -> Dewesoft Web page -> PRO Training.

ANALOG/DIGITAL OUT

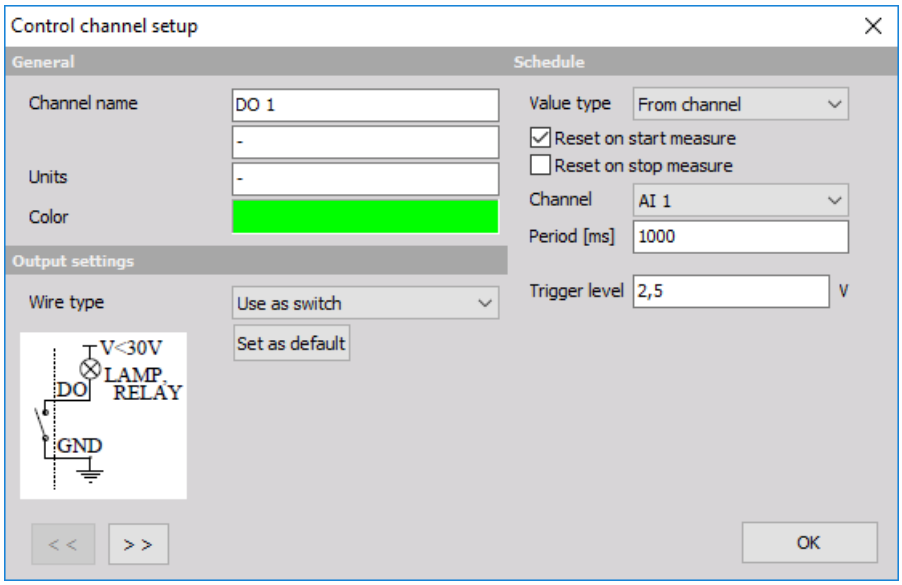
After adding a new module (for instructions on how to Add new module see -> Setup -> Add module) the A/D out tab on Dewesoft Ch.setup screen must be selected. When we open a new A/D out module a setup screen will appear, displaying all of possible analog and digital outputs supported by the connected device.



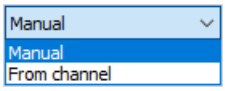
The A/D module is similar to Function generator, we can even use analog outputs.

Control channel setup

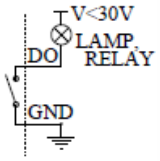
By opening the control channel setup we can set:



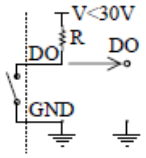
- General information
- Channel name
- Displayed name of channe
- Units
- Displayed units
- Color
- Color of channel
- Schedule
- Value type



- If from channel is selected additional settings like Channel, Period and Trigger level must be selected.
- Reset on start measure
- Reset the state of output on start of the measurement.
- Reset on stop measure
- Reset the state of output when the measurement is stopped.
- Output settings
- Use as switch



- Use as digital output


V<30V
R
DO
DO
GND

GPS

Dewesoft GPS Acquisition module offers the possibility to acquire GPS data. Depending on the used GPS system, different data is available like position, velocity, available satellites, and many more. GPS systems are used to determine the position on earth, velocity and absolute time.

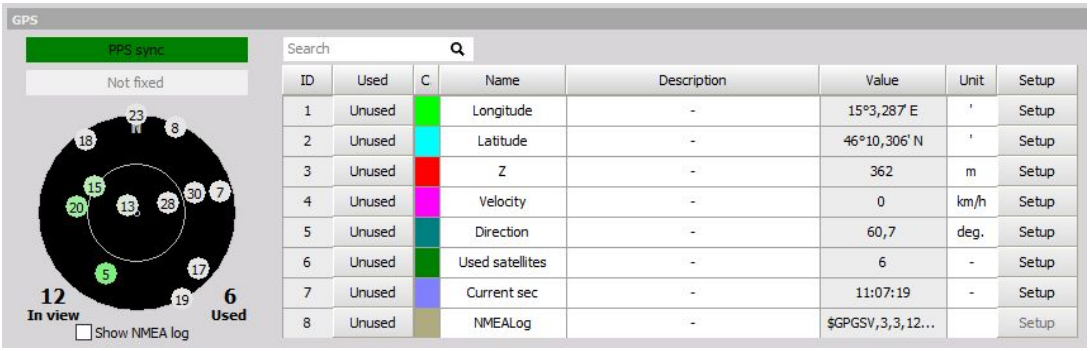
- **Required hardware** (VGPS - HS, VGPS-HSC, VGPS, Leane VSAT, Javad, Microsat or any NMEA compatible GPS)
- **Setup sample rate** - At least 1 kHz

For additional help with GPS Acquisition visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training .

For additional Dewesoft manuals visit -> Dewesoft Web page -> Support -> Downloads.

When you select GPS tab on Dewesoft Setup screen, on lower part of screen GPS settings will appear:

- GPS channel list - with information about GPS channel Slot number, On / Off state, Name, Value column and primary SETUP column with Setup button to call GPS channel setup window
- GPS information - display information about all inView and Used satellites constellation on the sky, PPS synchronization, SBAS or WAAS Differential mode and to show NMEA log information



ID	Used	C	Name	Description	Value	Unit	Setup
1	Unused		Longitude	-	15°3,287' E	'	Setup
2	Unused		Latitude	-	46°10,306' N	'	Setup
3	Unused		Z	-	362	m	Setup
4	Unused		Velocity	-	0	km/h	Setup
5	Unused		Direction	-	60,7	deg.	Setup
6	Unused		Used satellites	-	6	-	Setup
7	Unused		Current sec	-	11:07:19	-	Setup
8	Unused		NMEALog	-	$GPGSV,3,3,12...		Setup

After GPS channel setup is done press the Unused button - it will change now to Used - to switch on the acquisition.

GPS channels which are available (all channels are not available with specific types of receivers):

- **X absolute** - current longitude (the position from Greenwich - prime meridian, positive is east). Raw data in the data file is in minutes (from example above 15*60+3=903)
- **Y absolute** - current latitude (0 degree is equator, positive is north) raw data in the data file is in minutes (example from above 46*60+10=2770)
- **Z** - in meters
- **Velocity** - vector velocity of movement
- **Velocity Z** - velocity in Z coordinate
- **Direction** - direction of travel (0 is north)
- **Distance** - traveled distance calculated from velocity
- **Used satellites** - number of satellites used in GPS solution
- **Current sec** - number of elapsed seconds from midnight
- **Mark input** - status of external signal input (used sometimes for brake switch)

- **Acceleration** - current acceleration vector calculated from velocity
- **GPS fix quality** - description of GPS fix - only available for some receivers, it could be Standalone, WAAS/EGNOS sat differential, RTK float and RTK fixed. Second receives data from fixed satellites while the last two require differential data from the base station. RTK fixed is the best possible solution providing cm accuracy of the solution.
- **NMEALog** - raw text messages coming from GPS

GPS channel list

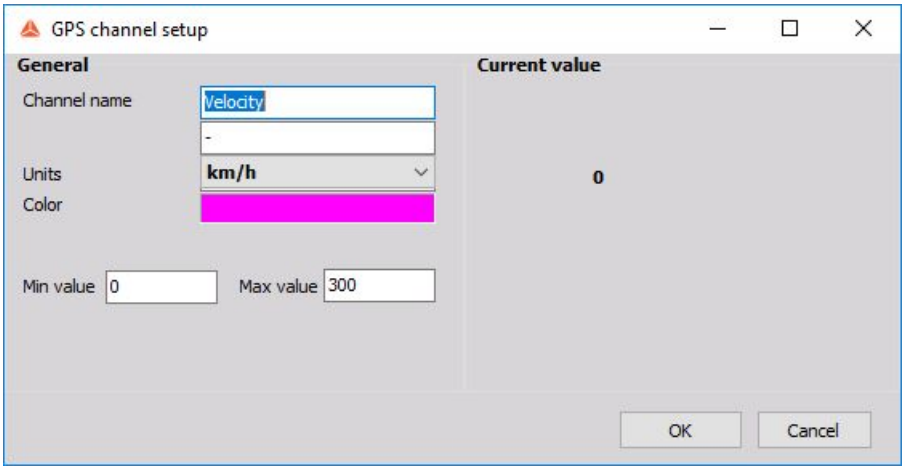
First let's look at the columns that are shown in the GPS channel list, and describe what each is for:

- **SLOT** - GPS channel input slot
- **ON/OFF** – activate/deactivate GPS channel - press the Unused button - it will change now to Used - to set the desired GPS channel active for GPS data acquisition and they are available in all instruments either as digital values or graph
- **C**– channel color selector - this color will carry through the text and graph representations of this channel throughout all DEWE software procedure screens
- **NAME** – GPS channel name - just click into the NAME field and you can enter the new name as usual text
- **VALUE** – current GPS channel value
- **SETUP** – GPS channel settings (see below)

For information about Slot, On/Off, C and Name see -> Setup -> Channel grid.

GPS channel setup

Press the Setup button of the GPS channel you want to change. The following GPS channel setup window will open:



In the General part of GPS channel setup screen you can enter:

- **Channel name** - can be Velocity, Distance, Coordinates,... and are entered as text
- **Units** - of measurement value: m, km/h,... are entered as text
- **Color** - of displayed channel - simply click on the colored field to open the color selector
- **Min value, Max value** - fields represent a definable standard display range and are entered as text

Edit and navigation keys are same as in other Windows application.

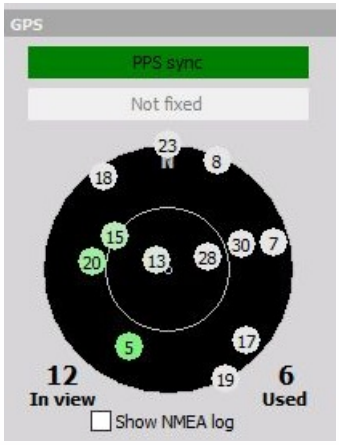
On right part of GPS channel setup window Current value of channel is displayed.

GPS information

Satellites information

On the bottom we have a sky map, which shows the current satellites constellation on the sky. The satellites which are currently used are drawn in green (if the receiver supports GLONASS, then the satellites are shown in red) and the color shows the strength of the signal. Pale green means weak signal and dark green is strong signal.

Under GPS Satellite map field is number of All satellites on the left and Used satellites on right.



Other GPS information

Whit PPS sync indicator we have information if PPS synchronization is available. This is information about the receipt of the pulse per second signal over the GPS interface (RS232 or USB), which can, if it is available, enhance synchronization to other data source.

If the receiver supports differential mode from SBAS or WAAS and differential mode is used in Hardware setup, this will be shown in Differential mode indicator. If this indicator is dark green, differential (higher accuracy) mode is used.

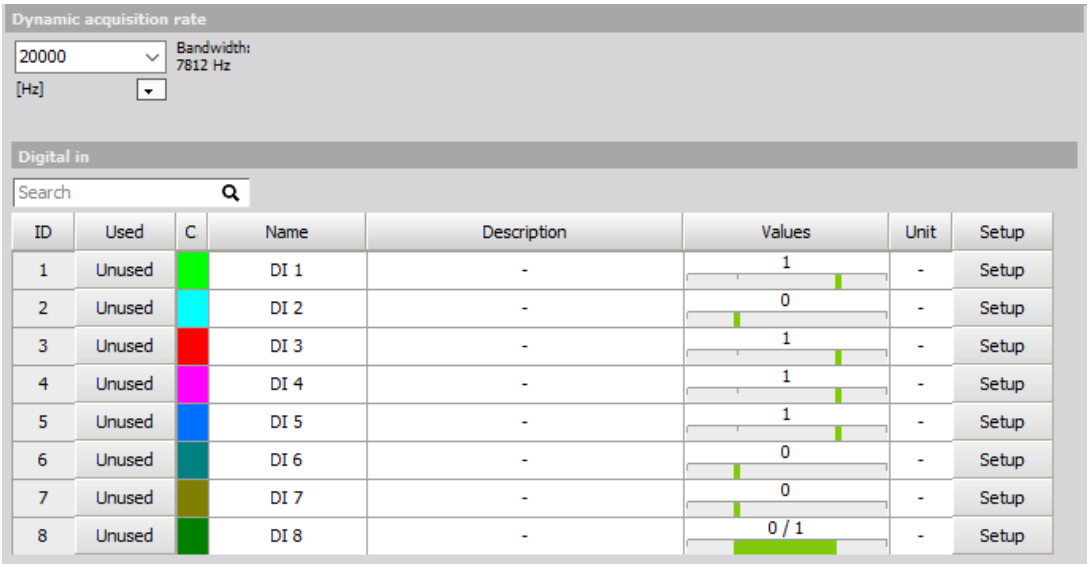
We can display GPS log information with Show NMEA log button. This function is used to see specific messages from the GPS receiver.

DIGITAL IN

The Digital input channels are used to measure only two states of the inputs: low and high. It is useful for measurement of status signals, for example relays.

For instructions on how to Add new module see -> Setup -> Add module.

When you select a Digital input tab on Dewesoft Ch.setup screen, a similar window will appear:

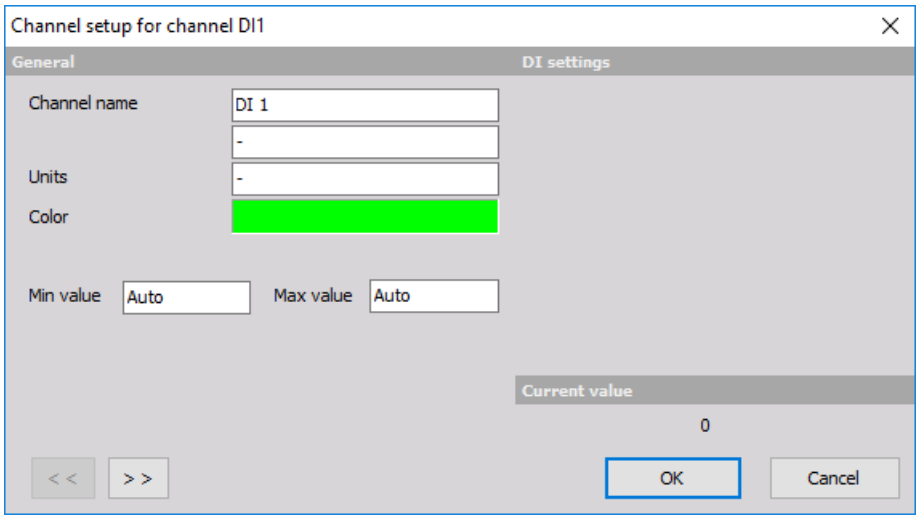


ID	Used	C	Name	Description	Values	Unit	Setup
1	Unused		DI 1	-	1	-	Setup
2	Unused		DI 2	-	0	-	Setup
3	Unused		DI 3	-	1	-	Setup
4	Unused		DI 4	-	1	-	Setup
5	Unused		DI 5	-	1	-	Setup
6	Unused		DI 6	-	0	-	Setup
7	Unused		DI 7	-	0	-	Setup
8	Unused		DI 8	-	0 / 1	-	Setup

For information about Channel grid see -> Setup -> Channel grid.

Channel setup for input channel

Press Setup to do the base Digital input settings in Channel setup window:



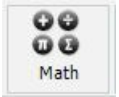
- **General parameters** - are similar to the analog input channel settings and contain the Channel name, Units, Color, Min and Max value. see -> Analog (in) Channel Setup.
- **DI settings** - We can define the signal filter (similar like counters) in this section.
- **Current value** - Displays digital value (0 or 1) of current Digital input channel.

MATH

Dewesoft has many different data acquisition sources. Taking raw data measurement is often not enough to come to the wanted result. Data processing is one of the most important features to be able to recalculate signals, filter them or make other signal processing. The following section gives an overview of basic mathematical modules.

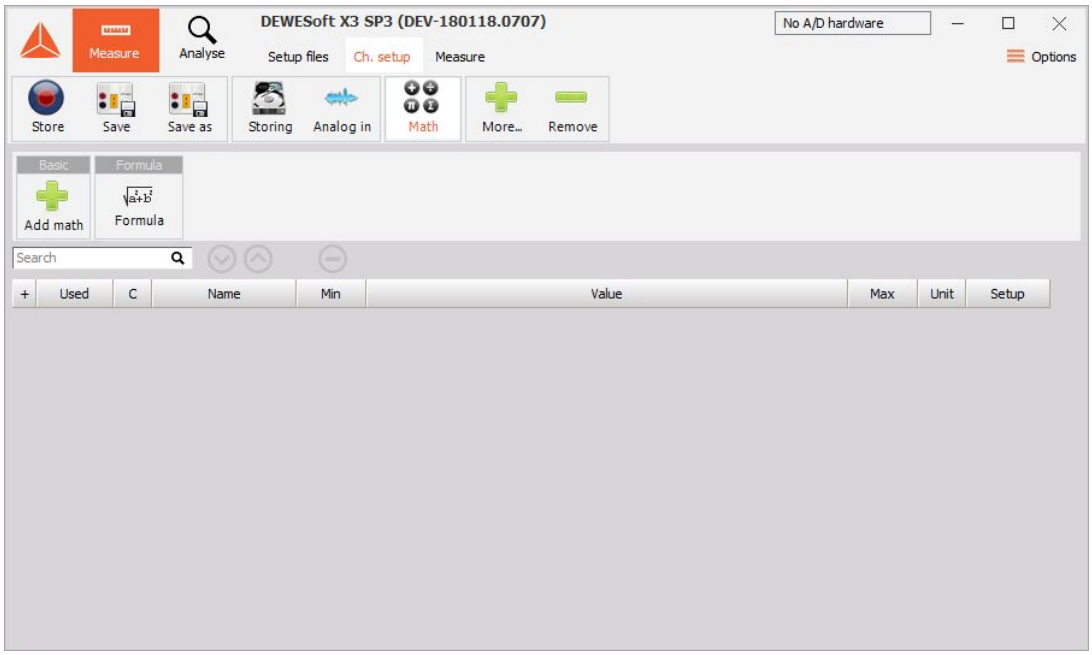
Enabling Mathematics and Filter

First you have to make sure that the mathematics module is enabled. On the Dewesoft Setup screen next to the Analog and the other available tabs, a Math tab has to be displayed. Otherwise please refer to Add module to enable the mathematics functions.



Math setup screen

When we press the math tab a setup screen is opened. Here different modules can be added.



There are also three function keys:

- Remove selected mathematical function or filter. If you want to delete the channel, simply select desired filter or function with a mouse click and then press this minus button above the channels.

- Move UP selected function within the list. If you want to move the channel upwards, simply select the desired filter or function with a mouse click and then press this button above the channels. For the first channel in the list this button is not available.

- Move DOWN selected function within the list. If you want to move the channel downwards, simply select the desired filter or function with a mouse click and then press this button above the channels. For the last channel in the list this button is not available.

Moving the modules up and down doesn't change the calculation order. Dewesoft automatically detects the references and orders the mathematical channels. So, in general, we can first select a filter on formula channel and after that define this formula channel. Dewesoft will then first calculate the formula and after that calculate the filter.

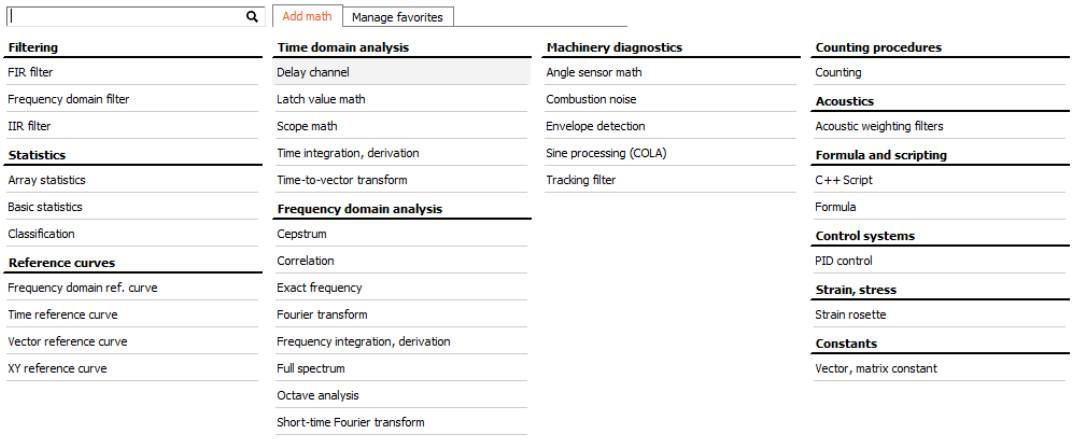
For extensive calculation, Dewesoft offers the possibility to calculate formulas and filters in separate CPU's. This is available only on multi-core processors. It will work best if the math

modules are not referencing each other, for example that we calculate the IIR filter of several input channels. Then the calculation can be nicely split into several cores. If the channels are referencing each other, this is not possible since the order of calculation is strict and well known.

Adding Math to setup screen

On the Dewesoft Math setup screen, you can add new mathematics by left clicking on Add math module.

A list of all functions appears:



To add a function to the Dewesoft Math setup screen just click on desired function from the list or search for it in the search window:

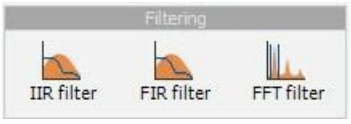
DIFFERENT MATH FUNCTIONS

Dewesoft X3 has a large array of different math and filtering options you can use to further improve the capabilities of your system.

Formula

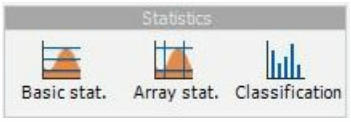
- Formula editor

Filtering



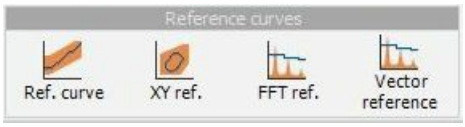
- IIR filter
- FIR filter
- Frequency domain filter

Statistics



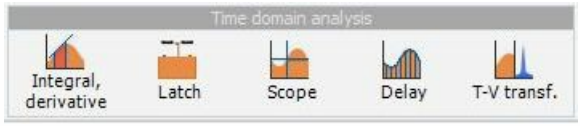
- Basic statistics
- Array statistics
- Classification

Reference curves



- Time reference curve
- XY reference
- Frequency domain reference curve
- Vector reference

Time domain analysis



- Integral derivative
- Latch value math
- Scope
- Delay channel
- Time-to-vector transform

Frequency domain analysis

- Exact frequency
- Correlation
- Cepstrum
- Full spectrum
- Frequency intergration and derivation
- FFT
- Short-time Fourier transform
- Octave analysis

Machinery diagnostics

- Combustion noise
- Angle sensor math
- Envelope detection
- Sine processing
- Tracking filter

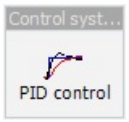
Counting procedures

- Counting

Acoustics

- Weighting filters

Control systems



- PID control

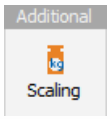
Strain, stress

- Strain rosette

Constants

- Vector matrix constant

Additional



- Scaling

NOTE: **For more details on general information how to add and setup the modules, see Setup screen and basic operation page.**

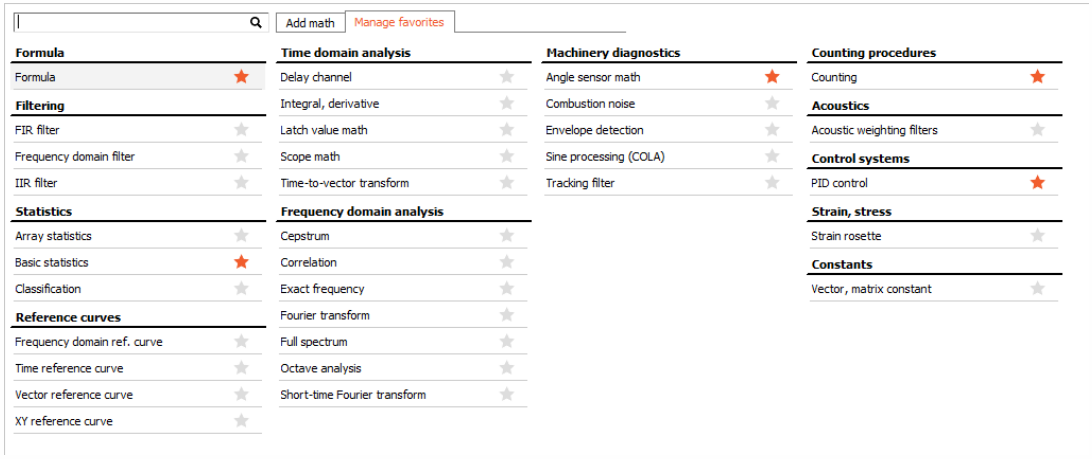
FAVORITES

You can add any Dewesoft math function to favorites bar beside the Add math button.

With this function you can access your most used math functions fast and easy.

Adding and removing

To add math function to favorites go to Math and then press Add math and Manage favorites.

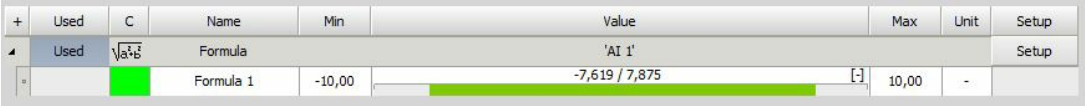


If a function is marked as favorite a star will be present beside the text.

If the star is not present the function is not in favorites bar.

SETUP SCREEN AND BASIC OPERATION

When you choose any of Dewesoft Math setup screen buttons with specific Math function (e.g. formula button), a new Math channel is displayed:



The columns in the new Math line look similar to the analog channel setup:

- Used/Unused - activate / deactivate the channels
- C- channel color selector
- Name - formula / output channel name
- Minimum- minimum default value of the graph
- Value - formula preview (symbolic description) and the calculated values
- Maximum - maximum default value of the graph
- Unit - units for your reference
- Setup - button to enter the formula in Formula setup window

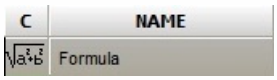
C - channel color selector

In first channel row of C column symbol of Formula is displayed and in second row of C column is C color selector for this channel (also show momentarily selected color) -> see below - Name column.

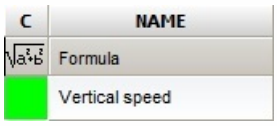
For information about Channel grid see -> Channel Setup - Channel color.

NAME

In first channel row of Name column the symbol with title Formula is displayed.

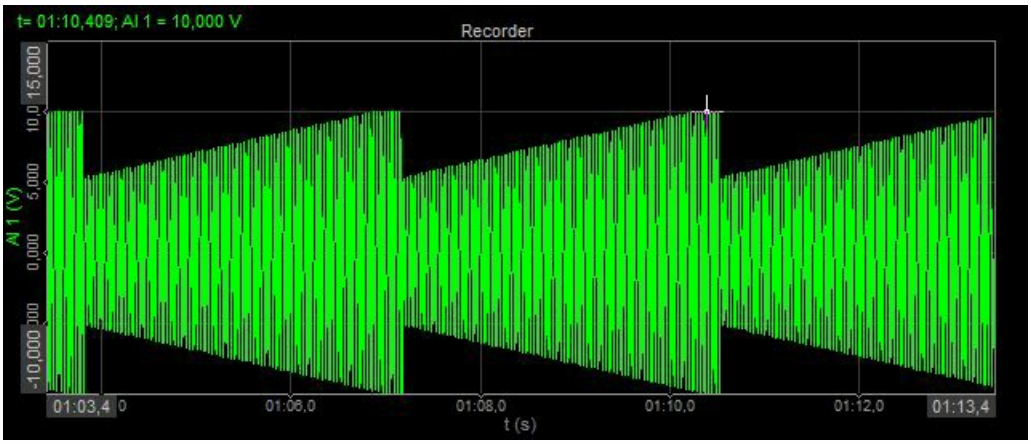


In second channel row of Name column the name of the output channel created with the formula is displayed.



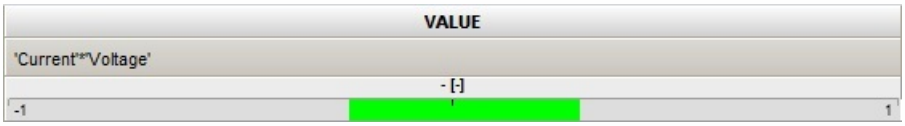
Minimum

In this section, you can change the predefined default minimum value of the graph scale.



Value

In the first channel row of Value column the description of math module is displayed. For formula a symbolic record of the formula is displayed:

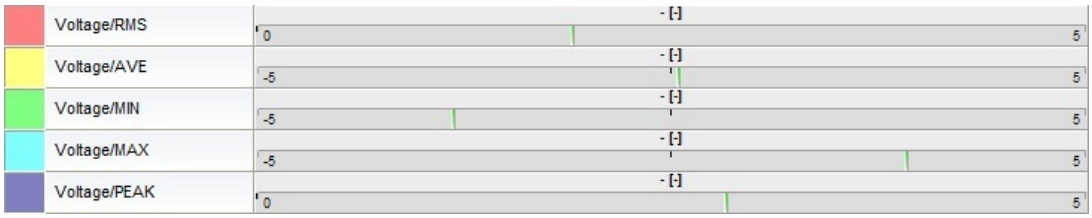


for the filter, the filter settings are shown:

In the second channel row of Value column, the live preview of the calculated value is displayed.

For the math modules with the possibility of selecting several input channels (like filters) or having several output channels for each input (like statistics), channel output section looks a little bit different - it shows one line for each output channel.

The math can also have several output channels for one input channel. The example is Statistics, where we can calculate RMS, AVE, MIN, MAX,... values for each input channel.



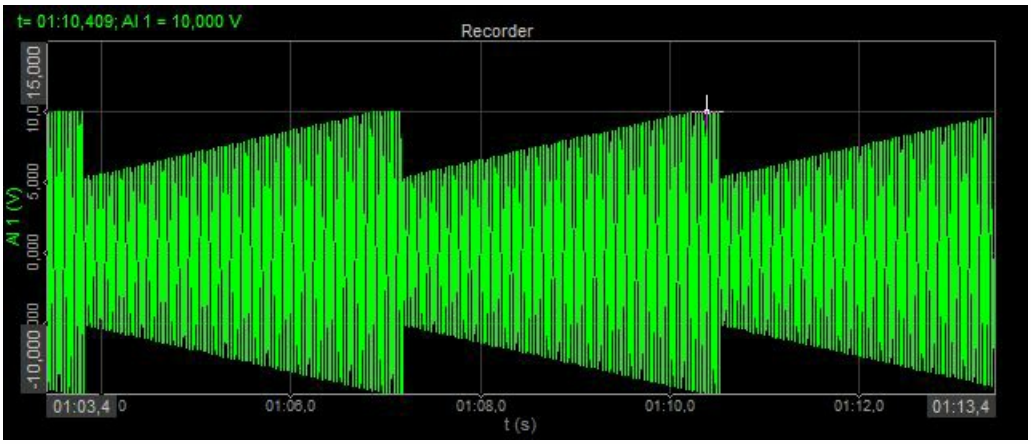
If there is an error in the math module, the error caption will appear:

There are several possible errors:

- **Channel not found** - input channel is not found (it was deleted or renamed, for example)
- **Syntax error** - the formula contains an error, for example brackets are not closed
- **Circular reference error** - the formula a references formula b while the formula b references formula a
- **Input channel error** - the input channel used in the formula has an error already (for example syntax error)

Maximum

In this section, you can change the predefined default maximum value of the graph scale.



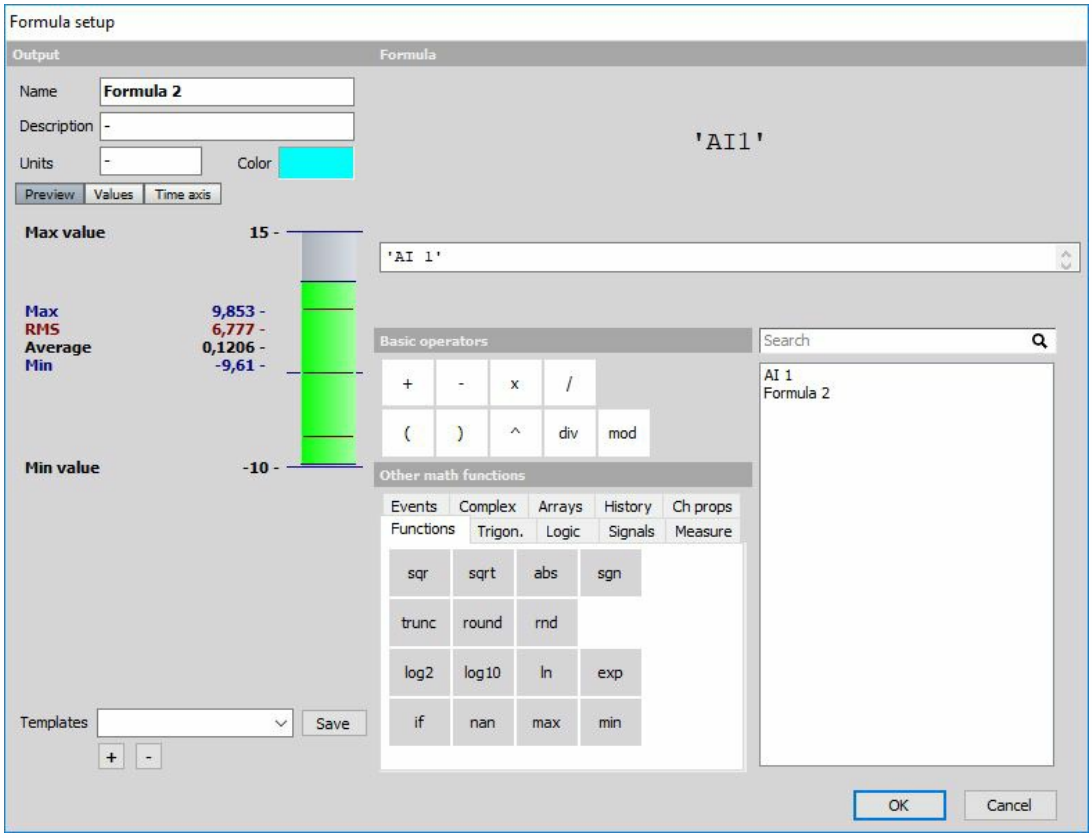
Unit

In the unit column, you can enter the unit of your choice, that will be displayed beside the calculated results.

Setup

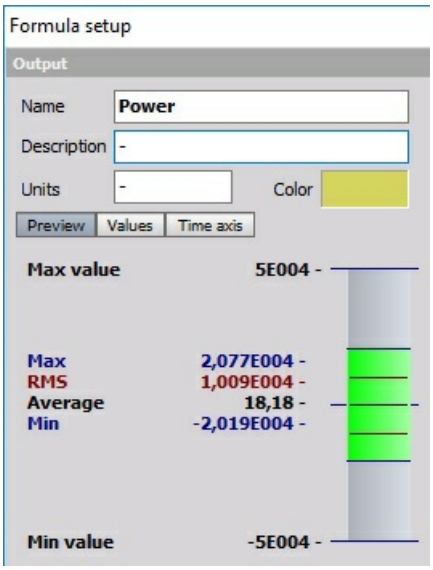
When you press the Setup button in SETUP column of Math setup screen, the window will appear depending on the selected module.

Example - Formula setup window:

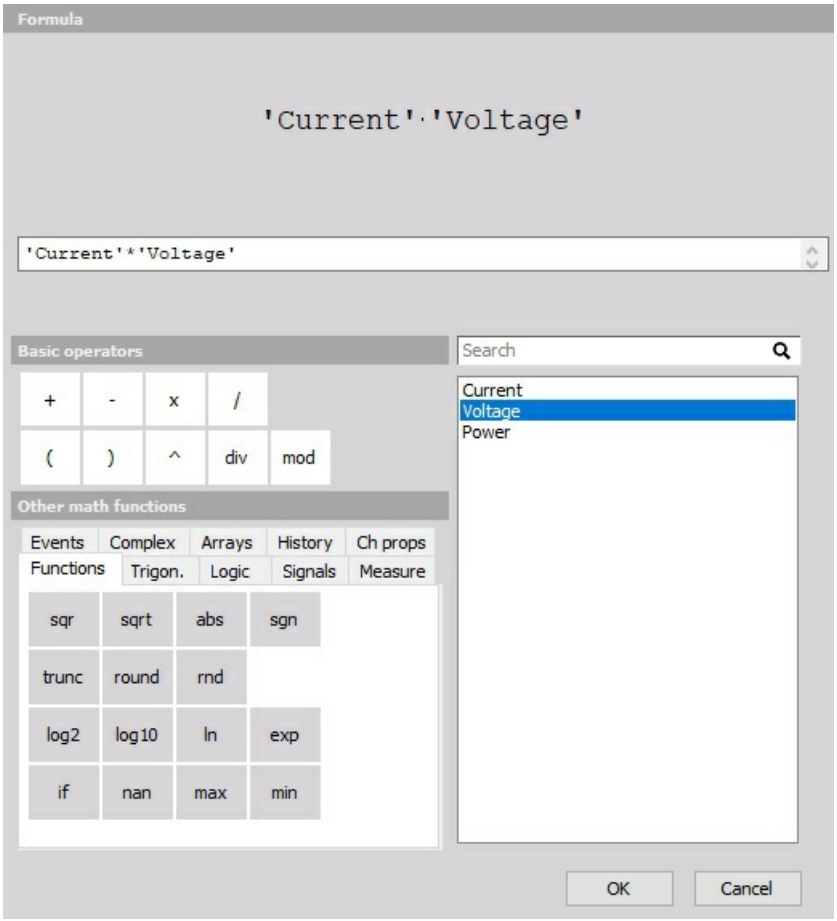


Formula Setup is comprised of two sections:

1. Output: General output settings on the top left side of the window.

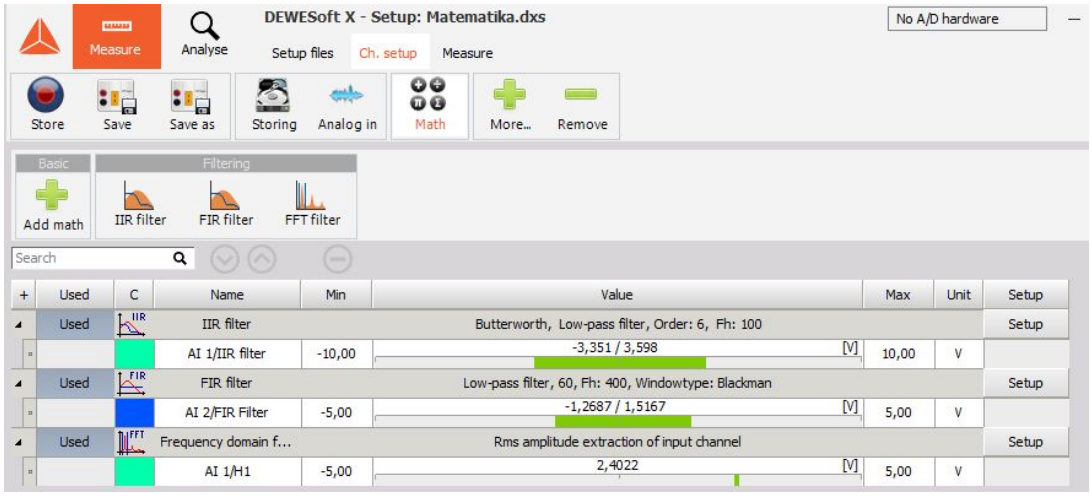


2. Formula: Formula editor on the right side of the window



FILTERING

New Filter can be added on the Dewesoft Math setup screen by selecting basic Add math button or from the favorites Filtering list (for details see -> Add new Filter) and new Filter line is displayed:



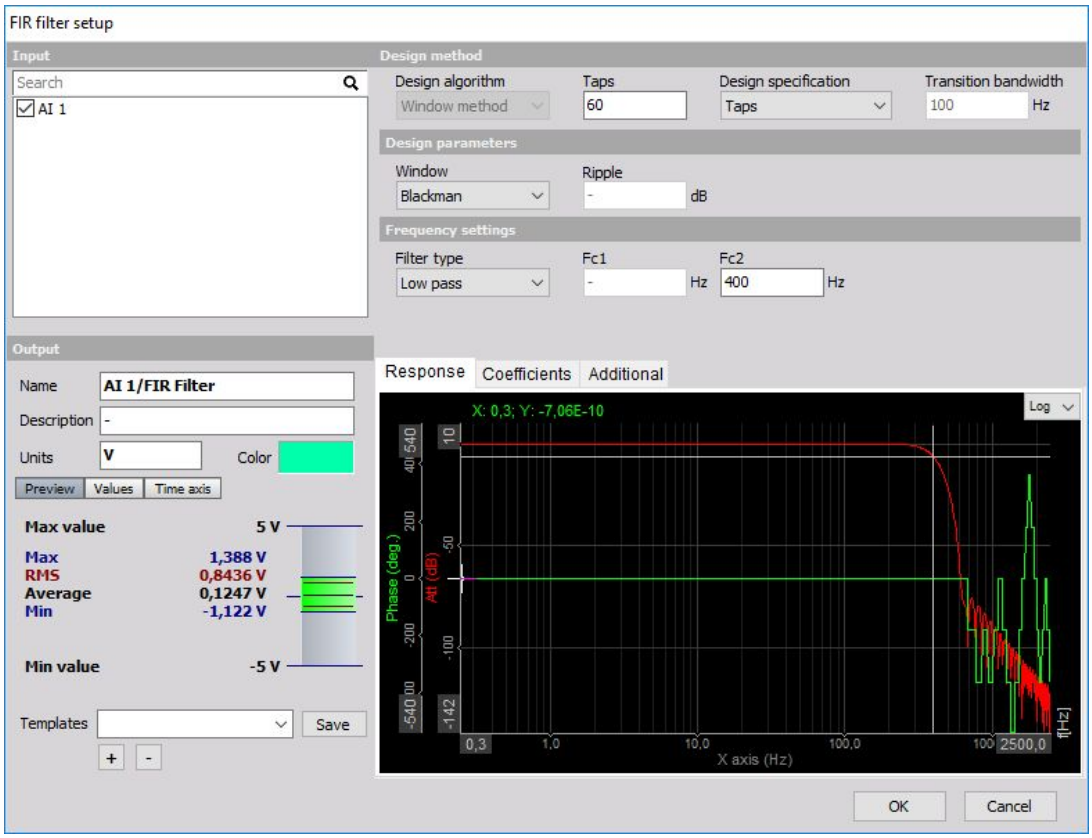
For detailed information about basic settings see -> Setup screen and basic operation

When you press the Setup button on activated new Filter line, the Filter setup window will open. This window depends on selected filter type and his name appears on the title line of the window:

- IIR Filter setup
- FIR Filter setup
- Frequency domain filter setup

FIR FILTER SETUP

When you press the Setup button on newly activated FIR Filter line, the following FIR filter setup window will open:



The filter supports multiple input channels.

For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

FIR stands for finite impulse response. In theory, it means that the response to the impulse will be zero after some time (exactly after the samples will equal to filter order).

Another nice property of the filters is that basically phase response is linear. The phase shift in time is half of the number of samples if the filter is calculated for the samples in the past.

Since Dewesoft has calculation delay, we can use a trick to compensate the filter delay and have absolutely no phase shift in pass as well as in the transition band of the filter. This is a major benefit compared to the IIR filter where we always have a phase shift. The drawback of FIR filters is that they will use more CPU power compared to IIR.

We will make a comparison between these two types a bit later; now let's take a look at basic properties how to set the filter.

FIR Filter settings

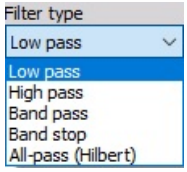
For FIR Filter you can set:

- Design method
- Design algorithm
- Window method (for now)
- Design specification
- Taps / Order
- Transition bandwidth
- Design parameters
- Window type
- Blackman
- Rectangle
- Hamming
- Hanning
- Kaiser - Ripple
- Flat top
- Frequency settings
- Filter type
- Low pass
- High pass
- Band pass
- Band stop
- All-pass (Hilbert)
- Additional
- Scale

You can see the effect of this setting directly on Response curve / Coefficients preview for Filter type: Low pass, High pass, Band pass, Band stop and different Window type:

Filter type

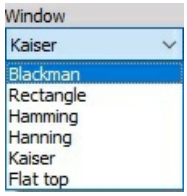
You can select Filter type from list between:



- **Low pass** - Low pass filters cut the high frequencies of the signals.
- **High pass** -High pass filters DC and low frequencies.
- **Band pass** - Band pass filter filters high and low frequencies, so there is only one band of values left.
- **Band stop** - Band stop filter filters only one section of frequencies.
- **All-pass (Hilbert)** - An all-pass filter is a signal processing filter that passes all frequencies equally in gain, but changes the phase relationship among various frequencies

Window type

You can select Window type from the list. The window defines the behavior of the filter in the transition and the stop band (the height of the sidebands and the width of the main band). For common usage, Blackman window is quite good choice, because the side bands are extremely low.



Taps / Order

In this field, you can enter Taps. The taps of the filter define the number of coefficients of the filter and that will directly affect the slope of the transition band. The filter taps are not directly comparable with FIR filter.

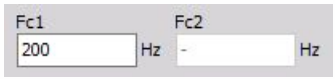
Kaiser window type - Ripple

When a Kaiser Window type is selected, new Ripple field appears on the right side of the Window type field. In this filed you can enter ripple value in dB. It tells the maximum allowed pass band ripple of the filter. The more this value is, the bigger will be non-linearity in the pass band, but the filter will be steeper.

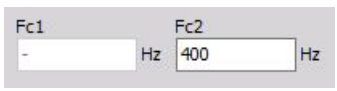
Cut-off frequency

The filter cutoff frequency defines the -6 dB point (half amplitude) of the filter. You can enter the Cut-off frequency in the field:

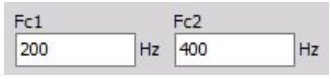
- **Fc1 - Low frequency** (You can enter Fc1 for Low pass, Band pass and Band stop filter).



- **Fc2 - High frequency** (You can enter Fc2 for High pass, Band pass and Band stop filter).



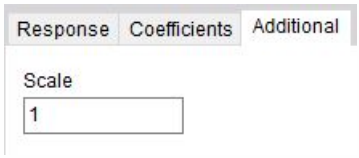
- **Both High and Low frequency** (You can enter Fc1 and Fc2 for Band pass and Band stop filter).



Fc1 value must be always lower than Fc2. These values are limited by filter stability. In Dewesoft the filters are calculated in sections, which enable the ratio between cutoff and sample frequency in a range of 1 to 100000. So we are able to calculate 1 Hz high pass filter with 100 kHz sampling rate.

Scale

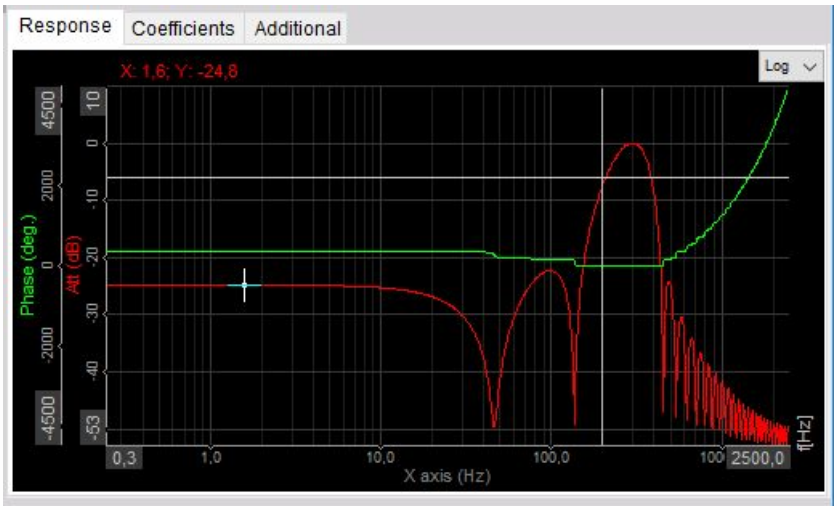
For filters, you can enter also Scale. Scale factor means the final multiplication factor before the value is written to the output channel. It helps us to change the unit, for example. A good example of using the Scale is shown in the Integration section.



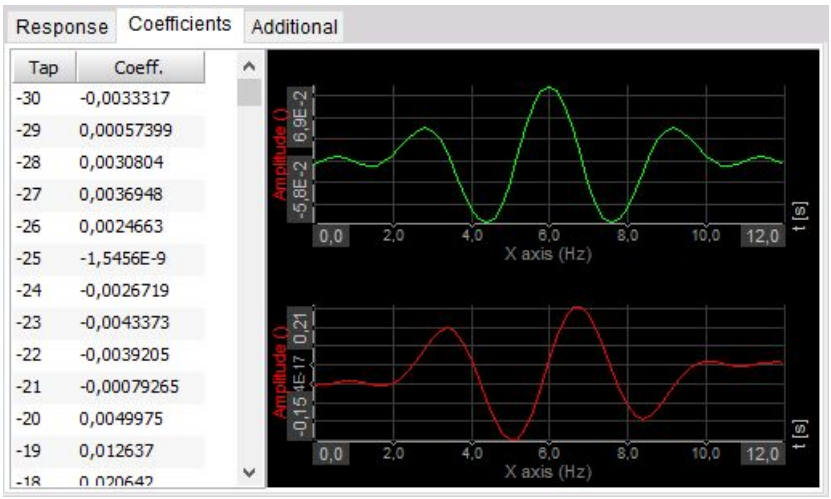
Response curve / Coefficients preview

You can choose between Response curve preview and Coefficients display.

The red response curve shows the amplitude damping of the filter. The amplification ratio is expressed in dB (similar to IIR filter). The green curve shows the phase delay. In the pass band as well as in the transition band the phase delay is always zero and in the stop band the phase angle is not even important because of high damping ratio.



The other display is the display of coefficients. The upper graph and the left table shows the filter coefficients with which the raw data is convoluted. The lower graph shows the response of the filter to the step response.



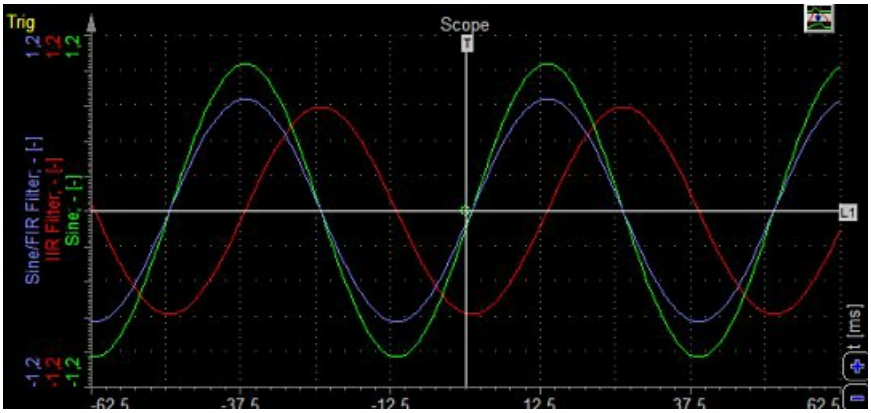
On Response curve preview you can choose between Logarithmic and Linear display, you can also edit coordinates value and auto scale Yaxis.

Filter comparison

Let's look at the difference of the FIR filter compared to the standard IIR filter. Let's take a very simple 20 Hz second order filter (at 1 kHz sampling rate).

The IIR filter is calculated with 6 coefficients, while similar FIR filter is calculated with 40 coefficients for the same damping. Therefore the FIR filter is more CPU demanding for the same performance.

Another fact is while we can get ratios of cutoff frequency to sample rate of 1/100000 and more, we can achieve only limited results with FIR filter. The ratio increases with the higher number of coefficients.



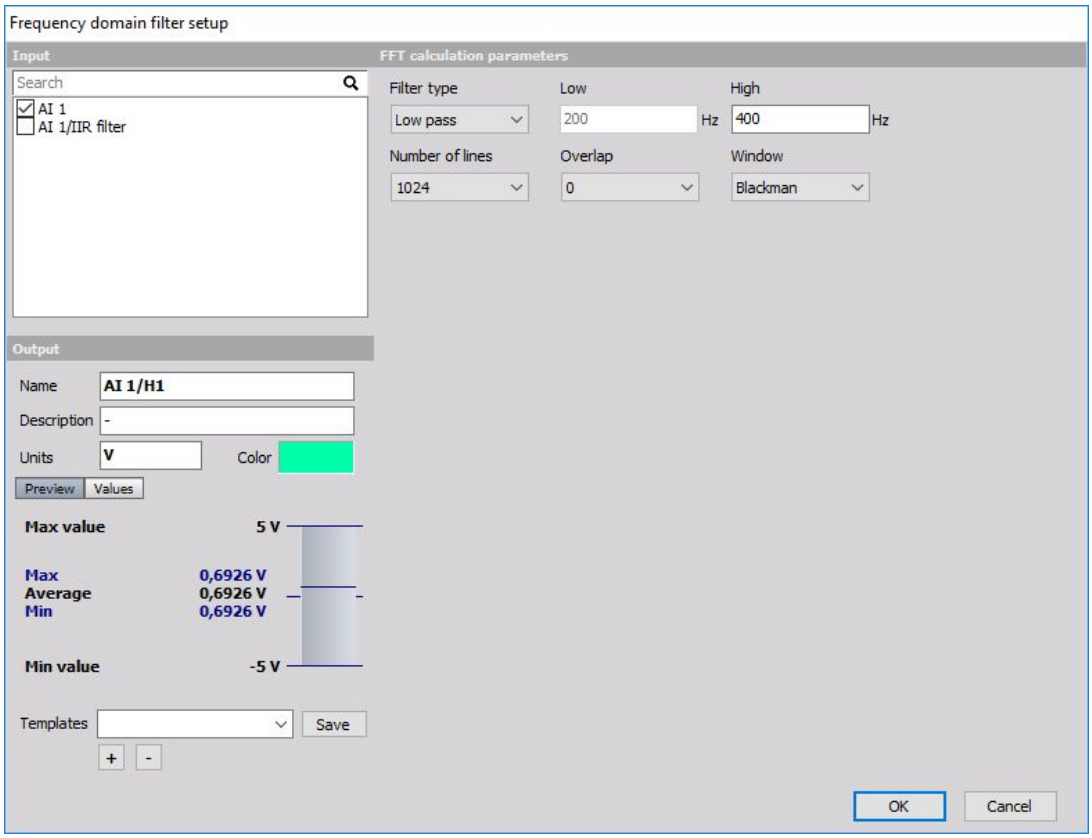
Enough of the downsides, let's look at the response graph at 20 Hz (exactly at the limit). The green curve is the original sine wave while the red one is calculated with IIR filter. We can clearly see the phase delay of the output.

The blue curve is the response of the FIR filter which has absolutely no phase shift. For lots of applications, it is very important that the signals are not delayed and there the use of FIR filters is

very advantageous.

FREQUENCY DOMAIN FILTER

When you press the Setup button on new activated FFT Filter line, the following FFT filter setup window will open:



The filter supports multiple input channels.

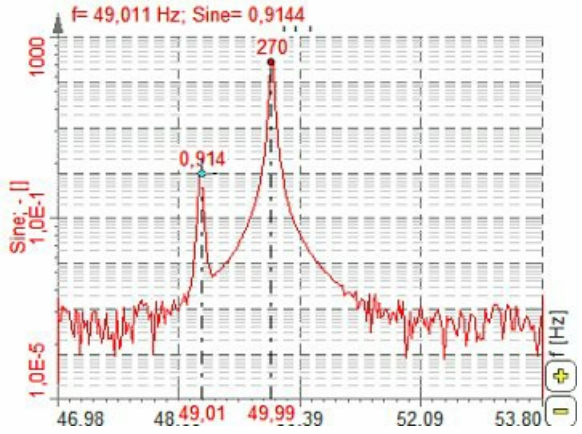
For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

FFT filter description

FFT filter is quite different to other types of filters. While IIR and FIR filters are time domain filters, FFT filter calculates the spectrum of the signal with specific number of lines and overlap and then extracts the RMS value of certain range of this signal. Therefore the result is not the full curve, but only one value per frequency spectrum.

The usage of this filter is to extract low peaks of signals where there are big harmonics nearby where it wouldn't be possible to choose IIR filter which would extract this low amplitude.

The example below shows the electromotive winding failure which can be seen as low values at the rotation frequency where the line frequency is very high:



We can design our own filter -> see -> Custom FFT Filter.

FFT filter parameters

To set the FFT filter, it is recommended to observe the signal in the FFT display and choose the right Number of lines and Window which fits the best and then set the filter parameters.

For FFT Filter you can set:

FFT calculation parameters

- Number of lines
- Overlap
- Window

Amplitude extraction parameters

- Frequency source
- Center frequency

only for fixed frequency source

- Delta frequency
- Frequency channel

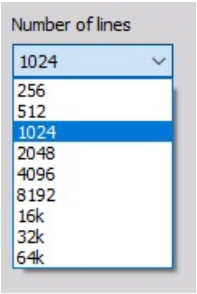
only for tracking frequency source

- Number of harmonics

only for Tracking Frequency source

Number of lines

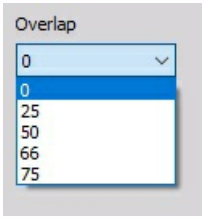
You can select Number of lines from the list.



This defines the resolution of the filter as well as the number of points in the calculation. The resolution needs to be high enough that the wanted harmonic can be clearly extracted, but not too high to have a higher result update.

Overlap

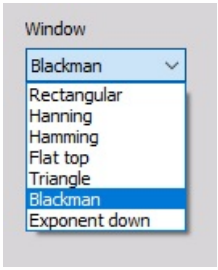
You can select Overlap from the list.



Overlap defines (same for FFT averaging) how many 'old' data is taken for next calculation. This increases the result update rate with the same number of lines.

Window type

You can select Window type from the list.

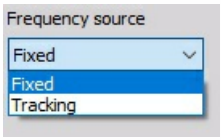


The window defines the behavior of the filter in the transition and the stop band (the height of the sidebands and the width of the main band).

Frequency source

You can select Frequency source from the list if Tracking filter type is enabled:

- **Fixed** - Fixed frequency will always take fixed value for center frequency.
- **Tracking** - Tracking frequency means that the center frequency will depend on a second input channel (for example rotation frequency).



Delta frequency

In this field, you can enter Delta frequency in Hz. This value depends on the wanted frequency band. It also depends on the window and number of lines (line resolution). In our example we would choose 0.5 Hz since we don't want that 50 Hz value will appear in the result.

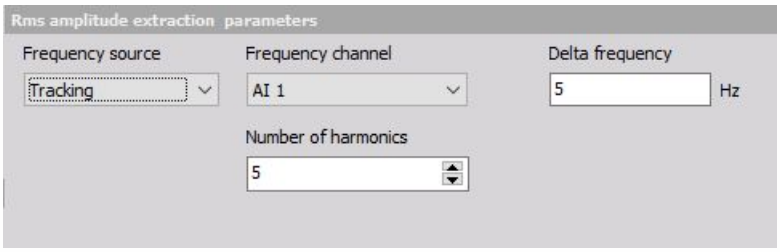
Center frequency

If we choose Fixed Frequency source, we need to enter Center frequency in Hz. The center frequency is the middle value of frequency for value extraction. In our example above we would

take 49 Hz as the center frequency.

Filter settings for Tracking frequency source

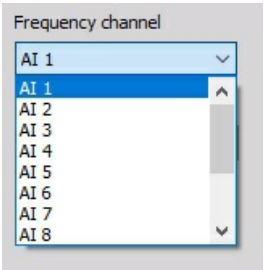
If we choose Tracking Frequency source, we need to enter the Frequency channel and Number of harmonics instead of Center frequency:



Frequency channel

Frequency channel is the channel with the current frequency which needs to be extracted. The unit of this channel must be in Hz.

Frequency channel can be selected from the list:

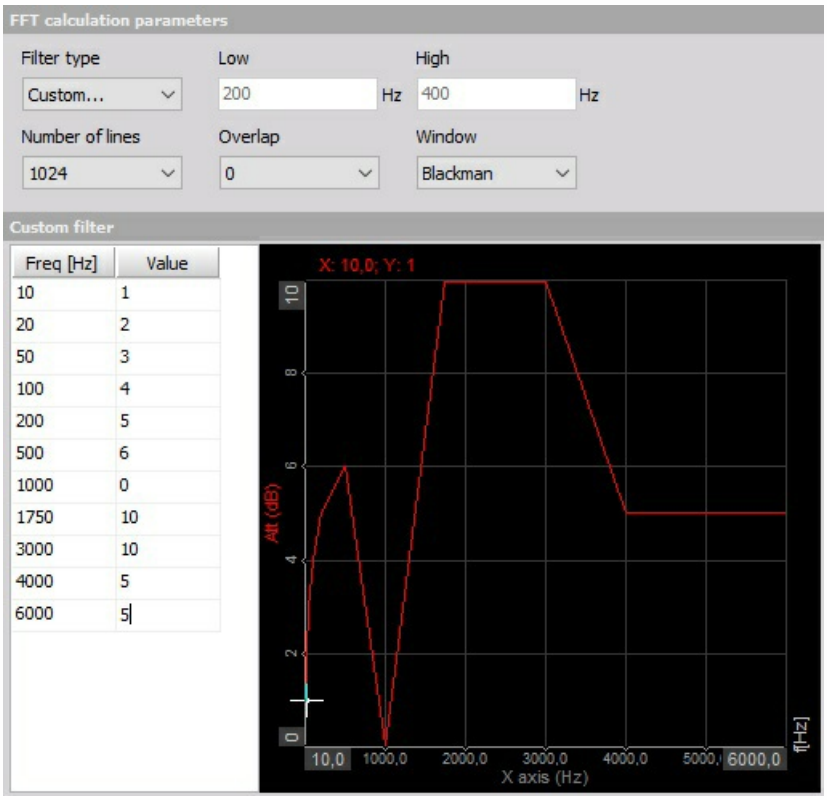


Number of harmonics

Number of harmonics describes how many harmonics need to be extracted from the spectrum. If we enter a value of 5, there will be 5 channels created for each input channel. The first channel will have the center frequency as the frequency channel; second will have twice the frequency of the input and so on.

Custom FFT filter

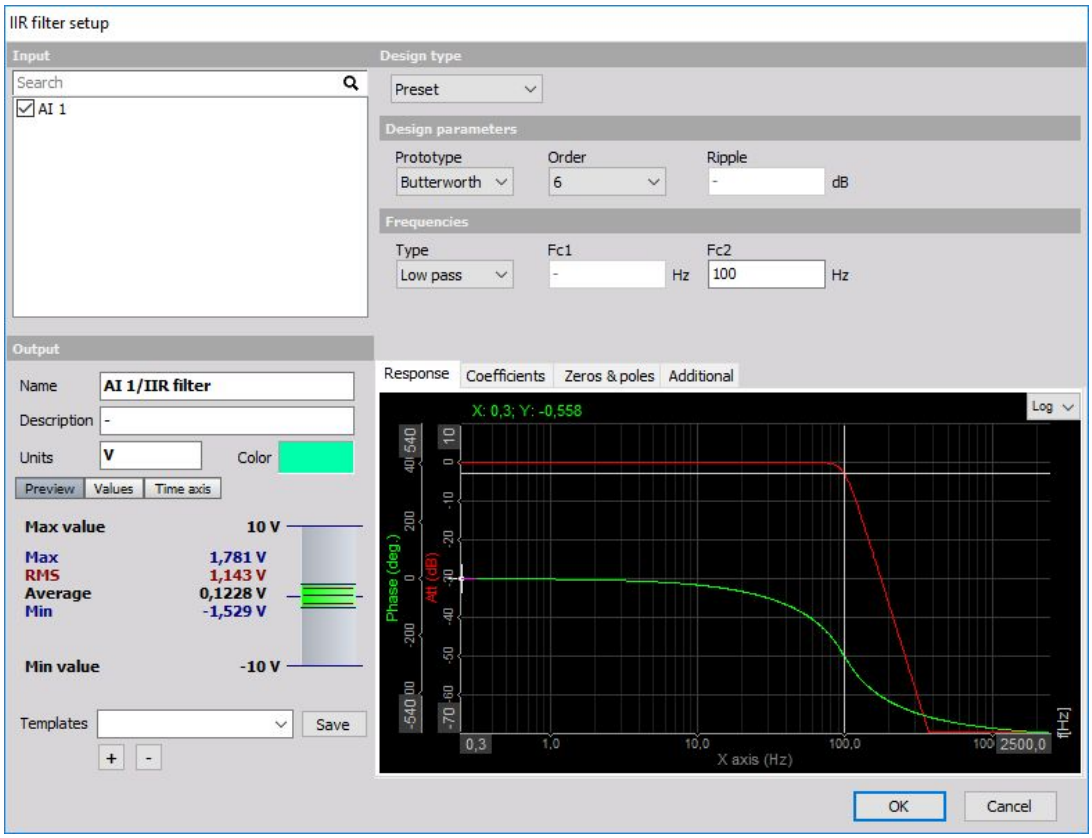
If Custom filter is selected from the Filter type, then we can design our own filter. With this option, we can create any type of filter curve in the frequency domain and calculate RMS value. Sometimes it is not easy to define filter characteristics in the time domain, but we have it defined in the frequency domain. Custom FFT filter is perfect for such case.



If the frequency source is external, we can define the channel where the frequency is defined and the filter will change the characteristic to always filter correctly like in the time domain. This is especially useful for example for CA noise calculation on the external clock.

IIR FILTER SETUP

When you press the Setup button on new activated IIR Filter line, the following IIR Filter setup window will open:



The filter supports multiple input channels.

For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

IIR is the abbreviation for infinite impulse response. It means that the response to the impulse will be non zero over an infinite length of time. The calculation behind is that the new filtered value is calculated from the current and previous input values as well as previous output values.

That's the theory. In practice, we can say that the IIR filter is a direct equivalent to the RC type's analog filters.

For example we can achieve the exact theoretical low pass, high pass, sound weighting and other standard filters which were achieved by analog circuits in classic 'old school' instruments.

IIR Filter settings

Click on the icons at the top to select between:

- Preset
- Butterworth
- Low pass
- Hi pass
- Band pass
- Band stop
- Chebyshev
- Bessel
- Manual
- Manual entry of grid properties

STANDARD FILTER

Standard filter basically allows high, low, band pass and band reject filters.

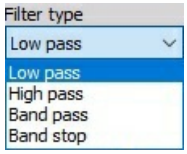
With Filter option on IIR Filter settings section you can set:

- Type and Prototype
- Order
- Cut-off Frequency
- Ripple
- Scale

You can see the effect of these settings directly on Response curve /Zeros & Poles display for Filter Type: Low pass, High pass, Band pass, Band stop and different Prototype.

Type of Filter

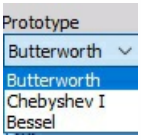
You can select Filter type from list between:



- **Low pass** - Low pass filters cut the high frequencies of the signals.
- **High pass** - High pass filters DC and low frequencies.
- **Band pass** - Band pass filter filters high and low frequencies, so there is only one band of values left.
- **Band stop** - Band stop filter filters only one section of frequencies.

Prototype of Filter

You can select Prototype of Filter from list between:

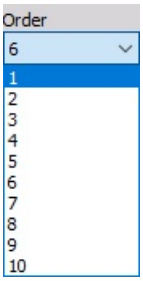


- **Butterworth** - Butterworth is without the ripple and maintains the shape with higher orders. Roll-off is defined with $\left(-20\frac{dB}{decade} \times order\right)$. It is also known as maximally flat magnitude, suggesting that the filter response is really flat in the pass band.
- **Chebyshev** - Sometimes the selection of the filters is defined by the application, but in general, the Chebyshev has a highest roll-off of all three, but has a ripple in the pass band and doesn't maintain the shape with higher orders.
- **Bessel** - Bessel filter is the filter with maximally linear phase response. The roll off however is the least step of all three filter types.

Order

You can select Order from the list. The order of the filter defines the steepness of the filter. For the Butterworth the roll-off is $(-20\frac{dB}{decade} \times order)$, so for the sixth order, the roll-off would be $-120\frac{dB}{decade}$. That would mean if the amplitude at 100 Hz (already in the stop band) is 1, the amplitude at 1000 Hz would be $1E^{-120/20} = 1E-6$.

The highest as the order is, more calculation power will be needed to calculate the filter. We need 6 multiplications for every two orders of the filter.

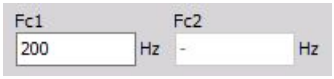


Cut-off frequency

The filter cutoff frequency defines the -6 dB point (half amplitude) of the filter. You can enter Cut-off frequency in field:

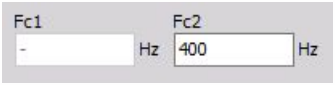
- **Fc1 - Low frequency**

You can enter Fc1 for Low pass, Band pass and Band stop filter.



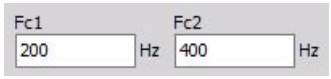
- **Fc2 - High frequency**

You can enter Fc2 for High pass, Band pass and Band stop filter.



- **Both High and Low frequency**

You can enter Fc1 and Fc2 for Band pass and Band stop filter.



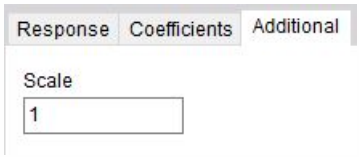
Fc1 value must be always lower than Fc2. These values are limited by filter stability. In Dewesoft the filters are calculated in sections, which enable the ratio between cutoff and sample frequency in a range of 1 to 100000. So we are able to calculate 1 Hz high pass filter with 100 kHz sampling rate.

Ripple

You can set also Ripple. Ripple is the maximum amplitude error of the filter in the pass band in dB.

NOTE: **This field appears only for Chebyshev filter Prototype.**

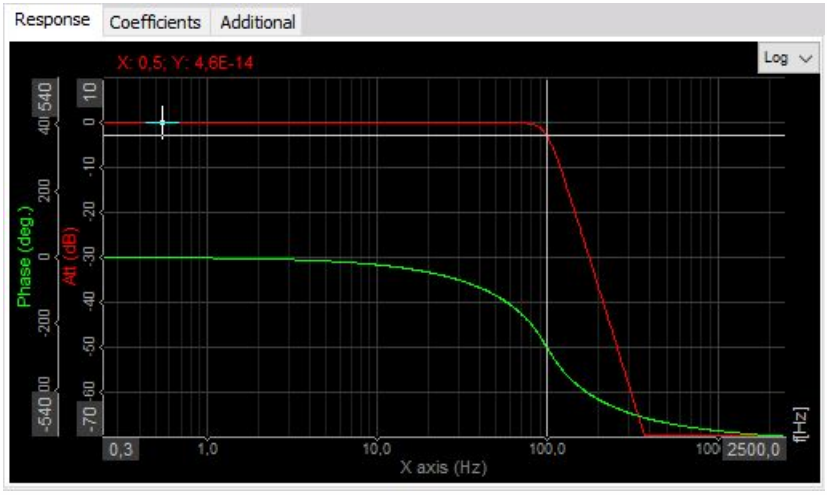
Scale



For filters, you can enter also Scale. Scale factor means the final multiplication factor before the value is written to the output channel. It helps us to change the unit, for example. A good example of using the Scale is shown in the Integration section.

Response curve / Zeros & Poles preview

On the lower side, we see some useful information of the chosen filter. First is the response curve.

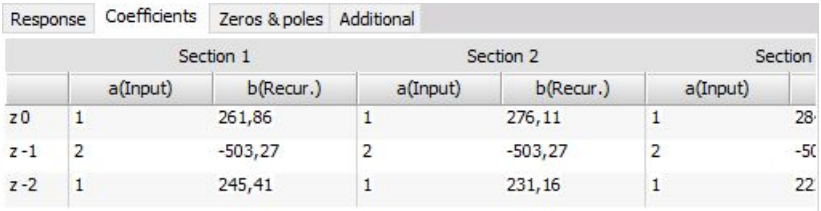


The red graph shows the amplification/attenuation of the filter in dB related to the frequency. To refresh the memory, dB scaling is calculated with equation $a[dB] = 20 \times log_{10}(A)$, so the attenuation ratio is calculated with $A = 10^{\frac{a}{20}}$.

If we read out the value of -34 dB as attenuation, the ratio between input at output at that frequency will be A=10^(-34/20)=0,02. So if the input is 1 V sine wave, the output will be 0,02 V sine wave.

The phase shows the delay of the signal in degrees.

Lower table shows the coefficients with which the filters will be calculated. The filter is split into several sections for increased stability, so the result from the first section is taken to the next section and so on. These coefficients can be also copy / pasted with a right mouse click on the table to be used from/in other calculation programs.



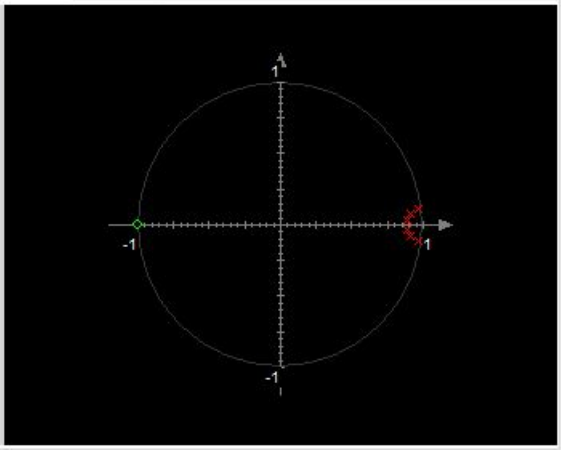
	Section 1		Section 2		Section
	a(Input)	b(Recur.)	a(Input)	b(Recur.)	a(Input)
z 0	1	261,86	1	276,11	1
z -1	2	-503,27	2	-503,27	2
z -2	1	245,41	1	231,16	1

On Response curve preview you can choose between Logarithmic and Linear display, you can also edit coordinates value and auto scale Y axis.

Zeroes & poles diagram shows the position of filter zeroes and filter poles and can suggest the stability of the filter.

Response | Coefficients | Zeros & poles | Additional

Zeros	Poles
-1+0i	0.96094+0.11726i
-1+0i	0.91135+0.081409i
-1+0i	0.88498+0.028936i
-1+0i	0.88498-0.028936i
-1+0i	0.91135-0.081409i
-1+0i	0.96094-0.11726i



CUSTOM FILTER - MANUAL

The Custom defined filter setup requires setting:

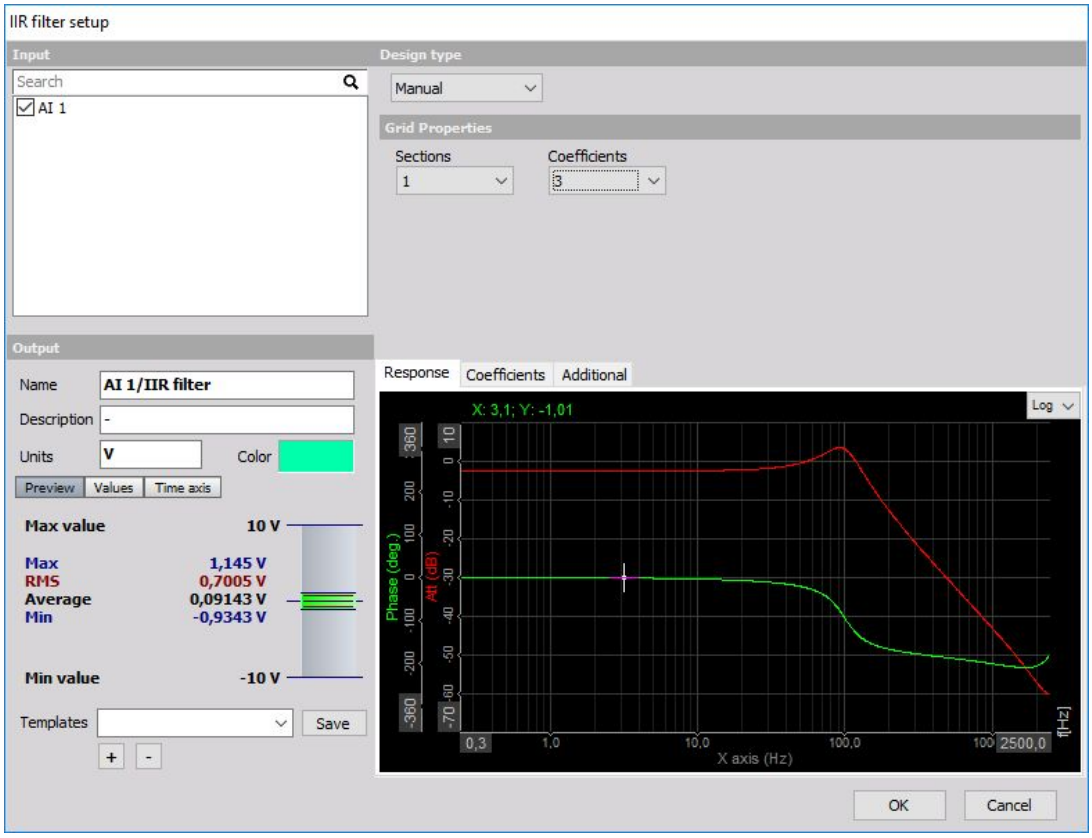
- number of Sections andCoefficients
- Scale

For information about this setting see -> IIR Filter - standard-> Scale.

Individual filter coefficients value It is possible to import custom filters - coefficients from Matlab, see -> Custom filter import from Matlab.

Usually, FIR filters consist from a single section. Multiple sections can be understood actually as several IIR filters combined into one (for example low pass and high pass). It also improves the stability of the filter since the coefficients are smaller. However, it is most common to use only one section for custom filters.

We define the number of coefficients per section which are the number of rows in the table. This basically defines the filter order.



The last things to define are the filter coefficients. Enter **a(input)** and **b(rekur.)** values in the z-plane and press the Update button to change the filter settings. We can also copy/paste the coefficients from the clipboard by right-clicking and choosing 'Copy to clipboard' or 'Paste from clipboard' menu item.

The big question is now: how to define the coefficients. The answer to this question lies in the

knowledge of filter design in s-plane and converting the filter to z-plane. We will try to shortly remember this procedure.

Usually, the filters are defined in the s-plane. Let's take a simple example of the general formula for second order filter:

1. $$H(s) = \frac{g_0 + g_1 s + s^2}{h_0 + h_1 s + h_2 s^2}$$

To get the filter coefficients in the z-plane (time domain coefficients) we need to use the bilinear transformation:

2. $$s = 2f_s \frac{1 + z^{-1}}{1 - z^{-1}}$$

where f_s is the sampling frequency. The upper equation reveals the important fact of filters defined in the z plane - they work only for one sample rate. Therefore if we need the filters at different sampling rates, the coefficients need to be recalculated.

If we substitute the s in **equation (1)** with the formula in **equation (2)**, we get for the upper part:

3. $$g(1 + z^{-1})^2 + 2g_1 f_s (1 - z^{-1})(1 + z^{-1}) + 4g_2 {f_s}^2 (1 - z^{-1})^2 =$$
$$(g_0 + 2g_1 f_s + 4g_2 {f_s}^2) + (2g_0 - 8g_2 {f_s}^2) z^{-1} + (g_0 - 2g_1 f_s + 4g_2 {f_s}^2) z^{-2}$$

The first third of the equation is valid for z^0 coefficient, second for z^{-1} and the third one for z^{-2} in the table above. The upper part of the equation (with g coefficients) is valid for input part while the lower part with h coefficients is valid for recursive part of the equation.

If we need higher order filter, we need to make equation similar to **(3)** with the larger number of the coefficients. The result will have also z^{-3} factor.

Let's now make a simple example for second order Butterworth filter. It has the following prototype in the s-plane:

4. $$H(s) = 1 / \left(1 + \sqrt{2}\frac{s}{\omega_c} + \left(\frac{s}{\omega_c}\right)^2\right)$$

where the ω_c is the cutoff frequency in rd/s. We have to adapt the cutoff frequency to the sample rate with pre warping:

5. $$\omega_c = 2f_s \cdot \tan\left(\pi \frac{f_c}{f_s}\right)$$

If we write out the factors for this filter:

6. $$g_0 = 1; \quad g_1 = 0; \quad g_2 = 0;$$
$$h_0 = 1; \quad h_1 = \frac{\sqrt{2}}{f_c}; \quad h_2 = \left(\frac{1}{f_c}\right)^2$$

Now let's make the following filter:

- cutoff frequency - $f_c = 100\ \mathrm{Hz}$
- sampling rate - $f_s = 1000\ \mathrm{Hz}$

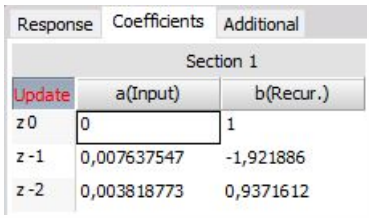
First we do the pre warping:

$$f_{cp} = 2 \cdot 1000 \cdot \tan\left(\pi \frac{100}{1000}\right) = 649{,}8\ \frac{\mathrm{rad}}{\mathrm{s}}$$

And now we need to calculate the coefficients for direct and recursive part of the filter with substituting factors (6) in equation (3):

$$a(z^{-2}) = 1 - 0 + 0 = 1 \quad b(z^{-2}) = 1 - 2 \cdot \sqrt{2} \cdot \tfrac{1000}{649{,}8} + 4 \cdot \tfrac{10002}{649{,}82} = 6{,}12$$
$$a(z^{-1}) = 2 \cdot 1 - 0 = 2 \quad b(z^{-1}) = 2 \cdot 1 - 8 \cdot \tfrac{10002}{649{,}82} = -16{,}944$$
$$a(z^{0}) = 1 + 0 + 0 = 1 \quad b(z^{0}) = 1 + 2 \cdot \sqrt{2} \cdot \tfrac{1000}{649{,}8} + 4 \cdot \tfrac{10002}{649{,}82} = 14{,}825$$

Finally, we set the number of coefficients to 3, number of sections to 1 and enter calculated 6 values in the table and press Update. All entered values are red color and button Update also flash until Update is pressed.



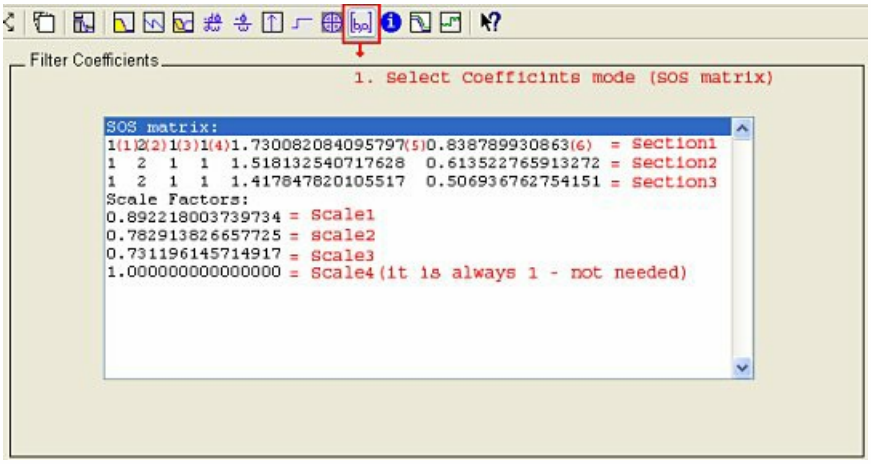
Remember, this is valid only for a sampling rate of 1000 Hz. For others, we need to recalculate the fcp and coefficients.

To make second-order Butterworth filter is much easier with Dewesoft standard filters, but if we need a specific filter, it is necessary to design it 'by hand'.

The next chapter describes how to import custom filters fromMatlab.

CUSTOM FILTER IMPORT FROM MATLAB

It is possible to import custom filter from Matlab (registered trademark of MathWorks company). Design IIR filter in Matlab and then click on the filter coefficients button. Filter coefficients should appear in SOS matrix form. That is how they are presented in Dewesoft.



In Dewesoft you can't enter the scale factors so you just have to include them in the filter. One section in Dewesoft equals one SOS section in Matlab. All you have to do is scale it the right way. First three coefficients in Matlab are input and are calculated in that way that you multiply the coefficient by the corresponding scale factor. The second three coefficients are recursive and all you need to do is just to copy them from Matlab.

These are calculation formulas for specific section i:

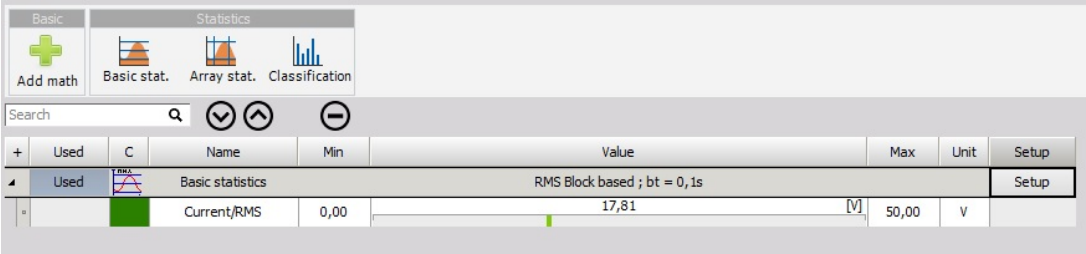
	a(input)	b(recur.)
z0	Scale i * Section i(1)	Section i(4)
z-1	Scale i * Section i(2)	Section i(5)
z-2	Scale i * Section i(3)	Section i(6)

The coefficients for our example are calculatedbelow:

Updat	Section 1		Section 2		Section 3	
	a(input)	b(recur.)	a(input)	b(recur.)	a(input)	b(recur.)
z0	0.892218	1	0.7829138	1	0.7311961	1
z-1	1.784436	1.730082	1.565828	1.518132	1.462392	1.417848
z-2	0.892218	0.8387899	0.7829138	0.6135227	0.7311961	0.5069367

STATISTICS

Available Statistics on Math setup screen are: Basic Statistics, Array Statistics and Classification. New Statistics line can be added by selecting:



When you press the Setup button on activated new Statistics line, the Statistics setup window will open. This window depends on the selected Statistics type and his name appears on the title line of the window:

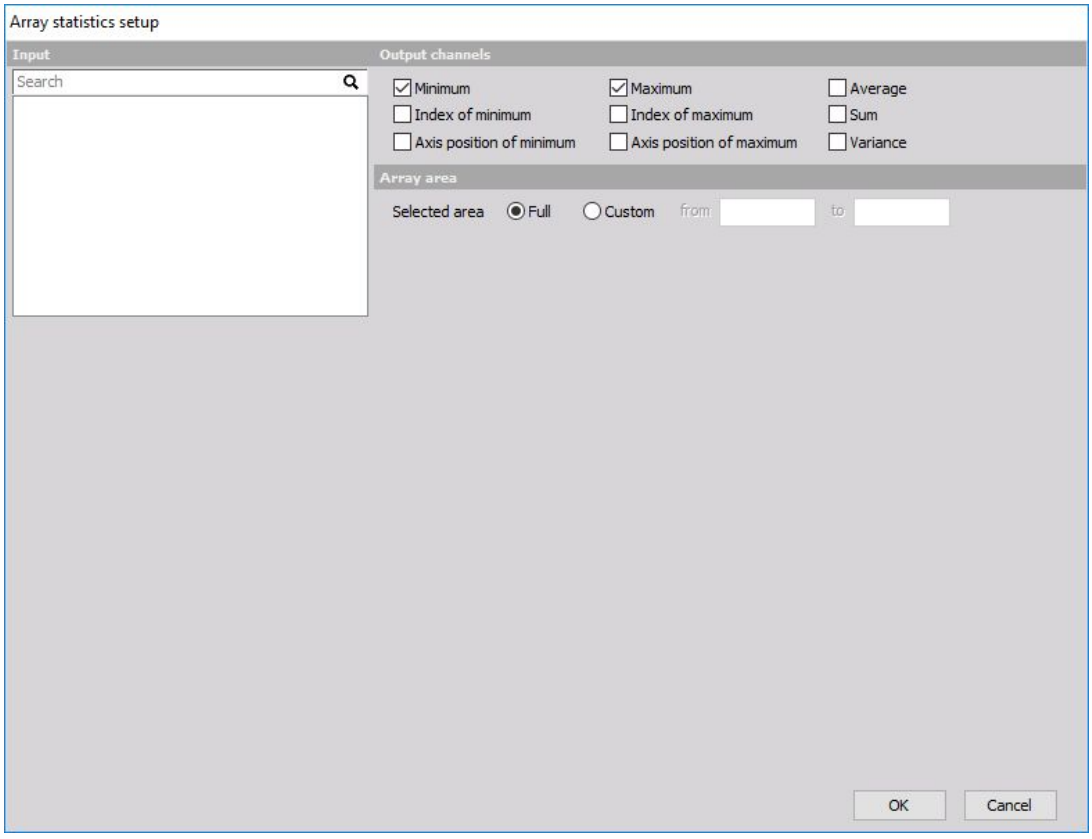
- Basic Statistics
- Array statistics
- Classification

ARRAY STATISTICS

The array statistics can calculate the statistical value from the array.

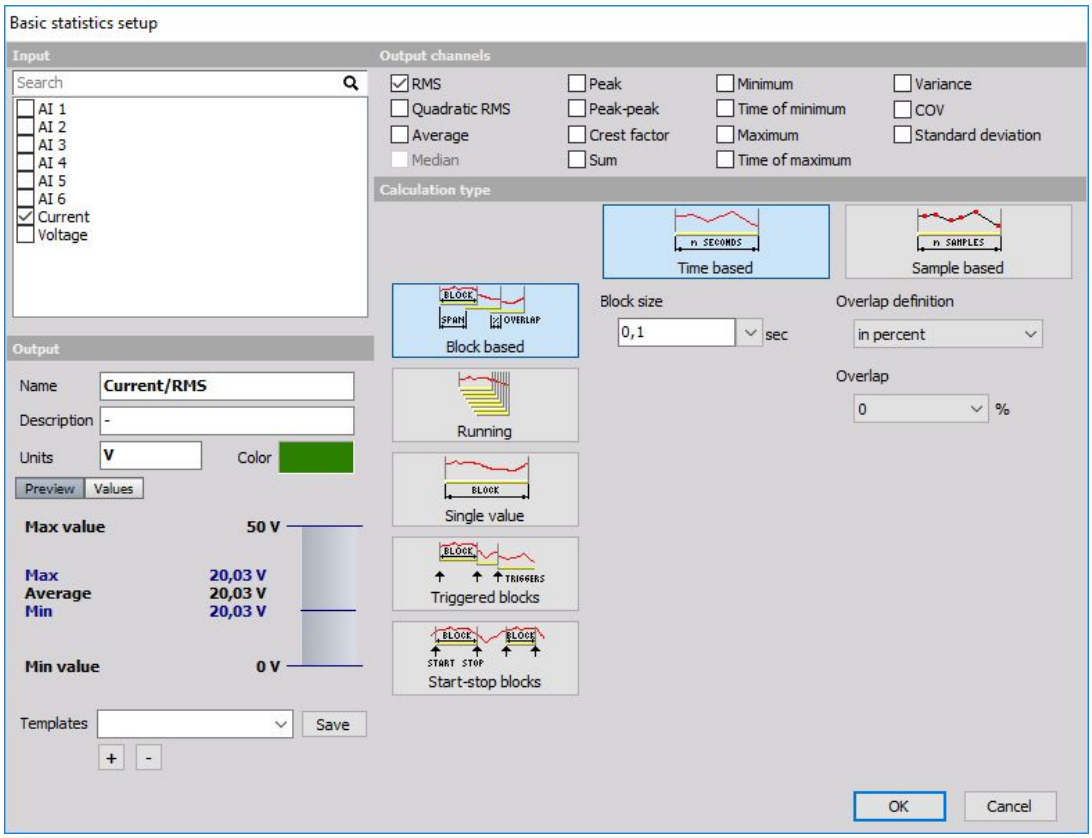
There are several options which can be chosen:

- **Minimum** finds minimum value from the array. There are two output channels created: class and value. The class will describe which index of the array holds the parameter and the value will be the minimum value itself.
- **Maximum** finds maximum value from the array. There are two output channels created: class and value. The class will describe which index of the array holds the parameter and the value will be the maximum value itself.
- **Average** calculates the average value of all elements from the array.
- **Sum** calculates the sum of all elements from the array.
- **Variance** calculates the variance of all elements from the array.



BASIC STATISTICS

When you press the Setup button on new Basic statistics line, the following Basic statistics setup window will open:



Basic statistics setup window has a standard layout with Input channels in left top corner, Output information in left bottom and output statistics channels on right top. Right bottom is populated by different basic statistic calculation settings.

The statistics support multiple input channels.

For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

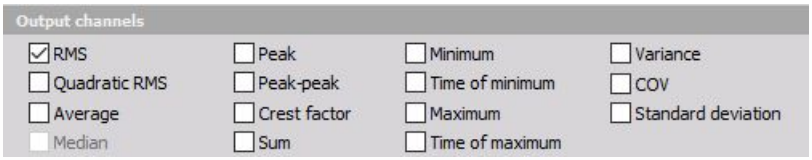
The statistic module provides basic statistical quantities of the signal. There are five basic calculation modes:

- Block based
- Running
- Single value
- Triggered blocks
- Start-stop blocks

After choosing INPUT channel, we need to select statistical function on OUTPUT CHANNEL section.

Output channel - Statistical functions

To select statistical function simply click (check) on the box beside its name on OUTPUT CHANNEL section:

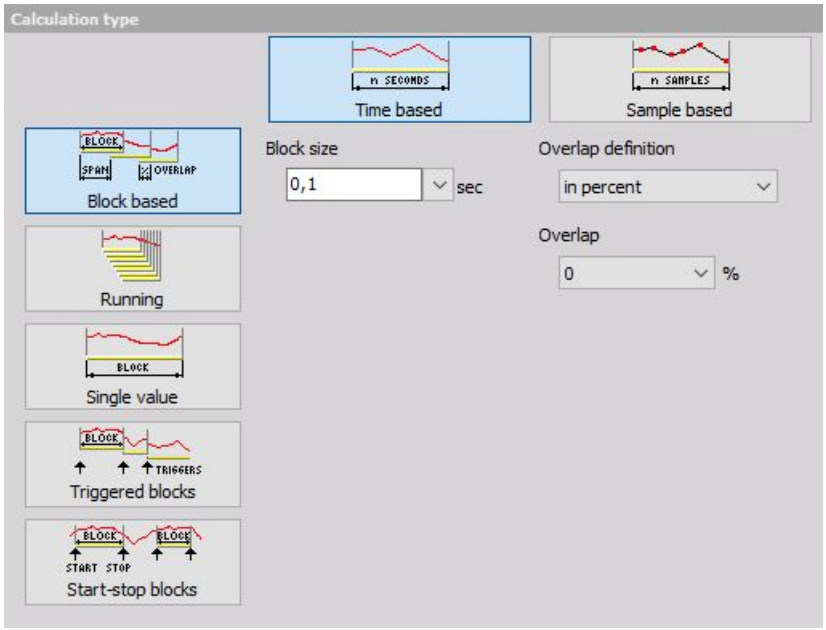


- **RMS** - Will calculate the root mean square value of the signal
- **Quadratic RMS** - This is similar to the RMS, except all the values are double squared and summed.
- **Average** - Average will calculate the average or middle point.
- **Median** - Is the value separating the higher half of a data sample, from the lower half.
- **Peak** - Peak is the maximum deviation of the signal from the average value.
- **Peak-peak** - Is the difference between the minimum and maximum.
- **Crest factor** - Is the ratio between the peak and RMS value. Crest factor gives an impression about the spikes in the signal. Pure sine waves have a crest factor of 1.41.
- **Sum** - Sum of all acquired values in respect of the selected calculation type.
- **Minimum** - The minimum will calculate a minimum value of the signal for the specified period.
- **Time of minimum** - On the end of the block, it displays the time at which the minimum value occurred.
- **Maximum** - The maximum will calculate a maximum value of the signal for the specified period. This is very intensive operation and therefore unavailable in Running mode.
- **Time of maximum** - On the end of the block, it displays the time at which the maximum value occurred.
- **Variance** - Indicating how possible values of a signal are spread around the expected value.
- **COV** - Coefficient of variation is the normalized measure of dispersion of probability distribution. It is calculated as the ration between standard deviation and the mean.
- **Standard deviation** - The standard deviation is a measure of the spread of the values of the signal away from its mean, measuring how widely spread the values in a data set is. If the data points are close to the mean, then the standard deviation is small (if all the data values are equal, then the standard deviation is zero).

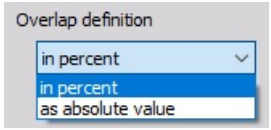
Calculation type

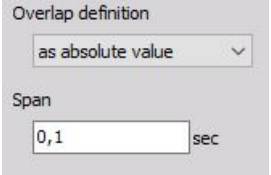
Block based

Block based calculation calculates the statistical quantity based on specific time interval defined by the block size.



- Block size in [sec] defines the time interval for calculation. 0,1 second in our case means that it will calculate the statistical quantities in 0,1 second interval. Therefore the resulting channels will have an update interval of 0,1 second.
- Overlap - Is useful when we need a specific time interval, but want to have a higher update rate of the resulting channels. In our case, the quantities will be updated in 0,1 second interval with 0% overlap and in 0,05 second interval with 50 % overlap. It means that the second block will not be calculated at the end of the first block, but half of the block before that. So the first block will be calculated from 0 to 0,1 second, second one from 0,05 to 0,15 second, third one from 0,1 to 0,2 second and so on.





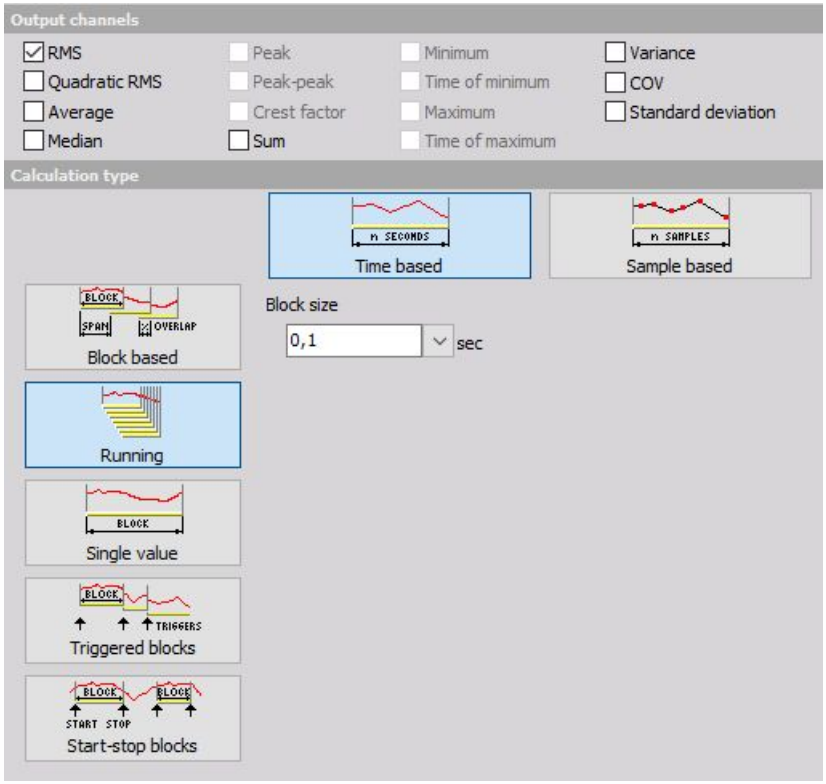
Overlap can be defined:

- in percent -> see above and also FFT Filter setup - Overlap
- as absolute value - Span in sec

Running

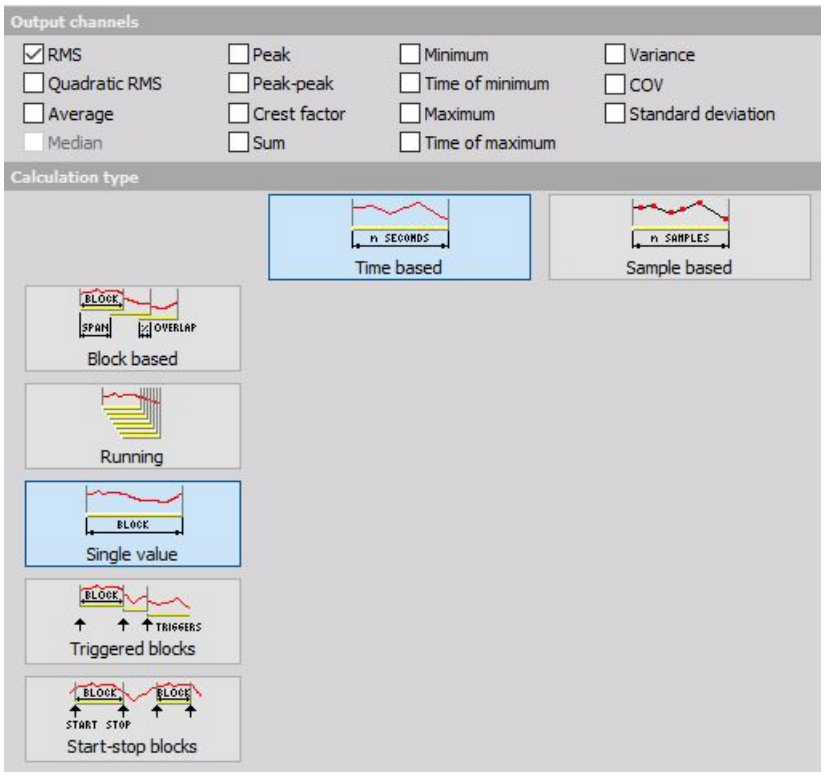
Running calculation is an 'extreme' version of overlapping. The second block is calculated after one sample after the first block. Block size has the same meaning as for block based calculation.

With this method, we can only calculate RMS, Average, Quadratic RMS, Variance and Standard deviation statistical functions, because all others would be too intensive (especially minimum and maximum while all others relate to those two).



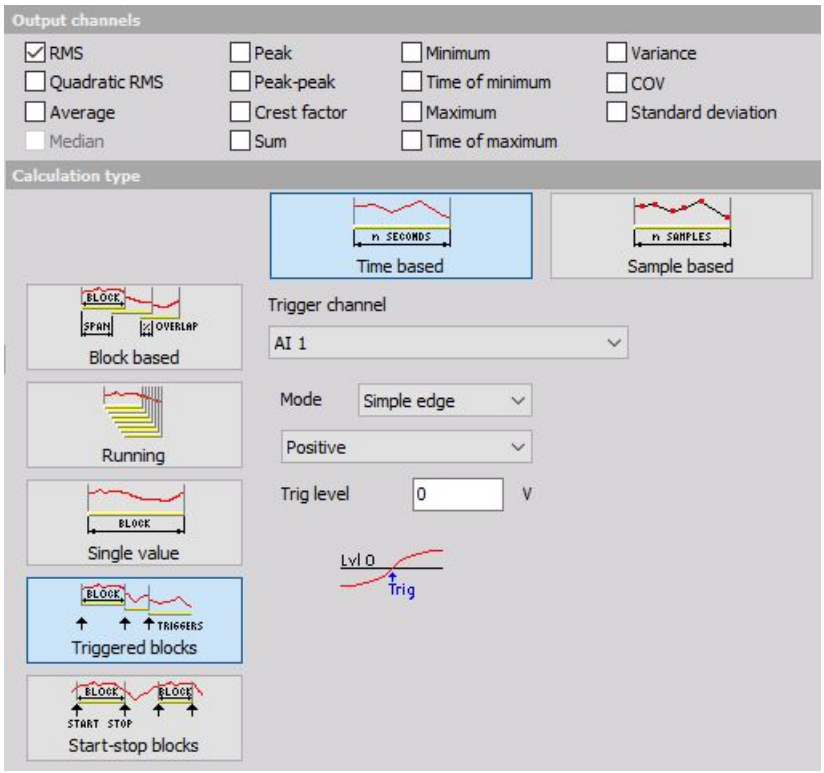
Single value

Single value is the simplest calculation and has no settings. It will produce only one value at the end of the measurement. The result will be updated also during the measurement, but only the final value will be stored in the data file.



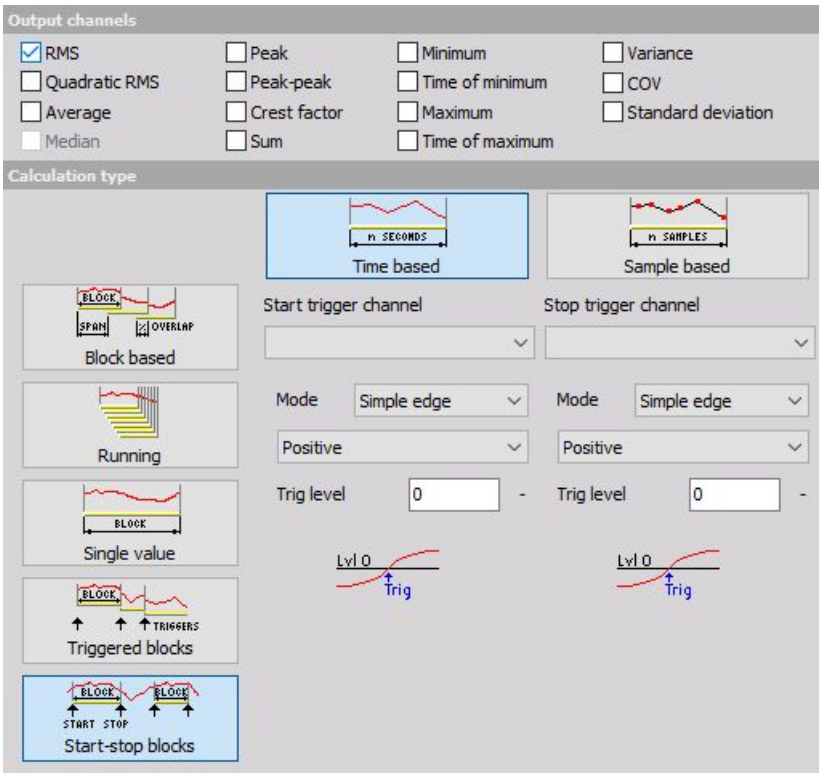
Triggered blocks

Triggered blocks option calculates the statistical value based on a specific trigger event. The calculation begins at the start of the acquisiton. When a trigger event is recognized, it stops the first calculation, writes the statistical value with its timestamp and then starts to calculate a new value. We can define any channel as the trigger channel and the settings for the trigger condition are the same as the alarm or storage triggers.



Start/stop blocks

Start/stop blocks option calculates the statistical value starting at a specific trigger event. When an event is recognized, it starts to calculate. When a stop condition is recognized, then the value is written to the resulting channel with the time stamp of stop event. It will wait with the calculation until the new start event is recognized. The start and stop channel can be any channel, also a different one and the trigger condition have the same options as the alarm or storage triggers.


Output channels
RMS
Quadratic RMS
Average
Median
Peak
Peak-peak
Crest factor
Sum
Minimum
Time of minimum
Maximum
Time of maximum
Variance
COV
Standard deviation
Calculation type
n SECONDS
Time based
n SAMPLES
Sample based
BLOCK
SPAN
OVERLAP
Block based
Running
BLOCK
Single value
BLOCK
TRIGGERS
Triggered blocks
BLOCK
BLOCK
START
STOP
Start-stop blocks
Start trigger channel
Stop trigger channel
Mode
Simple edge
Positive
Trig level
0
Lvl 0
Trig
Mode
Simple edge
Positive
Trig level
0
Lvl 0
Trig

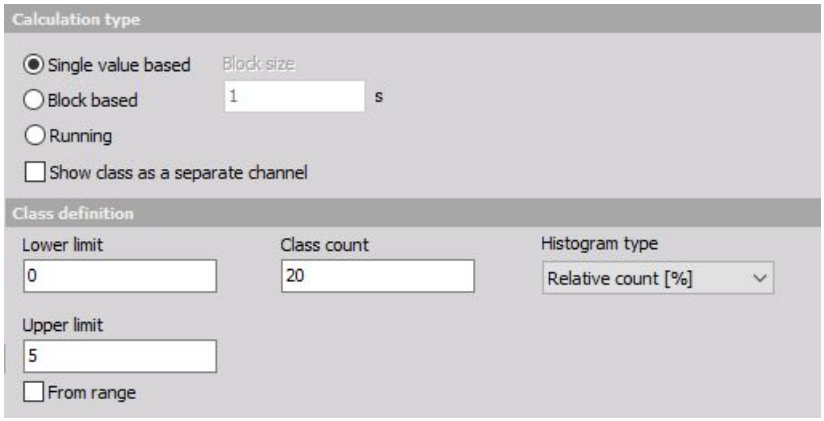
CLASSIFICATION

Classification is a procedure to count the values from the channel and puts them in the classes. A classical classification from the primary school is to create the classes and count number of student with specific weight or height.

Classification in measurement field is used for various applications, for example to find the distribution of power grid frequencies with time or to find the distribution of sound levels to which certain area or working place is exposed to.

First of all, we need to define what will be the result of classification. There are three options:

- **Single value based** - the result will be one array holding the result of the entire run
- **Block based** - the result will be set of the array each one added at the end of the defined block size. If we have for example 2 seconds block size and acquire data for 10 seconds, we will get 5 arrays of classification values, each valid for 2 seconds of data.
- **Running** - A running total is the summation of a sequence of numbers which is updated each time a new number is added to the sequence, by adding the value of the new number to the previous running total.
- **Show class as a separate channel**- option will create single value channels for each of the class element. This is a nice option to display the values in the multimeter.



For class definition we need to set the:

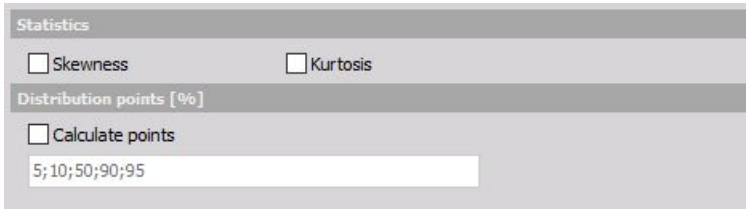
- **Lower limit** - this will set the lower limit for start counting - all values below this level will be counted in the first class
- **Upper limit** - this will set the upper limit for an end of counting all values above this level will be counted in the last class
- **Class count** - defines the number of classes. In the example above the width of each class will be 5/20=0,25. First and last class will have half-width, so it will go from 0 to 0.125. The second class has a middle value of 0.25 and it goes from 0.125 to 0.375 and so on.

Histogram type defines what will be the output of the data (amplitude):

- **Absolute count** - each class value has the number of samples within the class (value will always count up)
- **Relative count** - each class value has the value of samples with the class normalized to the

total number of counted samples (sum of all classes will be always 1)

- **Relative count [%]** - same as relative count, but expressed in percent (sum of all classes will be always 100)
- **Density** - provides empirical probability density each class value has the number of samples normalized to the total number of samples and divided by class width. In this case, the value is not depending on the number of classes within a range
- **Density [%]** - same as density but multiplied with 100
- **Distribution** - provides empirical probability distribution, each class value has the sum of all lower classes and the number of current samples, normalized to the total number of samples. The highest class has the value of 1.
- **Distribution [%]** - same as distribution, but expressed in percent. The highest class has the value of 100.

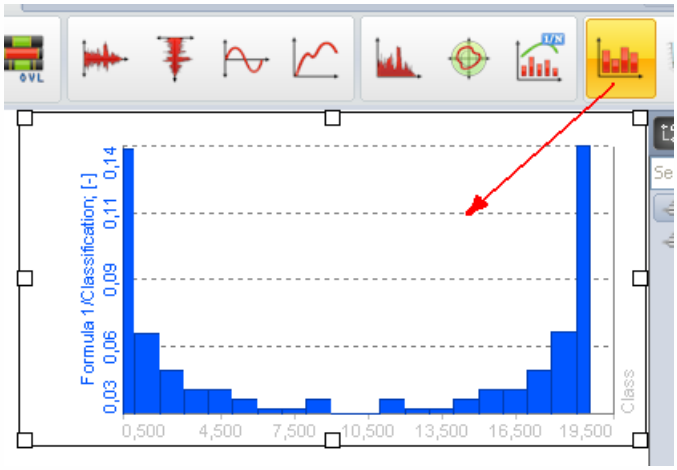


There are also several special output channels available. Two of them are Skewness (asymmetry of the probability distribution) and Kurtosis (a measure of "peakness" of distribution). Additionally, we can output a list of Distribution point values. Distribution points are the class values at which distribution reaches the entered value.

For the moment the distribution points work only if a distribution is chosen as the histogram type.

Histograms can be seen in the 2D graph during measurement and analysis.

If we choose block based calculation, we can also use the 3D graph to display the history of classifications.



REFERENCE CURVES

Available Reference curves on Math setup screen are:

- Frequency domain reference curve

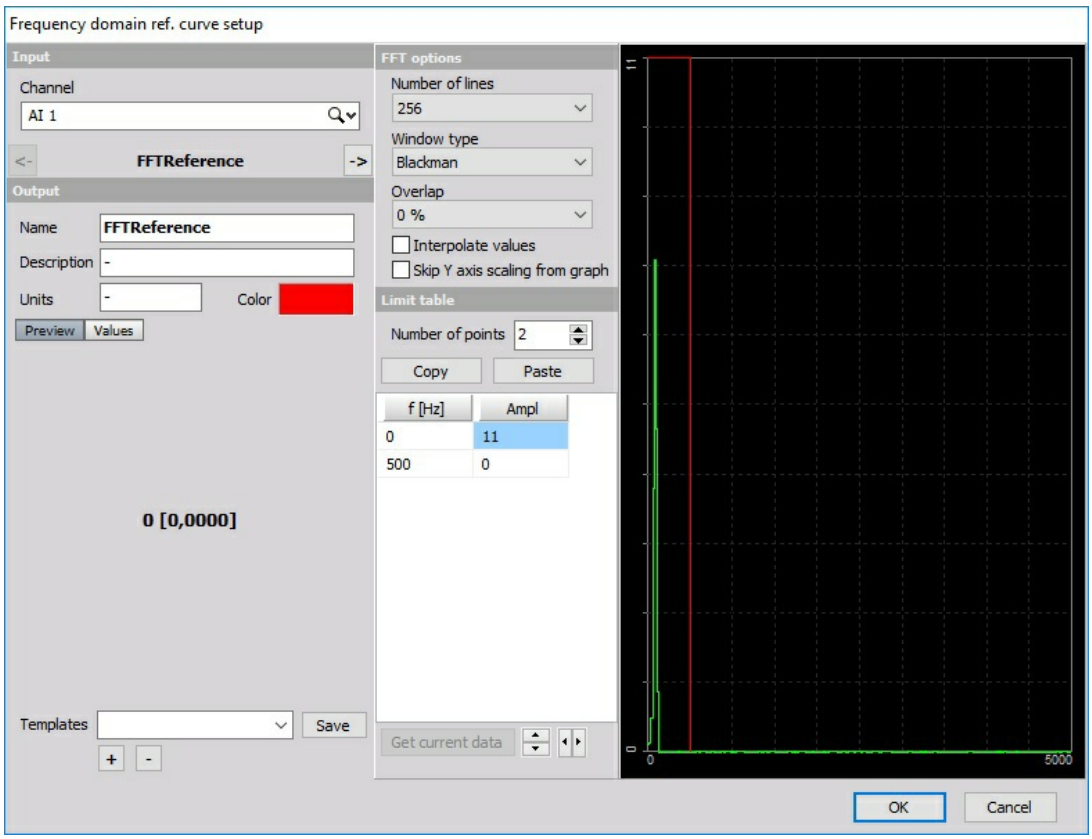
- Time reference curve

- XY reference curve

- Vector reference curve

FREQUENCY DOMAIN REFERENCE CURVE

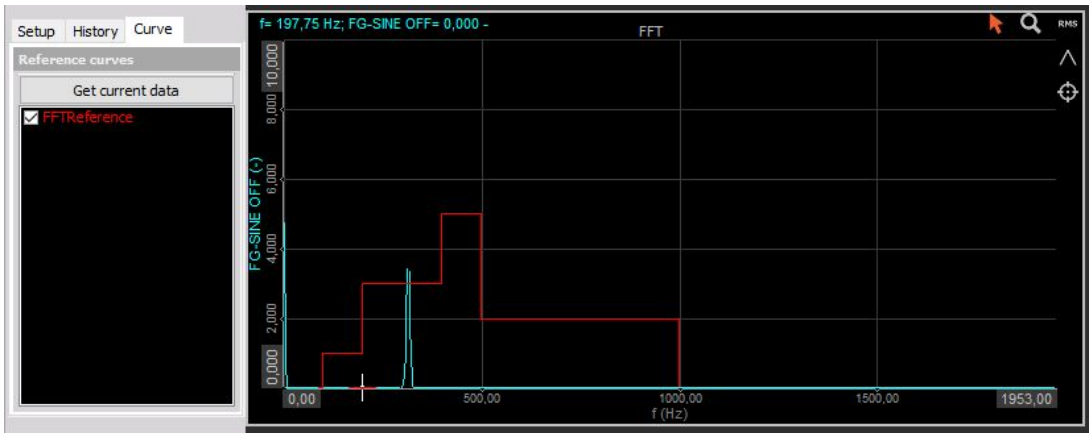
The FFT reference curve math module provides the reference curve for FFT display. It can be used for displaying the reference on FFT screen and also to provide a math output channel which goes high when the levels are exceeded.



For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

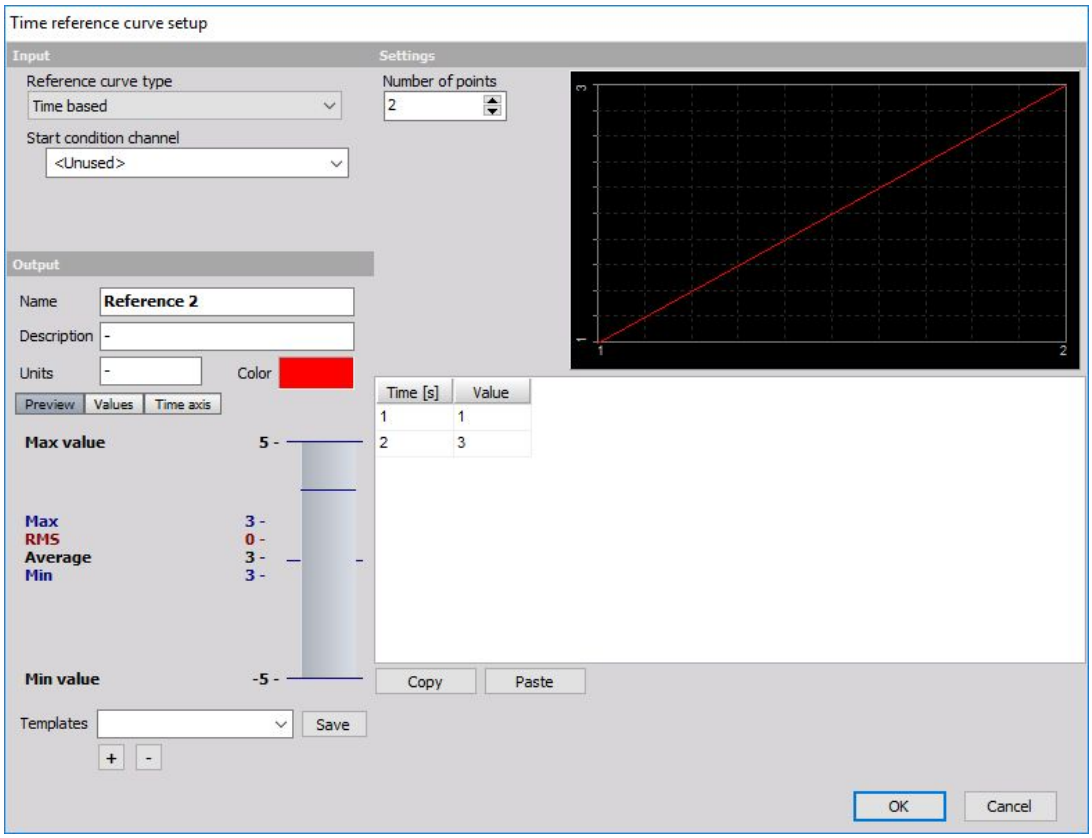
We define in the channel setup the maximum Number of lines, Window type and Overlap. These values will be used for calculation of FFT spectrum. Then we define the limits - Limit table. They can be entered value by value or from current data - Get current data button. This means that current FFT (in the setup screen) will be taken as the reference. Then we can define some offset to that reference (with up / down / left / right arrows button). Similar to the FFT trigger it will raise/widen the current transfer curve.

The FFT reference curve can be used in FFT display to show some reference lines, while the output channel (named Status in the screenshots) can be used for displaying either the current data is above or below the defined reference.


Setup
History
Curve
Reference curves
Get current data
FFTReference
f= 197,75 Hz; FG-SINE OFF= 0,000 -
FFT
RMS
FG-SINE OFF (-)
10,000
8,000
6,000
4,000
2,000
0,000
0,00
500,00
1000,00
1500,00
1953,00
f (Hz)

TIME REFERENCE CURVE

When you press the Setup button on newly activated Reference curve line, the following Reference curve setup window will open:

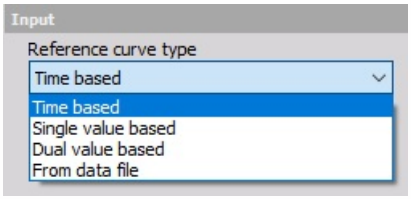


Input settings

Reference curve type

In this field we can select Reference curve type from the drop-down list:

- Time based
- Single value based
- Dual value based
- From data file

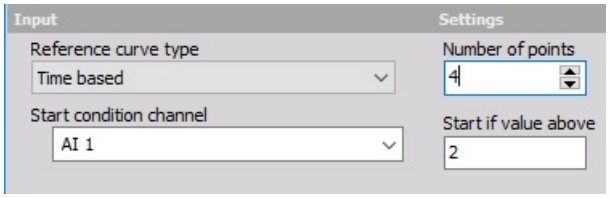


Time based reference curve

Time based reference curve is useful for defining a curve in time as a reference during the certain test, which has to follow the certain protocol. Let's say we have a test where we need to accelerate to 100 km/h in 10 seconds, drive constantly for 10 seconds and decelerate to 0 km/h in 10 seconds.

We can define for each curve starting criteria. If the Start condition channel is not defined, the curve will start at beginning of the measurement. Maybe it would be nice to define to start the test on a channel which measures vehicle velocity, but then we have also to define the limit Start if a value above (this field appear only when Start conditional channel is selected - default = <Unused>). In our example we could enter for example 2 km/h.

Next step is to define Number of points and the points themselves in the list. In our case we would have four points, so we enter this in the field.



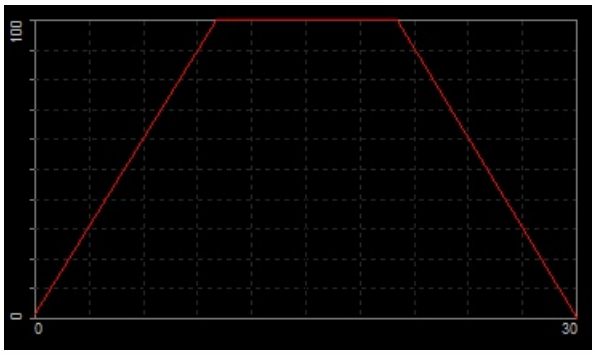
A table will show four empty rows. Now we enter the time points and the values. In our case it will be:

t [s]	Value
0	2
10	100
20	100
30	0

- first point: 0s, 2 km/h
- second point: 10 s, 100 km/h

- third point: 20 s, 100 km/h
- fourth point: 30 s, 0 km/h

Display on right upper part Reference curve setup window now shows this curve:



Then we can use the formula math channel to see if we are in between or out of the limit. We can for example calculate:

$$|\mathrm{RefCurve} - \mathrm{Velocity}|\ 5$$

This will give a number of 1 if the absolute difference from reference curve to the measured signal is above 5 km/h and 0 if the difference is lower.

Single value based reference curve

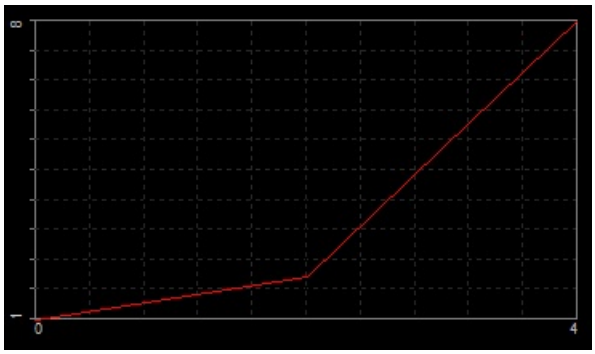
A single value based reference curve can be understood as a sort of non-linear scaling. We need to define a First reference channel,

Number of points and a table with the values of reference and output channel.

So in this case on right, if the input channel will be 3, the output value will be 5. The values will be interpolated in between the points. So if the input is 2.5, the output will be 3.5. If the value is below or above the lowest or highest point, data will be extrapolated from two lowest or highest points. If we have 0 at the input, we will have 0 at the output. If we have 5 at the input, we will have 11 at the output.

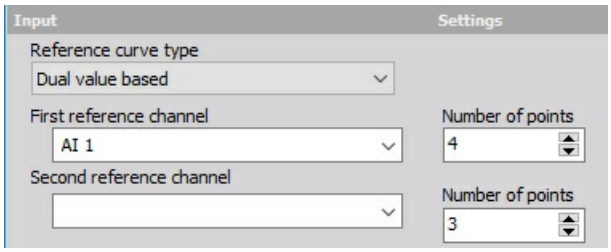
Ref. val	Value
0	1
2	2
3	5
4	8

Display on right upper part Reference curve setup window now shows this curve:



Dual value based reference curve

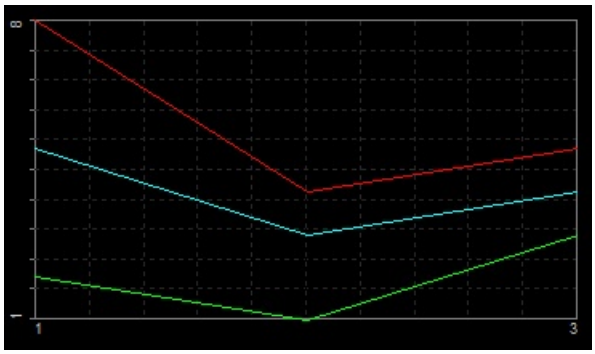
Dual value based reference curve has two inputs: First reference channels and Second reference channels. We can imagine as the tridimensional surface of reference points. X and Y are two reference channels while the Z is defined points. We need to define first the channels and the Number of points for each channel.



Then we need to enter the reference values in the list.

Ch. 1 \ Ch. 2	1	2	3
2	2	1	3
3	5	3	4
4	8	4	5

Display on right upper part Reference curve setup window now shows this curve:

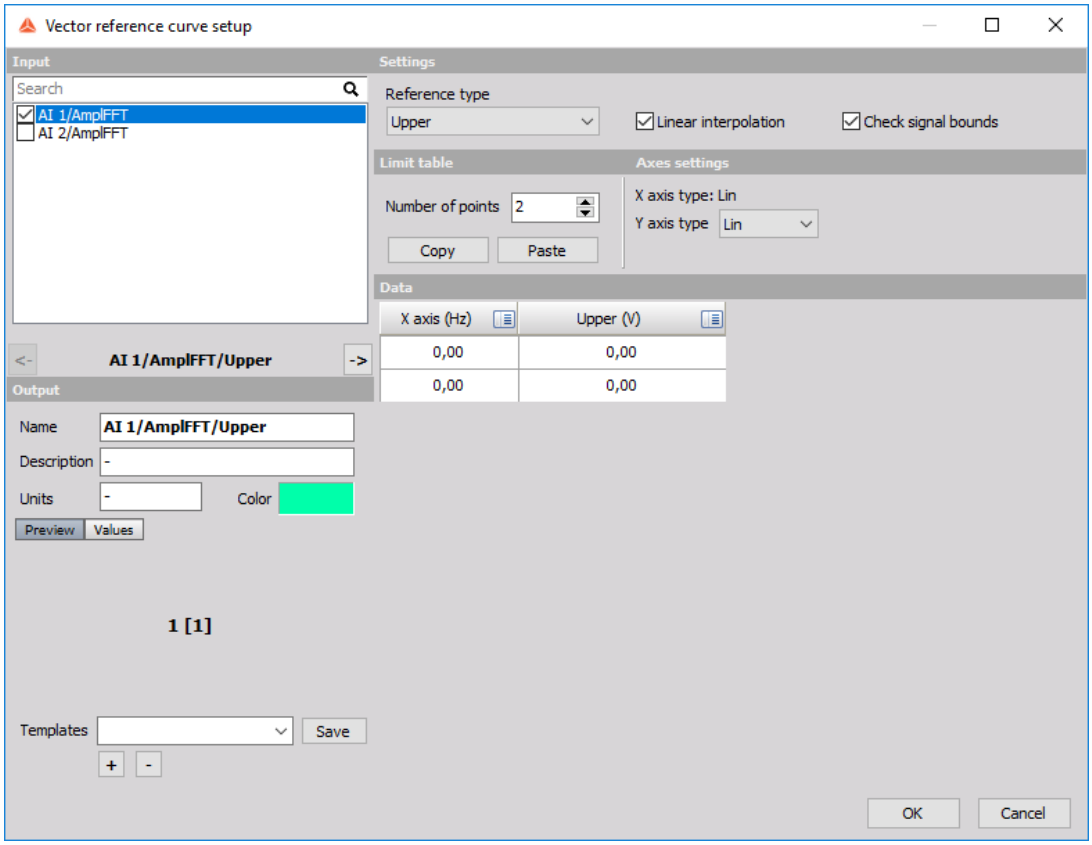


A good example of using dual reference curve is to define the limit oil pressure referenced to RPM and the oil temperature.

VECTOR REFERENCE CURVE

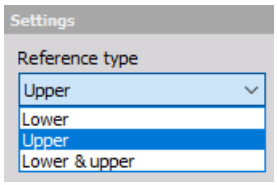
In the vector reference curve, we define the reference curves for the input channel, which is usually an array (FFT data, ...).

When you press the Setup button on newly activated Vector reference curve line, the following setup screen window will open:



Settings

Reference type



The reference curve type can be selected from:

- lower (define the lower limit)
- upper (define the upper limit)
- lower & upper (define the lower and upper limit)

Linear interpolation

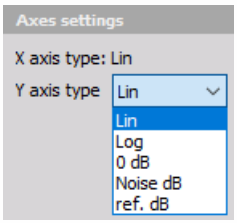
With linear interpolation, the program will draw a direct line between the two neighborhood points. It will give a smoother transition between the points on the curve.

Check signal bounds

This will create an additional output channel which will output 1 when the input is higher than the reference curve and will output 0 when the input is below the reference curve

Axes settings

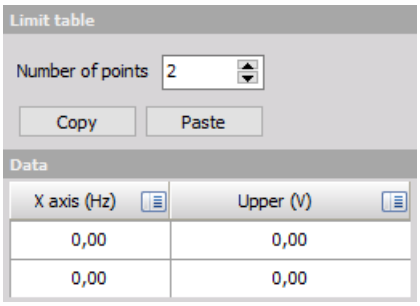
Axes settings can be selected from:



- linear
- logarithmic
- 0 dB (dB reference will be set from the range of the first channel)
- Noise dB (the reference value is 20 uPa)
- ref. dB (user enters the reference value)

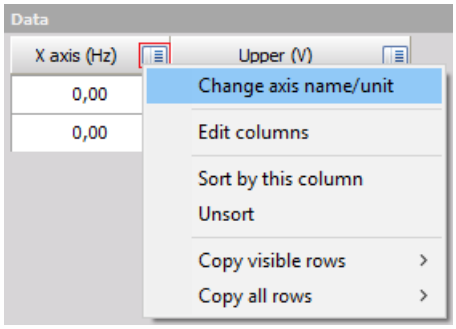
Limit table

Limit table points can be manually defined or by copy/paste from an external program.



Data

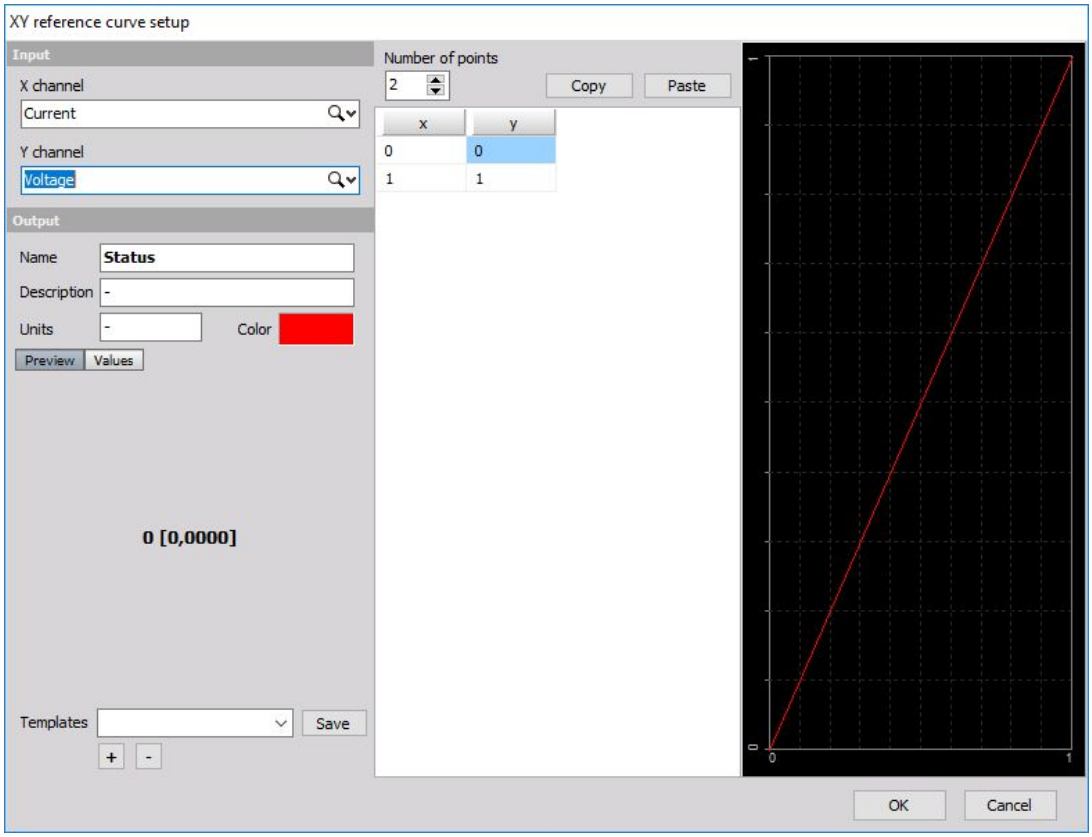
Rename the channels with the click on the icon in the top right corner and selecting the Change axis name/unit option.



XY - REFERENCE CURVE

The XY reference curve math module provides the reference curve for XY display. It can be used for displaying the reference (if we display maximum torque vs. RPM, for example).

When you press the Setup button on new activated XY reference curve line, the following XY reference curve setup window will open:



For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

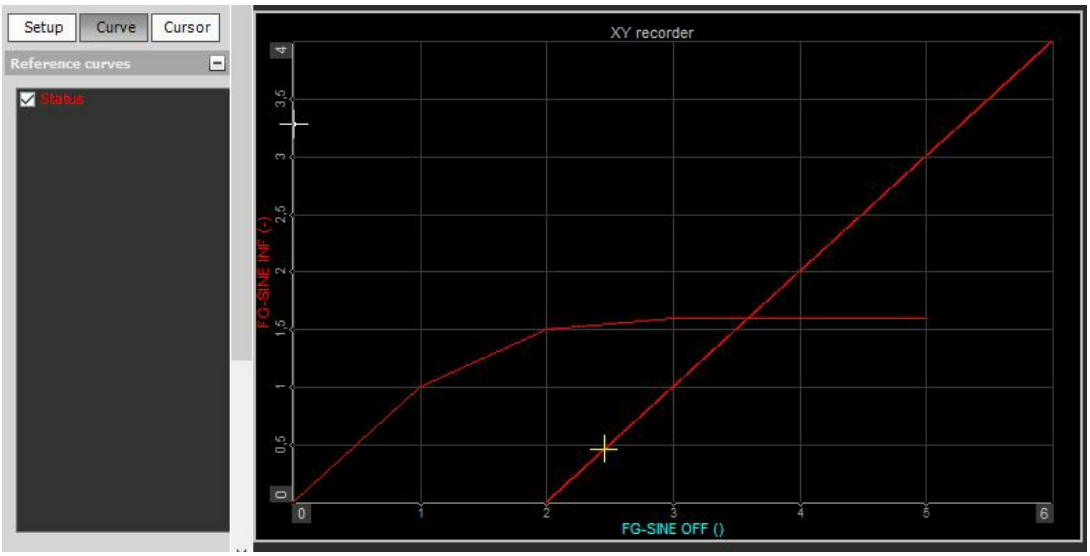
After choosing INPUT X channel and Y channel, we need to set Number of points for XY reference curve. The reference points can be entered value by value. We also have another option for easier transfer of points copy and paste function. Copy button copies all the points to the clipboard in tab-delimited style, so it can be easily pasted to any spreadsheet software. The values can be copied from other programs with the Paste command, where the data has to be tab delimited and each line should be terminated with carriage return/line feed character. The first line should have XTAB YCR LF.

In simple words, if we see the following picture in MS Excel, choose the data range and select Copy, the data can be easily pasted in Dewesoft.

x	y
0	0
1	1
2	1,5
3	1,6
5	1,6

Display on right upper part XY reference curve setup window shows the defined curve. Lower left part of XY reference curve setup window shows the selected OUTPUT channel settings.

The XY display can show the reference curve as seen in the picture below.



There is another advantage of the reference curve. The output channel (Status in our case) gives the value of 1 when the XY curve crosses the reference curve. This can be used for trigger criteria or for counting of events (with ECNT function in formula editor).

TIME DOMAIN ANALYSIS

In Time domain analysis we can chose from following mathematical functions:

- Integral, derivative

- Latch value math

- Scope math

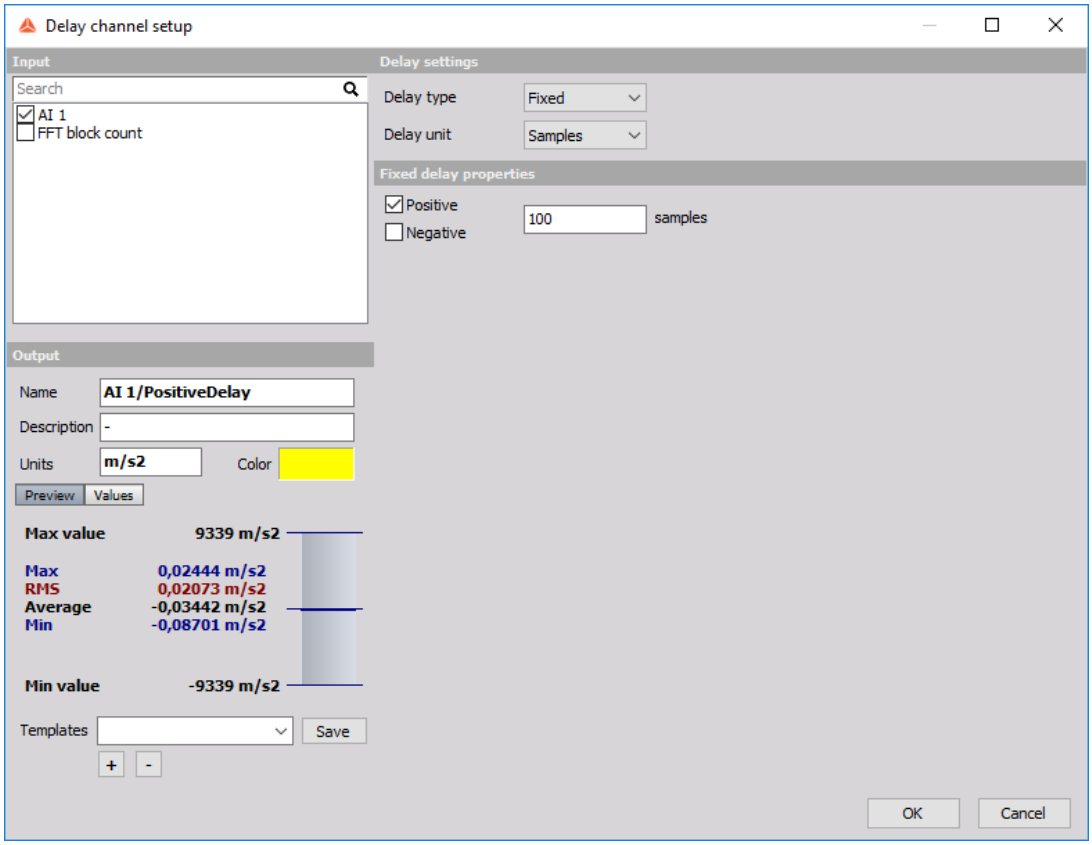
- Delay channel

- Time-to-vector tranform

DELAY CHANNEL

Delay channel is used for artificially delaying or advancing of a signal. Input channels can be synchronous or asynchronous.

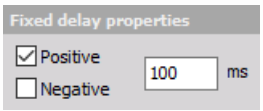
When you press the Setup button on newly activated Delay channel line, the following setup screen window will open:



Delay settings:

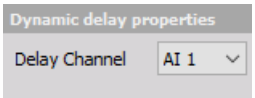
Delay type

- Fixed



We can delay (negative) or advance (positive) the signal

- Dynamic



Delay a channel in reference to another positive signal.

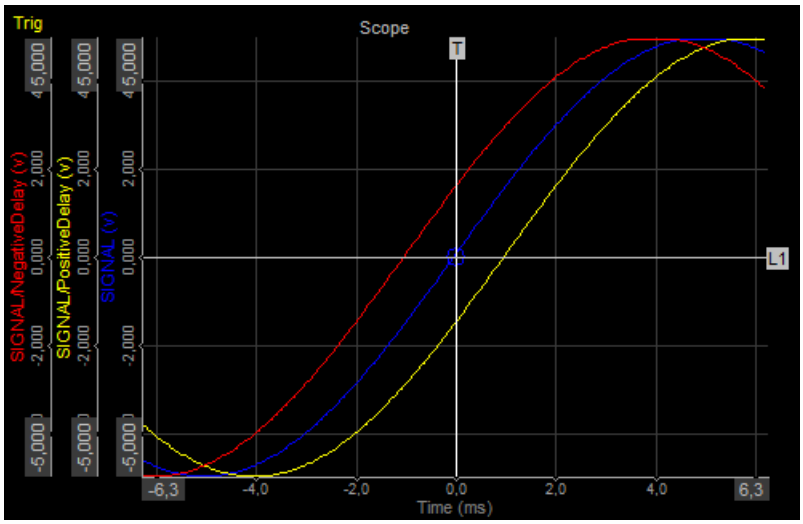
Delay unit

- Time

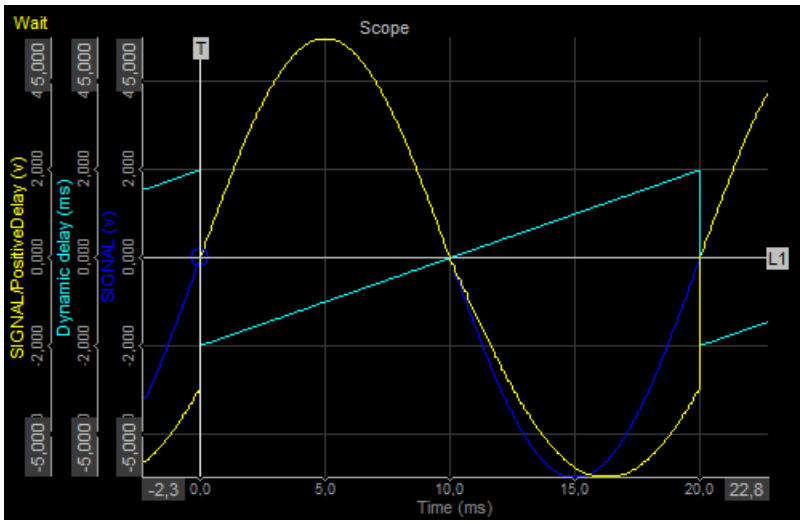
Switch between delaying on time or samples per your requirements.

- Samples

Example of the signal with fixed positive and negative delay:

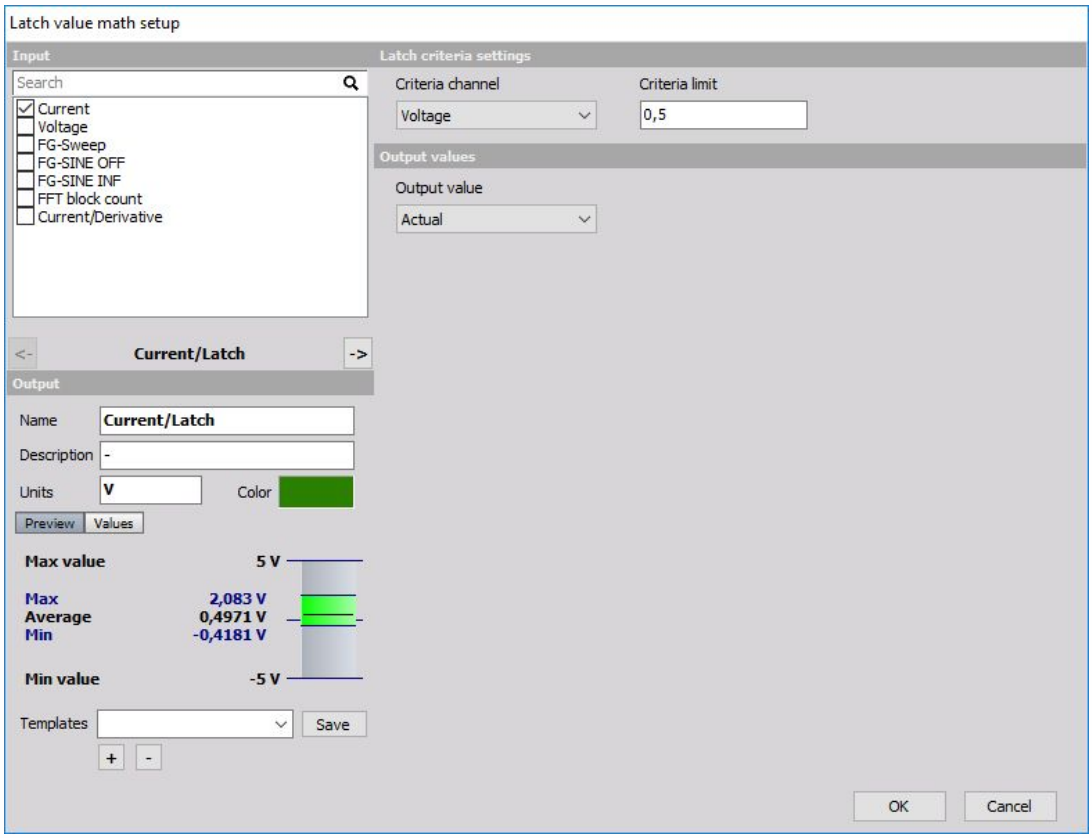


Example of signal and it's dynamically delayed signal:



LATCH VALUE MATH

When you press the Setup button on newly activated Latch math line, the following Latch math setup window will open:



For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

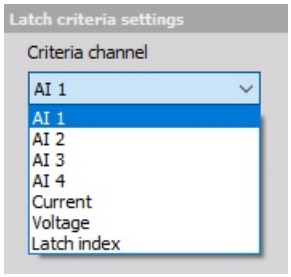
The Latch math will return the value of the channel when a latch condition is met. So we can use this for getting the values only at specific trigger position (for example a state of the test). These values can be nicely displayed in the tabular display.

After choosing INPUT channel, we need to set Latch criteria and Output value.

Latch criteria settings

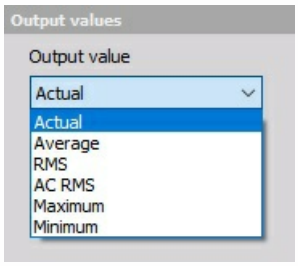
Select Criteria channel from the drop-down list of available channels.

After that enter Criteria limit value in the corresponding field.

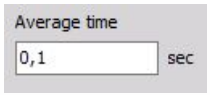


Output value

The select Output value from the drop-down list of available Actual and statistical values.

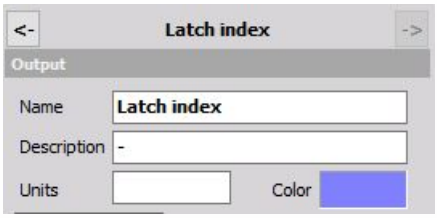


For all statistical values additional field appear to enter Time base in seconds:



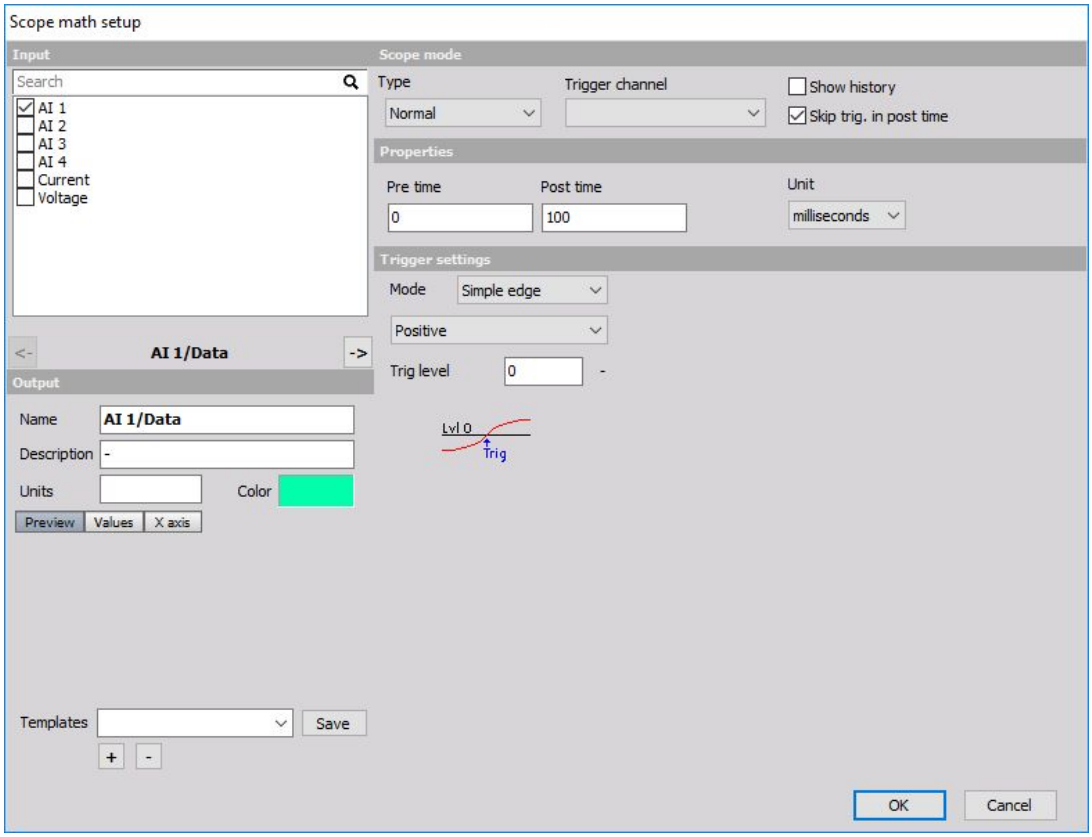
Latch index

Lower left part of Latch math setup window shows the selected output / Latch channel settings and predefined Index channel. Output selection is made with left and right buttons.



SCOPE MATH

Scope math is intended to extract matrix channel from the scope trigger shots on which the math functions can be performed. We will get either the last shot or the history of scope triggered shots.



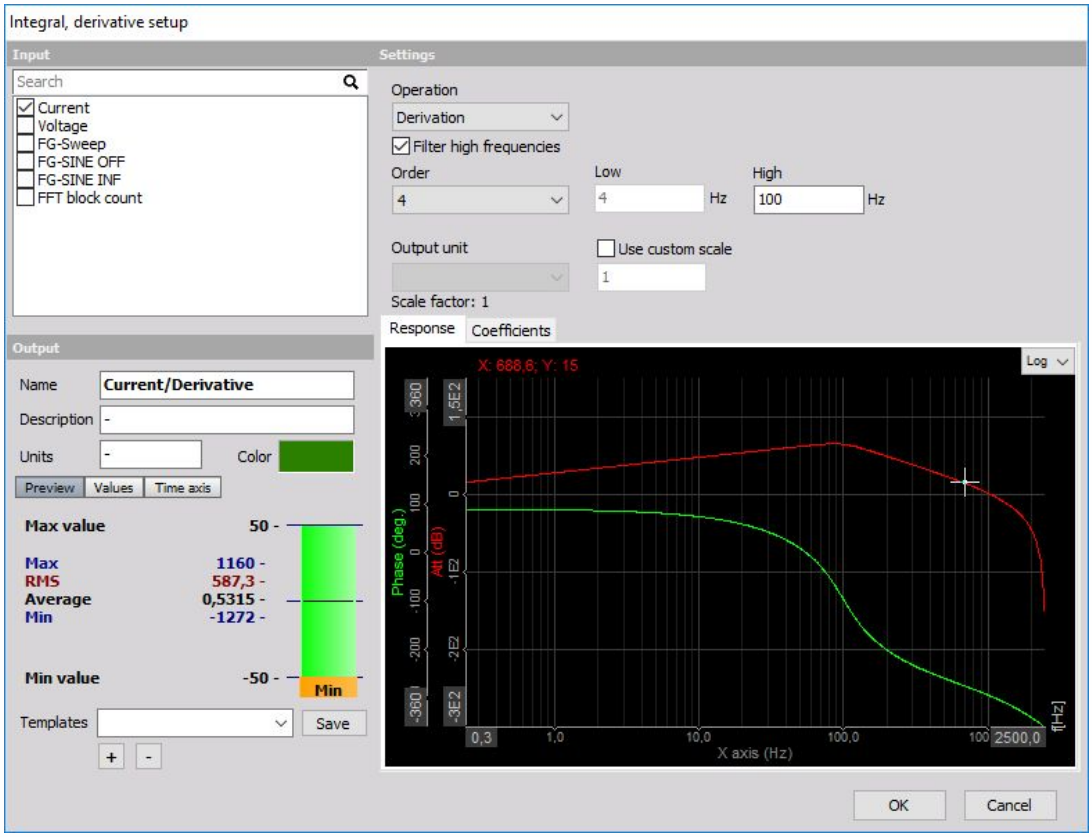
For that we need to define the trigger Type which has the same meaning as on the scope instrument:

- **Normal** - only output a scope shot when a real trigger occurs
- **Auto** - if a trigger is not found, output current values
- **Free run** - outputs scope pictures regardless of the trigger
- **Single shot** - outputs only at first trigger
- **Unit** - defines the x scale units either in milliseconds or samples.
- **Trigger channel** - defines the channel for triggering the shots. We can output many different channels (defined on the left upper side, but the trigger channel can be only one for all).
- **Show history** - defines if the output is single channel (always overwritten with latest data) or if the entire history is put into the channels.
- **Skip trig. in post time** - defines if the triggers are searched also when post time is running. If yes, data could overlap.
- **Pre time and Post time** - defines pre trigger and post trigger time.

Trigger settings are the same as in normal scope, alarms or storing triggers.

INTEGRAL, DERIVATIVE

When opening a new Integral/derivative math function the following setup window will appear:



Under settings we can select the type of operation:

Integral functions:

- Integration
- Double integration

Derivative functions:

- Derivation
- Double derivation

INTEGRATION AND DOUBLE INTEGRATION

Integration

There are two main purposes of integration: first is to get for example traveled distance from measured velocity or to get the energy from measured power. Another type of integration is to calculate the velocity vibration from the measured acceleration.

A clear difference between two basic functions is that in the first application the carrier of information is the DC value - the offset (traveled distance or spent energy). In other application the offset is only a measurement error and the carrier of information is the dynamic part of the signal-vibration velocity in our example.

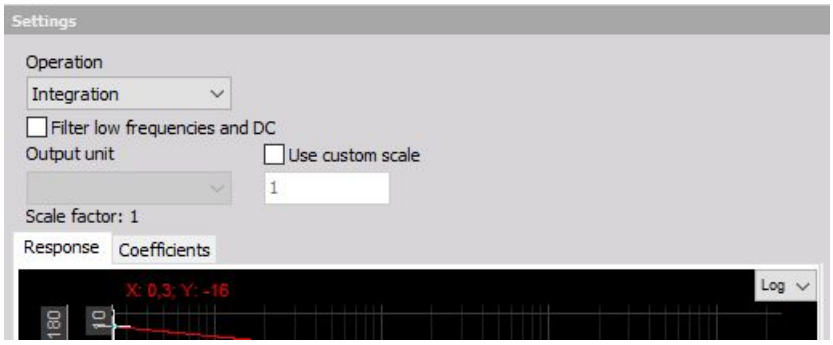
1. Let's first take a look at how to integrate distance from velocity. If we have for example velocity in km/h as an input channel, we choose a filter, integration, and switch OFF the option to 'Filter low frequencies and DC'. Now the only thing left is to define the Scale. We have to know that integration adds a sec to the unit.

EXAMPLE: **So if we have km/h as input, we have at output: km/h * sec = 1000 m / 3600 sec * sec = 0.278 m**.

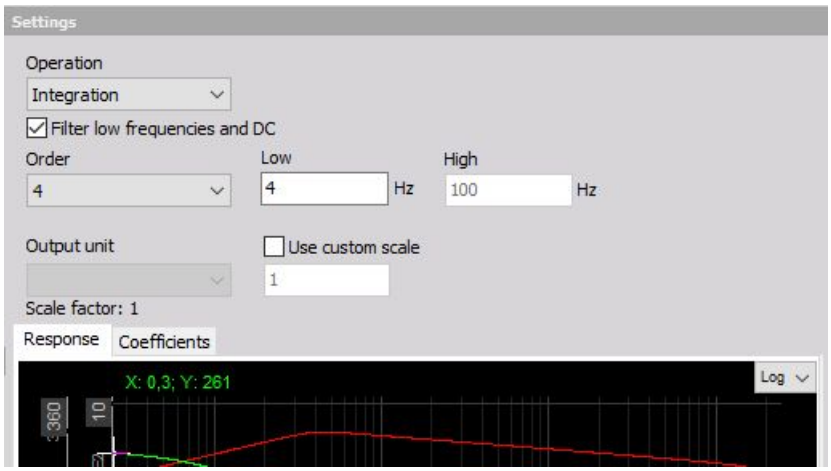
Therefore if we want to have the output in meters, we have to enter 0.278 as the scale factor. If we want to have output in km, we have to enter 0.000278.

2. Let's practice on another example - if we have input channel as a power in kW, we have at the output:

EXAMPLE: **kW * sec=kW * 1/3600 * h=0.000278 kWh - this is our scaling factor.**



The second application is to use integration on dynamic signals like vibration acceleration. If we have measured acceleration, there is always some offset because of an amplifier and AD converter offset. This offset will result in a drift of result, which is not wanted in this case. Therefore we need to use the option 'Filter low frequencies and DC' to cut this offset.



Then we need to define the Order. Be aware that integration is equivalent to filter with order 1, so we need to choose a high pass filter with order 2 or higher to really cut DC values. The Flow tells us where the resulting velocity will be cut off. High values of Flow (like 10 Hz) will result in very fast stabilization in the case of overload, but it might on the other side already cut the information we require. Low values (< 1Hz) will result in quite slow stabilization times (typically 5÷10 seconds for 1 Hz filter) but will pass through the virtually entire frequency range.

For vibration measurement, the usual value is from 3 to 10 Hz for general purpose measurement. For low-frequency vibration like the human body or building vibration, a value between 0.3 and 1 Hz is used. For special application, like sea sickness or high structures (like TV towers or cranes) movement a very low frequency like 0.1÷0.3 Hz is used, but we have to know that the stabilization will be a very long process. In this case, we also need to assure that the sensors we are using have such low-frequency range. General purpose ICP sensors have the cutoff frequency between 0.3 to 1 Hz and therefore are not useful in such applications.

Let's look at the Scale - scaling factor for this application. Let's assume that we measure the acceleration in g. If we want the results in mm/s, we need to have a scaling factor:

1 g * sec = 9,81 m/sec/sec * sec = 9,81 m/s = 9810 mm /s

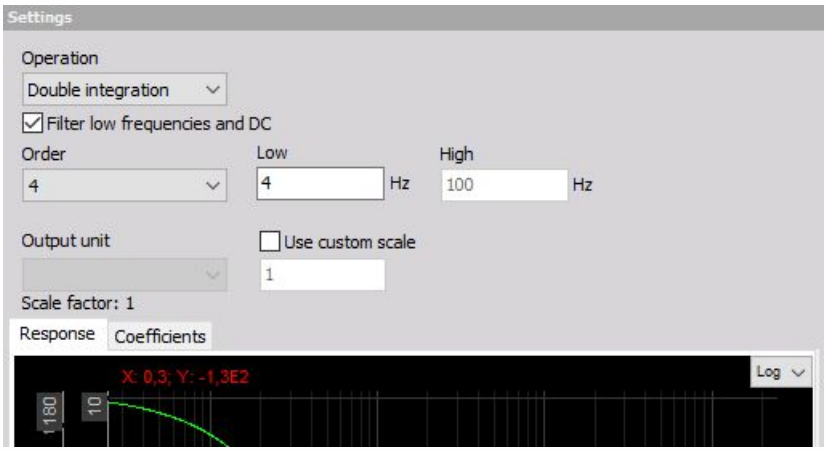
So we need to enter 9810 in the field scaling.

Double integration

Double integration is useful mainly to directly integrate displacement from acceleration, so to use it for dynamic signals. Our output unit will be multiplied with sec * sec. Therefore we need to again to choose the option to Filter low frequencies and DC, but we have to take care since the double integration is similar to second order filter, we need to choose Order 3 or higher for low-frequency filter.

Let's see how to calculate the Scale - scaling factor. Usually, we want the measurement result in µm (micrometers). So we have to enter a scaling factor of:

1 g * sec * sec = 9,81 m/sec/sec * sec * sec = 9,81 m = 9,81E6 µm



DERIVATION AND DOUBLE DERIVATION

Derivation

Derivation and double derivation is used, as the name already suggests, to calculation derivation of chosen input signals. Here the application range is not divided into two areas since the procedure is similar in all cases. The basic calculation is simple: we subtract current value from the previous one and divide by the time interval.

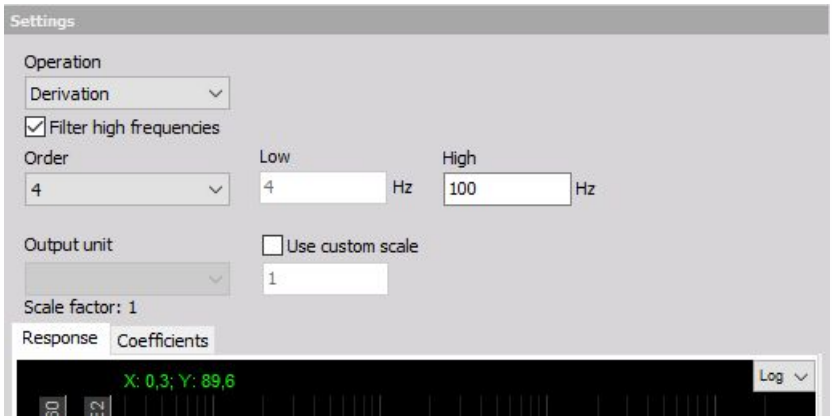
However, this might produce very noisy signals, especially with the high sampling frequency. We can look at the derivation as the 20 db / decade growing filter in the frequency domain. Sometimes it is therefore nice to cut the high-frequency contents. We can choose an option to Filter high frequencies. We choose the Order (at least 2 for derivation and at least 3 for double derivation) and FHigh - cutoff frequencies of the signal.

Let's assume the example of calculating acceleration out of speed of the vehicle. With a car it is a fact that we can't have the acceleration higher than for example 10 Hz coming from the real vehicle acceleration - higher values are basically vibrations.

So we can choose the 10 Hz as the FHigh high frequency and just get the real vehicle acceleration. Scale - scaling factor is similar, here the input is divided with second for derivation and divided with second squared for double derivation. So let's calculate the scaling factor for example above:

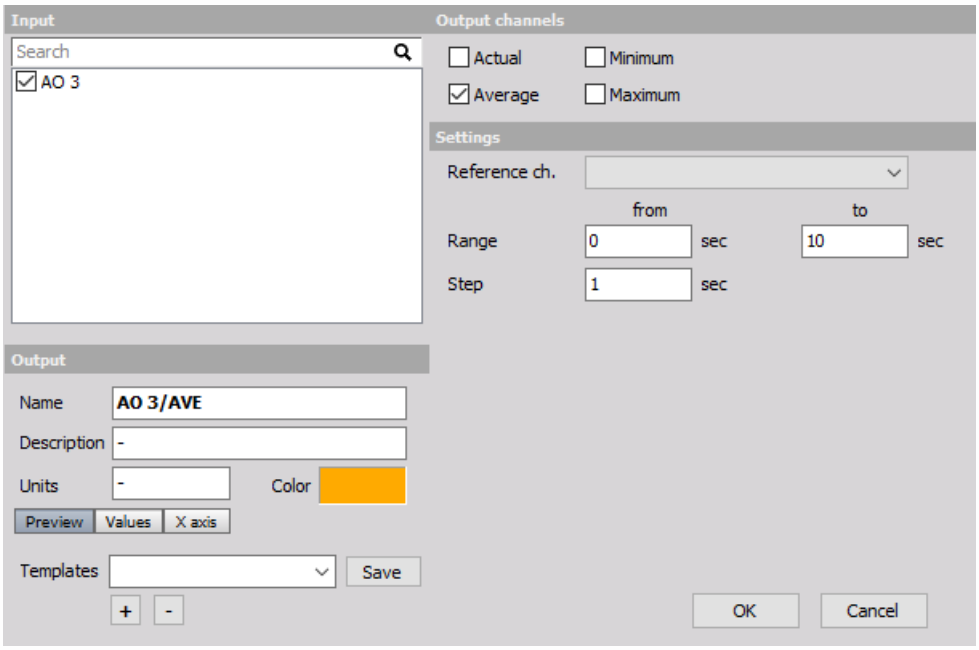
1 km/hour / sec = 1000 m / 3600 sec /sec = 0.278 m/s2 = 0.278 / 9,81 g = 0.0283 g

We have to enter this value in the scale field to get correct results.



TIME-TO-VECTOR TRANSFORM

When you press the Setup button on newly activated Time-to-vector transform line, the following setup screen window will open:



FREQUENCY DOMAIN ANALYSIS

In Frequency domain analysis we can choose from following mathematical functions:

- Cepstrum

- Correlation

- Exact frequency

- Fast Fourier transform

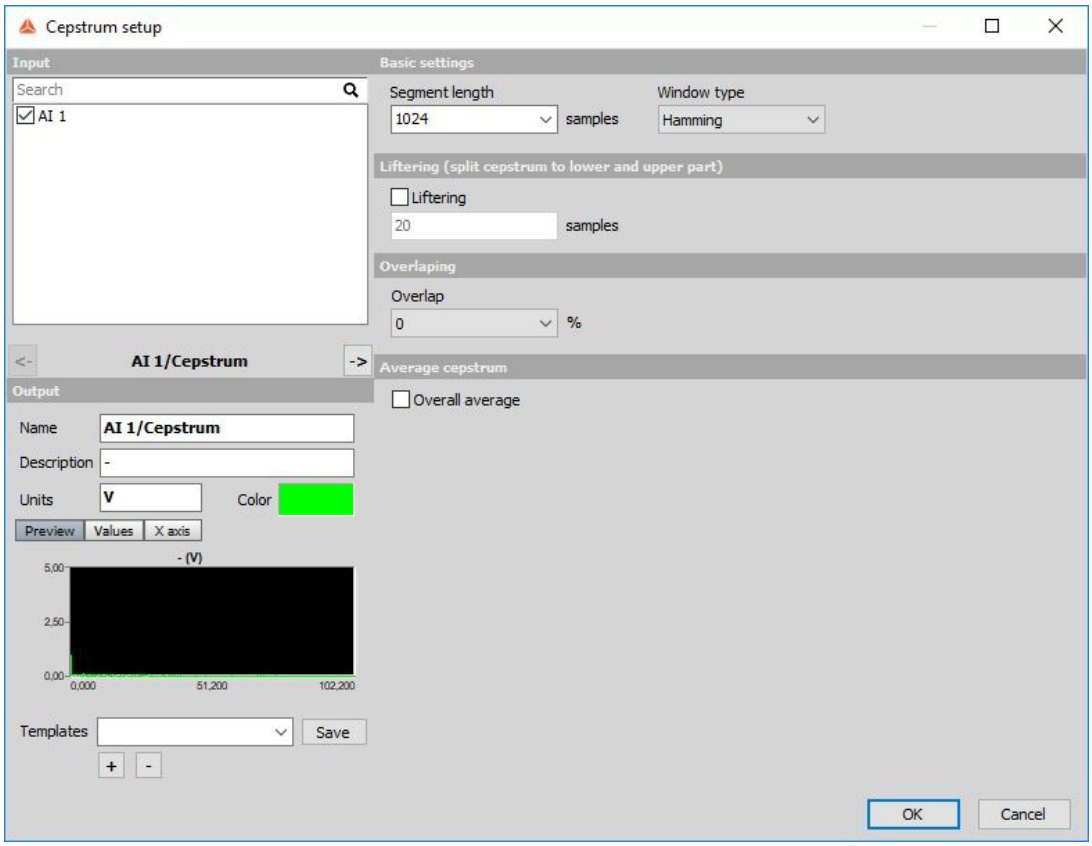
- Frequency integration and derivation
- Full spectrum

- Octave analysis

- Short-time Fourier transform

CEPSTRUM

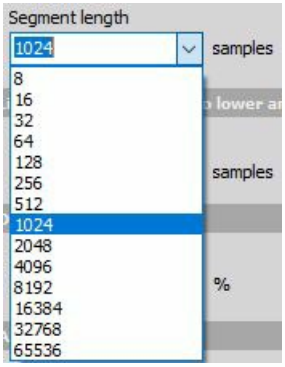
When you press the Setup button on newly activated Cepstrum line, the following setup screen window will open:



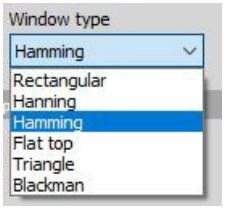
Module performs real cepstrum analysis (one-sided). The cepstrum is a representation used in homomorphic signal processing, to convert signals (such as a source and filter) combined by convolution into sums of their cepstra, for linear separation.

Basic settings

Segment length (number of samples used)

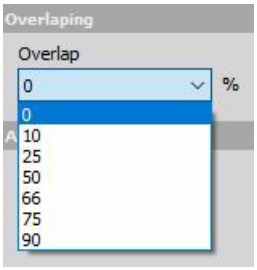


Window (Rectangular, Hanning, Hamming, Flat top, Triangle and Blackman)



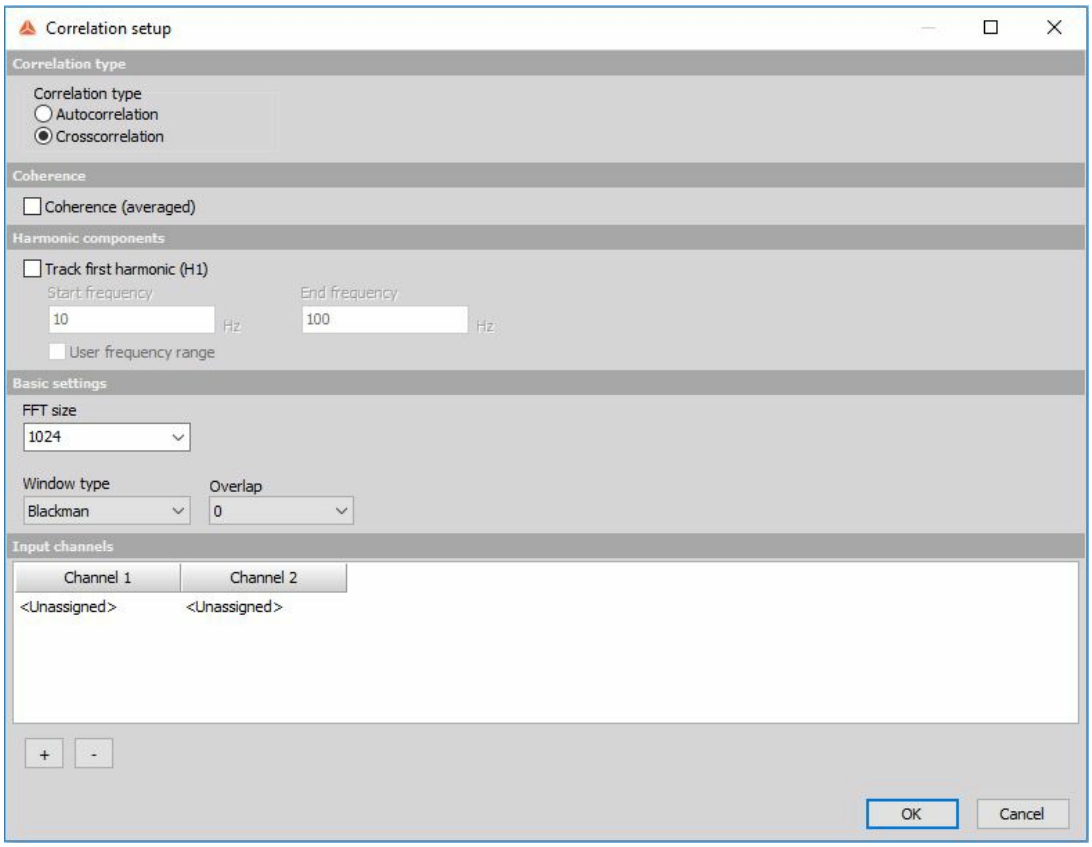
Overlaping

Overlap defines the percentage of time signal that has already been calculated and it is used again for calculation.



CORRELATION

When you press the Setup button on newly activated Correlation line, the following setup screen window will open:

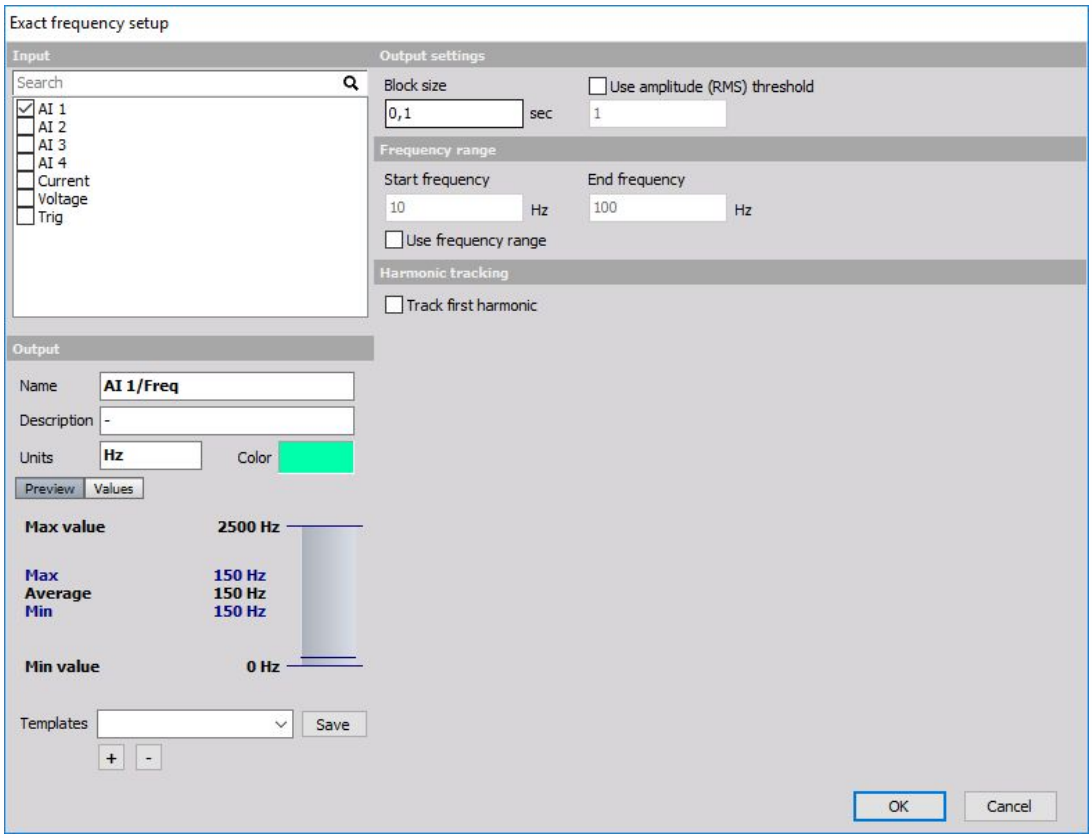


Two types of correlation can be computed:

- **Autocorrelation** - has one input channel.
- **Crosscorrelation** - has two input channels. When crosscorrelation is selected you can track first harmonic and/or output coherence function.

EXACT FREQUENCY

When you press the Setup button on new activated Exact frequency line, the following Exact frequency setup window will open:



The Exact frequency supports multiple input channels.

For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

The Exact frequency is a very nice procedure for determining the exact frequency of the signals. Even with lower sampling rates which don't allow methods like period measurement and even with signals with not that clear level crossing this method will work very nicely.

The method is based on finding the best fit to the theoretical sine wave. It can measure millihertz accurate with sampling rates only a few kHz and a low block size. The method works best on the signals close to the pure sine wave. It will search and lock to the highest harmonic component in the signal.

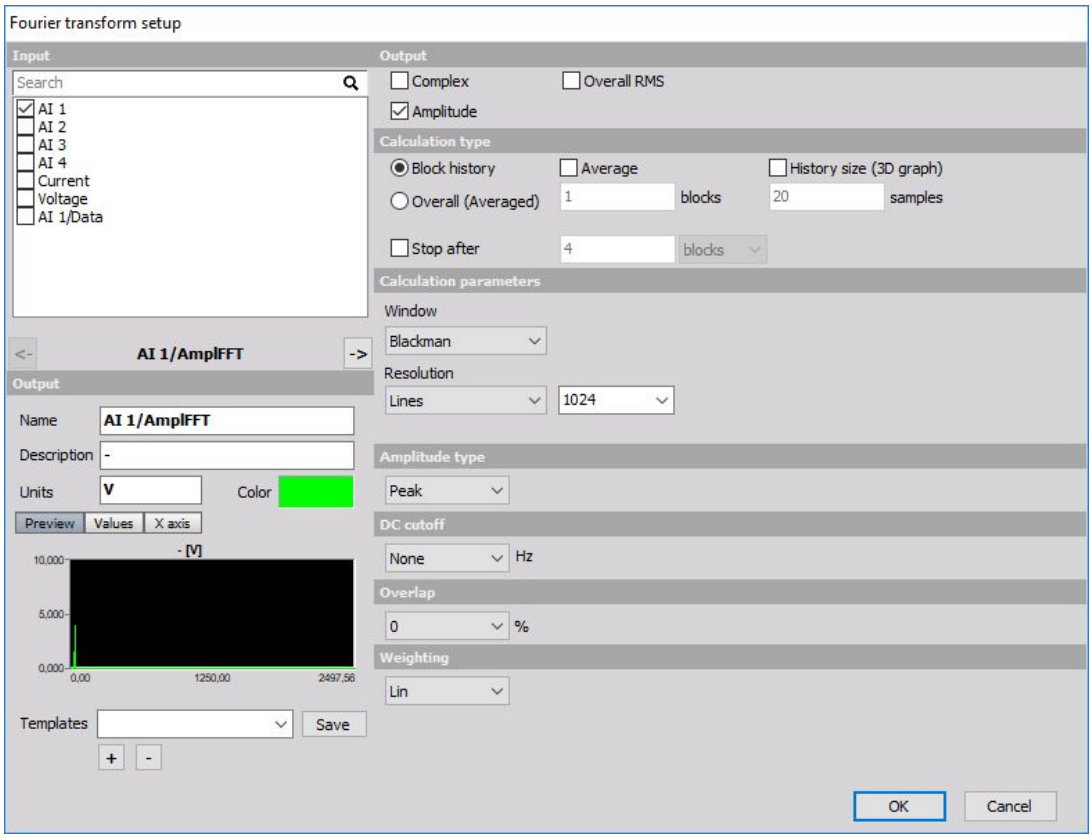
In the right area of the window are - Output settings - Calculation parameters- with:

- **Block size** - Block size defines the calculation interval. The resulting frequency will be updated with this speed, but we need to take care that this interval will be longer than 10 measured periods of the signal.
- **Use amplitude threshold** - Amplitude threshold defines the minimum amplitude where the signal is still calculated. If the RMS of the signal is below this limit, the frequency will not be

calculated at all.

FOURIER TRANSFORM

When you press the Setup button on FFT analysis math item, the following setup window will open:



The output of the FFT analysis could be Complex (real, imaginary), Amplitude, Phase or any combination of those.

Calculation type can be Overall (Averaged), where the result is one spectrum for the entire record. The second option is Block history, where the FFT's are acquired shot by shot and put into the buffer. In this case, they can be observed on the 3D graph. Number of Averages can be also set. When we enter a value of n, then it will calculate the average of n spectrum and put the result in the channel. Manual history count can override the normal settings for the number of shots to be kept in the memory during measurement.

Several Calculation parameters can be also set. One of them is the Window type, which describes the FFT window. There is a good description of the usage of different window functions in the tutorial. By default, we use Blackman, because it is a good compromise between the amplitude error and width of sidebands. Number of Lines defines the size of the FFT. The more lines we choose, the more accurate the frequency will be, but also longer time is needed for the calculation. The number of samples needed is always twice of the size of resulting FFT (number which is set).

IMPORTANT: **ANY number of lines can be entered, not only 2^n. We can manually enter 2000 or 10000 FFT lines. The only drawback is that the calculation will get**

slower if we choose strange numbers (like 12431).

Amplitude type defines what kind of amplitude is put in the channel. We have several options:

Amplitude type	Units	Description
Amplitude (Auto)	V	is the pure signal amplitude
RMS	V rms	is the RMS amplitude, calculated as Amplitude/sqrt(2)
Power	V * V	calculated as RMS value squared
PSD	V * V / Hz	calculated as RMS squared, divided by the line resolution and sqrt(2) - used for checking the noise
RMS SD	V / sqrt(Hz)	calculated as RMS value, divided by the square root of line resolution - also used for checking the nois

DC cutoff option will remove the static offset from the FFT. If it is set to None, it will output FFT as calculated, if not, it will cut the number of lines until frequency which is defined is reached.

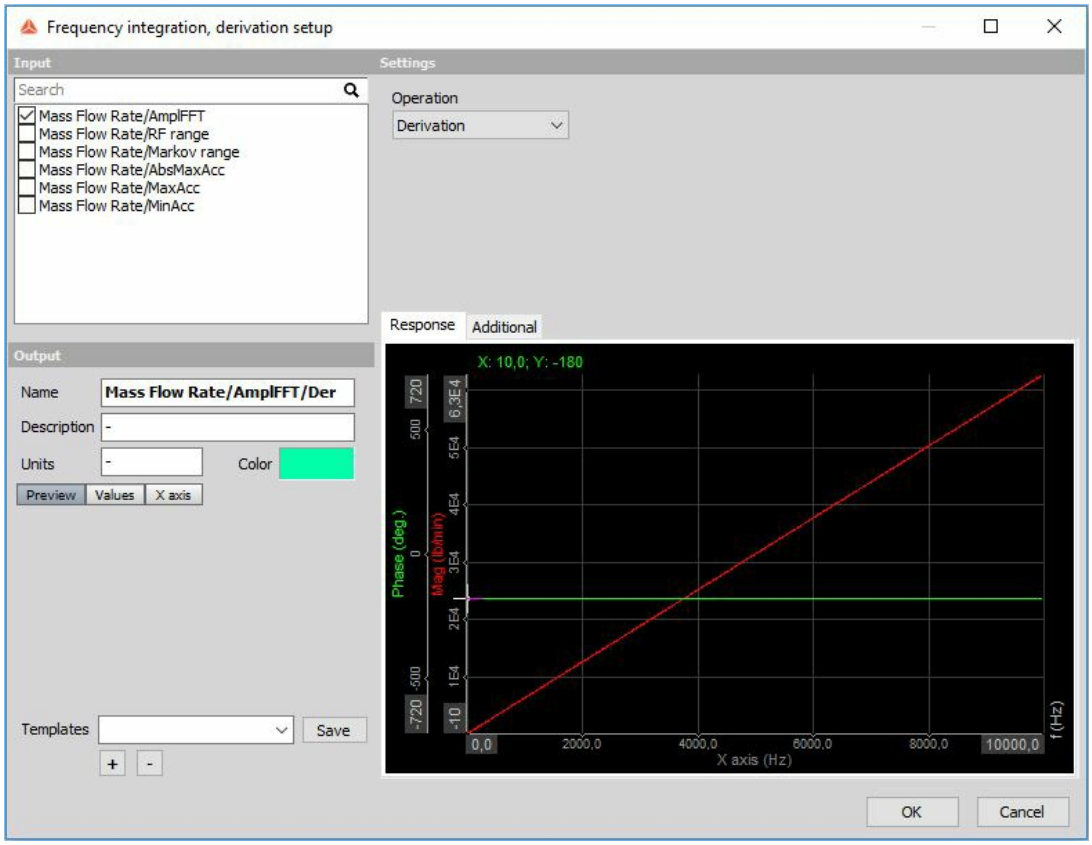
Overlap defines how much will two FFT shots overlap between each other. 50% is enough that all the samples will have the same weight as the result independent of the window which is used.

Weighting defines which sound weighting should be used on the resulting FFT.

As all matrix channels also FFT can be best seen on the 2D graph. Block-based FFT can be also put in the 3D graph for viewing the history.

FREQUENCY INTEGRATION, DERIVATION

When opening a new Frequency integration, derivation math function the following setup window will appear:



Under settings we can select the type of operation:

Integral functions

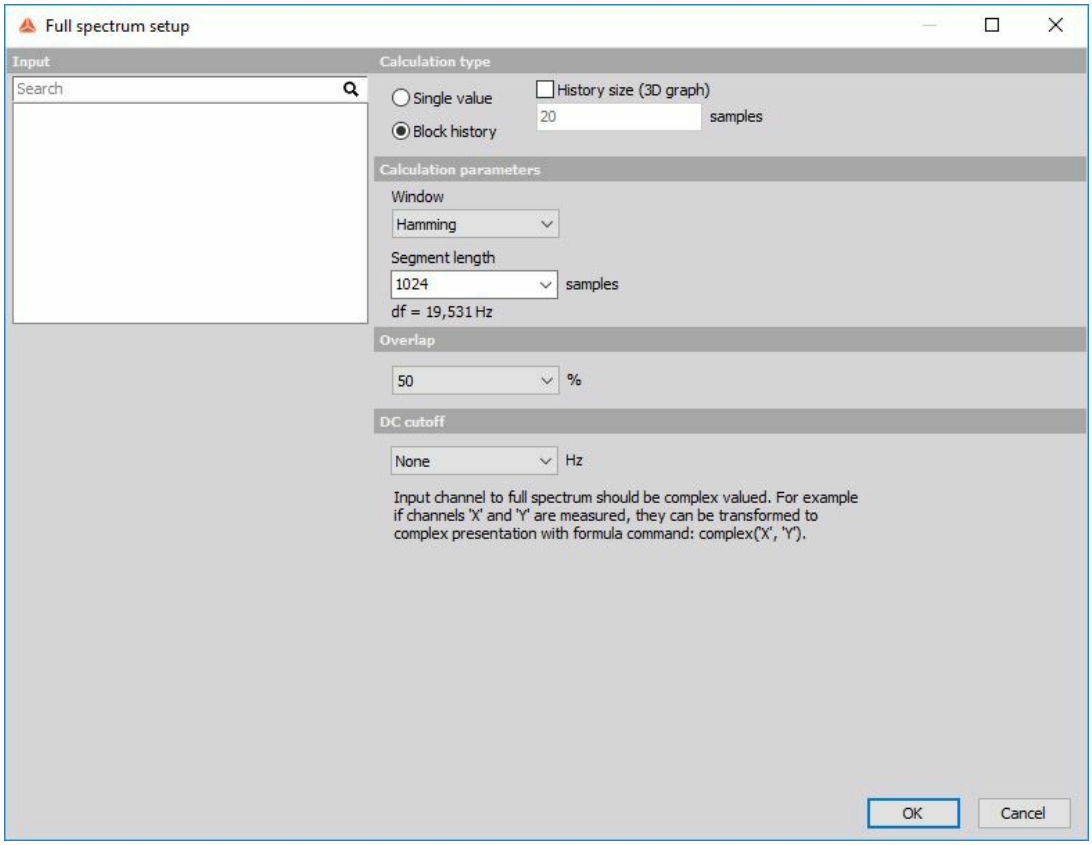
- Integration (used for functions where the carrier of information is the DC value (the offset) or the dynamic part of the signal.)
- Double integration (used for dynamic signals)

Derivative functions

- Derivation
- Double derivation

FULL SPECTRUM

When opening a new Full spectrum math function the following setup window will appear:



Full spectrum (two-sided Fourier) transform changes complex input signal from time to complex frequency domain. Input signal should be in the form of complex synchronous data (re, im). Real and imaginary part are usually gathered from two proximity probes installed in X and Y direction. Traditional spectrum provides no information regarding the direction of motion of individual frequency components on rotating shafts and here is where full spectrum steps in.

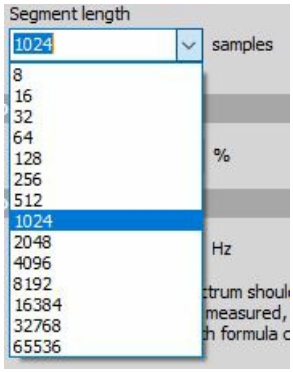
Calculation type

- **Block history** - uses blocks to calculate
- **Single value**

Calculation parameters

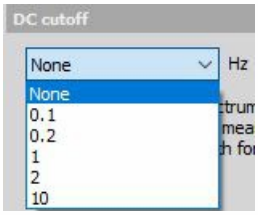
- **Window** (Rectangular, Hanning, Hamming, Flat Top, Triangle and Blackman)

- **Segment length** (number of smaples used)



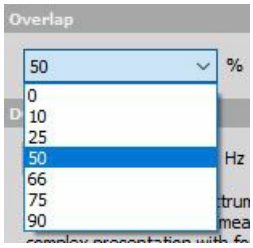
DC cutoff

To remove DC or low-frequency components, select from a drop-down list the DC cutoff filter - lower limit.



Overlap

Overlap defines the percentage of time signal that has already been calculated and it is used again for calculation.

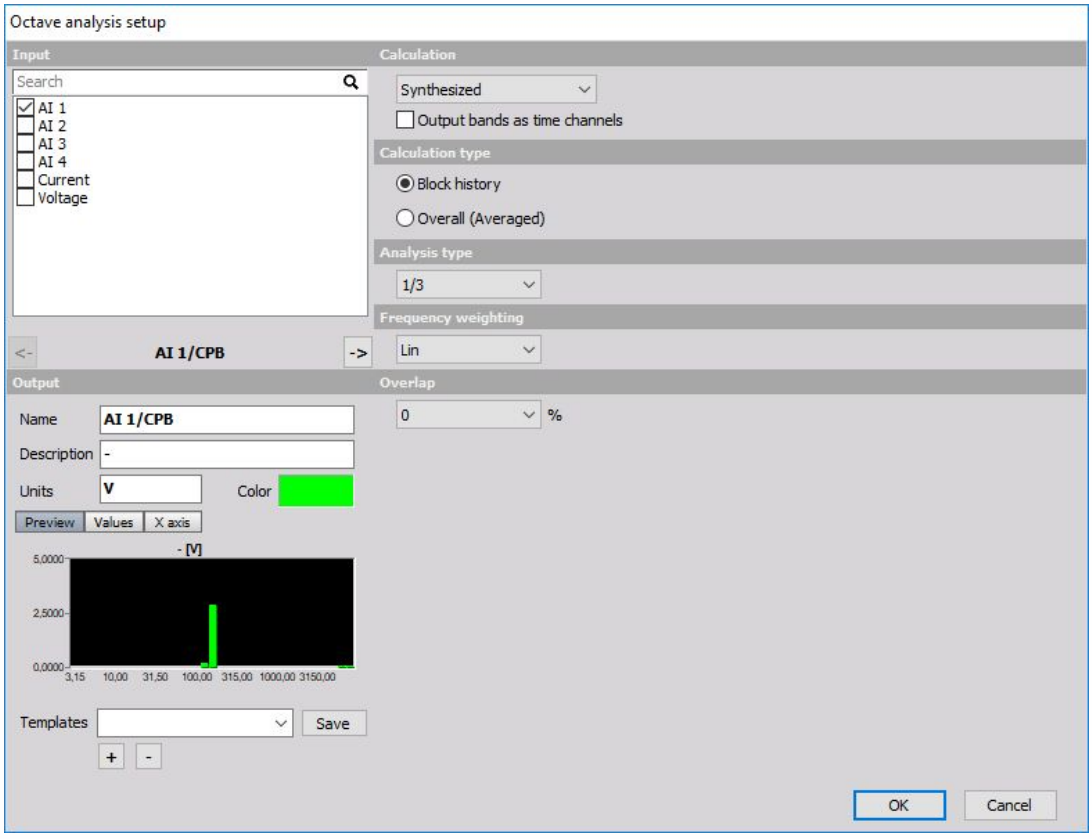


OCTAVE ANALYSIS

As opposed to FFT analysis, which has a specific number of lines per linear frequency (x axis), CPB (constant percentage bandwidth, called also octave) has a specific number of lines if logarithmic frequency x axis is used. Therefore lower frequencies have more lines than higher ones.

CPB analysis is traditionally used in sound and vibration field.

When you press the Setup button on CPB analysis math item, the following setup window will open:



Calculation principle can be selected as a synthesized or true octave. Synthesized CPB is calculated using FFT as the base for calculating octave bands. Therefore it is updated only with every FFT being calculated.

True octave uses filters sets as in old analog (very expensive) octave analyzers. It uses more computing power and the result if only average spectrum over the entire run is needed is virtually the same. But when we observe that in real time, the difference is like night and day. We really see the dynamic behavior of the input data.

Calculation type can be Overall (Averaged), where the result is one spectrum for the entire record.

The second option is Block history, where the CPBs are acquired shot by shot and put into the buffer. In this case, they can be observed on the 3D graph. The number of averages can be also set. When we enter a value of n, then it will calculate the average of n spectrum and put the result in the channel. Manual history count can override the normal settings for the number of shots to

be kept in the memory during measurement.

Analysis type defines the number of bands within one octave. One octave means that the next center frequency of the band is twice the value of the current one. If we have 100 Hz, next octave band will be at 200 Hz. Then these octaves are further divided by a number defined in the analysis type field. 1/3 octave will have three bands per one octave and so on. So the higher the number is, more precise frequencies will be possible to observe and more calculation power will be used, especially with true octave.

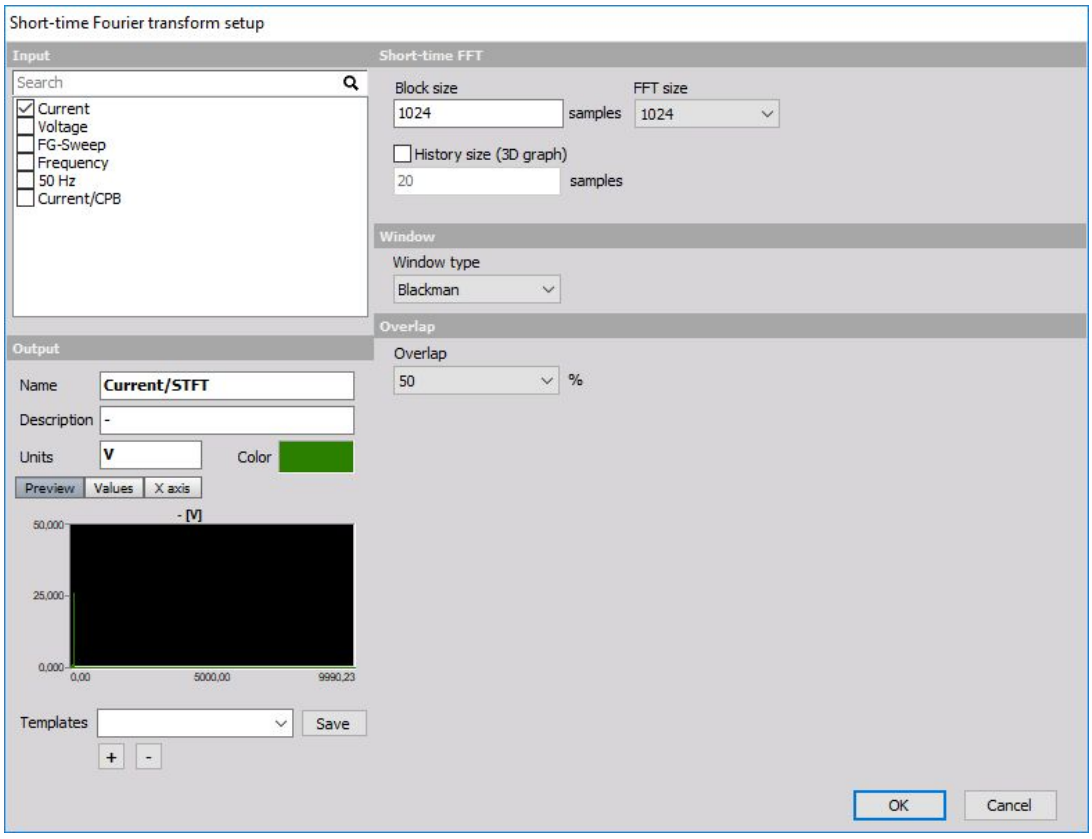
Two additional fields must be defined if the true octave is used. First is Exponential averaging time, which defines the speed that the averaging filter works after each band is calculated. The three values correspond to the noise standard values (0.035 is IMPULSE, 0.125 sec is FAST, 1 sec is SLOW), other values can be also freely entered.

The second field is the Lower band frequency, which defines what will be the lowest value for calculation.

As all matrix channels also CPB can be best seen on the 2D graph. Block-based CPB can be also put in the 3D graph for viewing the history.

SHORT-TIME FOURIER TRANSFORM

Short time Fourier transformation is the procedure which calculates more lines than the normal FFT. This is achieved by having a smaller real block size of data and larger FFT size. Real data is windowed and zero padded and then FFT is calculated. With this procedure, we can calculate more FFT's for the same time base. It is nice for use with fast transients.



- **Block size** - defines the number of real data samples to be taken for the calculating FFT.
- **FFT size** - defines the number of resulting lines and with that the ratio between real and zero padded lines.
- **Window type** - describes the FFT window to be used. There is a good description of the usage of different window functions in the tutorial. By default, we use Blackman, because it is a good compromise between the amplitude error and width of sidebands.
- **Overlap** - defines how much will two FFT shots overlap between each other. 50% is enough that all the samples will have the same weight as the result independent of the window which is used.

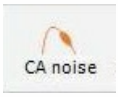
STFT as can be viewed in 2D and 3D graph.

MACHINERY DIAGNOSTICS

In Machine diagnostics we can choose from following mathematical functions:

- Angle sensor math

- Cumbustion noise



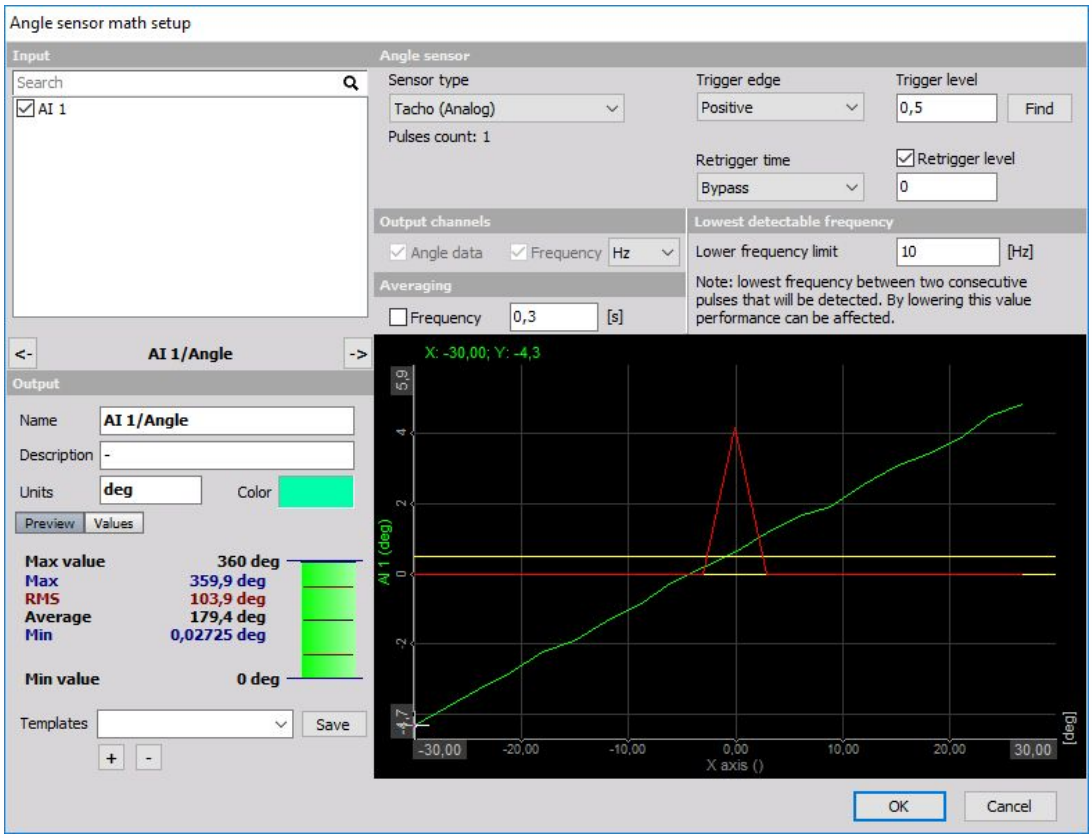
- Envelope detection

- Sine processing (COLA)

- Tracking filter

ANGLE SENSOR MATH

When you press the Setup button on new activated Angle sensor line, the following Angle sensor setup window will open:

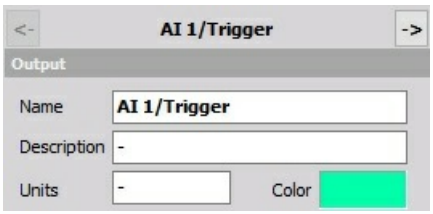


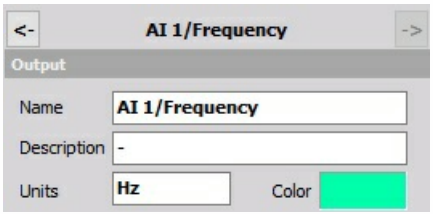
For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

The input for the angle sensor math can be any analog trigger signal (like tacho - 1 time per revolution or automotive 60-2 sensor). In both cases, we need some mathematics to calculate the current angle and the frequency of the incoming signal. In both cases, the signal needs to be connected to the analog input of the instrument. Then we need to select this signal, select the Sensor type from the list of the available sensor or by defining a new sensor type in the counter sensor form. Next, we can simply press Find function to determine the trigger levels. In most cases, this function works already, but in some cases, we need to define the trigger levels manually by settings the Trigger level and Re-trigger level input field. Next, we can define the Re-trigger time. This is a hold off in which the signal is not checked for triggers. This is useful to prevent trigger glitches. The graph below will show the signal levels for easier determination if the trigger is working fine.

The output from this math module is the:

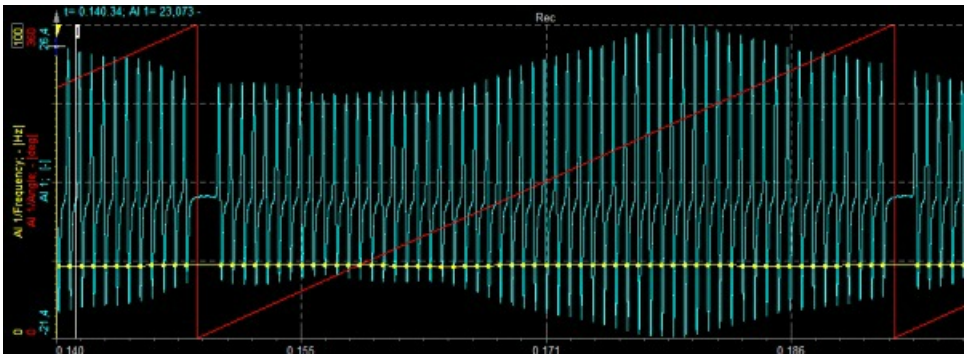
- **Angle**: Angle channel as we can see from the picture above and Trigger channel
- **Frequency**: Frequency channel





These channels can be selected by checking the appropriated box in the Output channels section.

This angle can be used in basically all advanced Dewesoft modules like CA, order tracking and torsional vibration or standalone just to get the signal frequency.

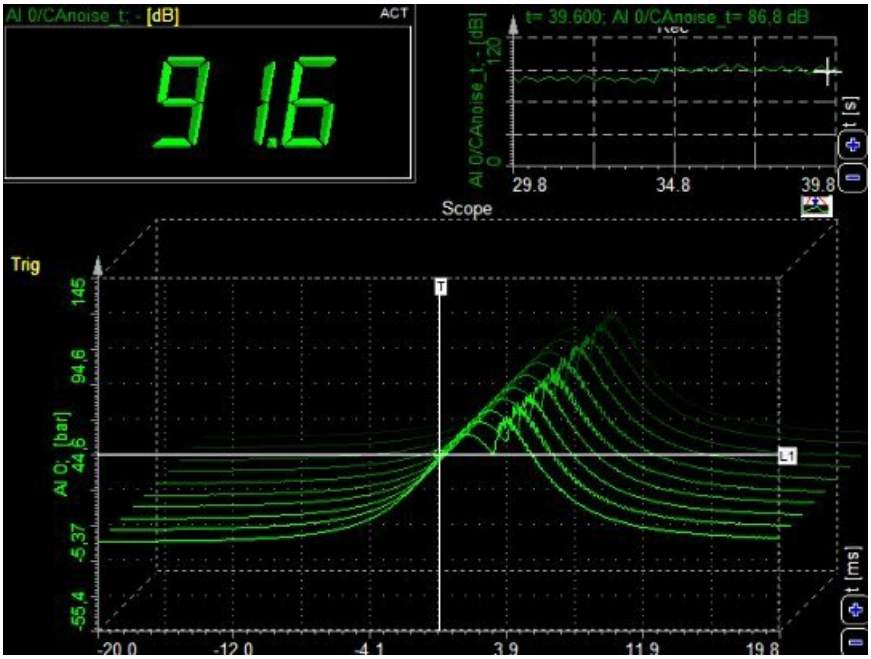


The nice feature of this module is that it implements calculation delay. For sensor like 60-2 it is not possible to determine the angle until the first gap is recognized. For that time the angle sensor math holds all the calculations and with the first gap also calculates the data for the first turn before the gap. This is crucial for some applications like the cold start of the engine.

Please be aware that the quality and the resolution of the data depend on the analog sample rate. If the sample rate is low, the output (especially the signal frequency) will not be exact. So it is recommended to use higher sampling rates (like 50 kHz) for these calculations. The results can be improved with the usage of Averaging on Frequency, which is defined in the setup screen. This will average the frequency results in the time period which we define.

COMBUSTION NOISE

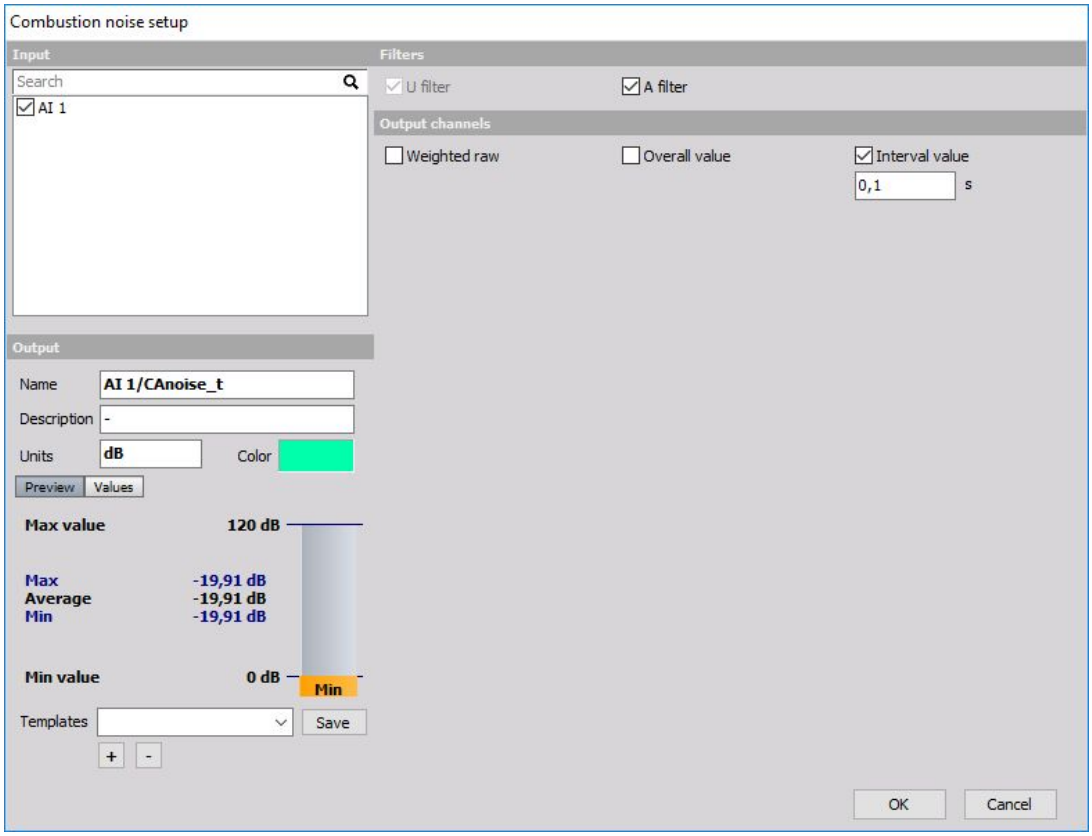
Combustion analyser noise description:



CA noise is a special calculation which can judge the noise resulting from the engine, based on the measurement of the cylinder pressure. The example above shows the typical modern cylinder pressure curve. The CA noise gives the approximate value of what kind of noise this engine will produce.

Settings

When you press the Setup button on activated new CA NOISE line, the CA NOISE setup window will open:

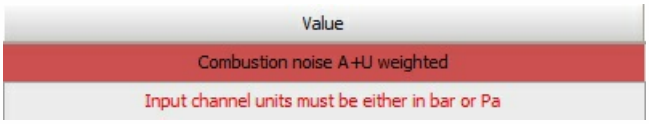


The CA NOISE supports multiple input channels.

For detailed information about basic settings of the input and output channels see -> Setup screen and basic operation.

CA noise is calculated with a special so-called U weighting filter. This is basically a band pass filter of the cylinder pressure. Additionally, we can use the A filter (human hearing filter) to determine the human perception to the noise made by the engine.

The scaling of the input pressure must be either in 'Pa' or 'bar' in order that the math works correctly afterward, otherwise error message appear on CA NOISE line in Value column:



Outputs

There are three types of output channels available:

- **Weighted raw** - the output is the time curve with applied U (and optionally A) weighting; this

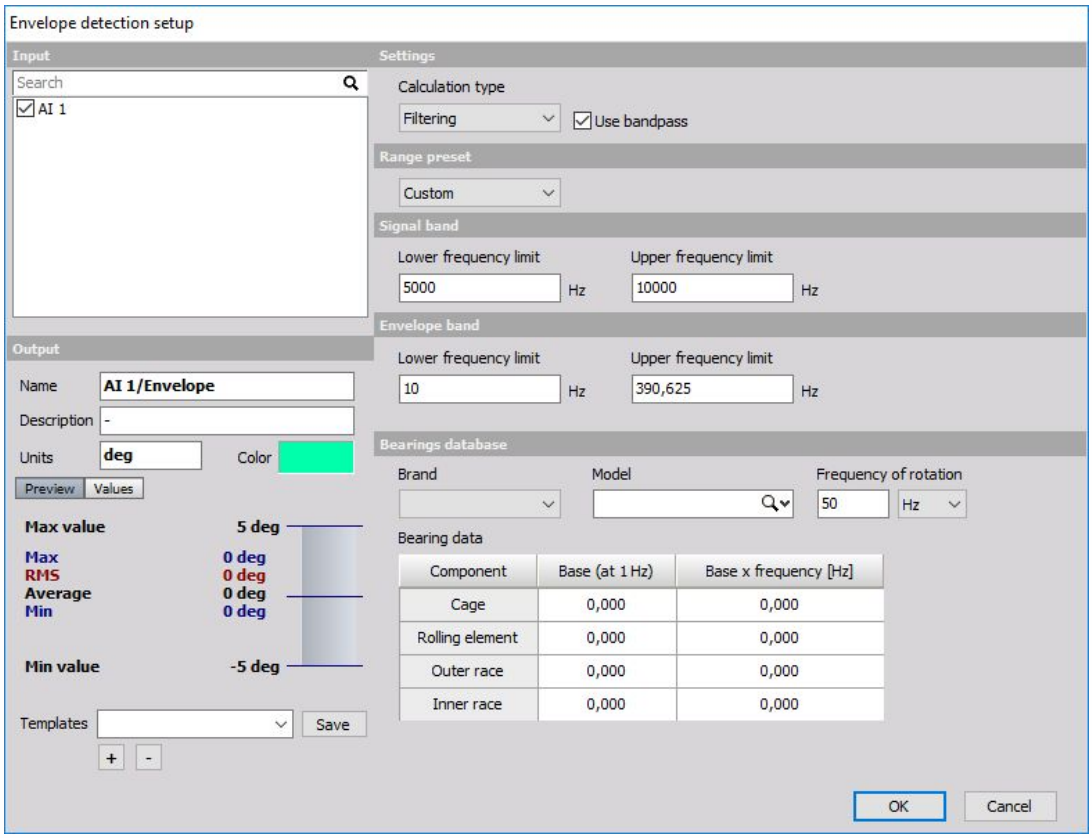
channel might be used in the Sound Level mathematics for advanced sound analysis.

- **Overall value** - this channel will have one value (single value) at the end of the measurement
- **Interval value** - this channel will have a value for each interval, which we can freely define in seconds.

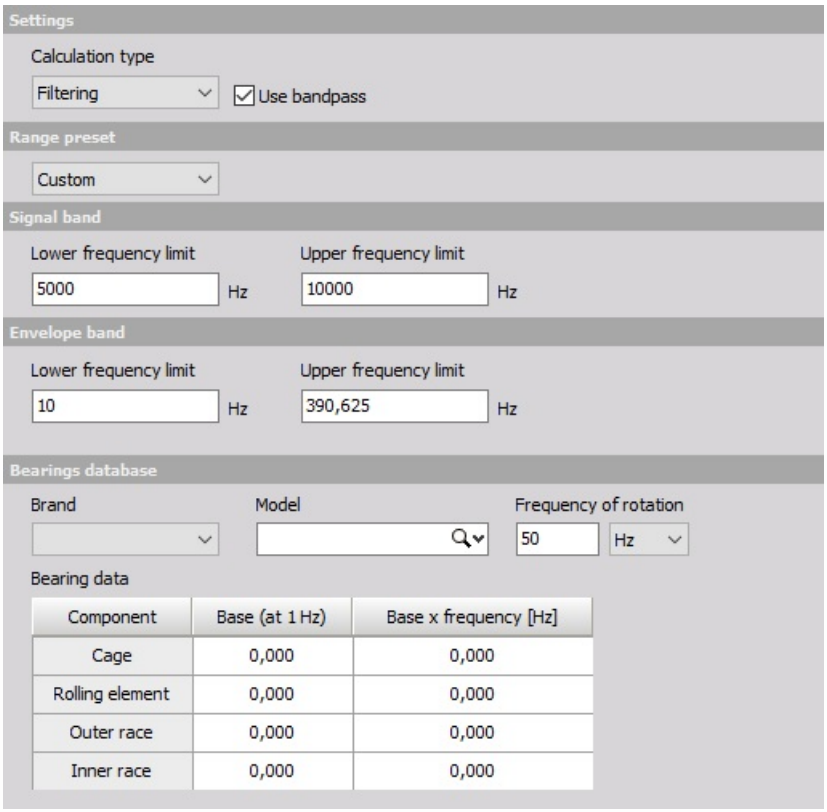
Please note that custom CA noise or CA noise on the external clock can be also calculated using the FFT filter.

ENVELOPE DETECTION

When you press the Setup button or open a new math Envelope detection line, the following setup window will open:



Envelope detection is a procedure for early detecting of faults on ball bearings. Envelope detector has several stages and for each stage, the parameters must be set:



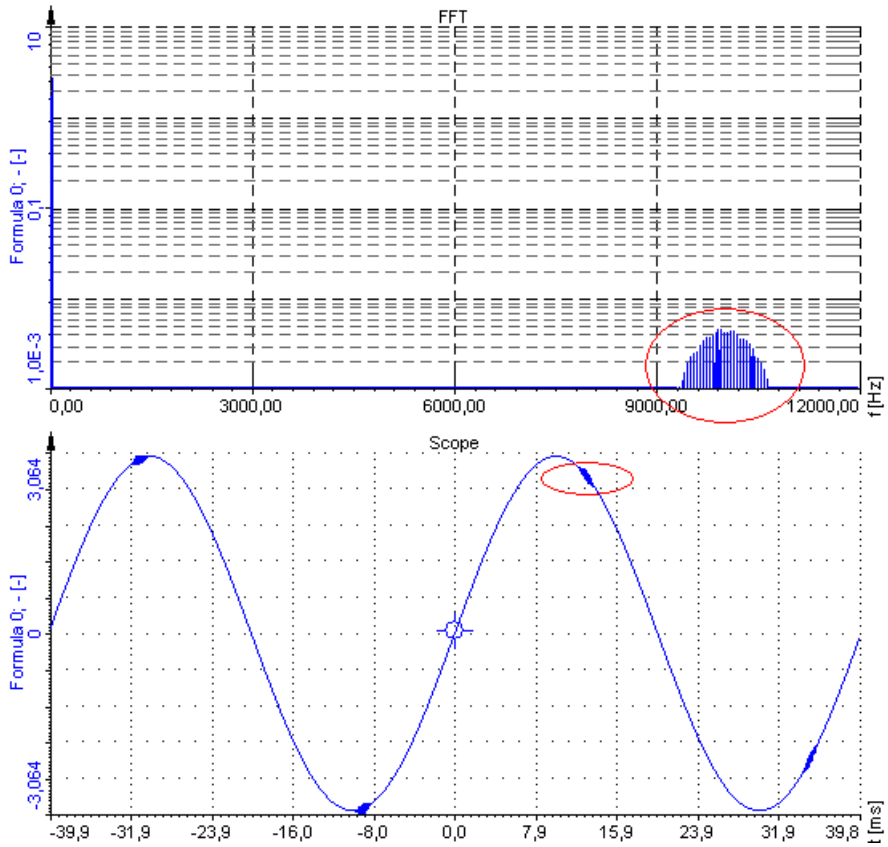
Calculation type defines the principle of calculation.

- **Filtering** - uses filter procedure for envelope calculation. Filtering is a standard procedure for calculating envelope used also in other implementations.
- **Peak detection** - uses the procedure of detecting peak values in the signal. Peak detection is a procedure which calculates amplitudes more exact than filtering.
- **Bandpass** - checkbox enables or disables the first stage of calculation - band pass filtering. |

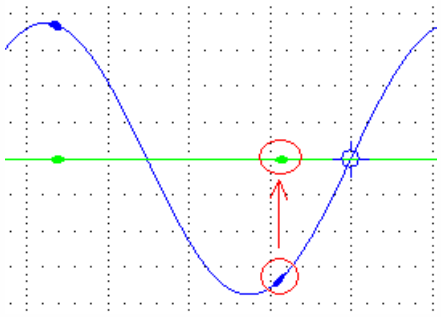
Acceleration sensor measures the entire frequency range and acquires unbalance, misalignment and other faults on the machine. Ball bearing errors have very low energy and therefore is a small contribution in the entire frequency spectrum.

When an error of the ball bearing occurs, it will produce ringing with a frequency which corresponds to its natural frequency. This ringing will repeat each time when a damaged part of the ball hits the ring or vice versa. We also have to know that inner ring, outer ring, cage and balls have different typical repeating frequency depending on the geometry of the bearing and the rotational frequency.

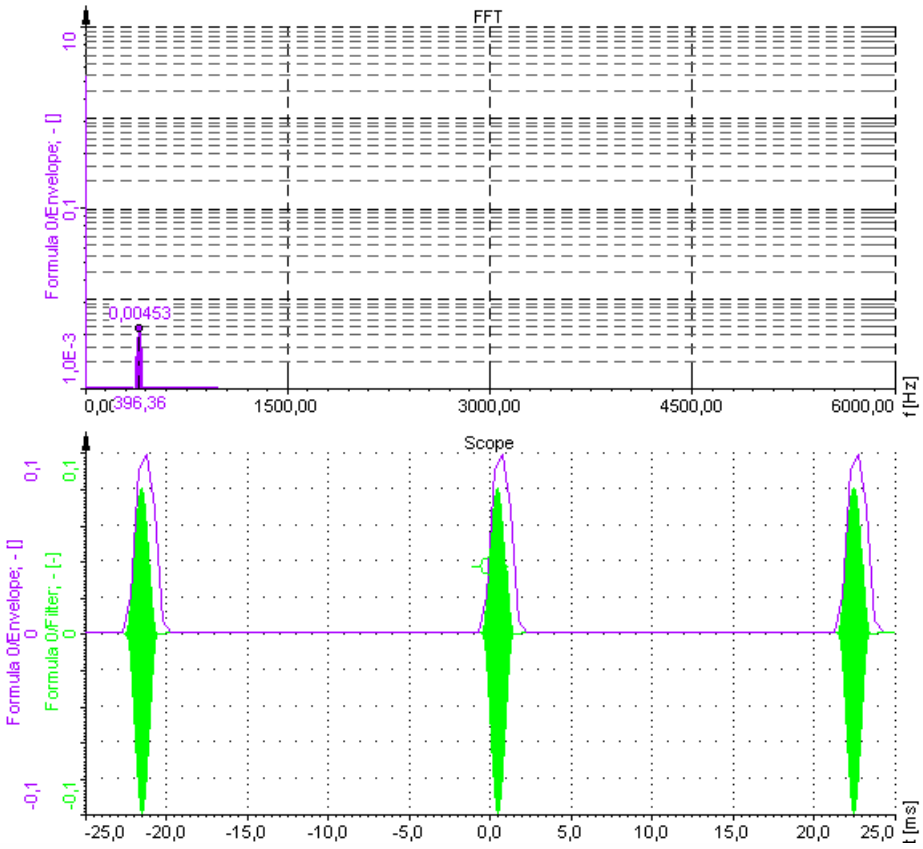
To only focus on these high frequencies of the ringing, we have to look at the original frequency spectrum. We have generated a sine wave which has a small 10 kHz rings on top. In the frequency domain, we don't see the frequency of repeating ringing, but only a major sine wave (could come from unbalancing) and very high frequency coming from the bearing.



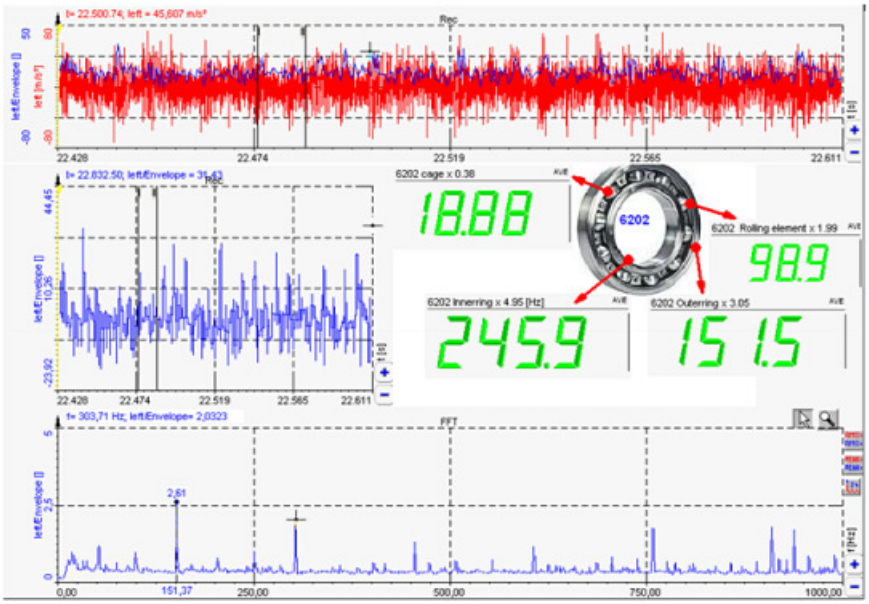
Bandpass filtering in the envelope detector must be set to remove all components except ringing of the ball bearing. This can be usually found around 10 kHz. In the example we have set the lower frequency limit to 6 kHz and upper limit to 12 kHz in order to get all the energy. The signal after filtering should look in the picture on the right.



Only high frequency remains, but we still don't see the main low frequency with which the rings are repeating. Therefore we have to apply envelope to the signal. Envelope will draw a curve around the peaks of the signal, producing only positive part of the data. To get a correct amplitude, we have to choose the Envelope band frequency. Bearings usually have typical frequencies up to 500 Hz and we also might want to Remove DC component in order to see a nice frequency spectrum without large DC value coming from DC offset. After this filter the signal looks like this and frequency spectrum of the envelope signal reveals the frequency of hits.



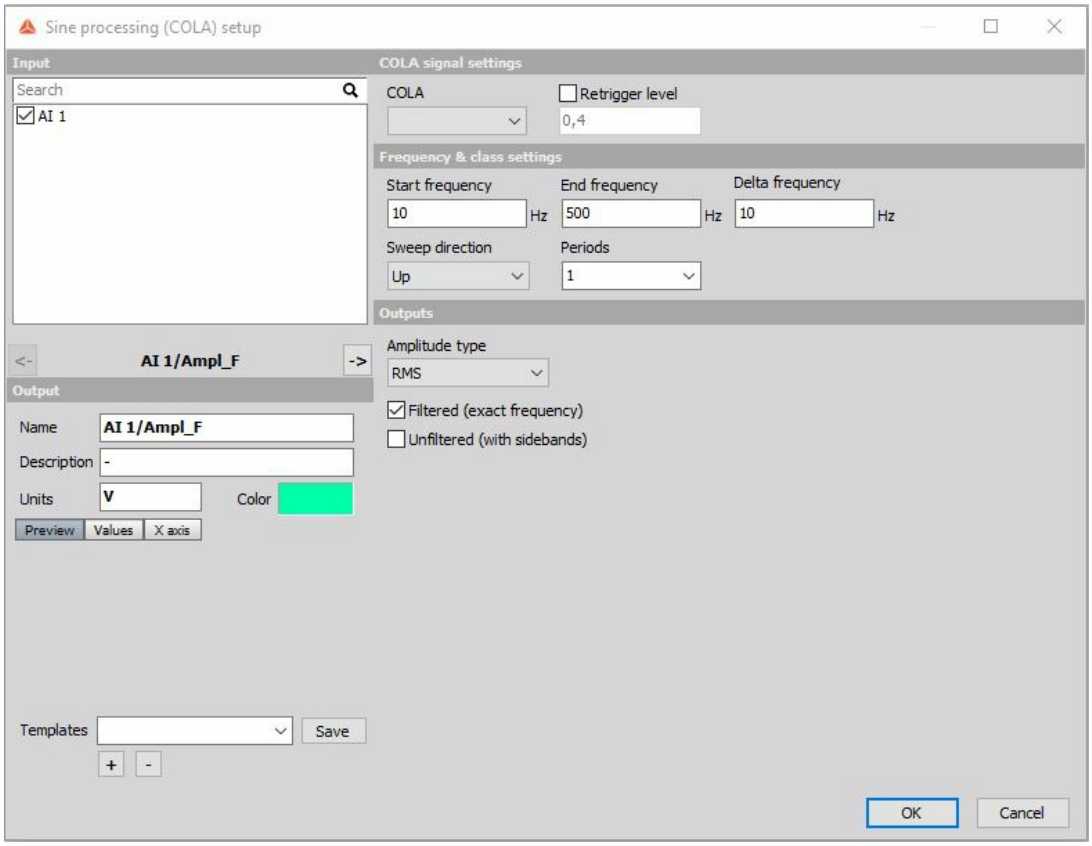
This was a simulated case to see the math procedure behind calculation. In reality the signal will look like this. Not much to see from the time signal, but with calculation of typical frequencies we can see that the outer ring frequency is clearly shown in the FFT of the envelope signal.



The following picture shows the typical damage of the outer ring of the large bearing (courtesy of Kalmer d.o.o. Trbovlje).

SINE PROCESSING (COLA)

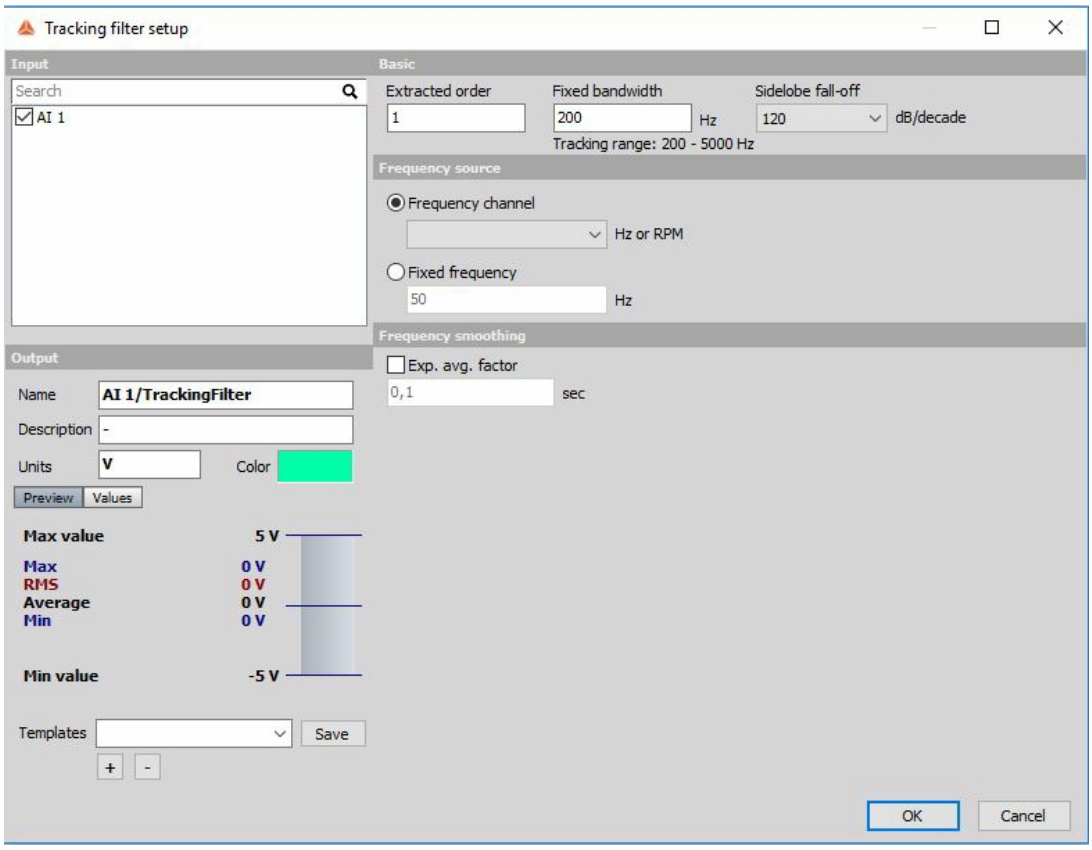
When opening a new Sine processing math function the following setup window will appear:



Sine processing function is a one-line DFT based amplitude and phase extraction procedure. Is uses COLA signal to calculate instantaneous frequency and then it extracts amplitude and phase at that frequency. Amplitude and phase are put into a histogram.

TRACKING FILTER

When opening a new Frequency integration, derivation math function the following setup window will appear:



Tracking filter has a center frequency that is always tuned to the current drive frequency. This allows att other signals to be rejected from measurement and control.

Tracking filters greatly reduce the noise and harmonic signals above and below the signal's center frequency. The narrower the bandwidth of the tracking filter. the more the rejection of harmonics and noise outside of the filter's bandwidth.

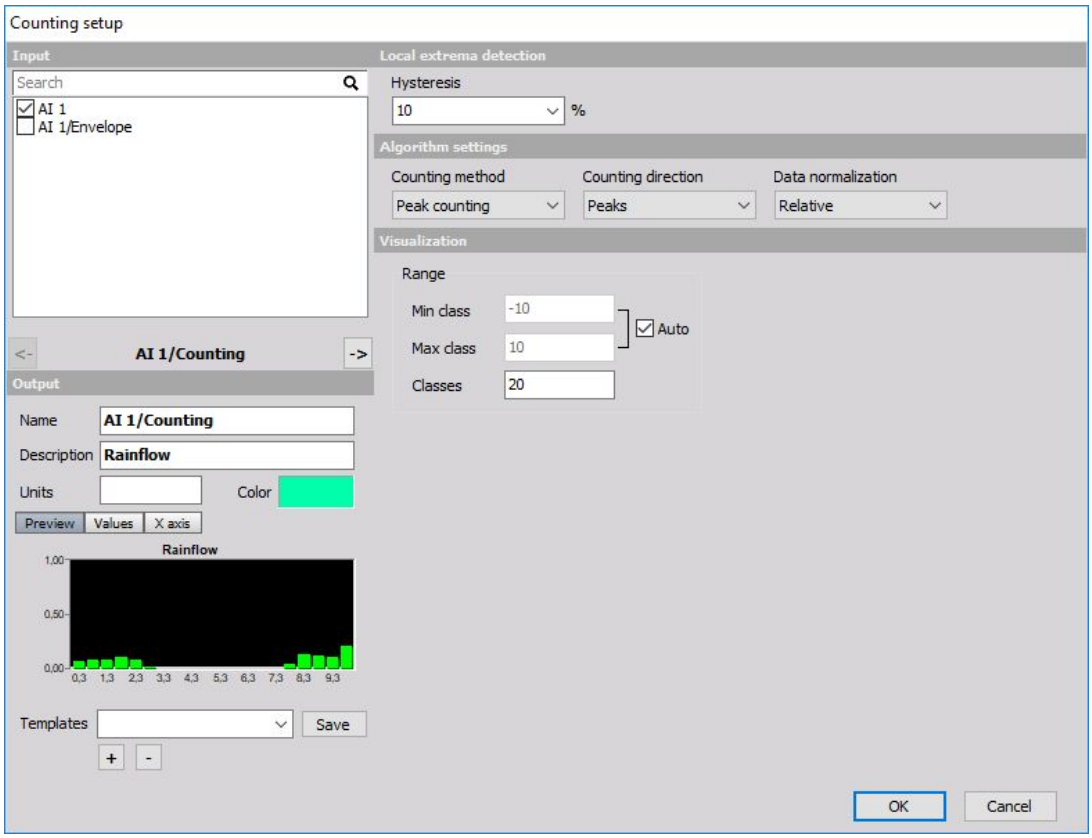
COUNTING PROCEDURES

In Counting procedures we can choose from following mathematical functions:

Counting

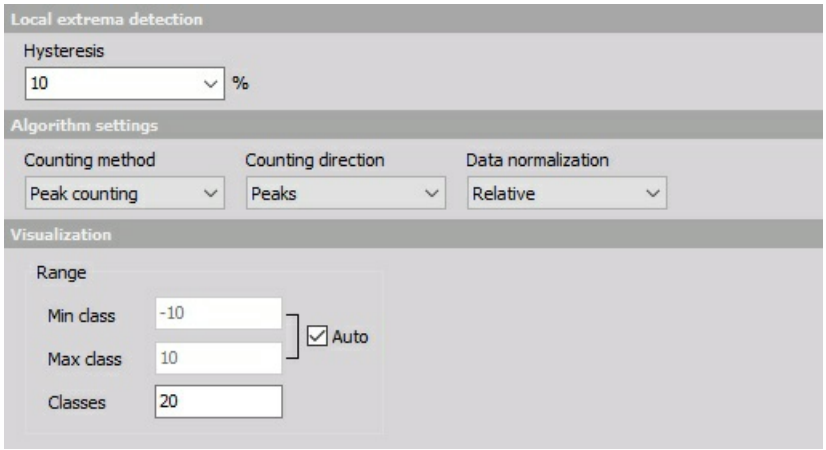
COUNTING

When you press the Setup button or open a new math Counting line, the following setup window will open:



Counting is the standard procedure to reduce the amount of data for analysis. It is for example used in the application of road load data collection where we have some static load and on top dynamic load.

The only interesting values are the height of load cycles and the average static load of that cycle. For that purpose, the rain-flow analysis is made.



The counting procedure counts the peaks and valley of the signal. The Hysteresis is defined in the percentage of the class width. This prevents too many false counts if the signal is noisy.

There are three possible output values (Normalization):

- **Absolute** - it outputs the number of cycles as a value - values will increase with time.
- **Relative** - it outputs the number of cycles normalized to the absolute number of cycles - the sum of all values will be always 1.
- **Relative [%]** - it outputs the number of cycles normalized to the absolute number of cycles multiplied with 100 - the sum of all values will be always 100.

There are several Counting Method to choose from:

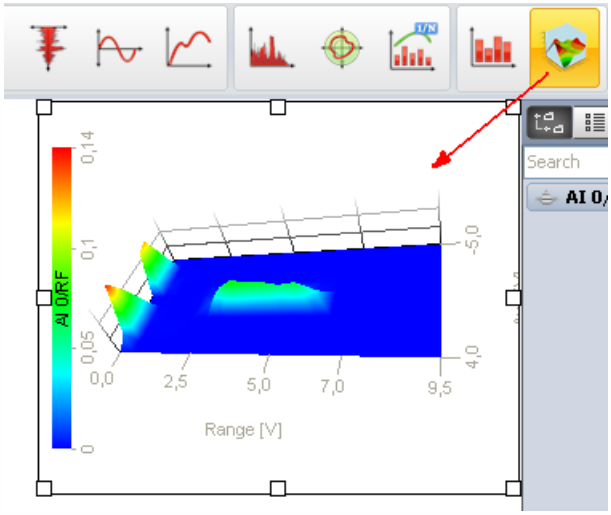
- **Peak counting** - counts the number of peaks in the signal in certain classes (for this option we can define to count peaks, valleys or both).
- **Range counting** - counts the range between successive peaks and valley pairs. Ranges are positive when the slope between peaks and valleys is positive. We can choose either to count positive, negative directions or both).
- **Level crossing** - counts the number of times when that signal crosses various levels.

First, four options output 2D matrix while the last one outputs 3D matrix.

For all options we have to define the number of classes for Average value, minimum - Min and the maximum - Max or we choose to define the Min, max value from the range of input parameters.

For 2D rainflow we have to define these parameters also for the third axis which counts Peak-peak value.

The peak, range, level and 1D rainflow can be best seen in 2D graph while we need to use 3D graph for rainflow.



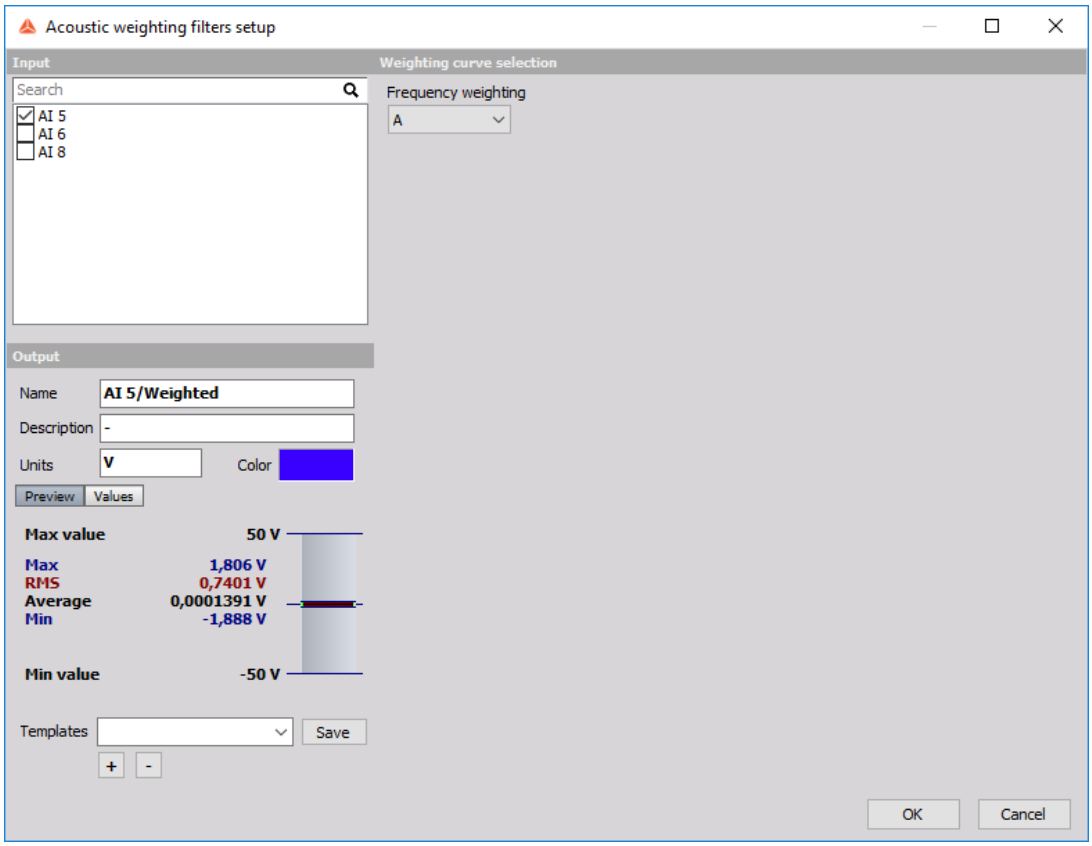
ACOUSTICS

In Acoustics we can choose from following mathematical functions:

Weighting filters

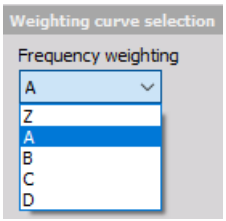
WEIGHTING FILTERS

When you press the Setup button or open a new math Acoustic weighting filter, the following setup window will open:



Weighing curve selection

To add a weighting curve select the required filter. The filter will be applied to the selected Input channels.



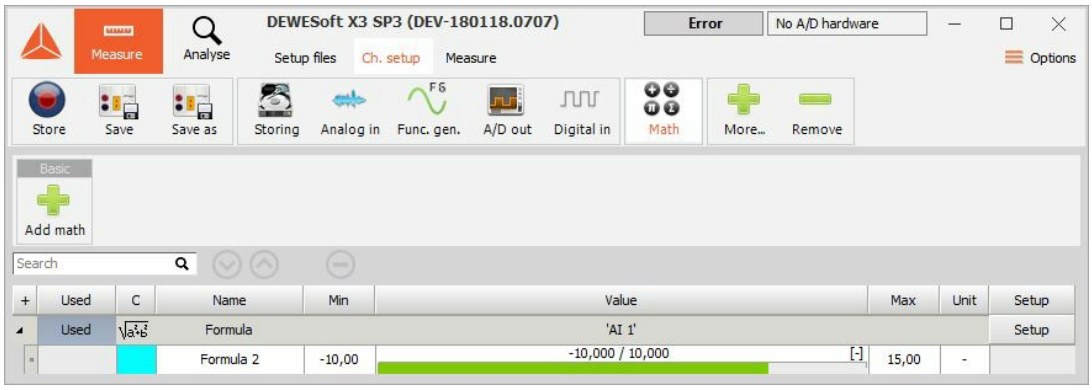
FORMULA AND SCRIPTING

Enter topic text here.

FORMULA EDITOR

The Dewesoft formula mathematics enables the user to use a formula parser with algebraic, logic, trigonometric and other functions to recalculate one or several input channels.

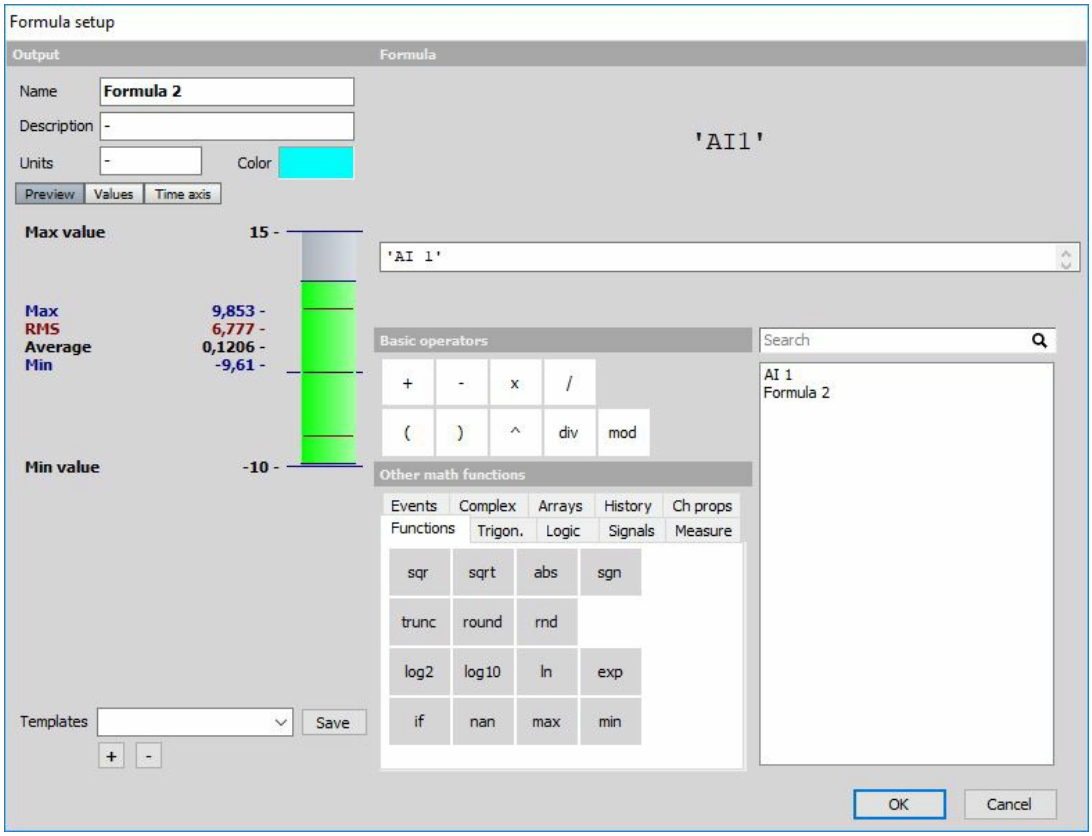
When you choose Dewesoft Math Formula, the icon below is displayed:



For detailed information about basic settings see -> Setup screen and basic operation.

Formula setup

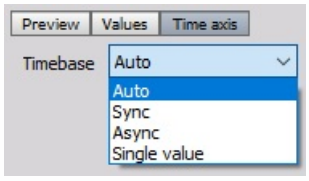
When you press the Setup button in SETUP column of Math setup screen, the window will appear depending on the selected module.



The left side of the display is standard display setup, described in Setup screen -> see -> Setup screen and basic operation. The only difference is the sample rate divider (see below), which can be defined for the output channel of the formula. The right side shows the formula editor.

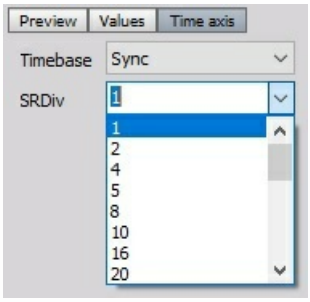
Sample rate divider

SR div is the sample rate divider of the channel with several options:

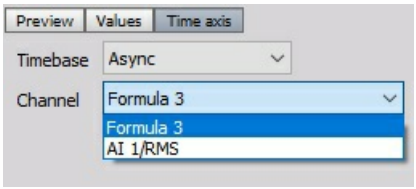


- **Auto** - The sample rate divider will be assigned automatically. For example:
- If the input is a synchronous channel with Sr div of 4, the resulting channel will also have a sample rate divider of four.
- If the input is an asynchronous channel, the output will also be an asynchronous channel.
- If there are more input channels, the output will be matched to the input channel with the highest rate.

- **Sync** - This function will override the logic of automatic Sr dDiv assignment and will set the sample rate divider manually. This is useful to make interpolation of input asynchronous channel because each asynchronous channel is interpolated before it is calculated. This gives a smoother curve and more precise results.
- **Async** - This function enables you to set the sample rate of the mathematical formula to the same as the selected asynchronous channel.
- **Single value** - The result of mathematics is only a single value. Single value is a special type of Dewesoft channel where the result is not a set of values with a time stamp, but always just one value. This is useful for calculating cumulative values. **Sync** (From the list, we can select Manual values from 1 to 500).

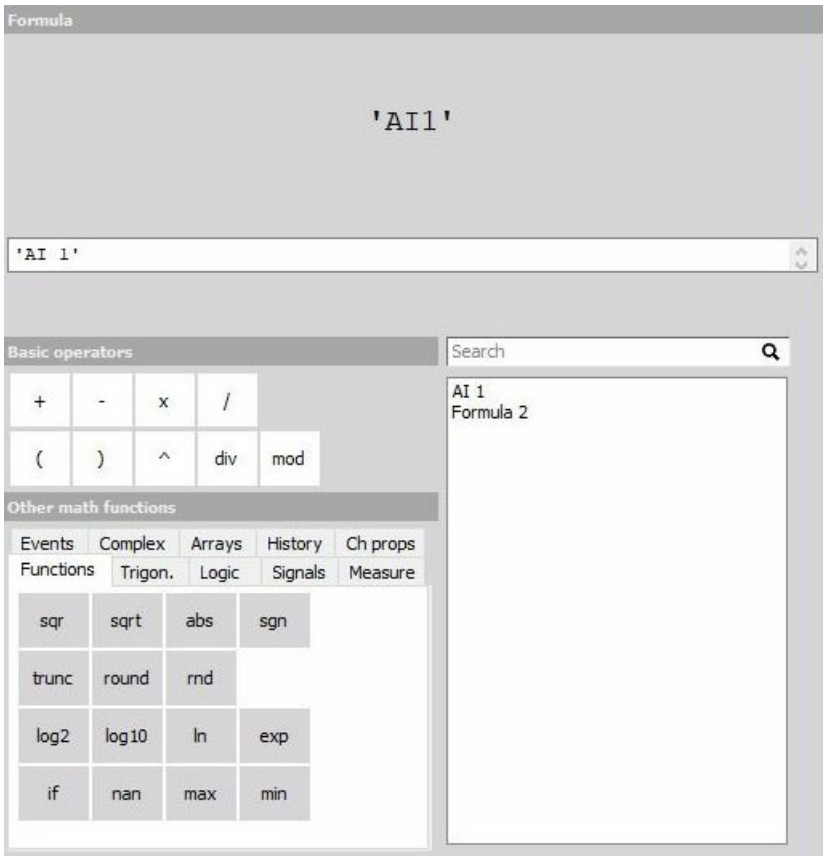


Async (In channel the drop-down menu, you can select the appropriate trigger).



Formula editor

The right side, "Formula" part of the Formula setup window contains:



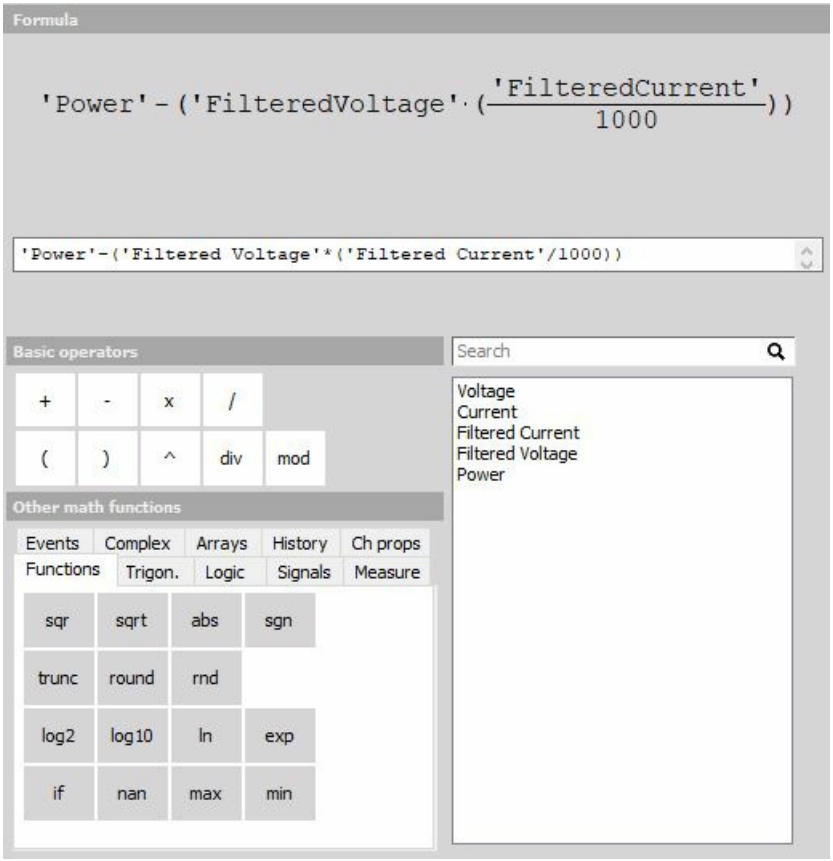
- a formula preview at the top
- the field to enter the formula directly
- the mathematical functions and operators with:
- Basics operators
- Other math functions (Functions, Logic, Trigonometry, Signals, Measure, Events)
- list of all available channels: analog channels, variables, constants

To enter a channel in the formula just:

- double-click on the channel (in list of all channels)
- single-click on the function (in 'Other math functions')
- single-click on the operator (In 'Basic operators')

Dewesoft allows doing mathematics on mathematics in any combination. The Formula interpreter allows also enter complex formulas.

If you do a mathematical operation on two channels with the different sampling rate, the two channels will be synchronized to each other. The values of the slower sampled signal will be repeated for the calculation with the faster-sampled signal until a new value is available.

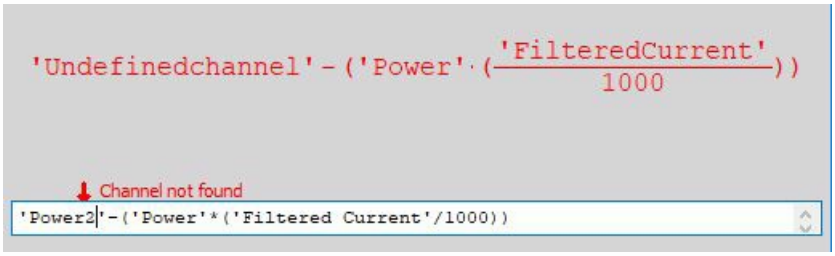


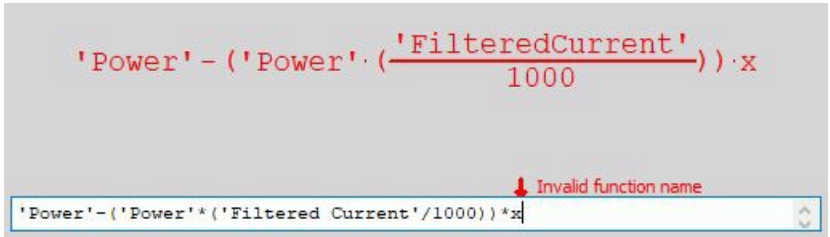
All available channels are displayed in the selection list. For Dewesoft it makes no difference where the data are coming from: analog input, CAN, mathematics, GPS,... - even different sampling rates are no problem.

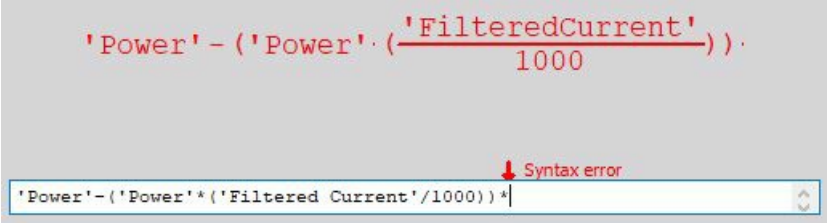
Dewesoft allows different types of mathematics, which are grouped in six tabs. Please learn more about these functions in -> Operators and mathematical function.

If you make an error, you will get an indicator which shows you immediately where the error is located and error description.

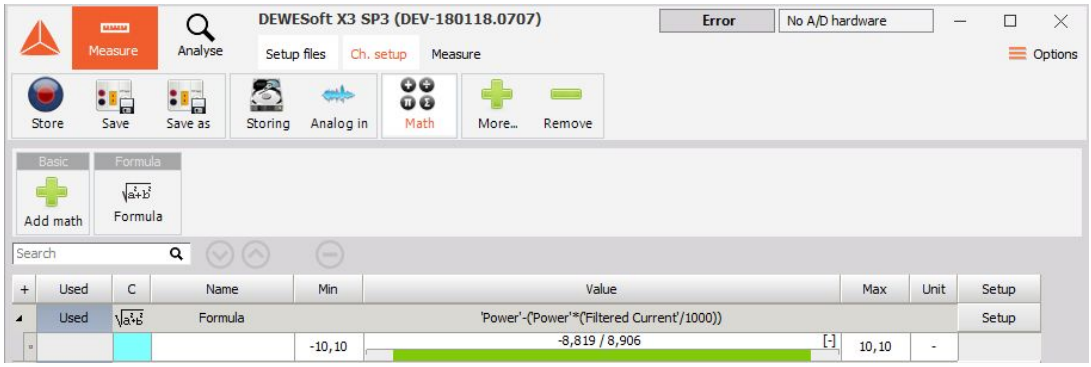
Some error examples:







When you have entered your formula, press the OK button to leave the setup. Now a new line has been added to the mathematics overview screen:



See next topic to have a complete list of available operators and mathematical functions.

OPERATORS AND MATHEMATICAL FUNCTION

Dewesoft beside General mathematics in Basics operators allows different types of functions, which are grouped in six tabs in Other math functions section of Formula editor:

- Functions
- Logic
- Measure
- Complex
- History
- Trigonometry
- Signals
- Events
- Arrays
- Channel properties

Dewesoft supports array channels -> see -> Array operations in formula.

For information about interesting use of mathematics function see also:

- Example of logic functions
- Example of measure functions
- Example of input events
- Example of stopwatch
- Example of signals

Basics operators

Dewesoft offers following general mathematical functions - Basic operators section:

General mathematics

Function	Syntax	Description
+	expression1 + expression2	expression1 plus expression2
-	expression1 - expression2	expression1 minus expression2
×	expression1 * expression2	multiplies expression1 by expression2
/	expression1 / expression2	divides expression1 by expression2
( )	(expression)	brackets expression
^	expression1 ^ expression2	raises expression1 to the power of expression2
DIV	expression1 DIV expression2	integer part of division of expression1 by expression2
MOD	expression1 MOD expression2	rest of a division of expression1 by expression2

DIV integer division (Delivers the integer part of the division)

- 420 DIV 720 = 0
- 740 DIV 720 = 1

MOD (Delivers the rest of a division)

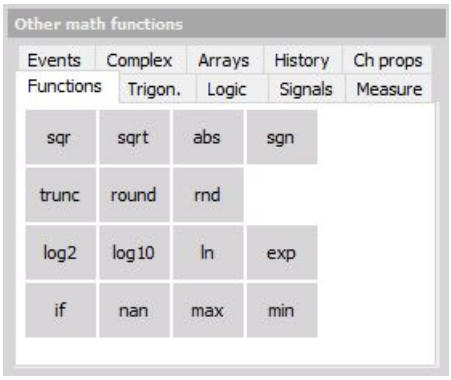
- 420 MOD 720 = 420
- 740 MOD 720 = 20

Mathematical functions types

All functions have a short hint. When we move the mouse over the icons, we get an explanation of the function and its syntax.

Functions

The Functions tab contains additional mathematical functions.



Function	Syntax	Description
SQR	SQR(expression)	square
SQRT	SQRT(expression)	square root
ABS	ABS(expression)	absolute value of expression
SGN	SGN(expression)	sign, expression > 0: output = 1, expression = 0: output = 0, expression < 0: output = -1
TRUNC	TRUNC(expression)	truncate to integer expression
ROUND	ROUND(expression)	round expression to nearest whole number

Function	Syntax	Description
RND	RND	random number between 0 and 1
LOG2	LOG2(expression)	logarithm (base 2)
LOG10	LOG10(expression)	logarithm (base 10)
LN	LN(expression)	natural logarithm
EXP	EXP(expression)	exponent (expression)
IF	IF(expression)	Create IF sentence
NAN		not a number
MAX	MAX(expression)	calculates the max value of expression
MIN	MIN(expresion)	calculates the min value of expression

Functions examples and their explanations:

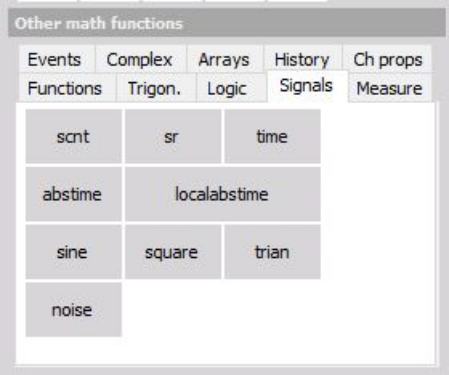
SQR - square	sqr(4)= 16, sqr(-3)= 9	The square of a number is the number multiplied with itself. No matter if the number is pos. or neg. the result will be always positive.

SQRT - square root	sqrt(16) = 4, sqrt(9) = 3	The square root of number (inverse function of SQR)

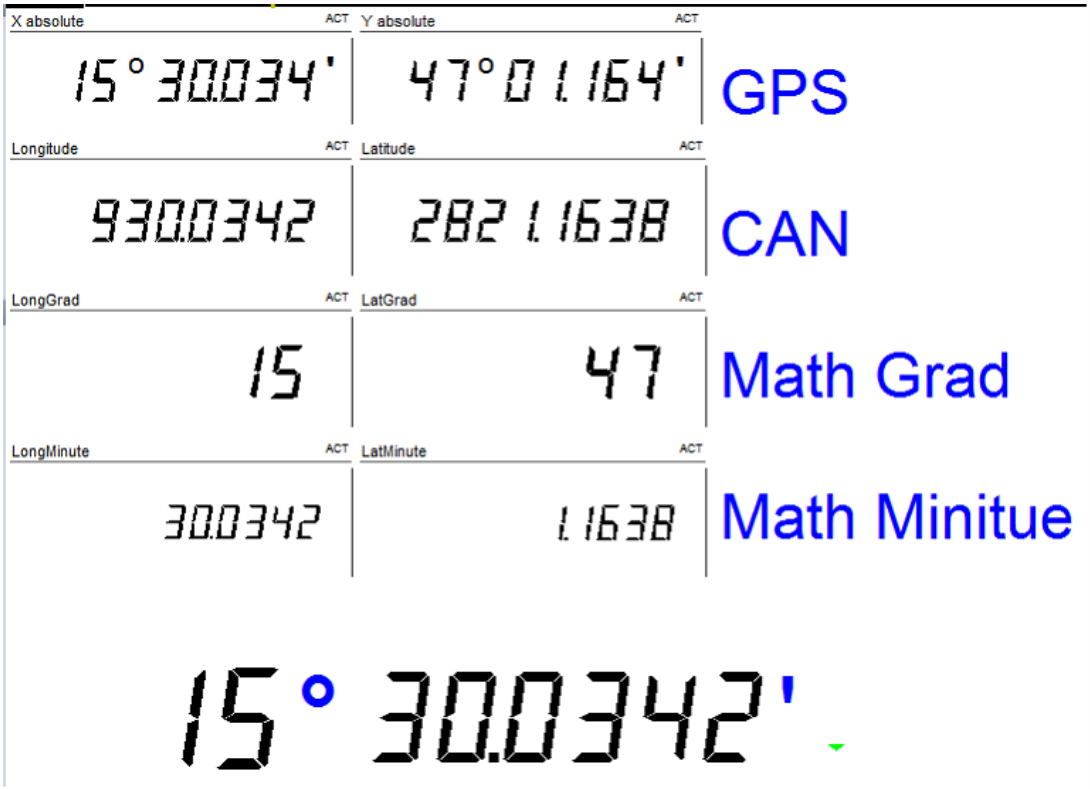
INFO: **SQRT of a negative number sqrt(-9) will always deliver 0 instead of a complex number in DeweSoft.**

ABS - absolute value	ABS(45.34) = 45.34, ABS(-33.12) = 33.12, ABS(0) = 0	Calculates the absolute value of number or a channel.
SGN - Sign	sgn(-8.124) = -1, sgn(19.345) = 1, sgn(0) = 0	Sign function delivers 0 if input channel or number is 0.
TRUNC - truncate function	Trunc(1452.457) = 1452, Trunc(-1452.457) = -1452	Converts a number into integer. So every number or channel which is converted with the Trunc function will loose the part after the comma. It will become an integer value.It is not rounded so either: Trunc(86.248) = 86 and also Trunc(86.848) will give 86

Example: Separate CAN GPS Signal to DEG:MIN,xxx



The VGPS Longitude and Latitude signals received over the CAN Bus should be separated into DEG:MIN,xxx. The CAN data is received in MIN like it is shown below. With math functions, the regular GPS display should be produced. Below the result is shown.



MATH: **trunc('Latitude'/60) dividing by 60 will convert it from minutes to deg, where the trunk function will eliminate the digits after the comma. -> result [°].**

(('Latitude'*100000) mod 6000000) / 100000 will provide the minutes and the rest.

Because the MOD function only delivers the rest in integer we have to multiply the latitude with 100000 and Use mod with 60 *100 000 = 6000 000 and divide the result again with 100 000 to get the result.

ROUND - round function	round(14.43) = 14, round(14.501) = 15, round(-14.492) = -14, round(-14.51) = -15	Rounds a number or channel depending on the digits after the comma to an integer value. 3.xxx if xxx is bigger than or equals 0,5 it will be rounded up to the next integer value.

HINT: **If you want to round 14.43 to 10, or 136.3724 to 140 simply divide the value or channel by 10 round it and multiply the result by 10.**

round(13.63724)*10 = 140

If you want to round 136.3724 to 136.4 multiply the value or channel with 10, round it, and divide it by 10.

round (1363.724)/10 = 136.4

RND - random	RDN	Creates random numbers with the selected sampling rate between 0 and 1. So if a sample rate of 1000Hz is selected, 1000 values per second are created.

LOG2 - Logarithm base 2	Log2(8) = 3, Log2(a) = b	Calculates the logarithm (base 2) of a number or an input channel. The logarithm extracts b from an equation 2b = a.
LOG10 - Logarithm base 10	Log10(100) = 2, Log2(a) = b	Calculates the logarithm (base 10) of a number or an input channel. The logarithm extracts b from an equation 10b = a.
LN- Natural logarithm base e	LN(100) = 2, LN(a) = b	Calculates the natural logarithm (base e=2.71828…) of a number or an input channel. The logarithm extracts b from an equation 2.71828b = a.
EXP - exponent function of e	EXP(1) = 2.71828, EXP(b) = a	Calculates the exponential function of e from a number or an input channel.The logarithm extracts b from an equation 2.71828b = a

IF/NAN - If sentence, not a number function	IF(condition,result1,result2) Outputs either the result1 or result2 depending on condition. condition: Expects a true false input.

EXAMPLE: (Channel >= 12)

So if the channel, could be a number or any Dewesoft channel, is equal or bigger than 12, the Condition becomes true and result1 will become rue. Otherwise, the condition is false and result will become true.

Example: IF('ID'>=1,'velocity','displacement')

'ID', 'velocity', and 'displacement' are analog input channels.

If the ID channel is equal or bigger than 1, the statement is true and the velocity channel will be chosen as output, otherwise displacement. So either velocity or displacement is used for output depending on the condition, both at the same time are not possible.

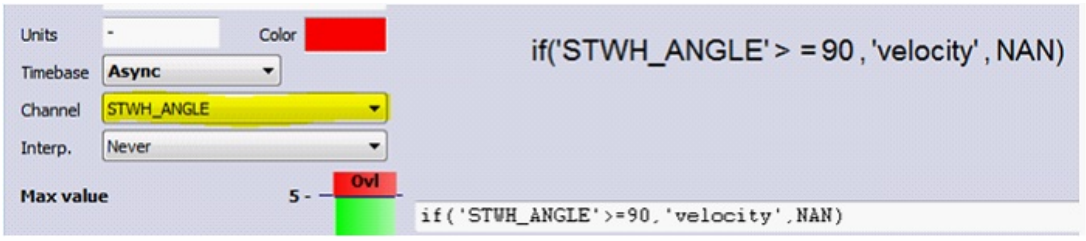
NAN(not a number)

Example: if('STWH_ANGLE'>=90,'velocity',NAN)

'STWH_ANGLE' = CAN Channel (asynchronous)

'Velocity' = analogue input channel (synchronous)

The channel will deliver only a NAN if STWH_Angle is smaller than 90deg. And if the time base of the math channel is forced to asynchronous output. Otherwise because of a sync. channel is used in the formula (velocity) the formula will output a Sync. channel and NAN will become zero.



The picture above is showing the Timebase setting of the math channel. The async. channel has to be used in the formula otherwise it is not possible to select it in the channel selector. The interpolation has to be set to never.

MAX - Maximum function of more channels	MAX(Channel1,Channel2)	Checks both channels and outputs the maximum value of one of the channels.

Example: max('pressure1','pressure2')

pressure1 and pressure2 are two analogue input channels.

pressure1	3	4	6	8
pressure2	2	5	4	7
output	3	5	6	8

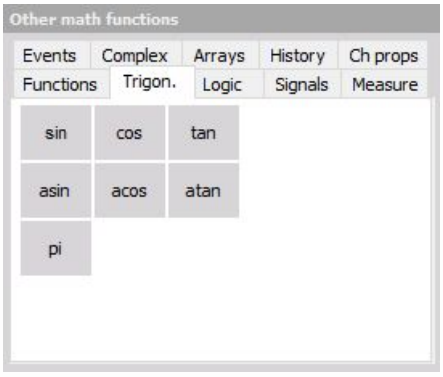
The higher value if both channels will be output.

NOTE: **Also, multiple max could be used in one formula.**

max(max('pressure1','pressure2'),'pressure3')

Trigonometry

The Trigonometry tab contains all required angle based functions.

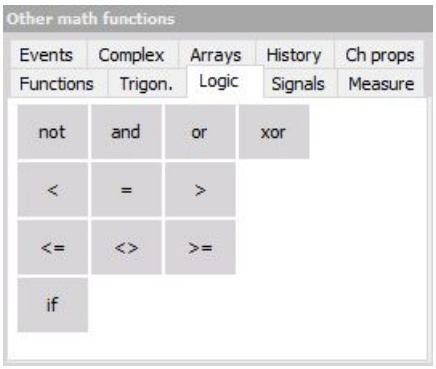


Function	Syntax	Description
SIN	SIN(expression)	sine of expression
COS	COS(expression)	cosine of expression
TAN	TAN(expression)	tangent of expression
PI	PI	constant piPI

Function	Syntax	Description
ASIN	ASIN(expression)	arc-sine of expression
ACOS	ACOS(expression)	arc-cosine of expression
ATAN	ATAN(expression) or ATAN(x,y)arc-cosine of expression	arc-tangent of expression or arc-tangent of x/y

Logic

The Logic tab contains required logic functions. Please be aware that the logic operators OR, AND, NOT and XOR works only with PURE digital values - 0 and 1. Therefore all analog channel must be first digitized before used on this operators (like 'velocity'<100).



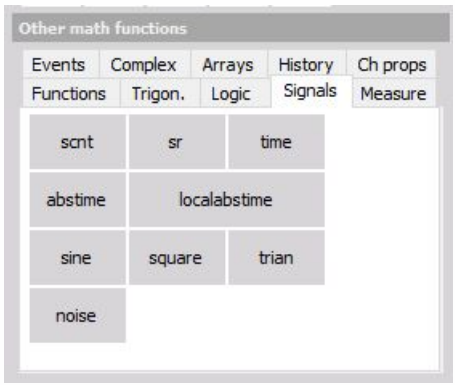
Function	Syntax	Description
<	expression1 < expression2	if expression1 is less than expression2 then output is 1, else 0
=	expression1 = expression2	if expression1 is equal to expression2 then output is 1, else 0

Function	Syntax	Description
>	expression1 > expression2	if expression1 is greater than expression2 then output is 1, else 0
<=	expression1 <= expression2	if expression1 is less than or equal to expression2 then output is 1, else 0
<>	expression1 <> expression2	if expression1 is less or greater than expression2 then output is 1, else 0
>=	expression1 >= expression2	if expression1 is greater than or equal to expression2 then output is 1, else 0
NOT	NOT expression	negation; expression = 0: 1; expression = 1: 0;
AND	expression1 AND expression2	logic and, 1 AND 1 = 1, 1 AND 0 = 0, 0 AND 1 = 0, 0 AND 0 = 0
OR	expression1 OR expression2	logic or, 1 OR 1 = 1, 1 OR 0 = 1, 0 OR 1 = 1, 0 OR 0 = 0

Function	Syntax	Description
XOR	expression1 XOR expression2	logic exclusive or, 1 XOR 1 = 0, 1 XOR 0 = 1, 0 XOR 1 = 1, 0 XOR 0 = 0

Signals

Depending on the application, generated signals may be required. Therefore you can select the common signal forms in the Signals tab.



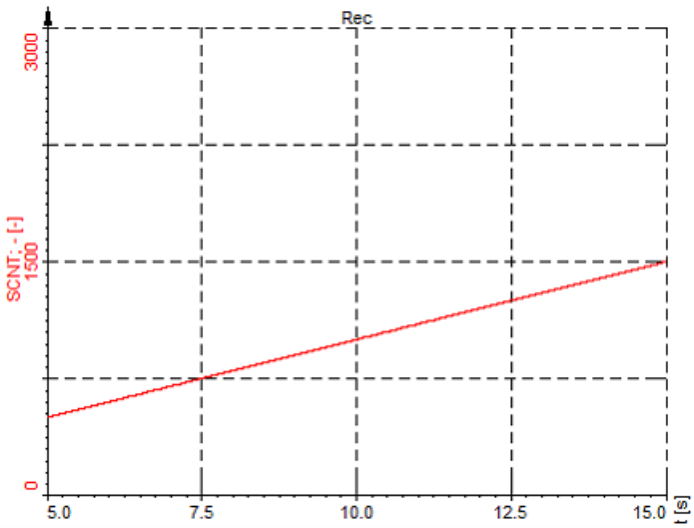
Signal functions and generation:

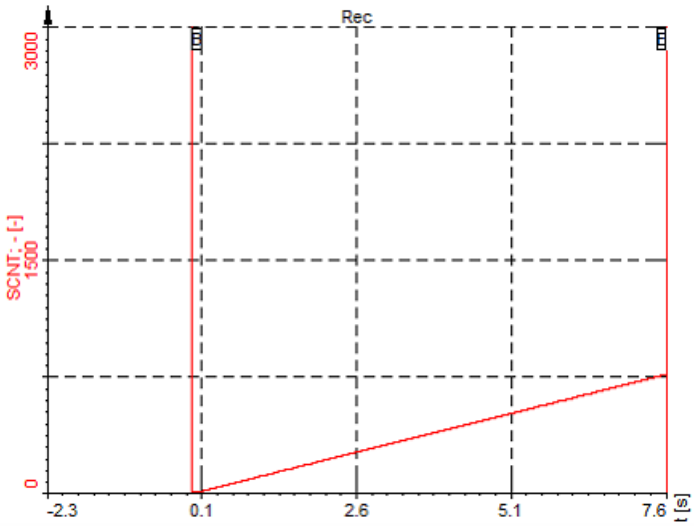
Function	Syntax	Description
SCNT	SCNT	Number of samples acquired
SR	SR	Sample rate of acquisition
TIME	TIME	Time elapsed (in seconds)
SINE	SINE(f [,ps])	Generates a sine wave with frequency f [Hz] optionally with phase shift ps [radian]

Function	Syntax	Description
SQUARE	SQUARE(f [,ps])	Generates a square wave with frequency f [Hz] optionally with phase shift ps [radian]
TRIAN	TRIAN(f [,ps])	Generates a triangle wave with frequency f [Hz] optionally with phase shift ps [radian]
NOISE	NOISE	Generates noise (random number between ±1)

Signals examples and their explanations:

Sample counter - Delivers the samples acquired from start of the measurement. The counter will be rest at start storing.





The above picture is showing the result, while at the bellow picture the reset after start storing could be seen.

Create Angle Signal

EXAMPLE: **If external clocking is used and the signal should be shown in an XY diagram angle based, together with the MOD function we can create an angle signal.**

Let's assume we are using an encoder with 720pulses/rev.

SCNT MOD 720 will deliver a sawtooth which runs from 0 to 720. To get the angle we have to multiply it by 0,5deg so at the end we get this formula.

SCNT MOD 720 * 0.5 this channel could be used in an XY diagram to show the resulting angle based.

The only disadvantage will be, if we get the wrong pulses from the encoder (noise, spikes), the angle signal will shift.

If a TRG pulse is also available, CLK and TRG, this signals could be also routed to a Counter, where this is eliminated.

Because the TRG PULS will reset the counter every revolution.

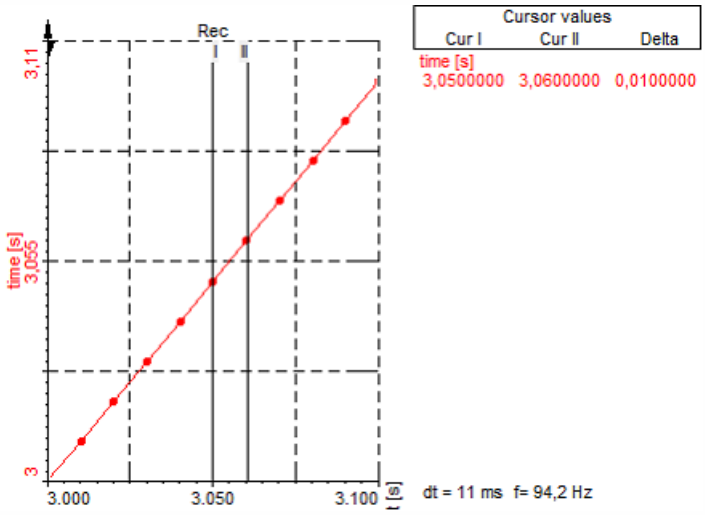
TIME	Time function	Is providing the elapsed time of the measurement in seconds.

This is a similar function to SCNT. The only difference is, instead of samples we get the time in seconds, independent which sampling rate is used.

Also, the TIME function is reset at start storing.

The resolution is linked with the sampling rate.

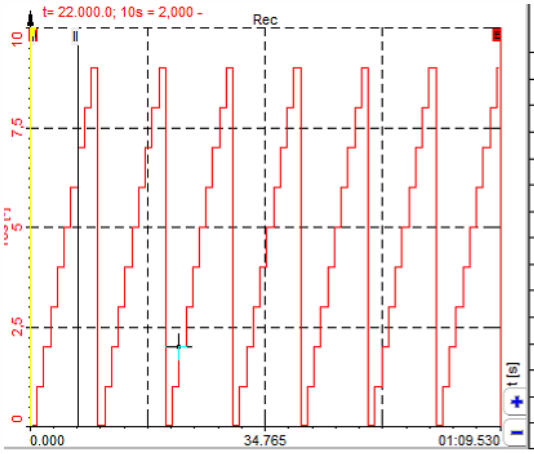
The screenshot below is showing the time function in a recorder. We could see that the resolution is 0,01s and also dt=10ms which results in a sampling rate of 100Hz. So the time will count up 0,01s after every sample at sampling rate of 100Hz or 0,001s at sampling rate 1kHz.



EXAMPLE: **Show actual value averaged every 10s in a list and export it to Excel.**

MATH: (Time MOD 60)

This will create a sawtooth with a period time of 60s. Look at the picture below. This is channel will be used as an event channel in the LATCH math to average the actual channel and show it in a list. The averaged values could be exported to Excel or TXT asynchronously. To export it asynchronously at Export only the asynchronous channels have to be selected.



Time	pressrue/Latch bar
06.09.2010 18:00:22.537	0,9997
06.09.2010 18:00:12.537	0,9997
06.09.2010 18:00:02.537	0,9998
06.09.2010 17:59:52.537	0,9998
06.09.2010 17:59:42.537	0,9998
06.09.2010 17:59:32.537	0,9998
06.09.2010 17:59:22.537	0,9998

Exported	Index	Type	Acq. rate	Dimension	Name
No	0	AI 0	10000	Scalar	pressrue
No	1	Math 0 (Formula)	single	Scalar	Frm0/Formula 0
No	2	Math 1 (Formula)	10000	Scalar	Frm1/10s
Yes	3	Math 2 (Latch math)	0,1	Scalar	Latch0/Latch index
Yes	4	Math 2 (Latch math)	0,1	Scalar	Latch0/pressrue/Latch

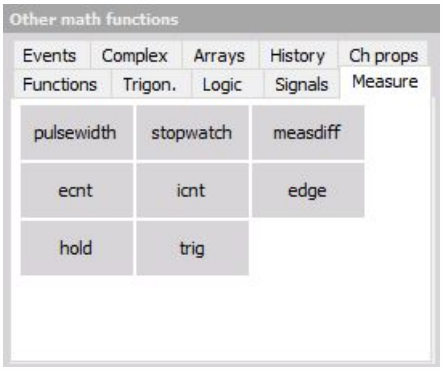
NOTE: **Generated signals can only be used for mathematics, display and storage. It is not possible to use them for analog output!**

If you require analog output (e.g. from a function generator) you have to use the DEWESOFT-OPT-FG option.

For details about Function generator see -> User Guide -> Channel Setup - Function Generator.

Measure

Different measurement values and information are available in the Measure tab. Again here the conditions have to be logic operators. It will again not work to write ECNT('Input TTL') and assume that the transition will be done on value bigger than 1. The correct equation would be ECNT('Input TTL'>2.5) where
'Input TTL'>2.5 give logical 0 or 1.



for examples see: -> Example of measure functions -> Example of stopwatch

PULSEWIDTH	PULSEWIDTH(cond [, rearm])	Measures time [s] between two cond edges (cond jumps from 0 to 1), rearm edge optional
STOPWATCH	STOPWATCH(cond1, cond2)	Measures time [s] between cond1 and cond2 edges (cond jumps from 0 to 1)

MEASDIFF	MEASDIFF(value, cond1, cond2)	Measures value difference between cond1 and cond2 edges (cond jumps from 0 to 1)
EDGE	EDGE(cond [, rearm])	Returns 1 when the cond changes from 0 to 1 and have again optional rearm conditions
ECNT	ECNT(cond)	Counts number of edges for cond (cond jumps from 0 to 1)
ICNT	ICNT(cond)	Counts all the samples where the cond have a logical value 1

HOLD	HOLD(value, cond [, rearm])	shows and holds value, when cond goes from 0 to 1 with optional reams condition
TRIG	TRIG	has a value of 1 when a store condition appears

Measure examples and their explanations:

HOLD - Hold function

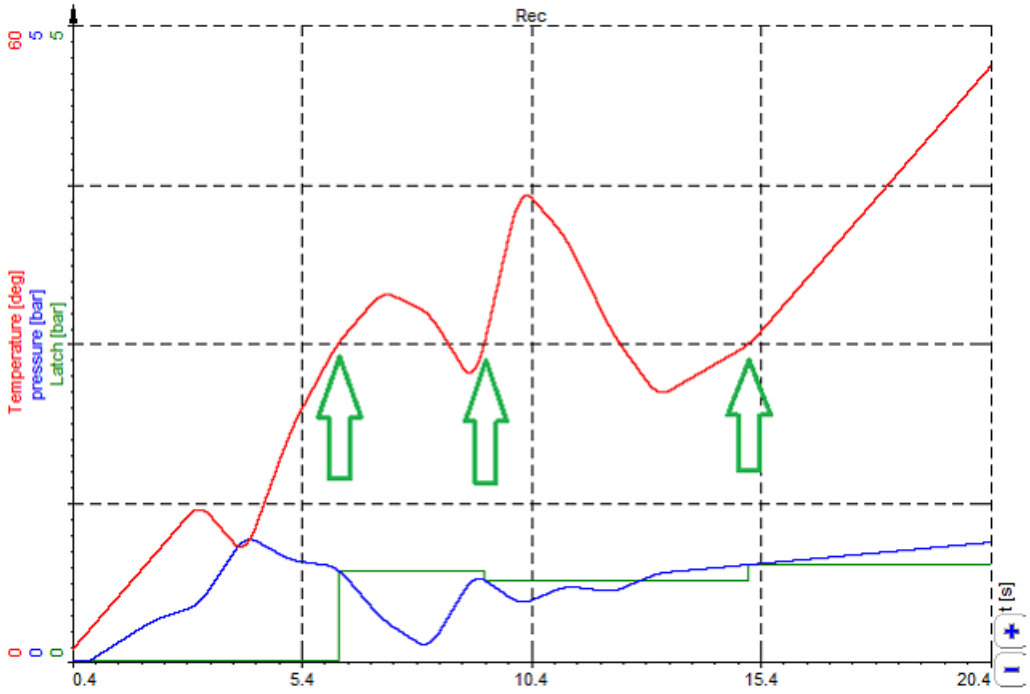
Hold(value,latch condition,[rearm condition])

The hold function is used to latch or hold a single value if a condition is met.

EXAMPLE 1: **Hold(‘pressure’, ‘Temperature’> 30)

Hold(value,latch condition)**

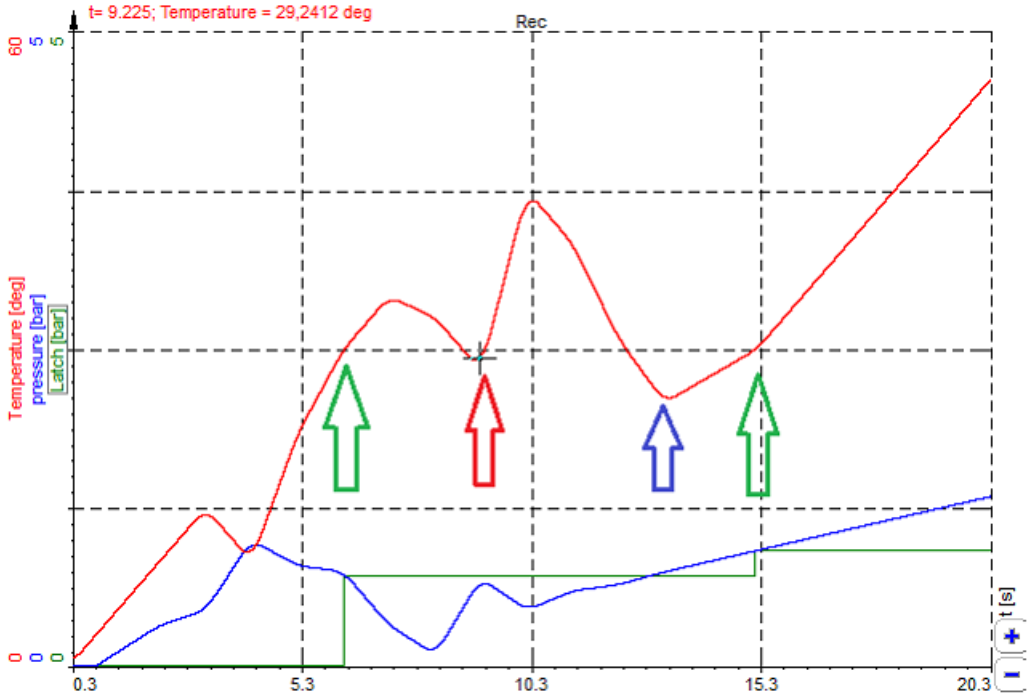
In the example above the function will hold the actual pressure if the temperature is higher or equal 45deg.



The picture above is showing the function in a recorder. All the time when the temperature exceeds 30deg (green arrows), the actual pressure channel is latched.

EXAMPLE 2: **Hold(‘pressure’, ‘Temperature’> 30, ‘Temperature’< 28) Hold(value, latchcondition, [rearmcondition])**

The function could be extended with an 'rearmcondition'. After the 'latchcondition' occurred, the 'rearmcondition' has to be met first before a new 'latchcondition' and therefore a new latch could occur. This is used to filer the 'latchcondition' a little bit. Imagine the 'latchcondition' channel has noise on it or is fluctuating around the level (30deg +-0,2deg) which would cause an unintentional LATCH.



The picture above will illustrate this function:

1. Temperature rises above 30 deg -> LATCH 1 - green arrow

2. Temperature goes below 30deg and back above 30 deg -> no LATCH because the temperature did no go under 28deg, therefore, the LATCH did not occur. - red arrow.

3. Temperature goes below 28 deg -> rearm condition is complied - blue arrow

4. Temperature exceeds again 30deg -> LATCH 1 is performed again - green arrow

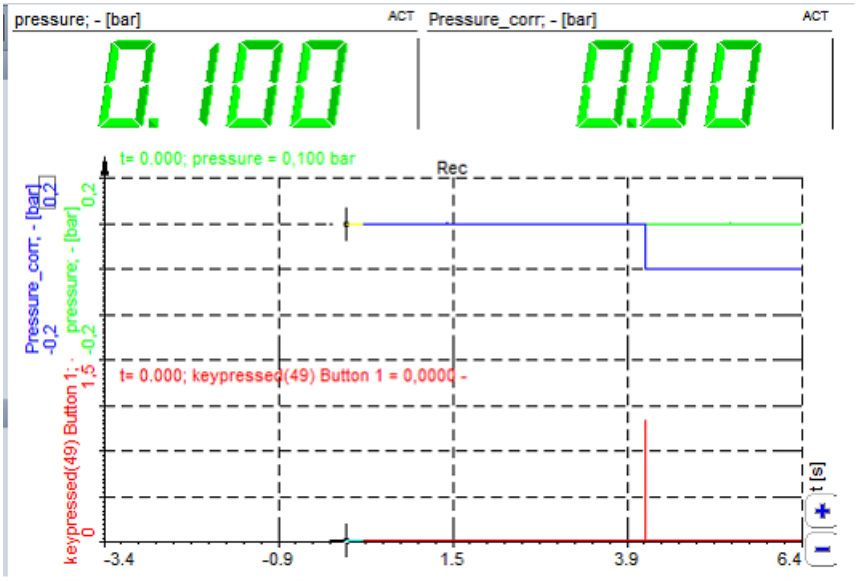
EXAMPLE 3: **Remove offset from a Channel.**

HOLD(channel, condition) ... will latch the actual value of the channel if condition become true.

MATH: 'pressure'-hold('pressure',keypressed(49)>0.5)

The actual pressure channel is subtracted with the value latched in the hold function. The hold function will latch the actual pressure if the second statement 'keypressed(49)>0.5' become true.

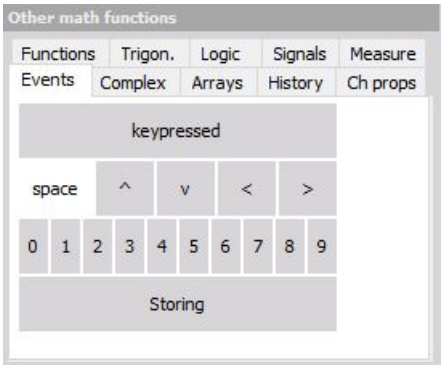
So even during the measurement, an offset compensation could be done by pressing a specific key. The picture below is showing an example. The 'keypressed(49)' channel is indicating the pressed key in an additional math channel to make it more obvious.



Events

The Events tab allows defining keys as signals. Each time when you press the defined key, you receive a peak in the signal. You can display that e.g. in the recorder, scope, etc.

for examples see -> Example of input events



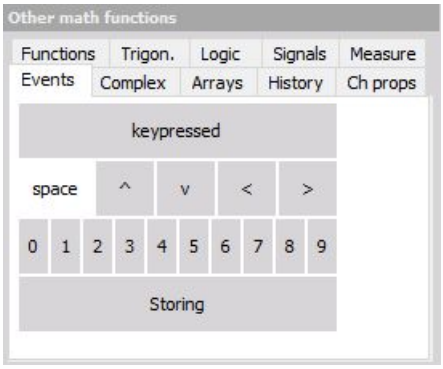
Function	Syntax	Description
KEYPRESSED	KEYPRESSED(ASCII code)	Generates pulse (transition 0 to 1 and back) if key with matching ASCII code is pressed.

EXAMPLE: **KEYPRESSED(32) outputs pulse if Space key is pressed.**

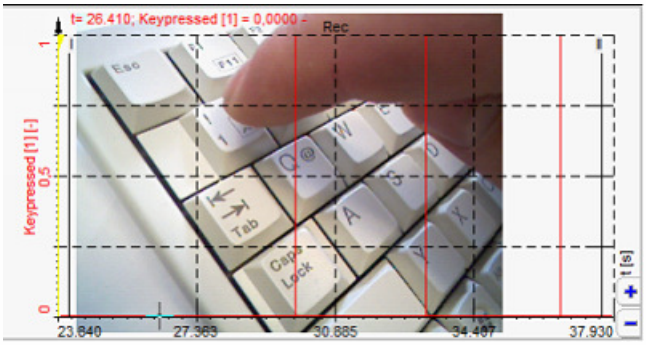
Events examples and their explanations:

keypressed(49) - produces a signal from 0 to 1 with a duration of 1 sample if [1] button is pressed.

A few predefined keys are already prepared as shown in the picture below. So if you press the [1] button below the line 'keypressed(49)' will be automatically added to your formula.



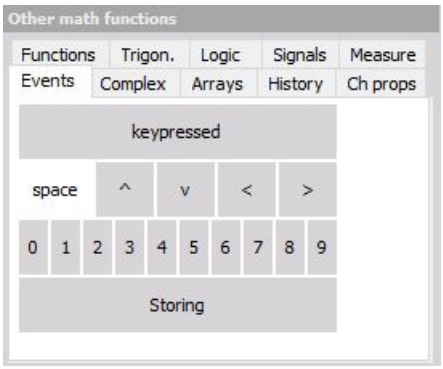
Every time the [KEY 1] is pressed on your keyboard a signal is produced.

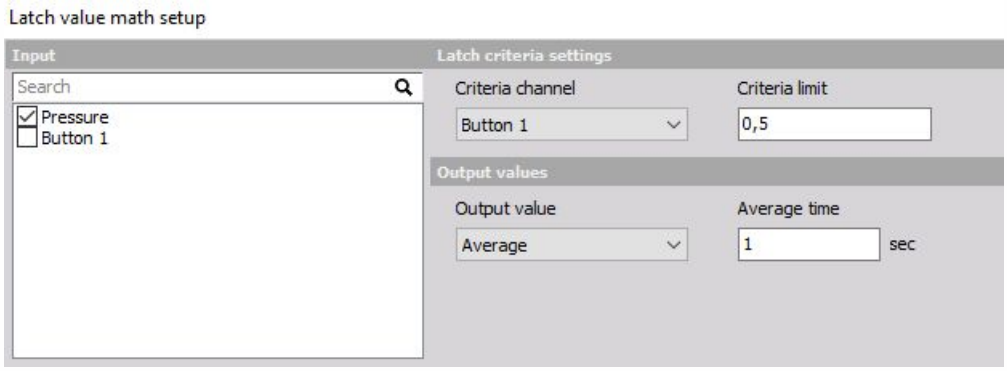


Almost any key could be used in the 'keypressed' function. The value in the brackets is reflecting the virtual key code in decimal.

Below you will find a list of the most popular keys. You have to convert them from Hex to decimal. So [Key 1] ->31hex -> 49dec. The decimal value has to be entered Into the keypressed function -> keypressed(49).

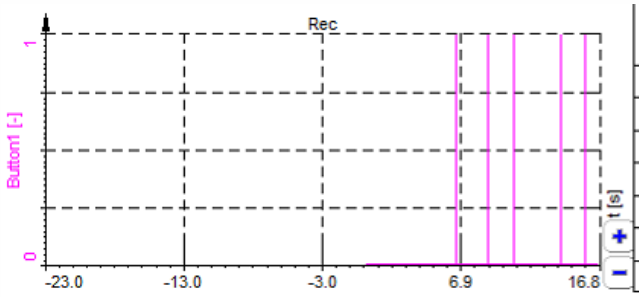
EXAMPLE: **Latch Value into List**





MATH: keypressed(49) produces signal from 0 to 1 with a duration of 1 sample if [1] Button is pressed.

This could be used in the latch Math to latch the actual or average value of an other channel(s) in a list.



Time	pressrue/Latch bar
06.09.2010 18:42:44.999	0,9998
06.09.2010 18:42:43.234	0,9998
06.09.2010 18:42:39.904	0,9998
06.09.2010 18:42:38.040	0,9998
06.09.2010 18:42:35.642	0,9998

ARRAY OPERATIONS

Array channels can come from scope trigger, FFT, CPB, STFT, classification, counting and other new procedures. The formula has operations which work on those array channels.

Extracting subset of the array

To extract a value from an array, use brackets with index. Let' s say we want to extract one single value of the FFT:

$$'AI0/AmplFFT'\[10$$

\]

Operation on arrays

Basically, all operation should work on arrays, but there are some limitations to it. We can, for example, do the following:

$$'AI0/AmplFFT' -' AI1/AmplFFT'$$

It is important to know that the product of the two vectors multiplies elements in the array. The resulting array has the same size as the input array.

$$'AI0/AmplFFT'*'AI1/AmplFFT'$$

We can also for example mix vector and scalar values:

$$'AI0/AmplFFT' + 2$$

This formula will add a value of two to each array element and will output array with the same size as the input.

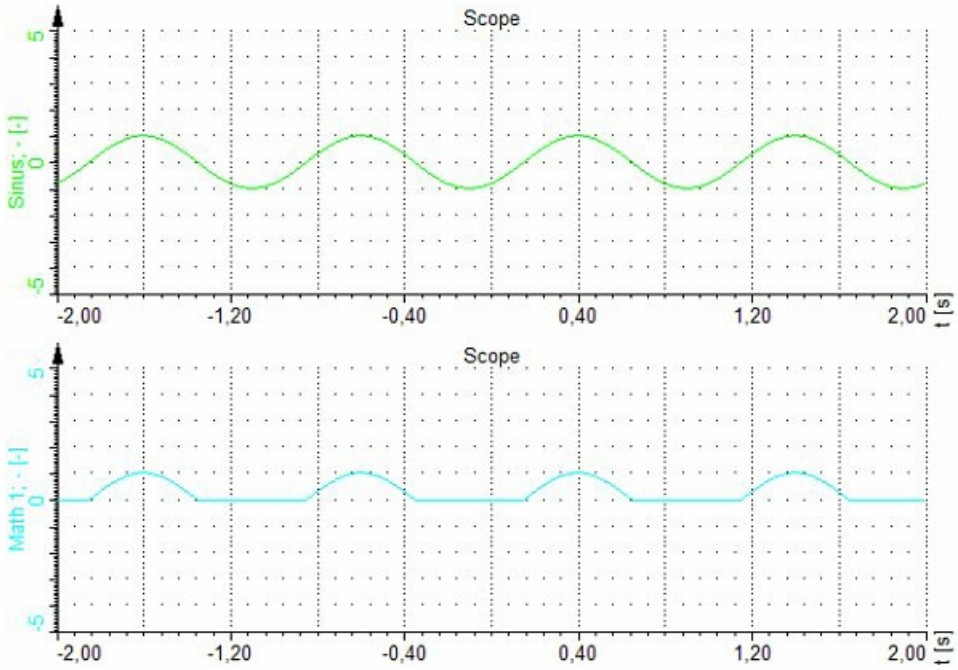
IMPORTANT: **Arrays with different sizes can't be combined into a single formula.**

EXAMPLE OF LOGIC FUNCTIONS

The result of logic operators is always 0 or 1. Therefore we can use this in further calculations:

```
('sinus' > 0) * 'sinus'
```

The first part of the following equation will return 1 when the input channel 'sinus' is positive and the value of 0 when the sinus will be negative. Multiplying this result with the original signal will result in cutting the negative part of the signal.

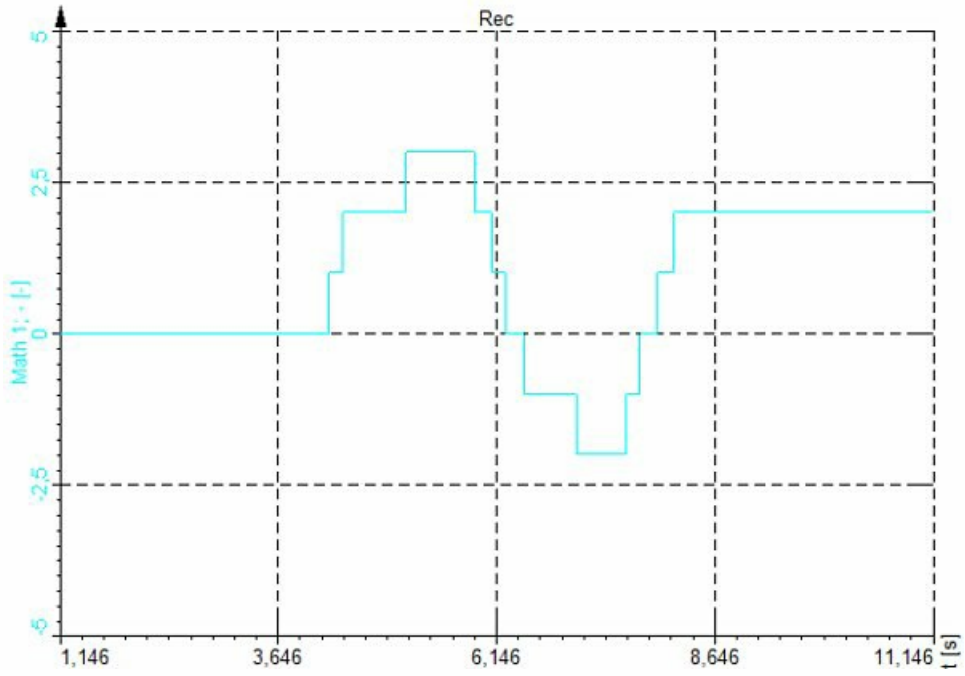


EXAMPLE OF INPUT EVENTS

The input events combined with count procedures are very useful for counting events (for example event counting).

```
ecnt (keypressed(39)) - ecnt (keypressed(37))
```

The example above counts up when the right cursor key is pressed and counts down when the left cursor key is pressed.

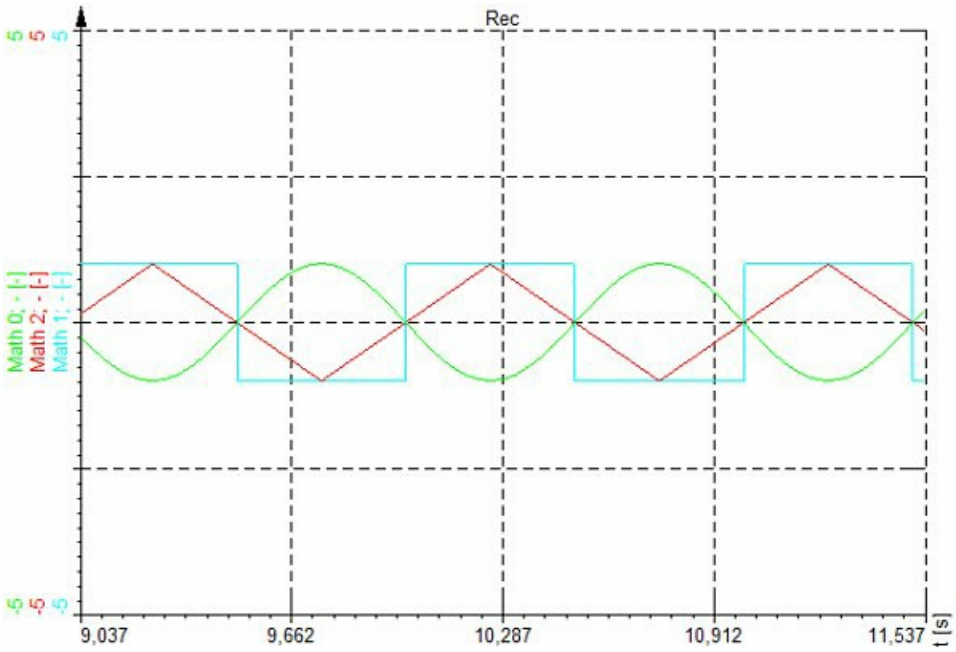


EXAMPLE OF SIGNALS

The signal generator is used for making a test signal for testing. We have basic functions like a sine wave, square, triangle, rectangle and noise. We can define also the phase for each individual channel. The phase is defined in radians, so in the example below the sine wave with 1 Hz frequency and 180 deg phase is defined.

```
sine(1 , Pi)
square(1)
trian(1)
```

The example above results in the following signals:



To generate signals with variable frequencies, it is better to use function generator in Analog out section. The Test AO is free of charge and can produce signals with variable frequencies, amplitudes and phases.

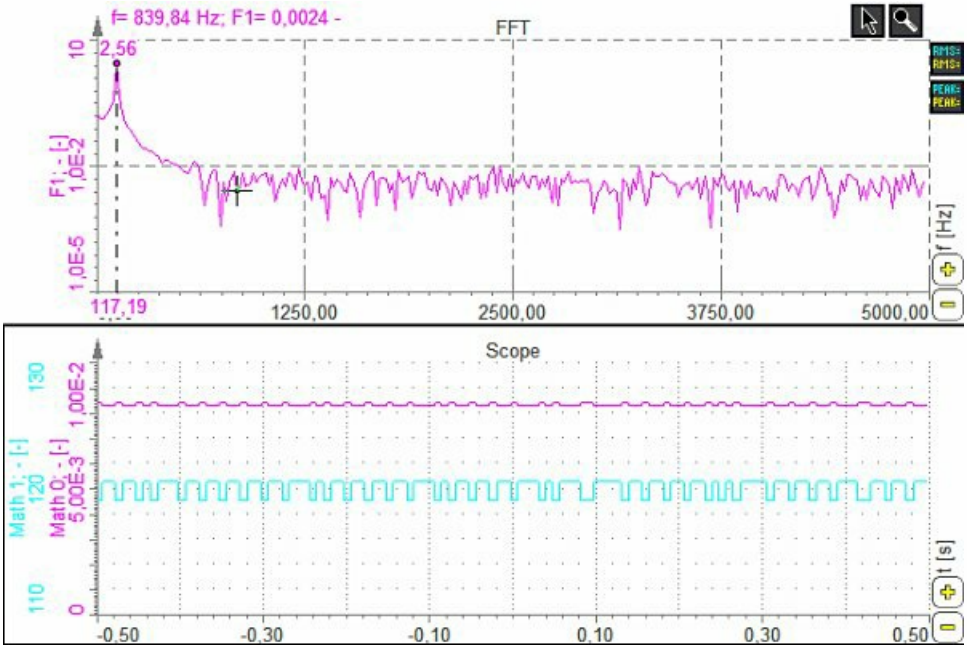
EXAMPLE OF MEASURE FUNCTIONS

The following example shows the measurement of the pulsewidth. Let's say that the input is a square wave and we want to measure the pulse width of this channel. First of all, we need to translate the input signal in the logic with 'F1'>0. This will give a value of 1 for positive and value of zero for negative values of the input signal. The pulsewidth function will then measure the time between each transition. If we want to measure the frequency, we can calculate the inverse of this function as seen below.

$$\text{pulsewidth('F1'} > 0)$$

$$\frac{1}{\text{pulsewidth('F1'} > 0)}$$

The graph shows a pulse width measurement (Math0) and frequency measurement (Math1).



'pulsewidth' function is useful for digital signal. The resolution of the measurement is limited by the sample rate.

If we want to measure the pulse width or frequency of sine waves, it is better to use Exact frequency math module, described in the following chapters.

EXAMPLE OF STOPWATCH

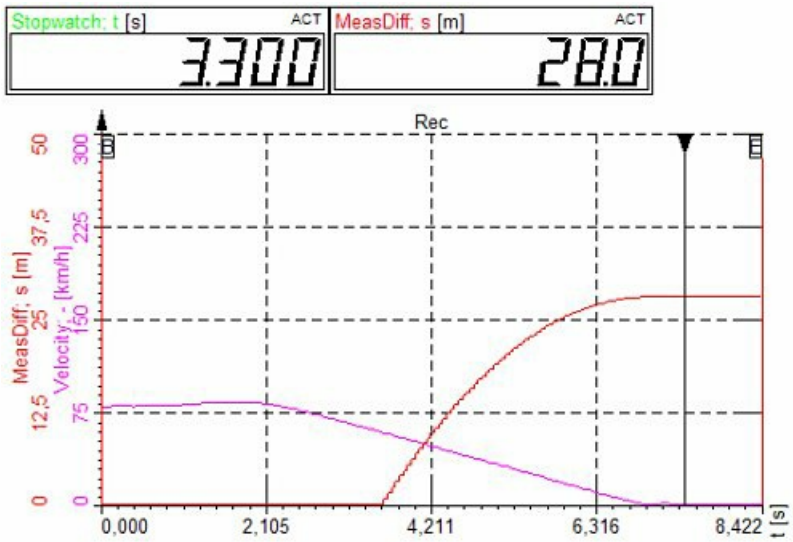
There are two functions in Measure section of math which helps to measure time and value between events. The Stopwatch functions enable us to measure the time between a start and stop event. The 'measdiff' function makes the same measurement, but we can define the measurement channel instead of time.

In the example shown we measure the brake time and brake distance. The first example gives the time of braking between 60 km/h and 2 km/h.

The second channel measures the distance between 60 and 2 km/h. This is achieved by defining the Distance channel as the calculated value.

```
stopwatch('Velocity' < 60 , 'Velocity' < 2)
measdiff(Distance', 'Velocity' <60 , 'Velocity' <2)
```

The graph below shows the example of measured data - input channels of velocity and distance. The meter shows the time and distance of braking.



Dewesoft 7 interpolates the velocity and distance channels. So for examples if measured velocity points are 61.2 and 59.2, it will take the time in between those two values as the start trigger.

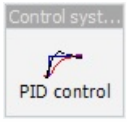
C++ SCRIPT

Please refer to the official C++ Script manual for help. It can be located under Support > Developers > Developer downloads > C++ script on the Dewesoft® website.

CONTROL SYSTEMS

In Control systems we can choose from following mathematical functions:

- PID control

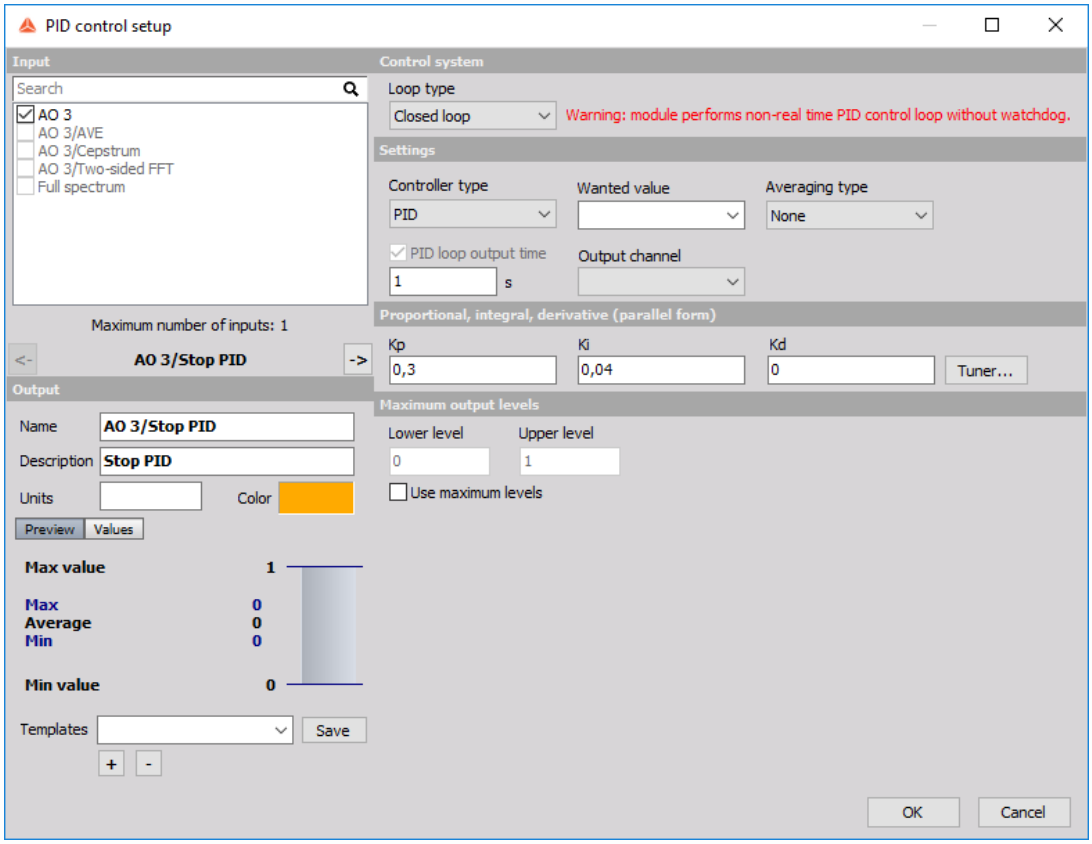


PID CONTROL

Dewesoft can be used as a PID controller by using the PID Control.

WARNING: **Dewesoft software running on Windows is not a real-time system because Windows is not a real-time operating system. This means that the delay of the controller is not always the same and can be unpredictable. For example Windows could give priority to another application instead of Dewesoft, which can dramatically increase controller delay.**

When you press the Setup button on newly activated PID control line, the following setup screen window will open:



Control system

Loop type - if Open loop is chosen, there is no feedback to the controller. In measure mode, the user can set a value to the controlling channel (analog or digital output, for example) and measure the response without controller interference. This option is useful for obtaining the step response of the system. If Closed loop is chosen, the feedback loop is turned on and additional settings appear.

Settings

- **Controller type** - we can choose between a PID controller and a PI controller with anti-windup (anti-windup will be described later in this tutorial). If only the P controller is needed, the PID type can be chosen and zeros filled in for I and D gains.
- **Wanted value** - the set point. In the figure above we created a user input channel (under Channel Setup, User Inputs) called ‘Set point freq’in order to be able to change the setpoint during measurement.
- **Output channel** - the controller output. Normally an analog or digital output channel would be chosen. In the figure above we named the analog output channel as “Control AO voltage”.
- **PID loop output time** - the period at which the controller will update the Output channel.
- **Averaging type** - the method of averaging the error (error = wanted value - input). Can be used if there is a lot of noise in the process value.

Maximum output levels

Maximum output levels set the minimum and maximum levels of the Output channel and can be set if desired.

Integrator windup limit

Integrator windup limit is visible when Controller type is set to PI with anti-windup and sets the limits of the Output channel in its units. When the Output channel is outside this limits, the error will not be integrated. There is another way to conquer a large dead time. In the PID setup, we can choose the PI controller type. This will prevent the accumulation of the integration error when the actuator is in saturation, which happens during dead time. If this type of controller is chosen, the Ziegler-Nichols method provides very good response also in the case of a large dead time.

For additional help with PID Control visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training -> PID Control.

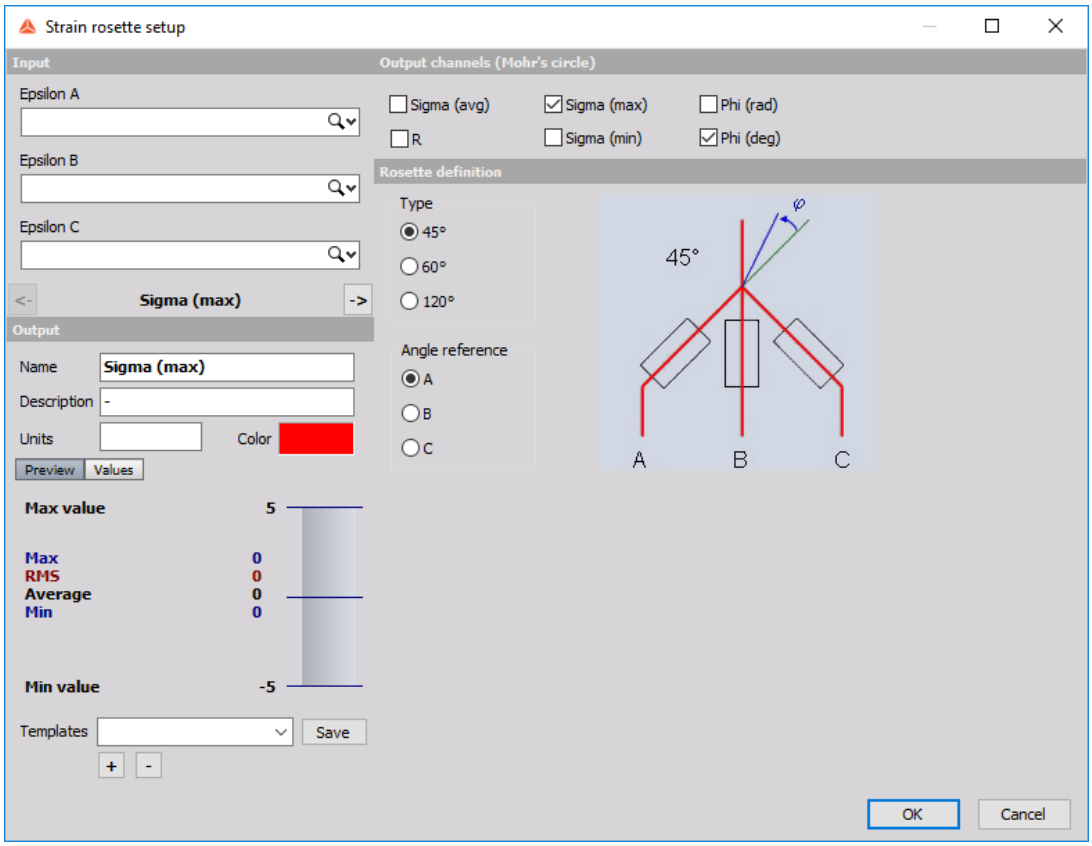
STRAIN, STRESS

In Strain, stress we can chose from following mathematical functions:

- Strain rosette

STRAIN ROSETTE

When you press the Setup button on newly activated Strain rosette setup line, the following setup screen window will open:



Strain rosette setup is divided into 4 parts:

- Input
- Output
- Output channels (Mohor's circle)
- Rosette definition

Input

Define the Input channel for A, B and C rosette.

Output

Edit the parameters of possible output channels.

Output channels (Mohor's circle)

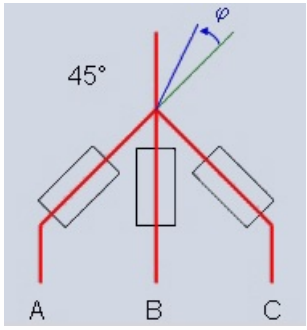
Activate the required calculations of strain rosette from list of possible options:

- Sigma (avg)
- R
- Sigma (max)
- Sigma (min)
- Phi (rad)
- Phi (deg)

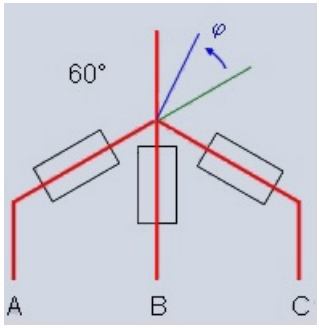
Rosette definition

Define the type of rosette by angle:

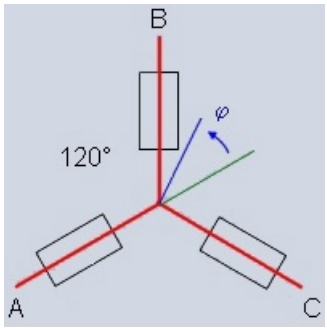
- **45°**



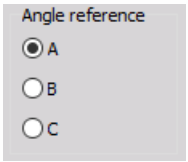
- **60°**



- **120°**



Define the angle reference point.



NOTE: **For more information please download the Rossete Math plugin manual.**

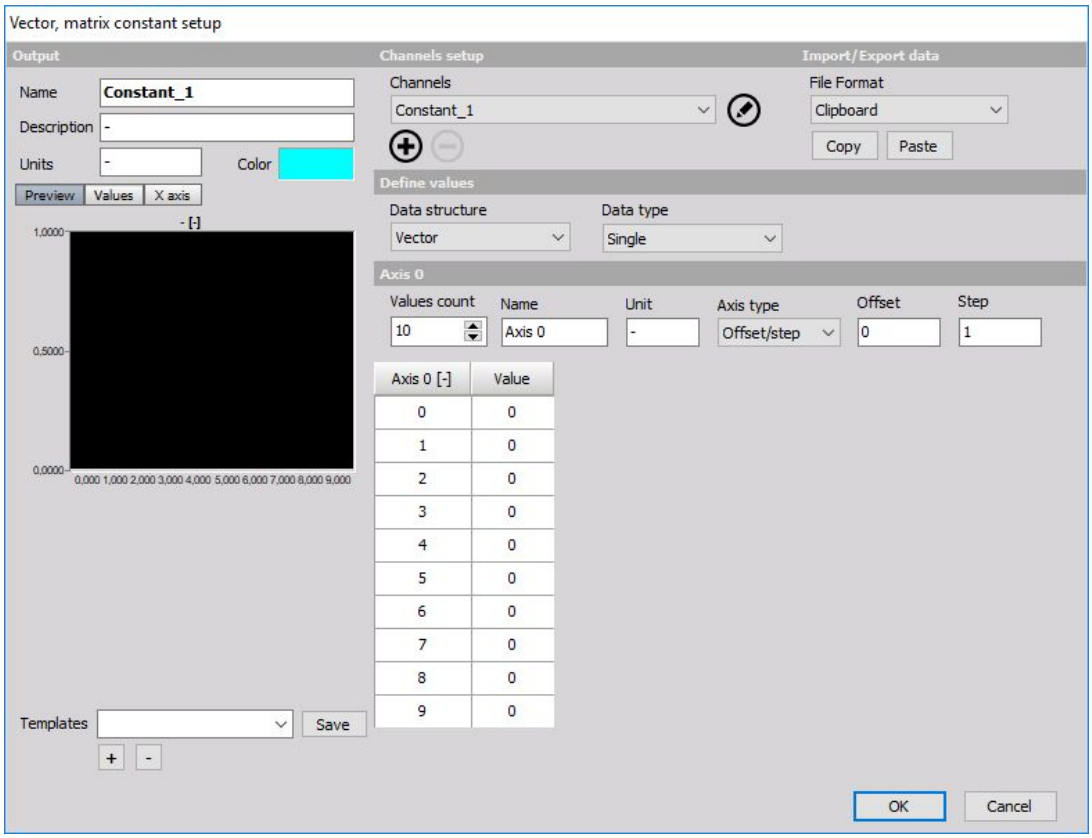
CONSTANTS

In Constants we can choose from following mathematical functions:

- Vector matrix constant

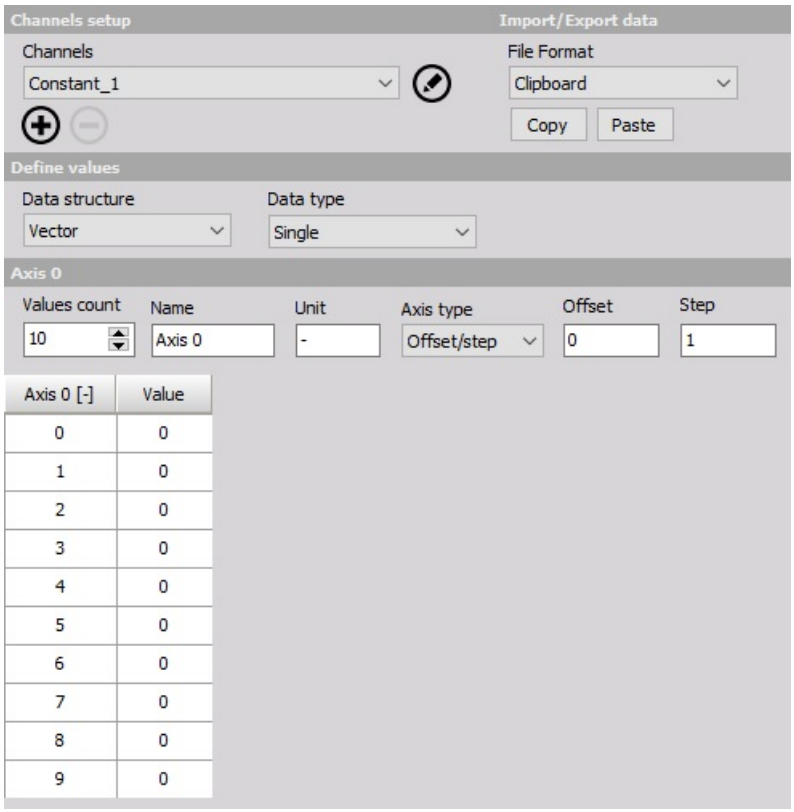
VECTOR MATRIX CONSTANT

When you press the Setup button or open a new math Matrix constant line, the following setup window will open:



The constant can define a set of values which will be as a result an array channel. We can add or remove points by increasing or decreasing Values count number and we can also Copy / Paste values from/to Excel.

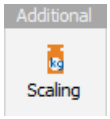
The resulting constant can be for example used when we want to multiply an array from FFT math with some filter parameters defined in the constant.


Channels setup
Import/Export data
Channels
Constant_1
File Format
Clipboard
Copy
Paste
Define values
Data structure
Vector
Data type
Single
Axis 0
Values count
10
Name
Axis 0
Unit
-
Axis type
Offset/step
Offset
0
Step
1
Axis 0 [-]
Value
0 0
1 0
2 0
3 0
4 0
5 0
6 0
7 0
8 0
9 0

ADDITIONAL

In Additional we can choose from following mathematical functions:

- Scaling



SCALING

The scaling module allows you to scale any scalar channel. The main purpose of the module is to allow you to scale your input data and be able to keep two channels, raw and scaled While the module is designed to scale data that is not linear, you can also use the same interface to scale linear data. There are three scaling types that you can choose from:

- From sensor
- Polynome
- Table

From Sensor

When this option is selected Dewesoft will automatically take the non-linear scaling parameters from the sensor that is mounted on the selected analog input channel. The purpose of this option is to allow you to have two channels for a single data source, unscaled and scaled. If you would simply use a non-linear sensor on your analog input channel, then there would be no way for you to re-scale the data in analysis, because Dewesoft does not store the raw signal, but only the scaled values.

To get around this, we have added an additional option to the Analog In grid called scaling. You can add it by right-clicking on the grid's header and selecting the Edit columns option. Once added, you can toggle the button to either keep the Raw signal or scale the data.

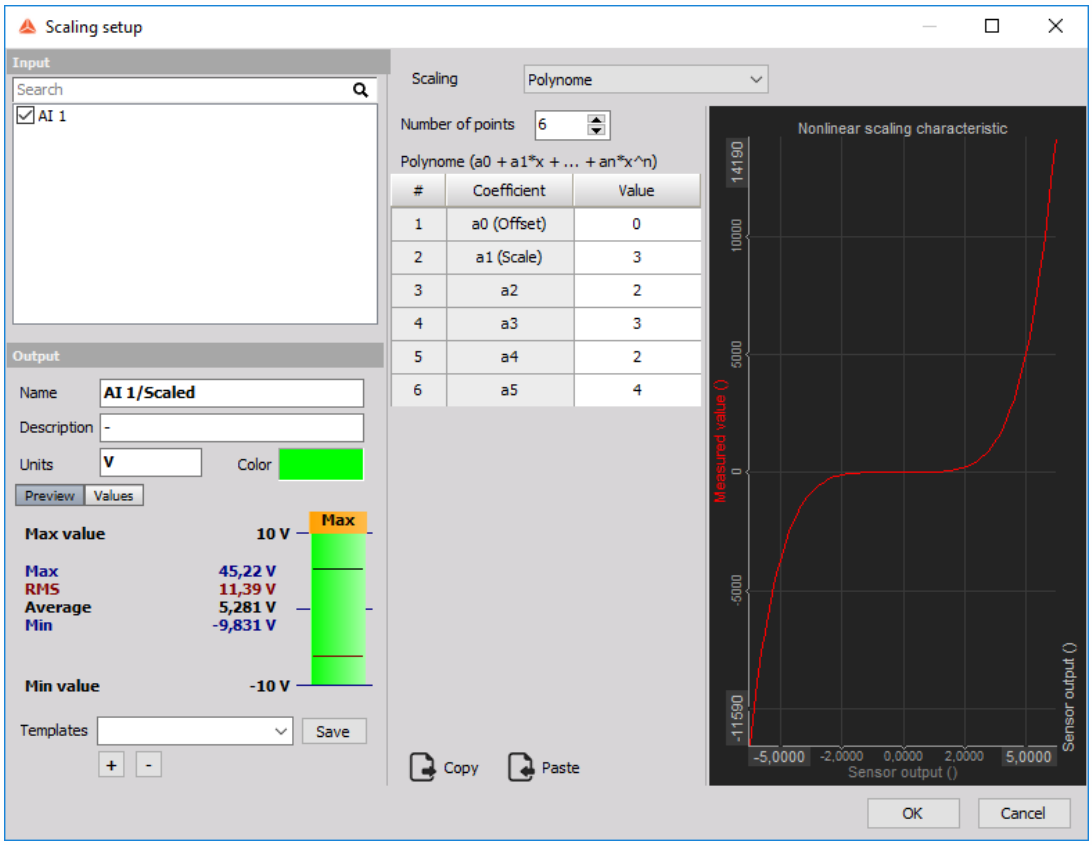
ID	Used	C	Name	Ampl. name	Range	Measurement	Min	Values	Max	Physical quantity	Units	Scaling	Zero	Setup
1	Used		AI 1	DEMO-SIRIUS-ACC	10 V	Voltage	-10,00	-1,991 / 2,252	10,00	Velocity	V	Raw		Setup
2	Unused		AI 2	DEMO-SIRIUS-ACC+	10 V	Voltage	-10,00	-2,826 / 3,063	10,00	Torque	Nm	Scaled	Zero	Setup

If you set the Scaling to Raw while at the same time having a non-linear sensor tied to it and select the channel in the Scaling math module, you will be able to use both channels in measure mode. You will also be able to re-scale the data in analysis in case you've made a mistake scaling the data originally.

Another benefit of this option is that you can use different scaling tables within the same module, since they are taken directly from sensors in tha Analog In tab.

Polynome

Polynome scaling allows you to enter polynomial coefficients that will be used to scale your input channels. You also get a nice preview of the function that you've entered and can copy/paste the scaling parameters.



Table

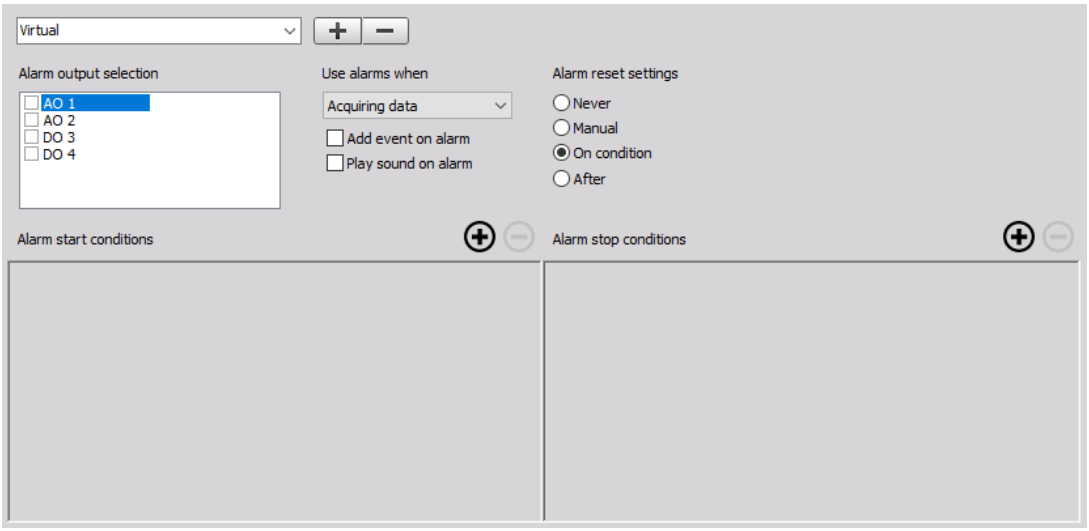
Table scaling allows you to enter coefficients that will be used to scale your input channels. You also get a nice preview of the table that you've entered and can copy/paste the scaling parameters.

ALARM MONITORING

Dewesoft set with Alarms the digital states according to acquired data and display alarm state on online screen. The alarm monitoring function can be also activated for digital output to A/D board.

When you select the Alarm tab on Dewesoft Setup screen, on the lower part of screen Alarms settings will appear to enter:

- **Basic settings** - to Add / Delete alarm; set eventual physically digital Alarm output selection; define Use alarms when and Alarm reset settings (set condition for reset alarm)
- **Alarm conditions** - conditions to Add (New) / Change / Remove and define Alarm start condition or “Don’t start” condition and eventual Alarm stop conditions



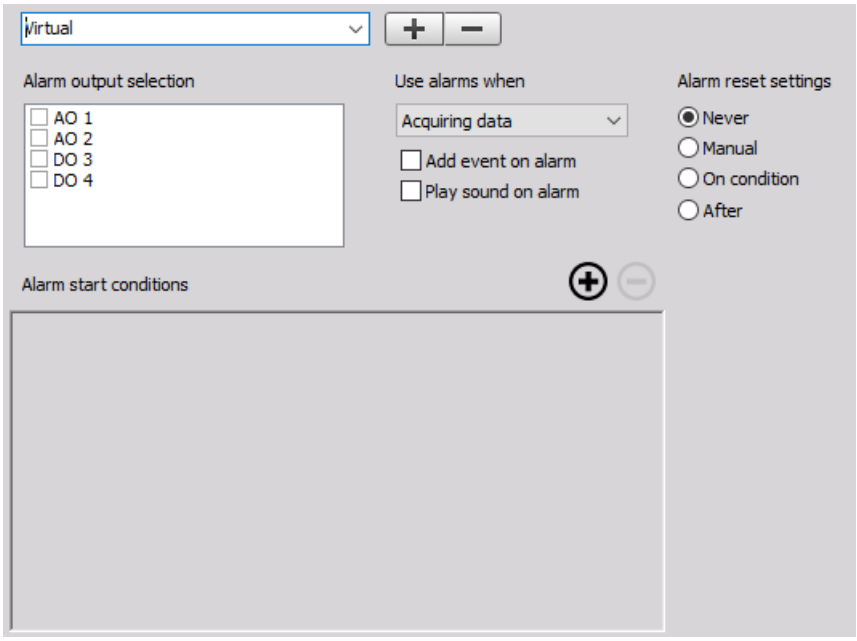
Basic settings

Add (New) / Delete alarm monitoring channel

When you select Alarms tab to define a new Alarm, on the lower part of the screen an empty alarm list appears:

It has two icons: * Plus - to add a new alarm condition * Minus - to remove the selected alarm condition.

When you press the Add alarm button, a new alarm condition is displayed:



To set up alarm we use:

1. STEP

Common settings - set eventual physically digital Alarm output selection; define Use alarms when and Alarm reset settings (set condition for reset alarm)

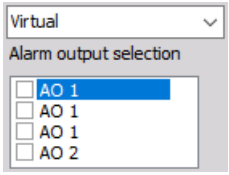
2. STEP

Conditions settings - to Add (New) / Change / Delete and define Alarm start condition or "Don’t start" condition and eventual Alarm stop conditions with Condition setup

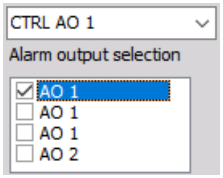
Alarm output selection

The Enable alarm monitoring and Use digital output box are checked.

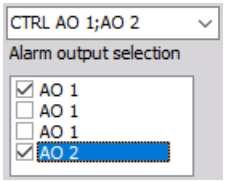
If no alarm is checked, the alarm will be virtual - only available for display.



After checking one output on Alarm output selection the alarms will be output on selected channel.



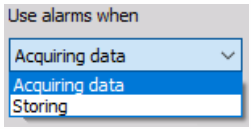
After checking two outputs on Alarm output selection the alarm will be output on two output channels



Use alarms when

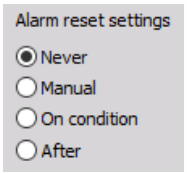
Select from Use alarms when the drop-down list:

- alarm while acquiring data
- alarm during storing



Alarm reset settings

Select the appropriate choice from the displayed list:

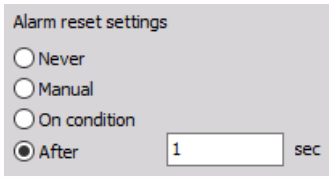


- **Never** - when Start condition carry out, Alarm remain on to end of acquiring or storing data.
- **Manual** - when Start condition carry out, Alarm goes on and on the first row of display screen red Alarm ON button appears:

To manual alarm reset alarm click on this button and display change to:

- **On condition** - next to Alarm start condition and "Don't start" condition on the right lower part of the screen also Alarm stop conditions section appear:

- **After** - when Start condition carries out, Alarm goes on and will be reset after the amount of sec, which is entered in the field beside this control:



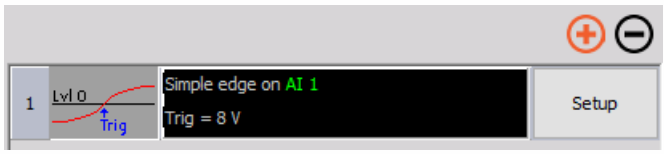
Set Alarm conditions

After you have configured add and basics alarm settings, alarm conditions on the lower part of the screen can be added/changed:

- Alarm start condition or "Don't start" condition
- Alarm stop conditions (this part is displayed only for On condition Alarm reset settings option

Add (New) Alarm conditions

When you press the Add alarm button beside Alarm start/stop condition or "Don't start" condition, a new alarm condition line is displayed:



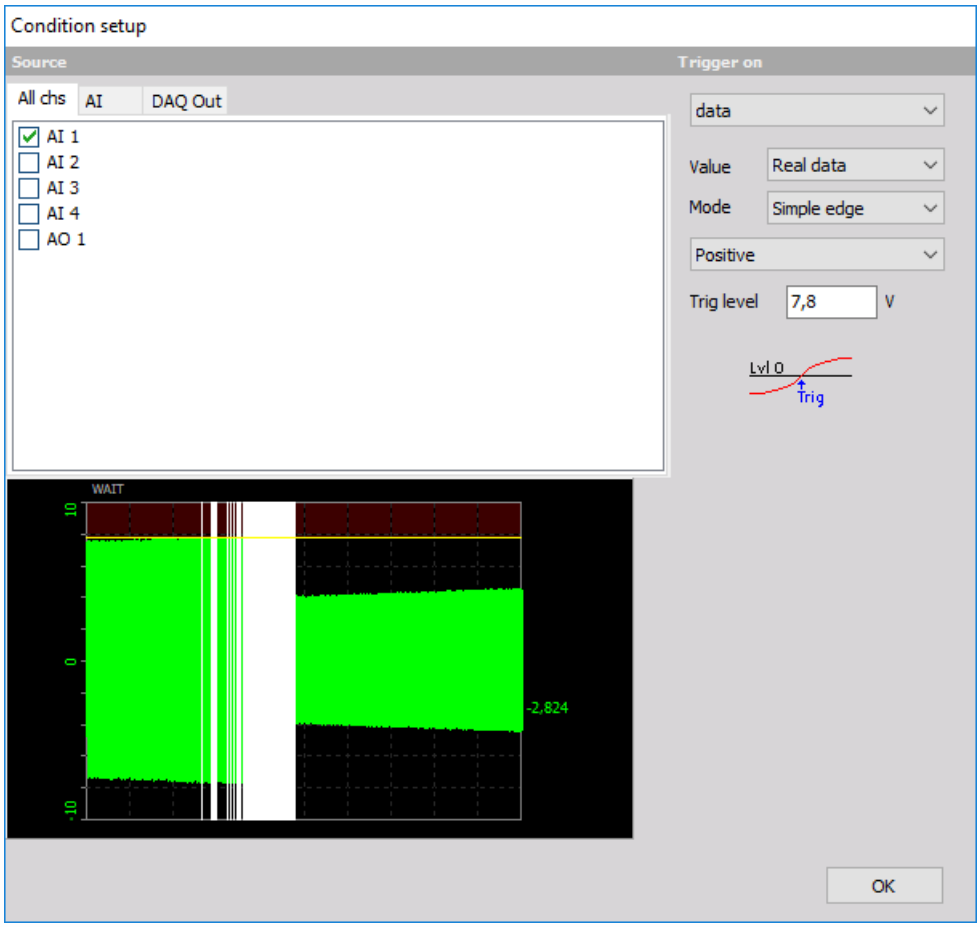
The alarm monitoring settings are similar to trigger conditions, for help with Trigger conditions see -> Setup -> Recording setup -> Setup - Triggers.

NOTE: **The alarm monitoring function has no influence on recording functions. It is only used to define the state of the digital output channels.**

Condition setup

To change the default setting, press the Setup button in condition line. The following window

appears:



Now you can define the alarm condition. In the left-lower area of the window, you can see the current signals, which should help you to set the condition faster. After you have done all settings, press the OK button to accept the changes. The new condition is now available.

If you have more than one alarm condition defined, they are combined by OR condition. There is no relation between the output channels; therefore you can use the same or different conditions for different output channels.

The alarm monitoring conditions are similar to trigger conditions.

For detailed information about Condition setup see -> Setup -> Recording setup -> Setup - Triggers.

Change Alarm conditions

To change alarm conditions, select the condition (click once on it) and press the Setup button again ton invoke Condition setup window where condition settings can be changed.

Remove Alarm conditions

To remove an alarm condition, select the output channel and the condition to delete by clicking

on it. After you have selected the condition in that way, press the Remove button.

VIDEO

Dewesoft Video Acquisition module offers the possibility to acquire video signals together with other sources. A wide range of cameras is supported.

- **Required hardware** - Any AD card, Web cam, GigE cam, Photron FastCAM cameras and Basler/Dewecam
- **Setup sample rate** - At least 1 kHz

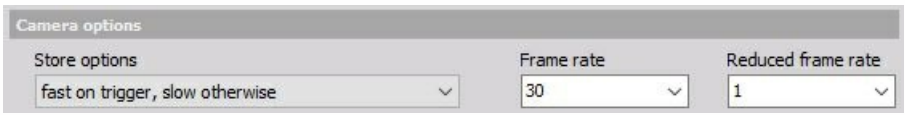
For additional help with Video Acquisition visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training.

For additional Dewesoft manuals visit -> Dewesoft Web page -> Support -> Downloads.

For detailed information about High speed Photron cameras setup see -> HS Video.

When you select a Video tab on Dewesoft Setup screen, on lower part of screen Video (camera) settings will appear:

- Store options - select mode to store your video data
- Frame rate - select frame rate (speed of the incoming frames)
- List of cameras - with information about camera Slot number, Name, Setting column and primary Set column with Setup button to call camera setup window **Video store Options:**



List of connected cameras:

After Camera setup is done press the Unused button - it will change now to Used - to switch on the acquisition.

Camera list

First let‘s look at the columns that are shown in the camera list, and describe what each is for:

Camera name - Manufacturers name of the camera connected to the system. **Camera status** - Displays the status of camera and the used frame rate.

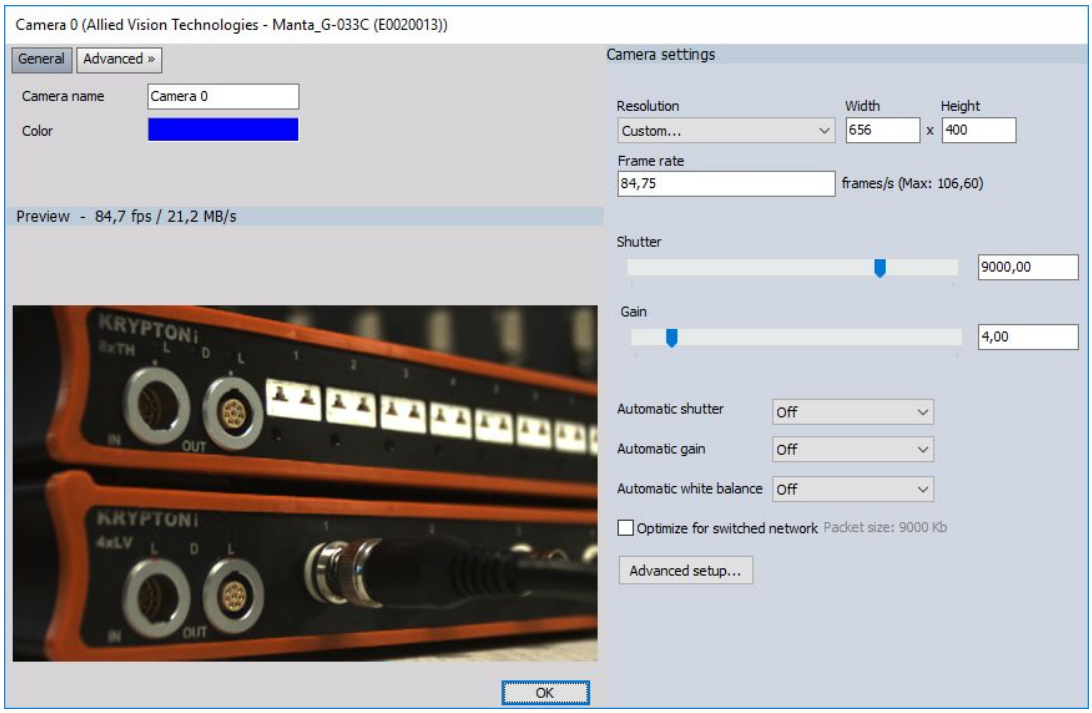
For information about Channel grid see -> Setup -> Channel grid.

Camera setup

Press the Setup button of the camera on the list of connected cameras you want to change and the Camera n[type] setup window will open. This window consists of three main part:

- **Basics settings** - Just click into the Camera name field and you can enter the new camera name as usual text. Navigation keys are the same as in other Windows application. You can also enter this text on the Video setup screen.
- **Preview** - Preview of the captured picture.
- **Cameras settings** - Depend on used camera (HW) - see below. It can contain: Mode setting, Frame cut settings, Compression and Resolution setting, Picture setup settings…

Camera setup:



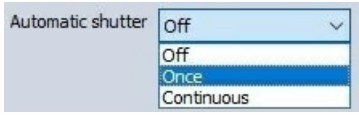
Mode - to set the Frame rate of fps, input desired value and confirm with keyboard key Enter.

Frame cut - when we select higher frame rate, the resolution can't be 640x480 (VGA), but it is reduced (e.g. by 200 fps is reduced to 640x242); with selecting Horizontal, Proportional or Custom, we can choose how the camera cuts pictures. The only way to increase the frame rate is to reduce the height of the picture (reducing the width will not increase maximum frame rate).

Picture setup - Shutter speed depends on the selected frame rate (e.g. can't be longer than 5 ms to be able to acquire 200 fps):

- lower shutter speeds will reduce the smearing of picture with fast movements, but will also reduce the brightness of the picture, therefore we will need either a strong light or we will need to increase the Brightness and Gain (on the other hand this will increase the noise on the picture and will reduce the picture quality).

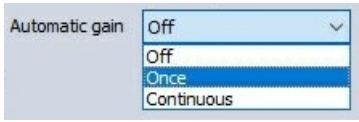
Automatic shutter



Automatic shutter can be set to

- Off(manual),
- Once (automatic adjustment of the shutter to current picture),
- Continuous (continuous adjustment of the shutter)

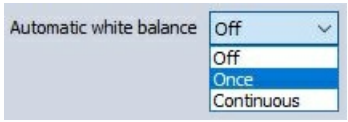
Automatic Gain



Automatic gain can be set to

- Off (manual),
- Once (automatic adjustment of gain to current picture),
- Continuous (continuous adjustment of gain)

Auto white balance



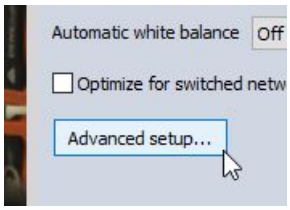
- Off (disabled)
- Once sets the color balance of the picture. This function assumes that the color average of the picture is white, therefore we should select this option withholding the white piece of paper in front of the camera for the best effect.
- Continuous (continuous adjustment of WB)

The settings of the web and handy cam depend on the capabilities of the camera. There is a huge difference between one webcam to another in terms of speed, picture quality and available functions. Some cameras have automatic shutter and automatic gain, there are few even with automatic focus.

Some cameras have different compression types like YUV or I420. This means that each pixel will not have 24 bits of data (8 bits of data per color), but less. In short, using such modes will result in smaller picture sizes and will, in the end, reduce the data file size, but the colors might not be as perfect as with RGB (uncompressed). However, the human eye is much more sensitive to scales of gray than to shades of colors. These compression algorithms use exactly this fact, therefore we might not even see any difference.

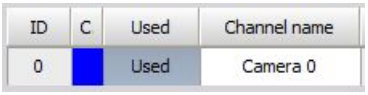
Characteristics of Web and handy cams changes and from themes of their settings so often that it doesn't make sense to write any camera brand in the manual. So, it is worth asking what is the latest and greatest. We are listing some typical settings.

- **Direct setup** - Depending on the camera and it’s capabilities you can change settings like: Compression, Resolution and Frame rate. When Advanced setup is selected you can adjust cameras Properties. This SW is created by the camera producer.



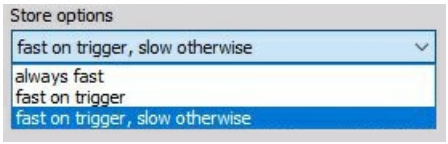
- **Picture setup** - Whit sliders you can change Exposure, Brightness, Contrast, Hue and Saturation or check Auto for automatic setting. With Default button, we call back default camera picture settings. The cameras usually offer Custom settings, which will show in the cameras Properties window all the special functions of the specific cameras like flipping or rotation the picture.

After you have done your settings press OK to leave the camera setup. Now you have to set the desired cameras active for video data acquisition. Press the Unused button - it will change now to Used - to switch on the acquisition:



Store options

Dewesoft offers three different ways - Storing strategies to store your video data. Select option from the drop-down list:

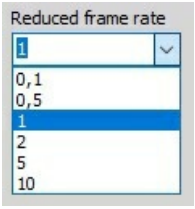


- **Always fast** - the data will be continuously stored all the time to the file with the selected full image acquisition rate - full speed.

If our data consists of events which can be captured, we can choose to store with two options for triggers. The trigger event can be defined in the software and then Dewesoft will wait for this event and store only the portion of interest.

- **Fast on trigger** - the video data will only be stored with the full image acquisition rate when trigger condition is true
- **Fast on trigger, slow otherwise** - to be able to acquire data with two speeds: stores data with a reduced image acquisition rate until the trigger condition became true - after then full speed is active. When you select this option new select field for Reduced frame rate will appear. Simply select the value from the drop-down list.

Example: if we select 0.1 seconds, this means that the video picture will be stored every 0.1 seconds



To set-up system triggers go to Dewesoft Storing tab.

For detailed information about set up trigger see Recording setup.

GIGE

Introduction

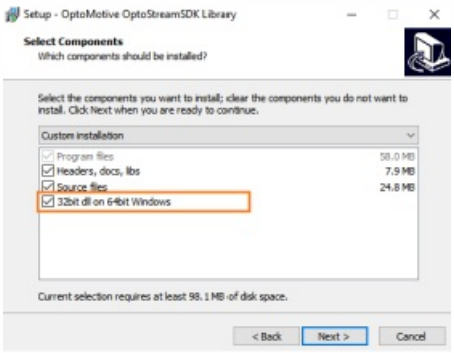
In Windows 10 NDIS driver 5.x is not available anymore, that means that GigE v3.6 and lower versions are not supported anymore (since those versions depend on NDIS 5.0 driver). GigE v4.1 with support for NDIS 6.x is intended to replace the old driver. The new driver supports Windows 7 and higher. It replaces the old package with the same functionality and will be developed further.

GigE driver v4.0 installation

If you are making a clean install, just install new SDK and Dewesoft driver:

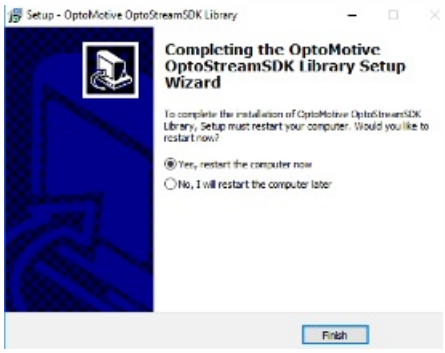
- OptoMotive_OptoStreamSDK_V2_7_4_0.exe (OptoStream SDK and FilterDriver)
- GigECamera.cdv (Dewesoft driver, copy the file to Addons folder)

During SDK installation make sure to check “32bit dll on 64 bit Windows” under installation.

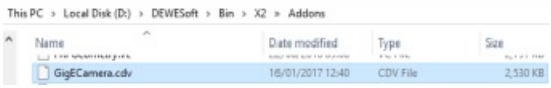


If you have the previous installation of GigE drivers (Smartek_GigEVisionSDK + GigeCamera v3.6 or lower driver) than you need to remove them first. That includes removing of SDK and FilterDriver.

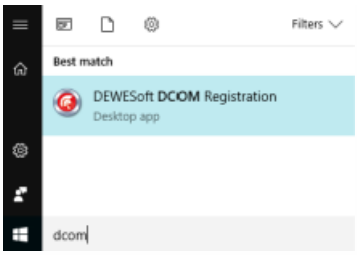
After the installation, reboot the PC:



After logging back in please copy "GigECamera.cdv" to the Addons folder in Dewesoft installation folder: e.g. "D:\Dewesoft\Bin\X2\Addons" folder.



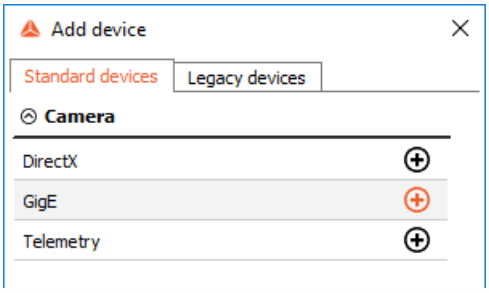
Run the "Dewesoft DCOM Registration” to register the plugin.



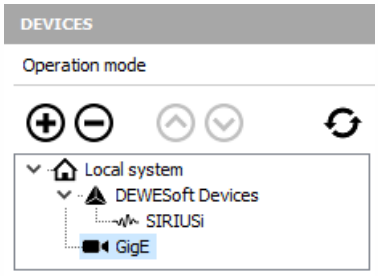
Check the Addons option and press the Register button to register new add-ons.

Setting up the GigE Add-On in Dewesoft

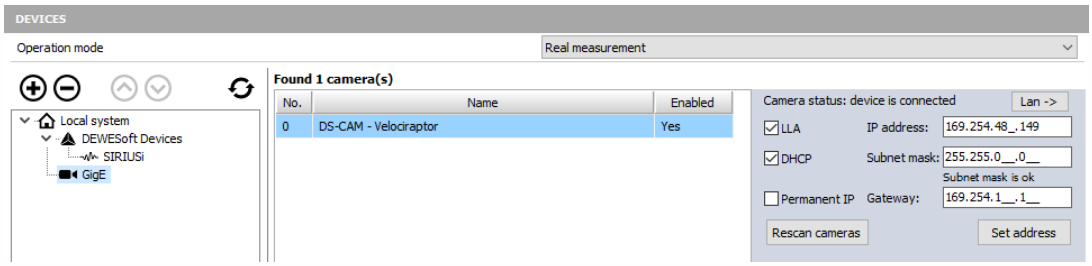
To add a new GigE camera go to Options -> Settings -> Devices -> Add device, and an Add device window will open:



Under Standard devices -> Camera, select and click on GigE tab, that will create a new system.

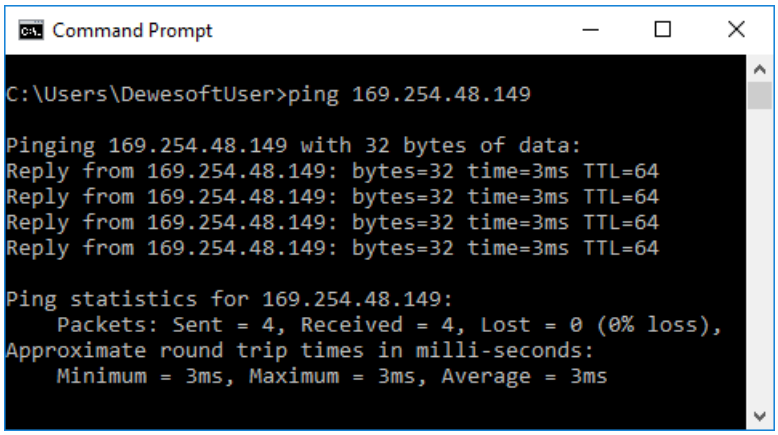


Please wait until the program finds the connected GigE camera. If the camera is not detected automatically you can use the refresh button.

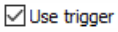


You have to enter the camera's IP address manually and you have to be in the same subnet. You also can use the command line to ping the camera's IP address to make sure the IP configuration is right.

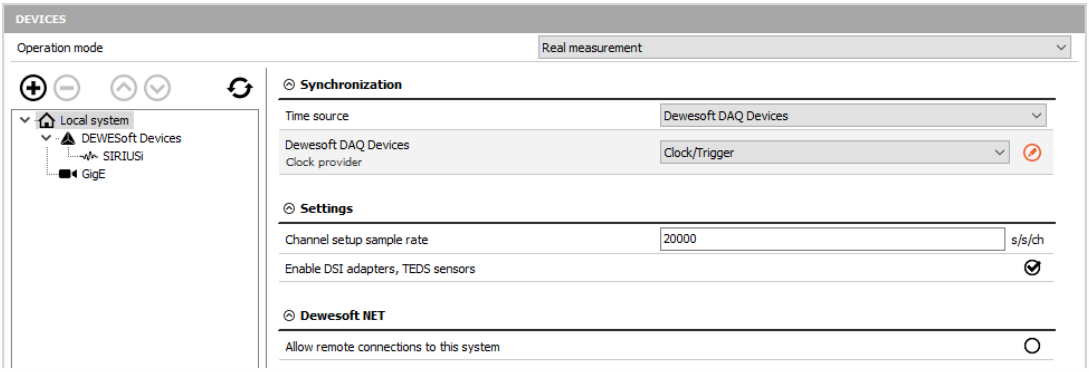
Here is an example:



Click on GigE option under Dewesoft devices. You will see the connected cameras and the status of devices.



To activate "Use Trigger" set your Dewesoft DAQ device (Sirius or DEWE-43A for example) to be the master clock/trigger provider.



Supported BASLER external cameras

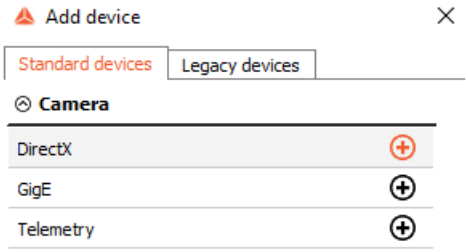
Supported camera models with GPIO (general purpose I/O) line (pin 3 is used as a GPIO line).

- acA640-300
- acA800-200
- acA1300-75
- acA1920-40
- acA1920-48
- acA1920-50
- acA2040-35
- acA2440-20
- acA2500-20

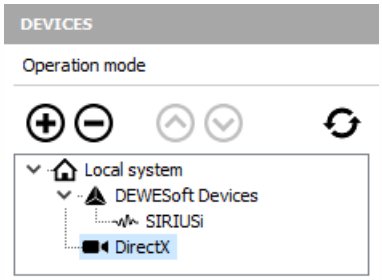
DIRECTX

Setting up the DirectX Add-On in Dewesoft

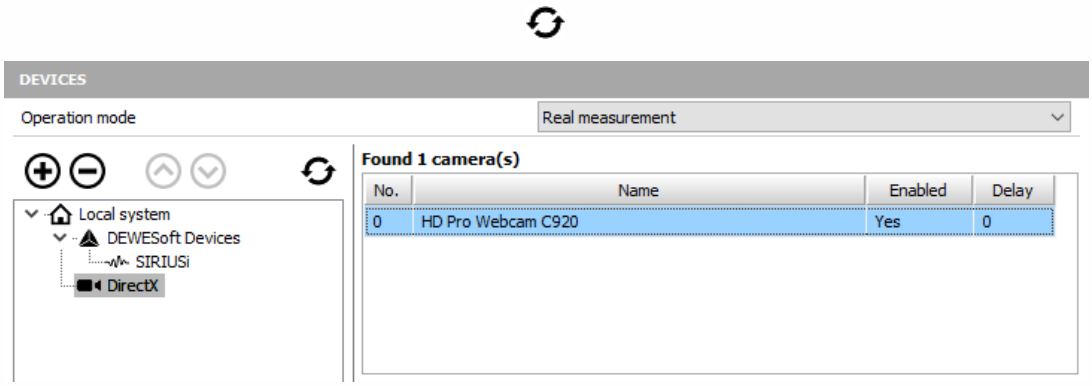
To add a new GigE camera go to Options -> Settings -> Devices -> Add device, and an Add device window will open:



Under Standard devices -> Camera, select and click on GigE tab, that will create a new system.



Please wait until the program finds the connected DirectX camera. If the camera is not detected automatically you can use the refresh button.



No other additional settings are required and you can close the Settings and proceed to the camera options in Video module.

PHOTRON

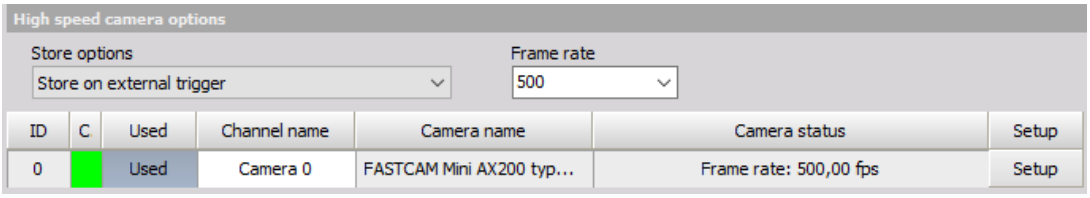
Features:

- supports Photron FastCAM cameras (on PCI, Ethernet and FireWire interfaces)
- supports multiple cameras simultaneously
- automatic synchronization with analog data
- supports Dewesoft triggers and external triggers

Installation

For installation and hardware setup please refer to the -> Dewesoft Web page -> Support -> Downloads -> Photron High speed-Camera Plugin - Manual.

Software setup



In the high-speed camera options section, there are two combo boxes. The first one is for Store options settings and the second one is for Frame rate settings.

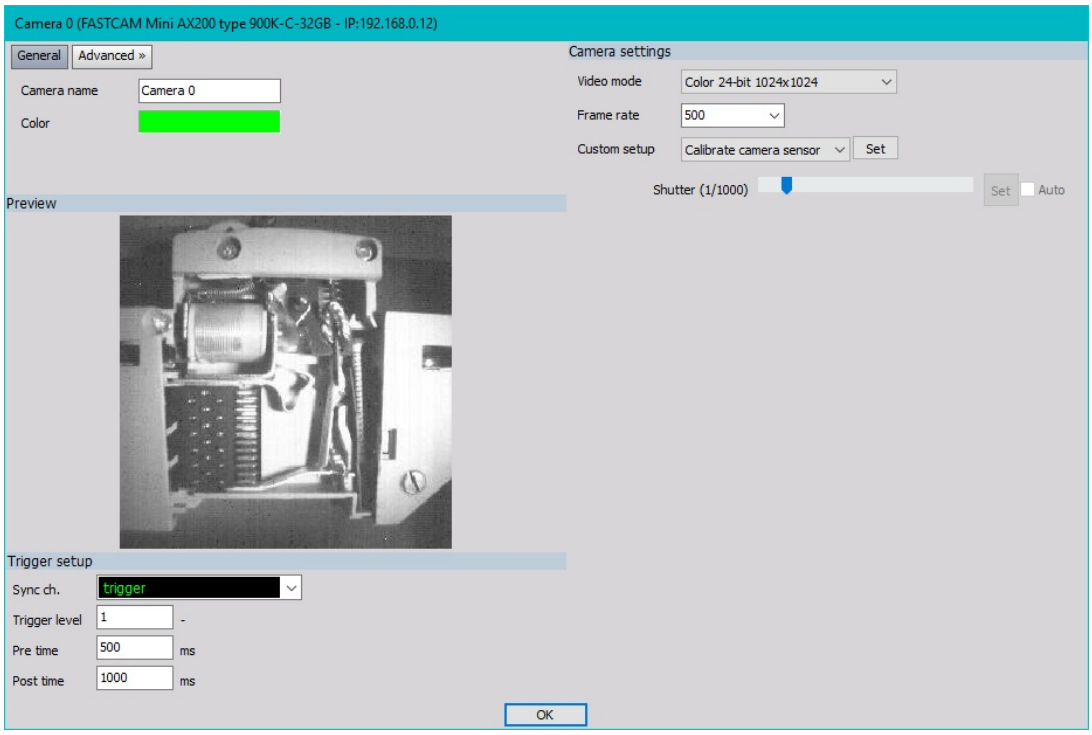
With store setting you select the way camera will be triggered. You can choose between the following options:

- **Store on start of acquisition** - the camera will start at the beginning of the recording.
- **Store on Dewesoft trigger** - the camera will be triggered by a trigger sent from Dewesoft (fired by some analog data or forced with trigger button in Dewesoft recorder).
- **Start on external trigger** - the camera will be triggered with the external trigger. Refer to Photron Manual for information on how to connect and apply external triggers.

Frame rate selected here will be applied to all cameras. If any camera does not support the selected frame rate, the closest one will be used. You can set the frame rate for every single camera in camera setup. There is no need that all cameras have the same frame rate.

Camera setup

There are two important groups of settings in Camera setup -> Camera settings and Trigger setup. You can also find Basic settings group where you can adjust camera name and Preview group where you can see preview used for picture setup.



Camera settings

Here you can set Resolution, Frame rate, Shutter speed and Sensor gain (if the camera supports hardware sensor gain) parameters. You can also calibrate the camera sensor here. The options are depending on the camera model and one from another. For example, at resolution 1024*1024 1000fps is the fastest sample rate, but at resolution 256*256, sample rates to 10000fps are possible.

PFV software can be used to change other parameters not supported by Dewesoft (if the parameter is not supported in Dewesoft, Dewesoft will leave it as it was).

The camera settings are applied to the camera as soon as something is changed, so the preview is always up to date with current settings.

Trigger setup

Synchronization channel and trigger level parameters take care of camera synchronization with analog data. If the camera is triggered with an external trigger then the same signal can be used for synchronization. If Dewesoft trigger is used then camera trigger out signal can be used for synchronization. The trigger signal is usually just 2-3us 5V pulse, so counter or high enough sample rate must be used to catch it.

Refer to Photron hardware manual for further information on camera out signals.

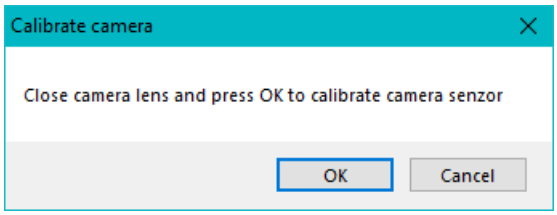
Pre and Post time are parameters for setting time of recording before the trigger and after the trigger. The maximum time of recording depends on Resolution, Frame rate and camera memory.

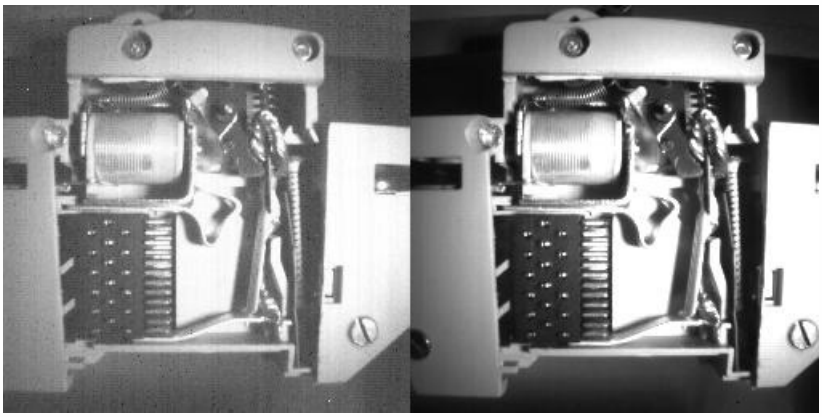
Camera sensor calibration

Each camera comes with two sensor calibration files. Here is an example how the calibration files are named (e.g. for FastCam 1024PCI with SN: 145509164):

- PixelGainData_Default_1024PCI_1455_0009_0164.gdf
- shadingdata_1024pci_1455_0009_0164.gdf

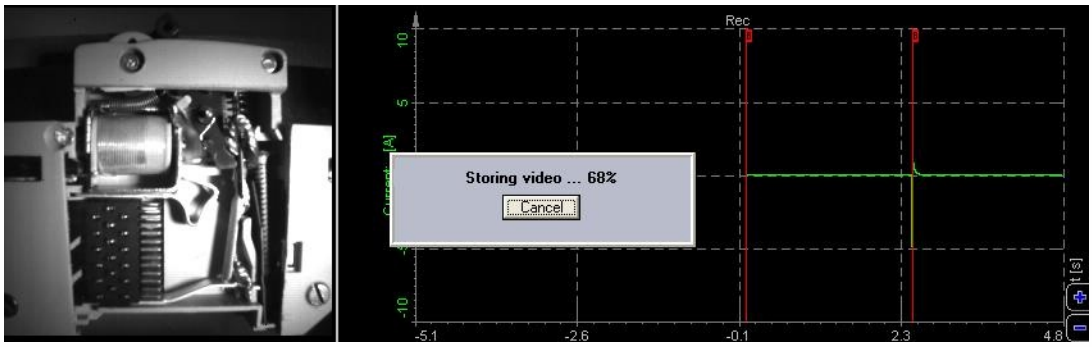
If you copy these files to Dewesoft Addons folder they will be used automatically. Files are optimized for 1000fps frame rate and the picture will not be optimal when other frame rates are selected. If you do not have these files or factory calibration data is not satisfying or from any other reason the picture is grainy and/or contains fixed shading pattern, you have to calibrate the sensor to get a better picture. You can do that by selecting Calibrate camera sensor from Custom setup and press the Set button. The Calibrate camera window will pop up. Just follow the instructions (Close camera lens and then press OK) and the sensor will get calibrated.





Recorded frames transfer

Frames data is recorded into camera memory. Downloading starts when Dewesoft recording stops. Download time depends on the recording time (Pre and Post time), resolution and frame rate and can take a lot of time. Progress is shown in the transfer window. Please wait while all data is transferred from the camera. For situations, where it is obvious that the test was not successful, there is also a Cancel button which will stop the transfer process.


Rec
10
5
-10
-5.1
-2.6
-0.1
2.3
4.8
t [s]
Storing video ... 68%
Cancel

SECURITY

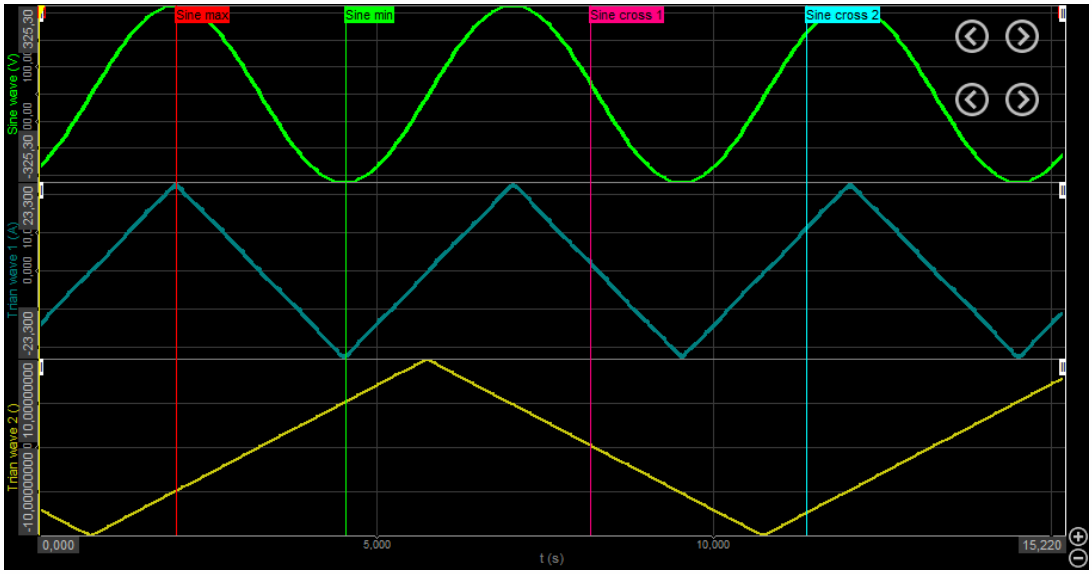
Enter topic text here.

USER INPUTS

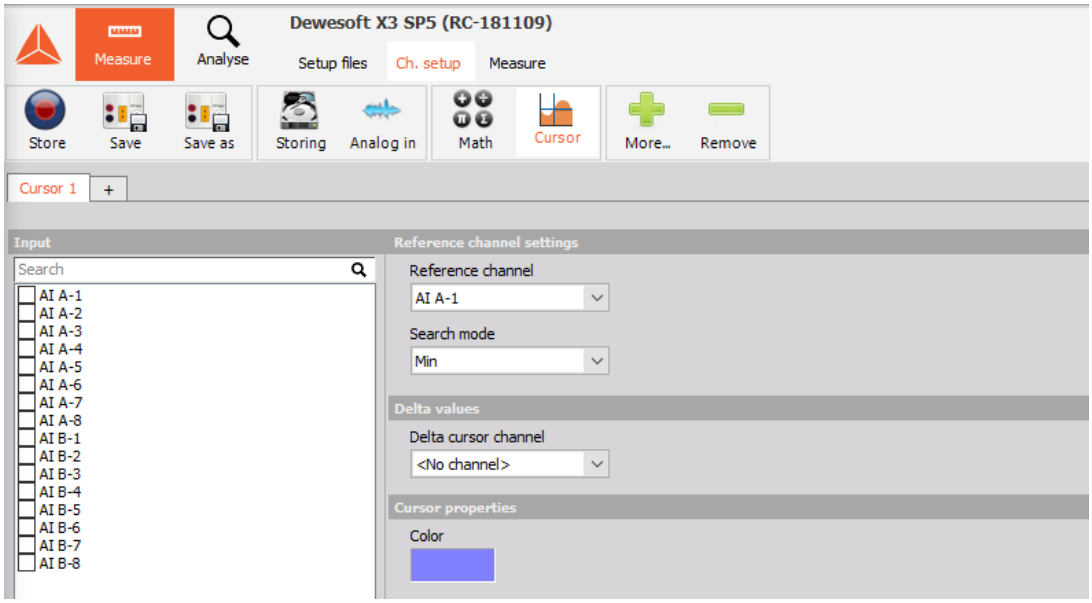
test

CURSOR

The cursor module allows users to interactively search for level crossings in analysis, as well as find local minimums or maximums of channels. These values are marked on Recorder and XY Recorder visual controls with specially labeled and colored cursors.



While this is a feature that is mainly meant to be used in analysis, the module can also be added and configured in measure mode by finding the Cursor label under the new module interface (for instructions on how to add a new module see -> Setup -> Add module). Once added you should see the following screen:



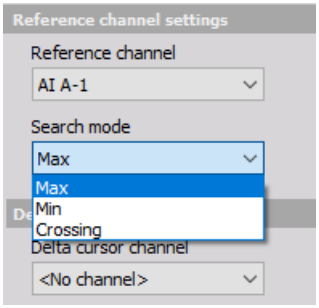
The Cursor module will return the value of any input channel when a condition for the defined reference channel is met. These channels can then be used in additional formulas or displayed on visual instruments, such as the digital meter. It is important to know that the extracted values are

single value channels, meaning that they only hold one value for the entire length of the data file.

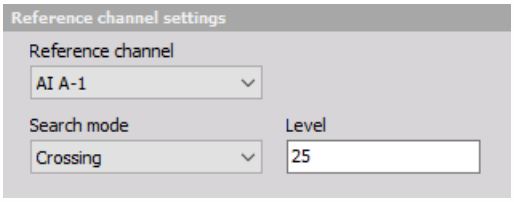
REFERENCE CHANNEL SETTINGS

The reference channel is the channel on which we are searching for specific values. Each cursor can only have one reference (conditional) channel, but we can tie multiple inputs (latched channels) to it. After selecting the reference, we should select the search mode. Three settings are available: Max, Min and Level crossing.

In the Max or Min mode, Dewesoft will find the maximum or minimum value of the reference channel in the visible timeline. Because of that, whenever you zoom into the file and recalculate, the returned maximum or minimum value will be the local extreme of the dataset.



Whenever the user selects the Crossing search mode, a new field will open up where the desired signal level should be entered. As the module can linearly interpolate data, it is able to extract values with a higher precision than just by taking the nearest synchronous or asynchronous sample.



For the above example, Dewesoft would find the two closest samples of the "AI A-1" channel, one that is above 25 and one that is below 25. Using linear interpolation, Dewesoft will extract the exact time between the two samples where the signal hits the level of 25. Based on that time, the module will also interpolate all of the input channels that the user had selected.

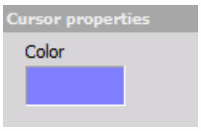
DELTA VALUES

When the user has defined two cursor modules, Dewesoft will allow you calculate delta values. This is done by selecting the delta cursor channel from the drop-down list of available cursor modules.

In the above image, we're editing the setup of the "Cursor 1" module, and we've defined the "Cursor 2" module as the delta cursor channel. That means that any inputs that the two modules have in common will be subtracted from one another ("Cursor 1" - "Cursor 2"), resulting in new channels. The module will also add an additional delta t channel, which will provide the time difference between the two events.

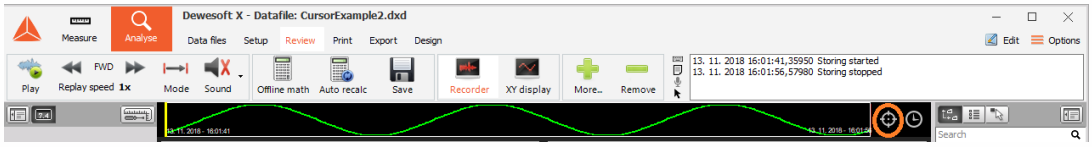
CURSOR PROPERTIES

Under cursor properties, users get a chance to adjust the color of the cursor that will be displayed on the Recorder and the XY Recorder, if the appropriate signals are added. The name of the cursor that is displayed is taken directly from the name of the module.

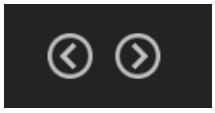


USAGE

You can access the cursor setup interface directly out of the review window using the cursor icon, which is located at the end of the data file preview and to the left of the time selector icon.



As there can be several level crossings within a data file, users can use control channels generated by the cursor module to move between events. To change the location, simply use the "Search position" channel in combination with the "Input control display". You can either enter the location index manually, but usually the "Next prev button" is a better choice, as it allows the user to move between cursor positions in either direction with a single mouse click.



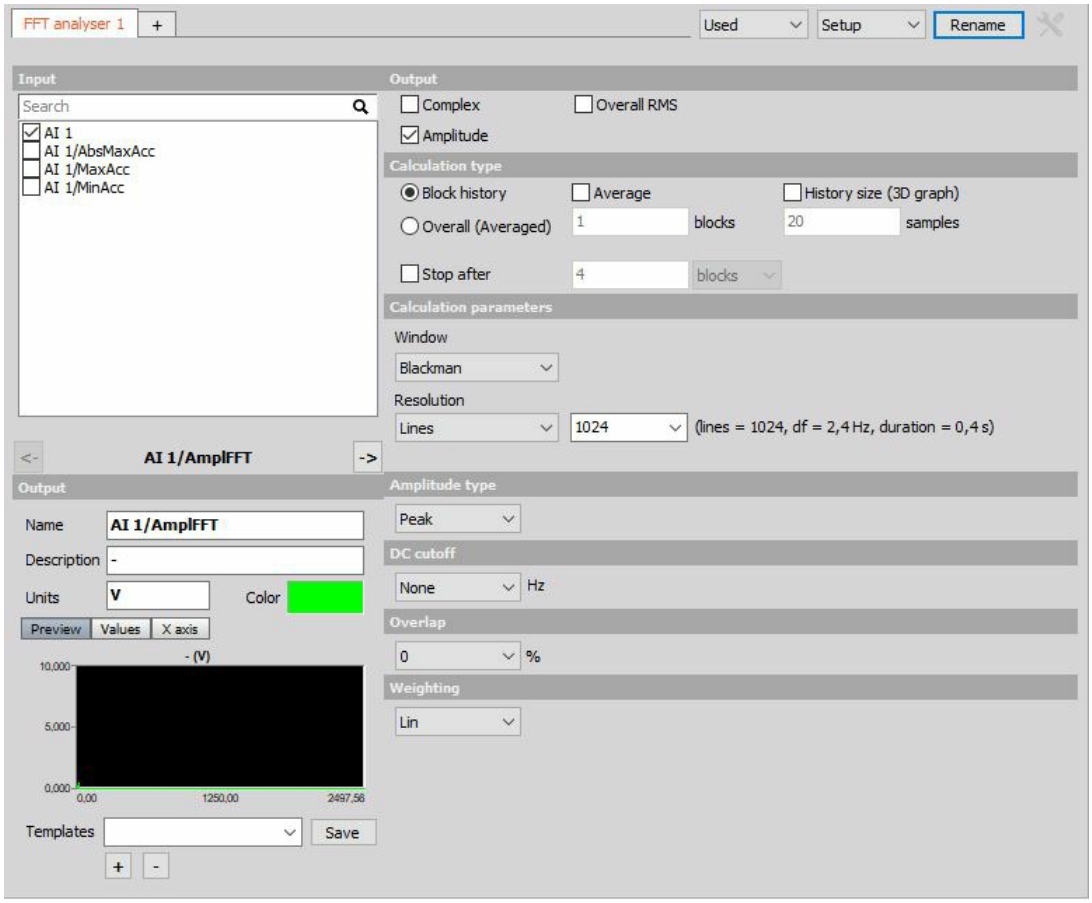
For more information about the "Input control display", please check out the Control element section of the manual.

FREQUENCY DOMAIN ANALYSIS

FFT analyser in Dewesoft has it all: top performance, advanced cursor functions, very high freely selectable line resolution, flexible averaging as well as many advanced functions for in-depth analysis.

FFT ANALYSER

When you press the Setup button on FFT analyser, the following setup window will open:



Output

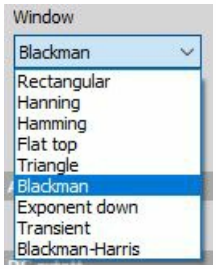
- **Complex** - outputs are phase, imaginary and real part of the signal.
- **Overall RMS** - output is only one value for the whole measurement.
- **Amplitude** - output is the amplitude of the signal.

Calculation type

- **Block history** - uses blocks to calculate FFT spectrum.
- **Overall (Averaged)** - gives only one averaged FFT spectrum at the end of the measurement. It will average all the blocks in the signal and the output will be only one FFT for the whole measurement.

Calculation parameters

- **Window** - Rectangular, Hanning, Hamming, Flat Top, Triangle, Blackman, Exponent down, Transient and Backman-Harris



- **Resolution** - defined in the number of Lines, with the delta frequency Df [Hz] or with Block duration [s].

Amplitude type

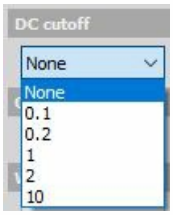
The Amplitude type display section defines display in Y-amplitude axis.

In Amplitude type we can choose from:

- **Amplitude** - is the pure signal amplitude [V].
- **RMS** - is the RMS amplitude, calculated as Amplitude/sqrt(2) [V].
- **Power** - calculated as RMS value squared [V*V]. **PSD** - calculated as RMS squared, divided by the line resolution and sqrt(2), used for checking the noise [V*V/Hz]. **RMS SD** - calculated as RMS value, divided by the square root of line resolution - also used for checking the noise [V/sqrt(Hz)].

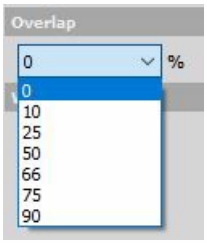
DC cutoff

To remove DC or low-frequency components, select from a drop-down list the DC cutoff filter - lower limit.



Overlap

Overlap defines the percentage of time signal that has already been calculated and it is used again for calculation (example: 50% overlapping means that the calculation will take half of the old data).



Weighting

As a standard, FFT analyser uses Linear Weighting. For sound analysis, special FFT weighting can be set. As opposed to the sound module in math, where the weightings will be calculated in the time domain, this will calculate the sound weighting in the frequency domain. Weighting can be of the following type:

- **Linear weighting** - is linear at all frequencies and it has the same effect on all measured values.
- **A weighting** - is applied to instrument-measured sound levels in an effort to account for the relative loudness perceived by the human ear, as the ear is less sensitive to low audio frequencies.
- **B weighting** - is used for intermediate levels and is similar to A, except for the fact that low-frequency attenuation is a lot less extreme though still significant (-10 dB at 60 Hz). This is the best weighting to use for music listening purposes.
- **C weighting** - is similar to A and B as far as the high frequencies are concerned. In the low-frequency range, it hardly provides attenuation. This weighting is used for high-level noise.
- **D weighting** - was specifically designed for use when measuring high-level aircraft noise. The large peak in the D-weighting curve is not a feature of the equal-loudness contours but reflects the fact that humans hear random noise differently from pure tones, an effect that is particularly pronounced around 6 kHz.

FFT markers

2D graph can display values of the currently selected point with the crosshair cursor. When clicking on such point with the left mouse button, the marker line will be added showing x axis value on the x-axis and showing y-axis vale of a certain point above the marked point. All points can be removed by pressing the right mouse button and select Delete selected marker.

For detailed information about FFT markers see -> 2D Graph.

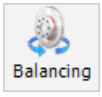
For additional help visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training -> FFT markers.

For detailed information about FFT Analysis see -> FFT Analysis.

MACHINERY DIAGNOSTICS

Dewesoft Machinery diagnostics covers a wide range of measurements in the field of structural dynamics, industrial acoustics and machine diagnostics.

- Balancing - explains procedures for working with Balancing module, which is used to determinate the amplitude & angle of unbalance and calculate correction mass with proper placement angle.



- Combustion analysis - explains procedures for working with Combustion analysis module used to calculate parameters of internal combustion engines.

- Human body vibation - explains procedures for working with Human vibration module, used to evaluate the effects of vibration on the human body.

- Modal test - explains procedures for working with Frequency response analysis used to determine natural frequencies of the system under test.

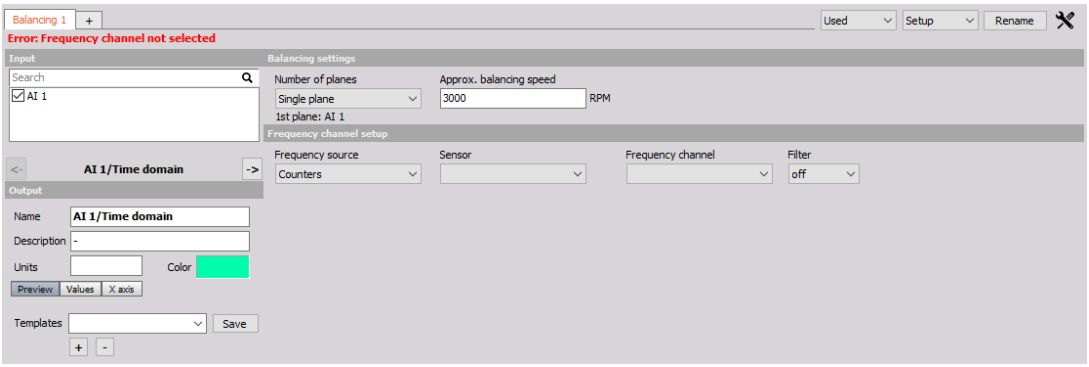
- Torsional vibration - explains procedures for working with Torsional vibration module used to measure dynamic and static bending and vibration of the shafts.

- Order tracking - explains procedures for working with Order tracking module used to extract the harmonics during machine run up and run downs.

For information how to initialize above special Dewesoft modules see -> Add module.

BALANCING

After adding new Balancing module a setup screen is displayed:



In balancing settings we can choose from:

- **Number of planes** - Select from Single plane or dual plane balancing, depending of your application requirements.
- **Approx balancing speed** - Approximate balancing speed is required for Order tracking calculations needed for balancing procedure.

Frequency channel setup lets us select:

Frequency source - Depending on your hardware, select from:

- Counters
- Analog pulses:
- Sensor

Choose the type of sensor used for your application:

- Encoder - 36
- Encoder - 512
- Encoder - 900
- Encoder - 1024
- Encoder - 1800
- Encoder - 3600
- CDM - 360
- CDM - 720
- Tacho (Digital)
- 60 - 2
- 36 - 2
- Tacho (Analog)

Frequency channel - Define the channel of frequency used for determination of rotational frequency.

Filter - Filter is needed to prevent glitches and spikes in digital encoder pulse signal. It can be set from 100ns to 5us.

For information about Counter setup see -> Setup -> Counters.

For information about Balancing procedure see -> Setup -> Theory of balancing.

For additional help visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training -> Balancing.

VISUAL CONTROL INSTALLATION

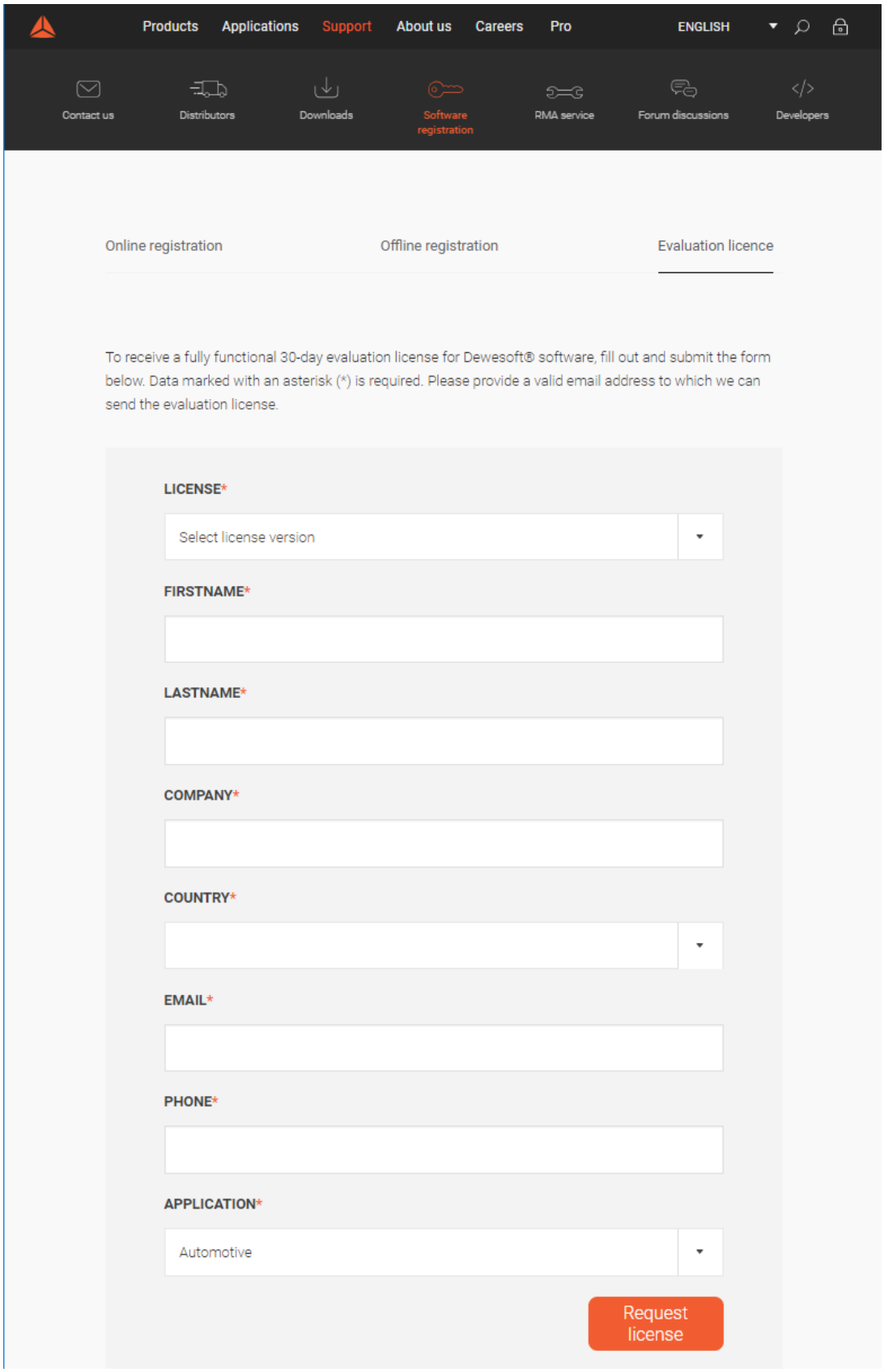
The balancing Add-on is included in the Dewesoft full installer, you just need a license key to enable it.

Download the latest version of the Dewesoft® Rotor balancer visual control instrument here and then extract it to your DEWESoft installation folder (e.g. D:\DEWESoft\Bin\X3\Addons).

The plugin is compatible with Dewesoft® X1, X2 and X3.

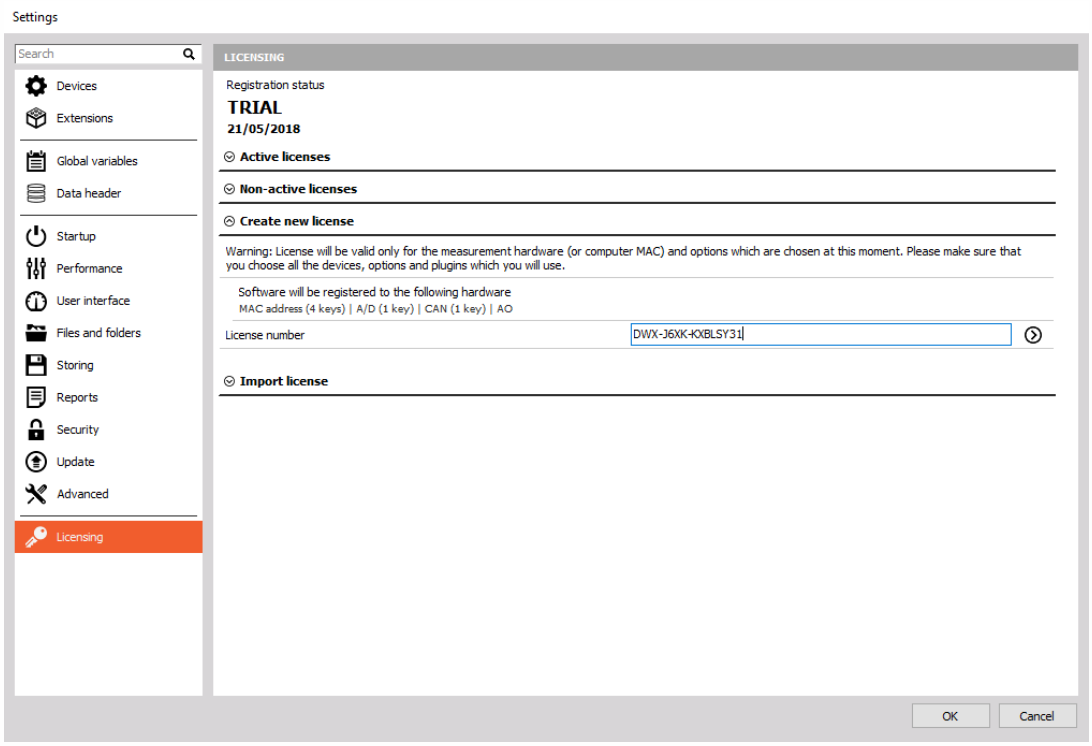
The plugin requires a valid Dewesoft® license. An additional license for the plugin is needed, it can also be written into the Dewesoft® device. To test the plugin you can use a 30-days-Evaluation license.

You can request an Evaluation license from our homepage: http://www.dewesoft.com/registration

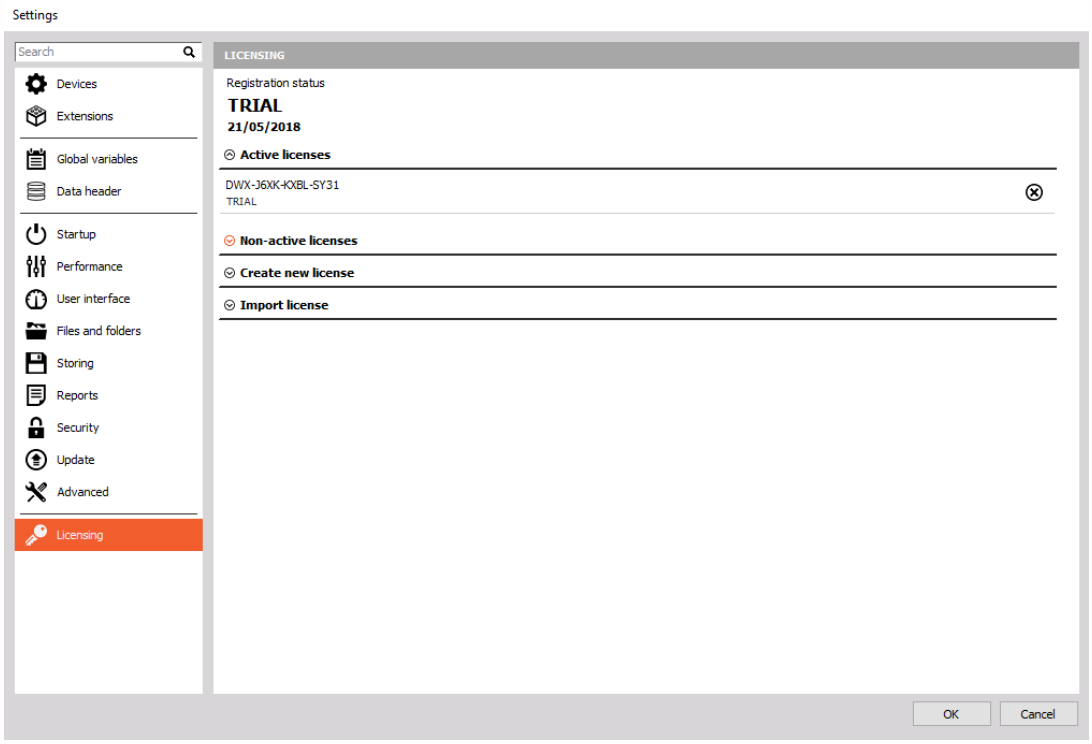


When you have received your trial license key, open DEWESoft®, go to Settings → Settings → Licensing.

Select “Create new license” and enter the license code. Click “ONLINE license registration”.

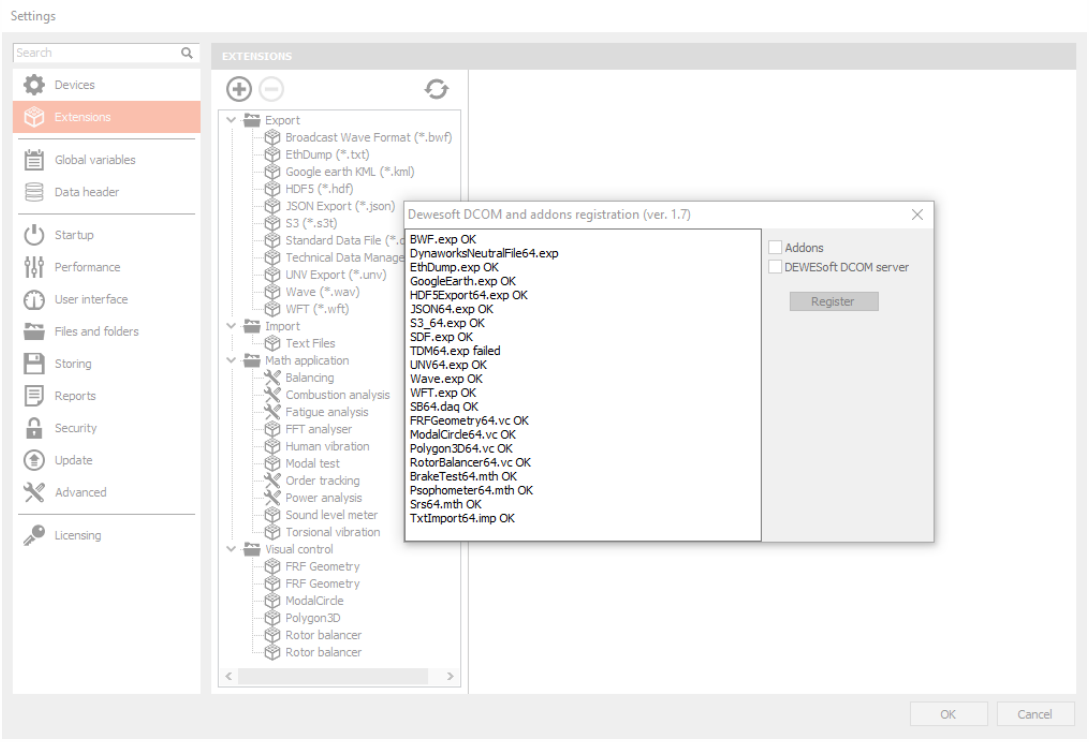


Then your new license key will show up in the list of Active licenses and should have the status TRIAL.



Please copy the latest version of the file “RotorBalancer.vc” into the Addons folder of your DEWESoft® installation. (e.g. D:\DEWESoft\Bin\X2\Addons\), then start DEWESoft®.

The plugin can be found under Settings → Settings → Extensions. If the list is empty, you have to register the plugins first.



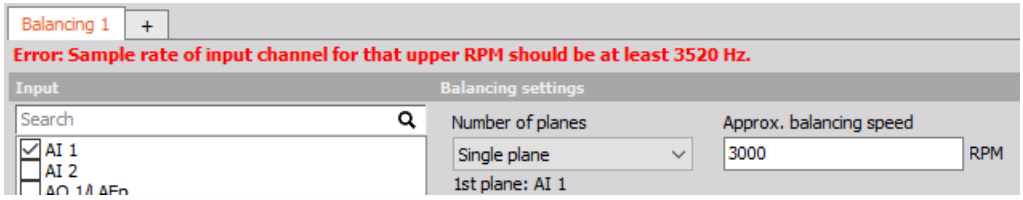
If you are using Windows 7, you must click the Register plugins button (the button with two curved arrows) once and restart DEWESoft before the plugin shows up in the list of Extensions. This usually requires admin rights.

THEORY OF BALANCING

Balanced rotors are essential for most kinds of rotating machinery. Unbalance will create high vibrations causing material defects and reducing the lifetime of a material. In most cases the rotor unbalance is the major problem of vibration, it is related to the first order (= rotational frequency).

We assume, that we consider the so-called “rigid rotors”, which is true for nearly all practical cases. That means the operating speed of the machine is below 70% of its first resonance frequency. The resonance frequency is the critical speed, where structural resonances cause heavy vibrations. At the resonance, the phase is turning quickly and it would be impossible to make a correct measurement.

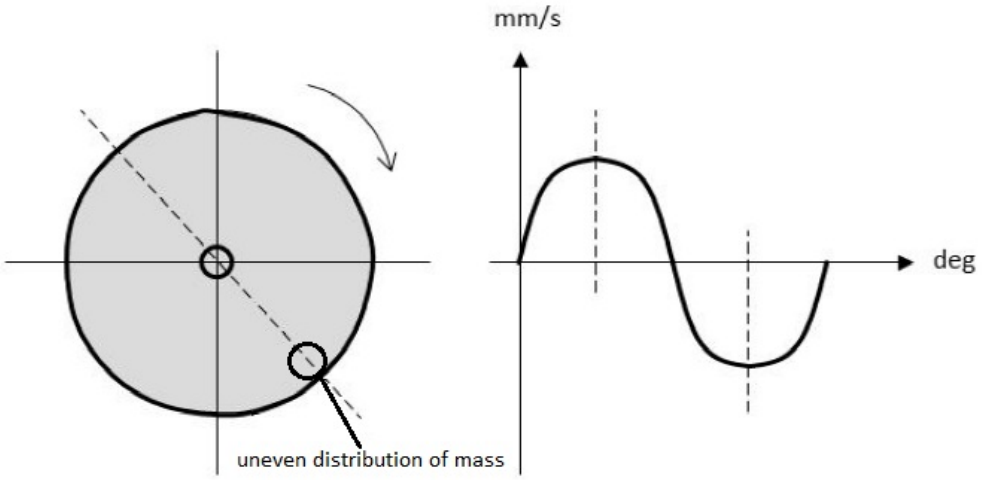
NOTE: **The requirement in terms of sampling rate depends on the first order (e.g. 3000 RPM/60 = 50 Hz → required sampling rate ≥ 3520 Hz). Also, a precise vibration sensor signal is mandatory**



NOTE: **The minimum sample rate is calculated from the balancing speed RPM (Max RPM + 10%) and the maximum order that is selected in the settings (32 is default). (3000RPM / 60) * 1.1 * 2 * 32 = 3520 Hz.**

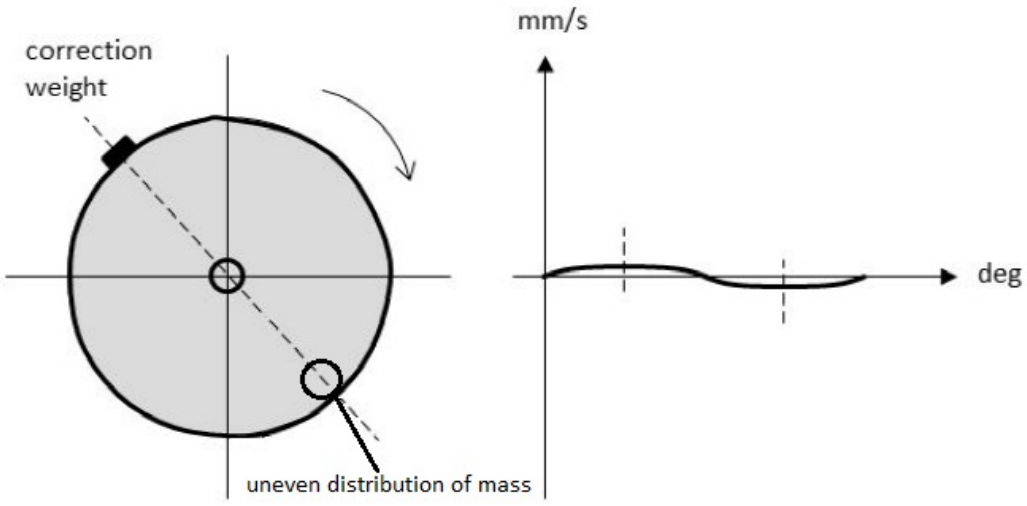
The goal of balancing is to minimize vibrations related to the first order. Basically, it works like this: We measure the initial state, then we add a trial weight of known mass, calculate the position and mass of a counterweight, remove the trial weight and put the calculated weight on the opposite side, to cancel out the imbalance.

When an unbalance exists, the first order (rotational frequency) can be seen clearly. As shown in the example below, on the rotor exists an uneven distribution of mass.



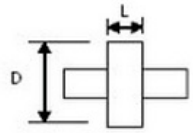
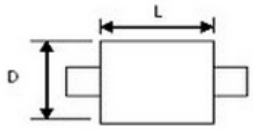
A correction weight is added (or material is removed) on the opposite side, which cancels out the

major part. This procedure can then be repeated until satisfaction.



Single and dual plane balancing

Depending on the machinery, single or dual plane balancing is used. Selecting one plane or two plane balancing generally depends on two factors. One of the factors is the ratio of the length of the rotor (L) to the diameter of the rotor (D). The other factor is the operating speed of the rotor. As a general rule of thumb, we can refer to the table shown below.

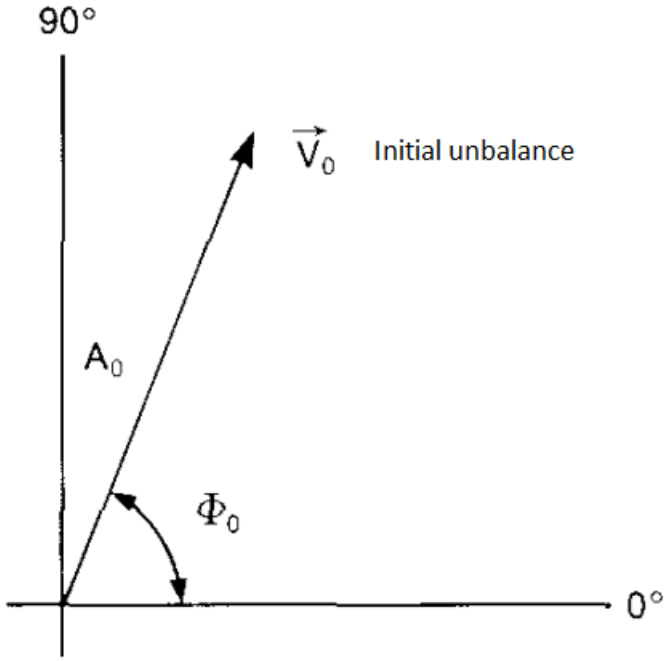
Rotor	L/D Ratio	Balance Correction	
		Single Plane	Two Plane
	Less than 0.5	0 - 1,000 RPM	Above 1,000 RPM
	More than 0.5	0 - 500 RPM	Above 500 RPM

The procedure for a single plane or dual plane balancing will be different, depending on which option is chosen. But basically the following steps have to be taken:

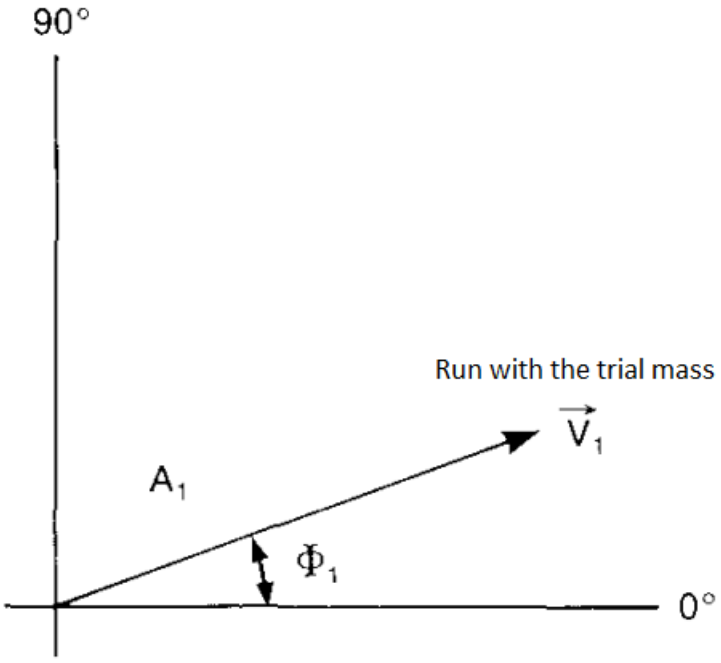
- initial run
- trial run
- correction run(s)

Step-by-step procedure

The first step of balancing procedure is to do an initial run. The machine has to be run up the operating speed. The vibration velocity is detected. The velocity level and phase angle give together a vector that represents the original unbalance of the rotor. The length of the vector is equal to the vibration amplitude and its direction is given by the phase angle.

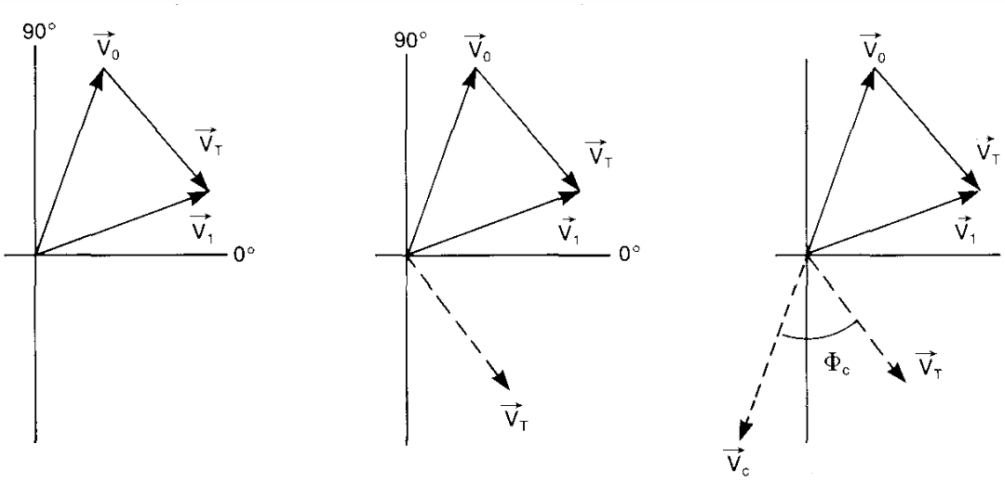


The second step is to add a trial mass. A trial mass has a known weight and it is fixed at a known radius at an arbitrary angular position on the rotor. The machine is again run up to operational speed. We get a new vibration velocity level and a new phase angle. These values represent the resultant effect of the initial unbalance and the trial mass.



The tips of vectors V_0 and V_1 are joined by means of the third vector V_T, which is marked so that it indicates the V_0 to V_1 direction. This vector represents the effect of the trial mass alone. A vector is drawn parallel to the vector V_T, with the same amplitude and direction, but starting at the origin.

In the opposite direction to V_0, there is a vector V_C and it represents the position and magnitude of the mass required to counteract the original unbalance.



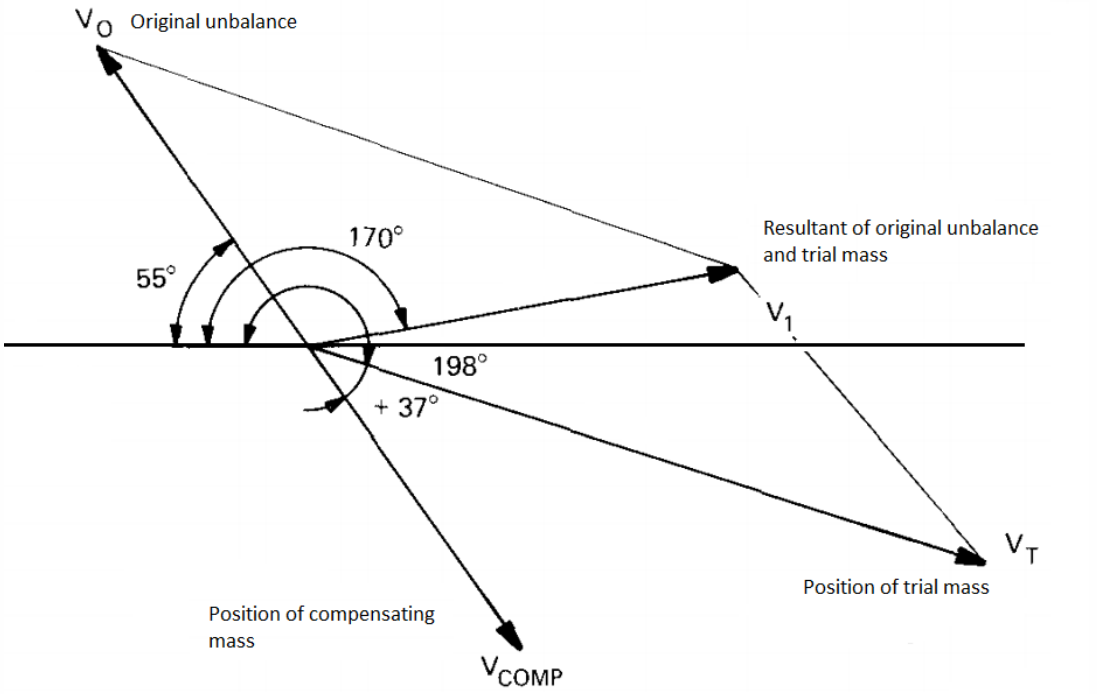
If we assume that the amplitude of the vibration is proportional to the unbalance mass, we get the expression that enables us to find the value of compensating mass (M_{COMP}).

$$\frac{M_{\mathrm{T}}}{\vec{v_{\mathrm{T}}}} = \frac{M_{\mathrm{COMP}}}{\overrightarrow{v_{\mathrm{COMP}}}} = \frac{M_0}{\vec{v_0}}$$

$$M_{\mathrm{COMP}} = M_0 = \frac{\vec{v_0}}{\vec{v_T}} \cdot M_T$$

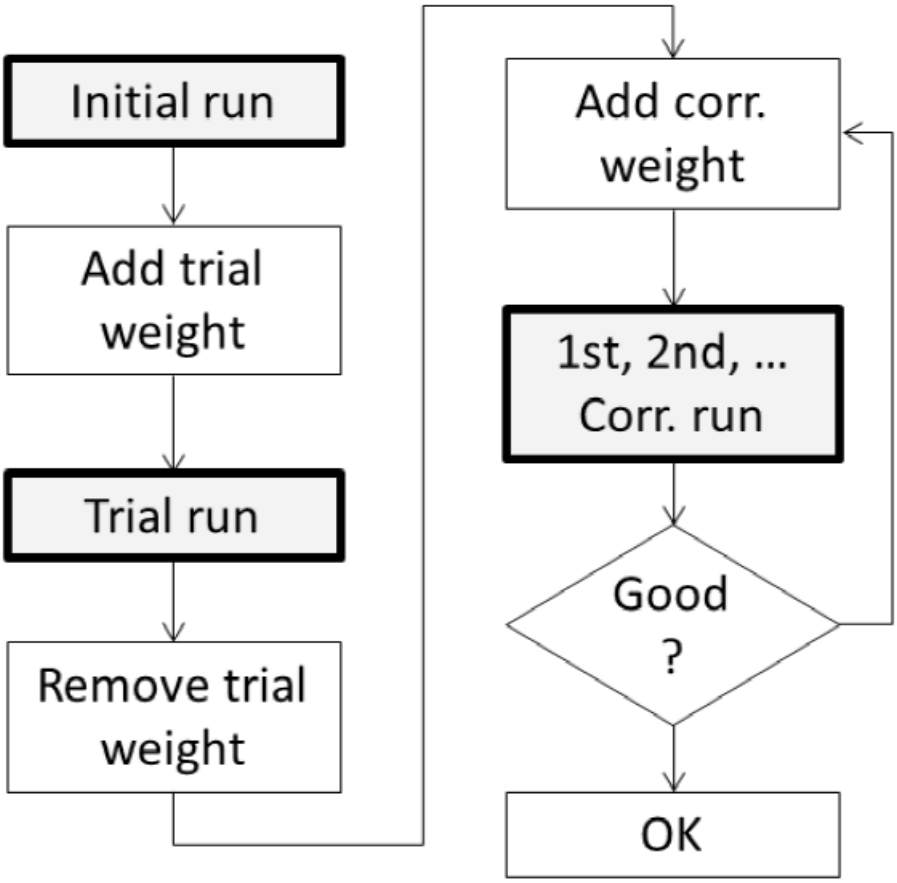
The position of the mass relative to the position of the trial mass can be determined from the vector diagram.

Now we have sufficient information for the vector diagram with vector lengths proportional to the measured vibration velocity levels.



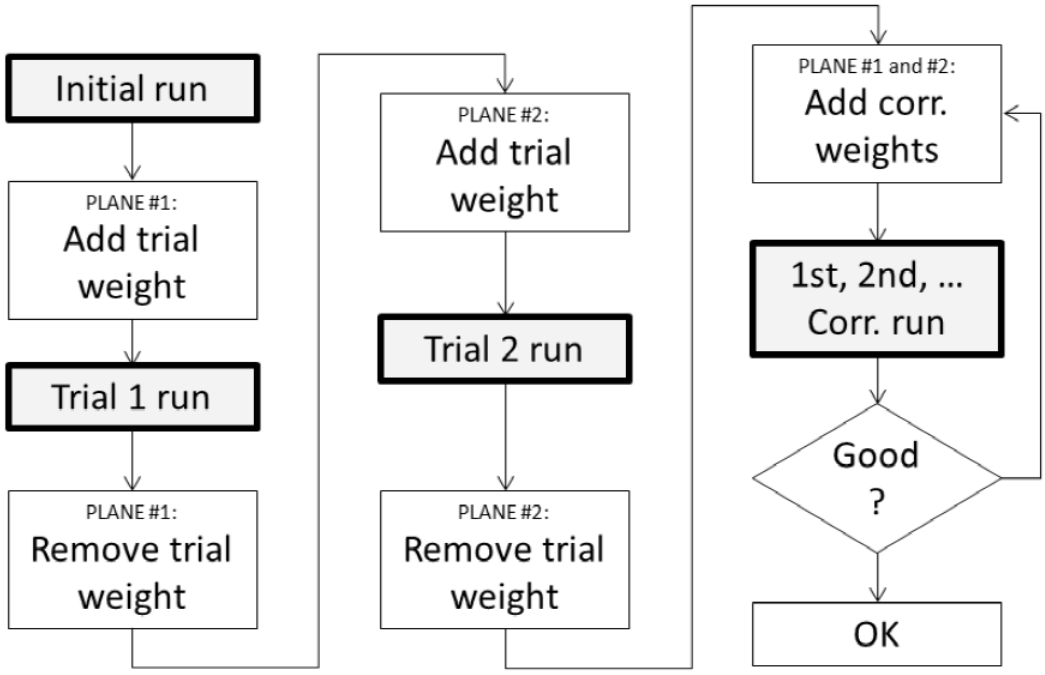
The procedure in DEWESoft is guided by the visual control instrument. The flowcharts below show the routines for single and dual plane balancing. Instead of adding correction weights you can also remove material from the opposite position (angle + 180°).

Single plane balancing procedure



Dual plane balancing procedure

By adding the correction weights for both planes at the same time, we save one additional step.



COMBUSTION ANALYSIS

Analysis of internal combustion engines is possible with Dewesoft Combustion analysis package. During the measurement we can see typical combustion values like: Cylinder pressures, MEP values, heat release (TI, TQ, burn angles), knocking factors,....

- **Required hardware** - Dewesoft Sirius
- **Required software** - Dewesoft PROF or higher and CA option

For additional help with Combustion analysis visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training -> Combustion analysis.

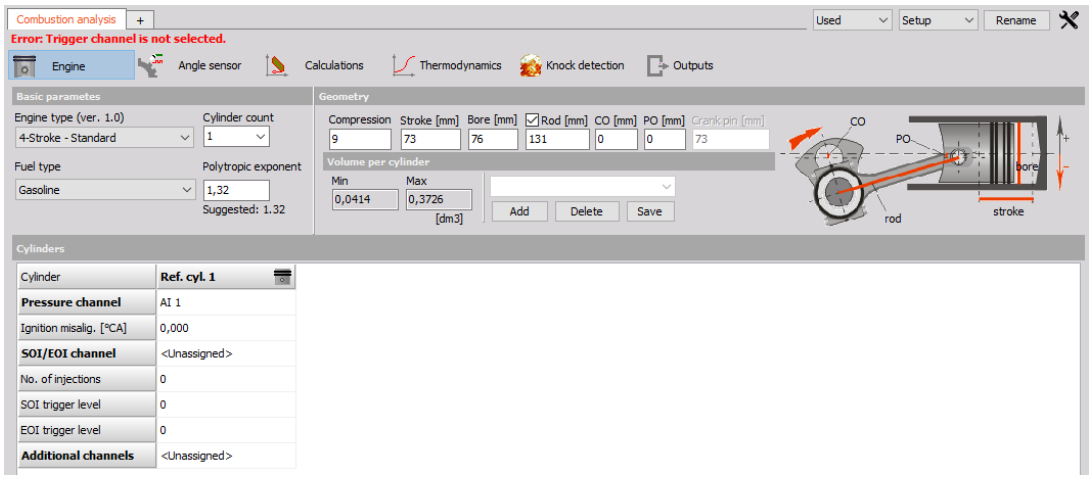
To work with Dewesoft Combustion analyzer we need to do:

- **Activate hardware or modify hardware settings** - analog and counter channels on Hardware setup window
- **Combustion module** - add or delete module.
- **Combustion analysis setup** - to select CA type: Engine, Angle sensor, Calculations, Heat release, Knock detection and then define all settings.

For additional Dewesoft manuals visit ->Dewesoft Web page -> Support -> Downloads -> Combustion Analyzer.

Combustion analysis setup

After adding the new Combustion analysis module initial (Engine) setup screen is displayed:

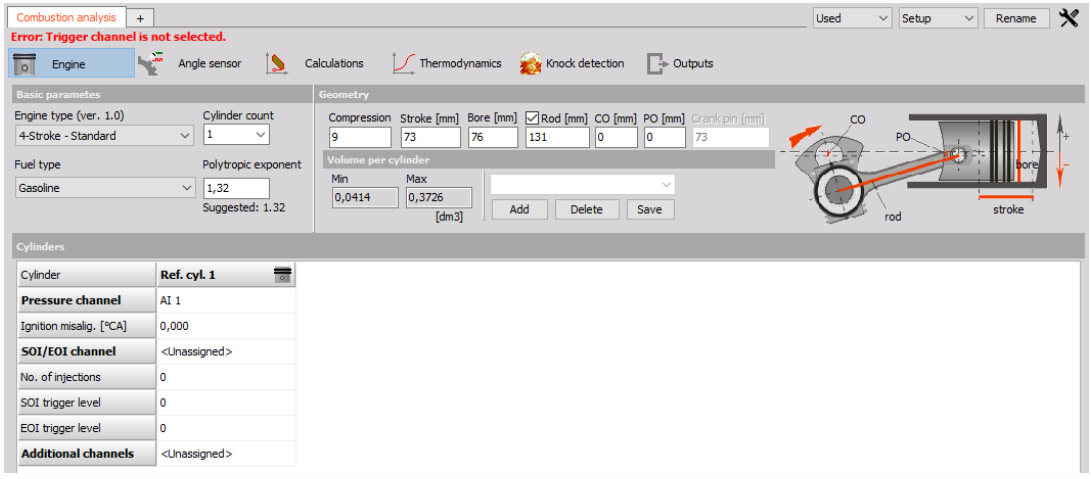


We can set Combustion analysis (CA) in any way we want, but usually, the easiest way would be to select CA type and then define all settings of a particular type:

- Engine
- Angle sensor
- Calculations
- Thermodynamics
- Knock detection

1. Engine setup

After selecting Engine type CA or after new CA module is added following setup screen is displayed:



Settings and entered values for Engine CA type are divided into the following sections:

Basic parameters

Geometry

- Engine type
- Cylinder count
- Fuel type
- Polytropic exponent
- Compression
- Stroke
- Bore
- Rod
- CO
- PO

Volume per cylinder

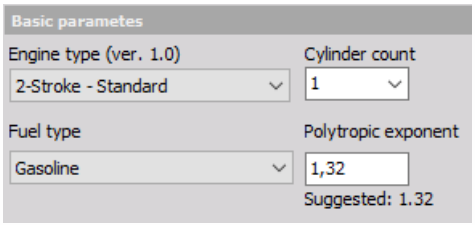
- Min
- Max

Engine templates

Cylinders

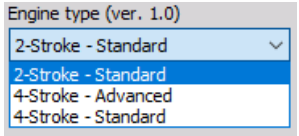
- Pressure channel
- Ignition misalignment
- S01/E01 channel
- Number of injections
- S01 trigger level
- E01 trigger level
- Additional channels

Basic parameters

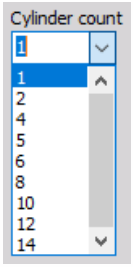


Drop-down list options:

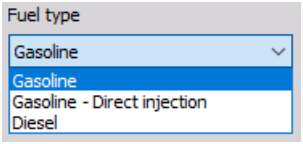
- **Engine type** - Engine type can be 4-stroke (for example car) or 2-stroke (motorbike).



- **Cylinder count**



- **Fuel type** - Fuel type defines calculation procedures, so it is important to select the correct setting.



Compression is the ratio between total and compressed volume:

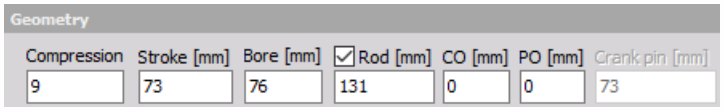
$$R_c = \frac{V_d + V_c}{V_c}$$

V_d = swept volume

V_c = clearance volume

Polytropic exponent - The polytropic exponent is a fixed value for entire stroke. It is important for calculation of thermodynamic zero and for heat release. If you don't know the polytropic exponent for your engine, take the suggested value for each engine type.

Geometry

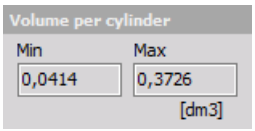


On the drawing on the right, you can see what values should be entered in corresponding fields.

- Compression [bar] - Cylinder compression
- Stroke [mm] - Stroke of the piston
- Bore [mm] - Bore of the cylinder
- Rod [mm] - Rod length from the center of crankshaft to the center of piston pin
- Piston offset - PO [mm] - This is offset of the piston pin (usually zero).
- Crankshaft offset - CO [mm] - This is the offset of the crankshaft pin (usually zero).
- Crank pin [mm] - Diameter of the crankpin

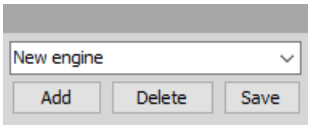
Calculated volume

As soon as you enter the previous values and define the cylinder pressure channel, minimum and maximum volume of the engine will be calculated and shown here.



Engine templates

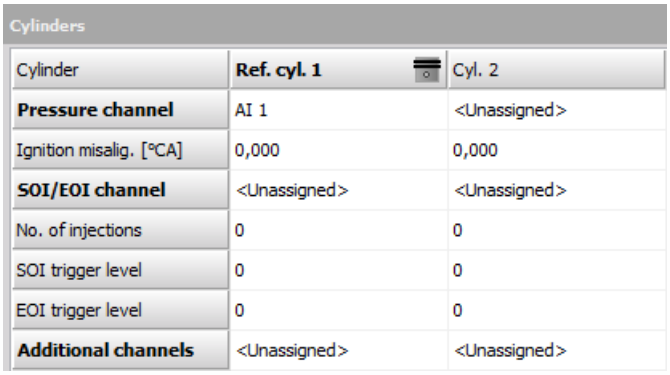
All that information about engine type and geometry is stored in Dewesoft setup file. But you can save it to XML file (in Dewesoft directory CAEngines.xml) for easier handling of the geometry data.



Buttons:

- **Add** - adds new template - after Adding new template the name should be entered
- **Delete** - deletes currently selected template
- **Save** - saves changes to currently selected template

Cylinders

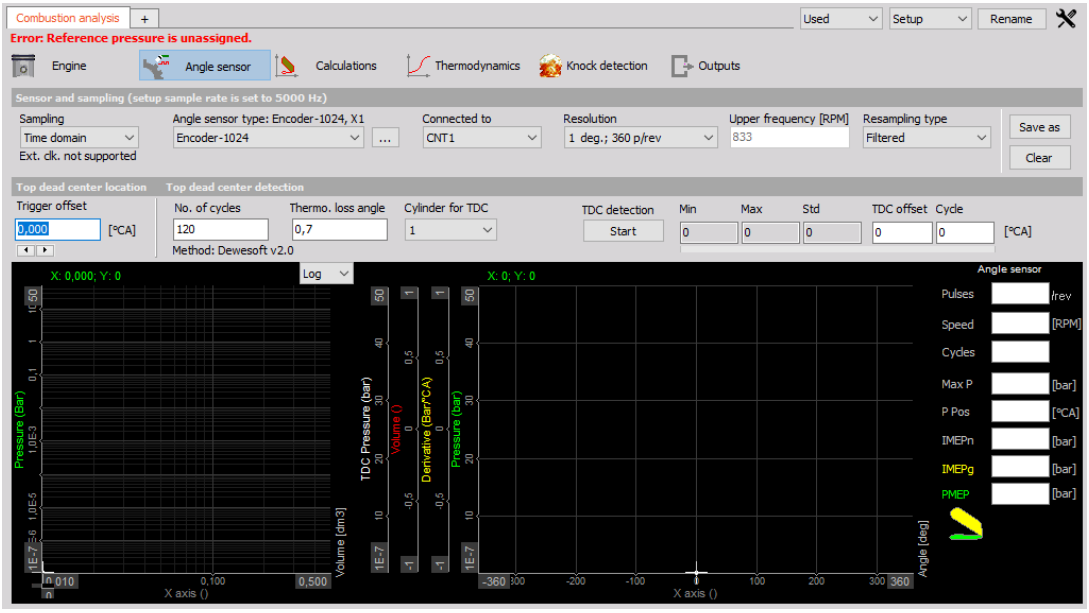


- **Cylinder** - Reference cylinder is the one that is used for all calculations as base (it is the one which has zero misalignment). Select it by clicking in the cylinder row. In our example that is cylinder 1 (grayed).
- **Pressure channel** - In this row you select pressure channel for a given cylinder. At least reference cylinder needs pressure channel for correct calculations. Channels must be set correctly in the Analog setup (scaling, name, color).
- **Ignition misalig. [°CA]** - This is how cylinders are fired, with what delay (in degrees). You must know this for your engine.
- **SOI / EOI channel** - Channel for calculation of start and end of injection.
- **No. of injection** - Number of injections - we will have the same number of output channels for angle of start and end of injection.
- **SOI trigger level** - Trigger level for start of injection (always takes positive edge).
- **EOI trigger level** - Trigger level for end of injection (always takes negative edge).
- **Additional channels** - Additional channels are not used in any calculations but can be shown in displays.

Channels must be set correctly in the Analog setup (scaling, name, color).

2. Angle sensor setup

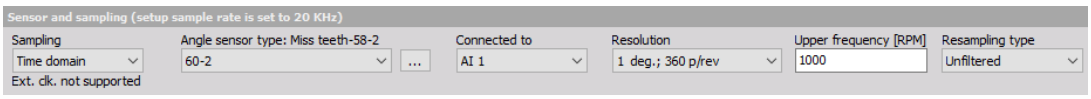
After selecting Angle sensor type CA following setup screen is displayed:



Settings and entered values for this Angle sensor CA type are divided on sections:

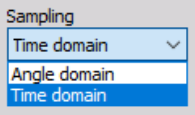
- Sensor and sampling
- Angle sensor type
- Connected to
- Resolution
- Upper frequency [RPM]
- Resampling type
- Top dead center location
- Top dead center detection
- Trigger offset
- Number of cycles
- Thermo loss angle
- Cylinder for TDC
- Display graphs
- Values table

Sensor and sampling



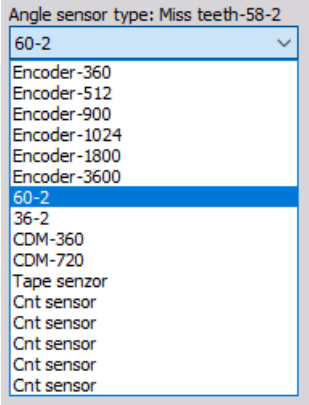
Drop-down list options:

- **Sampling type** - Sampling type is the most important selection for the behavior of combustion analysis. Internal clock will acquire the data in the time domain and recalculate to angle domain. It will be slower, but we are able to do all time-based calculations.

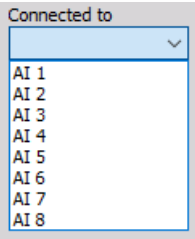


- **Angle sensor type** - Angle sensor depends on what sensor we connected.

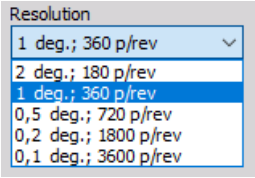
When three dot button is selected, the Counter sensor editor window appears to define counter sensor.



- **Connected to** - Here we define the physical connection of the sensor. If use an external clock with CA-CPU, it will show the CA-CPU inputs. If we use the internal clock, it will offer the counter inputs for encoder and CDM sensors and analog inputs for Geartooth with double or missing teeth.



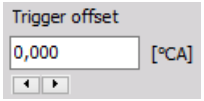
- **Resolution** - The resolution will define the number of points per one engine revolution. A higher number will give higher accuracy, but will also bring more calculation load.



For information about Angle sensor setup see -> General mathematics modules -> Angle sensor.

Top dead center location

Trigger offset



Trigger offset is the offset from the trigger to top dead center of the reference cylinder.

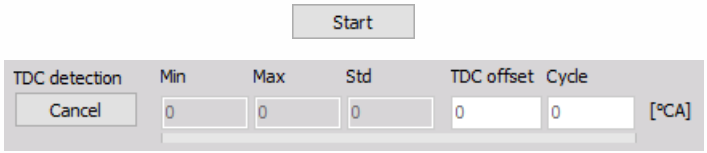
The value in this field can be decreased and increase with buttons.

Top dead center detection

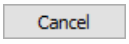
- **No. of cycles** - Number of cycles defines the number of averages to take in TDC detection procedure.
- **Thermo. loss angle** - TDC detection procedure will search for pressure peak, but this peak is delayed for some angle compared to the top dead center. We enter this offset in this field.

TDC detection

When Start button is selected, table with values appear instead on right side table:



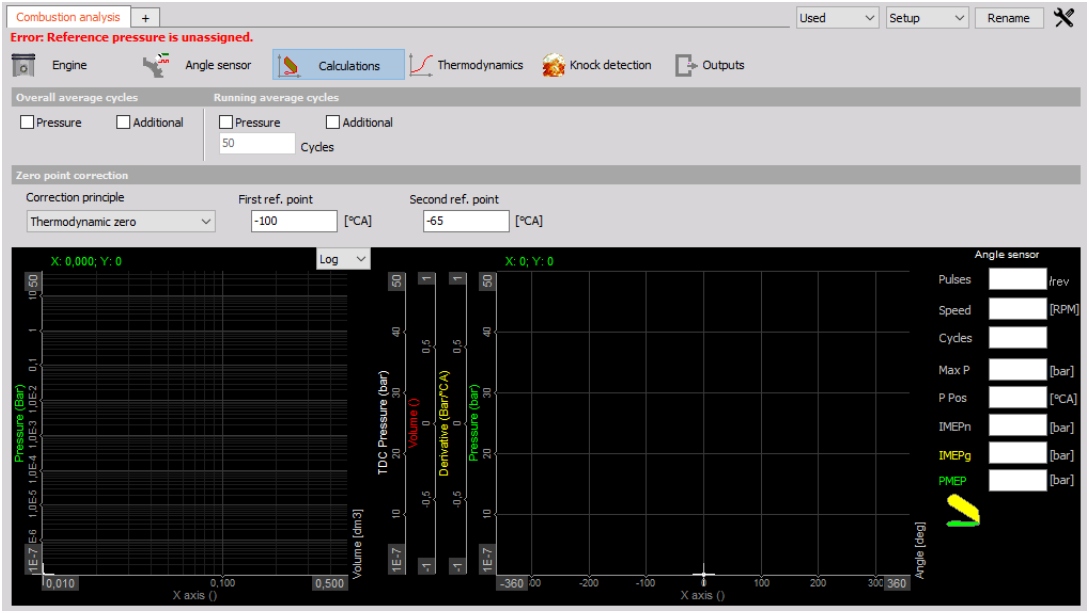
Now the Start button changes to Cancel button and with selecting this we can stop TDC detection (button change again to Start).



When TDC detection is finished, the Average offset will be taken as the trigger offset.

3. Calculations setup

After selecting Calculations type CA following setup screen is displayed:



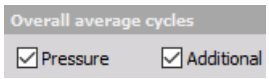
Settings and entered values for this Calculations CA type are divided into sections:

- Overall average cycles
- Running average cycles
- Pressure
- Additional
- Pressure
- Additional
- Number of cycles
- Zero point correction
- Correction principle
- First reference point
- Second reference point
- Display graphs
- Values table

Overall average cycles

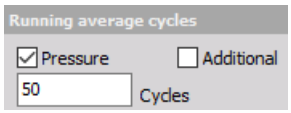
Statistics are selected by default since these values are needed also for further calculation. The maximum pressure and position are included.

Derivation calculates the pressure derivation and adds value and position of pressure rise. When Derivation is selected, on graph appears also this value.

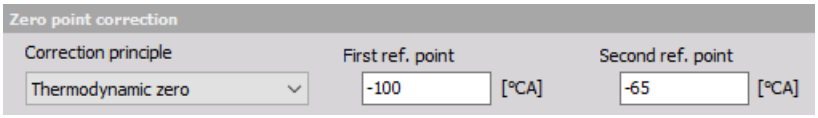


Running average cycles

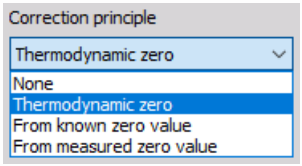
The Running average cycles calculate the mean value of the last n cycles. This basic statistic is available for the pressure and for the Additional channels of each cylinder. The result is a vector with the angle as a reference.



Zero point detection

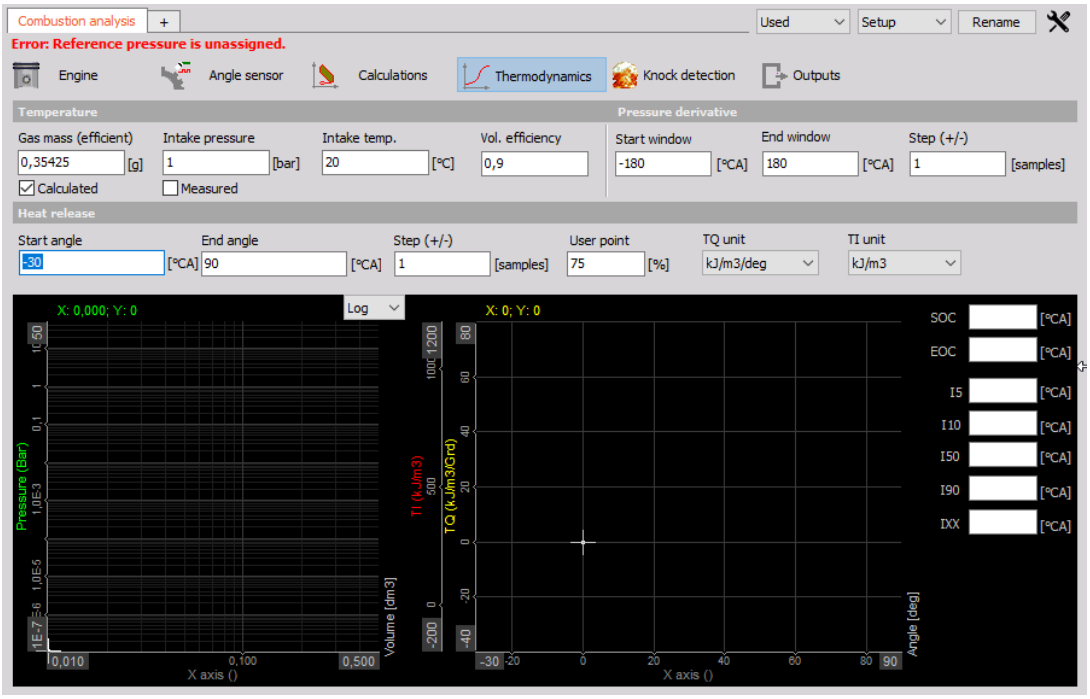


Correction principle - There are three basic correction principles for the zero point correction. Thermodynamic zero assumes polytropic compression and finds the absolute pressure offset according to it. The polytropic exponent, the start and end angle for this calculation needs to fit the real compression (without ignition).



4. Thermodynamic setup

After selecting Thermodynamics type CA following setup screen is displayed:



Settings and entered values for this Thermodynamics CA type are divided into sections:

- Temperature
- Pressure derivative
- Gas mass (efficient)
- Intake pressure

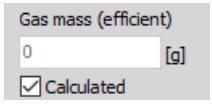
- Intake temp
- Volumetric efficiency
- Start window
- End window
- Stef (+/-)
- Heat release
- Start angle
- End angle
- Step (+/-)
- User point
- TQ unit
- TI unit
- Display graphs
- Values table

Temperature

The temperature inside the combustion is calculated based on the ideal gas law and is only valid at around TDC.

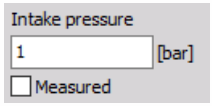
Gas mass (efficient)

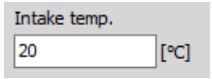
For the Temperature calculation, the gas mass is required. This can be either manually entered, or calculated.

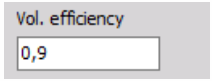


Intake pressure, Intake temp and Volumetric efficiency

If from Calculated is used, the intake temperature, intake pressure, and also the volumetric efficiency (0.9= 90% filled) must be entered.

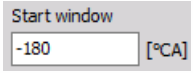


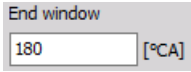


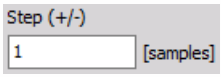


Pressure derivative

For Pressure derivative, the start-angle, the stop-angle and also the step size must be defined.



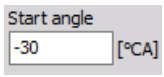




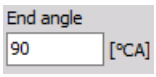
Heat release

For setting up the Heat release calculation the start and stop angle must be defined. The typical range is from -30° to +60°. An earlier injection start angel must be set according to the real injection point.

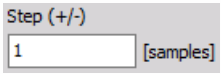
- Start angle - The angle of the crank on the start of heat release.



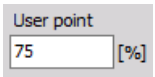
- End angle - The angle of the crank on end of heat release.



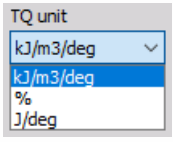
- Step(+/-) - The step input field defines the calculation width: e.g. Step 1 means the calculation is based on ±1 sample (or angle resolution value). A higher value smooths the result.



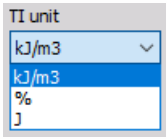
- User point - Heat release creates several output channels with angle values for certain amplitude values - 5, 10, 50 and 90% (called I5, I10, I50 and I90). Additionally, we can define one user point called IXX where the XX is the percentage value of the heat release defined here.



- TQ unit - We can either have the physical unit for the heat release or have it expressed in percentage.

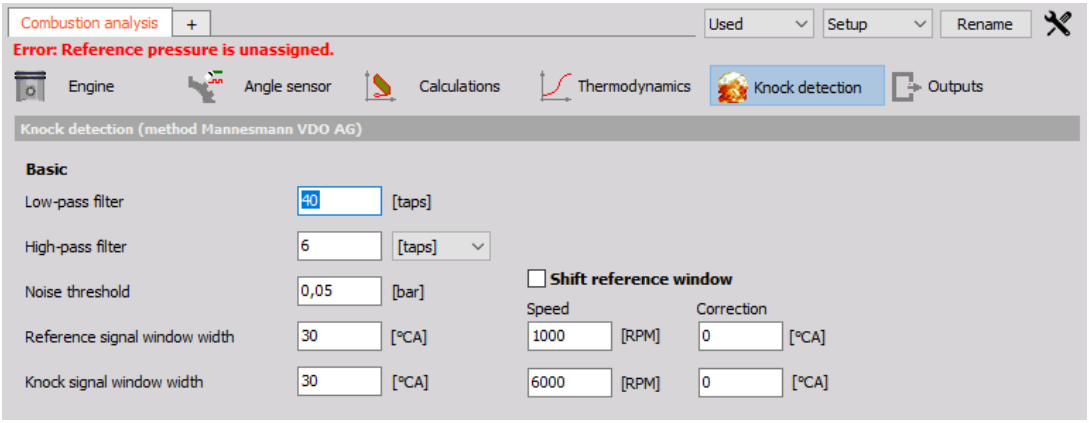


- TI unit - We can either have the physical unit for the integrated heat release or have it expressed in percentage.



5. Knock detection setup

After selecting Knock detection type CA following setup screen is displayed:



Knock detection is the principle for finding the amount of knocking due to late ignition.

Settings and editable values for this Knock detection CA type are:

- Knock detection (method Mannesmann VDO AG)
- Low-pass filter
- Shift reference window
- High-pass filter
- Noise threshold
- Knock signal window width

Low-Pass and High-pass filter

In these fields, we define Number of taps (according to angle resolution) of the average filter to cut low and high pass part of the signal.

Noise threshold

Is the value which defines the lower limit for calculation of values.

Knock signal window width

The width of the window on the left and right side of the pressure signal.

Shift reference window

Is two-point linear scaling offset definition how much the calculation window is shifted for different speeds.

HUMAN VIBRATION

Dewesoft Human vibration application module allows measurement of effect of vibrations to human body. Especially on work places exposed to vibrations there is a big chance of permanent damage to some parts of human body. The human vibration module provides measurements to be able to judge the risk of such damage.

Measurement is based on an ISO 2631-1 (dated in 1997) standard which defines basic procedures, ISO 8041 (dated 2005), which defines exact procedures for measurements and ISO 2631-5 (dated 2005) which defines calculations of lumbar spine response to the vibrations.

- **Whole body measurements** - are measured with the help of the so called seat sensor, where we need to install the triaxial sensor in the rubber adapter on which we sit on.
- **Hand arm measurements** - is measurement of hand arm where the sensors are installed on special adapters for holding them on the handle or between fingers.

Both measurements are performed with triaxial accelerometers (it is very common to use 50 g sensors) and using special adapters. For working places with high vibrations (for example impact hammers) it is necessary to use high g sensors (500 g or more).

- **Required hardware** - Sirius, Dewe 43
- **Required software** - SE or higher + HBV option, DSA or EE
- **Setup sample rate** - At least 5 kHz

For additional help with Human vibration Application visit Dewesoft PRO training.

Add / New Human vibration module

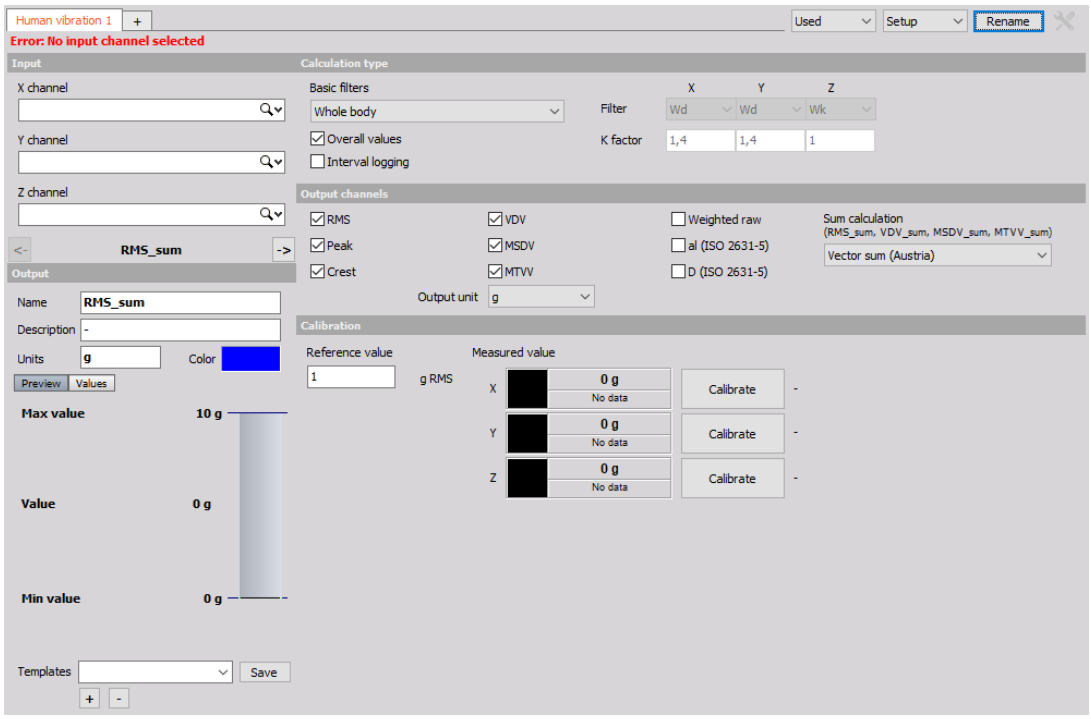
For instructions on how to Add new module see -> Setup -> Add module.

Basic procedures of Human vibration application setup are:

- Human vibration module setup for applied hardware
- Calibration

Human vibration module setup

After we add a new Human vibration module a screen like this will appear:



This screen has the following main sections:

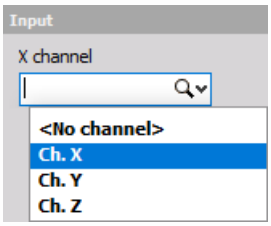
- Input channel selection
- Calculation type
- Ouput channels
- Calculated parameters definition
- Triaxial accelerometer calibration

With using View Channel List button we can display a different view on Human vibration module setup - Channel list with all defined output channels appear.

Input channel

First of all we need to assign the input channels to triaxial accelerometer sensor by selecting from drop down lists for x channel, y channel and z channel separately.

It is important that the z axis is a vertical direction since it is weighted differently than x and y.

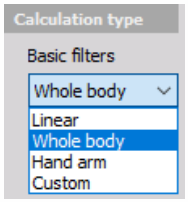


Calculation type

On Calculation type section we have two parameters to define the measurement:

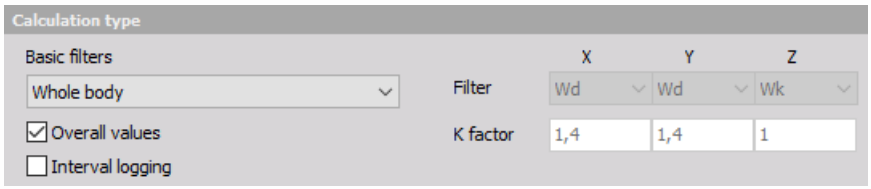
1. Measurement modes

- **Basic filters** - Different modes define different Basic filters used to simulate the human response to vibrations. Those filters are defined from numerous measurements of natural frequencies of certain parts of the human body. Select desired mode from the drop-down list.

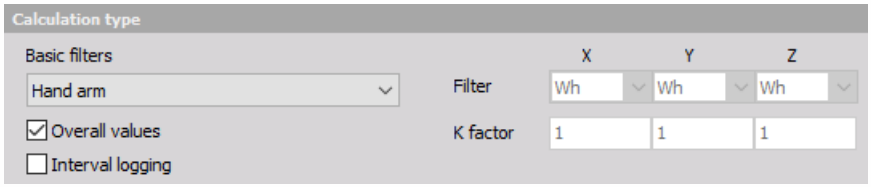


There are two basic modes of operation:

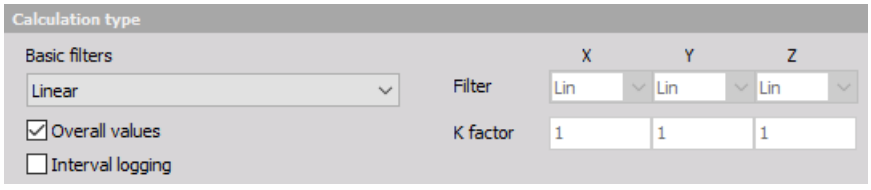
- **Whole body mode** (Individual Filter and K factor settings can't be chosen, these are predefined).



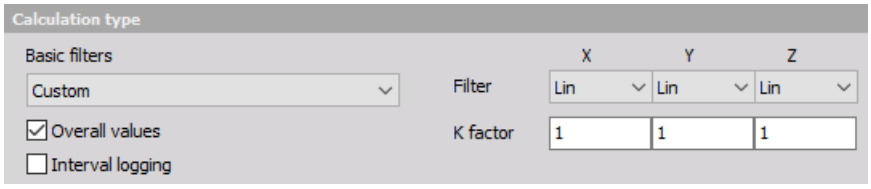
- **Hand arm mode** (Individual Filter and K factor settings can't be chosen, these are predefined).



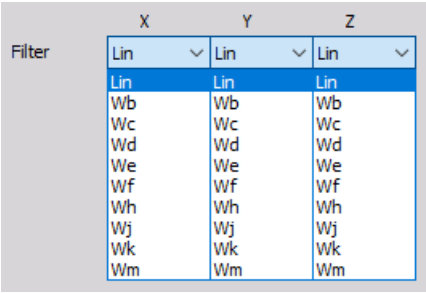
- **Linear filter to check the measurement chain** (Individual Filter and K factor settings can't be chosen, these are predefined).



Custom filters



Filter X, Y and Z value: this individual value can be selected from the drop-down lists on the right side to do special measurements. Filters are defined as follows:



- **Lin** - unweighted linear
- **Wf** - motion sickness, z axis
- **Wb** - vertical whole body, z axis (older ISO 2631-4)
- **Wh** - hand arm, all directions
- **Wc** - horizontal whole body, x axis
- **Wj** - vertical head vibration, x axis
- **Wd** - horizontal whole body, x or y axis
- **Wk** - vertical whole body, z axis
- **We** - rotational whole body, all directions
- **Wm** - building vibration; all directions.

With custom filter, we need to define a weighting K factor. This is a multiplication factor for each axis when calculation vibration sum.

NOTE: **We need to take care about the high pass frequency limit of the sensor and used amplifier. For hand arm mode, the high pass frequency is 6.4 Hz, which is easy for any sensor. For the hand arm, the frequency limit is 0,4 Hz, where we need to choose sensor carefully. We can also use higher filters (like 3 Hz), if we know there is no frequency content below this limit. This will help to perform measurement faster and with less error (lower frequency filters means longer settling times). A special care must be taken for Wf filter for motion sickness (for example on ships) where the frequency limit is only 0,08 Hz and we need a very special sensor to measure this low frequency vibration.**

NOTE: **The recommended sampling rate of the measurement also depends on the application. For hand arm the minimum sampling rate is 5 kHz, while for all the others 1 kHz is enough.**

2. Calculated parameters

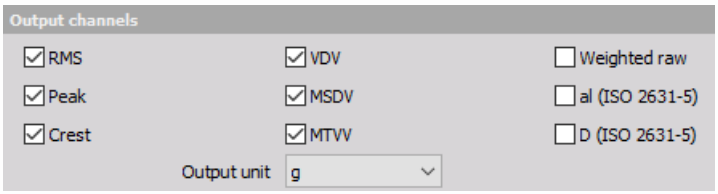
On the lower part of Calculation type section choose type of parameter for calculation (at least one must be selected for calculated channels):

- **Overall values** - we have only one value at the end of the measurement
- **Interval logging** - the time interval for logging is defined.
- **Both**

For example if we select to have interval logging with 5 seconds interval, we will get a new value after each 5 second. After that the value is reset and the calculation is started again.

Output channels definitions

We can define the output channels, additionally to basic filters and k factor channels.



- **RMS** - calculate the root means square value of the weighted signal
- **Peak** - the maximum deviation of the signal from the from the zero line
- **Crest** - the ratio between the peak value and RMS value. Crest factor gives an impression about the spikes in the signal; pure sine waves have a crest factor of 1.41.
- **VDV** - fourth power vibration dose value
- **MSDV** - motion sickness dose value
- **MTVV** - the maximum transient vibration value, calculated in one second interval
- **Weighted raw** - full speed time signal weighted with chosen filter. We can use those channels for calculation of FFT or CPB spectrum.
- **al (ISO 2631-5)** - the lumbar spine response from excitation measured in all three directions
- **D (ISO 2631-5)** - acceleration dose, measured from the lumbar spine response

NOTE: **al and D are the values based on ISO 2631-5 which describes the calculations and the limits for lumbar spine response to vibrations. The base for this standard is that the professional drivers of buses or trucks are exposed to vibrations when driving on rough roads or over the bumps. Multiple shocks cause transient pressure changes at the lumbar vertebral end plates that cause damage after years of driving. al and D are enough to evaluate human vibration exposure according to ISO 2631-5.**

Output channels settings

The lower-left area displays the output channel settings with fields like in the analog channel setup: Name, Units, Color, Min val and Max val, also symbolic display of signal values.

For help with Output Channel list see -> Setup -> Output Channel list.

Human vibration module can have more output channels according to:

Calculation type section

Basic filters: For each selected axis individually a Filter and a K factor channel is created. Each output channel is created with a default name and can be changed in output name row.

- Filter output channels

- K factor output channels

k_X	k_Y	k_Z

NOTE: **These output channels are created in any case, though in Calculated parameters and Output channels sections none whatever choice is selected.**

Calculated parameters section

These parameters have only influence upon creating output channels for calculated value selected in Output channels sections -> see below.

Output channels section

For each calculated and in this section selected value additional to basic filters channels one output channel for each axis individually is created.

Peak output channels

- For selected Overall values in Calculated parameters section:

PEAK_X	PEAK_Y	PEAK_Z

- For selected Interval logging in Calculated parameters section:

PEAK_X_t	PEAK_Y_t	PEAK_Z_t

Crest output channels

- For selected Overall values in Calculated parameters section:

Crest_X	Crest_Y	Crest_Z

- For selected Interval logging in Calculated parameters section:

Crest_X_t	Crest_Y_t	Crest_Z_t

The RMS, MSDV, VDV and MTVV are calculated for sum of all three axes, therefore a value for output channel is created.

RMS output channels

- For selected Overall values in Calculated parameters section:

RMS_sum	RMS_X	RMS_Y	RMS_Z

- For selected Interval logging in Calculated parameters section:

RMS_sum_t	RMS_X_t	RMS_Y_t	RMS_Z_t

VDV output channels

- For selected Overall values in Calculated parameters section:

VDV_sum	VDV_X	VDV_Y	VDV_Z

- For selected Interval logging in Calculated parameters section:

VDV_sum_t	VDV_X_t	VDV_Y_t	VDV_Z_t

MSDV output channels

- For selected Overall values in Calculated parameters section:

MSDV_sum	MSDV_X	MSDV_Y	MSDV_Z

- For selected Interval logging in Calculated parameters section:

MSDV_sum_t	MSDV_X_t	MSDV_Y_t	MSDV_Z_t

MTVV output channels

- For selected Overall values in Calculated parameters section:

MTVV_sum	MTVV_X	MTVV_Y	MTVV_Z

- For selected Interval logging in Calculated parameters section:

MTVV_sum_t	MTVV_X_t	MTVV_Y_t	MTVV_Z_t

NOTE: **When creating output channels for above-listed Output channels option Overall values and/or Interval logging in Calculated parameters section must be selected.**

Weighted raw ouput channels

- Independently of selection in Calculated parameters section:

X_w	Y_w	Z_w

al (ISO 2631-5) output channels

- Independently of selection in Calculated parameters section:

alX	alY	alZ

D (ISO 2631-5) output channels

- None of choices is selected in Calculated parameters section:

alX	alY	alZ

- For selected Overall values in Calculated parameters section:

DX	DY	DZ

- For selected Interval logging in Calculated parameters section:

DX_t	DY_t	DZ_t

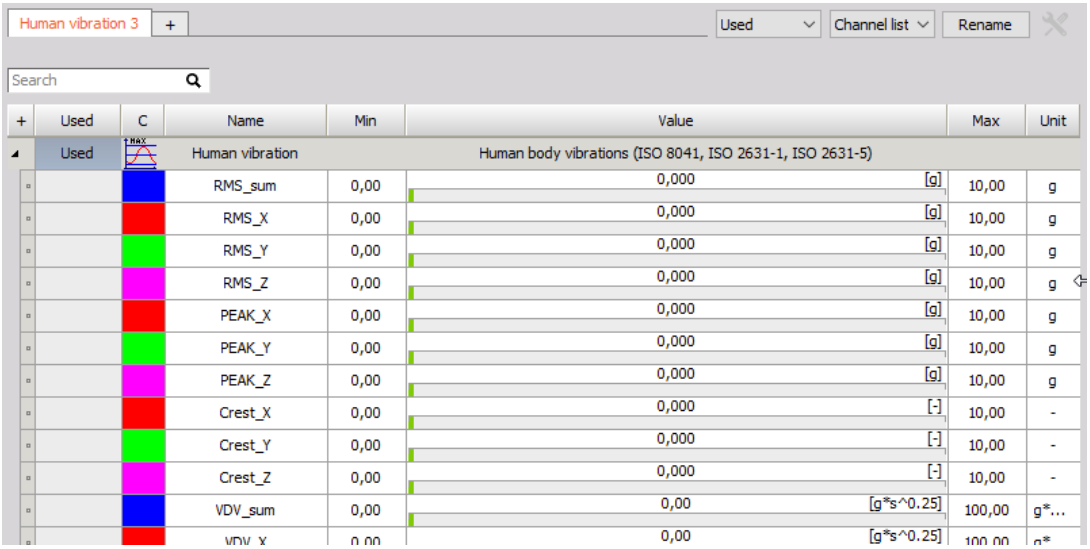
Use the Prevoius and Next buttons to navigate between the output channels in OUTPUT channel settings part of the screen.

<-

->

Channel list view

If we open Channel list the following output channels (if filters selected) will appear:



+	Used	C	Name	Min	Value	Max	Unit
	Used		Human vibration		Human body vibrations (ISO 8041, ISO 2631-1, ISO 2631-5)		
			RMS_sum	0,00	0,000 [g]	10,00	g
			RMS_X	0,00	0,000 [g]	10,00	g
			RMS_Y	0,00	0,000 [g]	10,00	g
			RMS_Z	0,00	0,000 [g]	10,00	g
			PEAK_X	0,00	0,000 [g]	10,00	g
			PEAK_Y	0,00	0,000 [g]	10,00	g
			PEAK_Z	0,00	0,000 [g]	10,00	g
			Crest_X	0,00	0,000 [-]	10,00	-
			Crest_Y	0,00	0,000 [-]	10,00	-
			Crest_Z	0,00	0,000 [-]	10,00	-
			VDV_sum	0,00	0,00 [g*s^0.25]	100,00	g*...
			VDV_X	0,00	0,00 [g*s^0.25]	100,00	g*

For help with Output Channel list see -> Setup -> Output Channel list.

Calibration

The triaxial accelerometer can be calibrated in two ways.

1. Scaling with calibration certificate

If we don't use the calibrator, but have the sensitivity of triaxial accelerometer, we can define it directly in the channel setup using the values from the triaxial accelerometer calibration sheet. This can be done from Dewesoft Setup -> Analog in tab Channel setup for the Human vibration Application module used channels.

For information about Calibration analog input channel see -> Channel setup - Calibration.

NOTE: **First, as usual, we enter the 'Units of measurement'. We have a sensitivity of the sensor expressed either in mV / m/s2 or mV/g (or both) for IEPE sensors and in pC/g for piezoelectric (charge) sensors. Therefore we need to scale it to physical quantity. The Reference sensitivity is the key value to enter in the Dewesoft setup. Then it is best to go to scaling 'by function' tab, check the Sensitivity and then enter value from calibration data sheet. We need to change also the amplifier range to best fit the current acceleration values and the filter to fit the sampling rate. If Sirius is used, the filter can be set to the highest values (because amplifiers A/D converter provides sharp anti aliasing filters) unless we intentionally want to have lower frequency range.**

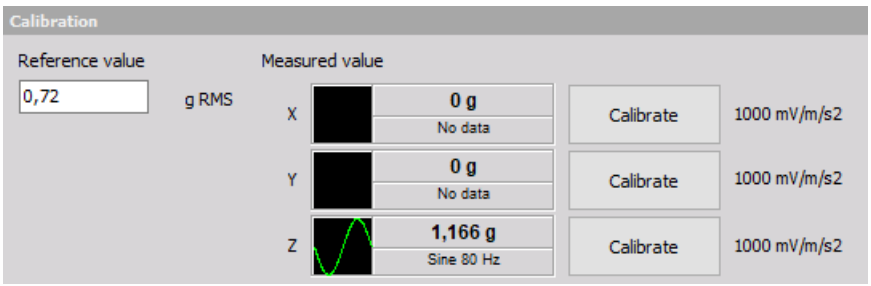
2. Calibrating the triaxial accelerometer with calibrator

If we want to perform calibration with the calibrator, we can perform it here - in Human vibration module itself. First enter the Reference value of vibration. This can be read from the calibrator.

NOTE: **Usually the calibration levels are 10 m/s2 peak, which is 7,07 m/s2 RMS or 0,72 g. We need to enter this value in Reference value field.**

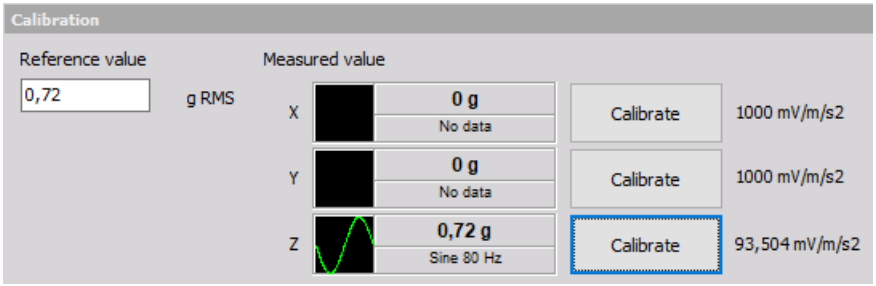
As soon as we mount the sensor on the calibrator, we should see the amplitude and the frequency of the signals. In this case we see that we applied the sensor in Z direction.

The frequency of calibration is below 200 Hz (80 or 160 is typical, in our case it is 80 Hz). This is a simple check that everything is OK.



Next simple step is to press the Calibrate button near the axis which is currently calibrated. As soon as we do this, we will see sensor sensitivity in mV/g which can be checked against the

sensor calibration certificate. Little percent difference is acceptable.



Then we can see the calibrated live RMS value of the vibration (in example above 0,72 g).

NOTE: **We need to use setup sample rate in Dewesoft at least 5 kHz or higher to make successful accelerometer calibration. This can be changed in Dewesoft Analog in.**

MODAL TEST

Modal test is a process to determine the transfer characteristic of a system. It is most often used to find the natural frequencies of a mechanical structure using hammer or shaker excitations.

- **Required hardware:** Sirius, Dewe43
- **Required software:** Dewesoft DSA or SE, PROF and FRF option

Modal test (introduction)

Modal test is a mathematical module that allows us to calculate the transfer function (H) between two signals.

$$H = \frac{\text{Output}}{\text{Input}}$$

Input and Output signals are usually mechanical or electrical nature. Mechanical means exciting structure with hammer or shaker and measure response with an accelerometer. Electrical can be for example input and output voltage of the amplifier in order to get transfer function of the amplifier.

Transfer function (H) is a relation between two signals in a complex form (real and imaginary vs frequency).

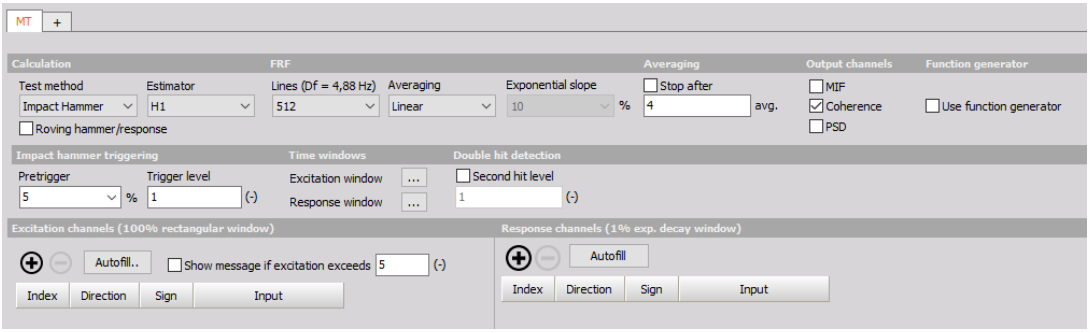
Data collection:

Calculations can be done in three ways:

- Impact hammer (Triggered) - roving hammer/accelerometer
- Shaker (Free run)
- ODS (Operational deflection shapes)

Setup

For instructions on how to Add new module see Add module.



Calculation

Transfer function is calculated via the standard Frequency response function algorithm or FRF for short. It is possible to calculate all excitation/response pair at once or in groups.

Test method

- **Impact Hammer (Triggered acquisition)** - In triggered acquisition, blocks of data are taken when trigger level is exceeded.
- **Shaker (Free run)** - In Free run blocks of data are taken continuously (one following another with overlap if specified).
- **ODS (Operational Deflection Shapes)** - A single accelerometer is used as means of excitation measurement, others are used for the response. This method is used if excitation by hammer or shaker is not an option.

Use one excitation channel and several response channels or several excitation channels and one response channel in a single measurement.

Do multiple measurements with one or more excitation points to measure one or more responses.

FRF estimator

- **H1** - The most commonly used estimator is the H1-estimator, which assumes that there is no noise on the input and consequently that all the input measurements are accurate. All noise is assumed to be on the output.
- **H2** - Alternatively, the H2 estimator can be used. This assumes that there is no noise on the output and consequently that all the output measurements are accurate. Noise is assumed to be only on input.

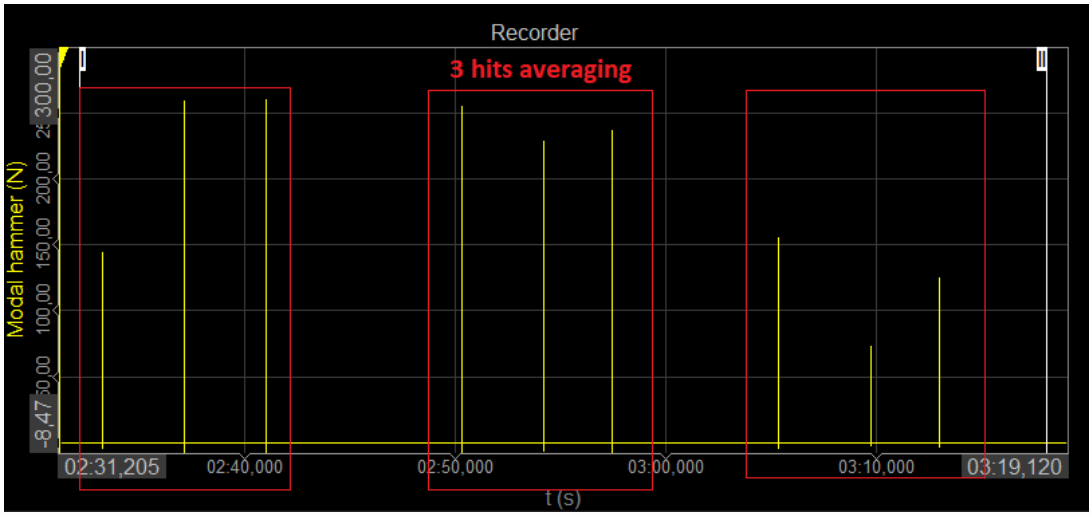
FRF (frequency response function)

- Lines - The number of lines for FFT (block size equals two times lines).

- Averaging type - Select the averaging type from Linear, Peak hold, Exponential
- Exponential slope - Define slope falloff in per cent if Exponential is used
- Stop after - Stop after the set number of averages

Stop after

Select the check-box to enable Averaging and input the maximum number of measurements. This will enable you to take the predefined number of measurements and average them for a combined result. If the number of averages is not reached, the result will be calculated from the completed measurements. After the number is reached, you will not be able to do additional measurements for that group.



Output channels

- **MIF** - Mode Indicator Function is a function ranging from zero to one and shows where resonances are.
- **Coherence** - is a real value between zero and one. If the value is one then response power is caused totally by input power. A value less then one indicates that measured response power is greater than that from input power (noise…).
- **PSD** - Power Spectral Density is the measure of signal's power content over frequency.

Function generator

In any case, we can excite the system with help of Function generator. By checking the use function generator an additional row of configuration options is opened for shaker control or directly for input of electric circuits.

Impact Hammer

- **Burst** - Define Hold time (s)
- **Chirp** - Define Start and Stop frequency

Shaker

- **Sine** - Define Start and Stop frequency
- **Noise** - Outputs random noise

Further adjustments can be done inside of Function generator module.

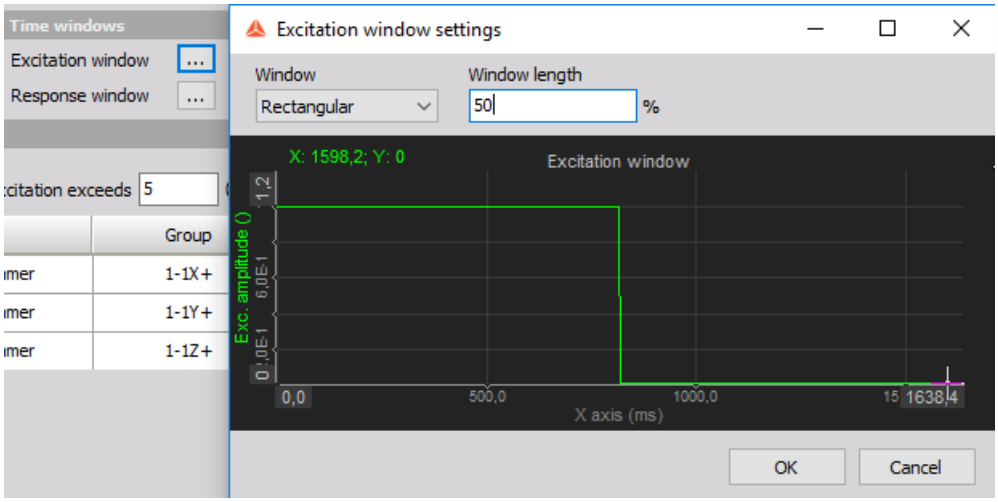
Impact hammer (Triggered)

Impact hammer triggering

- **Pretrigger:** how much data will be taken before a trigger is reached (in per cent of block size). If this is set to zero we will probably miss some data on the beginning.
- **Trigger level:** trigger level in units of input channel.

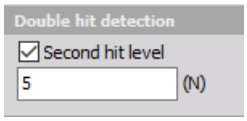
Time windows

Excitation & Response window: Define the type (Rectangular or Exponent) of the window and its length/decay in per cent. Rectangular is predefined for Excitation and Exponent for Response as this is the typical option. If your system requires you can change it.



Double hit detection

Enable or disable second hit level detection with the check box and enter the desired value in the field below. The signal will be searched for peaks and if there is more than one peak which is higher than double hit detection level you will get a warning that double hit happened. You can check it on excitation spectrum. Then you can reject this measurement or continue.

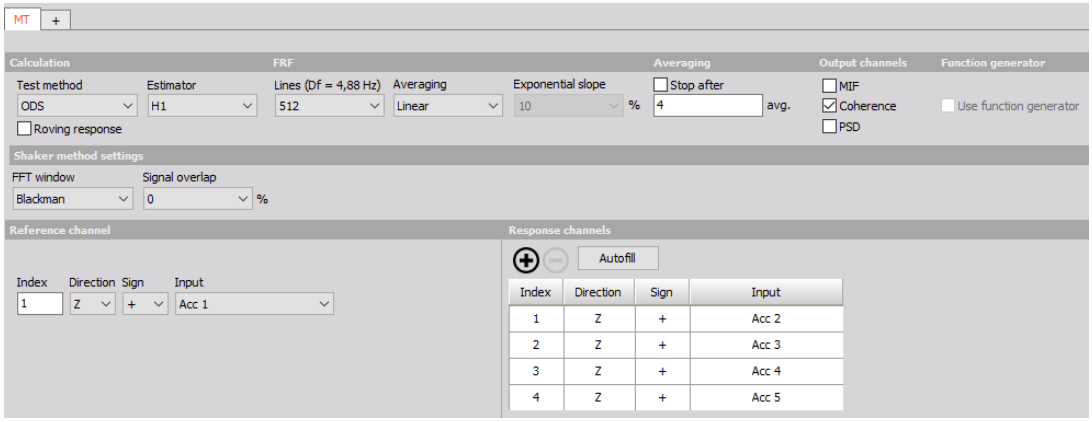


Shaker (Free run)

- **FFT window** - Standard Dewesoft window functions for FFT leakage reduction.
- **Signal overlap** - Block of data can also be overlapped.

Operational deflection shapes (ODS)

In ODS mode settings are similar to the **Shaker** method, except that we can't use the Function generator and we are limited to one Reference "excitation" channel.



Excitation, response and reference channel settings

Show message if excitation exceeds - This is just a safety that we don't damage our structure. When the response acceleration from our system is above the limit analogue out and measurement are stopped.

Excitation and response configurations

- **Index** - index of measured point (according to geometry).
- **Direction** - in which direction (x, y, z) we measure certain point (according to geometry).
- **Sign** - in which direction of axis we measure (+, -).
- **Input** - Input channel that is physically connected force or an acceleration sensor.
- **Group** - Define the measurement group when using Roving hammer/response
- **AO channel** - Analog Output channel which is used for driving shaker.
- **Autofill** - Autofill setup is used to add a large number of channels

Autofill setup

NOTE: **If you wish to use Autofill for triaxial sensors you must connect the response channels (Triaxial accelerometers) in a manner of: 1st sensor X, Y, Z, 2nd sensor X, Y, Z, and so on.**

Input channels settings

AI start node input channel - Select the channel that will measure the first node of the structure Triaxial sensors (only for response ch.) - If multi-axial sensors are used, check the box to define the used channels under Geometry settings / Directions

Geometry settings

- Start node index - Define the starting point node of the structure
- Node index increment - Set the increment by which the nodes will advance
- Direction(s) - Select the direction(s) of the measurement (if Triaxial response you can select multiple)
- Sign - Define if the excitation will be positive or negative regarding the Direction.
- Sensors in group - Define the number of sensors in one rowing group
- Roving count - Define the number of positions, how many moves of the sensor group

Outputs

For each measured point

- Amplitude - signal amplitude.
- Phase - signal phase.
- Real - the real part of a signal.
- Imaginary - the imaginary part of a signal.
- Coherence - is a real value between zero and one. If the value is one then response power is caused totally by input power. A value less then one indicates that measured response power is greater than that from input power (noise...).

For excitations

Excitation spectrum - FFT spectrum of an excitation signal.

Common channels

- Info - information about which point is currently measured.
- Ave Count - a number of collected averages.
- Reject last - this is a channel which can be used with visual control to reject last sample (overload of channels, double hit...).
- Next point - this is a channel which can be used with visual control to go to the next point when grouping is used.

Measure

Visualize transfer function calculation

How to visualize transfer function calculation with FRF Geometry visual control see Geometry display.

Reject Last: in the triggered mode it is also possible to reject the last block of data if we made a false measurement. This can be done via a control button in the measurement. If we reject a block then the average count decreases by 1.

FRF geometry

Model

- **Load UNV**
- **Load UNV file of your desired structure.**
- **Edit**

Opens UNV editor, where you can create the structure in Cartesian or Cylindrical coordinate system.

Node selection

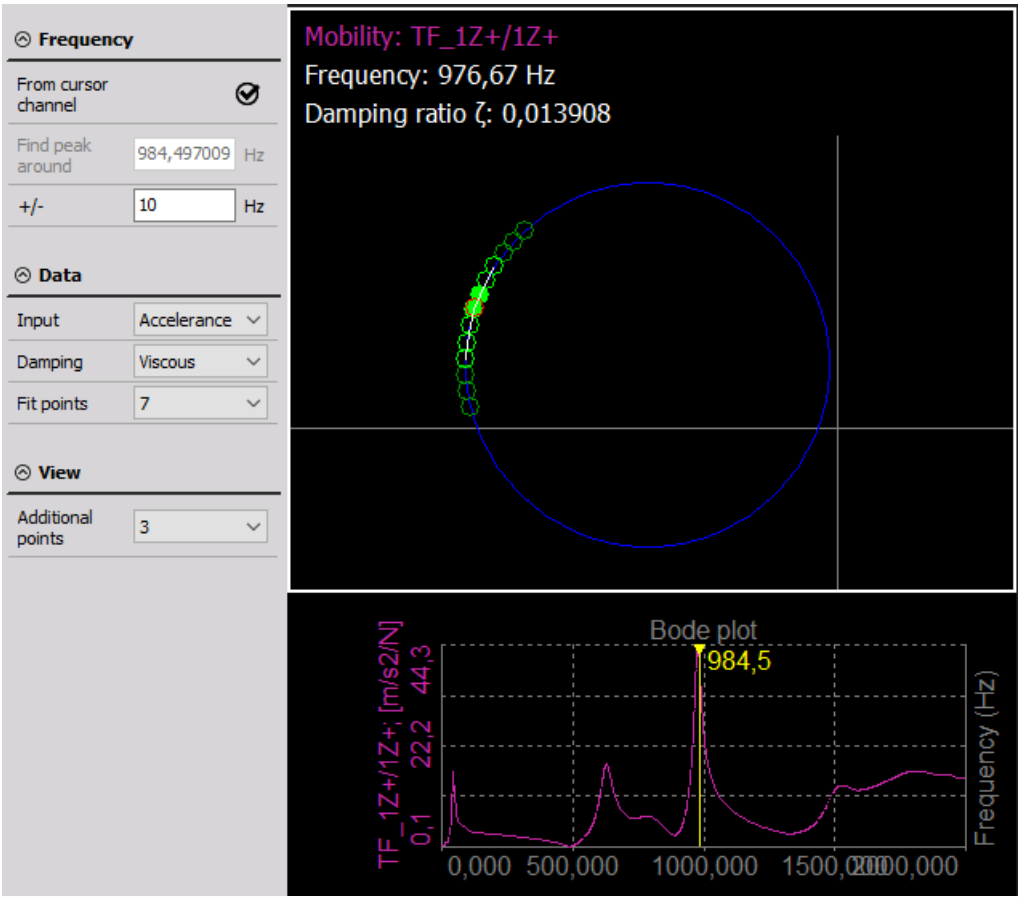
Excitation - Select the excitation point you want to observe **Response** - Select the response point you want to observe **Show responses in direction** - Select the directions to be animated. Possible if measured in multiple axes.

View

- **Coordinate system** - Display or hide the X, Y, Z axes
- **Nodes** - Display or hide nodes / measured points
- **Node numbers** - Display or hide node numbers
- **Lines** - Show connecting lines between nodes
- **Surfaces** - Display or hide surfaces
- **Rotate/ Scale** - Switch between rotate and scale with left mouse button
- **Select position** - Select between different viewing positions
- **Animation** - Define the scale and speed of the animation or switch to manual and adjust the phase by hand.
- **Frequency** - Animate the structure at frequency of cursor channel or Manually by inputting the frequency in the bottom field.
- **Advanced** - Lets us select the Complex, Real or Imaginary mode.

Circle fit (exact damping estimation)

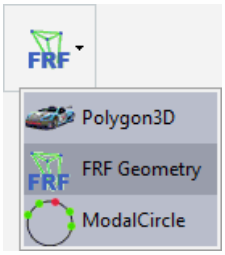
This is a visual control for basic modal analysis. You can use the yellow cursor in a 2D graph and set it to the frequency content of interest. Circle fit will find the nearest peak (resonance) and fit the circle into that data set. Then it will calculate the damping factor and more exact frequency (between line resolution of FFT).



- **From cursor channel** - the yellow cursor on the 2D graph is taken for frequency point determination.
- **Manual** - you can manually enter frequency point for circle fit procedure.
- **Peak search (manual mode)** - an area in which we will search for peak for circle fit.
- **Neighbour count** - a number of neighbours taken into account when doing the circle fit.

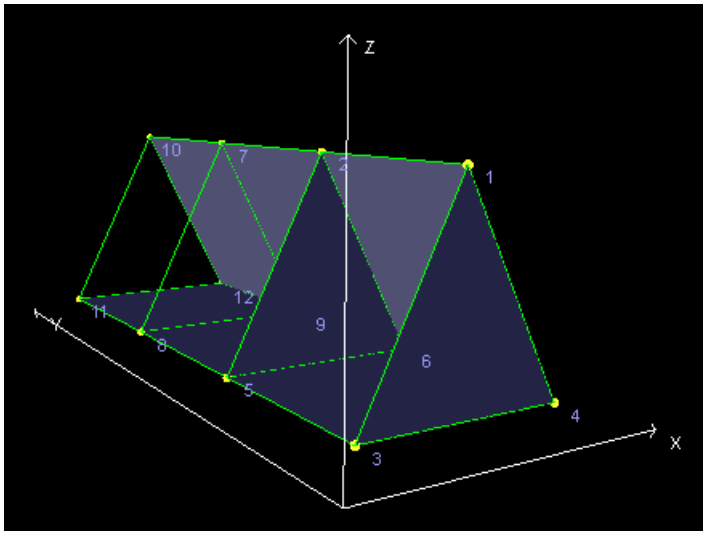
GEOMETRY DISPLAY

FRF Geometry visual control is used to visualize transfer function calculation. It can be found on custom control drop down menu in design mode. If the icon does not appear, the visual control FRFGeometry.vc must be added to Addons folder of Dewesoft.



Mouse controls

- Left click: rotate structure
- Right click: translate structure
- Left+Right click: zoom
- Mouse wheel: zoom



Geometry options

There are several options which can be set for the geometry:

- Data
- Load UNV - Loads structure data from external UNV (universal file format) file
- Editor - Opens FRF Geometry editor

Mode

Here you can select mouse left button behavior. It can be set to rotate, translate or scale the object.

Button Reset view will return back to default position.

View

Scale and Speed values define how much fast and how much nodes should move. Default value for both fields is 1.

Icons in the View section are used to enable/disable view options.

- Nodes
- Node labels
- Trace lines
- Surfaces
- Coordinate system

Show frequency

- **From cursor channel** - animated frequency is taken from yellow cursor on 2D graph
- **Manual** - manually define animated frequency

FRF Geometry Editor

Explanation of terms

For easier understanding, main terms are explained here.

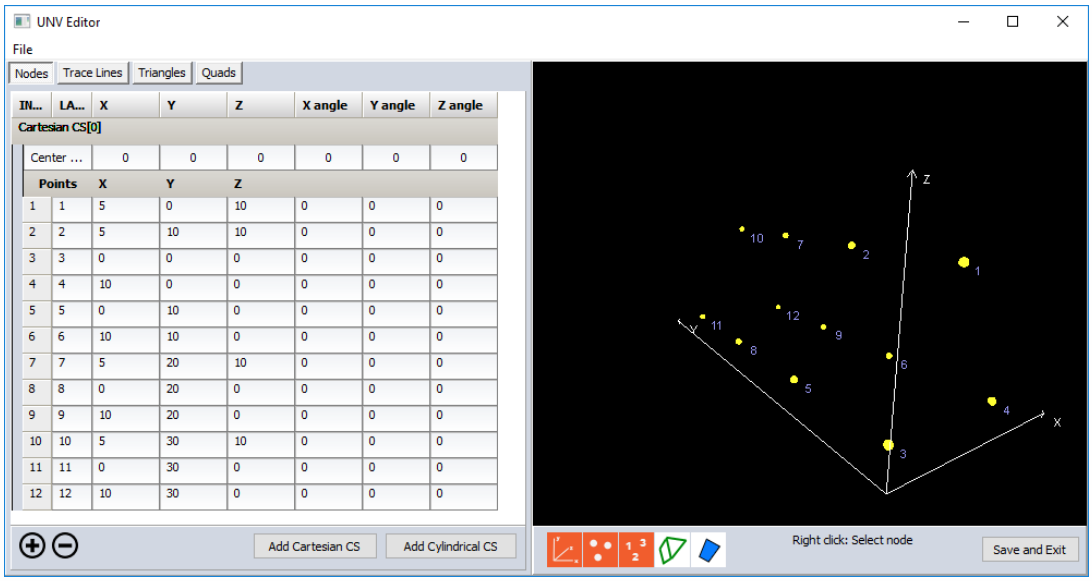
- **Node** - Node is point where sensor is positioned on object. Node is defined with location (X, Y, Z) and rotation around axes (X angle, Y angle, Z angle).
- **Trace line** - Trace line connects two nodes together. **Triangle** - Surface defined with 3 nodes
- **Quad** - Surface defined with 4 nodes.
- **Cartesian coordinate system** - Usually nodes are presented with Cartesian coordinate system. This means you have X, Y, Z position and rotation around all three axes. Coordinate system can be used for grouping nodes, because you can later rotate or translate them with Center point.
- **Cylindrical coordinate system** - Cylindrical coordinate system is used for easier creation of round objects. Points are defined with radius, angle and z (height) around coordinate systems center point.
- **Creation of structure** - If structure data is already loaded and we want to make a new one, we click on File -> Clear structure.

Nodes

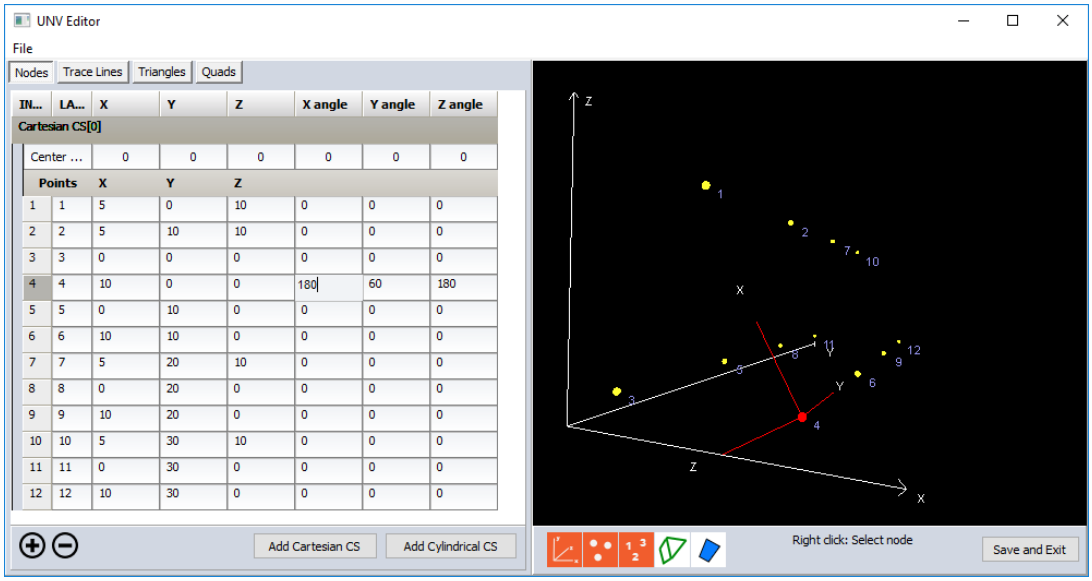
Creation of structure always begins with nodes, so we have to switch to Nodes tab. Now we have to create coordinate system in which we will define our nodes. This can either be Cartesian or cylindrical.

After coordinate system is created, we can add nodes with Plus button.

In the picture below, you can see Cartesian coordinate system with 12 nodes.



After nodes are created we can change their rotation (according to how sensor is rotated on object) with all three axes. Node can be selected with selection in node table or with right mouse click on structure preview window. When node is selected rotation is shown with small coordinate system located directly on node. In the picture below you can see selected and rotated node.



When node is selected, we can remove it by pressing the Minus button on nodes tab.

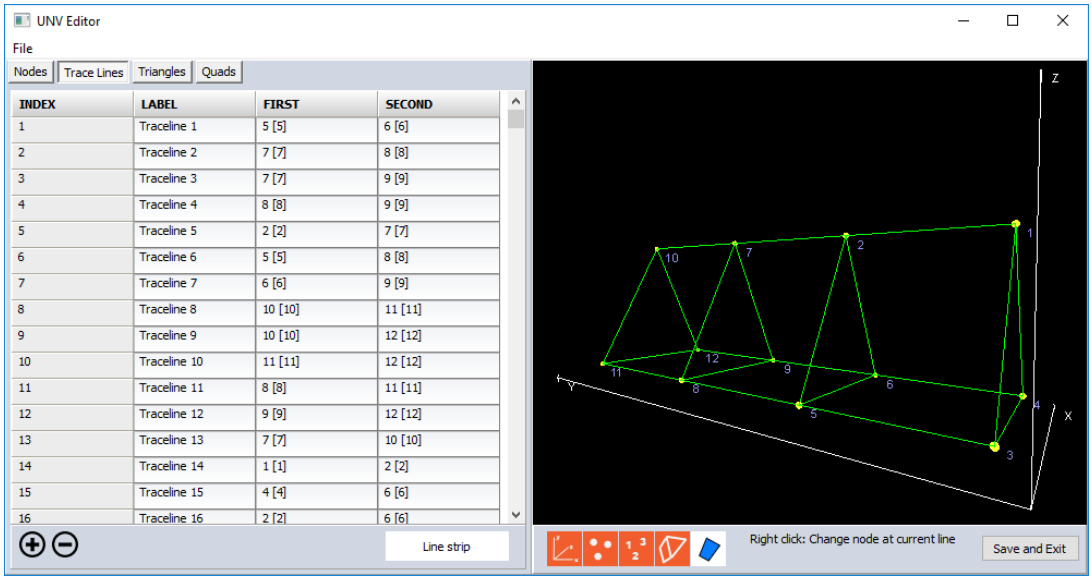
Trace lines

When basic nodes are defined we can go ahead and add trace lines. Easiest way to create trace lines is with Line strip button. After Line strip button is enabled, we can add trace lines automatically with right clicking on nodes in preview window. With each click, new trace line will be created connecting new node with previous one. When line strip is finished, click on Line strip button again to disable it – with that we come back to normal behavior.

If we don't want to draw connected line strip, we can also manually add trace line by pressing on the Plus button in Trace lines tab.

With that new trace line is added and we have to select nodes which connect them – we can do that either with selecting nodes in the left table or by right clicking in preview window.

In the picture below we can see object with some trace lines added.



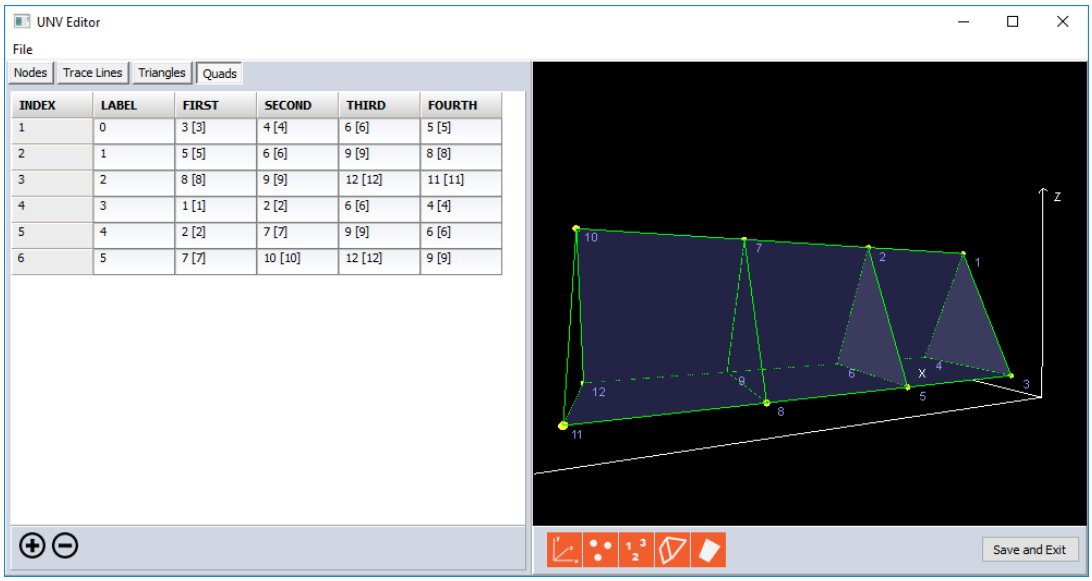
Surfaces

Currently selected trace line is marked with red color in the preview window. This helps selecting nodes trough table since we always see what is selected in 3d preview window.

To improve object preview we can also add surfaces (triangles and quads). Both are added in the same way.

For example, to add a quad, we select Quads tab, press on the Plus button and then select 4 nodes with selecting them in the left table.

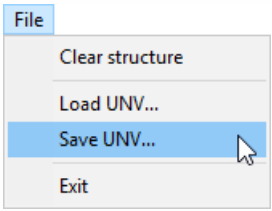
Below you can see the same object as in previous screenshots with added surfaces.



Saving structure

After structure is defined we can store it in external UNV file by clicking on File -> Save UNV…«. If structure is stored in external file, we can later use it another project by selecting File -> Open UNV…. If we need structure just for one setup, we can also just click on Save and exit button in the lower right corner. With that, structure is stored within Dewesoft setup and data file.

If you want to have structure stored just in setup, don’t forget to press Save setup later in Dewesoft channel setup screen.



ORDER TRACKING

Dewesoft order tracking method is used to extract the harmonic components related to a rotational frequency of the machine. With order extraction, we can see a specific harmonic component which relates to certain machine fault.

The mechanical vibration pattern is a mixture of excitation frequencies, usually related to rotational speed, such as unbalance, eccentricity, bearing faults and other and machine response function, which relates to machine natural frequencies based on the structure and mounting of that machine.

Machine vibration pattern - is a mixture of excitation frequencies, usually related to rotational speed, such as unbalance, eccentricity, bearing faults and other machine response functions, which relate to machine natural frequencies based on the structure and mounting of that machine.

Harmonic component - which relates to certain machine fault: - the first order (harmonic) usually relates to unbalance of the machine - the second harmonic often relates to eccentricity - if we have for example 9 rotor blades, 9th harmonic relates to errors on the blades - if we have for example 31 teeth on the gear, then the 31st harmonic will show the gear mesh frequency...

Final measured vibration - The ratio between excitation and system response is defined by the system transfer curve. So the final measured vibration of the system is a product of excitation force and system transfer curve. Since the transfer curve is fixed, we get different responses for excitations at different rotation speeds. When the excitation passes natural frequency, we get so-called resonance with increased vibration amplitudes which could be fatal to the machine.

NOTE: **To measure Order tracking, we need at least one accelerometer for measuring the vibration and angle sensor for measuring RPM.**

- **Required hardware** - Sirius, Dewe 43
- **Required software** - SE or higher + ORDTR option, DSA or EE
- **Setup sample rate** - At least 10 kHz

For additional help with Order tracking Application visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training.

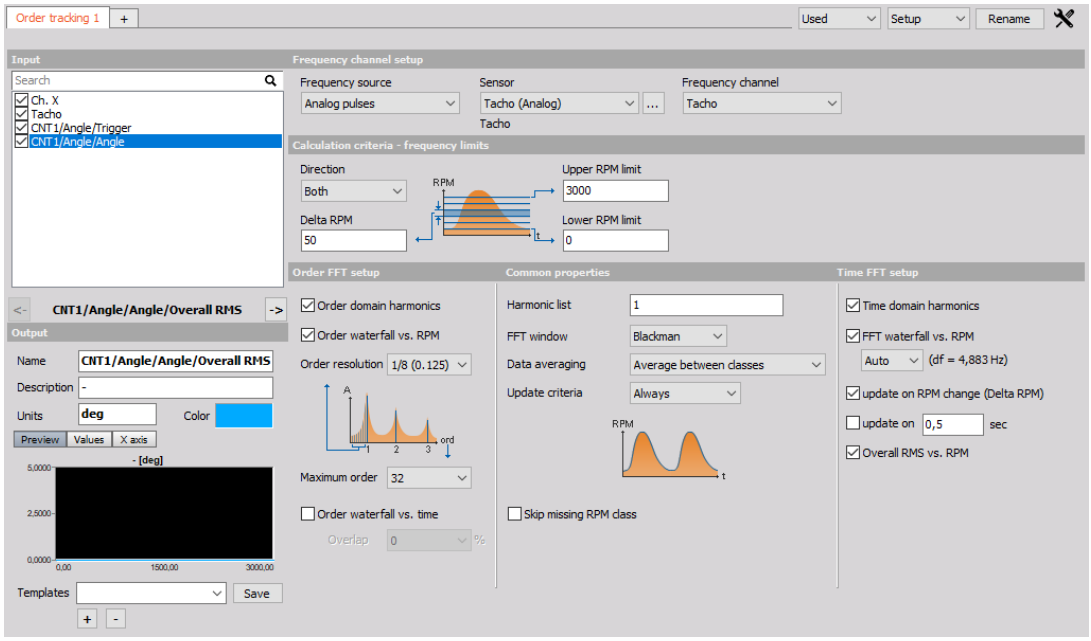
For help with Counters see -> Setup -> Counters.

Add / New Order tracking module

For instructions on how to Add new module see -> Setup -> Add module.

Setup

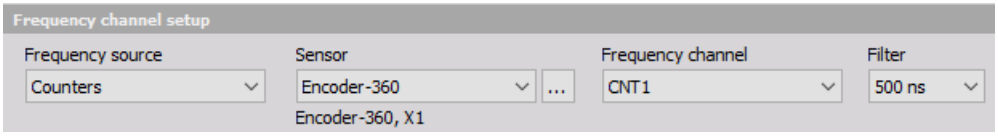
When an order tracking module is added, we get the following window with the settings for order tracking.



This screen has the following main sections:

- Input channel selection
- Output channel settings
- Frequency channel setup
- Calculation criteria - frequency limits
- Order FFT setup
- Common properties
- Time FFT setup

Frequency channel settings



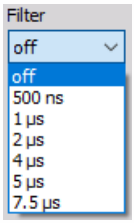
There are several possible sources of the frequency which can be selected from Frequency source drop-down list:

We can have the frequency source as Counters. In this case, the angle sensor should be an encoder or gear tooth. Then we select the sensor itself from Sensor drop-down list and the source where this sensor is connected to.

If we choose the encoder, then we need to connect the encoder to the counter input.

By selected encoder on this section, we have two additional settings. Input Filter defines the

digital filter of the data to prevent double triggers (same function as in counters).



The second option is to select the Analog pulses as the source. In this case, the sensor should be connected to an analog input and the angle sensor math is used for getting the angle and the trigger from analog data. Please be careful that the sample rate is high enough to catch the pulses also at high frequencies.

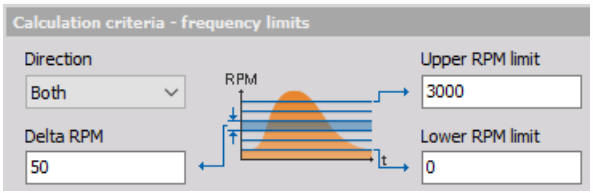
Define the trigger levels by pressing the " ... " button which appears on the left side of the Frequency channel field.

When the " ... " button (on the right of Sensor field) is selected, we get Counter sensor editor window where angle sensor should be defined. see -> Counter sensor editor.

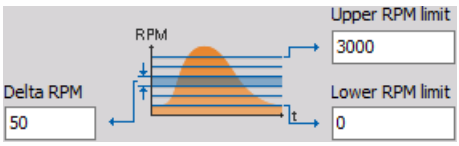
The third option is the RPM channel. In this case, the frequency source comes directly from an analog channel which measures the frequency.

The fourth option - Torsional vibration is available when we have torsional vibration. The source can come directly from torsional vibration.

Calculation criteria - frequency limits

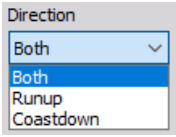


In this section we need to define the lower (Lower RPM limit) and upper (Upper RPM limit) rotation speed which we would like to see and the difference (Delta RPM) in rotation speed. These three parameters are important for reserving memory of waterfall FFT based on the order tracking.

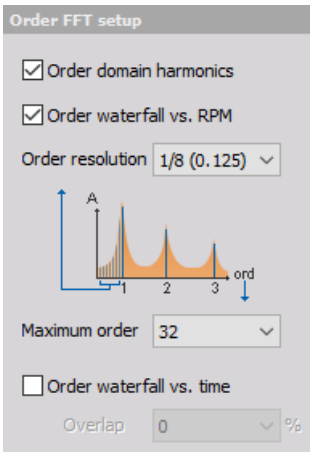


We can also define the Direction of triggering either to capture only Run-up, Coast-down or Both. This criterion will tell the order tracking module when to take the measurements.

Here can we also Skip missed RPMs with checking this box.

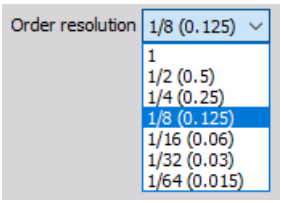


Order FFT setup

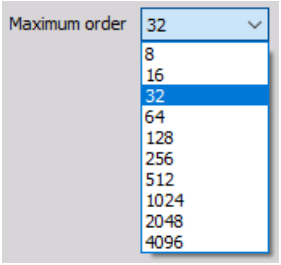


In this section, we define the order history FFT properties, but also the properties for order extraction.

First, we define the Order resolution, which reflects also the minimum number of revolutions acquired for a single calculation. If we set 1/16 order for the resolution, we need to acquire at least 16 cycles for a single data point.

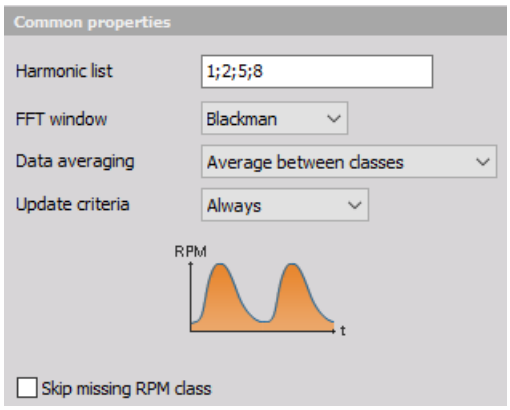


The Maximum order defines how many orders (harmonics) we will get and can be selected from the drop-down list.



This value and the maximum RPM defines the sample rate needed for acquisition. If the sample rate is not high enough, this will be written as the error message in the caption.

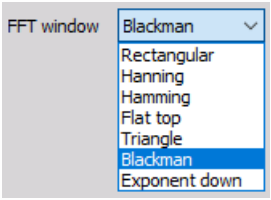
Common properties



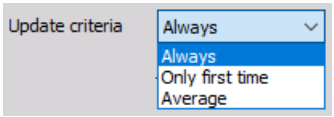
Order extraction setup - We will see all the orders in the history FFT, but if we want to draw the Bode or Nyquist run-up plots (in the x-y recorder), we need to have the data available as the channels. In this section, we define the orders (or harmonics) which will be available as channels From the list (see right) or All.

If we have the frequency source with a reference point (tacho or encoder), we can also get the phase, real and imaginary component of each harmonic.

We set (select from drop-down list) the FFT window type for calculation of the history FFT.

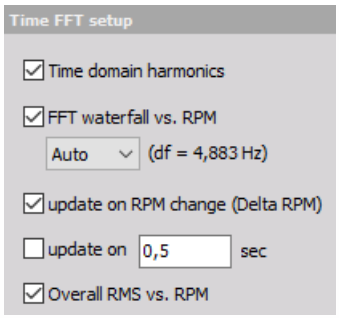


In this section, we define the time Update criteria from the drop-down list for taking new samples. We can define to acquire the data Only at first time Run-up / Coast-down or Always when passing through some frequencies.



We can also define the Maximum time limit. If the frequency doesn't change within a specific time, we will still get another point for harmonics and 3D history.

Time FFT setup



- **Time domain harmonics** - This creates a complex output channel showing the amplitude of harmonics in a time domain. In the harmonics list section, we define the harmonics that we want to extract. **FFT waterfall vs. RPM** - he resolution of the FFT waterfall vs. RPM can be defined by the number of lines (delta frequency is also shown). It can be also set to Auto - the resolution is calculated from the max. order, sample rate, and order FFT size.
- **Update on RPM change (Delta RPM)** - Delta RPM will define when a new update of the waterfall spectrum and also the extracted order domain harmonics are calculated.
- **Update on** - In case, the RPM is not changing only channels from Time FFT setup can be calculated according to the maximum time limit setting.
- **Overall RMS vs. RPM** - This option will create a channel that shows the overall RMS amplitude over the range of RPMs

Output channels settings

The lower-left area displays the output channel settings with fields like in the analog channel setup: Name, Units, Color, Minimum value and Maximum value, also a symbolic display of signal values.

For help with Output Channel list see -> Setup -> Output Channel list.

Order tracking can have more output channels according to the selection of Output extracted harmonics as channels.

TORSIONAL VIBRATION

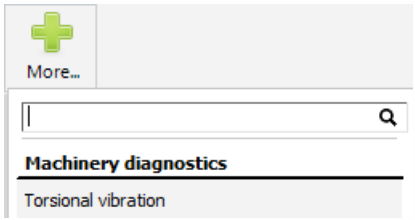
Dewesoft Torsional vibration application module allows measure dynamic and static bending and vibration of the shafts. The torsional vibration also measures the twist of the shaft with higher RPM. With torsional vibration module two different parameters can be measured: rotational vibration and torsional vibration.

- **Rotational vibration** - is simply the dynamic part of rotation speed. If we measure the rotation speed of shaft with high precision, we will notice that in some regions of the run up we get high deviation of rotation speed. This is caused by angular vibration crossing angular natural frequency of the shaft. It is calculated with cutting off the DC component of the rotation speed or rotation angle.
- **Torsional vibration** - is an oscillation of angular motions (twist) which occur along rotating parts such as gear trains, crank shafts or clutches. We need two encoders to measure the torsional vibration, so the torsional vibration is actually a difference between angles of two encoders.
- **Extract order** - from Torsional vibration or Rotational vibration see -> Extract order.

To measure torsional or rotational vibration, we need a Sirius module with two counter inputs, because all other methods do not have precision needed to do this.

- **Required hardware** - Sirius with counter expansion, Dewe43
- **Required software** - Dewesoft DSA license (OPT-ORDTR and OPT TORVIB option) or 1 month Evaluation license.
- **Setup sample rate** - At least 10 kHz

Dewesoft Torsional vibration measurement is activated by selecting the Torsional vibration calculation in Modules menu. Ch.setup -> More... (Add module) -> Torsional vibration tab.



For instructions on how to Add new module see -> Setup -> Add module.

After selecting this option, a tab labeled Torsional vibration appears in the Dewesoft Setup screen (see picture below) to define and setup Torsional vibration module(s).

To use Torsional vibration module we don't need to set anything on analog or counter channels.

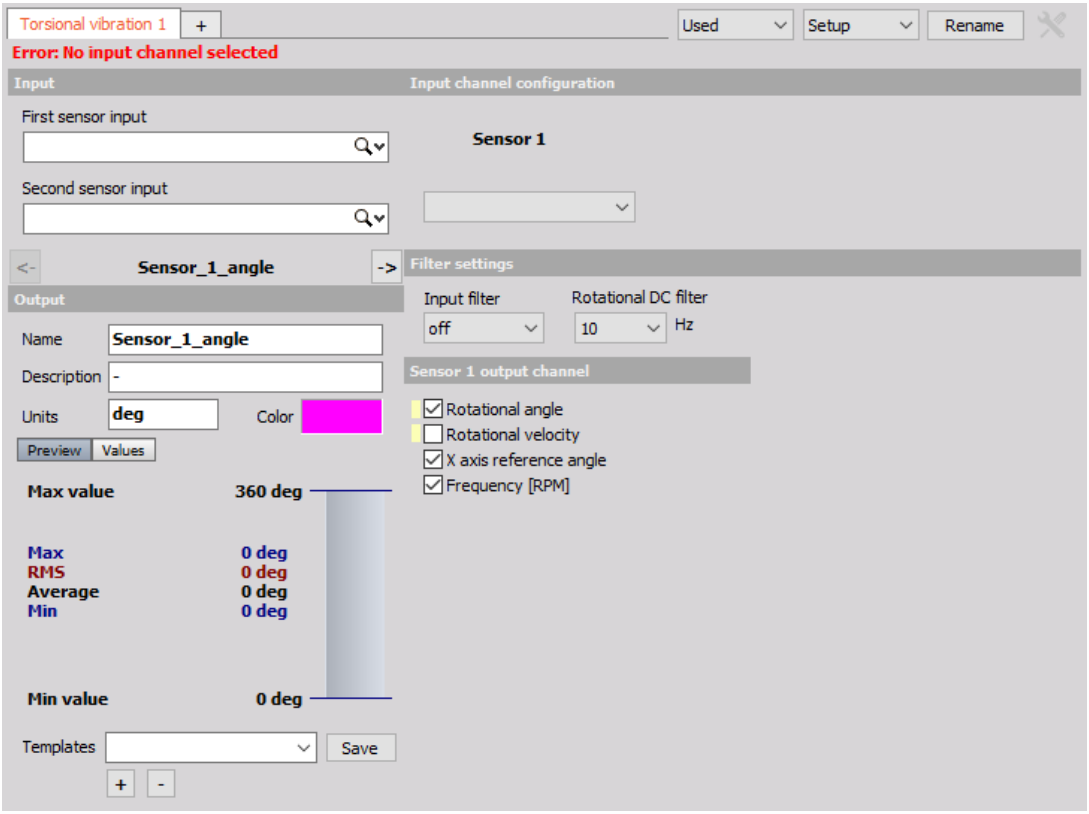
Torsional and rotational vibration can be measured with either encoder (up to 3600 pulses per revolution) or special RIE sensor which has less resolution (up to 720 pulses per revolution) but is much less sensitive to vibrations which could damage standard encoders.

Two different procedures of Torsional vibration module setup are:

- Rotational vibration setup
- Torsional vibration setup

Rotational vibration setup

After add new Torsional vibration module screen like this will appear:



This screen has five main sections:

- Input channel - sensor selection
- Output channel settings
- Input channel configuration - for Sensor 1
- Filter settings
- Sensor output channel definition for Sensor 1

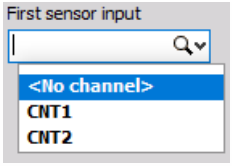
With using View Channel List button we can show different view on Torsional vibration module setup -> Channel list with all defined output channels.

Input channel - sensor

First of all select the input channel - sensor 1 (the first connected sensor) from the First sensor input drop down list.

Since we have connected in example above the first sensor to CNT0, we need to select it from list.

Please note that CNT1, CNT3 and CNT5 are not available, since they will be used internally to calculate exact frequency.

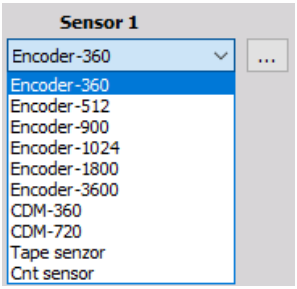


Input channel configuration

Since we only have one(first) Rotational vibration sensor connected, on Rotational vibration setup screen there is only one sensor 1 Input channel configuration section.

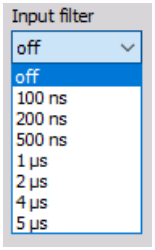
On this section we can set:

Sensor type - select the right Encoder sensor type from drop down list:



Both sensors must count in positive direction. If we use encoders, there is a chance that the counting direction of a sensor is Negative.

Input filter for the counters



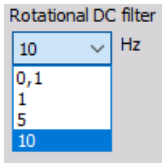
Input filter is needed to prevent glitches and spikes on the signal, is also one important setting to prevent double counts. The Input filter can be set in a range between 100ns and 5µs. The optimal settings are derived from the following equation:

$$\mathrm{InputFilter}\ \backslash[\mathrm{s}] \leq \frac{10}{\mathrm{RPM_{MAX}} \cdot \mathrm{PPR}}$$

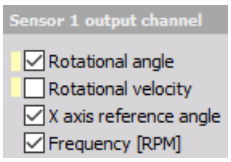
- $\mathrm{RPM_{MAX}}$ - max. revolution s per minute [s-1]
- PPR - pulses per revolution [-]

Rotational DC filter needs to be set to cut the DC component of the RPM. We need to set the

filter to include all wanted frequencies, but not too low, otherwise we will have static DC deviations on the output signal.



Sensor output channel



The Sensor 1 output channels can be:

- **Rotational angle** -filtered angle value of vibration
- **Rotational velocity** - filtered velocity vibration value
- **X axis reference angle** - the reference angle which is always from 0 to 360° and can be used as reference in angle based XY diagram
- **Frequency [RPM]** - frequency in RPM unit

Select desired option with checking box beside caption. For each selection in this section one separate output channel is created (see below).

Output channel settings

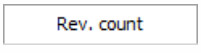
The lower-left area displays the output channel settings with fields like in the analog channel setup: Name, Units, Color, Min val and Max val, also symbolic display of signal values.

Torsional vibration module can have more output channels according to selection in Sensor 1 output channel. For each selection in this section one separate output channel with default name is created:

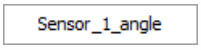
For Sensor 1 output channel section selected:

Rotation angle choice

Rotation velocity choice



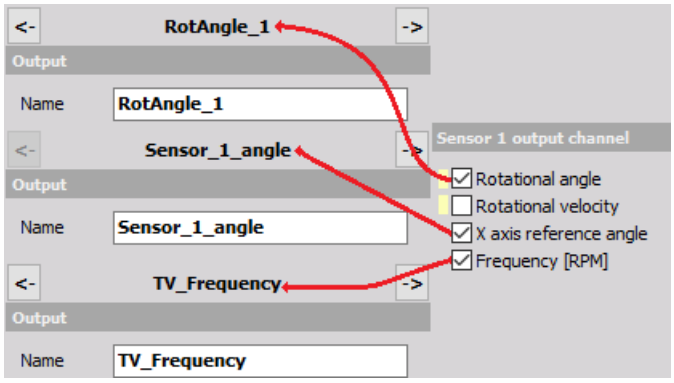
X axis reference angle choice



Frequency [RPM] choice

TV_Frequency

Example:



At least one output channels must be checked.

Use the Previous and Next buttons to navigate between the output channels in OUTPUT channel settings part of screen.

->

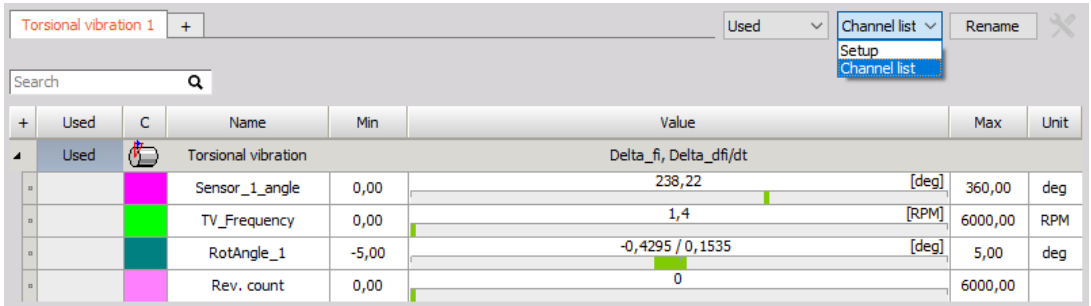
<-

Channel list view

After adding new Torsional vibration module we can navigate to Channel list.

For help with Output Channel list see -> Setup -> Output Channel list.

When this button is selected on Rotational vibration module setup, Channel list screen with all defined output channels appear:



For information about Channel grid columns see -> Setup -> Channel grid.

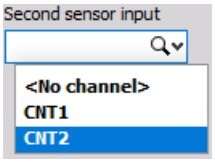
Torsional vibration setup

Input channel - sensor

To measure the torsional vibration we need to select both input channels in torsional vibration setup.

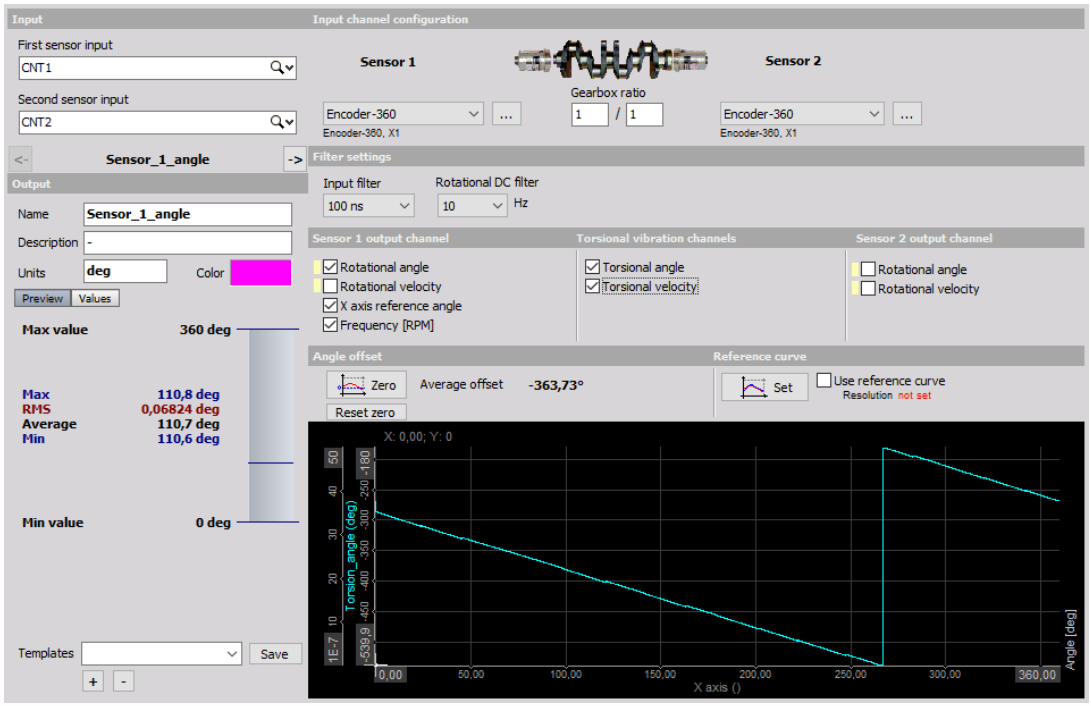
Beside first input (channel - sensor 1) also second input (channel - sensor 2) from the Second sensor input drop down list must be select.

Since we have connected in example below the second sensor to CNT2, we need to select it from list.



Please note that CNT1, CNT3 and CNT5 are not available, since they will be used internally to calculate exact frequency.

After selecting a second input channel - sensor 2 screen like this will appear:



This screen has the following main sections:

- Input channel - sensor selection
- Output channel settings
- Input channel configuration - for Sensor 1 and Sensor 2
- Filter settings

- Sensor output channels definition for Sensor 1 and Sensor 2
- Torsional vibration channels definition
- Angle offset / Zeroing
- Reference curve configuration

With using View Channel List button we can show different view on Torsional vibration module setup -> Channel list with all defined output channels.

Input channel configuration

Since we have two (first and second) Torsional vibration sensors connected, on Torsional vibration setup screen beside setup for Sensor 1, a second Input channel configuration for Sensor 2 section appeared.

On these setups we need to separately define Sensor 1 and Sensor 2 properties:

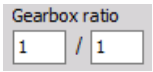
- Pulses per revolution
- Sensor type

For information about Pulses per revolution and Sensor type settings see -> Rotational vibration setup.

Some configuration settings are common for both sensors:

- Input filter for the counters
- Rotational DC filter

Gearbox ratio - If we have gearbox in between sensors, we need to enter Gearbox ratio.



NOTE: **Both sensors must be set to positive counting and both input filter must be set to same value!**

Sensor output channels

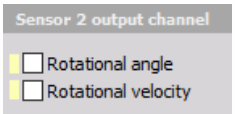
Sensor 1 output channel

The Sensor 1 output channel is same as for Rotational vibration setup.

Sensor 2 output channels

The Sensor 2 output channels can be:

- **Rotational angle** - filtered angle value of vibration



- **Rotational velocity** - filtered velocity vibration value

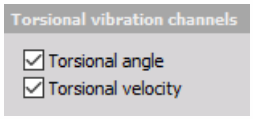
Select desired option by checking box beside caption.

For each selection in Sensor 1 and Sensor 2 section one separate output channel is created.

Torsional vibration channels

The Torsional vibration channels can be:

Torsional angle - dynamic torsional angle that is the angle difference from sensor 1 to sensor 2



- **Torsional velocity** - difference in angular velocity from sensor 1 to sensor 2

Select desired option by checking box beside caption.

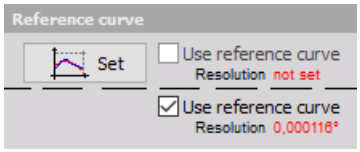
For each selection in this section one separate output channel is created (see below).

Angle offset / Zeroing

In Average offset field Angle offset (difference) is displayed. A click on the Zero button removes the angular difference (offset) between the two sensors (set angle offset to 0).

Reference curve

A click on the Set button records the current torsion angle over one revolution as reference. When Use reference is checked, the recorded reference is subtracted in angle domain from the current torsion angle. In this way you can overcome torsion errors caused by the sensors or their fixing.



Output channel settings

The lower-left area displays the output channel settings with fields like in the analog channel setup: Name, Units, Color, Min val and Max val, also symbolic display of signal values.

For help with Output Channel list see -> Setup -> Output Channel list.

Torsional vibration module can have more output channels according to:

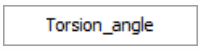
- **Sensor 1 output channel section**

The output channel is same as for Rotational vibration setup.

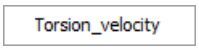
- **Torsional vibration channels section**

For on Torsional vibration channels section selected:

Torsional angle choice



Torsional velocity choice



- **Sensor 2 output channels section**

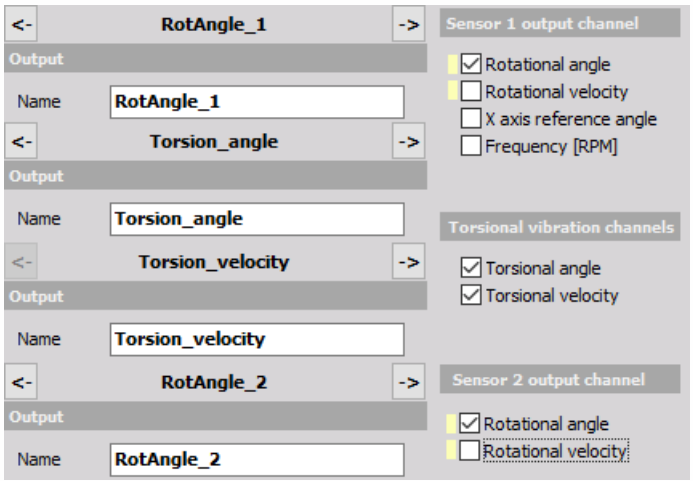
for on Sensor 2 output channels section selected:

Rotational angle choice

Rotational velocity choice

For each selection on these sections one separate output channel is created.

Example:



At least one output channels must be checked.

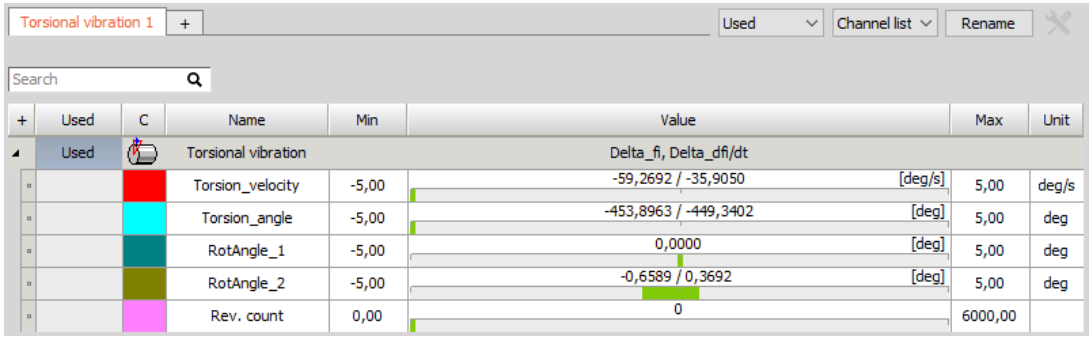
Use the Previous and Next buttons to navigate between the output channels in OUTPUT channel settings part of screen.

Channel list view

After adding new Torsional vibration measurements we can navigate to Channel list.

For help with Channel list see -> Setup -> Output Channel list.

When this option is selected on Torsional vibration module setup, Channel list screen with all defined output channels appears:



+	Used	C	Name	Min	Value	Max	Unit
	Used		Torsional vibration		Delta_fi, Delta_dfi/dt		
			Torsion_velocity	-5,00	-59,2692 / -35,9050 [deg/s]	5,00	deg/s
			Torsion_angle	-5,00	-453,8963 / -449,3402 [deg]	5,00	deg
			RotAngle_1	-5,00	0,0000 [deg]	5,00	deg
			RotAngle_2	-5,00	-0,6589 / 0,3692 [deg]	5,00	deg
			Rev. count	0,00	0	6000,00	

For information about columns in Torsional vibration channel list see -> User Guide -> Rotational vibration setup.

Extract order

NOTE: **To extract orders from rotational vibration we need to add order tracking module. For frequency source we need to define the same counter input as in rotational vibration.**

NOTE: **To extract orders from torsional vibration we choose the Torsion angle or Torsion velocity or both for the input channel of order tracking math module.**

ELECTRICAL MEASUREMENTS

In Electrical measurements we can choose from following modules:

- Power grid analysis

POWER GRID ANALYSIS

Dewesoft Power is one of the most complex mathematical modules inside Dewesoft modules which provide all functions which are needed for an analysis of power grids.

Some Dewesoft power module advantages are:

- When measuring voltage and current there are several ways to calculate the formula. Even with knowing basic equations, real world measurement is by far not that simple. Transducers and amplifiers have different transfer characteristics, which needs to be corrected, lots of AD cards have also phase shifts between channels and the line frequency is not ideal but varies with time. Power module compensates all possible errors and is checked to be perfect tool for power measurements even for most demanding applications.
- Dewesoft power module can not only handle the line to earth voltages but also the line to line voltages. Depending on the selected type of wiring schematic the conversion is done inside.
- In addition to the phasors also the RMS values of voltage and current, the phase angles and the power values can be shown for different power systems.
- Based on a calculation in the frequency domain the input channels can be calibrated very efficient in amplitude as well as in phase. Internal amplifiers, external transformers and clamps can be corrected with the use of this function.
- The range of applications for the systems is very wide:
  - P, Q, S, D
  - $\cos\phi$, Power factor
  - $P, Q, \cos\phi$ for each harmonic
  - Symmetrical Components (positive, negative and zero sequence components)
  - Period values (½ cycle, cycle,...)
- All frequencies can be analyzed. In addition to harmonics FFT a full frequency based FFT is available.
- If more then one power module is defined each one can have it‘s own frequency. So it is possible to do power measurements on different frequency systems with only one instrument at the same time.

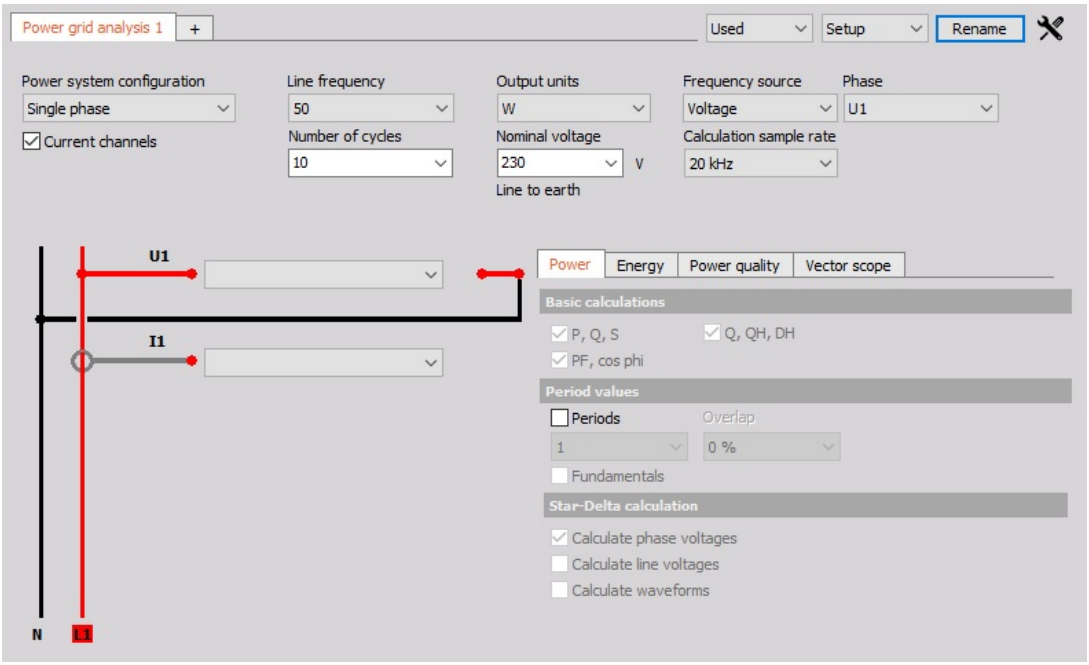
To measure Power, we need at least two input channels connected to your instrument for measuring voltage and current.

- **Required hardware** - Dewesoft Sirius
- **Required software** - SE or higher + POWER option or EE
- **Setup sample rate** - At least 5 kHz

For additional help with Power module visit Dewesoft PRO training → Dewesoft Web page → PRO Training.

Power module

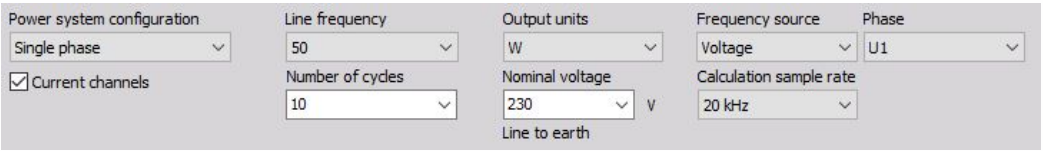
After adding the Power module and opening it by clicking on icon, following setup screen is opened:



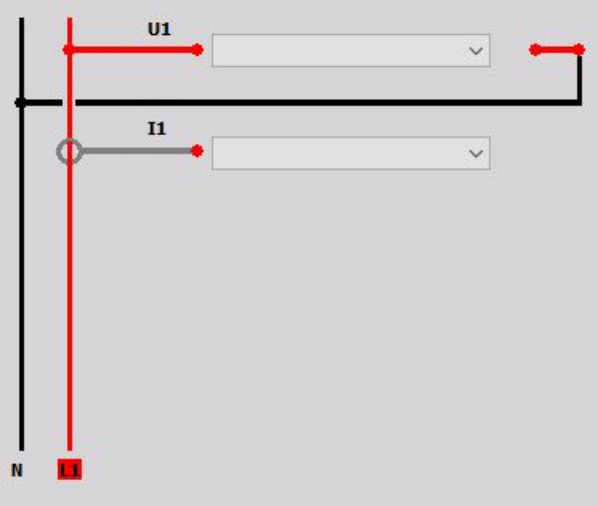
For detailed information on how to initialize Power module see → Add module.

Power grid analysis setup is divided in three main categories:

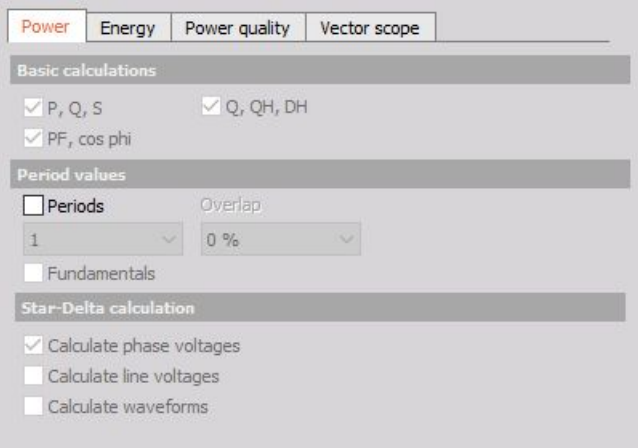
- Configuration



- Wiring



- Calculations



NOTE: **We need to have the correct unit for voltage and current. Voltage units could be V, kV and MV while the current units could be either A or kA. These values we entered in the Units field.**

Voltage measurements

We can split voltage measurements into several sections: high voltage grid measurements (in kilovolts), where we need voltage transducers, direct low voltage grid measurements (120/230 V) and low voltage measurements (up to 50 V). The high voltage converters convert kilovolts voltage signals to a measurable range - up to one kilovolt.

It is very important that the measurement Range is chosen according to the expected signal. We need to take special care for the settings of the Low-pass filter. If this setting is lower than half of the sample rate, it will cut the signals already in the range of the measurements. Sometimes this is needed, but more often this filter is set by mistake to the low range and then the measurement results will be invalid.

Current measurements

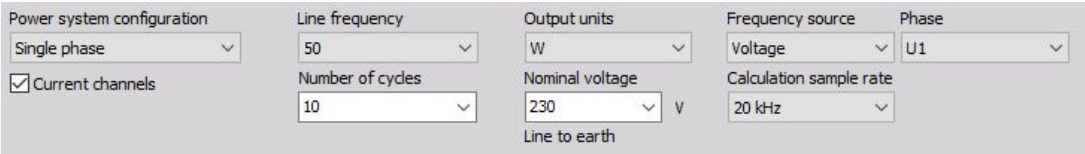
Basic procedures of Power application setup are:

- Power module setup for applied hardware
- Calibration of current

For additional information visit: Dewesoft WEB Power Manual.

CONFIGURATION

After adding new Power module, a screen like this will appear:



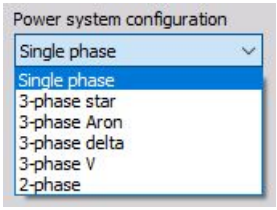
This screen is divided to the following two main sections:

- Power system configuration
- Common Power module settings

Depending on the type of system, Dewesoft calculates and outputs different channels. For Single Phase, 2-phase and 3-phase with default settings and additional channels if extra calculations are selected.

Power system configuration

We can define type of power system from Power system configuration drop down list.



Dewesoft enable following types of power modules:

- DC measurement
- Single phase AC measurement
- 2-phase AC measurement
- Five types of 3-phase AC systems:
  - Star configurations
    - 3-phase star
    - 3-phase Aron (only two current measurements)
  - Delta configurations
    - 3-phase delta
    - 3-phase V (only two currents)
    - 3-phase with two wattmeters (only two voltage and two currents)

All AC types can have the current measurement either enabled or disabled. Multiphase systems also allow calculating the line-to-line voltages.

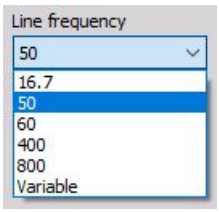
For closed description of these types see → Wiring scheme.

For information about Current channels and Calculate line voltages see → Power module calculations.

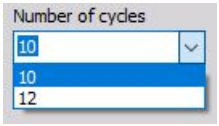
Common Power module settings

In this section you can define several settings:

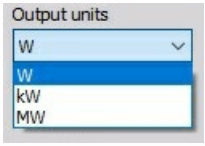
- **Power system configuration**
- **Current channels and Calculate line voltages** For information about Power system configuration see → Configuration, Wiring scheme, Calculations. For information about Current channels and Calculate line voltages see → Power module calculations.
- **Line frequency** - Line frequency depends on the grid frequency. We can have 16 ⅔ Hz (in trains), 25 Hz, 50 Hz (Europe), 60 Hz (USA); 400 and 800 Hz (vehicle power systems) or Variable (for frequency converters).



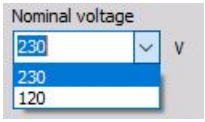
- **Number of cycles** - Choose from the drop-down list or freely enter any value. Standard values are 10 for 50 Hz grids and 12 for 60 Hz grids. This basically defines the measurement interval.



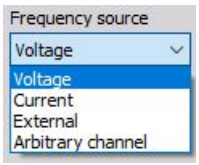
- **Output units** - Choose power unit for your system from the drop-down list.



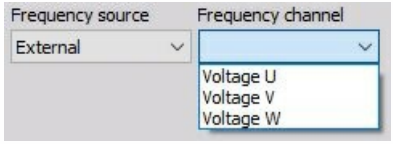
- **Nominal voltage** - Choose nominal voltage of your system from the drop-down list or enter any other value.



- **Frequency source** - Source for frequency measurement can be: Voltage, Current or External input channel and can be selected from the drop-down list.



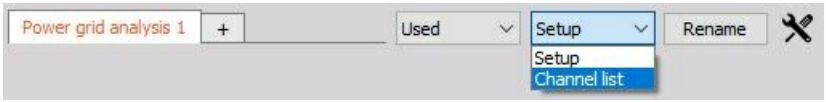
When we select External as source, input channel for this source can be selected from the drop-down list of available channels.



Voltage is the most common source for frequency measurement. Current is used most widely with frequency converters. External frequency is mostly used on distributed systems when we want to have only one source for frequency.

SINGLE PHASE OUTPUTS

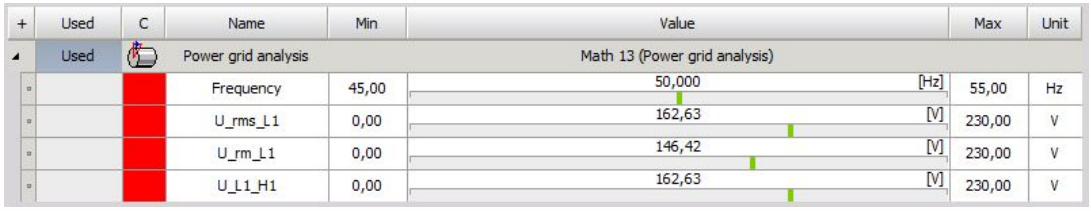
To display all channels go to second tab (Setup) in edit module section and select the Channel list from the drop down menu.



Basic output channels

If no additional options are checked Dewesoft automatically creates following output channels:

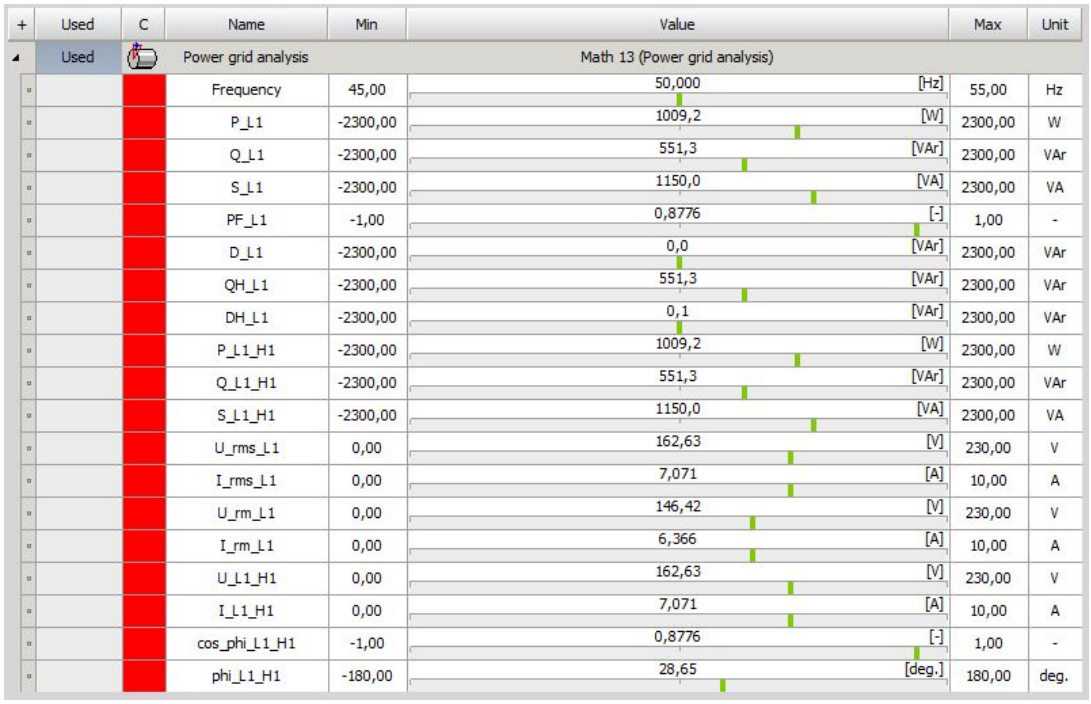
Current channels = disabled



+	Used	C	Name	Min	Value	Max	Unit
◢	Used		Power grid analysis		Math 13 (Power grid analysis)		
▫			Frequency	45,00	50,000 [Hz]	55,00	Hz
▫			U_rms_L1	0,00	162,63 [V]	230,00	V
▫			U_rm_L1	0,00	146,42 [V]	230,00	V
▫			U_L1_H1	0,00	162,63 [V]	230,00	V

To ‘Used’ analog channels voltage (U_H1 and U_rms) and frequency channels are added.

Current channels = enabled



+	Used	C	Name	Min	Value	Max	Unit
◢	Used		Power grid analysis		Math 13 (Power grid analysis)		
▫			Frequency	45,00	50,000 [Hz]	55,00	Hz
▫			P_L1	-2300,00	1009,2 [W]	2300,00	W
▫			Q_L1	-2300,00	551,3 [VAr]	2300,00	VAr
▫			S_L1	-2300,00	1150,0 [VA]	2300,00	VA
▫			PF_L1	-1,00	0,8776 [-]	1,00	-
▫			D_L1	-2300,00	0,0 [VAr]	2300,00	VAr
▫			QH_L1	-2300,00	551,3 [VAr]	2300,00	VAr
▫			DH_L1	-2300,00	0,1 [VAr]	2300,00	VAr
▫			P_L1_H1	-2300,00	1009,2 [W]	2300,00	W
▫			Q_L1_H1	-2300,00	551,3 [VAr]	2300,00	VAr
▫			S_L1_H1	-2300,00	1150,0 [VA]	2300,00	VA
▫			U_rms_L1	0,00	162,63 [V]	230,00	V
▫			I_rms_L1	0,00	7,071 [A]	10,00	A
▫			U_rm_L1	0,00	146,42 [V]	230,00	V
▫			I_rm_L1	0,00	6,366 [A]	10,00	A
▫			U_L1_H1	0,00	162,63 [V]	230,00	V
▫			I_L1_H1	0,00	7,071 [A]	10,00	A
▫			cos_phi_L1_H1	-1,00	0,8776 [-]	1,00	-
▫			phi_L1_H1	-180,00	28,65 [deg.]	180,00	deg.

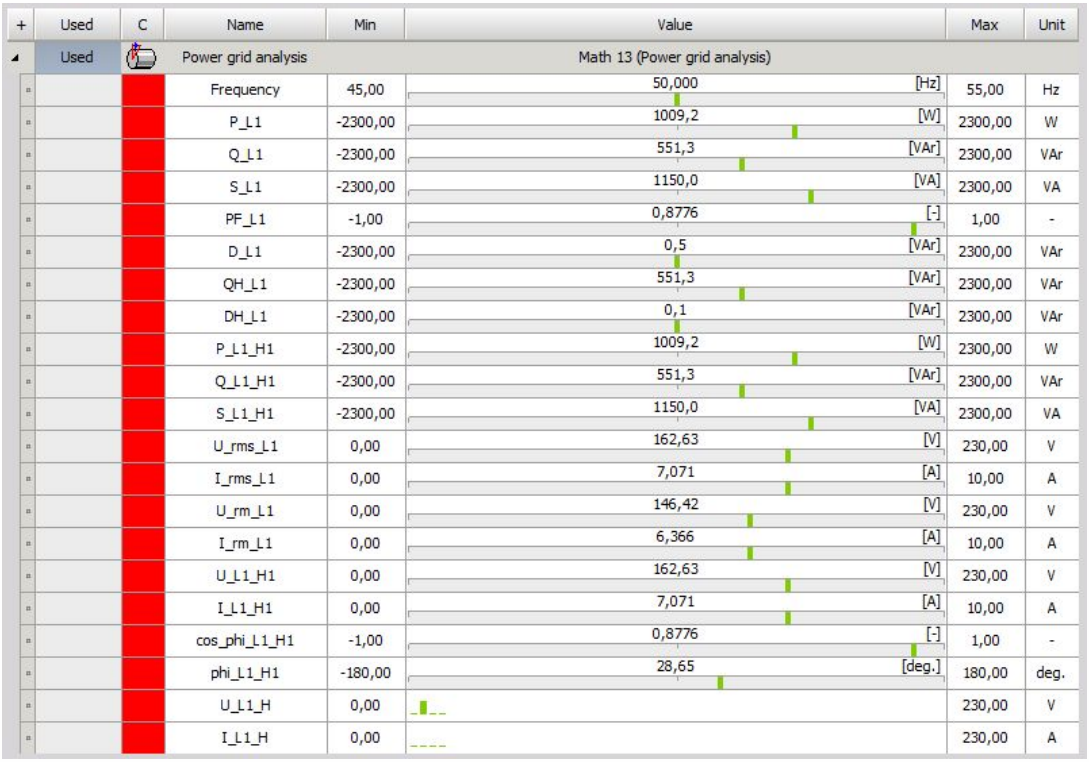
To basic channels emphasized channels are added -> current and power channels.

NOTE: **H1 harmonics channels are created anyway for: voltage and current channels (U and I) and power channels (P, Q, S, phi, cos phi).**

Harmonics output channels

Dewesoft creates following harmonic output channels for Single phase Power module (example for 3 harmonics and Current channels choice = enabled):

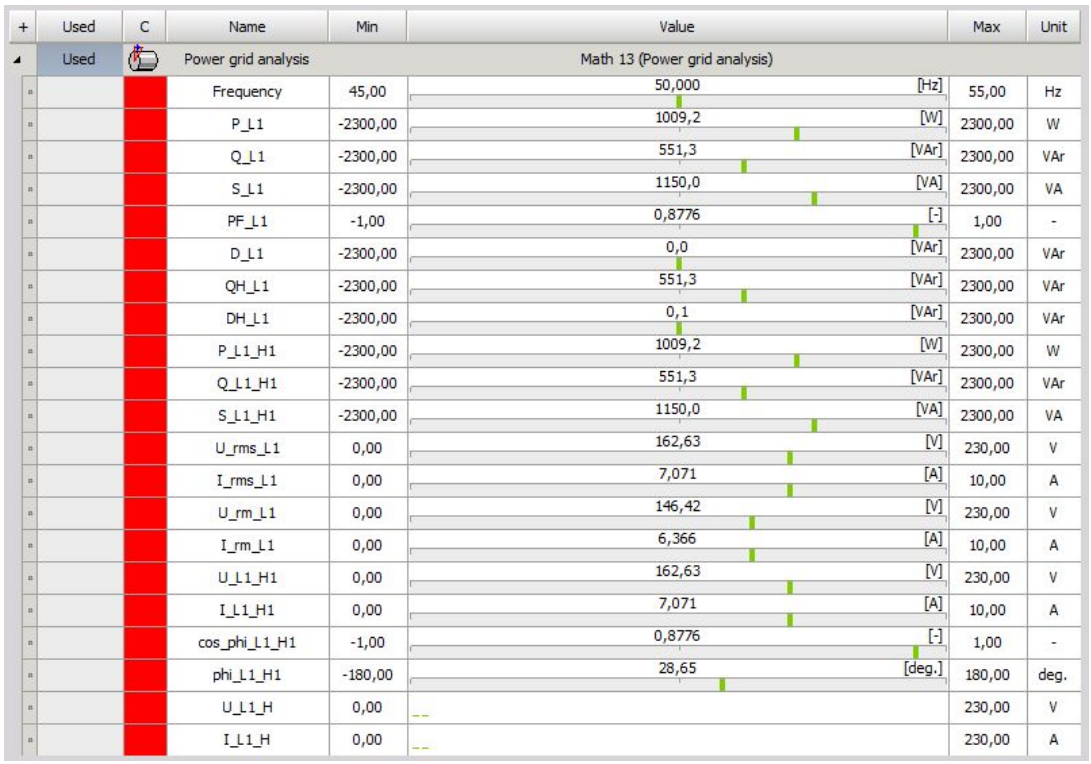
To basic channels for All Harmonics emphasized channels are added -> current and power harmonics channels.



+	Used	C	Name	Min	Value	Max	Unit
◢	Used		Power grid analysis		Math 13 (Power grid analysis)		
			Frequency	45,00	50,000 [Hz]	55,00	Hz
			P_L1	-2300,00	1009,2 [W]	2300,00	W
			Q_L1	-2300,00	551,3 [VAr]	2300,00	VAr
			S_L1	-2300,00	1150,0 [VA]	2300,00	VA
			PF_L1	-1,00	0,8776 [-]	1,00	-
			D_L1	-2300,00	0,5 [VAr]	2300,00	VAr
			QH_L1	-2300,00	551,3 [VAr]	2300,00	VAr
			DH_L1	-2300,00	0,1 [VAr]	2300,00	VAr
			P_L1_H1	-2300,00	1009,2 [W]	2300,00	W
			Q_L1_H1	-2300,00	551,3 [VAr]	2300,00	VAr
			S_L1_H1	-2300,00	1150,0 [VA]	2300,00	VA
			U_rms_L1	0,00	162,63 [V]	230,00	V
			I_rms_L1	0,00	7,071 [A]	10,00	A
			U_rm_L1	0,00	146,42 [V]	230,00	V
			I_rm_L1	0,00	6,366 [A]	10,00	A
			U_L1_H1	0,00	162,63 [V]	230,00	V
			I_L1_H1	0,00	7,071 [A]	10,00	A
			cos_phi_L1_H1	-1,00	0,8776 [-]	1,00	-
			phi_L1_H1	-180,00	28,65 [deg.]	180,00	deg.
			U_L1_H	0,00		230,00	V
			I_L1_H	0,00		230,00	A

When Current channels are disabled, current channels are not created (e.g. above only U_... channels).

To basic channels for Even Harmonics emphasized channels are added.



+	Used	C	Name	Min	Value	Max	Unit
◢	Used		Power grid analysis		Math 13 (Power grid analysis)		
			Frequency	45,00	50,000 [Hz]	55,00	Hz
			P_L1	-2300,00	1009,2 [W]	2300,00	W
			Q_L1	-2300,00	551,3 [VAr]	2300,00	VAr
			S_L1	-2300,00	1150,0 [VA]	2300,00	VA
			PF_L1	-1,00	0,8776 [-]	1,00	-
			D_L1	-2300,00	0,0 [VAr]	2300,00	VAr
			QH_L1	-2300,00	551,3 [VAr]	2300,00	VAr
			DH_L1	-2300,00	0,1 [VAr]	2300,00	VAr
			P_L1_H1	-2300,00	1009,2 [W]	2300,00	W
			Q_L1_H1	-2300,00	551,3 [VAr]	2300,00	VAr
			S_L1_H1	-2300,00	1150,0 [VA]	2300,00	VA
			U_rms_L1	0,00	162,63 [V]	230,00	V
			I_rms_L1	0,00	7,071 [A]	10,00	A
			U_rm_L1	0,00	146,42 [V]	230,00	V
			I_rm_L1	0,00	6,366 [A]	10,00	A
			U_L1_H1	0,00	162,63 [V]	230,00	V
			I_L1_H1	0,00	7,071 [A]	10,00	A
			cos_phi_L1_H1	-1,00	0,8776 [-]	1,00	-
			phi_L1_H1	-180,00	28,65 [deg.]	180,00	deg.
			U_L1_H	0,00	__	230,00	V
			I_L1_H	0,00	__	230,00	A

In above example odd channels are not created (except ..._H1 -> see note above).

Additional harmonics output channels

When phase angles, P,Q, Impedance and Interharmonics option is selected, Dewesoft create following additional harmonics output channels if:

a) phase anglesoption is checked

To basic channels for All Harmonics (see above left) emphasized phase angles pfi channels for each harmonic are added:

For Even Harmonics odd channels are not created (except …_H1 -> see note above).

b) P,Q option is checked

To basic channels for All Harmonics (see above left) emphasized P - active power and Q - reactive power channels for each harmonic are added:

For Even Harmonics odd channels are not created (except …_H1 -> see note above).

c) Impedanceoption is checked

To basic channels for All Harmonics (see above left) emphasized Impedance Z channels are added

For Even Harmonics odd channels are not created.

d) Interharmonicsoption is checked

To basic channels for All Harmonics (see above left) emphasized Interharmonics …_IH channels are added

For Even Harmonics odd channels are not created.

THD outputs

▫			THD_U_L1	0,00	0,00	[%]	100,00	%
▫			THDEven_U_L1	0,00	0,00	[%]	100,00	%
▫			THDOdd_U_L1	0,00	0,00	[%]	100,00	%
▫			THD_I_L1	0,00	0,00	[%]	100,00	%
▫			THDEven_I_L1	0,00	0,00	[%]	100,00	%
▫			THDOdd_I_L1	0,00	0,00	[%]	100,00	%
▫			K_U_L1	0,00	0,00	[%]	100,00	%
▫			K_I_L1	0,00	0,00	[%]	100,00	%

To basic channels emphasized THD channels are added (not only for voltage and current but also their odd and even harmonics.

Period value outputs

To create period values output channels, Periods option must be checked.

To basic channels emphasized period values ...per channels are added (not only for voltage and current but also for all powers: S - apparent, P - active, Q - reactive).

			U_L1per	0,00	162,63	[V]	230,00	V
			I_L1per	0,00	7,071	[A]	10,00	A
			P_L1per	0,00	1009,2	[W]	2300,00	W
			Q_L1per	0,00	551,3	[VAr]	2300,00	VAr
			S_L1per	0,00	1150,0	[VA]	2300,00	VA
			PF_L1per	0,00	0,8776	[-]	1,00	-

Flicker output

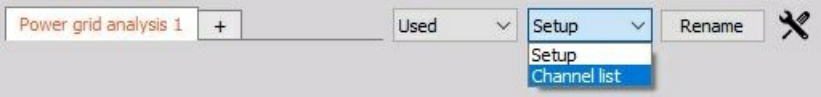
To create flicker output channels, Voltage flicker and Flicker emission option must be checked.

To basic channels emphasized flicker (powers) P... channels are added.

		PF5_L1	0,00	0,03 [-]	100,00	-
		Pst_L1	0,00	0,0000 [-] No data	5,00	-
		Plt_L1	0,00	0,0000 [-] No data	5,00	-
		Synth_U1	-460,00	-325,27 / 325,27 [V]	460,00	V
		I_PF5_L1	0,00	0,000	10,00	
		I_Pst_L1	0,00	0,000 No data	10,00	
		I_Plt_L1	0,00	0,000 No data	10,00	

3-PHASE OUTPUTS

To display all channels go to second tab (Setup) in edit module section and select the Channel list from the drop down menu.



NOTE: **Following description and examples are valid for all types of 3 phase systems: 3-phase star, 3-phase Aron, 3-phase delta and 3-phase V.**

Dewesoft automatically creates following Basic output channels

Current channels choice = disabled

Calculate line voltages choice = disabled

+	Used	C	Name	Min	Value	Max	Unit
◢	Used		Power grid analysis		Math 14 (Power grid analysis)		
			Frequency	45,00	50,000 [Hz]	55,00	Hz
			U_rms_L1	0,00	162,63 [V]	230,00	V
			U_rm_L1	0,00	146,42 [V]	230,00	V
			U_L1_H1	0,00	162,63 [V]	230,00	V
			U_rms_L2	0,00	162,63 [V]	230,00	V
			U_rm_L2	0,00	146,42 [V]	230,00	V
			U_L2_H1	0,00	162,63 [V]	230,00	V
			U_rms_L3	0,00	162,63 [V]	230,00	V
			U_rm_L3	0,00	146,42 [V]	230,00	V
			U_L3_H1	0,00	162,63 [V]	230,00	V

To ‘Used’ analog channels voltage (U_Ln_H1 and U_Ln_rms) and frequency channels are added.
Current channels choice = disabled

Calculate line voltages choice = enabled

+	Used	C	Name	Min	Value	Max	Unit
◢	Used		Power grid analysis		Math 14 (Power grid analysis)		
			Frequency	45,00	50,000 [Hz]	55,00	Hz
			U_rms_L1	0,00	162,63 [V]	230,00	V
			U_rms_L12	0,00	282,15 [V]	230,00	V
			U_rm_L1	0,00	146,42 [V]	230,00	V
			U_rm_L12	0,00	253,61 [V]	230,00	V
			U_L1_H1	0,00	162,63 [V]	230,00	V
			U_L12_H1	0,00	282,15 [V]	230,00	V
			U_rms_L2	0,00	162,63 [V]	230,00	V
			U_rms_L23	0,00	282,15 [V]	230,00	V
			U_rm_L2	0,00	146,42 [V]	230,00	V
			U_rm_L23	0,00	253,61 [V]	230,00	V
			U_L2_H1	0,00	162,63 [V]	230,00	V
			U_L23_H1	0,00	282,15 [V]	230,00	V
			U_rms_L3	0,00	162,63 [V]	230,00	V
			U_rms_L31	0,00	280,77 [V]	230,00	V
			U_rm_L3	0,00	146,42 [V]	230,00	V
			U_rm_L31	0,00	253,61 [V]	230,00	V
			U_L3_H1	0,00	162,63 [V]	230,00	V
			U_L31_H1	0,00	280,77 [V]	230,00	V

To basic channels (see on left) emphasized channels are added -> between lines voltages channels.

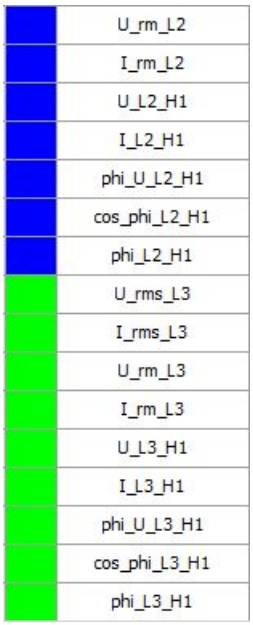
Current channels choice = enabled

Calculate line voltages choice = disabled

	Frequency
	P_L1
	Q_L1
	S_L1
	PF_L1
	D_L1
	QH_L1
	DH_L1
	P_L1_H1
	Q_L1_H1
	S_L1_H1
	P_L2
	Q_L2
	S_L2
	PF_L2
	D_L2
	QH_L2

	DH_L2
	P_L2_H1
	Q_L2_H1
	S_L2_H1
	P_L3
	Q_L3
	S_L3
	PF_L3
	D_L3
	QH_L3
	DH_L3
	P_L3_H1
	Q_L3_H1
	S_L3_H1
	P
	Q
	S

	PF
	D
	QH
	DH
	P_H1
	Q_H1
	S_H1
	U_rms_L1
	I_rms_L1
	U_rm_L1
	I_rm_L1
	U_L1_H1
	I_L1_H1
	cos_phi_L1_H1
	phi_L1_H1
	U_rms_L2
	I_rms_L2



To basic channels current and power channels are added.

When you compare channels on above picture with created channels for Single phase, you can see that for 3 Phase systems only the same kind channels for second - L2 and 3rd - L3 phase line are added.

NOTE: **H1 harmonics channels are created anyway for: voltage and current channels (U and I), power channels (P, Q, S, phi, cos phi).**

Harmonics output channels -> Harmonics option is checked

For information about this option settings and output channels see -> Single phase Harmonics option.

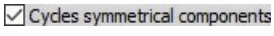
NOTE: **Additional to Single phase Harmonics channels only the same kind output channels for second - L2 and 3rd - L3 phase line are created.**

THD output channels -> THD option is checked

For information about this option settings and output channels see -> Single phase THD option.
NOTE: **Additional to Single phase THD channels only the same kind output channels for second - L2 and 3rd - L3phase line are created.**

Symmetrical components -> Symmetrical components option is checked

We can calculate Symmetrical components by selecting the following check box.



Symmetrical components are available only on three phase systems and tell us the unbalance of

the grid.

		u2	0,00	0,33	[%]	100,00	%
		u0	0,00	0,32	[%]	100,00	%
		u2_1	0,00	0,32	[%]	100,00	%
		u0_1	0,00	0,32	[%]	100,00	%
		U_0	0,00	0,53	[V]	230,00	V
		U_1	0,00	162,64	[V]	230,00	V
		U_2	0,00	0,53	[V]	230,00	V
		i2	0,00	0,33	[%]	100,00	%
		i0	0,00	0,32	[%]	100,00	%
		i2_1	0,00	0,32	[%]	100,00	%
		i0_1	0,00	0,32	[%]	100,00	%
		I_0	0,00	0,023	[A]	10,00	A
		I_1	0,00	7,071	[A]	10,00	A
		I_2	0,00	0,023	[A]	10,00	A

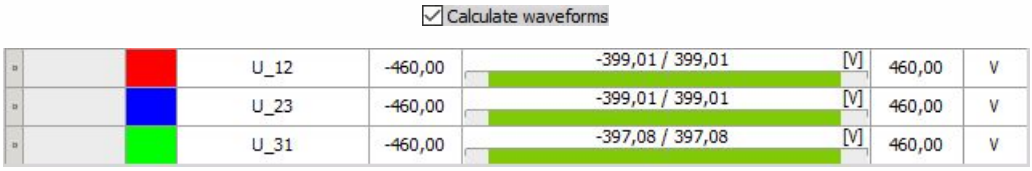
Period values output channels -> Period values option is checked

For information about this option settings and output channels see -> Single phase Period values option.

NOTE: **Additional to Single phase Period values channels only the same kind output channels for second - L2 and 3rd - L3 phase line are created.**

Waveforms -> Waveforms option is checked

If we measure line to ground voltages, we can calculate Waveforms for the line to line voltages by selecting the following option:



Flicker output channels -> Flicker option is checked

For information about this option settings and output channels see -> Single phase Flicker option.

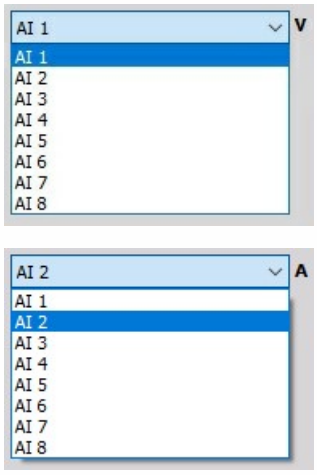
NOTE: **Additional to Single phase Flicker channels only the same kind output channels for second - L2 and 3rd - L3 phase line are created.**

WIRING

On this left lower part of the Dewesoft Power Setup screen principle module wiring scheme with input channels selection fields are displayed.

Input channels selection

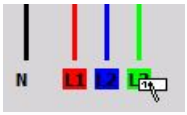
Dewesoft provides module wiring scheme with needed input channels selection fields for selected type of power module. All Inputs voltage and current channels can be selected from appropriate drop down list:



Wiring scheme colors

To change the color of the phase in the scheme and at the same time in the Vector plot, simply click on desired Lx and select the color from the opened window.

Note that N can’t be changed and is set to default BLACK.

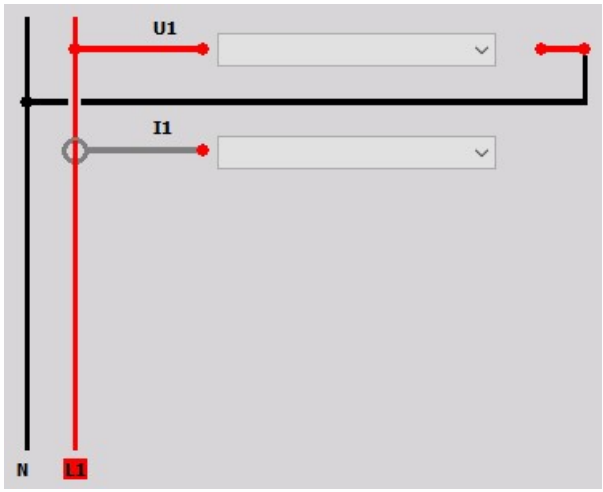


Module wiring scheme

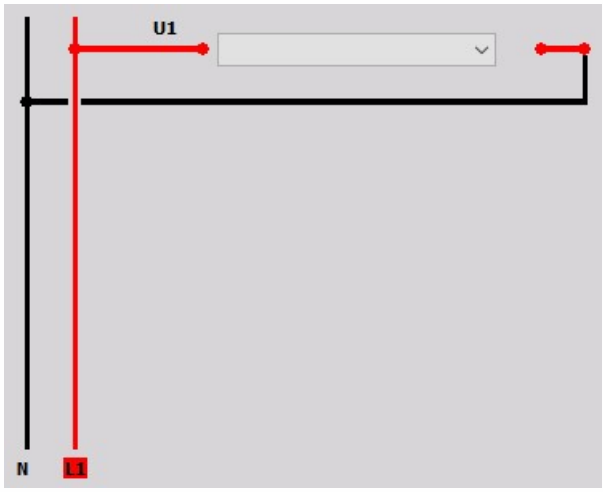
This scheme depends from selection in Power system configuration and Current channels field:

- **Single phase**

Current channels choice = enabled (default)

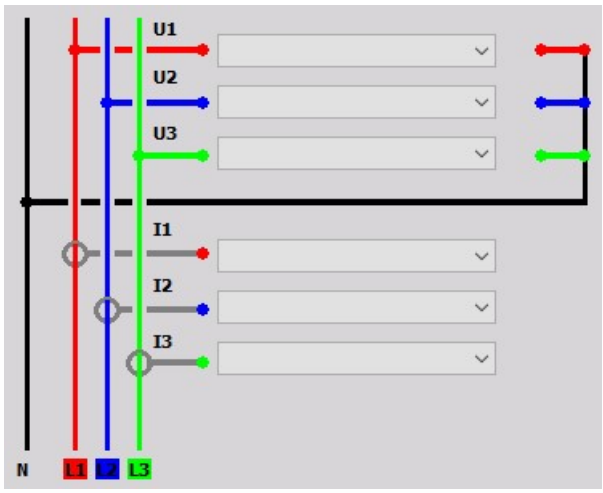


Current channels choice = disabled

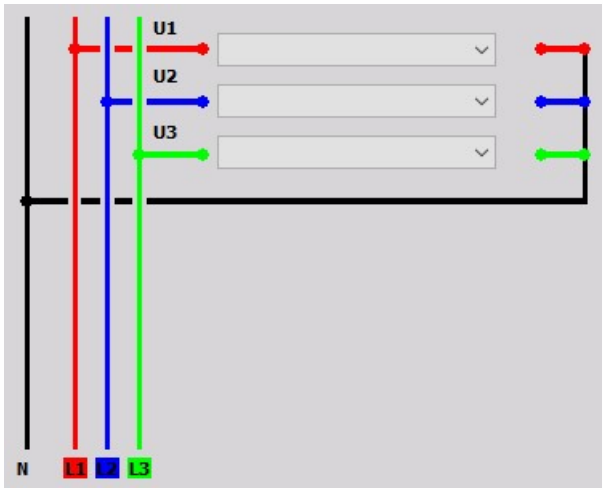


- **3 - Phase star**

Current channels choice = enabled (default)

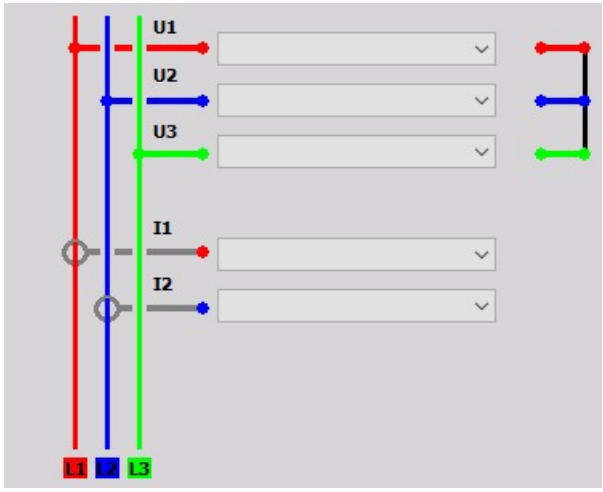


Current channels choice = disabled

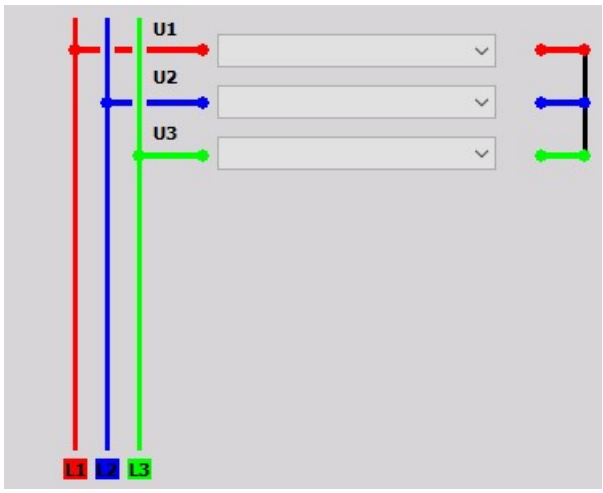


- **3 - phase Aron**

Current channels choice = enabled (default)

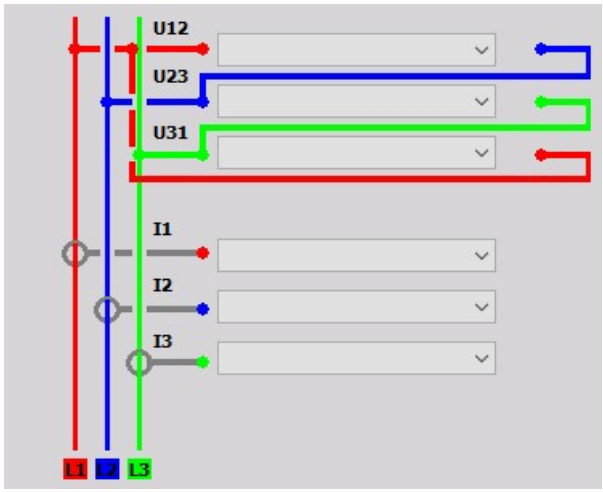


Current channels choice = disabled

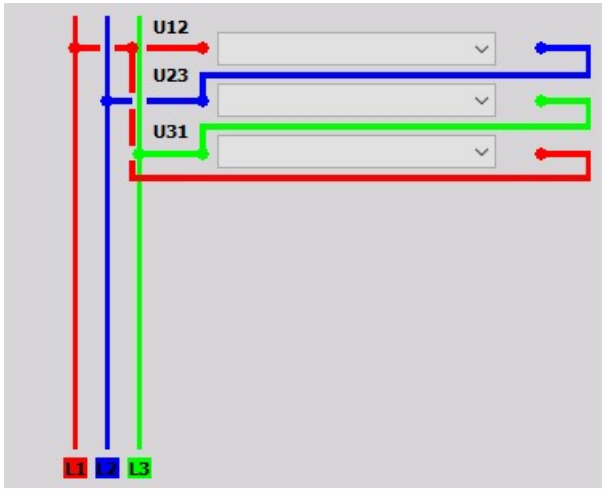


- **3 - phase delta**

Current channels choice = enabled (default)

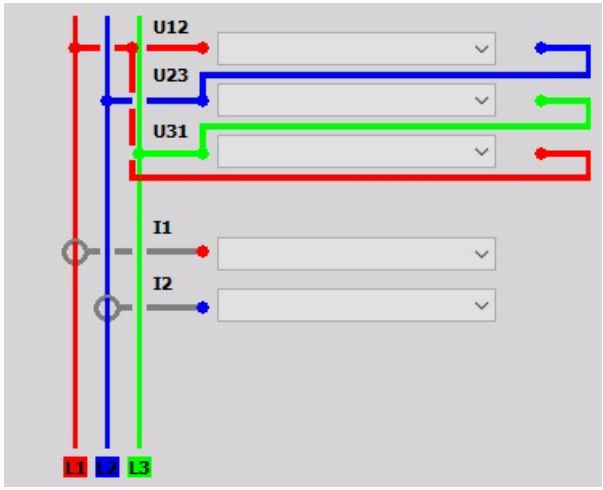


Current channels choice = disabled

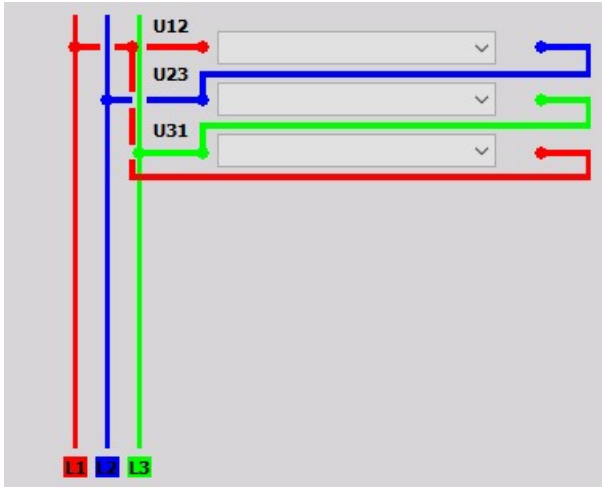


- **3 - phase V**

Current channels choice = enabled (default)

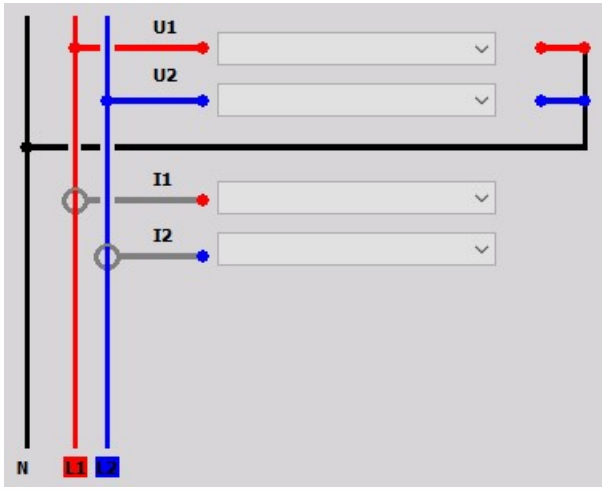


Current channels choice = disabled

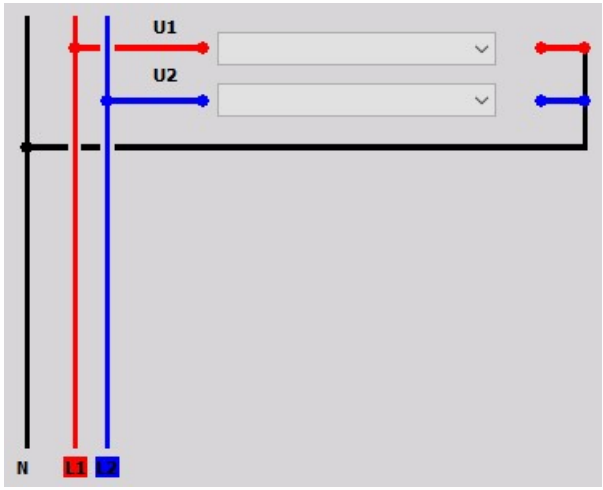


- **2 - phase**

Current channels choice = enabled (default)



Current channels choice = disabled

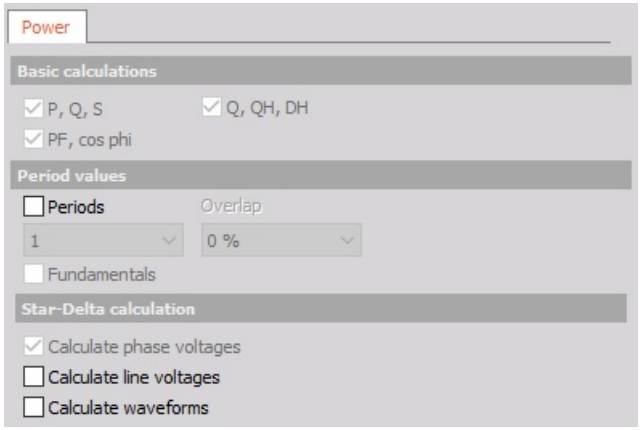


CALCULATIONS

In the calculations part of setup we can choose between:

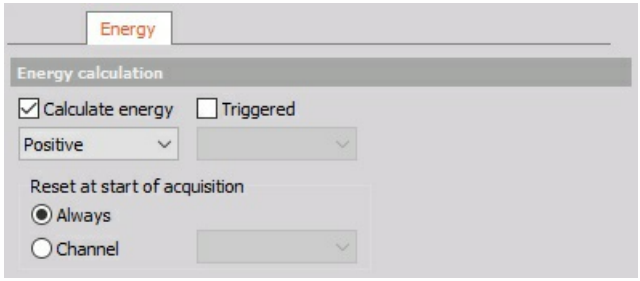
Power

- Basic calculations
- Period values
- Star-Delta calculation



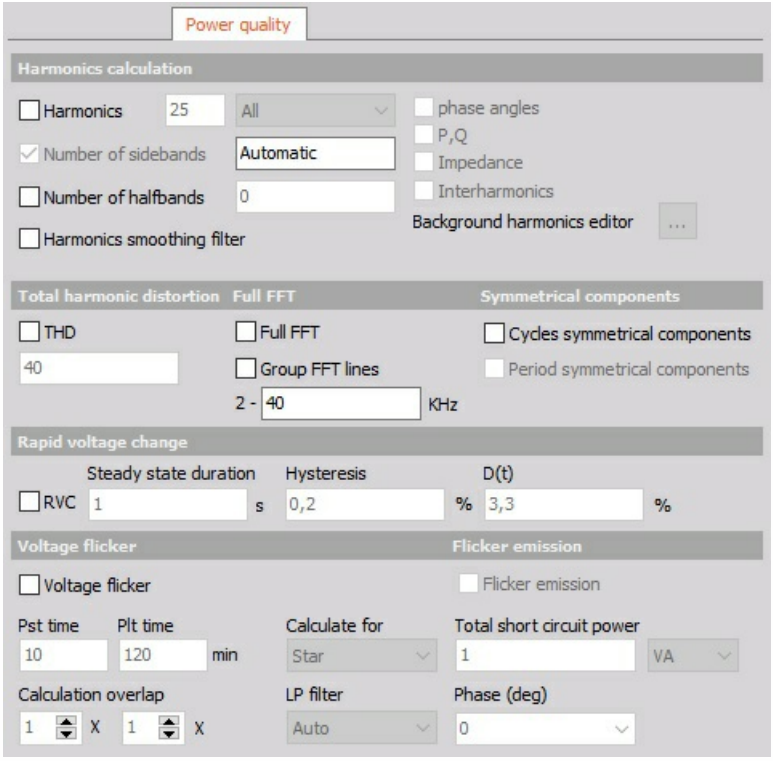
Energy

- Energy calculation

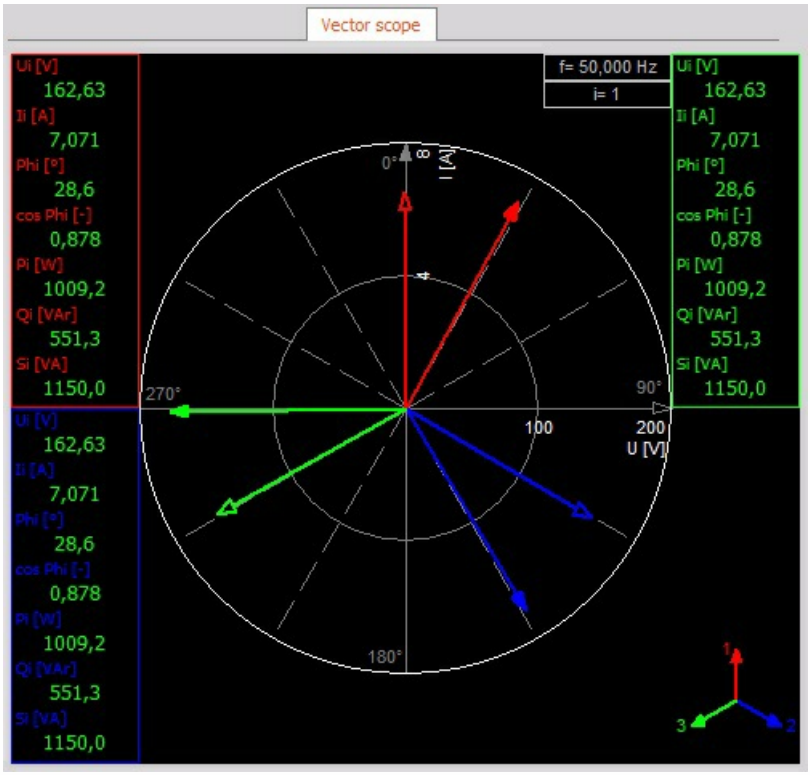


Power quality

- Harmonics calculation
- Total harmonic distortion
- Full FFT
- Symmetrical components
- Rapid voltage change
- Voltage flicker
- Flicker emission



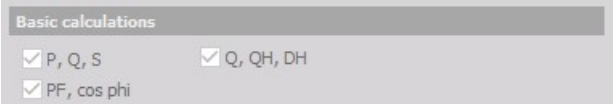
Vector scope



NOTE: **Dimmed options are unavailable for selected type of measurement.**

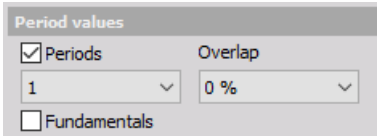
POWER

Basic calculations



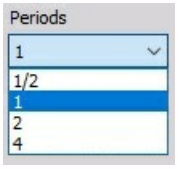
Basic calculations are always calculated and displayed in Channel list.

Period values

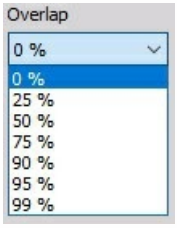


We can calculate Period values = values for voltage, current and power for:

- **Periods** = each = 1, 1/2, 2 or 4 period can be selected from drop down list



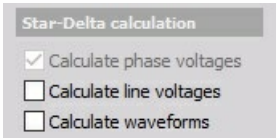
- **Overlap** = 0%, 25%, 50% or 75%, 90%, 95%, 99% which can be selected from drop down list



This is very helpful for triggering.

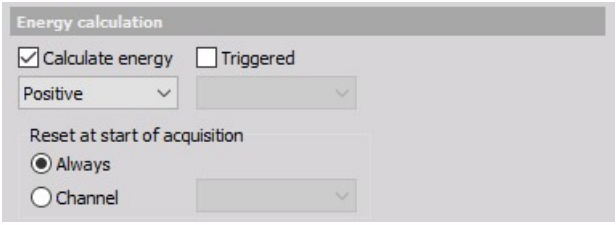
Star-Delta calculation

- To calculate line voltage check Calculate line voltage option.
- If we measure line to ground voltages, we can calculate Waveforms for the line to line voltages by selecting Calculate waveforms option.

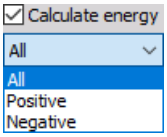


ENERGY

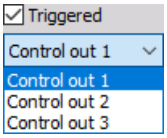
Energy calculation



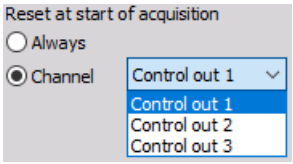
- **Calculate enery** - In Dewesoft the energy can automatically be calculated using the power module. It's possible to calculate the total energy consumption or you can even split up into positive (energy consumption) and negative energy (energy delivery).



- **Triggered** - In addition to the basic power calculation it is also possible to start/reset the energy calculation at trigger events. Otherwise, the energy will be calculated as soon as you switch into measure mode.
  NOTE: **A User input channel is required!**

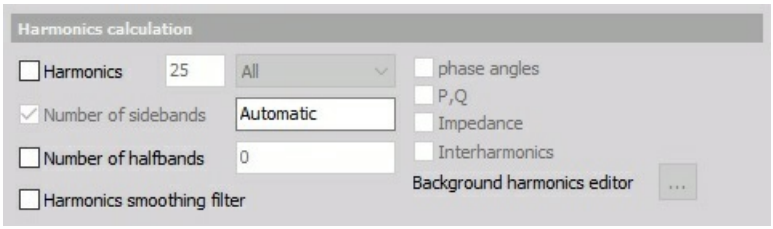


- **Reset at start of acquisition** - This is an option that enables us to reset the measurement in addition to start on acquisition, also later by triggering a User input channel.



POWER QUALITY

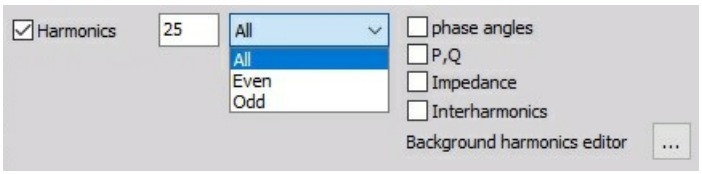
Harmonics calculation



Harmonics

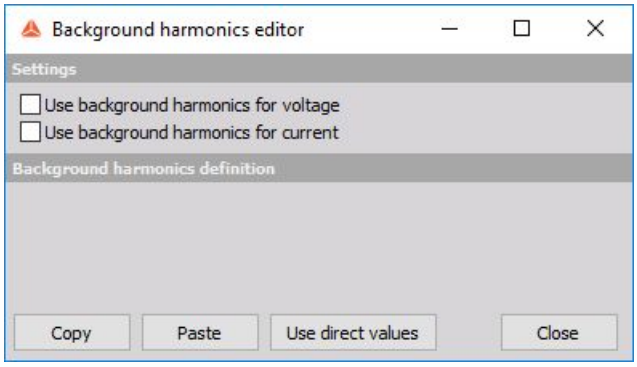
The line voltage isn't perfect and can have distortions, which are nicely shown as harmonics in the frequency spectrum.

When the Harmonics option is checked, like in sample, we can:

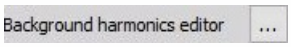


- define number of harmonics (with enter value in this field)
- select All or only Even or Odd harmonics from drop down list
- select additional harmonics option: phase angles, P,Q, Impedance, Interharmonics.

Background harmonics



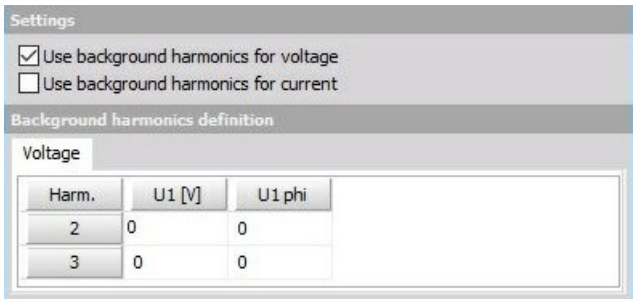
On upper picture you can see additional button, which appeared by checking Harmonics option.



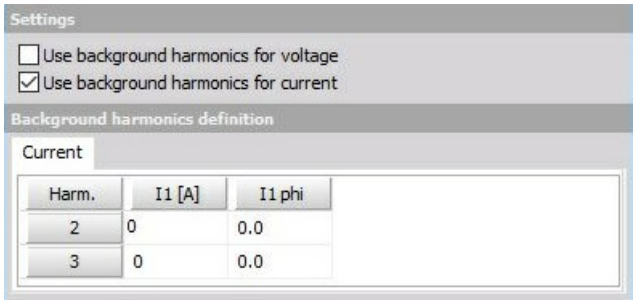
The background harmonics are used to compensate normal load and to see the difference from this normal conditions.

When we open the editor two options can be selected to use and define background harmonics:

- Use background harmonics for voltage



- Use background harmonics for current

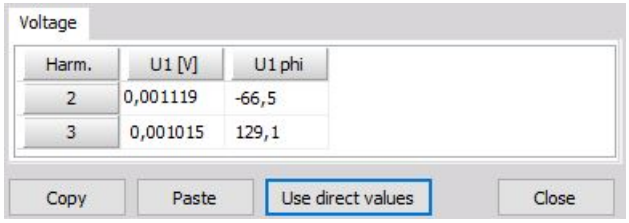


When one from this options (or both) is checked, table with voltage (U1 in V unit and U1 phi - phase angle) / current values (I1 in A unit and I1 phi - phase angle) for defined number - 1 background harmonics appear.

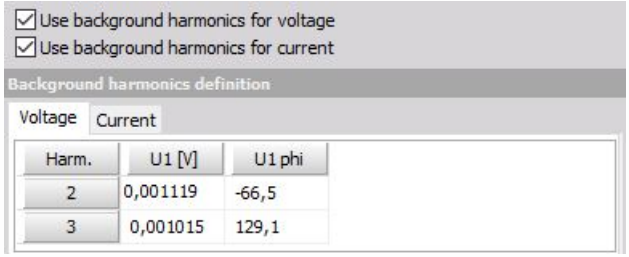
In example above we have selected 3 harmonics, therefore voltage / current value for second and 3rd harmonics can be defined.

Table Voltage / current values can be defined with:

- selecting cell and enter value
- Copy and Paste function
- selecting Use direct values button to enter current value in table cells



When both options for background harmonics are checked, table with current values appear in separate tab.



Other harmonic calculations

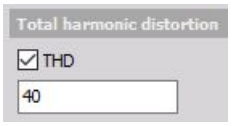
In Harmonics calculation we can make additional calculations like:

- **Number of sidebands** - The basic idea of sidebands is that a certain frequency range is considered as one harmonic.
- **Number of halfbands** - The IEC 61000-4-7 requires for the grouping of the harmonics sidebands where only the square root of the quadratic half of them should be added. This is required for the lowest and highest line and is defined as halfbands in Dewesoft.
- **Harmonics smoothing filter** - This options enables the low-pass filter which is required in IEC 61000-4-7 standard.

Total harmonic distortion

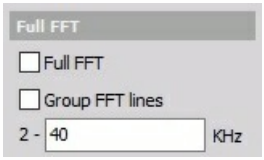
We can calculate THD = total harmonic distortion (sum of all harmonic values).

To create THD output channels this option must be checked. In field on right value the number of harmonics for THD can be entered (40 = default).



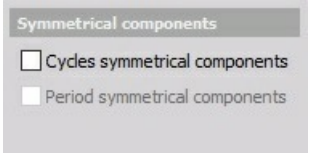
Full FFT

This options calculates the Full FFT which can be exported to the database and displayed via a 2Dgraph.



Symmetrical components

Symmetrical components are available only on three phase systems and tell us the unbalance of the grid.

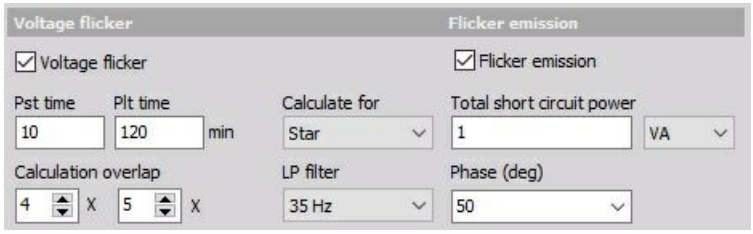


Rapid voltage change

The Rapid Voltage Changes are parameters which are added as a supplement to the flicker standard. Rapid Voltage Changes describe all voltage changes which are changing the voltage for more than 3% at a certain time interval. These voltage changes can afterward's be analyzed with different parameters (depth of voltage change, duration, steady state deviation, etc.).

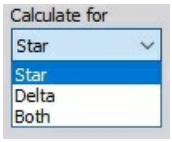
Voltage flicker and Flicker emission

The flicker is actually power quality parameters measuring low frequency distortions of the voltage.



For this option we can set:

- **Pst time** - Short calculation time interval, defined in minutes
- **Plt time** - Long calculation time interval, defined in minutes
- **Calculation overlap** - multiplier of calculation overlap for Pst time and Plt time; with up arrow beside value field we can increase and with down arrow we can decrease multiplier value
- **Calculate for** - from drop down list we can select type of calculation - for Star or Delta or Both:



This function is only available if calculations Calculate line voltage and Calculate waveforms in Power / Star-Delta calculations are used!

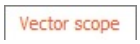
- **LP filter** - 6 th order low pass Butterworth filter's cutoff frequency is auto selected according to standard, or manually selectable (35 Hz or 42 Hz)

Flicker emission calculates the influence of the current on the flicker (contribution of the device being measured on the flicker).

We need to enter impedance in Ohm and Phase as two additional parameters for calculation.

VECTOR SCOPE

When Vector scope button is selected the vector scope instrument is displayed to check if everything is connected correctly.

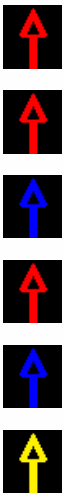


The Vector scope shows the phasors of voltage and current. In addition also the RMS values of voltage and current, the phase angles and the power values are shown. It is also possible to show the phasors for three phase systems.

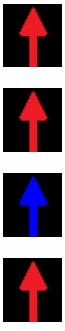
Phasors for voltage and current:

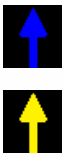
- 1 phase
- 2 phase
- 3 phase
- L1
- L1
- L2
- L1
- L2
- L3

Voltage:

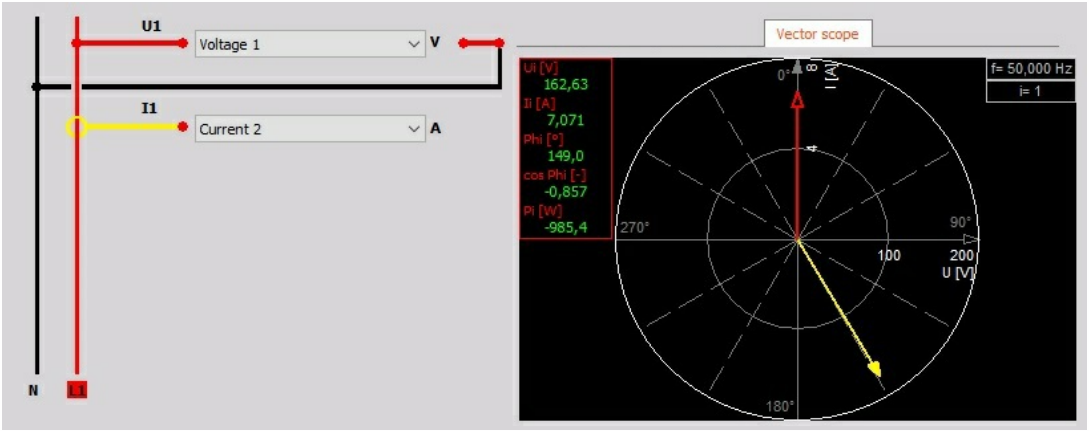


Current:



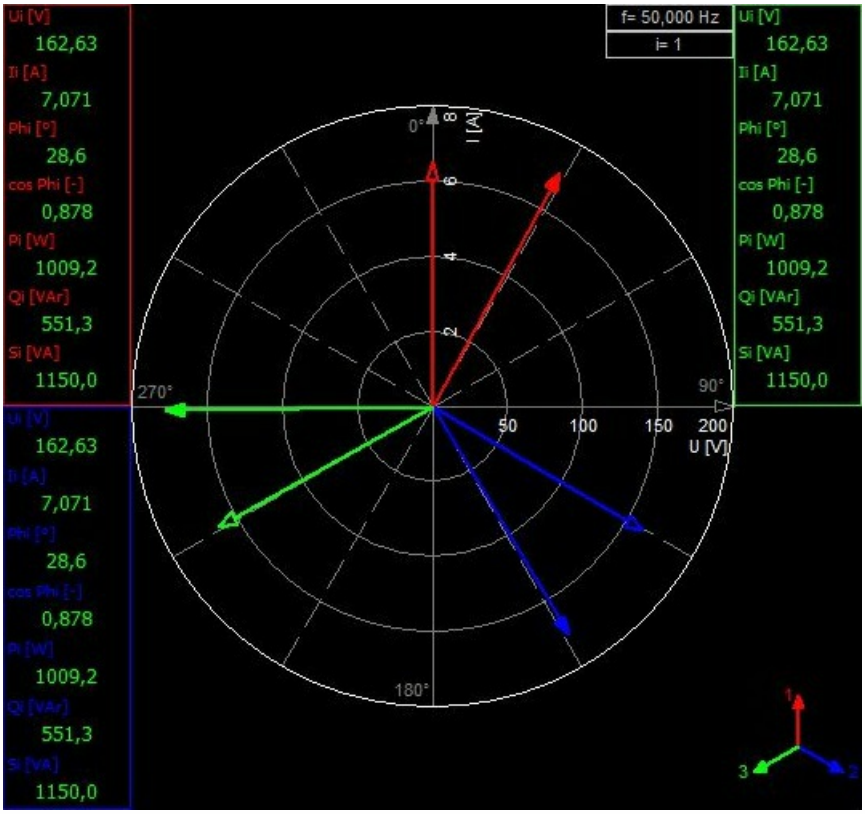


Example for Single phase system:



Only for this example the current of I1 is colored YELLOW. Normally Ux & Ix of Lx are the same color.

Principle example for Three phase system (each single phase line is displayed in different color):



CURRENT MEASUREMENT

The current measurements can be divided in two categories:

- high currents with two principles Rogowsky coil or current clamps

  Rogowsky coil can be used for AC current measurements. Directly it measures the derivative of current, therefore an integrator circuit or software filter module must be used.

  Current clamp works on the Hall effect principle and it outputs the voltage proportional to current.

- low current measurements is often done with shunt resistors, where we need to cut the wire to include the shunt in series. We have to be very careful not to exceed maximum current of the shunt.

Both principles include a phase shift of the output.

Scale the current clamps

This can be done from Dewesoft Setup → Analog tab Channel setup for the current measurement used channels.

For information about Calibration analog input channel see → Channel setup - Calibration.

NOTE: **First, as usual, we enter the 'Units of measurement'. In the channel setup we enter that we measure current I in unit A. It is very important that the measurement range is chosen according to the expected signal. If we choose too high range, the inaccuracy of the current clamps will be too high to make correct readings.**

Scale the shunt resistor

Module setup can be done from Dewesoft Setup → Analog tab Channel setup.

For information about Modules shunt resistor settings see → Channel setup - Module.

Sensor correction

When we are using the current clamps, we have amplitude and phase errors. So the current clamp is in this case the main source of calculation errors. Obviously we need to tell Dewesoft that we have a sensor which is not perfect and enter somehow the transfer curve of our sensor. Transfer curve gives information about amplitude and phase for sensors at certain frequencies and from this information Dewesoft can compensate the errors.

When we go back to analog setup after saving the sensors with Save file button and then choose the sensor for current channel, open the Sensors tab and choose in Sensor field the serial number of the sensor previously entered in the sensor editor we can't enter the normal scaling or sensitivity anymore. However we do have a chance to enter a Transducer scale, which we can use for reversing the polarity of the sensor by entering a value of -1.

For the next setup we don't have to define a sensor anymore, but can just choose it from the sensor list.

VEHICLE PERFORMANCE TEST

Enter topic text here.

BRAKE TEST

Enter topic text here.

STRAIN, STRESS

NOTE: **In Strain & stress it is important to correctly setup the Analog IN.**

Introduction

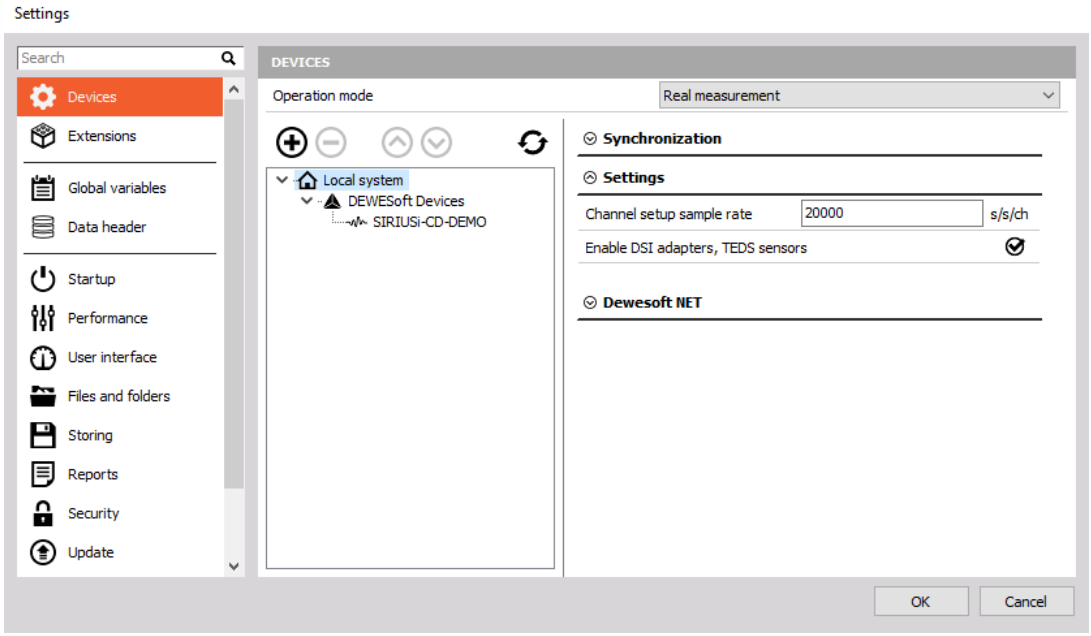
In this section we will cover how to setup the bridge operations like:

- Balance
- Reset
- Short
- Shunt
- Shunt cal check
- Compensate

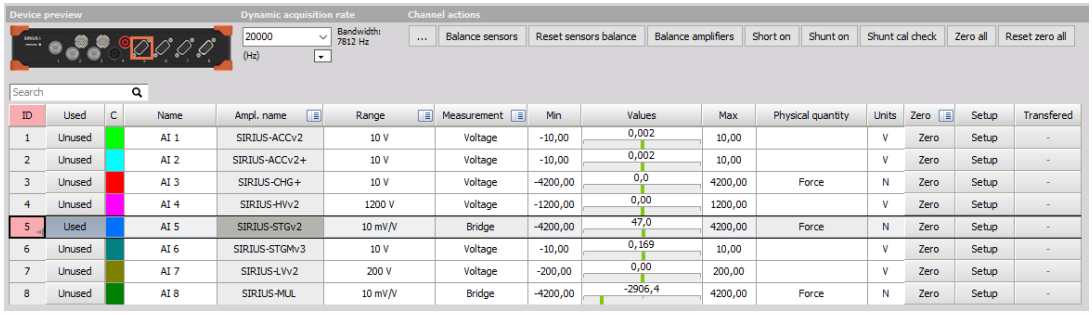
It also explains bridge sensors with TEDS interface.

Set up the sensor (with TEDS)

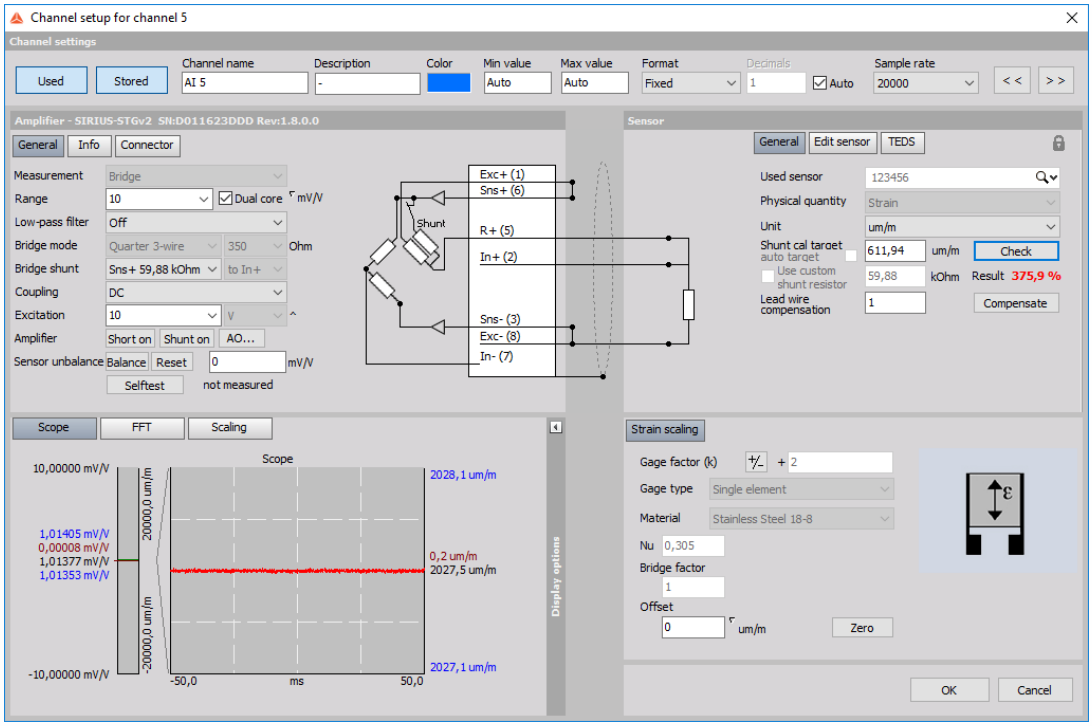
For using the benefits of automatic sensor detection (automatic reading of scaling, serial number, ...) please activate the check-box "DSI adapters/TEDS sensors" in DEWESoft -> Settings -> Hardware Setup:



Start DEWESoft; here the sensor with TEDS (a tuning fork with strain gauge applied + TEDS chip inside sensor connector) is plugged into channel 5:



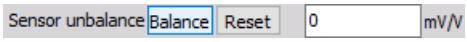
All the settings like "Measurement Type", "Bridge mode", "Excitation" and "Scaling" are read out and set automatically. Note the grayed, locked fields, which cannot be changed (at first):


Channel setup for channel 5
Channel settings
Used
Stored
Channel name
AI 5
Description
-
Color
Min value
Auto
Max value
Auto
Format
Fixed
Decimals
1
Auto
Sample rate
20000
< <
> >
Amplifier - SIRIUS-STGv2 SN:D011623DDD Rev:1.8.0.0
General
Info
Connector
Measurement
Bridge
Range
10
Dual core
mV/V
Low-pass filter
Off
Bridge mode
Quarter 3-wire
350
Ohm
Bridge shunt
Sns+ 59,88 kOhm
to In+
Coupling
DC
Excitation
10
V
Amplifier
Short on
Shunt on
AO...
Sensor unbalance
Balance
Reset
0
mV/V
Selftest
not measured
Exc+ (1)
Sns+ (6)
Shunt
R+ (5)
In+ (2)
Sns- (3)
Exc- (8)
In- (7)
Sensor
General
Edit sensor
TEDS
Used sensor
123456
Physical quantity
Strain
Unit
um/m
Shunt cal target
auto target
611,94
um/m
Check
Use custom
shunt resistor
59,88
kOhm
Result
375,9 %
Lead wire
compensation
1
Compensate
Scope
FFT
Scaling
Scope
10,00000 mV/V
1,01405 mV/V
0,00008 mV/V
1,01377 mV/V
1,01353 mV/V
-10,00000 mV/V
20000,0 um/m
-20000,0 um/m
2028,1 um/m
0,2 um/m
2027,5 um/m
2027,1 um/m
-50,0
ms
50,0
Display options
Strain scaling
Gage factor (k)
+/-
+ 2
Gage type
Single element
Material
Stainless Steel 18-8
Nu
0,305
Bridge factor
1
Offset
0
um/m
Zero
OK
Cancel

Balance the bridge

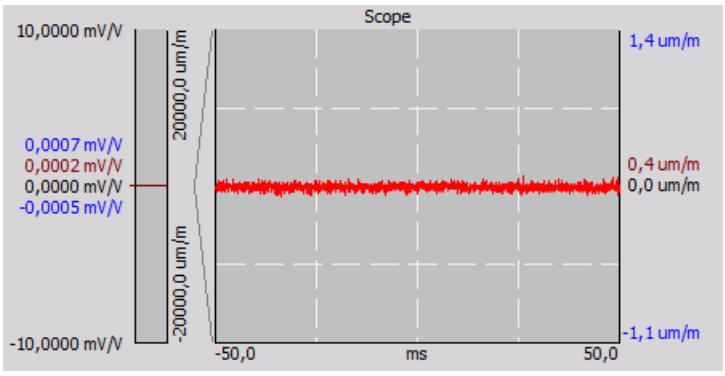
From the screenshot above you see that the current sensor offset (stored in the TEDS) is 0 %. In the scope screen above the scaled value currently read (right side) is 2027 um/m. That is because the strain gauge does not have exactly 350 ohms, which is normal, due to mechanical tolerances.

Click on "Balance" to zero the bridge.



The unbalance will be measured and shown. If it is out of limit, the field will get red. Check the strain gauge. Wrong resistance value?

The input signal is now 0 mV/V.

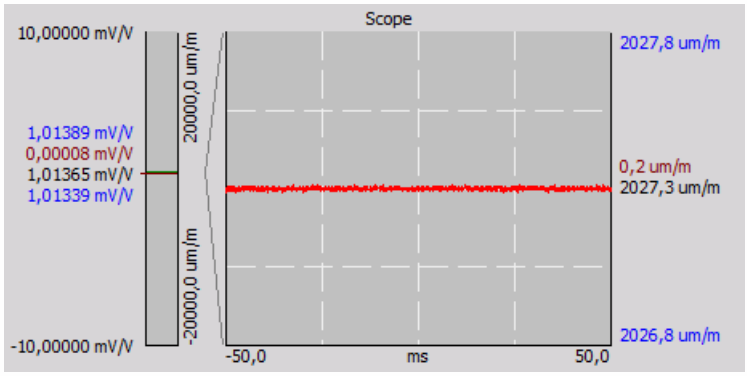


Balance is done. That's all! Now you can START your MEASUREMENT! To perform it on a number of channels at the same time, use the >> Group operations.

Reset the balancing

Click on "Reset". The sensor offset is removed, initial state...

... and you see the unbalance of the sensor again.



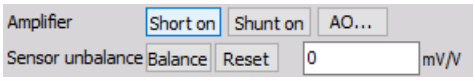
"Balance" and "Reset" are opposite operations.

Short

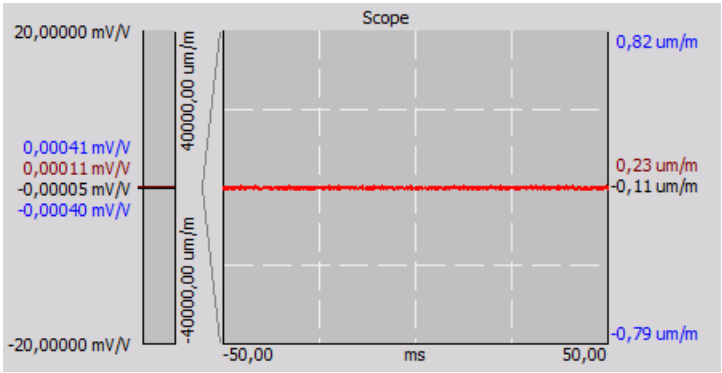
The Short and Shunt operations are mainly thought for checking if the connected strain gauge and the measurement amplifier are OK. When using the Short operation, the pins 2 and 7 (the input pins of the amplifier) are internally shorted and you see the offset of the amplifier.

First perform a "Reset" to cancel the stored offset.

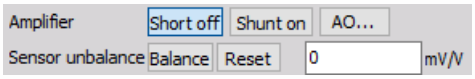
Then click "Short on".



If the output (see left side) is close to 0mV/V, the amplifier is OK.

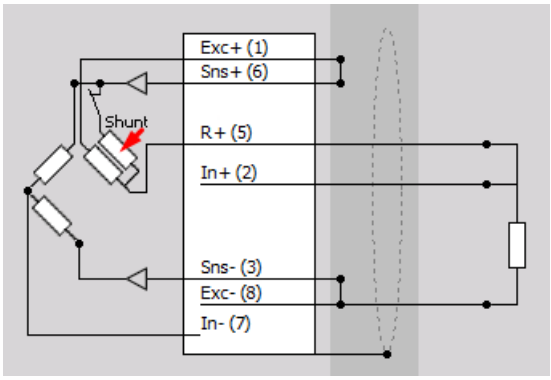


Disable the short again by clicking "Short off".



Shunt

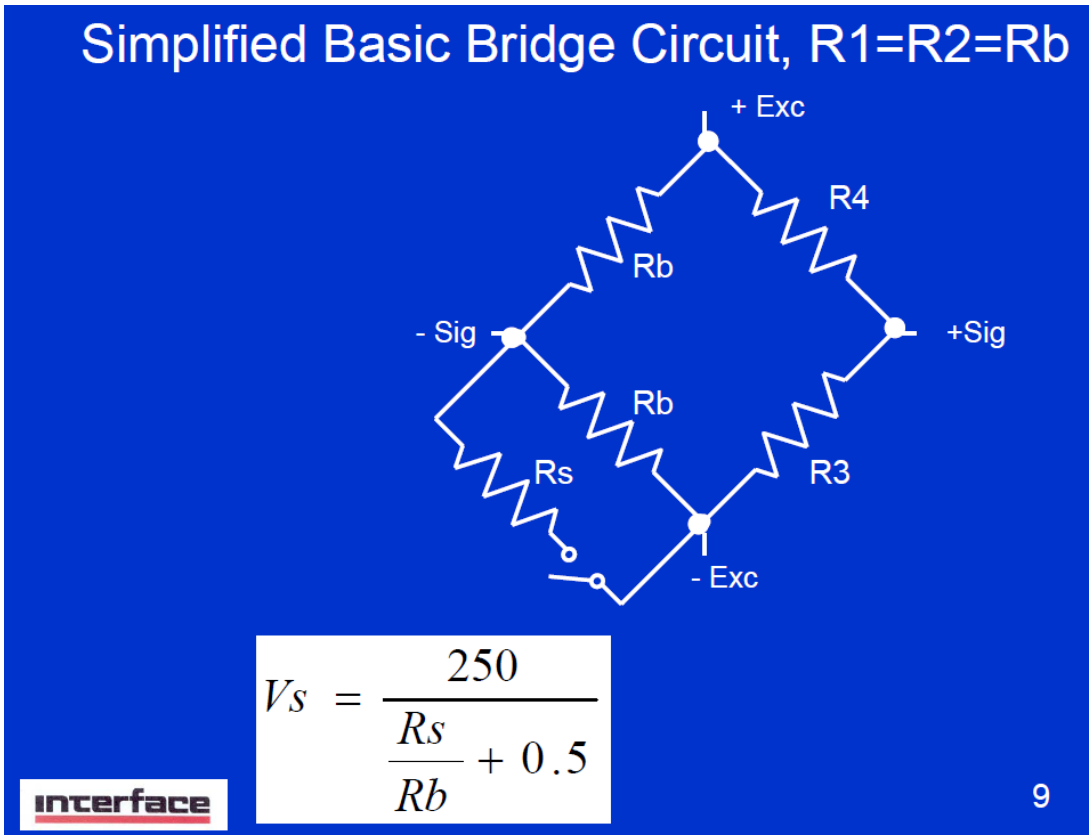
The Shunt operation is thought for checking if the connected strain gauge is OK. From the wiring schematic in DEWESoft X you see, that the amplifier already comes with the integrated shunt resistor.



The idea behind is to "shunt" a resistor of known value parallel to one resistor of the bridge to achieve a known, calculable unbalance. With the "Shunt calibration" we can automatically check the measured value against a predefined one (from sensor database or TEDS).

For the measurement, this internal resistor is disconnected again, of course.

Let's check out the formulas:

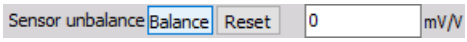


("Shunt calibration for Dummies; a reference guide" by LaVar Clegg, Interface Inc.)

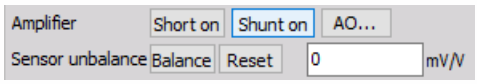
R3 and R4 are part of the bridge completion, internally of the amplifier. The upper Rb is the connected strain gauge, the lower Rb also internally of the amplifier together with the shunt Rs.

With a 350 ohms quarter bridge and an internal shunt resistor of 59k88 SIRIUS STG module) the expected unbalance should be: Vs = 250 /(59880/350 + 0,5) = 1,45699775 mV/V

First click on "Balance", because the formula is only valid on a balanced bridge (both Rb = 350 ohms).



Then click "Shunt on".



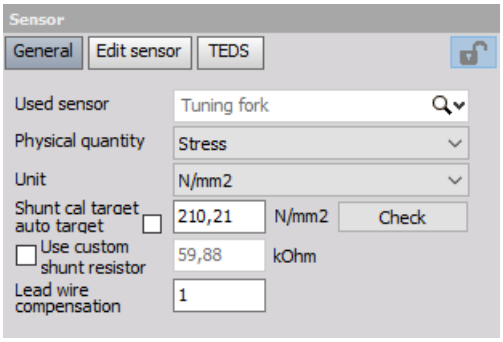
The output value comes very close to the expected value (1,4566 mV/V).

But how big exactly is the error?

Shunt cal check – preparation

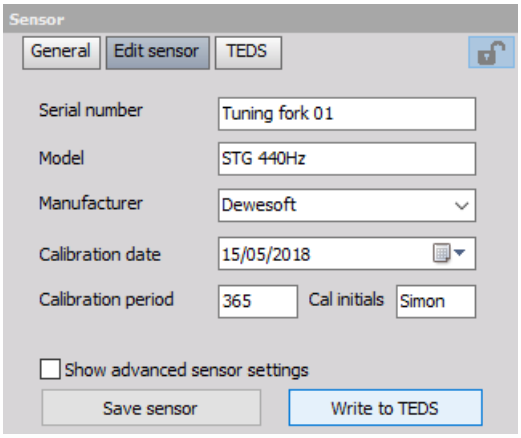
Shunt cal check is done on scaled values, therefore we take a look on the strain scaling. The max input from the graph above is 10 mV/V; the max scaled output signal is 20 000 um/m. So, the scaling factor is 2000. Our target value of 1,45699775 mV/V would equal (x 2000) = 2913,9955 um/m.

In the right upper section of the channel setup window, unlock the TEDS settings (or settings of the sensor database if you don't have TEDS).



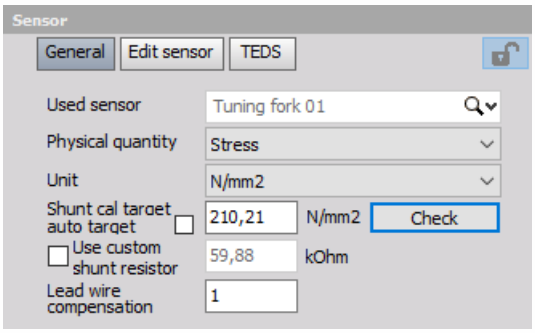
Enter the calculated 2913,9955 um/m in the field "Shunt cal target" and press <Enter>. The resistor is OK with 59,88k.

Then switch to the "Edit sensor" tab and click "Write to TEDS" (or "Save Sensor" if you don't have TEDS). Of course this procedure has only to be done the first time.

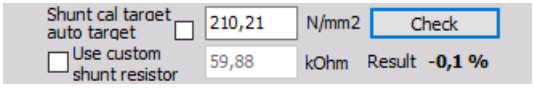


Shunt cal check

Now perform the "Shunt cal check":

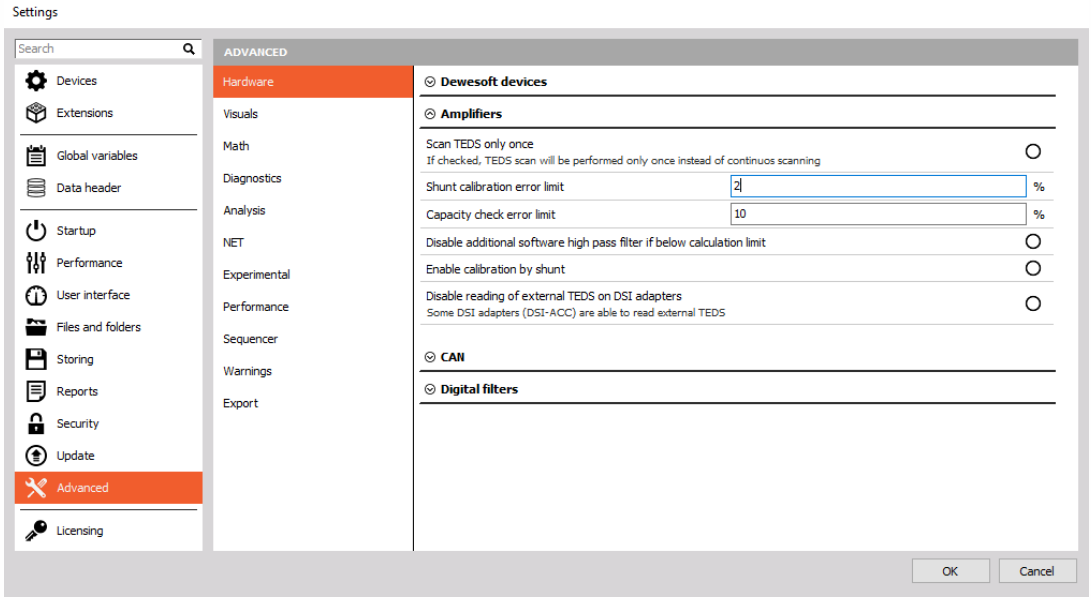


The result is shown, and looks very promising in this case (-0,1 %). Strain gauge is OK.



If the value is bigger than 2%, it will become red.

Adjusting the 2% limit is now possible in DEWESoft: go to Settings -> Advanced -> Hardware -> Amplifiers -> Shunt calibration error limit [%].



Shunt cal check – with custom shunt

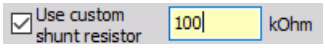
What do you do if the expected shunt cal result is already calculated in the datasheet / calibration sheet of the sensor, but for a different shunt resistance, e.g. 100 k?

You can just use it!

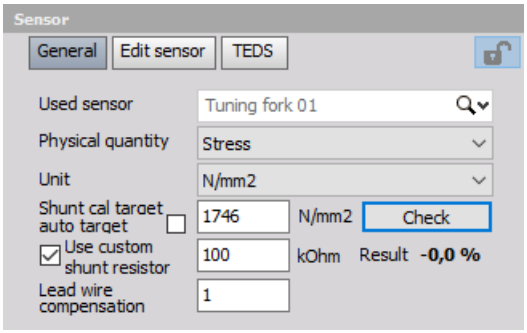
For this example, we take the well known formula from the "Shunt" chapter, and calculate the

shunt result with 100k shunt resistor. Vs = 250 / (100000/350 + 0,5) = 0,873 mV/V, with scaling of 2000 -> 1746,943 um/m.

Enter the values as shown below. When entering the resistor value, the field gets yellow. Please confirm the value with pressing <Enter>.



Then perform the "Check".



As you see, the software automatically takes care about different resistor values. The asterisk (*) symbol after the result indicates that the check has been performed with a different resistor value than physically available.

Lead wire compensation

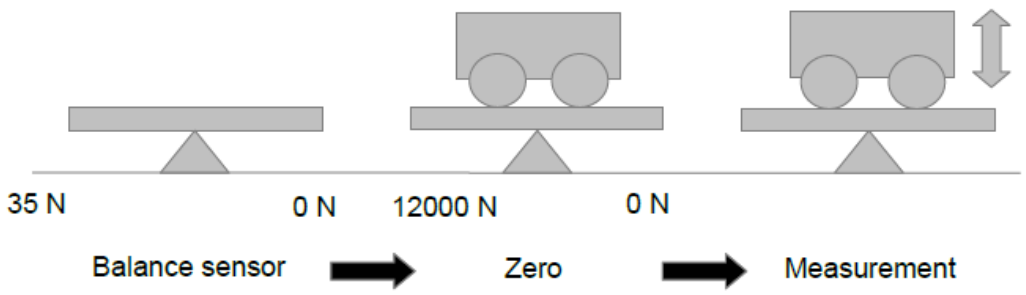
If you have a long cable to the sensor and "quarter bridge 3 wire mode", DEWESoft can also cancel out the wire resistance. (In the other modes the wire loss is canceled already because Sense and Excitation lines are connected directly at the sensor).

Here you see a quarter bridge strain gage with 3-wire connection. In each of the 3 lines we have built in a resistor of 11 ohms for showing the principle. Click the "Compensate" button.

The correction factor is calculated (1,031 in our case) and the measured resistance (10,8 ohms) displayed. (Please take into account, that this function is only available, if the Shunt resistor inside the module is connected to the Exc+ line, which is the default).

Zero

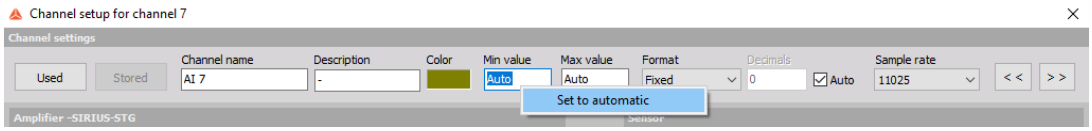
There is also a function called "Zero", let's look at the difference to "Balance".



Imagine, we have a force transducer with strain full bridge output. It will measure the weight in our experiment. In the first picture, we measure just the unbalance of the bridge sensor, e.g. 35 N. Let's do a "Balance". The output is now 0 N. A vehicle is put on the test bed. We measure the weight, it is 12 000 N. For our measurement only the changing of the weight is of interest (e.g. vibrations of the vehicle), so we cancel out the fix offset with the Zero function.

Click the "Zero" button in Channel Setup (can be reset by right mouse-click). The output is now zero again. Note, that this is a pure software subtraction. If the range was set to "Automatic", the range is automatically adapted to -52000...+28000.

The range can be set to "Automatic" in the channel setup window of the appropriate channel (right mouse-click).

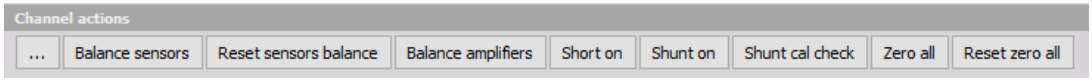


Now all offsets are canceled and we start the measurement. After the measurement you can go back to channel setup and remove the Zero. If it still shows the same weight, you can be sure, the sensor is still OK.

This function can also be accessed in the Measure mode (but NOT while storing!), see >> Group operations.

Group operations

On top of the Channel setup there is a bar with channel actions named Channel actions.

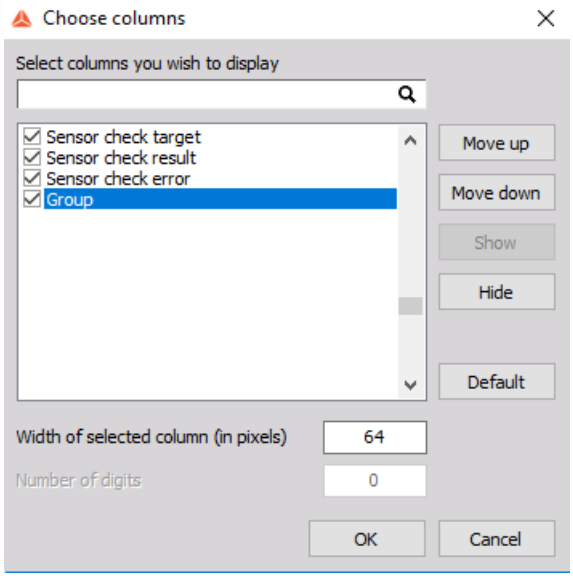


Depending on which amplifiers are used, more or less buttons are visible. Clicking on a button performs an action on ALL the channels.

For the following steps, please activate some more columns on the channel setup table. Click on one of the small icons, and select "Edit columns"

For information about Channel grid see -> Setup -> Channel grid.

Select the fields "Sensor check target", "Sensor check result", "Sensor check error" and "Group"



Group – Bridge balance

In the custom SIRIUS slice below there are 2 STG modules, each with a sensor connected. Click on the "Balance sensors" button.

As you see, all bridges are balanced with one click, input values become 0.

Group – Shunt Cal Check

From the picture above, note that there has to be a ShCal Target entered before doing the check (1,45699775 * 210 * 2 = 611,939 N/mm²; see section "Shunt").

Now click on the "Shunt cal check" button.

After a few seconds, you see the result in the additional columns we added.

Group – Balance amplifiers

The "Balance amplifiers" operation is applied on all channels at once. It is used to correct the amplifier offset before measurement, e.g. the drift on long-term measurements, or if the device is put to extremely different temperature conditions in short time (e.g. performing a measurement at -20°C, and immediately after that at +40°C). Depending on the usage, it does not have to be done before each test, generally only from time to time.

1. Let's assume we have a SIRIUS STGM amplifier in bridge mode, set to the smallest range of 2mV/V. When there is nothing connected, we will see only noise, so please click the "Short on" button. The IN- and IN+ will be shorted internally to determine the amplifier offset, in this case 51,1 µV.

2. Then exit the channel setup and do the "Balance amplifiers", it takes some seconds.

3. Repeat step 1 and check the amplifier offset again. In our example it is only 0,05 µV.

4. If you want to reset the offsets again, please go to Settings -> Hardware Setup -> Analog -> and click the "Set card..." button on the right side. In the tab "Ampl. Balance" you see a list of all the applied offset corrections for each channel.

We used the STGM module on the last channel in Range3, as expected the offset is 51µV. The HG (high-gain) and LG (low-gain) values show the offsets of both dual-core ADC stages.

The button "Reset offsets" below deletes all correction values and the whole tab disappears.

Group operations with different groups

In the Group column assign one channel to "Group 1", the other to "Group 2".

Now you can apply (bridge) operations group-wise.

Group operations in Measure mode

Now, with one channel assigned to Group1, another to Group2, let's go to the Measure mode.

There are two new buttons called "Zero" and "Bridge".

If the "Zero" button is not there, most likely sensors from the sensor database / TEDS sensors are used, and changing the offset is not allowed.

Check sensor settings in Channel setup or Sensor editor. If the "Bridge" button is not there, no bridge amplifiers are set to "Used" in channel setup.

Now you can do a selective "Balance" on the group you want or on "All chs".

Note, that DEWESoft is in Acquisition mode, but NOT storing! When pressing "Store", of course no balancing is possible during measurement any more.

Shunt/Short at beginning and end of measurement

At the end of a measurement you may want to check if the strain gage and the amplifier are still OK.

Maybe you also want to see, if the bridge was drifting over time due to temperature or other effects.

Start the measurement, click the Store button. The "Zero" button will disappear, because zeroing also changes the channel min/max limits, and that is not allowed during measurement.

Also balancing the bridge is not possible at this state any more. Do a "Short on for 1s", wait a little bit, then press "Shunt on for 1s".

At the end of the measurement – when you are still storing (!) – do again a "Short on for 1s" followed by a "Shunt on for 1s".

Stop the measurement and go to Analyse mode.

Activate the cursors in the properties of the recorder instrument (on the left side).

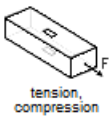
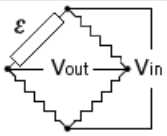
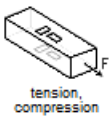
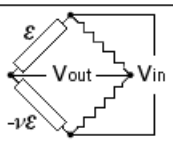
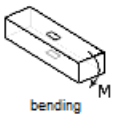
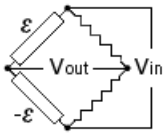
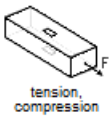
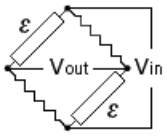
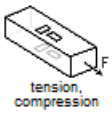
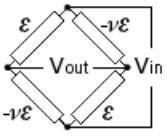
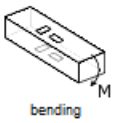
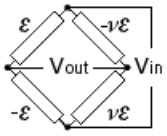
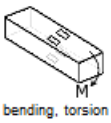
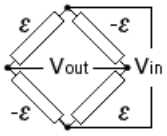
Move the white Cursor I to the Short position on start, and Cursor II to the Short position at the end (gray arrows). You can also lock the cursors to not lose them when zooming in and out of a longer measurement.

On the right side the Delta will be shown. In our case it is 0,0 – measurement OK.

Then move the cursors to the Shunt positions (black arrows), repeat the procedure to also check the Delta of the Shunt measurement.

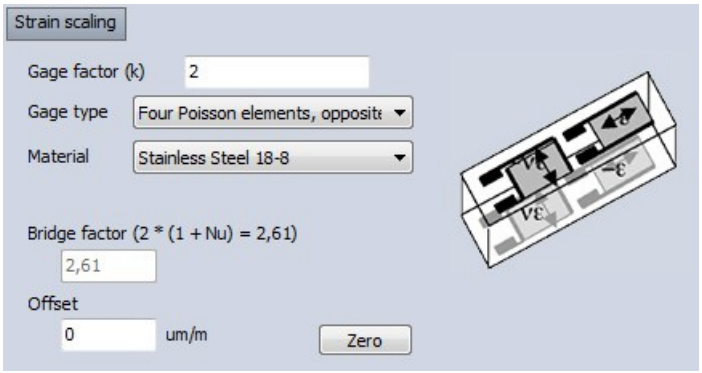
HINT: **Strain gage configurations in DEWESoft X**

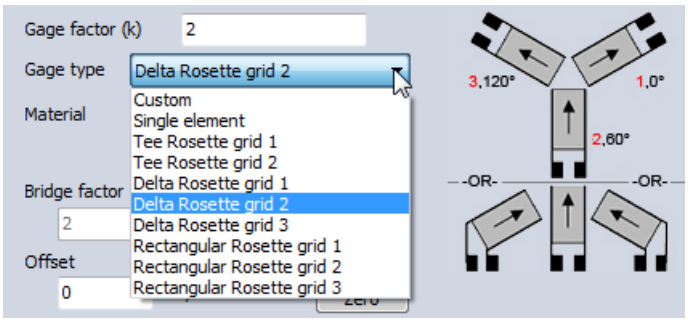
Until now it was rather difficult to select the proper strain configuration. You had to find out the bridge factor in the tutorials table and look for materials coefficients.

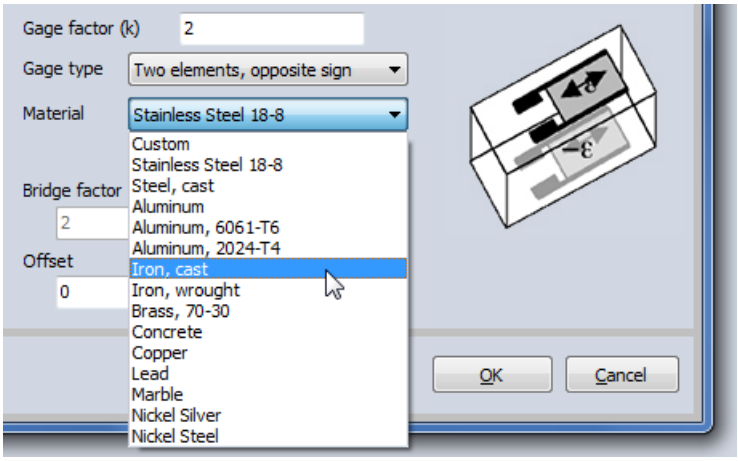
MEASURES	TYPE	BRIDGE	EQUATION Vout/Vin	BRIDGE FACTOR	LINEAR	DESCRIPTION
 tension, compression	quarter		$\frac{k \cdot \varepsilon}{4+2 \cdot k \cdot \varepsilon}$	1	no	Single gage measuring tension and compression - basic configuration
 tension, compression	half		$\frac{k \cdot \varepsilon \cdot (1+\nu)}{4+2 \cdot k \cdot \varepsilon \cdot (1-\nu)}$	$(1+\nu)$	no	One gage in principal direction and one in transverse direction - usually used for temperature compensation
 bending	half		$\frac{k \cdot \varepsilon}{2}$	2	yes	Two gages with opposite strain - usually used for measurement of bending
 tension, compression	half		$\frac{k \cdot \varepsilon}{2+k \cdot \varepsilon}$	2	no	Two gages with same strain - usually used for bending cancellation
 tension, compression	full		$\frac{k \cdot \varepsilon \cdot (1+\nu)}{2+k \cdot \varepsilon \cdot (1-\nu)}$	$2 \cdot (1+\nu)$	no	Two pairs of gages where one is in the principal direction and the other one is in transverse direction used in temperature compensation and bending cancellation
 bending	full		$\frac{k \cdot \varepsilon \cdot (1+\nu)}{2}$	$2 \cdot (1+\nu)$	yes	Two pairs of gages where one is in the principal direction and the other one is in transverse direction used in temperature compensation and tension cancellation
 bending, torsion	full		$k \cdot \varepsilon$	4	yes	Two pairs of gages in opposite strain - usually used for measurement of bending

Depending on the selected "Bridge mode" in the amplifier settings (Full, Half, QuarterBridge), it will now graphically show a various number of possible bridge configurations, and furthermore, you can select the material for using the correct Young's modulus.

Another error source is eliminated.



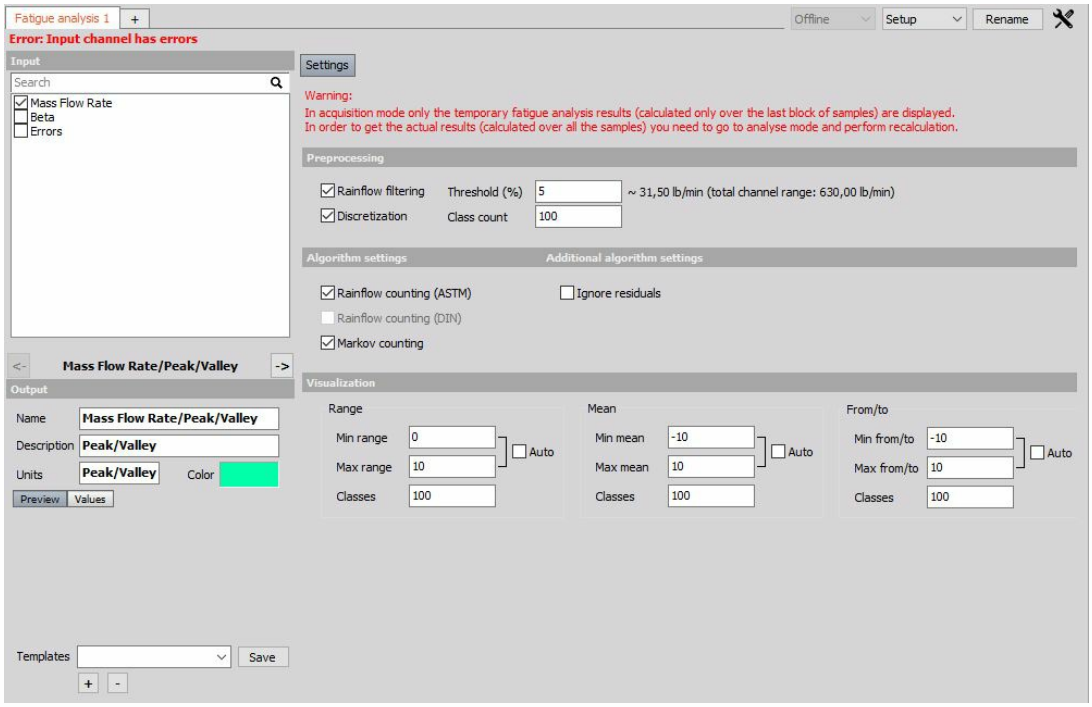

Gage factor (k)
2
Gage type
Delta Rosette grid 2
Material
Custom
Single element
Tee Rosette grid 1
Tee Rosette grid 2
Delta Rosette grid 1
Delta Rosette grid 2
Delta Rosette grid 3
Rectangular Rosette grid 1
Rectangular Rosette grid 2
Rectangular Rosette grid 3
Bridge factor
2
Offset
0
3,120°
1,0°
2,60°
-OR-
-OR-


Gage factor (k)
2
Gage type
Two elements, opposite sign
Material
Stainless Steel 18-8
Custom
Stainless Steel 18-8
Steel, cast
Aluminum
Aluminum, 6061-T6
Aluminum, 2024-T4
Iron, cast
Iron, wrought
Brass, 70-30
Concrete
Copper
Lead
Marble
Nickel Silver
Nickel Steel
Bridge factor
2
Offset
0
OK
Cancel

FATIGUE ANALYSIS

Fatigue Analysis Module supports a wide range of fatigue analysis features and utils.

After we press add new Fatigue analysis module screen like this will appear:



The module is organized into three main sections:

- Preprocessing
- Counting methods
- Visualization

NOTE: **In acquisition mode, only the temporary fatigue analysis results (calculated only over the last block of samples) are displayed. In order to get the actual results (calculated over all the samples) you need to go to analyse mode and perform recalculation!**

Preprocessing

Rainflow filtering

Rainflow filter (also called the hysteresis filter) removes small oscillations from the signal. All turning points that correspond to the cycles with the ranges below the given threshold are removed. Higher the threshold, more turning points are filtered out, and vice versa. The rainflow filtering can significantly reduce the number of turning points, which can be of importance for both testing and numerical simulation. Threshold parameter of value T corresponds to the T% of the absolute load signal range.

Discretization filter

Discretization filter divides the range of a load signal into N equidistant bins and assigns each turning point to its closest bin. Load signals are typically very stochastic, therefore, discretization is often applied to make them flatter and less stochastic. The number of bins is adjusted with the Class count parameter.

Counting methods

Markov counting

Markov counting is one of the most straightforward cycle counting methods. It operates on consecutive pairs of turning points and calculates absolute ranges between them.

Rainflow counting

Rainflow counting is generally accepted as being the best cycle counting method up to date and has become the industrial de facto standard.

The idea behind the method is to detect the hysteresis loops in the load signal, as depicted in the figure below. We'll refer to the hysteresis loops as the closed cycles. Parts of the signal that do not correspond to the closed cycles are the so-called open cycles or residuals. Simply said, closed cycles and residuals could be visualised as full-cycles and half-cycles, respectively.

Visualization

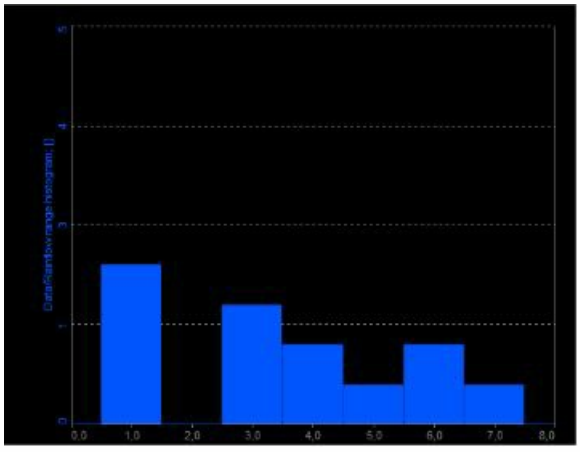
In order to graphically represent the results the Fatigue Analysis Module uses the most common visualization techniques used in fatigue analysis: range histograms, from/to matrices and range/mean matrices. When an input channel corresponding to a load signal of interest is selected, the following new channels are mounted:

- Preprocessing
- Peak/valley channel
- RF filter channel
- Discretization channel
- Rainflow counting
- Range histogram channel (2-D)
- Range/mean matrix channel (3-D)
- From/to matrix channel (3-D)
- Markov counting
- Range histogram channel (2-D)
- From/to matrix channel (3-D)

NOTE: **The channels marked with 2-D and 3-D correspond to 2-D and 3-D graphs, respectively. The channels corresponding to the Preprocessing section represent intermediate fatigue analysis results and serve for information purposes only.**

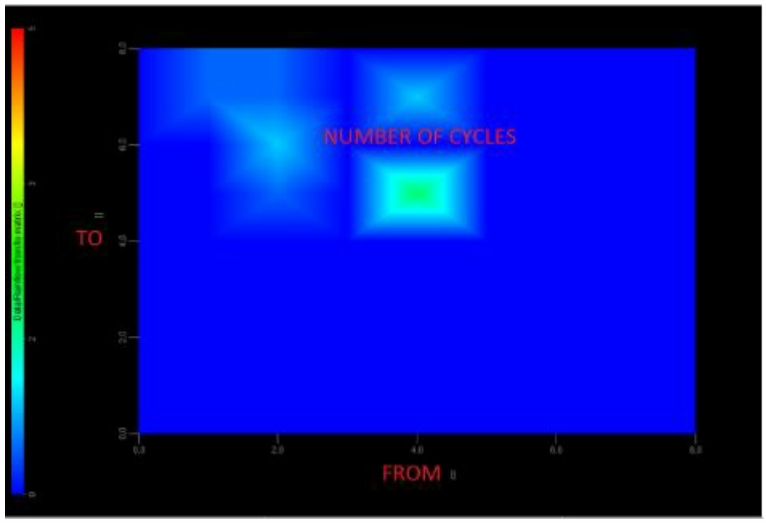
Range histogram

Range histogram is a 2-D graph representing the range distribution of a load signal. An example of a range histogram is depicted in the figure below. The X-axis and Y-axis of a range histogram correspond to the range and number of cycles, respectively.



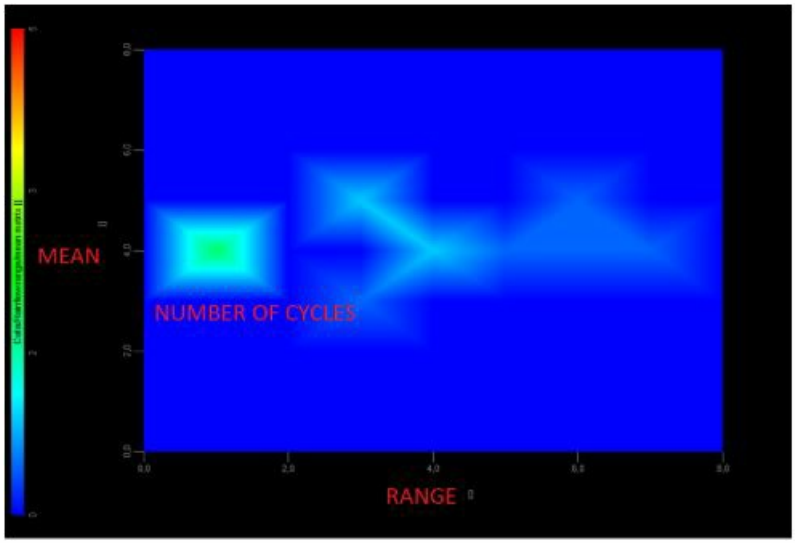
From/to matrix

From/to matrix is a 3-D graph representing the cycle distribution of a load signal. An example of a from/to matrix is depicted in the figure below. The X-axis, Y-axis and Z-axis of a from/to matrix correspond to From (cycle minimum), To (cycle maximum) and number of cycles, respectively.



Range/mean matrix

Range/mean matrix is a 3-D graph representing the cycle distribution of a load signal. An example of a range/mean matrix is depicted in the figure below. The X-axis, Y-axis and Z-axis of a range/mean matrix correspond to cycle range, cycle mean and number of cycles, respectively.



Visualization settings

The visualization section is organized into three groups:

- Range,
- Mean,
- From/To.

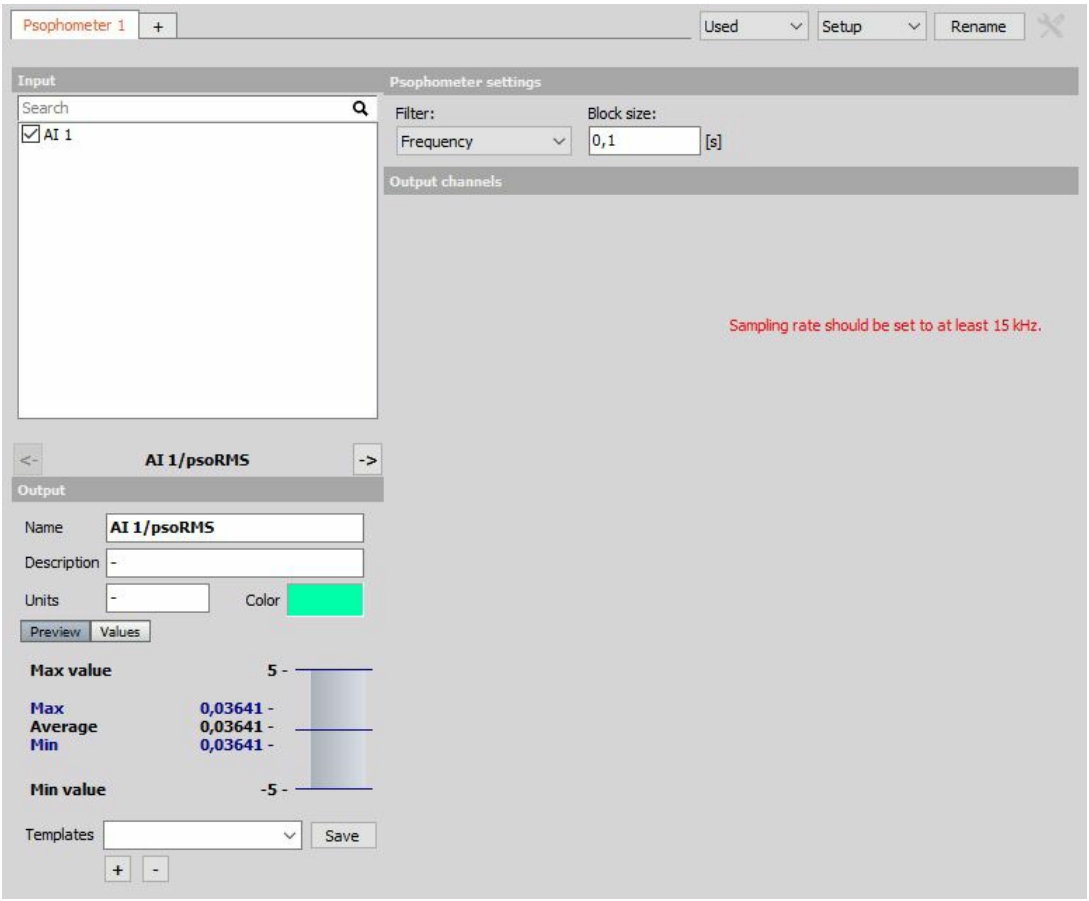
Each group controls the appearance of the corresponding graphs. For instance, Range group controls the appearance of the range histograms and the range/mean matrices. Min range and Max range parameters specify the minimum and maximum of the range axis, whereas Classes parameter specifies the number of classes the range is divided into. When Auto check box is

ticked DEWESoft automatically calculates Min range and Max range from the input signal. Exactly the same concept goes for the Mean and From/To groups as well.

For additional help visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training -> Fatigue analysis.

PSOPHOMETER

When you press the Setup button on Psophometer, the following setup window will open:



Psophometer is a plugin for measuring noise in telephone circuits. Psophometric weighting filters conform to internationally agreed characteristics laid down in CCITT 1951, CCITT P53.

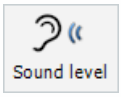
In Psophometer settings, we can set:

- Filter
- Frequency
- Time
- Block size

ACOUSTICS

In Acoustics we can choose from following modules:

- Sound level meter



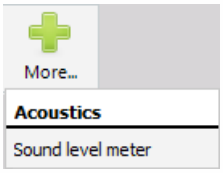
SOUND LEVEL

Dewesoft Sound level application module allows calculating typical parameters for sound level measurements from a single microphone. It allows Dewesoft to be used as the typical sound level meter. With appropriate hardware it can easily fulfill all the requirements for Class I sound level meter.

- **Required hardware** - Dewe 43, Sirius
- **Required software** - SE or higher + SNDLVL option, DSA or EE
- **Setup sample rate** - At least 10 kHz

For additional help with Sound Level Application visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training.

After required hardware - microphone installation and setup (this procedure is to be performed only after installation of new or changed hardware), Dewesoft Sound level measurement is activated by selecting More modules and under Acoustics selecting Sound level meter.



For instructions on how to Add new module see -> Setup -> Add module.

After selecting this option, a tab labeled Sound level appears in the Dewesoft Setup screen to define and setup sound levels module(s).

To use sound level module, please select and set up one or more analog channels in Analog tab to measure the sound.

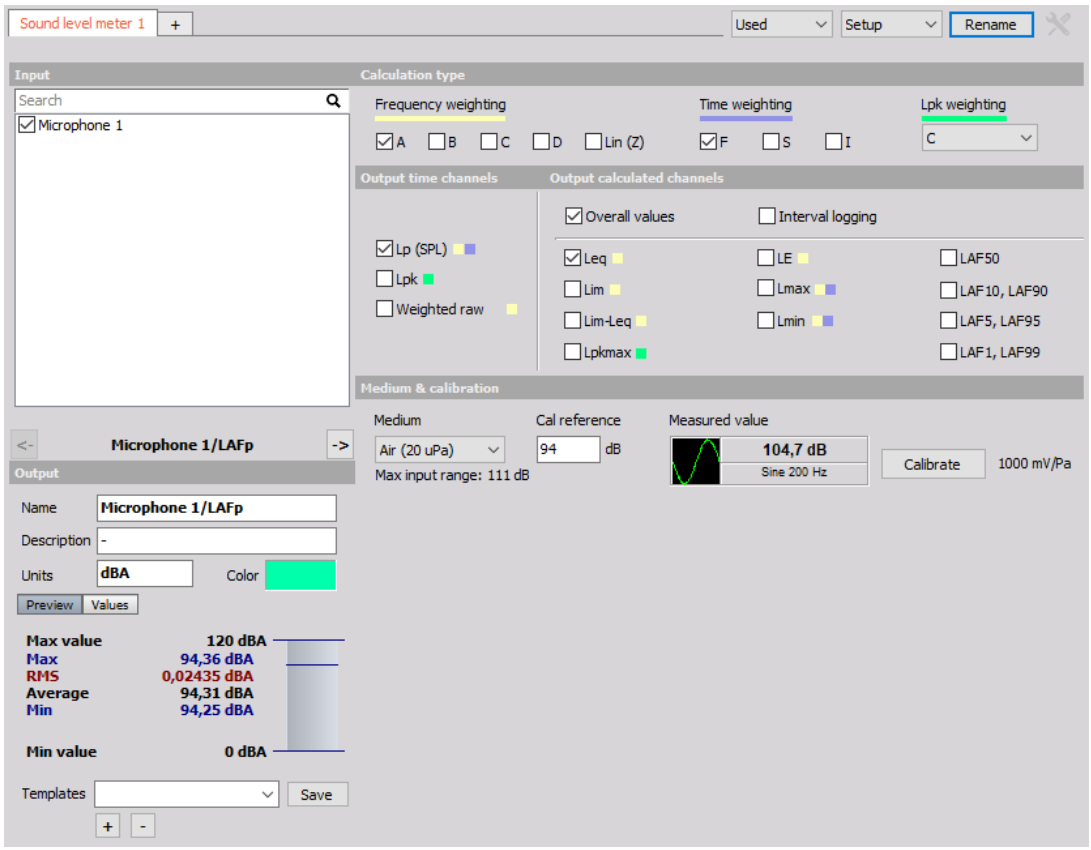
For detailed information see -> Analog in Channel Setup.

Basic procedures of Sound level application setup are:

- Sound level/channel setup for applied hardware
- Microphone calibration

Sound level / channel setup

After we press add new Sound level module screen like this will appear:



This screen has the following main sections:

- Input channel selection
- Output channel settings
- Calculation type
- Output time channels definition
- Output calculated channels definition
- Microphone Calibration

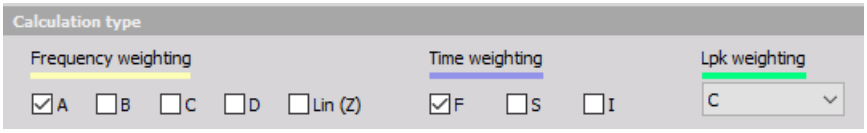
With using View Channel List button we can show different view on Sound levels module setup -> Channel list with all defined output channels appear.

Input channels

First of all select the input channels which should be measured at the upper left side of the display. In our case only AI 0 is selected. We can select multiple channels and then have several output channels with the same settings. Please take care that the input channels must be scaled in pascal (unit=Pa).

Calculation type

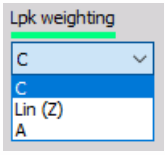
We have several options to choose in Calculation type section:



Any combination of:

- Frequency weighting (frequency response to human hearing) and
- Time weighting (time averaging) can be selected.

We can also select the weighting especially for Lpk from linear, A and C from drop down list.



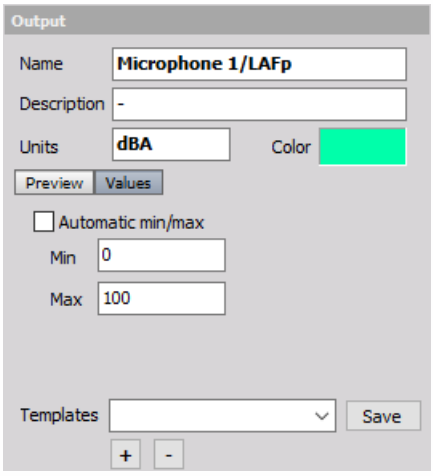
Output channels

The lower-left area displays the output channel settings, like the channel Name, the Units, and also the Color.

- **Name** - The first line of channel name holds the name of the output channel. This name will appear in the channel setup list and in the channel selector for the display. The second line of the channel name usually holds the channel description which will be shown in displays.
- **Unit** - The units of the channel describe the physical measured units. The default units (mostly dB) are already set.

For detailed information about this General setting see also -> Channel Setup.

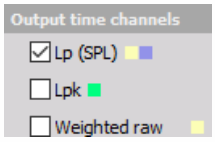
Values - The user scale Minimum and Maximum values can be set here. Minimum and maximum scale is used in the display as the default and full display range. The maximum and minimum can be set automatically or manually by clicking on check-box and entering the value in the fields.



Sound level module (also with only one input channel) can have more output channels according to selection in:

Output time channels

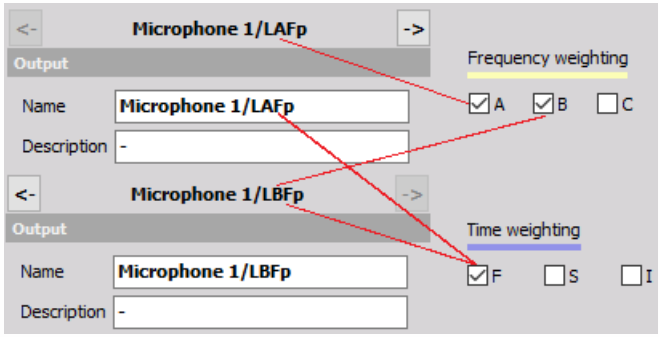
Select desired option with checking the box beside caption.



We have three types of Output time channels:

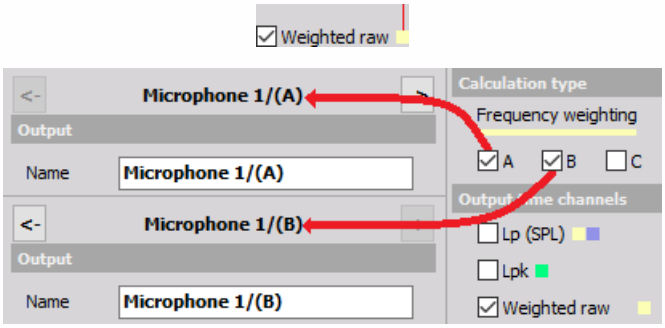
- **Frequency weighting** - LFTp - time and frequency weighted sound pressure level already scaled to dB which depends on Calculation type selection - Frequency weighting and / or Time weighting. For each combination of selection in this sections one separate output channel is created:

- **Time weighting**

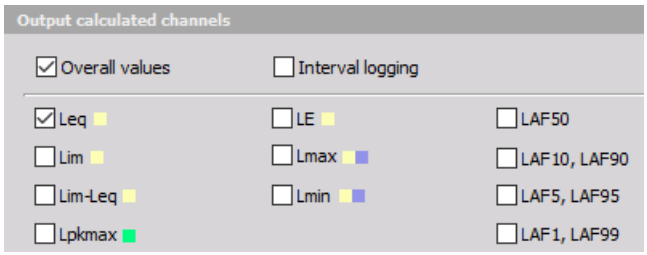


- **Lpk weighting** - LLpkpk value, which shows the current maximum value of the sound levels and depends on Calculation type selection - Lpk weighting. For selected choice in this section one separate output channel is created.

- **Frequency weighting raw** - F weighted raw value shows the frequency weighted time curve of sound in Pascal. This value depends on Calculation type selection - Frequency weighting. For each selected choice in this section one separate output channel is created:

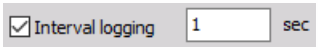


Output calculated channels



Select desired option with checking box beside caption. First choose upper calculation parameter (at least one must be selected for calculated channels):

- **Overall values** - we have only one value at the end of the measurement
- **Interval logging** - the time interval for logging is defined.

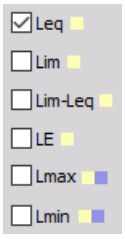


For example if we select to have interval logging with 5 seconds interval, we will get a new value after each 5 second. After that the value is reset and the calculation is started again.

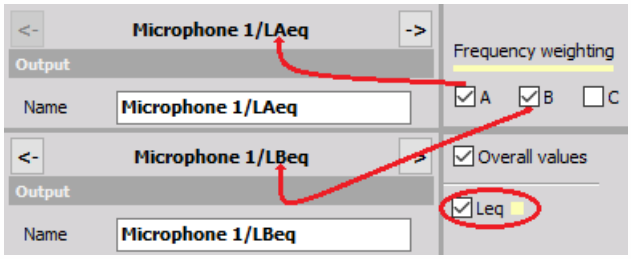
The values which can be calculated are:

- **Frequency weighting**
- **Time weighting**

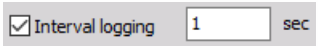
Frequency weighted LFeq value, which is equivalent sound level.



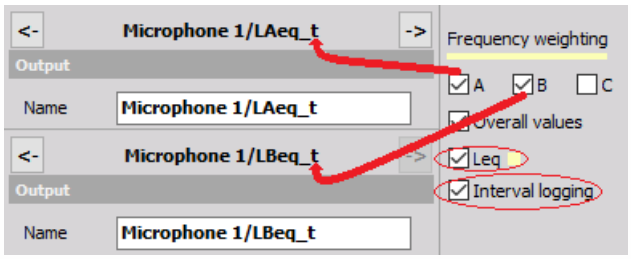
LFim tells the impulsivity of sound and is impulse weighted equivalent; the difference between those two values is calculated as LFim-LFeq. LFE is frequency weighted sound energy. LFTmax and LFTmin is time and frequency weighted minimum and maximum level of sound pressure. These values depend on Calculation type selection -> Frequency weighting and / or Time weighting. For each combination of selection in this sections one separate output channel is created.



When Interval logging is selected and defined:

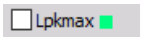


also for these combination of selection in this sections one separate output channel is created:

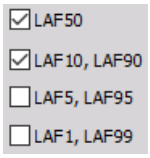


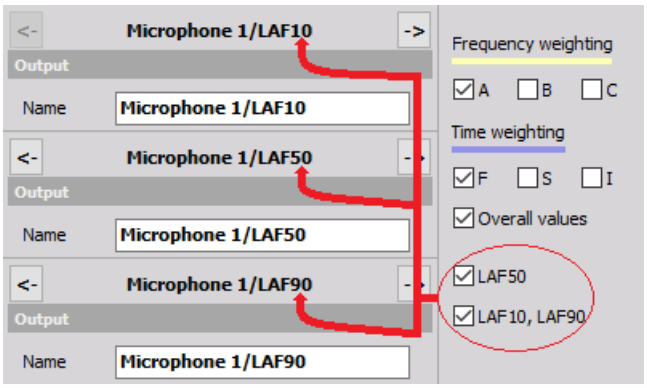
NOTE: **Interval logging is designated with _t characters on end of channel name.**

- **Lpk weighting** - Lpkmax is wither C or linear weighted maximum peak value of sound; depends on Calculation type selection -> Lpk weighting. For selected choice one separate output channel is created.



- **Classified sound levels** - Classified sound levels. Each calculated value is put in the classes and then we can choose to see LAF 1, 5, 10, 50, 90, 95 and 99% classes of values. For these options must be checked at least one or both: Calculation type -> Frequency weighting=A and Time weighting=F must be checked. Overall values or Interval logging. For each combination of selection in this sections one separate output channel is created:

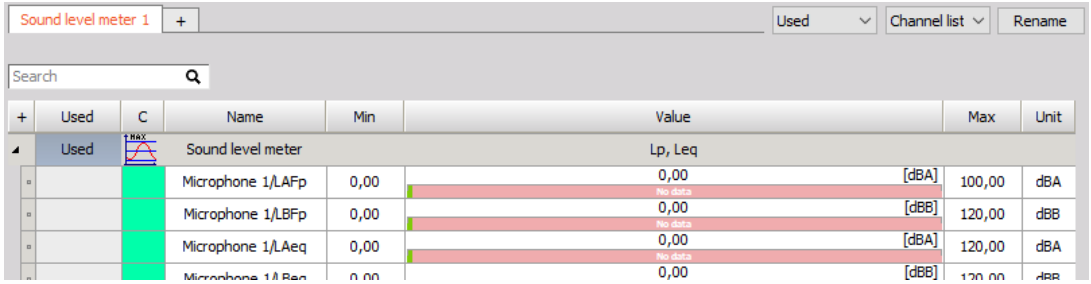




In case of more output channels, lower left part shows also the general output channel settings, where the only difference is that we have also two more buttons Prevous and Next to navigate between the output channels.

Channel list view

If we switch to channel list, a similar list of selected channels appears:



For information about Channel grid see -> Setup -> Channel grid.

Microphone calibration

The microphones can be calibrated in two ways. First of all we have to know that the direct value of measurement from the microphone is sound level in Pa. Therefore we need to scale it to physical quantity.

a) Scaling with calibration certificate

If we don't use the calibrator, but have the sensitivity of microphone, we can define it directly in the channel setup using the values from the microphone calibration sheet. This can be done from Dewesoft Setup - Analog tab Channel setup for the Sound Level Application module used channels.

For information about Calibration analog input channel see -> Channel setup - Calibration.

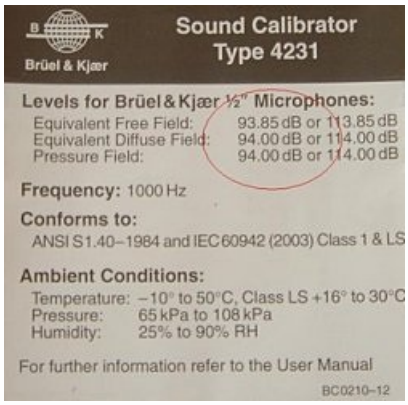
NOTE: **First, as usual, we define and enter the physical 'Units of measurement' - Pa is defined as the Units. Therefore we need to scale it to physical quantity. The Reference sensitivity is the key value to enter in the Dewesoft setup. Then it is the best to go to scaling 'by function' tab, check the Sensitivity and then enter in V/Pa value from calibration data sheet.**

b) Calibrating the microphone with calibrator

Another way is to calibrate the microphone with using the calibrator - make the real measurement. In this case the known parameter is the sound level emitted by the calibrator.

In example below it is 94 dB.

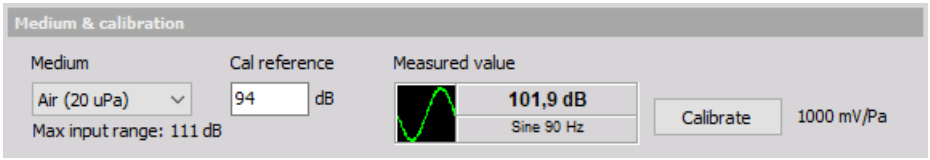
This value is directly entered in the Reference value field of Calibration section in the sound level module:



Then we connect the calibrator to the microphone and switch it on. We directly see the signal on the small overview in this section. This should be a sine wave with frequency of 1000 Hz. Since all the frequency weighted curves are referenced to 1000 Hz, this is very usual frequency for calibrating the microphones.

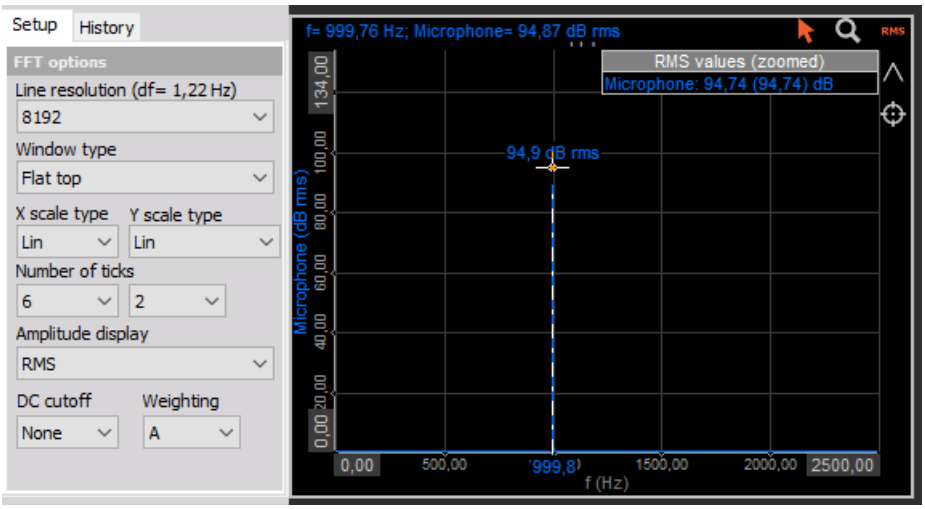
NOTE: **We need to use setup sample rate in Dewesoft at least 5 kHz or higher to make successful sound calibration. This can be changed in Dewesoft Tuner Utility.**

After we see that the sound is correctly recognized as the sine wave at 1000 Hz, we can press Calibrate button to perform a calibration. The sound module will calculate from highest FFT amplitude and reference value the sensitivity of microphone.



Sensitivity will be directly corrected already in the source channel and therefore no additional analog scaling is necessary. We can directly check the calibrated sensitivity with the information found on calibration certificate.

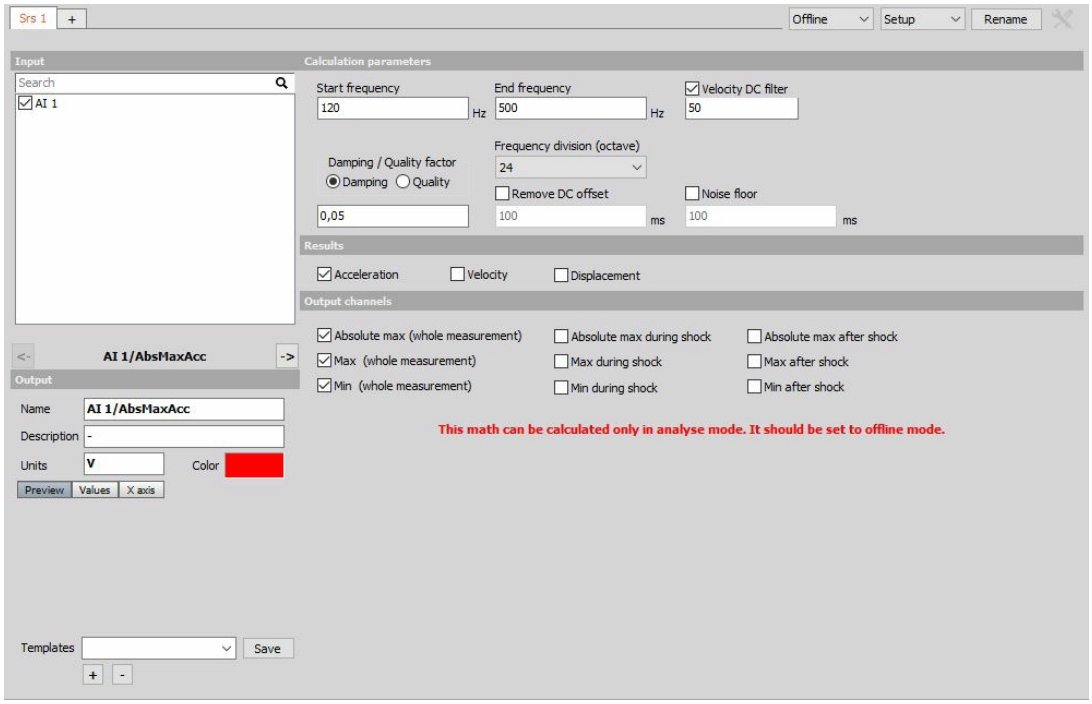
Now we have to check if the calibration was successful. Set a sampling rate of at least 2 kS/s - we would recommend 5 kS/s - and enter the FFT analysis (see -> FFT analysis). Set the FFT options to Flat top filter and the Y scale type to dB Noise. Now press the RMS icon to display the RMS values within the FFT graph. Switch on again your microphone calibrator and the RMS values should display 94 dB.



If there are mismatches please do the calibration again.

SHOCK REPONSE SPECTRUM

When you press the Setup button on SRS, the following setup window will open:



A shock response spectrum is a useful tool for estimating the damage potential of a shock pulse, as well as for test level specification.

Calculation parameters

- **Start frequency and End frequency** - define the calculation range for the frequency spectrum. The upper limit is set to 2000 Hz.
- **Velocity DC filter** - removes the DC component of the calculated time signal “InputChannel”/VelocityHistory.
- **Damping/Quality factor** - Different damping factors produce different SRSs for the same shock waveform.
- **Frequency division** - defines the resolution of the spectrum in frequency domain. Higher the number, better the resolution.
- **Remove DC offset** - removes the DC component of the input time signal.
- **Noise floor** - will calculate the noise floor of the input time signal.

Results

- Acceleration
- Velocity
- Displacement

Output channels

- **Absolute max (whole measurement)** - composite/maximax (the worst-case extrema for both instances). The composite shock response for the absolute maxima is the maximax.
- **Max (whole measurement)** - composite/maximax (the worst-case extrema for both instances). The composite shock response for the absolute maxima is the maximax.
- **Min (whole measurement)** - composite/maximax (the worst-case extrema for both instances). The composite shock response for the absolute maxima is the maximax.
- **Absolute max during shock** - primary (forced response, computation during the shock).
- **Max during shock** - primary (forced response, computation during the shock).
- **Min during shock** - primary (forced response, computation during the shock).
- **Absolute max after shock** - residual (free vibration, computation after the shock).
- **Max after shock** - residual (free vibration, computation after the shock).
- **Min after shock** - residual (free vibration, computation after the shock).

FILESTREAMER

The FileStreamer plugin reads channels from a binary file on an FTP server when their timestamp changes and displays their values as Dewesoft async channels.

Signal file format

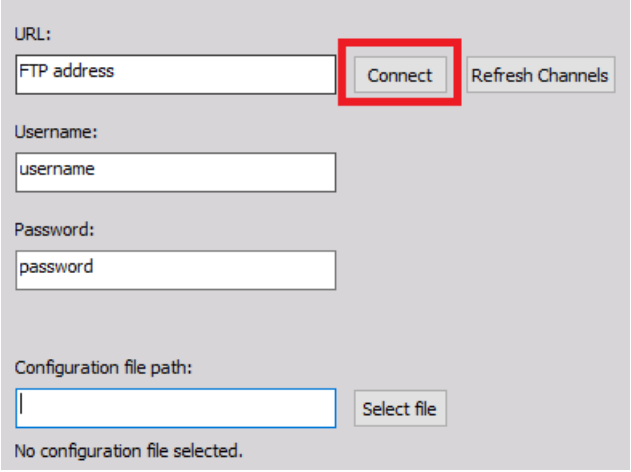
FileStreamer plugin requires a specific binary file format to successfully read required values. The file in question consists of a number of channels with a length of 18 bytes in the following format:

4 bytes (Longword)	4 bytes (Longword)	4 bytes (Longword)	4 bytes (float)	1 unsigned byte	1 unsigned byte
Not used	Timestamp	Not used	Value	Not used	Not used

The file needs to contain one or more of these 18 byte channel structures in order to be successfully opened. The plugin only uses timestamp and value data. Timestamps are required so that the FileStreamer can detect when a channel has changed. When the change is detected, the FileStreamer will read the channels from a file in the same order as they are written and display their values as async Dewesoft channels.

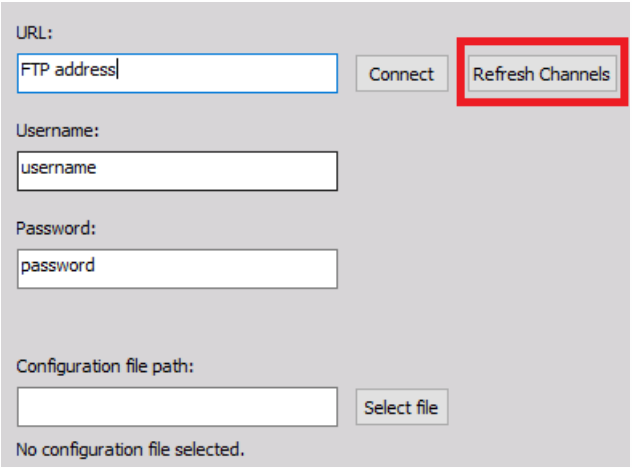
Connecting to the FTP server

To connect to an FTP server which hosts a file that you want to read, you first need to enter the full URL address of the file, username and password. The URL address does not have to start with "ftp://". After you have entered required information just click on the Connect button.



Refreshing channels

After the connection to the FTP server has been successfully established you need to click on the Refresh Channels button to mount channels contained in a file.

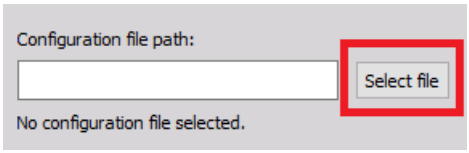


If the file has been successfully read, then you will be notified of the number of channels the file contains.

If a number of channels in a file changes, you need to click the Refresh Channels button again to show the new number of channels. Channels are mounted in the same order as they are written in the file.

Configuration file

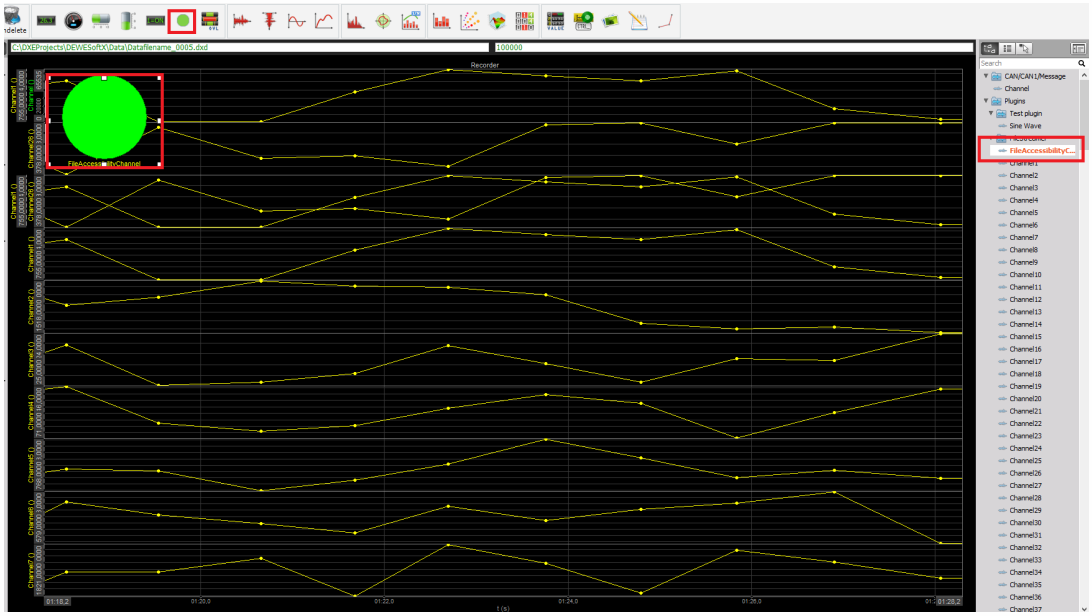
The configuration file is an ordinary .txt file which contains custom channel names in every line. You can select a configuration file by clicking the Select file button.



If a configuration file is not selected then the channel names will be set to default: Channel 0, Channel 1, Channel 2, etc,... By selecting a configuration file, channel names are replaced by those written in the configuration file in the same order. For example, the first channel mounted will be given a name from the first line of the configuration file.

File accessibility channel

An additional channel is mounted called FileAccessibilityChannel which is used to monitor if the file is being successfully written to and if it is accessible on the FTP server. If the file is being written to at least every 4 seconds and if it is accessible on an FTP server then the FileAccessibilityChannel will output a 0. If file has not been written to for 5 seconds or if it is not accessible on a FTP server then the FIleAccessibilityChannel will output a 1. The FileAccessibilityChannel can be connected to an Indicator lamp visual control which will change color based on the value of the channel.



MEASURE

Display design

Recently acquired data

NET acquisition data

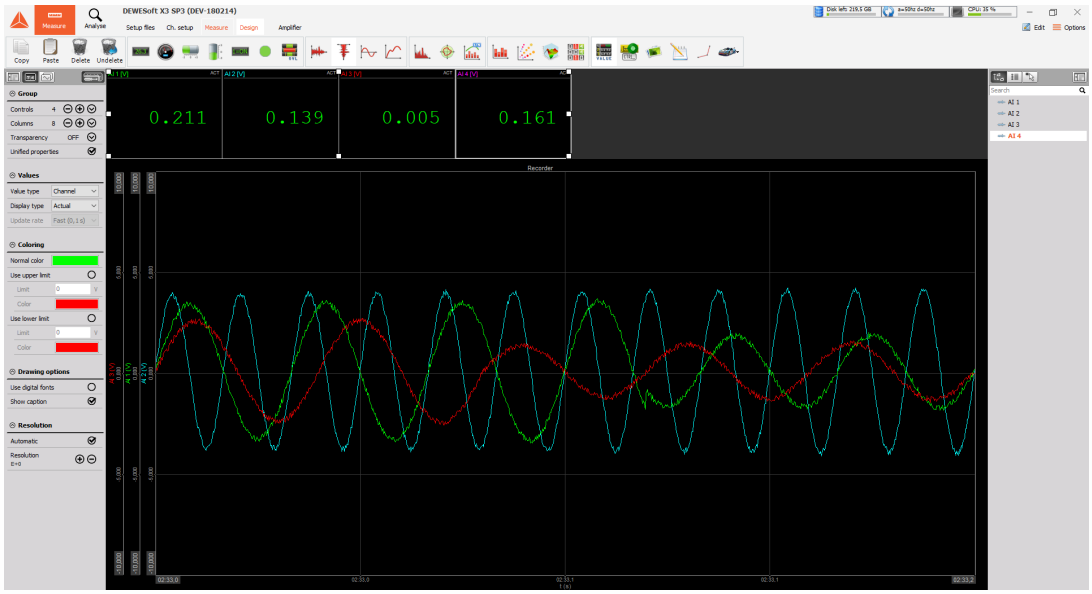
DISPLAY DESIGN

Primary goal of Dewesoft design online display screen is to create a clear and intelligible appearance of acquired and calculated data in different instruments on screen for review and analyzing of these data. Dewesoft allows you to set up different instruments for each input, for example digital meter, recorder, FFT analyser,... and arrange them freely in front of any graphic in your system to have a simple, but an efficient overview of your signals and measurement.

In Dewesoft we know basically four types of visual controls:

- controls which show only one value (digital meter, bar meter, analog meter, indicator lamp)
- controls which typically show all the data (recorder, vertical recorder, XY recorder, GPS map)
- controls which show the part of data directly or calculated (scope, FFT, octave, vector scope, harmonic FFT, tabular display)
- additional visual controls like picture, text or lines

All controls can be combined on one single screen or we can build several screens for specific part of measurement. Dewesoft has few predefined displays but these screens can be altered and own specific displays can be createdwith different appearance. The picture below shows typical display with standard elements for designing the display.



PREDEFINED DISPLAYS

Dewesoft provide some default screens with predefined displays of different instruments for each input, for example scope, recorder, FFT analyser,...

These instruments are built as a jump start that software acts like a classic instruments like strip chart recorders, classic oscilloscopes and so on. We have the following predefined displays - screens:

- **Overview** screen includes one meter for all active channels by default. An Overview display is intended to be defined and supplemented by the user with adding and arranging meters and other instruments.

Overview

- **Scope** - oscilloscope screen has predefined one Scope instrument, which is usually used for displaying fast, short-time events. Like in a traditional scope you can define trigger conditions.

Scope

- **Recorder** screen has predefined the recorder instrument displays for all of the active channels by default. This screen is meant to display the time-history plotting capabilities and to get an idea of the acquired signals over a long time - trend.

Recorder

- **Vertical recorder** screen has predefined the vertical recorder instrument displays for all of the active channels by default. It is similar to recorder display, but the time plots are shown vertically. It replaces old fashion paper recorders mainly used in telemetry.

Vert Rec

- **FFT display** (Fast Fourier Transformation) screen has predefined FFT and one scope instrument displays for the all of active channels by default (each graph can display up to 4 signals). The basic idea of the FFT display is to display the frequency components of your acquired data. Scope helps you to interpret the FFT data.

FFT

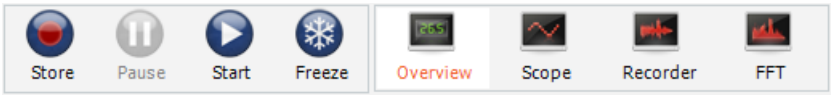
- **Power analysis** display screen has predefined Vector scope of selected signals and one scope instrument displays for the all of active channels by default. The basic idea of the Power display is to display of voltage and current as well as the phase between them of your acquired data immediately. The time domain based graph helps you to interpret the Power data.

- **Video** screen has predefined Video display and one recorder instrument display for the all of active channels by default (recorder graph can display up to 4 signals). The basic idea of the Video is to observe a video together with a recorder view for time domains display your acquired data.

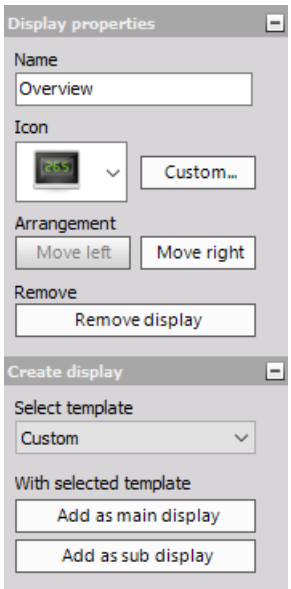
- **GPS** screen has predefined GPS display, digital meters for GPS channels and one recorder instrument display. The basic idea of the GPS screen is to observe a graphical representation position data together with a display of acquired values in digital meters and a recorder view for time domain display of acquired data.

With new setup, there are only a few displays which are added. These are Overview, Scope, Recorder and FFT.

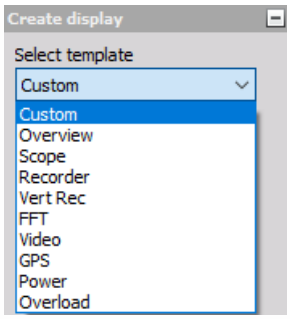
Displays can be changed by selecting icon on Dewesoft toolbar.



A great feature is that displays can be added according to the needs of specific setup. Additional predefined displays can be added by selecting the display icon.



A new predefined display can be added by choosing from display templates. Here all the standard displays named above are shown.



These screens can be altered to meet the user's requirements and arrange them freely in front of any graphic in your system to have a simple and efficient overview of your signals and measurement. Please look at the following section Creating own display and Display settings for more information on this topic.

DISPLAY SETTINGS

All Dewesoft pre-defined and added displays can be adapted to meet the user's requirements and to have a simple, but an efficient overview of your signals and measurement.

Pre-defined display is excellent starting point to create your own display with different appearance of acquired signals on screen with:

- Add instrument
- Standard instruments
- Additional instruments
- Additional screen edit functions
- Channel selector - different view on all used channels, channels assigning or reassigning to instruments, resizing the channel selector, user-defined channel groups
- Display menu functions - to manage displays: full-screen mode, add another display, Rename and Delete display
- Instrument properties - define control properties to set up predefined and new added instruments
- Screen edit function - to define Instruments appearance on screen, common and additional control and function

Design mode

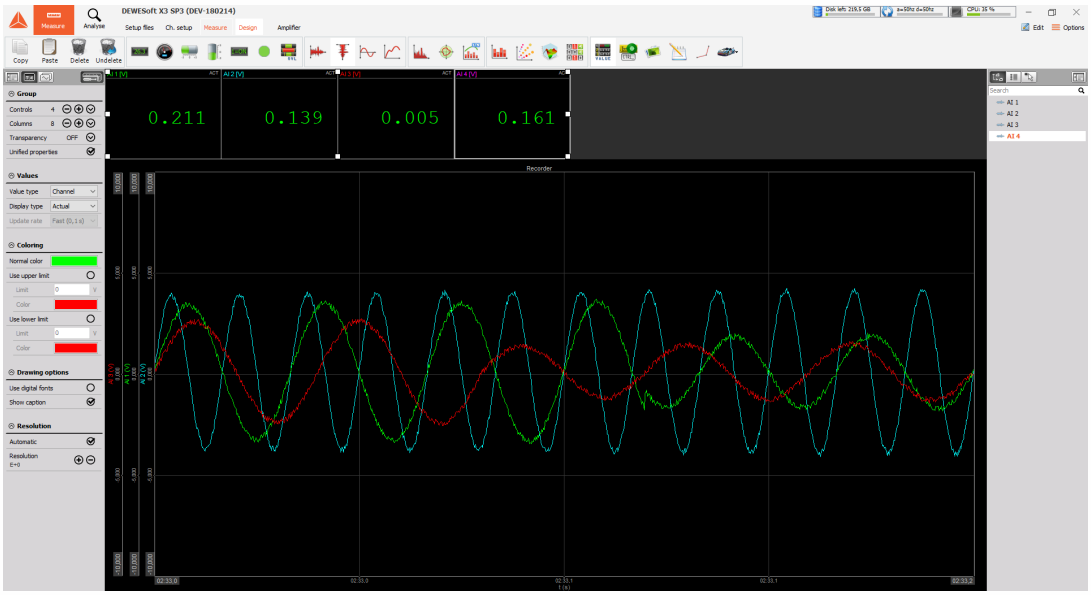
When you press the Design button or when selecting Design mode from the menu bar, design mode is selected and you are allowed to create your own display appearance of channels data on the screen.

After you press Design button, next to this button 'add instrument' design toolbar appears.

As soon as you enter any Visual control settings on Pre-defined display screen, you can enter the Design mode.

An Overview display is intended to be defined by the user and is best starting point to create own display, but also other predefined screens can be altered to meet the special requirements.

Design mode is selected as a default with Overview display:



WARNING: **When you have finished the adaptation, please press the Design button again to fix the layout, otherwise you won't have the full functionality of the instruments.**

After you press Design again, Design mode toolbar disappear.

All pre-defined displays can be adapted to your own requirements with:

- Shown predefined standard instruments for selected pre-defined display (and for all used channels) on the lower right part of this screen
- Control properties and Design tool button on upper left part of this screen to enter Design mode - Add instruments for this type of display with button on “add instrument” design toolbar
- Instrument setting on middle left part of this screen
- Assigning channels to new instruments with channels selector on right part of this screen
- Using additional screen edit function on design toolbar of this screen and Displays menu edit

display option
- Instrument appearance setting: position, size, grouping,...

Add instruments

To add an instrument to your display, after selecting Design mode just click on the desired icon on "add instrument" design toolbar, described at Instruments setup.

Every time when you click on the instrument icon, a new instrument will be added at the left-top corner of the instrument area.

You can add instruments according to your requirements. Due to the flexibility of the Dewesoft display screens, you can arrange the instruments according to your requirements with nearly no limitation.

For additional help visit Dewesoft PRO training -> Dewesoft Web page -> PRO Training.

Assign / Reassign channels to instruments

As a standard, channels will be automatically assigned to new instruments. But usually you don't want these channels to be in the new instrument and you want to reassign another channel.

Make sure that the new instrument is still selected - this is indicated by a white box around the instrument. If it is not selected, click once on it. Then click on any input from the channels selector on the right side of the screen to assign this channel to the new instrument. Some instruments like the Recorder or Scope allow more channels assigned to one graph.

To reassign any instrument from one channel to another, first click the instrument to select it, then unclick the currently selected channel from the channels selector (depending on instrument type, it will display unassigned or simply remove the channel), then click on any other channel from the channels list.

You can have the same channel assigned to multiple instruments if desired.

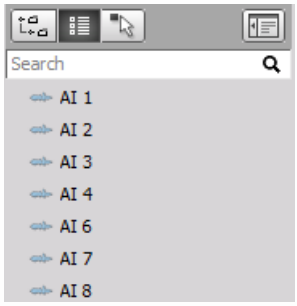
CHANNEL SELECTOR

Dewesoft offers a very powerful and flexible CHANNELS selector. It can display the available channels as already known in a channel list or in a grouped form. CHANNELS selector offers the following capability:

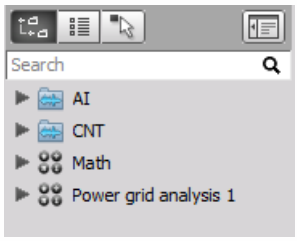
- Channel view / Group view - different view on all used channels in channel selector
- Select /deselect channels - assign or reassign channels to instruments

Examples of the channel selector:

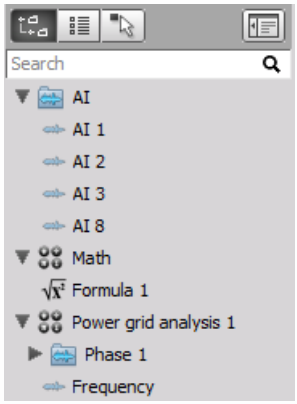
Channel list view:



Group view:



Group view with subchannels list:



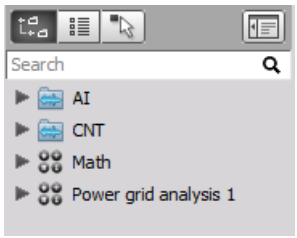
Channel (list) view / Group view

When you select the Group view (left picture below), channels will be grouped according to its source. we will have groups for analog input (AI), CAN, math and others. We can see the channels by expanding the group simply by clicking on it.

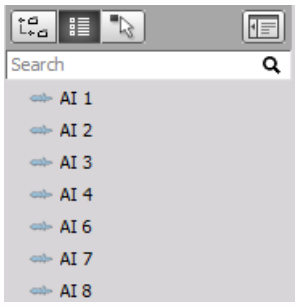
Another view of the channels is the channel view (middle below). Here the channels will be listed in alphabet order regardless of its source.

A third view is the list of all selected channels (right side below). This will show the channels which are currently selected on the display. This provides an easy way to deselect channels for current display by simply clicking on each item.

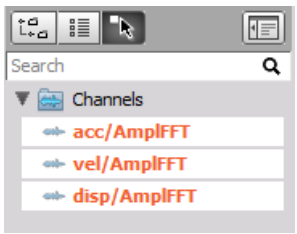
Group view



Channel view



Selected channels view

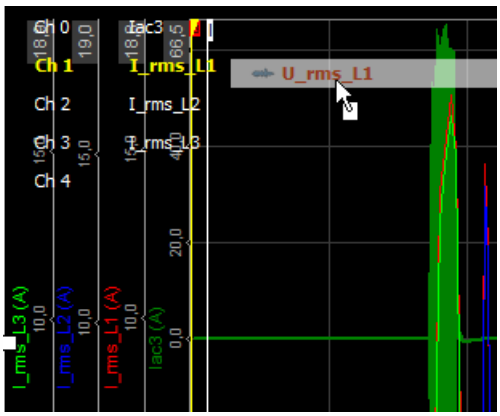


NOTE: **If there is only one hardware group available (e.g. AI) the channels are always displayed in List view!**

Select/deselect channels

The channels, which are shown on currently selected visual control can be selected by clicking on the channel. If the channel is already on the visual control, it will be marked and clicking on it will remove it from visual control.

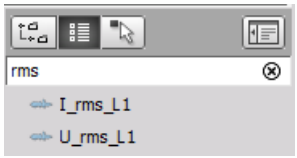
There are several other ways to assign channel to visual control. We can drag and drop the channel to the visual control. When we click and hold the left mouse button, channel will be selected and we can drag it over to any visual control. We can drop it to any place and that will add the channel to the display or we can drop it on the display's channel list to replace the channel. In the example below the Ch1 will be replaced with the U_rms_L1 channel.



In Design mode, we can drag and drop channel to an empty place on the display. This will add new visual control from the type currently selected on the Design bar and automatically assign the channel to it.

Searching for channels

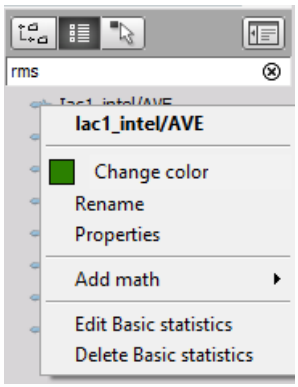
Dewesoft setups can easily have hundreds and even thousands of channels. To find channels we have added a feature to search for them by the name. If we enter the keyword in Search field, Dewesoft will look for channels with those keywords in the name (at any place). We can cancel the search by pressing the X button on the right side of the search bar.



Additional settings in analyse mode

In analyse mode the list is expanded and we can add any math from the channel selector. For example, when we want to add a filter to any channel, right click on the channel and select Add Math -> Filter. The filter window will be opened and the channel which was selected will be automatically chosen.

If selected channel source is math, then we have additional two options to edit the math function or to delete the math.


rms
lac1_intel/AVE
Change color
Rename
Properties
Add math
Edit Basic statistics
Delete Basic statistics

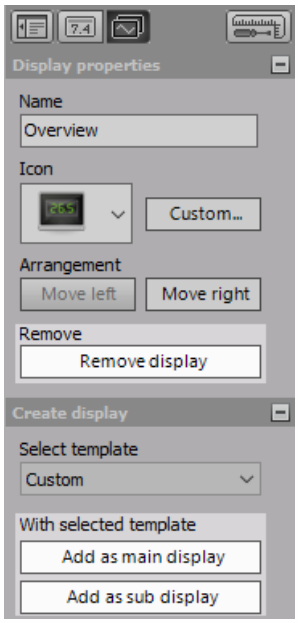
DISPLAY MENU FUNCTIONS

Full screen

Active online display screen can be enlarged on full Dewesoft window by pressing Ctrl-F. To escape from this mode press Esc button on your keyboard.

Add new display

Displays can be managed by choosing the display icon (shown in green color below).

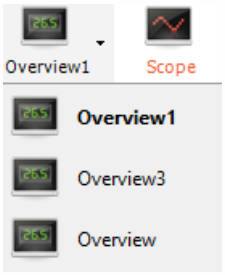


If we want to create a standard display, then choose it from the template list. If we want to create just an empty display, then Custom template must be chosen. We can add the display as main display, which will add a new screen right next to the selected display on the display menu bar. Then we can change the name and icon by changing Display properties.

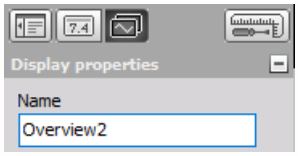
We can also add the display as a sub-display, which will add a new display as a child item of currently selected screen. The display icon will get a drop-down button to change the currently shown display.

Additional display screens

When you have added additional displays, you can select them directly with the down arrow next to the appropriate display icon:



Name of the displays can be changed by entering a new name in the Display name field. Icon of the display can be changed by setting one of the standard icons or by selecting any file when choosing the Custom item from the Icon drop-down and then selecting any image file. 32x32 is the best resolution for custom icons. We can also move display left or right by choosing the Move left or Move right button.



Delete display

Selected display can be deleted with selecting Remove display option from the Remove menu; after selecting this option Warning window is shown.

SCREEN EDIT FUNCTIONS

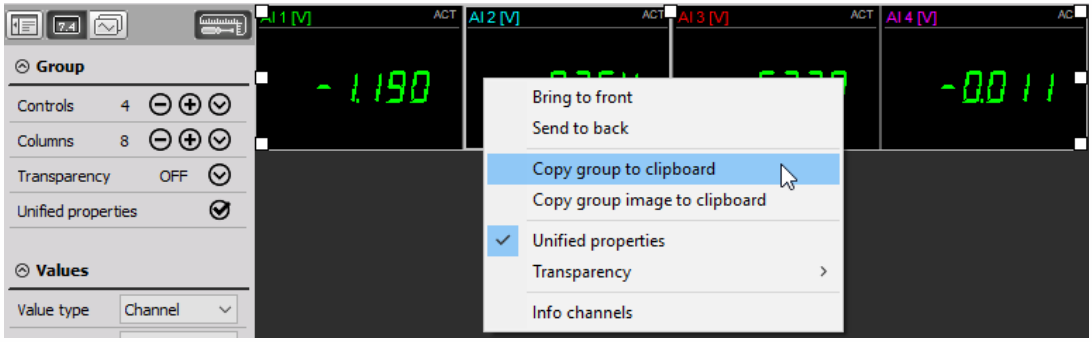
Dewesoft offers an additional screen edit function to create intelligible display all channels data on the screen.

- Instrument appearance - arrange elements - instruments with Positioning, Size and define Transparency.
- **Common function** - to Copy & Paste, Delete and Undelete elements.
- Additional controls - to illustrate your measurement with Background picture; to write text on the screens with Text element; to draw lines and shape, connect different elements,... with Line element.

Copy & Paste existing group (instrument)

You can use the Copy function to create a new group/element. The new elements will have exactly the same settings and can be edited as all other.

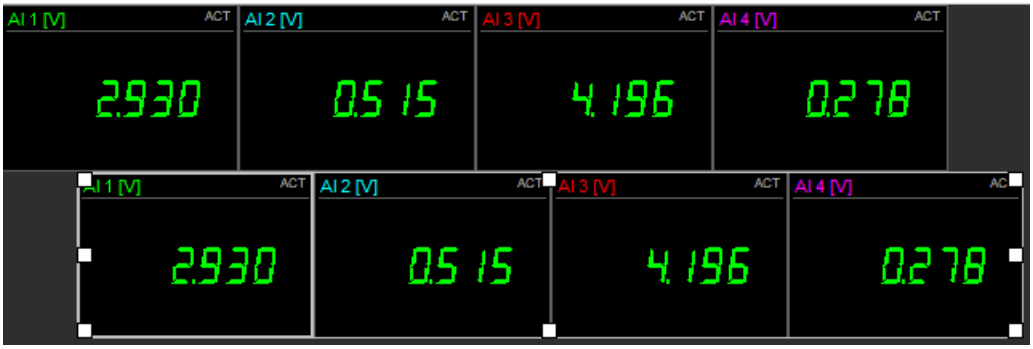
To copy the element, right-click on the element and select Copy group to Clipboard from the menu list:



or select Copy group to Clipboard from the Edit menu.

Then move the cursor to the desired position (must be on screen outside group), right-click again and select button which appears on screen.

That's all - the new element is available:



Delete instruments

You may want to delete instruments from the display screen.

To delete any instrument, just click it once to select it, and then click the Delete Instrument button on Design mode toolbar or press the Delete button on your keyboard.

WARNING: **When one instrument in group is selected, with using this function WHOLE group will be deleted!**

Undelete instruments

If you have deleted an instrument from the display screen and want to undo that select Undelete from the Edit menu.

WARNING: **Be aware that this function works only for the last deleted instrument!**

INSTRUMENT APPEARANCE

When you have to display dozen of channels in many elements - instruments, you must for clear appearance arrange all this with:

- Position elements
- Size elements
- define Transparency

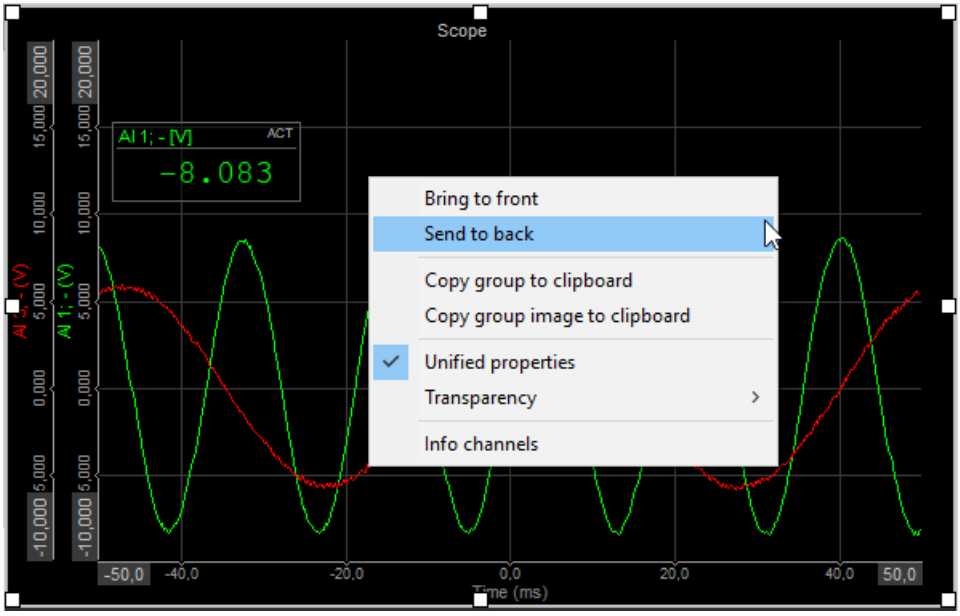
Position

Make sure that your instrument is still selected (simply click on the element to select it, a white box around it will indicate the selection). Now you have two possibilities:

- **Drag the element** and drop it where you need it
- Use the **arrow keys** to move it in the desired direction

HINT: **use the SHIFT key simultaneously with the arrow keys to increase moving speed.**

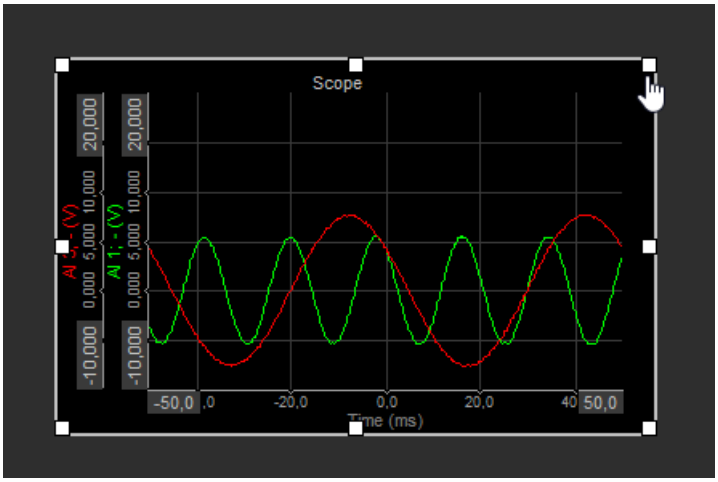
If the element comes in conflict with other elements, you can bring the element in front of the other one or behind it; therefore right-click on the element and then select Bring to front / Send to back from the appearing menu list.



The Design mode offers a help: when you have to arrange dozen of elements, it is nearly impossible to line them up. Therefore the instruments have a "magnetic" behavior. When you move one element by mouse and come next to another element, it will automatically snap to its outline.

Size

You can define the size of your instrument completely free. Note that you have 8 handles around the recorder. We can adjust the size of the control with dragging those handles. When the instrument is selected, simply click on one of the six white boxes at the outline of the instrument (the cursor will change to double arrow - see picture below), keep the mouse button pressed and move the mouse into the desired direction.

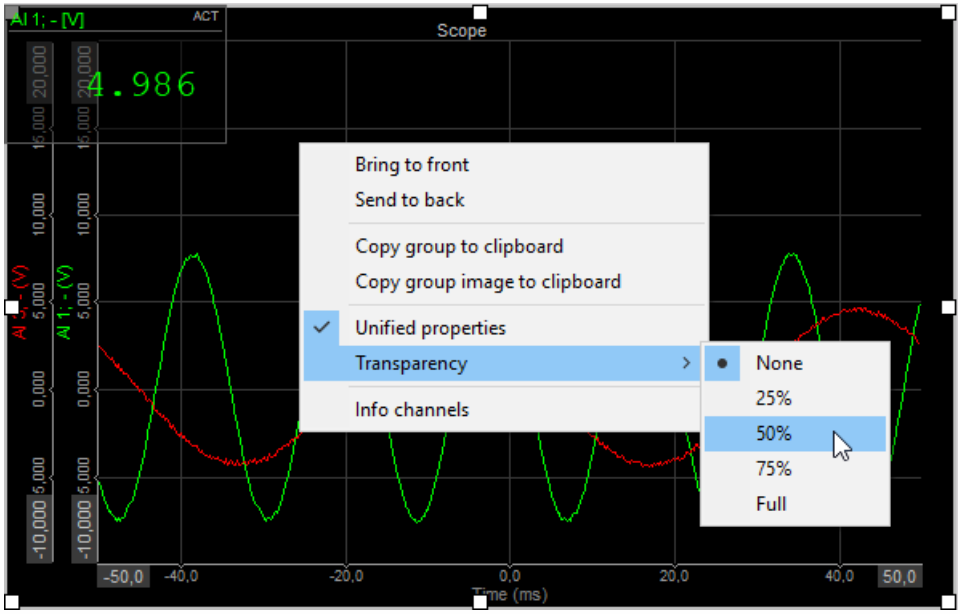


Transparency

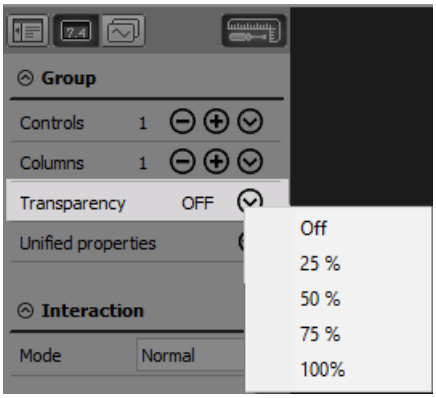
Now we have defined the instrument, its channel(s), size and its position. But what to do when instruments overlap one another?

Dewesoft offers a transparency for each element, which can be set to none, 25%, 50%, 75% or Full transparency. It can be set in two ways:

- use the Transparent checkbox in the Control properties bar and select the transparency value from the drop-down list
- right-click on the desired element, select Transparency and the desired value from the appearing menu:



After setting Transparency from this menu also Transparency field in Control properties is updated to this value:



For detailed information about Control properties -> see -> Common graph settings and Group properties.

The following example should give you an impression of the function - the Digital meter transparency has been set to 75%:

TEMP_OUTSIDE: - [deg C]
21.0
PSvel, v [km/h]
89.26

VISUAL CONTROL SETTINGS

Dewesoft allows you to set up different instruments for each input, for example digital meter, recorder, FFT analyser,... and arrange them freely in front of any graphic in your system to have a simple, but efficient overview of your signals and measurement.

To add an instrument to your display, after selecting Design mode click on the desired icon on "add instrument" design tool bar.

Digital meter

Recorder

Vector scope

Horizontal/vertical bar graph

Vertical recorder

Harmonics analysis

Vertical bar graph

X-Y recorder

Video

Analog meter

Scope

GPS

Indicator lamp

FFT analysis

Overload indicator

2D graph

Octave

Tabular display

3D graph

Orbit plot

CA p-v

Control element

CA scope

Discrete display

GROUP PROPERTIES

Dewesoft offers a Design Control properties to create a complex view all acquired and calculated data on the online display screen.

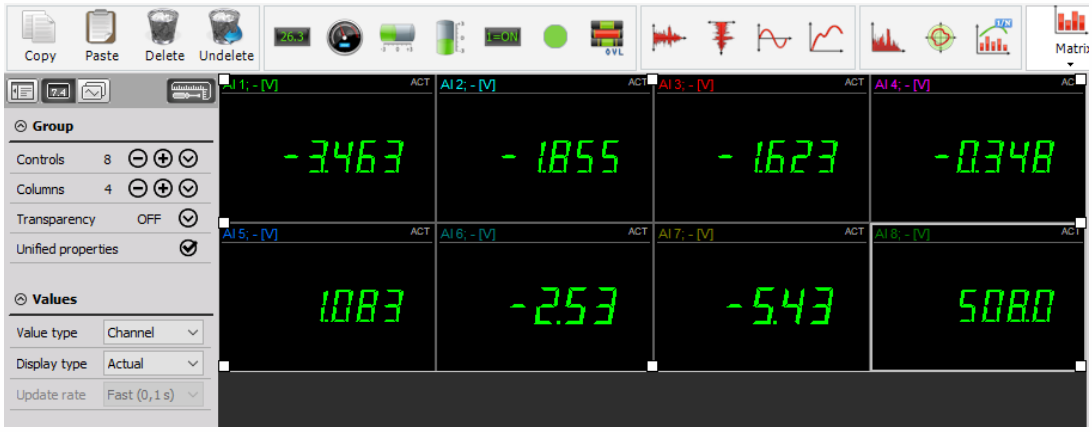
With Control properties Dewesoft allows to create and arrange elements - instruments in box - Group of instruments with:

- define Number columns in group
- Add / Remove instruments to/from group
- define Transparency of instruments
- define Unified properties (different settings for each instrument)

Create an instrument group

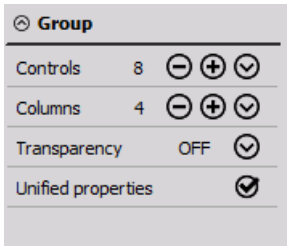
Placing instrument by instrument on the display can take some time, especially when you have to display dozen of channels. Therefore Dewesoft allows creating instrument groups, which are boxes with the same sort of instruments.

Example digital meter:



Columns number in group

With the Cols the drop-down list, you can define how many columns of instruments should be allowed in group.



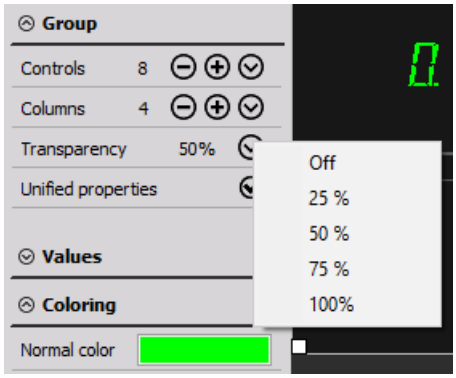
Add / Remove instrument to / from group

Sometimes you might require more graphs - instruments on one display according to your requirements.

After defined column number, simply press the Plus button to add same instrument or the Minus button to remove the instrument from group and from display.

Transparency

Dewesoft offers a transparency for each element, which can be set to none, 25%, 50%, 75% or full transparency. It can be set by checking the Transparent checkbox in the Control properties bar and select the transparency value from the drop-down list.



Unified properties

As a standard, all instruments within one instrument array - group have the same properties.

For example all digital meters within an array are set to 50% transparency, show the peak-peak value with very slow (5s) average time.

Use the Unified properties checkbox to allow different settings for each instrument within the selected group.

In our upper example now you can set e.g. peak-peak value for the first instrument, RMS for the second, Max for the third...

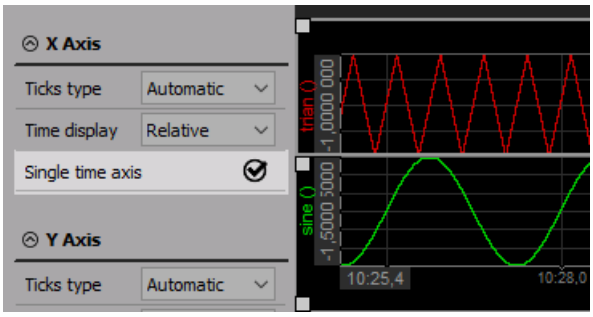
COMMON GRAPH SETTINGS

Single / Multiple Time axes

Dewesoft offers two different time base types for multiple graphs: single time axis or multiple time axes. Press the Single time axis check-box to change between these two modes.

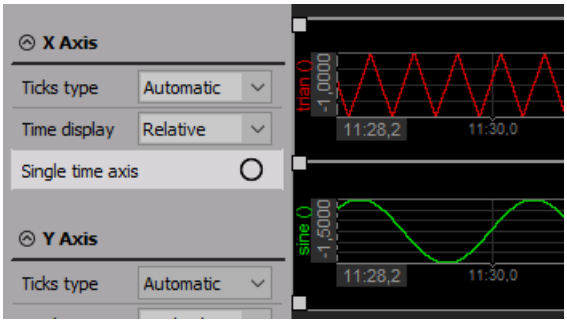
- **Single time axis** -> Single time axis box checked

The advantage of the single time axis is to have more space to display the data.



- **Multiple time axis** -> Single time axis box unchecked

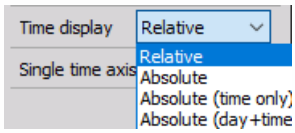
The advantage of the multiple time axes is to make it easier to read time-related information.



Appearance by vertical instruments is the same, except Time axis is vertical.

Time axis type

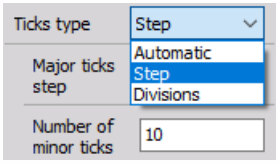
You can show either absolute or relative time in the Time axis (in case of Vertical recorder in vertical direction) for each graph. Click on any graph to select it and then use the selector to make your choice.



- **Relative** - elapsed time since recording or monitoring began
- **Absolute**- current time and date; this format is compatible with the timing information according to UTC
- **Absolute (time only)** - current time only
- **Absolute (day+time)** - number of the current day in the year and the time

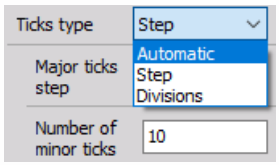
Grid resolution

To make the analysis more comfortable, Dewesoft contains the possibility to change the ticks type in the recorder display. Depending on range and signal, it may be interesting to change the ticks type from Automatic to Step or Divisions.



Display type

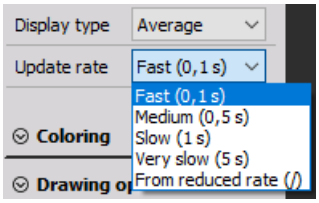
Your acquired signal may not be only a static signal; if you have ever tried to display a high-dynamic signal with digital values, you know that you have to make some kind of statistic to get representative values. Therefore all instruments offer different display types, which can be selected from Display type drop-down list (last three choices only for Digital meter).



WARNING: **Be aware that all these display types represent only statistic values for the online display. The settings have no influence on the other displays or the data storage.**

Average time

With the Average time drop-down list, you can define the update rate for the selected instrument. As a standard, the values for the display type will be calculated Fast (0.1 s) over a period of 0.1 seconds, which represents also the internal minimum calculation period.



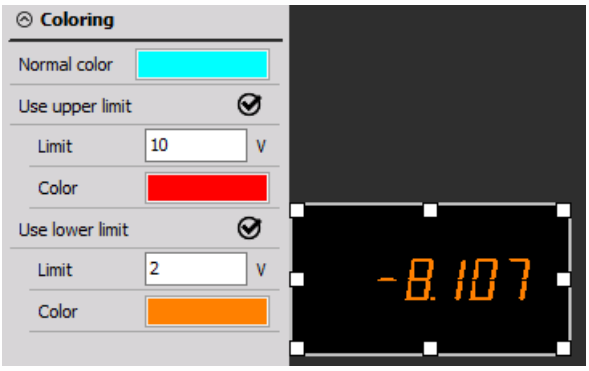
The system will still run at the dynamic sample rate and Dewesoft will acquire the data with full speed, calculate minimum, maximum, average and RMS for this time interval, however, every data point will NOT be displayed on the instrument and stored, but only these calculated values.

When From reduced rate mode is selected, the system will reduce the data continuously according to the static/reduced rate selected in STATIC/REDUCED RATE drop-down list in the channel setup. For help see -> Setup -> Recording setup.

Upper & Lower limit

You can set high and/or low alarm limits for each instrument, which will cause the change of color of instrument number to selected color when it is either above the high limit, or below the low limit.

If you want to use limits for an instrument, select it by clicking on it once. Use the Use check boxes to activate/deactivate high or low limit detection. If enabled, you can enter the alarm level for each.



For example, if we want to have the signal change to red when it either falls below 10 or rises above 7000 rpm, we set this meter up accordingly (see picture above).

Note that the number changes to red as a standard when it is outside of both limit, and remains green when it is within the limit(s). You can set Lower limit, Upper limit, or both limits for each meter.

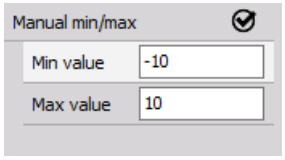
If you want to change the limit color simply click on the colored field next to the high or low limit field and choose the desired color from the appearing color selector window.

NOTE: **The upper and lower limit is only a visual effect; it is completely independent from the Alarm monitoring function.**

Minimum and maximum shown value

You can define a Minimum and a Maximum shown value for the bar graph; this feature is very important to get a better scaling for the display.

For example your temperature measurement range is 0 to 1000 °C, but you measure only between 0 and 40 °C.



Just enter both values and the bar graph scaling will change immediately.

Run mode Time axis scaling

WARNING: **This way changing Time axis can't be done in Design mode -> available only in Run mode.**

Use the blue PLUS and MINUS symbols to expand or compress the time axis. Clicking MINUS will show more time across the graph. You can show the entire length of recording, regardless of how long it is! Even days and days of data can be shown here.

By unchecked Single time axis - multiple time axis this control appear for each graph.

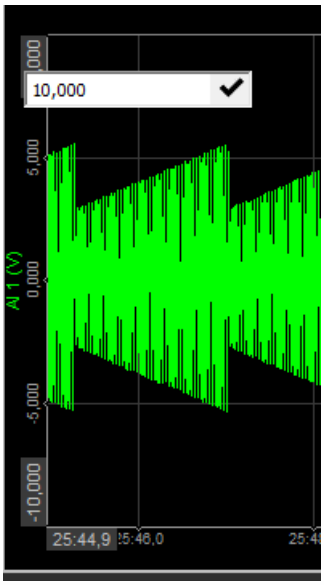
Appearance and functionality by vertical instruments are the same, except Time axis is vertical.

Run mode Changing the Y-axis scale/auto scale

WARNING: **This way changing Y-axis can't be done in Design mode -> available only in Run mode.**

If you want to change the Y-axis for any input, the recorder offers two possibilities:

- **Enter values**

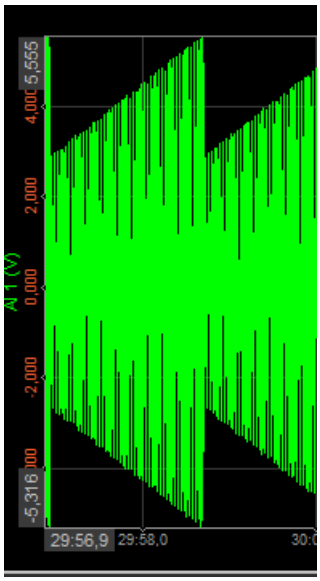


You may enter the values by yourself - simply click on the number at both extreme and then type in a new number.

This is possible when the cursor has following appearance:

A gray outlined box around the scale value indicates the selected channel.

- **Auto scale**



Move cursor to the axis scale.

When you reach the auto scale area, the cursor changes to the following appearance:

A gray outlined box around the scale value indicates the selected channel.

Press left mouse button to activate auto scale for this channel or press right mouse button to undo auto scale.

The auto scale function always calculates the minimum and maximum value of the currently displayed signal and use these values for scaling. The scaling will be only updated when you press the left mouse button!

Additional auto scale functions

Dewesoft offers additional auto scale functions when you press keys together with the left or right mouse button:

- **Left mouse button** - Auto scales the selected channel (see also above).
- **CTRL + Left mouse button** - Selects all available channels and scales all channels to the same maximum and minimum (highest and lowest value within all channels).
- **CTRL + ALT + Left mouse button** - Selects all available channels and scales each channel to its own maximum and minimum.
- **SHIFT + Left mouse button** - Scales the selected channel symmetrical around zero (± values are the same).
- **CTRL + SHIFT + Left mouse button** - Scales all available channels symmetrical around zero (all ± values are the same)
- **CTRL + SHIFT+ ALT + Left mouse button** - Selects all available channels and scales each channel symmetrical around zero (each channel to its own ± value).
- **Right mouse button** - Undo auto scale for the selected channel (see also above).
- **CTRL + Right mouse button** - Selects all available channels for undo auto scale.

Simple measurement cursor functions

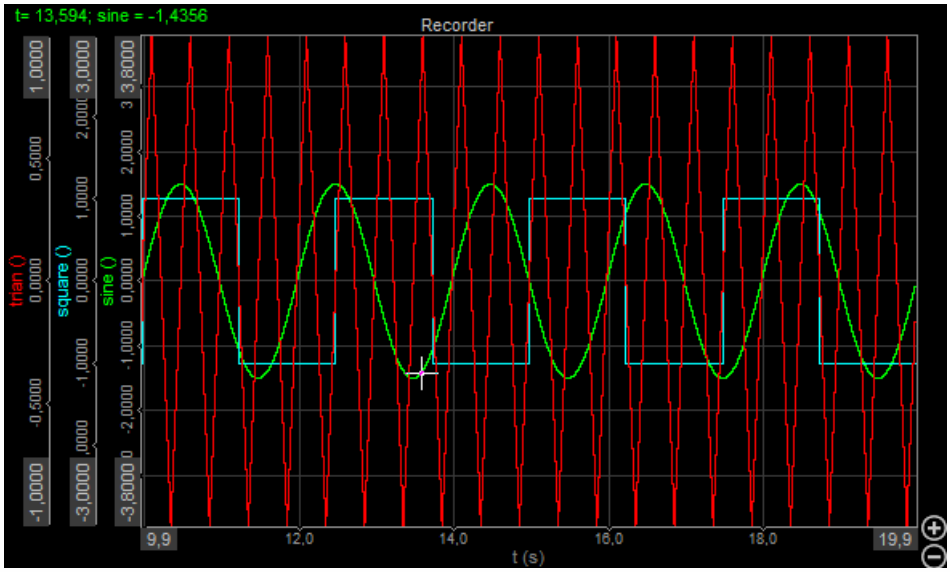
WARNING: **This way displaying X- and Y-axis information can't be done in Design mode -> available only in Run mode.**

If you move the mouse cursor over the signal, a white cross appears, showing the current position of the measurement cursor. As soon as you stop moving the mouse or trackball, the cursor will fix its position within the signal and move with the time. The cursor readout is displayed above the graph and shows:

- **X axis information** - the absolute or relative time at the cursor position, depending on the selected axis type
- **Y axis information** - the signal value at the cursor position, including units

As one graph may contain up to four signals, the measurement cursor always tracks to the signal next to the current position.

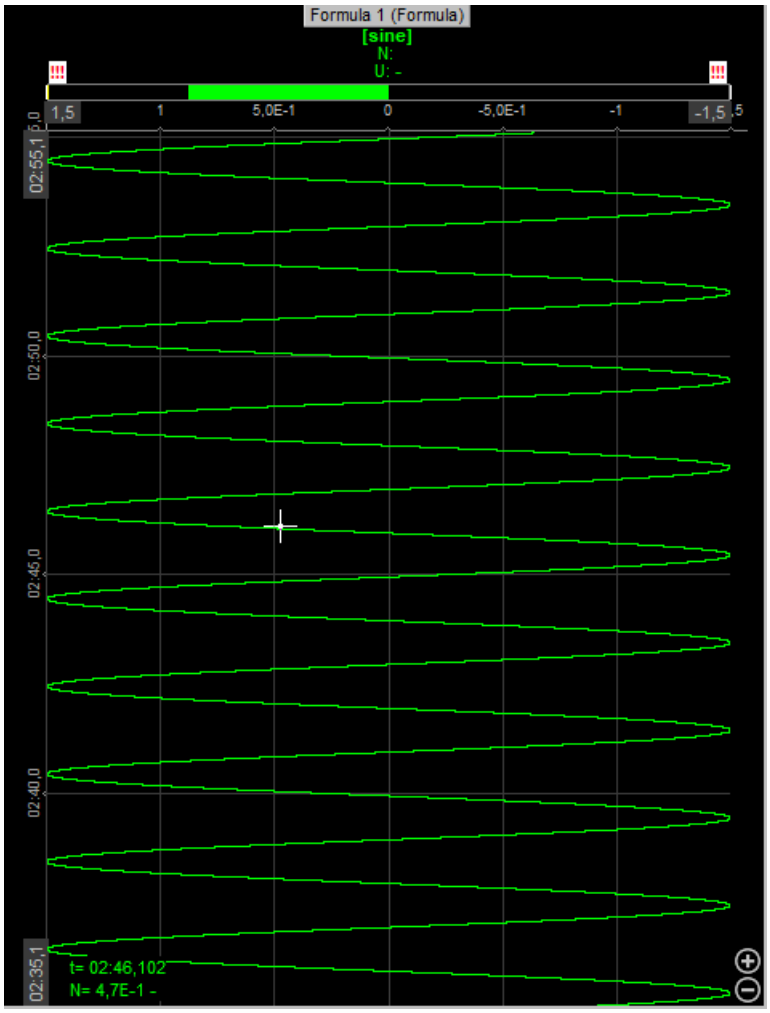
The next display gives an impression of the cursor functionality:



Appearance and functionality by vertical instruments are the same, except:

- Y-axis information is absolute or relative time at the cursor position and X-axis information is the signal value at the cursor position
- the cursor readout is displayed below the graph

The display below gives an impression of the cursor functionality for vertical graph.



WARNING: **The measurement cursor is not available when the time base is set to a short time. It's only working for long time measurements.**

DIGITAL METER

The digital meter is designed to show the current or averaged value of the channel.

When you select a digital meter in the design or run mode, following settings will appear on left and right part of the screen:

- **Control properties**

For detailed information about digital meter Control properties: grouping, number of column, Add / Remove instruments, transparency,... see -> Control properties.

Digital meter settings

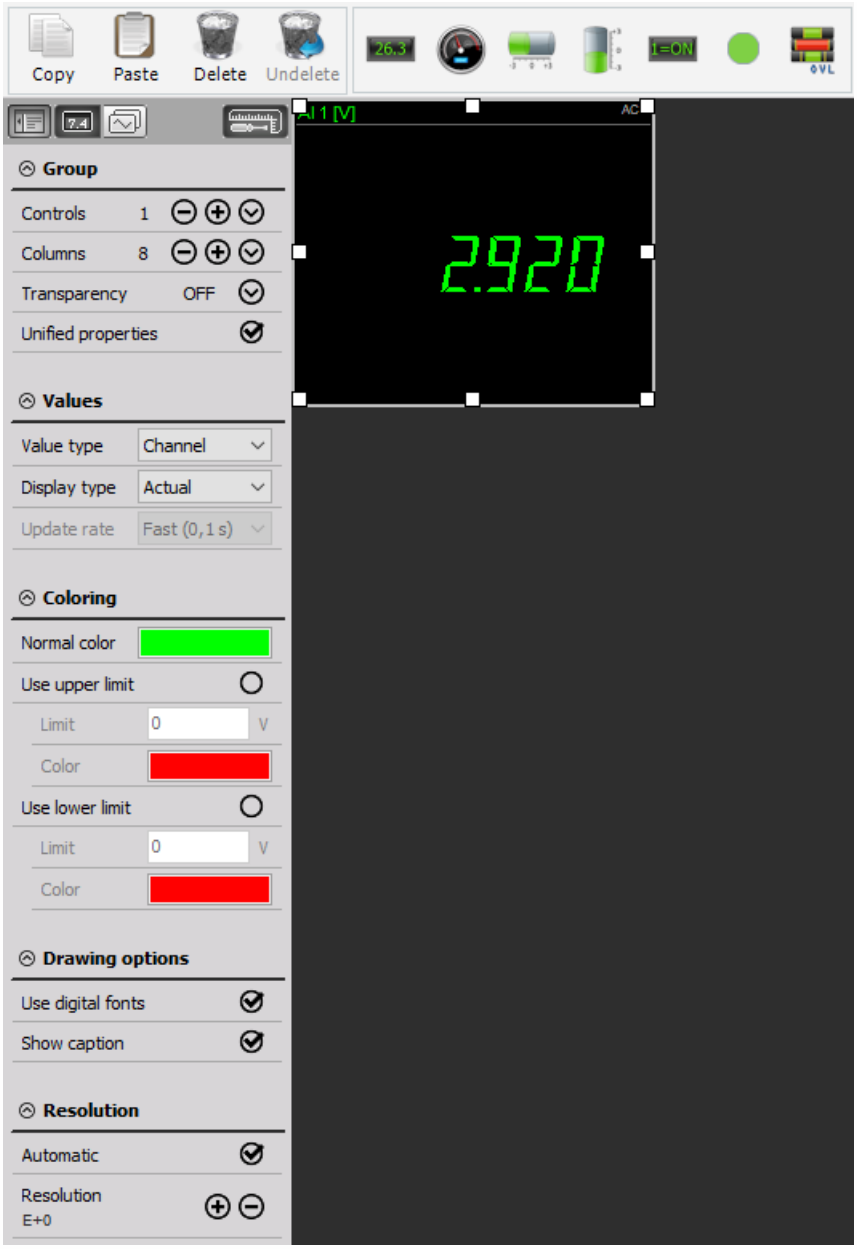
Typical digital meter setting are:

- Display value
- Display type
- Average time
- Upper / Lower limit
- Drawing options
- Resolution

Channels selector

For detailed information about assigning / reassigning channels to/from digital meter -> see -> Display settings.

Example Digital meter in Design mode


Copy
Paste
Delete
Undelete
26.3
1=ON
0 VL
AI 1 [V]
AC
2.920
Group
Controls 1
Columns 8
Transparency OFF
Unified properties
Values
Value type Channel
Display type Actual
Update rate Fast (0,1 s)
Coloring
Normal color
Use upper limit
Limit 0 V
Color
Use lower limit
Limit 0 V
Color
Drawing options
Use digital fonts
Show caption
Resolution
Automatic
Resolution
E+0

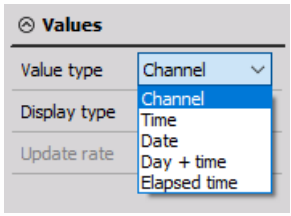
Appearance on screen

The digital meter has only one appearance: the channel name and unit is displayed at the top left, the display type at the top right side. The main space is used by the value letters itself.

For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Setting Display value

The digital meter can display different values. Besides the current measurement value, also timing information can be displayed:

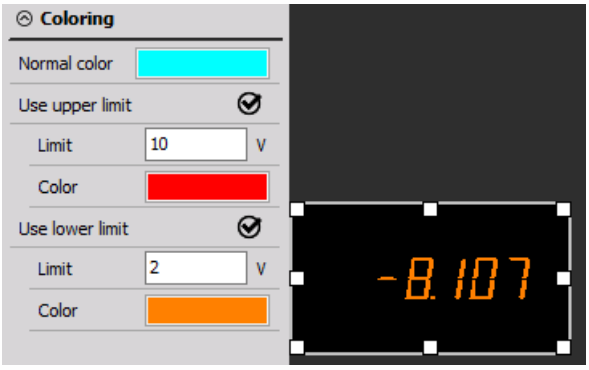


- **Channel** - current measurement value, assigned to an acquisition channel
- **Time** - current time of the acquisition system
- **Date** - current date of the acquisition system
- **Day + time** - number of the day within the current year and time of the acquisition system; this format is compatible with the timing information according to UTC
- **Elapsed time** - time elapsed from the beginning of measurement

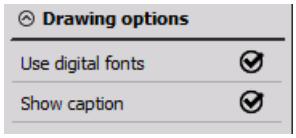
Coloring

Simply click on the Coloring check-box to select the normal color and upper or lower limit.:

- normal color
- upper limit
- lower limit



Drawing options

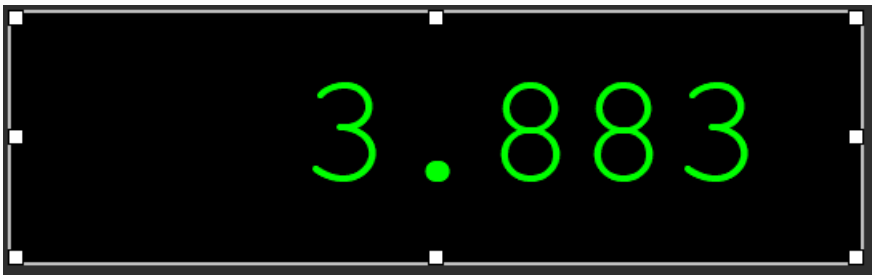


Simply click on the Drawing options check-box to show or hide the selected digital meter:

- caption
- digital fonts

Example caption and frame ON:

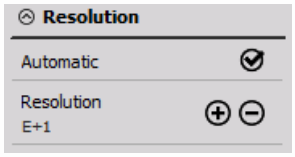
Example caption and frame OFF:



Resolution

Sometimes it is useful to be able to add one or two digital of display resolution or delete one or two, according to the type of data being displayed.

Select a meter by clicking once on it. If Auto is checked, click either Inc(Increase) button or Dec(Decrease) button to add or delete digits to the right of the decimal point of the meter's display.

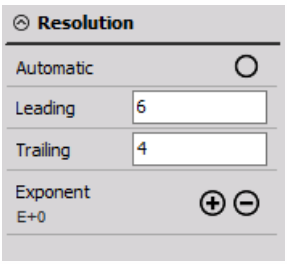


If Automatic Resolution is not checked, Leading and Trailing field is displayed to enter the number of digits. Below this also Exponent section appear.

When Manual Exponent is checked, two buttons appear:

- Plus button to define exponent E in step by +3.

- Minus button to define exponent E in step by -3.



HORIZONTAL / VERTICAL BAR GRAPH

The Horizontal / Vertical bar graph is designed to give a graphical indication of the current value of a signal within the defined display range.

When you select a Horizontal / Vertical bar graph in the design or run mode, following settings will appear on left and right part of the screen:

Control properties

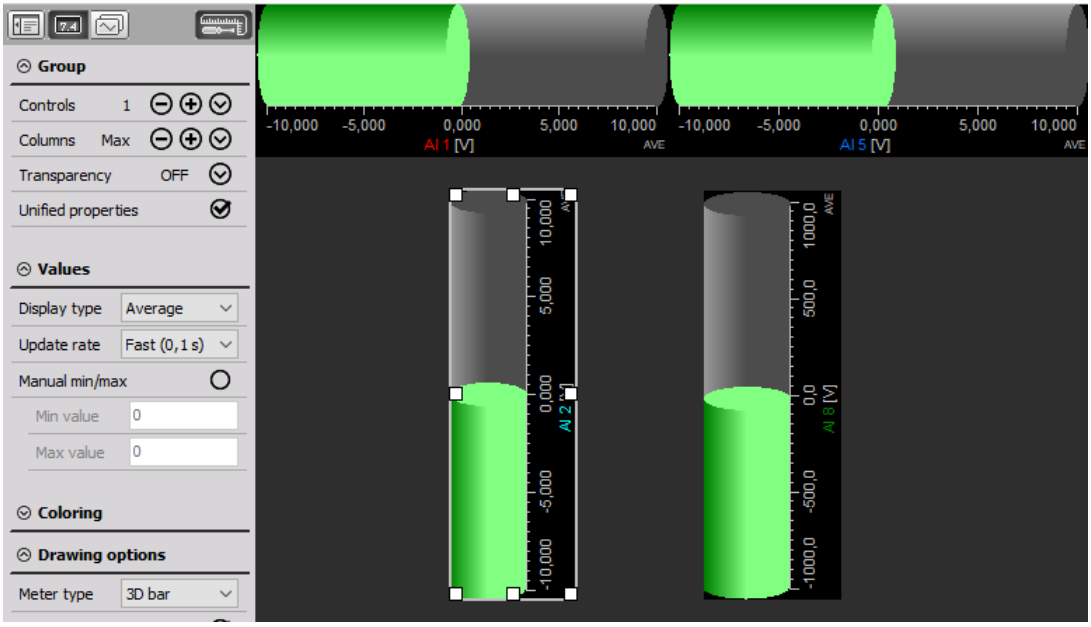
For detailed information about Horizontal / Vertical bar graph Control properties: grouping, number of columns, Add / Remove instruments, transparency,... see -> Control properties.

Horizontal / Vertical bar settings

Typical settings (same for Horizontal and Vertical bar) are:

- Display type
- Meter type
- Average time
- Upper / Lower limit
- Minimum / Maximum shown value
- Channels selector

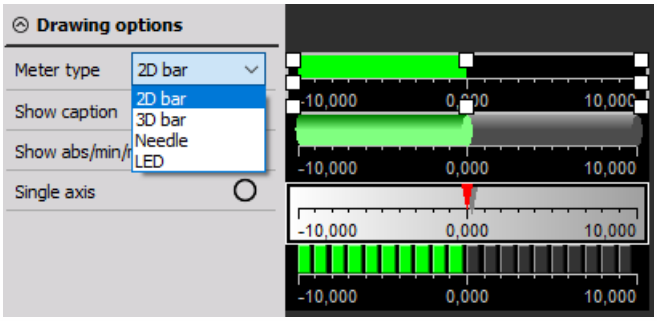
For detailed information about assigning / reassigning channels to Horizontal / Vertical bar -> see -> Display settings.



Meter type & Appearance on screen

The bar graph has four different basic appearances: as standard - Bar, 3D, Needle or LED bar graph and can be selected from the Meter Type drop-down list.

They all contain the channel name, display type and unit beside the measurement values.

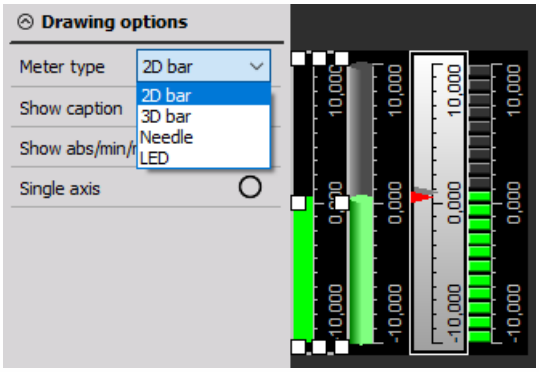


All bar graph basic appearances can be in:

- **horizontal** - Horizontal bar (see above)
- **vertical format** - Vertical bar (see below)

The orientation depends on the instrument type selected by the instrument icon in Design toolbar.

Changing the orientation is not possible; you have to select the right instrument when you click on the icon.



For detailed information about instruments positioning, size and transparency see > Screen edit functions.

ANALOG METER

The Analog meter is designed to give a graphical indication of the current value of a signal within the defined display range.

When you select an Analog meter in the design or run mode, following settings will appear on left and right part of the screen:

Control properties

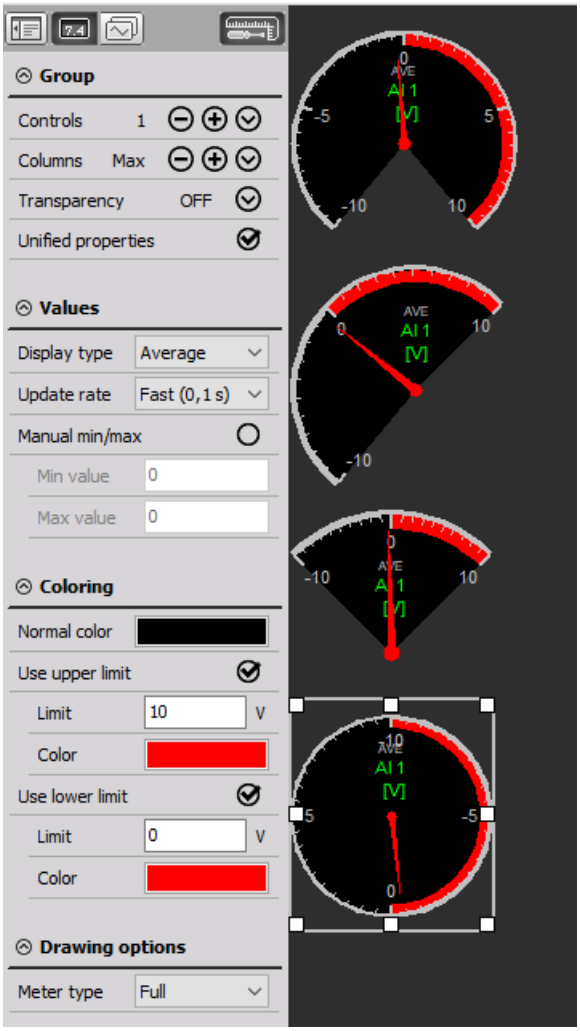
For detailed information about Analog meter Control properties: grouping, number of columns, Add / Remove instruments, transparency,... see -> Control properties.

Analog meter settings

Typical settings for Analog meter are:

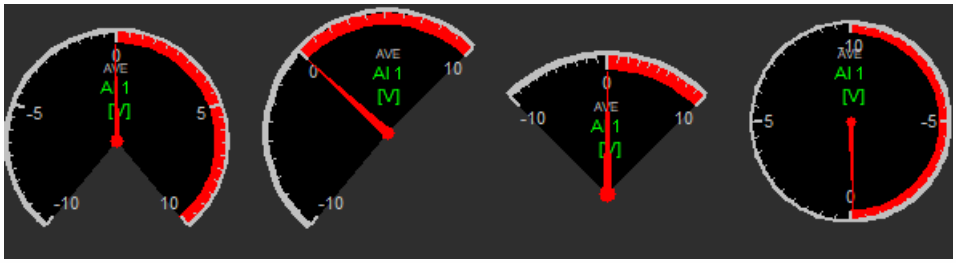
- Display type
- Analog meter type
- Average time
- Upper / Lower limit
- Minimum / Maximum shown value
- Channels selector

For detailed information about assigning / reassigning channels to Analog meter see -> Display settings.


Group
Controls 1
Columns Max
Transparency OFF
Unified properties
Values
Display type Average
Update rate Fast (0,1 s)
Manual min/max
Min value 0
Max value 0
Coloring
Normal color
Use upper limit
Limit 10 V
Color
Use lower limit
Limit 0 V
Color
Drawing options
Meter type Full
AVE
AI 1
[V]
0
-5
5
-10
10

Analog meter type & Appearance on screen

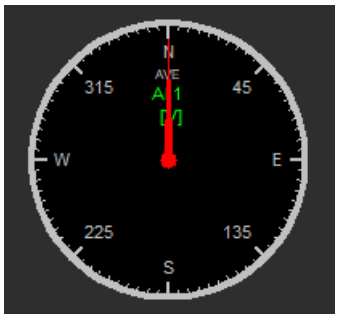
The analog meter offers three different appearances and can be selected from Analog meter type list.



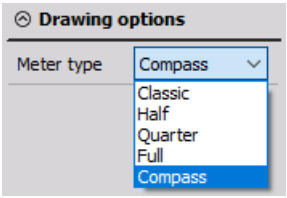
They all contain the channel name, display type and unit beside the measurement values.

Fourth Analog meter has two forms:

- **full circle instrument**
- **compass instrument**



To switch to this form Compass box must be checked:



For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

INDICATOR LAMP

The Indicator lamp indicator lamp is designed to show the state of digital signals, to supervise the value analog signals in a true / false way, but also to show Discrete values.

When you select Indicator lamp in design or run mode, following settings will appear on left and right part of the screen:

Control properties

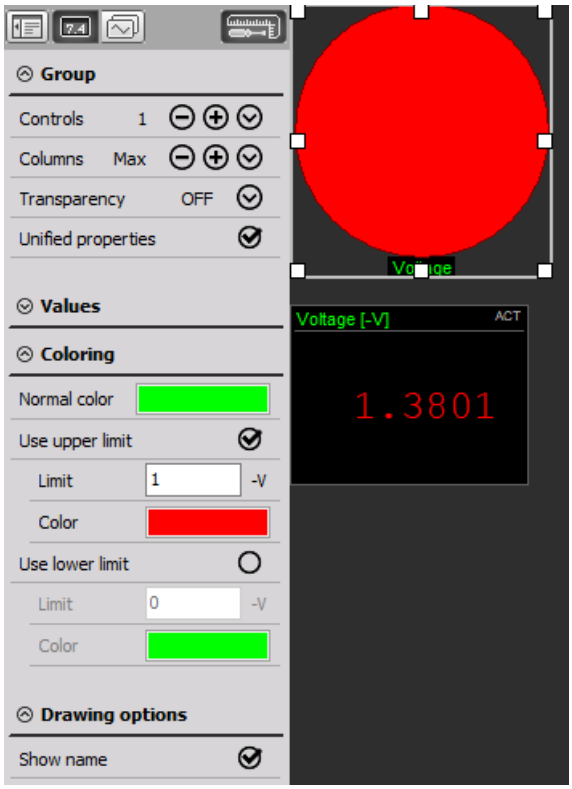
For detailed information about Indicator lamp Control properties: grouping, number of columns, Add / Remove, transparency,...see -> Control properties.

Indicator lamp Display mode settings

Typical setting for Indicator lamp Display mode are:

- Display mode
- Display type
- Average time
- Upper / Lower limit
- Minimum / Maximum shown value
- Channels selector

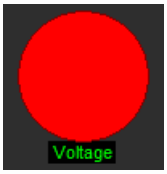
In indicator lamp mode the channels can be assigned only if Display value - Channel option is chosen; for detailed information about assigning/reassigning of channels see -> Display settings.



For example, when the Voltage drops below 1V, the lamp becomes green. As soon as the Voltage grows above 1V, the lamp becomes red.

Appearance on screen

The Indicator lamp is quite a simple element and displays next to the channel name only a colored lamp, depending on the settings.



For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

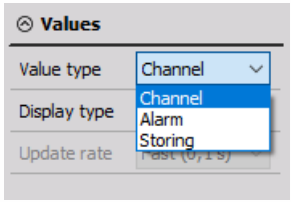
Display mode

Indicator lamp shows the state of digital signals and to supervise the value of an analog signal in a true/false way.

Indicator lamp Values

Possible Indicator lamp settings depend from selection in this field.

For Indicator lamp you can select three basic Value types:



1. Channel

- show the state of digital signals
- to supervise the value of analog signal current measurement value assigned to an acquisition channel in a true/false way

Three colors can be chosen - below first limit, in between limits and above second limit.

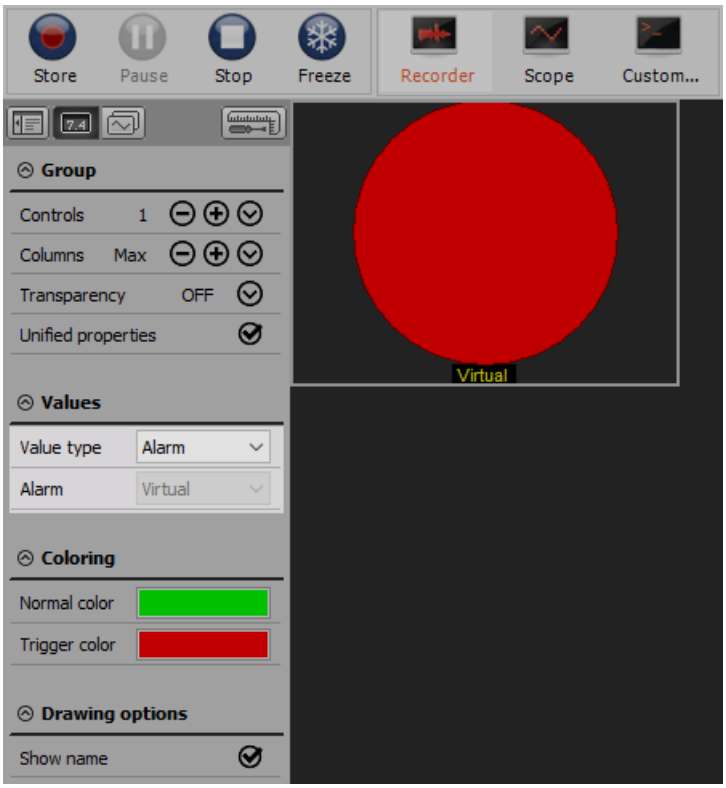
For Channel option you be able to set (see picture above):

- Display type
- Average time
- Upper / Lower limit

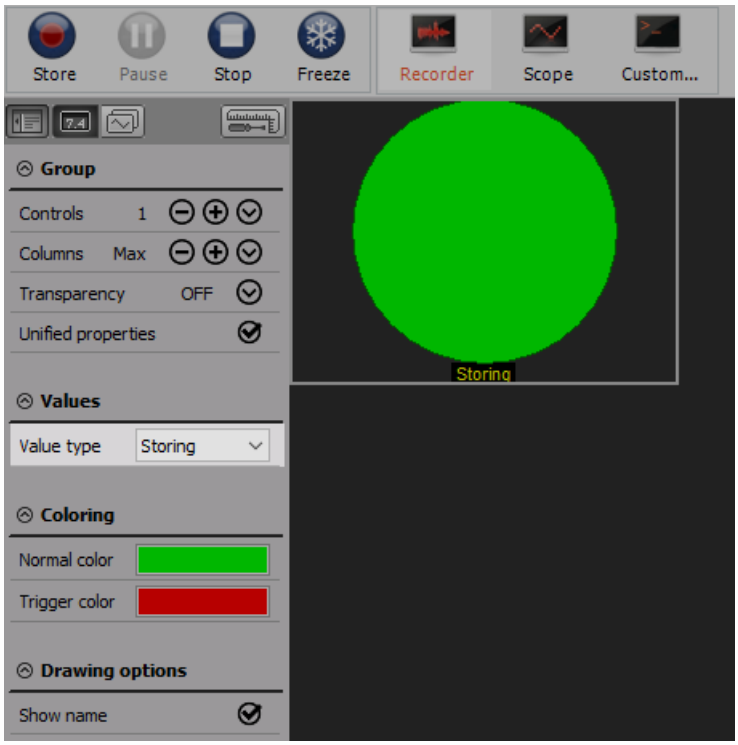
2. Alarm will show the state of alarm signals

For Alarm option you be able to set: Alarm from list; alarms are defined in Channel setup -> Alarms tab which is visible if alarm option is selected in channels setup:

- Display type
- Average time



3. Storing show the state of storing data. If the data is stored, the lamp will go red, otherwise, it will stay green



DISCRETE DISPLAY

The Discrete Display is designed to show defined discrete value, to supervise the value analog signals in a true / false way.

Typical setting for Discrete Display Display mode are:

- Discrete display
- Display mode
- Discrete values
- Options

Discrete Display

Each channel can have a set of predefined values - for example a binary CAN channel for ABS can have three states with codes 0, 1 and 2 which represents ON, OFF and ERROR.

Discrete Display mode setting

The Discrete Display instrument show defined discrete value.

For Discrete display mode you be able to set (see picture on right):

- Values
- Drawing options
- Discrete values

Let's take few CAN channels which have defined discrete channels. There are few channels which describe the bit value if the car is running in idle (MO1_Leergas) and if the clutch is pressed (MO1_Kup_schalt). The channels have defined discrete values which are loaded from the DBC library.

--- CAN ---			
4		Motor1	Setup
5		IDLE	Setup
6		CLUTCH	Setup
7		ENG_TORQUE	Setup
8		ENG_RPM	Setup
9		WANTED_TORQUE	Setup
10		Motor2	Setup

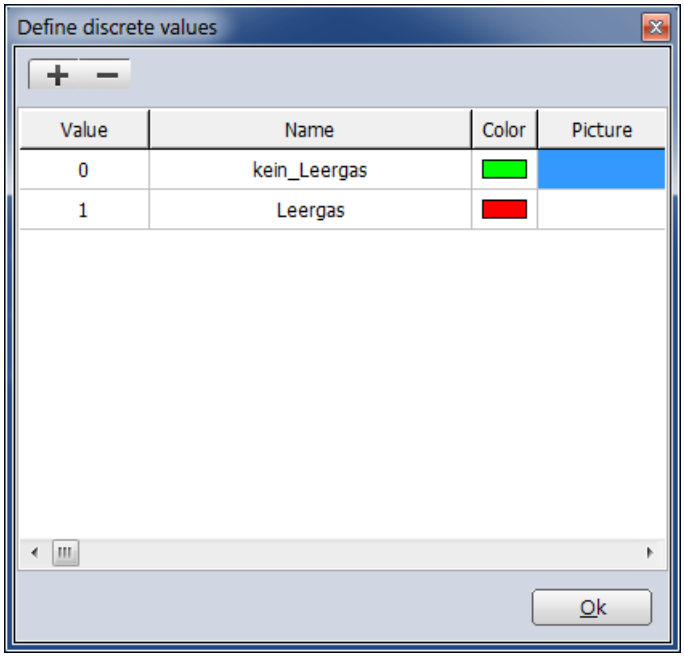
Now let's create a display with few controls to monitor the position of the clutch and gas pedal. So let's create a discrete display and put the MO1_Leergas in it. Since discrete values for this channel is already defined, we see already the status of the channel. The bar and recorder show the values of the status and gas pedal position channels.

The first picture shows the acceleration:

The second picture shows the gear switching:

Define Discrete values

If the discrete values are not defined, we can define them for selected channels with pressing Define button. Define discrete values window will appear:



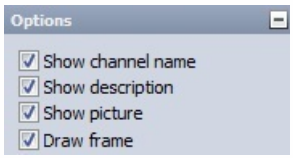
In this window you can define:

- Value - numerical code for each state
- Caption - description of each state
- Color - color of the certain state
- Picture - after click on this field Load graphics window appear to define .bmp or .jpg picture. The picture will be shown at the certain code in discrete display (a green lamp for on and a red lamp for off, for example).

With the Plus button you can add another value, and with the button delete value (table row).

Options

With checking appropriate box you can change Discrete Display appearance on online screen.



All options switched on.

Show channel name option shows the channel name caption. If it is switched off, it will not display the caption above.

Draw frame draws the frame around the control. This option is useful especially when 'Show channel name' is not checked to only display a discrete value.

Show picture option switches on and off the display of the picture, if they are available.

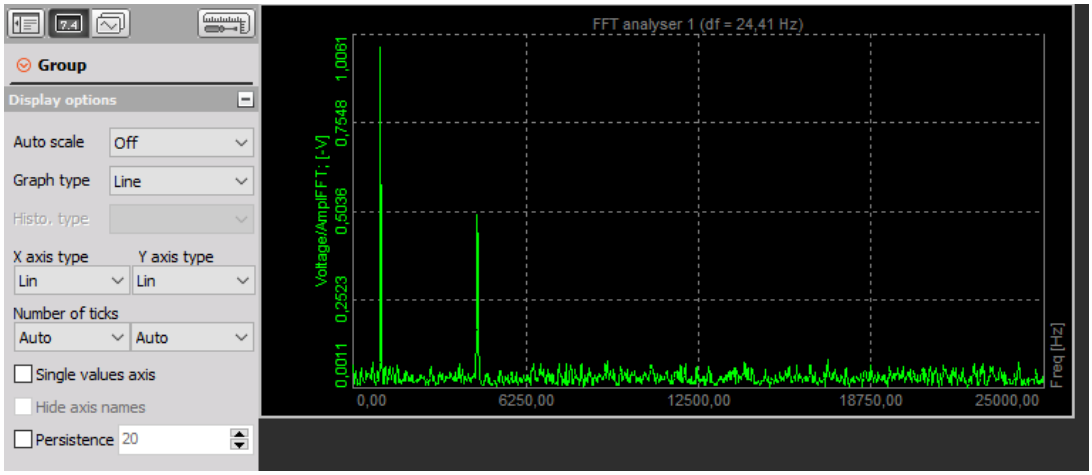
Show description option switches on and off the description of the state. If it is switched off, only the bitmap with state image will be shown.

2D GRAPH

The Dewesoft 2D graph shows the drawing of any vector channels. Some typical examples are FFT created from math channels, classification and others.

When you select 2D graph, the following settings will appear on left and right part of the screen:

- Control properties - For detailed information about Control properties: grouping, number of column, Add / Remove, transparency.
- Drawing options - Selects which parameters will be shown on the graph
- Channels selector



The input to the 2D graph can be:

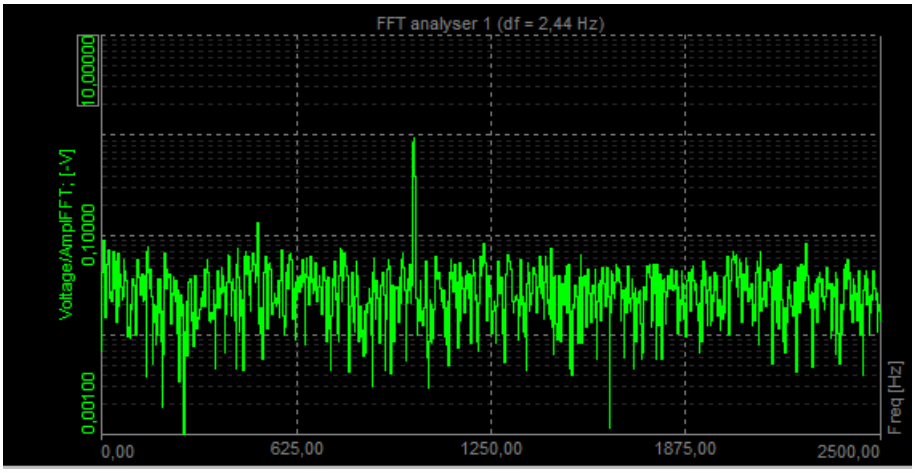
- FFT math
- STFT math
- CPB math
- classification
- counting
- scope trigger
- FRF math
- SRS math
- CA pressure and other channels

In short, 2D graph can show any array channel created by Dewesoft.

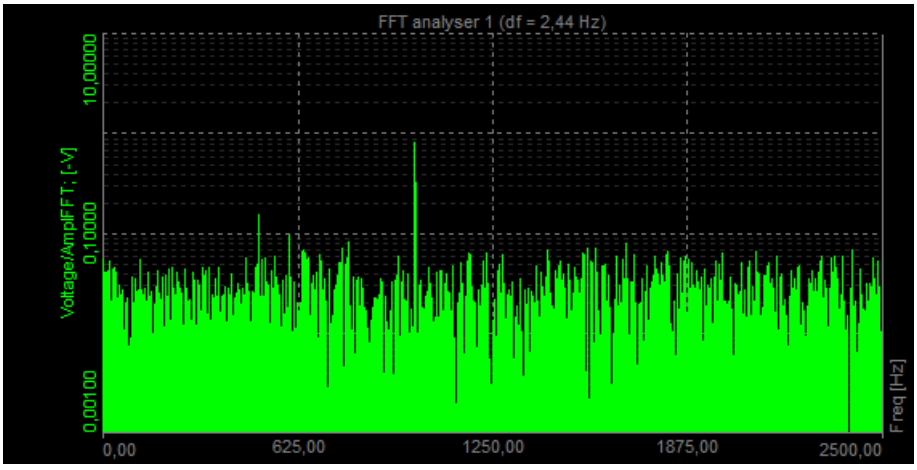
Properties

There are several properties which can be set to 2D graph.

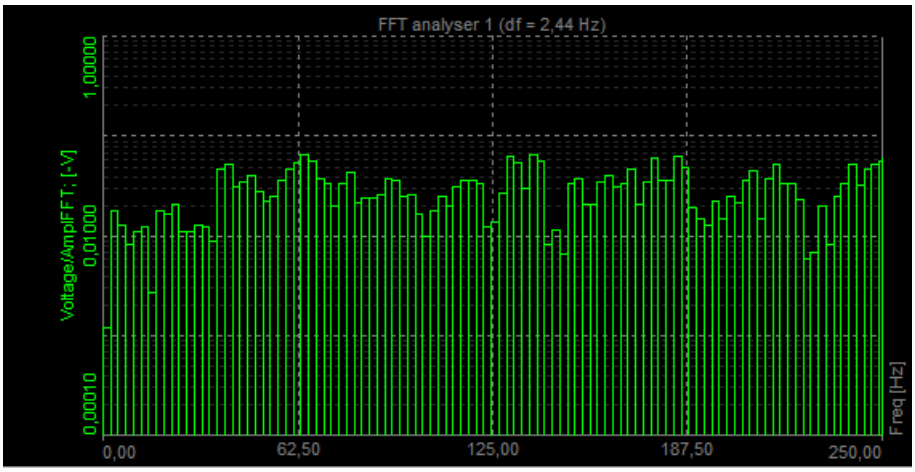
- **Auto-scale** - will automatically scale y axis
- **Graph type** - Automatic will set the graph type to what is set in the input channel. For example, FFT has the default graph type of lines while CPB has histogram. We can override these settings by manually defining either Lines or Histogram.



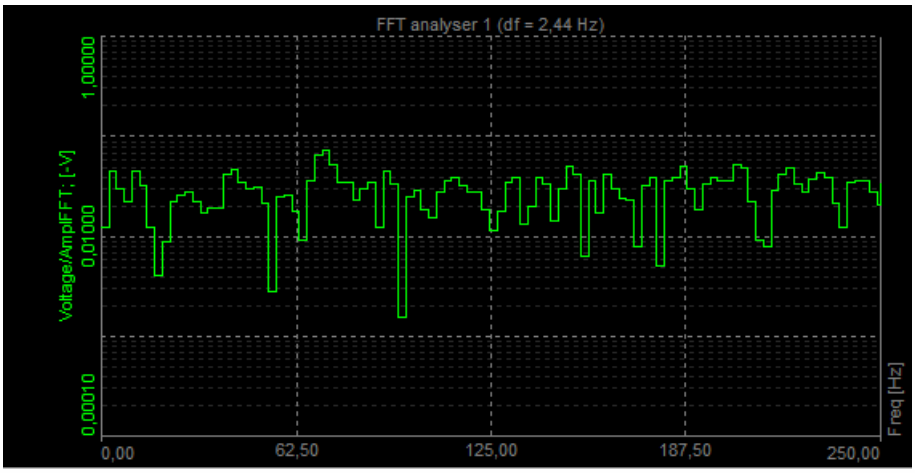
- **Histogram type** - for histogram type, we can define to either fill the bars with Full option, or to draw empty bars with Empty option or to simply draw the Line at the top for a very classical instrument look. Full histogram:



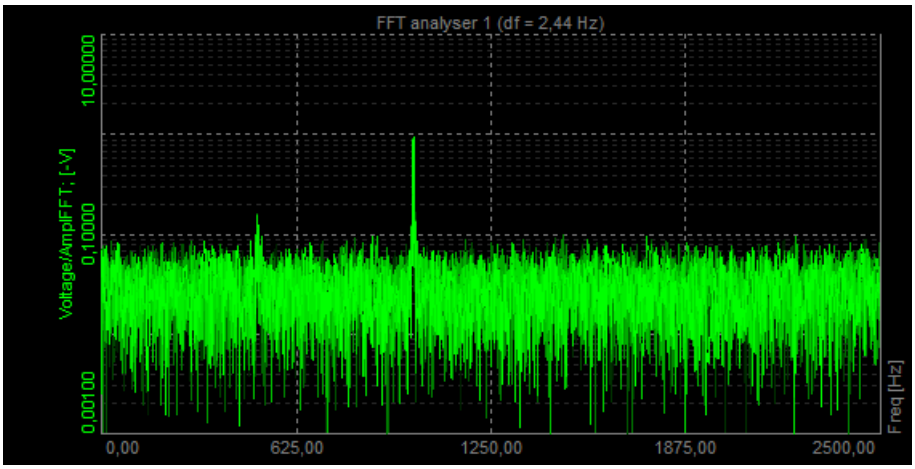
Empty histogram:



Line histogram:

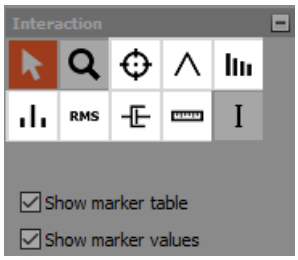


- **X Axis type** - can be either linear or logarithmic
- **Y Axis type** - can be either linear, logarithmic, Amp dB where the 0 dB is the full scale or Power dB
- **Number of ticks** - defines either automatic or manual number of graph divisions for x and y axis. Division for y axis can be freely defined only for linear scaling, log scaling defines number of ticks from minimum and maximum axis value.
- **Single value axis** - will set one y axis scale for all channels in the graph
- **Persistence** - will slowly fade the old data on the graph. We can define number of old arrays to be shown. The larger the number, more history will be seen. Persistence graph:


FFT analyser 1 (df = 2,44 Hz)
10,00000
0,10000
0,00100
Voltage/AmplFFT; [-V]
0,00
625,00
1250,00
1875,00
2500,00
Freq [Hz]

Interaction

The 2D graph can display values of the currently selected point with the markers. When clicking on such point with the left mouse button, the marker line will be added showing x-axis value on the x-axis and showing y-axis value of certain point above the marked point. All points can be removed by pressing the right mouse button.



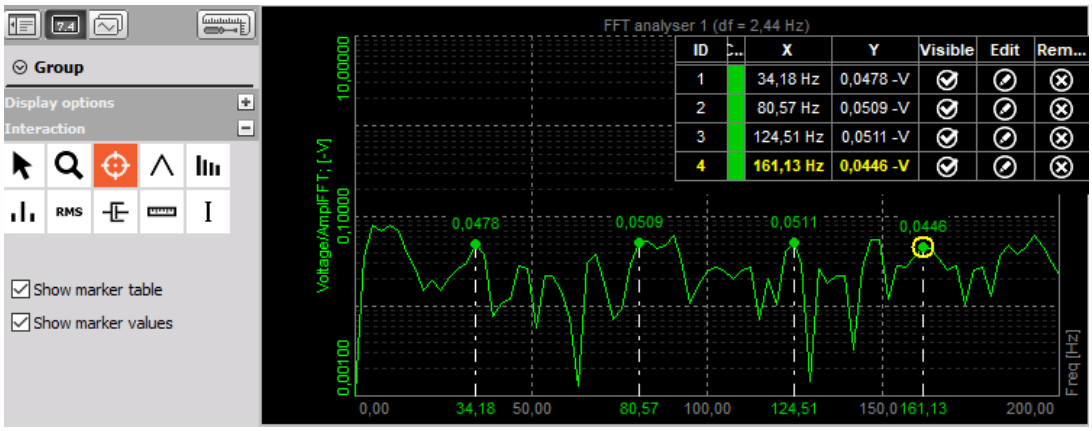
The type of markers:

- selection
- zoom
- free marker
- max marker
- harmonic marker
- sideband marker
- RMS marker
- damping marker
- cursor channel
- bearing marker

Free marker

Select an icon for Free marker and click on the FFT graph to add it.

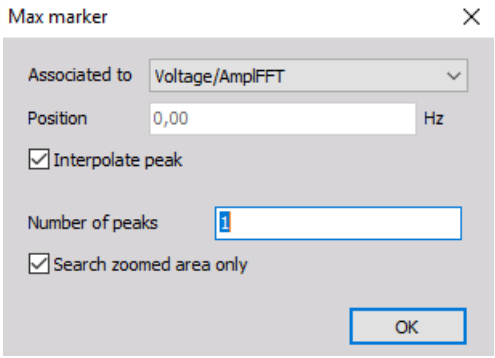
Free markers can be freely added. Marker shows us frequency of the peak at which it stands and its amplitude. With Show marker table selected you can see the table of markers - its ID, type, channel, color, its frequency (X-axis) and its amplitude (Y-axis). You can select if you want markers to be visible or not, you can also edit and remove it.



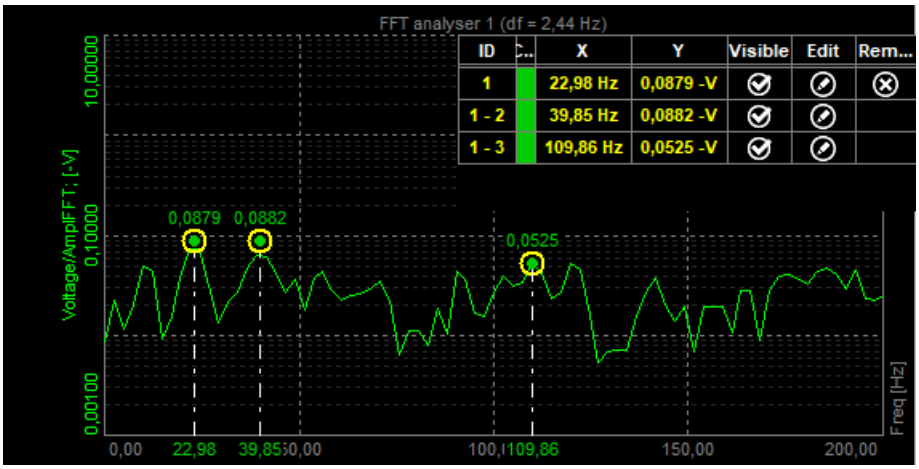
Max marker

Max marker finds the highest amplitude in the spectrum. Move the mouse to FFT graph and select icon for Max marker.

When we select Max marker and click on the FFT graph with the left mouse button the following setup opens.



First, we select the FFT curve to which the marker is related to. The position is calculated by the program. Depending on the selected window type, the frequency component (actual peak) can fall in between two adjacent lines. Then we select a number of peaks we want to find. If that number is 1 program will only find the peak with maximum amplitude. If that number is 2 it will find also the peak with second highest amplitude. The picture below shows the max marker with 2 number of peaks selected.



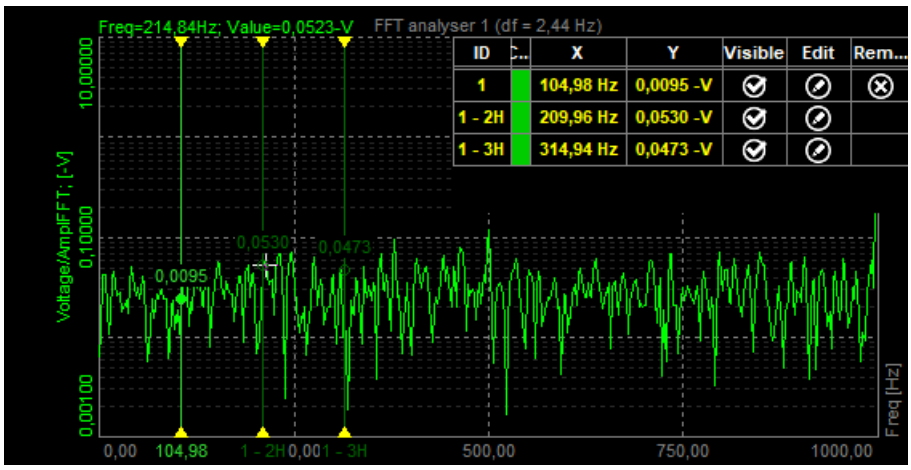
Harmonic marker

The harmonic marker is a great help when identifying the fundamentals of the frequency.

The harmonic marker can be enabled at any frequency. The harmonic marker will mark the harmonics of the selected frequency. The base marker of the harmonic marker can be selected and moved to any other frequency with the harmonics updated live.

Move the mouse to FFT graph and select the icon for a Harmonic marker. Then select the base frequency with the mouse and add a harmonic marker with the click on the left button.

We select the first peak at 104.98 Hz. If we select Number of harmonics as 3, we will see lines at 104.98 Hz, 209.96 Hz (2 x 104.98 Hz) and at 314.94 Hz (3 x 104.98 Hz). And the theoretical harmonics also nicely match with our measurement results - first three harmonics are nicely seen.



You can also pick and drag the fundamental frequency through the FFT spectrum. Harmonics will automatically follow.

Sideband marker

Sideband marker monitors the modulated frequencies to the left and right from the selected center line.

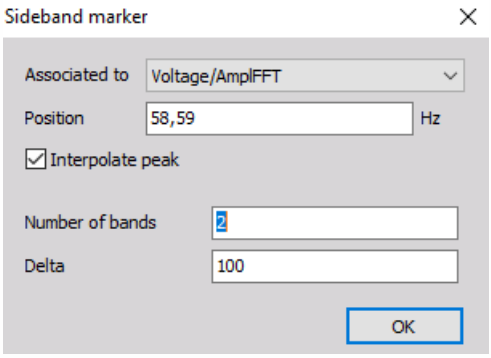
Let's generate an amplitude modulated signal with a carrier frequency of 1000 Hz and the base-band signal with a frequency of 100 Hz.

Sideband markers have a center marker and several equally spaced sideband markers. By selecting the center marker, you can drag the sideband markers to different positions while still maintaining the individual sideband space.

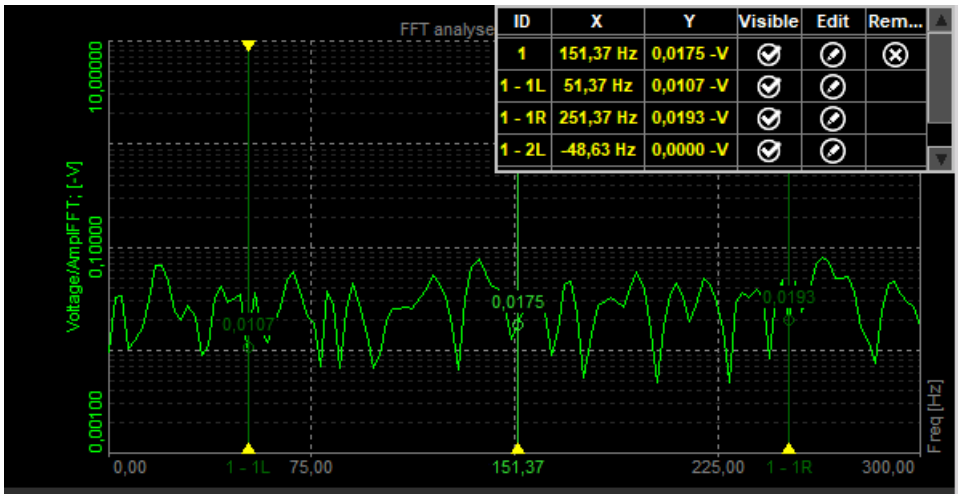
Each sideband cursor can be selected and moved to a different frequency hence changing the individual ratio of the side bands with respect to that of the center cursor.

On the FFT, graph select the icon for Sideband marker:

Sideband marker draws markers around the selected peak. We have to define the Number of bands (for how many bands in each direction we want to see drawn lines) and Delta (distance between bands in Hz). For example, selected position is 1000 Hz, a number of bands is 1 and Delta frequency is 100.



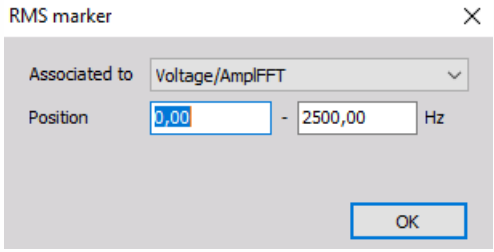
We can see that the central position is at 1000 Hz and we have one band in each direction. So the line on the left side is at 900 Hz and line on the right side is at 1100 Hz. Distance between the lines can be defined by the user, in our example it was 100 Hz.



RMS marker

RMS marker will sum up all the FFT lines in the selected band and calculates the RMS value. Move the mouse to FFT graph and select the icon for RMS marker.

RMS marker calculates RMS value of the channel between cursors or between defined area.

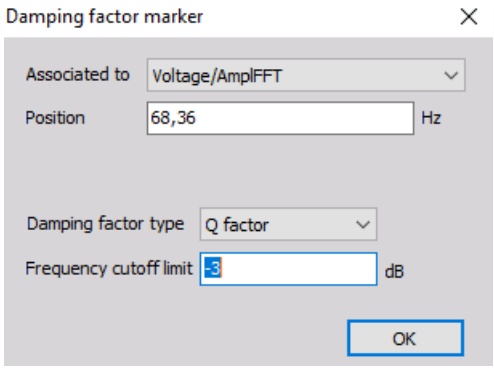


The RMS value of the channel between cursors can also be adjusted by dragging cursor with a mouse. RMS will be calculated automatically if the area changes.

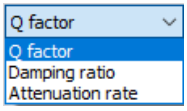
Damping marker

Damping markers are best to use in modal testing when we want to find out how our transfer curve is damped. We select it when we are interested in the quality factor, damping ration or attenuation rate of a selected peak.

Move the mouse to FFT graph and select the icon for Damping marker. Then click on the mouse button to the position, where you want to add a damping marker.

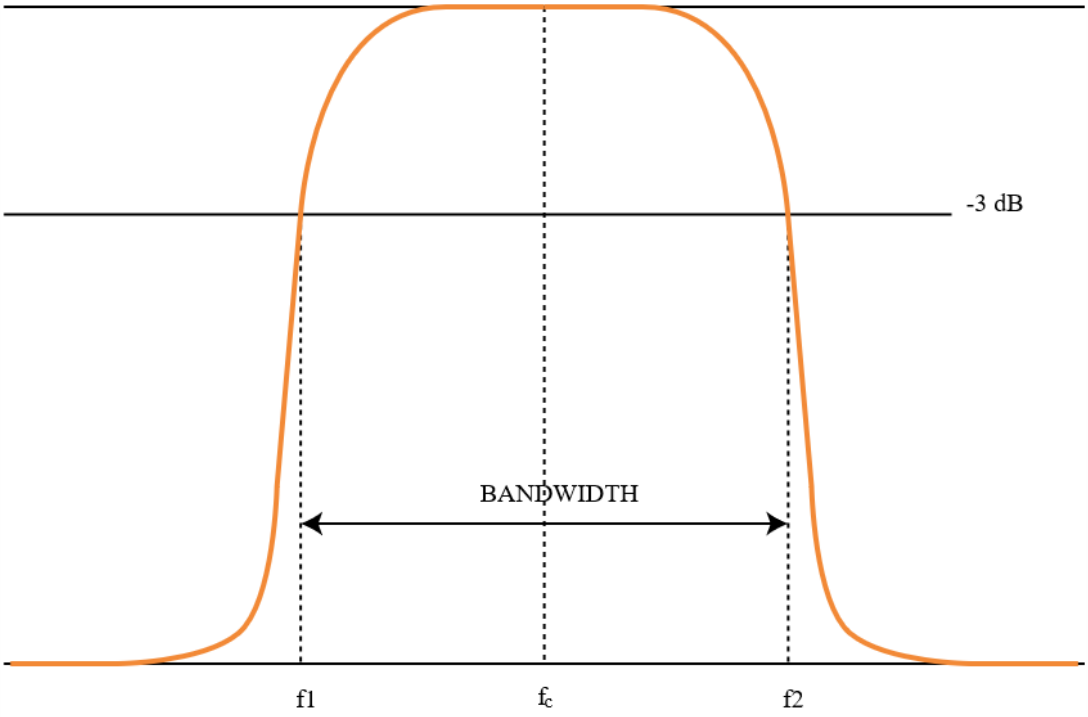


When selecting the damping marker the following setup appears:



Damping factor type can be selected from the following options:

- **Q factor**



The Q (quality) factor of the damped system is defined as:

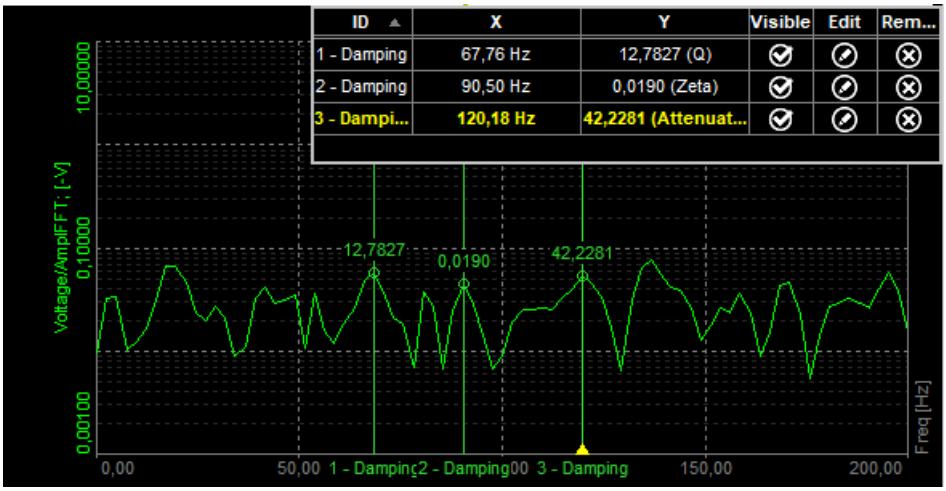
$$Q = \frac{f_C}{f_2 - f_1} = \frac{f_C}{\Delta f}$$

The higher the Q, the narrower and 'sharper' the peak is.

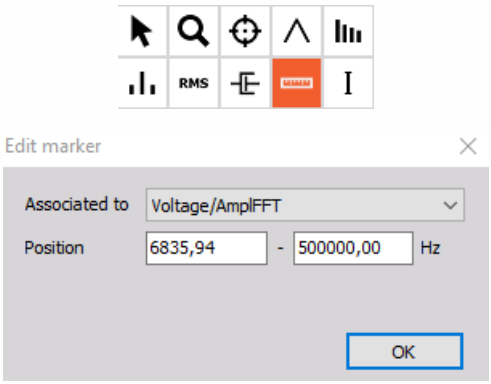
- **Damping ratio** - Damping ratio and quality factor Q are related through equation:

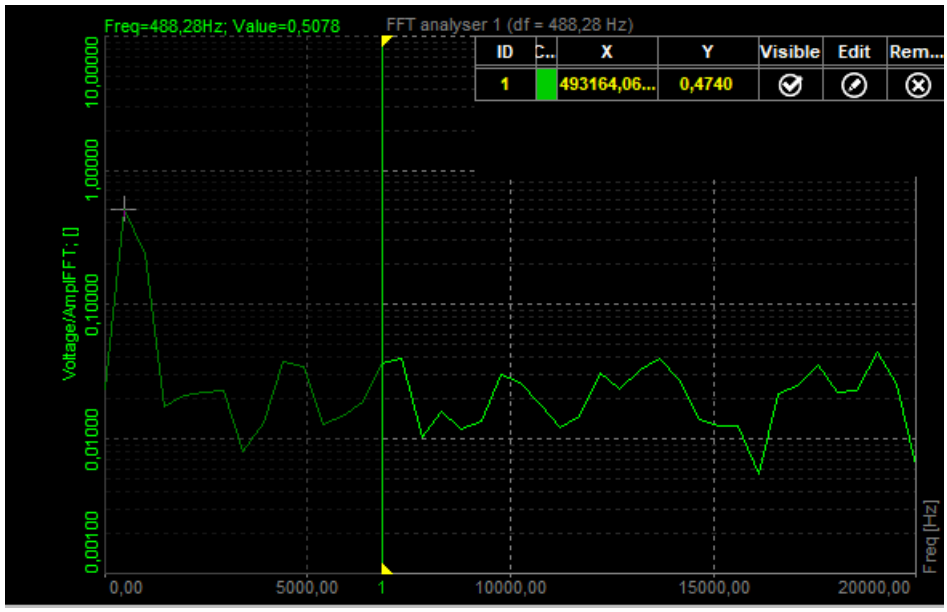
$$\zeta = \frac{1}{2Q}$$

- **Attenuation rate** - Attenuation is the gradual loss in intensity of any kind of flux through a medium. It is usually measured in units of decibels per unit length of the medium. In the picture below we can see a transfer curve of a beam. On each of the peak, we attach a damping factor and in the marker table we can see the quality factor (Q), which tells us, how much the transfer curve is damped. If Damping factor type is chosen as Damping ratio, the result is Zeta for each peak. If Damping factor type is chosen as Attenuation, the esult is attenuation ratio for each peak.



Delta marker



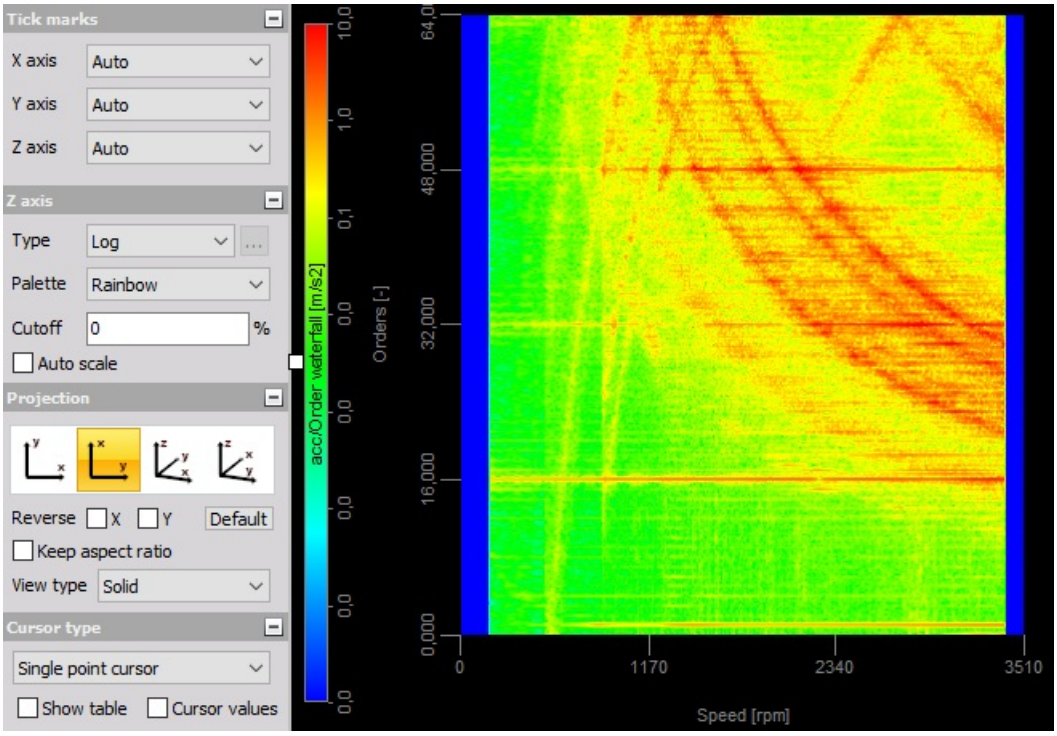

Freq=488,28Hz; Value=0,5078
FFT analyser 1 (df = 488,28 Hz)
ID
C...
X
Y
Visible
Edit
Rem...
1
493164,06...
0,4740
10,00000
1,00000
0,10000
0,01000
0,00100
Voltage/AmpFFT; []
0,00
5000,00
1
10000,00
15000,00
20000,00
Freq [Hz]

3D GRAPH

The Dewesoft 3D graph shows three-dimensional arrays or arrays with history. With this graph we can show FFT history, order tracking, rainflow count and even thermo vision data.

When you select 3D graph, following settings will appear on left and right part of the screen:

- Control properties - For detailed information about Control properties: grouping, number of column, Add / Remove, transparency.
- Drawing options - Selects which parameters will be shown on the graph
- Channels selector



The inputs to the 3D graph could be:

- block based FFT math
- STFT math
- block based CPB math
- order tracking order and frequency based history
- 3D rainflow counting
- FLIR thermo vision picture (requires special plug-in).

Please also note that 3D view might not be available on computers which doesn't have DirectX installed or the graphics card doesn't support 3D features required from graph.

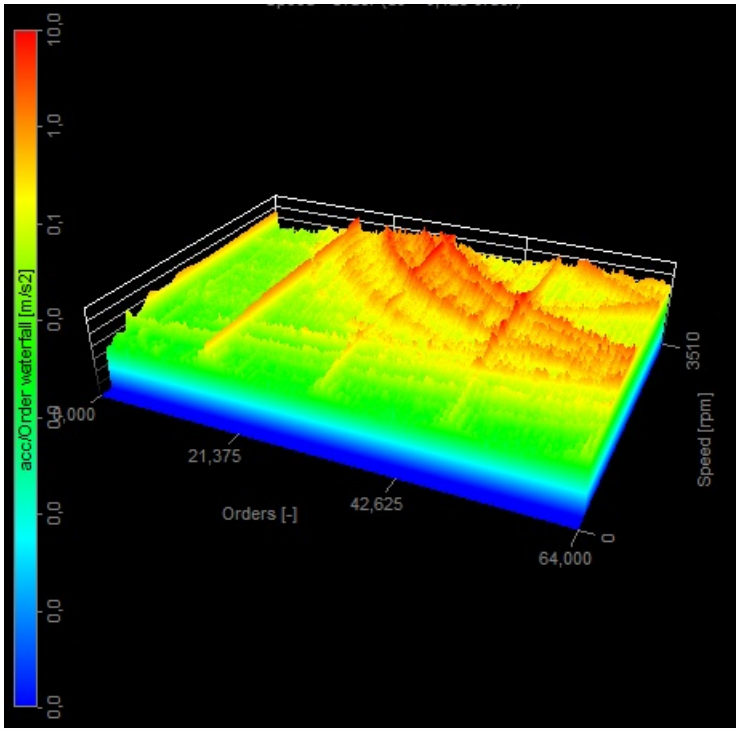
Properties

- Auto scale will automatically scale z axis.
- History count defined number of lines which will be shown on the display. Number of lines depends on the amount of memory reserved by the channel. Please note that increasing the value might result in very slow display since it requires lots of computing performance.
- Z axis type can be set to either logarithmic or linear. Minimum and maximum of each scale can be defined by clicking on the min and max value like in any graph. That also works for the z scale which is on the left side of the display.
- Z axis palette can be either in rainbow or grayscale color.
- The Projection of the axis can be changed. First icon (x up, y right) is planar view and is mostly used when time-based data is shown like FFT history, for example. The second one (x left, y up) is useful when displaying matrix channels like rainflow count or thermo vision picture. Order tracking is in between, some users prefer first while other prefer the second option. There are also two three dimensional views.
- Three-dimensional view can be rotated by pressing and holding the left mouse button to rotate it. Scrolling the mouse wheel or pressing Shift and left button will zoom in or out the display when moving the mouse up and down. Right-click and moving the mouse will rotate the graph around the display plane.

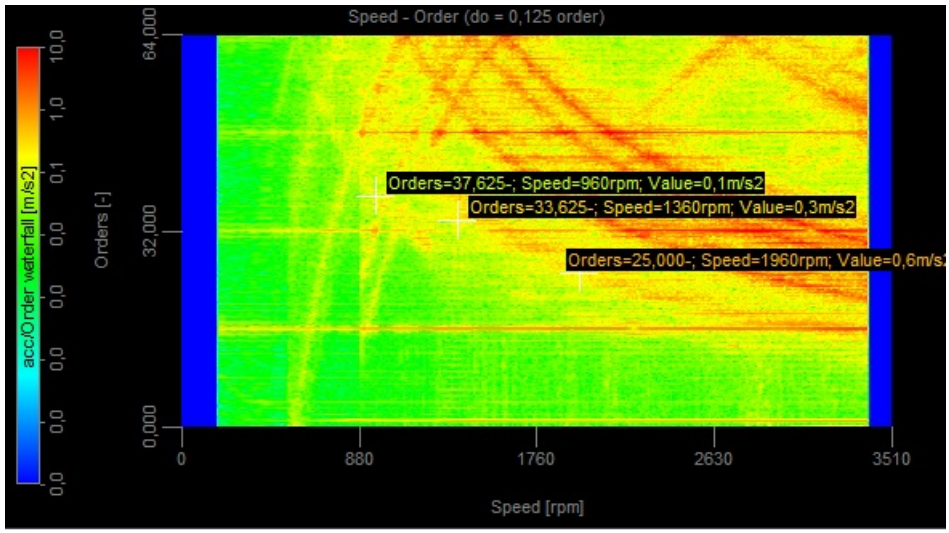
Cursors

Moving the mouse button around on the graph will place the crosshair cursor on the nearest point on the graph.

When clicking on the point for a short period of time (long click will rotate the display), the crosshair will be held in place and the value of all three axis will be displayed near the cursor. All the cursors can be removed with right mouse click.



Additionally, we can change the cursor to calculate slope between x and y axis. First we click on the first point, then on the second point and the value on the left will show (in our case) speed of frequency change over the time.



Cursor table

Orders [-]	Speed [rpm]	Value [m/s2]	Remove
33,63	1360,00	0,31	Remove
37,63	960,00	0,08	Remove
25,00	1960,00	0,59	Remove

RECORDER

The Recorder instrument is designed to show the time-history like the traditional strip chart recorder,but with enhanced display and analysis capabilities.

When you select Recorder instrument in the design or run mode, following settings will appear on left and right part of the screen:

Control properties

For detailed information about Recorder Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.

Recorder settings

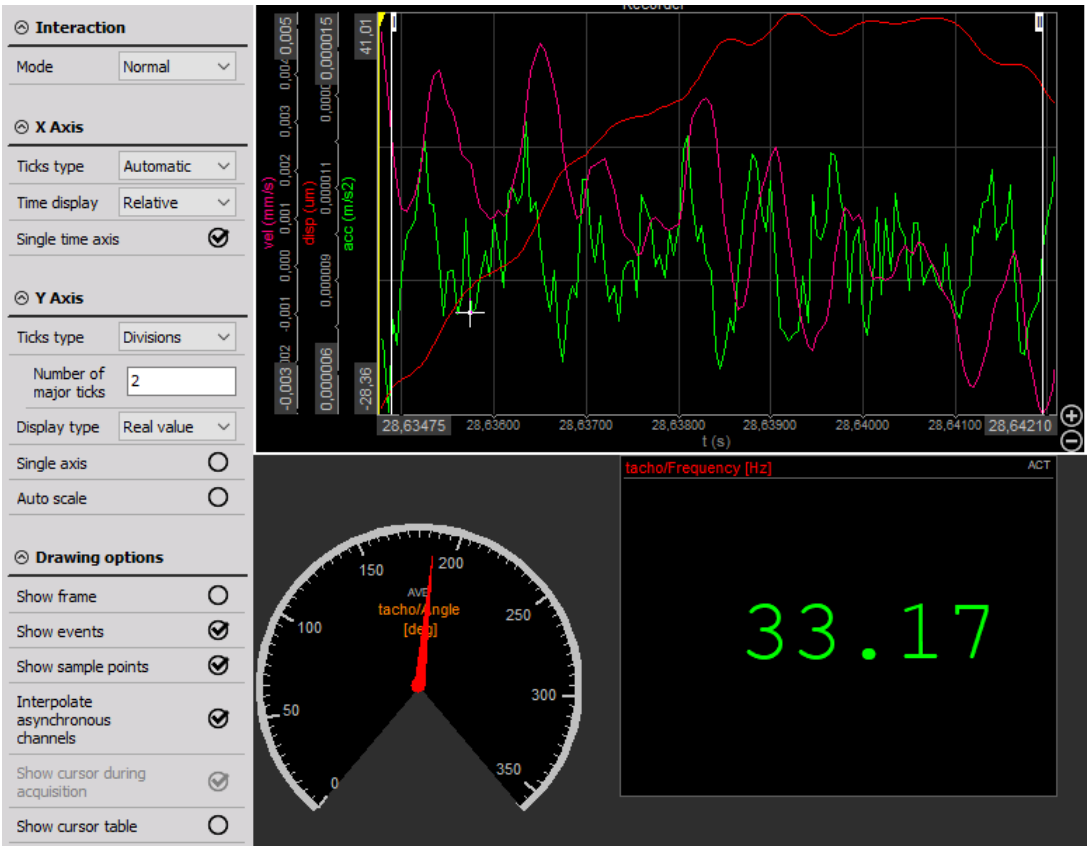
Typical Recorder setting are:

- Single/Multiple Time axis
- Display type
- Time axis type
- Y scale divisions
- Single value axis

Available appearance setting for Recorder instrument in Run mode are:

- Time axis scaling
- X-/Y-axis information
- Y-axis scale/auto scale
- Channels selector

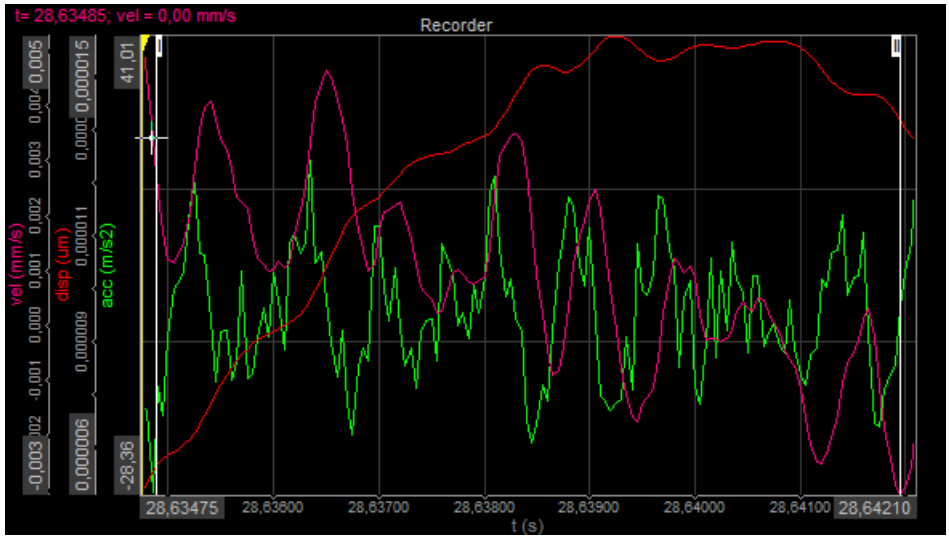
For detailed information about assigning / reassigning channels to/from Recorder -> see -> Display settings.


Interaction
Mode
Normal
X Axis
Ticks type
Automatic
Time display
Relative
Single time axis
Y Axis
Ticks type
Divisions
Number of major ticks
2
Display type
Real value
Single axis
Auto scale
Drawing options
Show frame
Show events
Show sample points
Interpolate asynchronous channels
Show cursor during acquisition
Show cursor table
vel (mm/s)
disp (um)
acc (m/s2)
41,01
-28,36
28,63475
28,63600
28,63700
28,63800
28,63900
28,64000
28,64100
28,64210
t (s)
tacho/Frequency [Hz]
ACT
33.17
AVE
tacho/Angle [deg]
0
50
100
150
200
250
300
350

Appearance on screen

The Recorder element offers all important information:

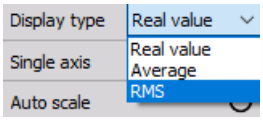
- channel name(s)
- unit(s)
- time information
- zoom functions…



For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Display type

Because the Recorder is typically used to show longer periods of time - minutes or even hours or days - it has the added ability to show data in one of several fashions: Real Data, RMS, or Average.



This is useful when monitoring AC signals, which are going to look like a solid band when a long duration is shown - not very useful to look at (unless you're just looking for overall amplitude envelopes or obvious drop-outs). Changing the display type for this graph to RMS will show a more useful representation of the data.

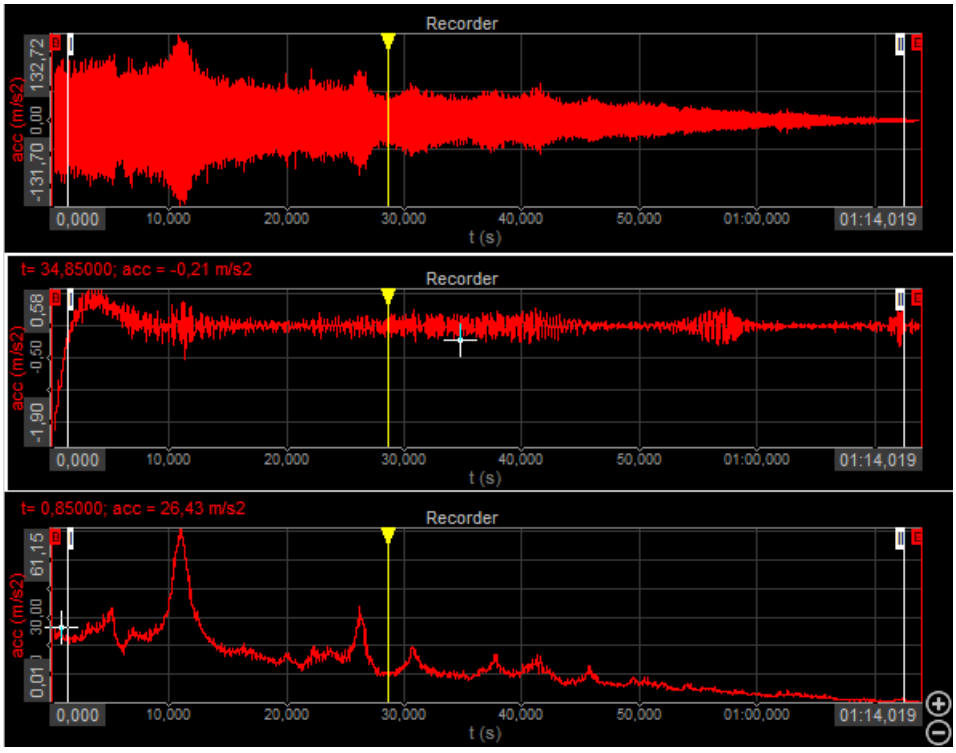
For noisy DC signals, selecting Average can clean up the display.

WARNING: **Be aware that all these display types represent only statistic values for the online display. The settings have no influence on the other displays or the data storage.**

Select any graph by clicking on it, and then select the Display type from the selector:

- Real value
- Average
- RMS

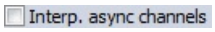
Just have a look at the following screen where you can see the difference between the three display types.



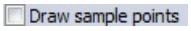
Use the Show events check-box to enable or disable view of event markers. You can also see all your events - keyboard, notice, and voice types - on the event list at the top right, directly below the replay control buttons.

For information about Events see -> Analyse -> Events.

Use the Interp. async channels check-box to interpolate the asynchronous channels. If an asynchronous data source like CAN is used, the values are interpolated between two sample points. But for digital signals (for example states) this is not wanted and with this option, we can disable the interpolation and the value will stay at the same level until next value is available.



Use the Draw sample points check-box to display individual sample points in analyse mode.



Single value axis

The Single value axis check-box can be used to set all active channels of a recorder grid to only one Y-axis. If set, all channels will use the same scaling and as a visual result, there will be only one axis with values left.

This function is very helpful when there is only small space for channel names and scaling and the channels use the same scaling. If this option is not used, recorder can display 4 channels at the same time. With this option, it can display up to 16 channels.

VERTICAL RECORDER

The Vertical recorder instrument is designed to emulate the time-history plotting capabilities of the traditional strip chart or recorder, but with vertical Time axis orientation and enhanced display. On each vertical recorder only one channel can be displayed.

When you select Vertical recorder instrument in the design or run mode, following settings will appear on left part of the screen:

Control properties

For detailed information about Recorder Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.

Vertical recorder settings

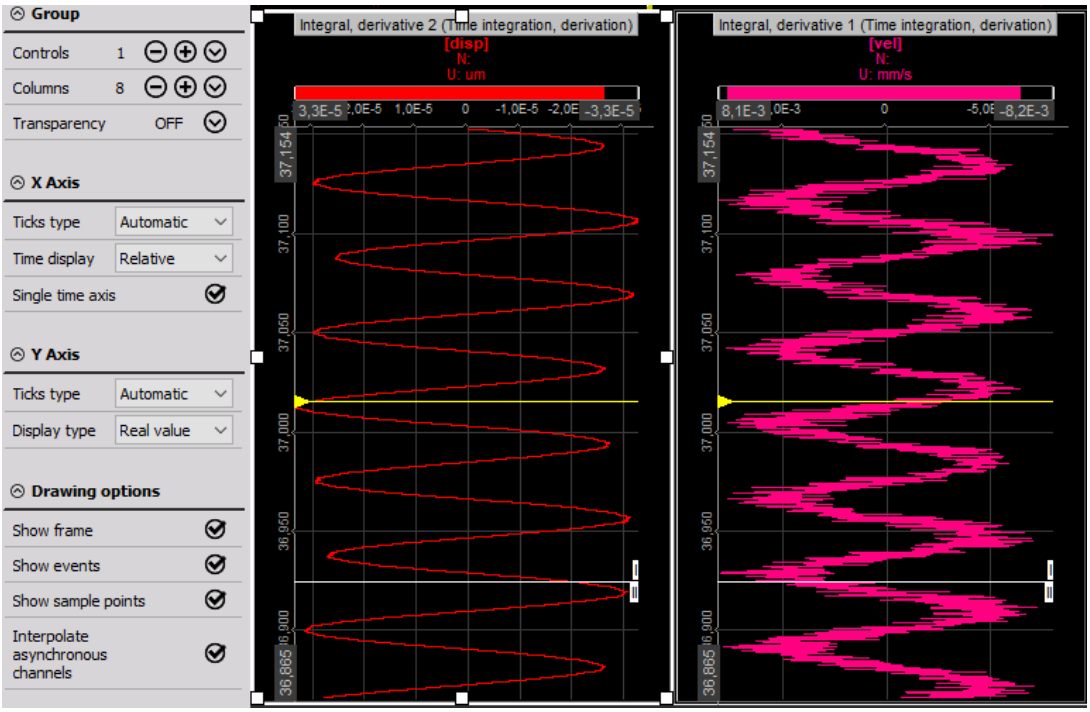
Typical Vertical recorder setting are:

- Single/Multiple Time axis
- Display type
- Time axis type
- X scale divisions
- Alarm levels

Available appearance setting for Vertical recorder in Run mode are:

- Time axis scaling
- X-/Y-axis information
- Y-axis scale/auto scale
- Channels selector

For detailed information about assigning / reassigning channels to/from Recorder -> see -> Display settings.

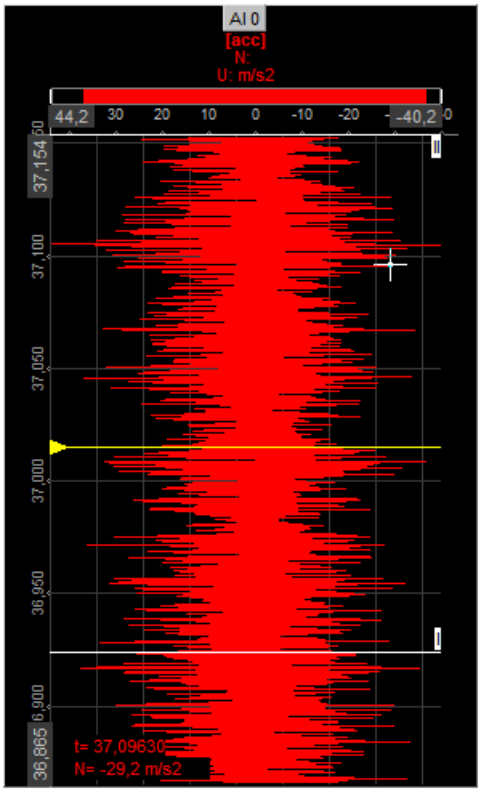

Group
Controls 1
Columns 8
Transparency OFF
X Axis
Ticks type Automatic
Time display Relative
Single time axis
Y Axis
Ticks type Automatic
Display type Real value
Drawing options
Show frame
Show events
Show sample points
Interpolate asynchronous channels
Integral, derivative 2 (Time integration, derivation)
[disp]
N:
U: um
3,3E-5
1,0E-5
0
-1,0E-5
-3,3E-5
37,154
37,100
37,050
37,000
36,950
36,900
36,865
Integral, derivative 1 (Time integration, derivation)
[vel]
N:
U: mm/s
8,1E-3
0
-8,2E-3
37,154
37,100
37,050
37,000
36,950
36,900
36,865

Appearance on screen

The Vertical recorder element offers all important information:

- channel number (group, description and slot)
- channel name(s)
- unit(s)
- time information
- zoom functions…

Appearance is like for Recorder, except Time axis is vertical.



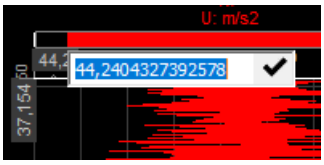
For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Run mode Changing the X-axis scale / auto scale

WARNING: **This way changing X-axis can't be done in Design mode -> available only in Run mode.**

If you want to change the X-axis for any input, the recorder offers two possibilities:

- **Enter values** - You may enter the values by yourself - simply click on the number at both extreme and then type in a new number.



This is possible when the cursor has following appearance:

- Auto scale

Move cursor to the axis scale. When you reach the auto scale area, the cursor changes to the following appearance:

Press left mouse button to activate auto scale for this channel or press right mouse button to undo auto scale. The auto scale function always calculates the minimum and maximum value of the currently displayed signal and use these values for scaling. The scaling will be only updated when you press the left mouse button!

Additional auto scale functions

Dewesoft offers additional auto scale functions when you press keys together with the left or right mouse button:

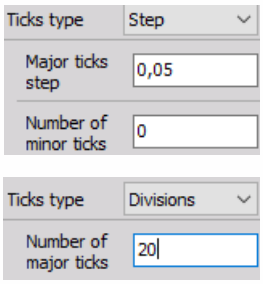
- Left mouse button - Auto scales the selected channel (see also above).
- SHIFT + Left mouse button Scales the selected channel symmetrical around zero (± values are the same).
- Right mouse button - Undo auto scale for the selected channel (see also above).

Display type

Display types of Vertical recorder are same as by Recorder, except Time axis type is vertical.

X scale divisions

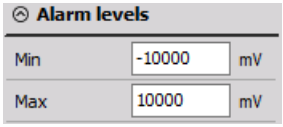
The Vertical recorder offers X scale types. You can select automatic, steps or divisions, if you choose the steps you have to define numbers of major ticks step and number of minor ticks, if you choose divisions you can choose from one to twenty (= dashed white line which divides graph area) from the drop-down list.



Alarm levels

You can set high and/or low alarm limits for each Vertical recorder, which will cause the recorder to mention the alarm above the grid.

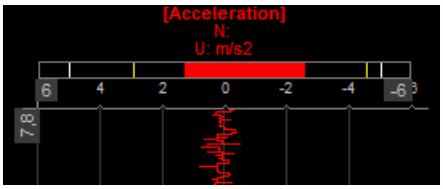
As a standard, the alarm values are set to the minimum and maximum range for the appropriate channel. If you want to use own limits, just enter the alarm level according to your requirements.



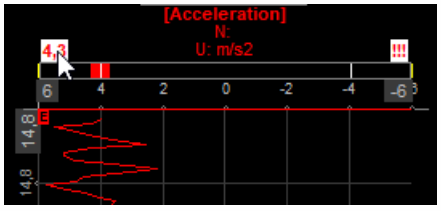
For example, if we want to supervise the signal within ±1 mm, we enter the following settings: -> see above.

Now we have set the limits to +1 and -1 mm, indicated by the two small white lines directly above the scale. The two yellow lines show the minimum and maximum value during this acquisition, which may be even outside the displayed time window. Finally, a line in the color of the channel indicates the current value; a small line indicates just small changes in the signal for the last 0.1 sec, the wider the line the higher the change in the signal.

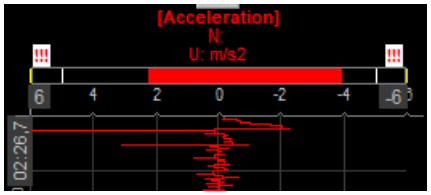
Example 1 - The current signal is within the defined limits of ±1 mm, indicated by the two white markers.



Example 2 - The signal is out of the defined limit, the current value (at the moment 1.37) is displayed in red letters as long as the signal is out of the limits.



Example 3 - The signal is back within the limits, the previous exceed of the limit is indicated by the three red !!!. You can also still see the maximum level indicator out of the limits.



X-Y RECORDER

The X-Y recorder is designed to display channels versus channels; there is no direct time relation.

When you select X-Y recorder instrument in the design or run mode, following settings will appear on left part of the screen:

Control properties

For detailed information about X-Y recorder Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.

X-Y recorder settings

Typical X-Y recorder setting in Setup tab are:

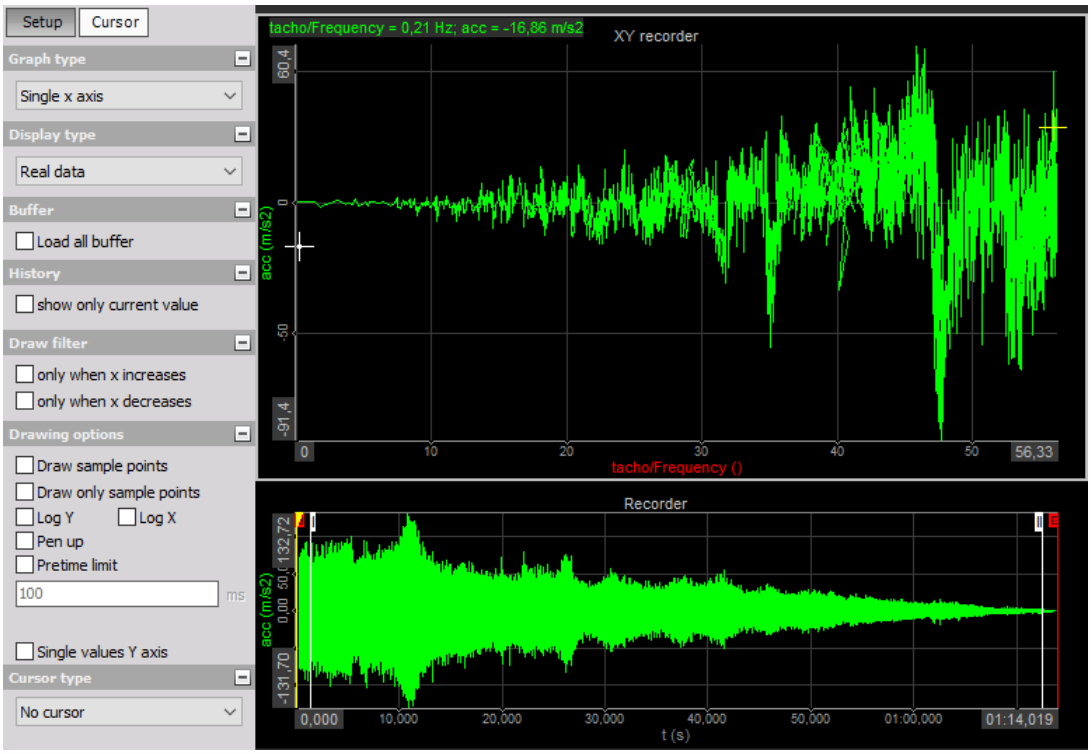
- Graph type
- Display type
- X and Y scale divisions
- Time scale
- Drawing options
- Draw filter
- History

Y channels cursor readouts are displayed in Cursor tab.

Available appearance setting for X-Y recorder instrument in Run mode are:

- X-axis scale/auto scale - logical same as for Vertical recorder
- Y-axis scale/auto scale - common instruments tools
- Channels selector

For detailed information about assigning / reassigning channels to/from X-Y recorder -> see -> Display settings.


Setup
Cursor
Graph type
Single x axis
Display type
Real data
Buffer
Load all buffer
History
show only current value
Draw filter
only when x increases
only when x decreases
Drawing options
Draw sample points
Draw only sample points
Log Y
Log X
Pen up
Pretime limit
100
ms
Single values Y axis
Cursor type
No cursor
tacho/Frequency = 0,21 Hz; acc = -16,86 m/s2
XY recorder
60,4
0
-50
-91,4
acc (m/s2)
0
10
20
30
40
50
56,33
tacho/Frequency ()
Recorder
132,72
0,00
-131,70
acc (m/s2)
0,000
10,000
20,000
30,000
40,000
50,000
01:00,000
01:14,019
t (s)

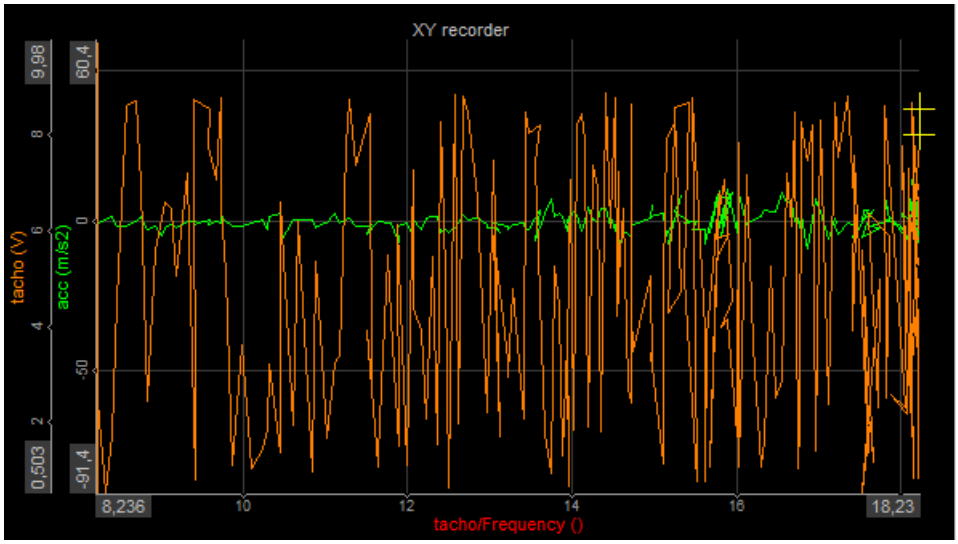
Appearance on screen

NOTE: **The only difference to time-based displays is that the first selected channel is always used as the X-axis channel.**

The X-Y recorder element can display up to eight Y-axis channels, related to one other channel on the X-axis at the same time.

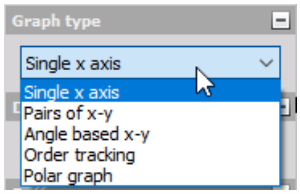
The yellow cross indicates the latest displayed value.

X-Y recorder appearance with only current value see -> History (below).



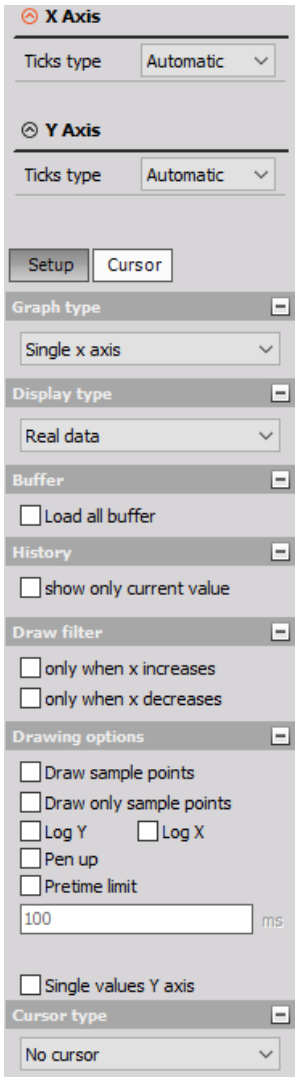
For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Graph type



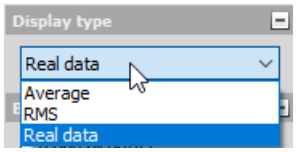
The Graph type knows four modes:

- **Single x-axis** - Allows only one channel x-axis. The first assigned channel is used for the x-axis, up to 4 following channels the y-axis.
- **Pairs of x-y** - Allow defining multiple ‘sets’ of x-y channels. First select the channel for x-axis, then select channel for the y-axis. This function allows referring different channels to different x-axis. Be aware that you still have only one x- and y-scaling! For example: x = distance 1, y = pressure 1. Now do the same for the second set, e.g. x = distance 2, y = pressure 2.
- **Angle based x-y** - The first x axis channel is the reference channel of the angle. It should go from 0 to 360°. The second (and further channels) are angle related data (like rotational vibration - XY recorder now displays the rotational angle of current revolution). This XY recorder is like a scope, but with angle reference instead of time reference.
- **Polar graph** - In a polar coordinate grid will be a series of circles extending out from the pole and many different lines passing through the pole to represent the angles at which the exact values are known for the trigonometric functions.


X Axis
Ticks type
Automatic
Y Axis
Ticks type
Automatic
Setup
Cursor
Graph type
Single x axis
Display type
Real data
Buffer
Load all buffer
History
show only current value
Draw filter
only when x increases
only when x decreases
Drawing options
Draw sample points
Draw only sample points
Log Y
Log X
Pen up
Pretime limit
100
ms
Single values Y axis
Cursor type
No cursor

Display type

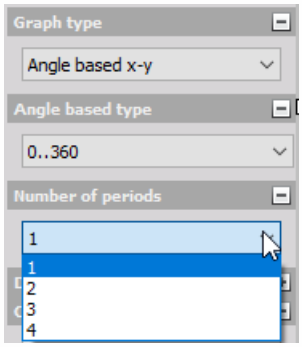
Dewesoft X-Y graph is now being able to display the Real data. You can display also Average (for slow signals) or RMS values (for dynamic signals). As a standard, the values for the display type will be calculated over a period of 0.1 seconds.



Select any graph by clicking on it, and then select this value from the Display type drop-down list.

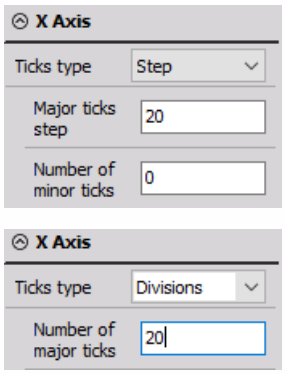
WARNING: **Be aware that all Average and RMS display types represent only statistic values for the online display. The settings have no influence on the other displays or the data storage.**

Angle based x-y graph - The Display type for this Graph type is different as for other Graph types. You can display signals over a 1 period or 2, 3 or 4 periods which can be selected from the drop-down list.

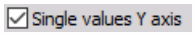


X and Y scale divisions

The X-Y recorder offers X and Y scale step/divisions. You can select the step or divisions and define the number of Major ticks step and the Number of minor sticks or X,Y scale divisions from one to twenty (= dashed white line which divides graph area), which can differ between X and Y axis.

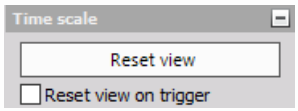


The Single value axis check-box can be used to set all active channels of an X-Y recorder grid to only one Y-axis. If set, all channels will use the same scaling and as a visual result, there will be only one axis with values left. This function is very helpful when there is only small space for channel names and scaling and the channels use the same scaling.



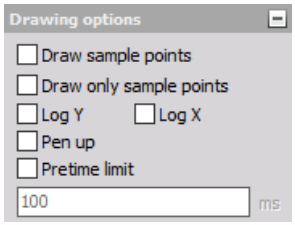
Time scale

Recorder or scope displays show their content only for a limited time. But what about the X-Y graph? How long should it display data? Decide it by yourself. Simply click on the Reset view button in Time scale section to clear the graph.



Drawing options

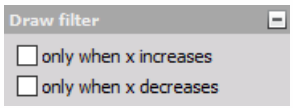
Dewesoft allows Log Y and/or Log X (logarithmic) axis types. Select the axis type (with checking appropriate box) according to your application.



For detailed information and using hints about X and Y scale type -> see -> Properties of the Fourier transform.

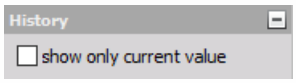
Draw filter

Especially for run-up tests you should use the Only when x increases check-box in Draw filter section. This is only a drawing filter and avoids any values displayed which become smaller than before. Unselect this feature to receive also decreasing X-axis signals again.

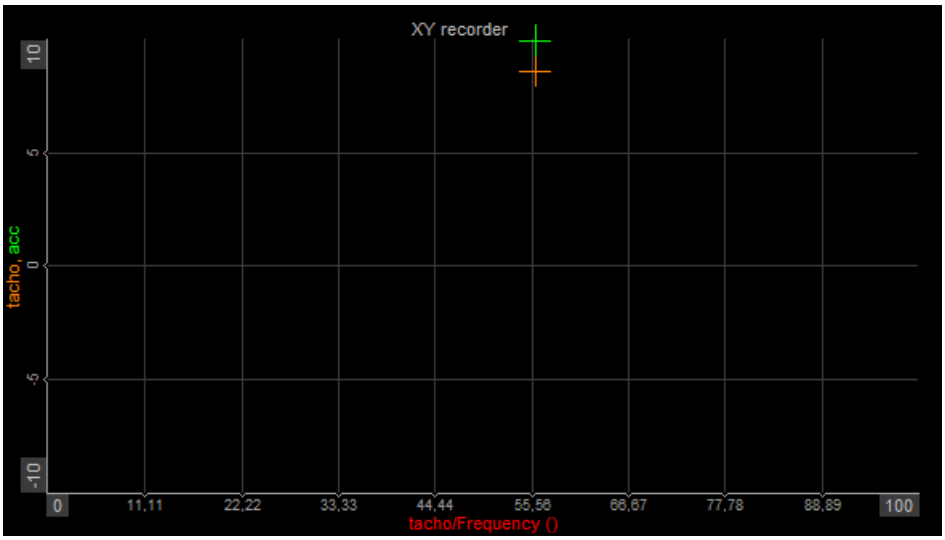


History

When the signal is changing very strong, it may be helpful to click the Show only current value check-box in History section; this removes all displayed values from the X-Y graph and shows only the current value. Unselect this feature to receive the whole signal history again.



Example: appearance X-Y recorder with enabled show only current value



XY Recorder Cursor

By selected Cursor tab the cursor readouts for each Y channel in CURRENT POSITION section is displayed.

SCOPE

The Scope instrument is used for displaying fast, short-time events. Like in a traditional scope you can define trigger conditions. Up to 16 inputs can be displayed at once in each graph.

When you select Scope instrument in the design or run mode, following settings will appear on left part of the screen:

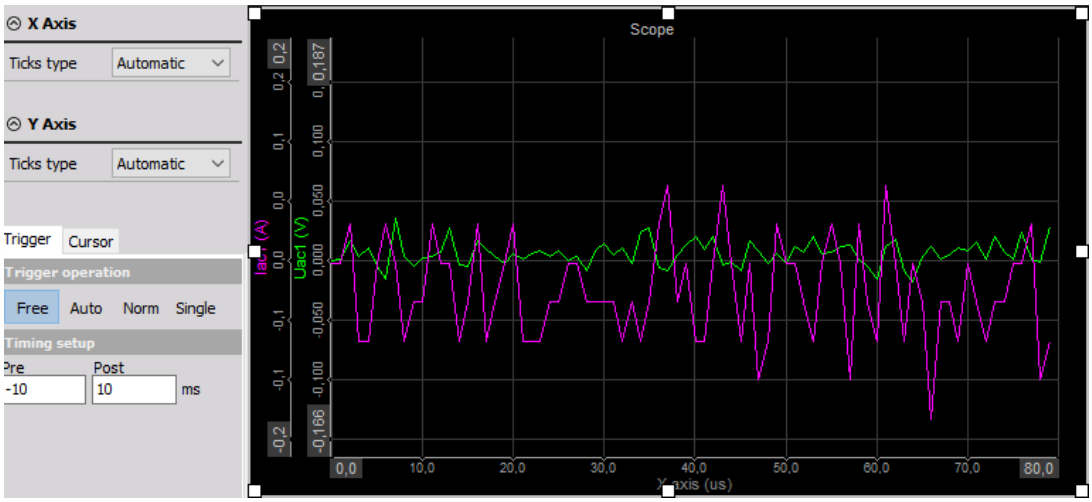
- Control properties - For detailed information about Scope Control properties: grouping, number of column, Add / Remove, transparency, see -> Control properties.
- Scope settings

Typical setting for Scope instrument are in three main groups:

- Trigger
- Cursor
- Scale
- History

Available appearance setting for Scope instrument in Run mode are:

- Y-axis scale/auto scale (Common instruments tools)
- Time axis scaling (Common instruments tools)
- Zoom (change the current position and scroll through the whole acquired data)
- Channels selector



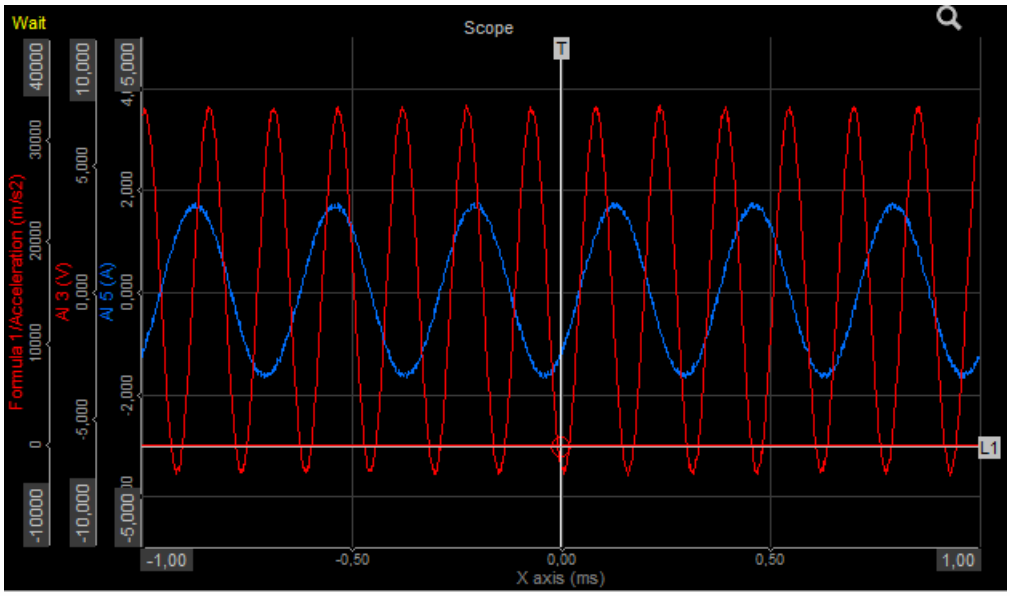
For detailed information about assigning / reassigning channels to/from Scope -> see -> Display settings.

Appearance on screen

The Scope element in the overview offers all important information:

- channel name(s)
- unit(s)
- time information
- zoom functions…

When the scope is not triggering, the bar on the right side shows the current levels of the signal so we can optimize the trigger level according the normal values (we can also use Auto trigger mode). When the trigger is lost for some seconds, data will be shown none triggered.

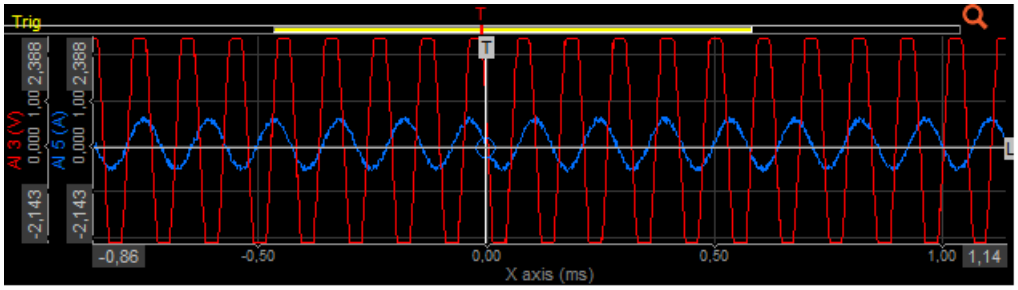


For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Run mode Zoom (additional appearance setting)

At the top right above each graph in Norm or Single Trigger mode you maybe have already noticed a small icon Pressing it enables/disables the zoom view during acquisition.

Up to now, when you press the black buttons at the bottom right side of each graph, you also changed the memory depth used for the acquisition.

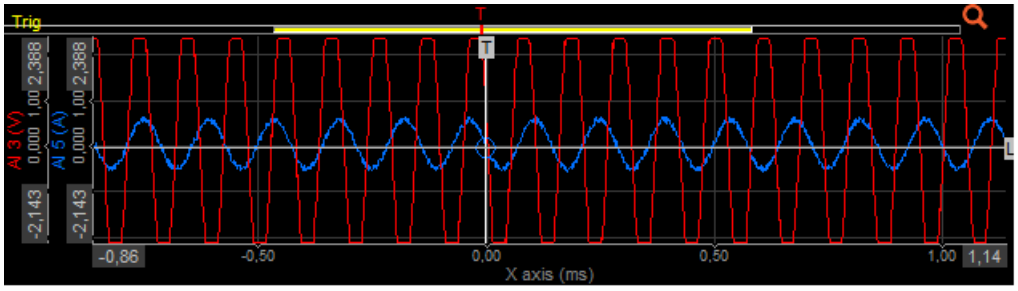


Normal Scope trigger view

If we want to see the event now more detailed, just click the zoom icon. At the top of the graph you will now see a scroll bar indicating the current displaying position within the whole acquired signal.

Press the Plus button to zoom in:

Or Minus button to undo zoom:



Zoomed Scope trigger view

When you move the mouse over the scroll bar at the top, it will change its appearance to a "hand".

When you press the left mouse button and move the mouse, you can change the current position and scroll throug the whole acquired data of the current trigger shot.

SETTING - DISPLAY TOOLS

The Scope instrument typical settings include three main groups:

- Trigger
- Free run
- Auto
- Norm
- Single - settings of triggered acquisition
- Cursor - cursor measurement to show the cursor readouts for each channel within the selected scope; with Reference curves possibility
- Scale - to change displayed offset and scaling of signals
- History - to display the trigger events in different ways -> history type, to select how many trigger events will be used, to browse through the trigger events, to export the acquired data

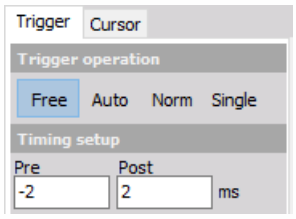
Trigger setting

Dewesoft knows four types of Trigger operation -> Triggered acquisition:

- Free run
- Auto
- Norm
- Single

Free run

All values are displayed, no trigger active. There are no additional settings.



Auto

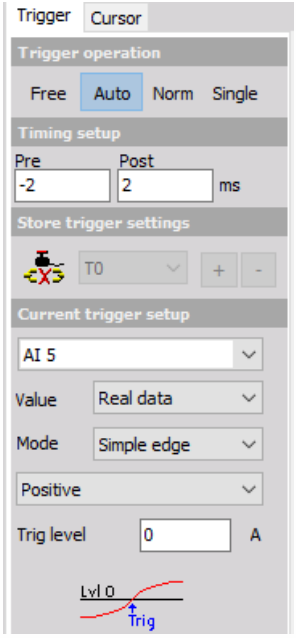
The auto trigger displays values if the trigger condition is true; when there is no trigger within some time, it displays the current value.

For this type of Trigger operation can be set:

Timing setup

Current trigger setup with:

- select the desired channel
- define the Value
- define the Mode - trigger type
- setup trigger condition for selected trigger type:



Mode:

- Simple edge
- Filtered edge
- Window
- Pulse-Width
- Window and pulse-width
- Slope

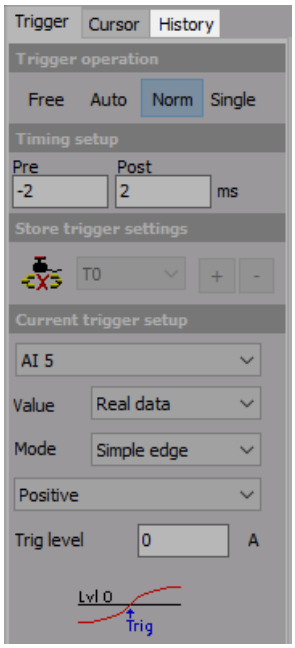
Store trigger settings

Norm

The normal trigger displays only values if the trigger condition is true.

For this type of Trigger operation can be set the same setting as for Auto trigger -> see above.

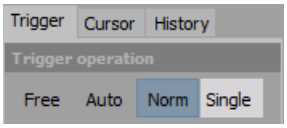
When the Norm (or Single) trigger is selected, another tab appears -> the History.



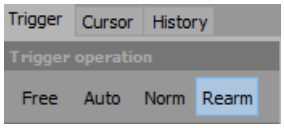
Single

This function can be used to acquire single events.

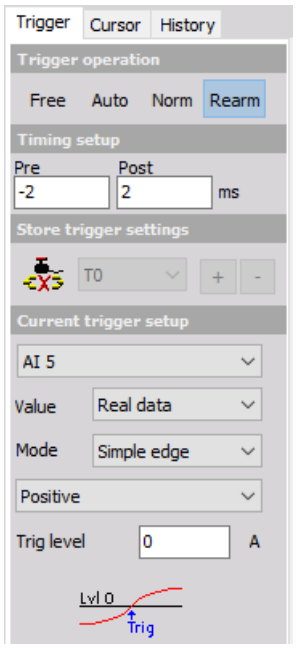
After selecting single button:



and event appear this button changes to Rearm:

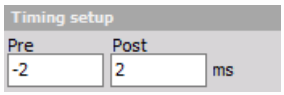


Press it to get another single shot event.


Trigger
Cursor
History
Trigger operation
Free
Auto
Norm
Rearm
Timing setup
Pre
Post
-2
2
ms
Store trigger settings
T0
+
-
Current trigger setup
AI 5
Value
Real data
Mode
Simple edge
Positive
Trig level
0
A
Lvl 0
Trig

Timing setup

The Timing setup can be used to define the displayed Pre and Post trigger time in milliseconds.



NOTE: **Like the trigger level, the trigger position can be changed within the displayed time window by moving the white vertical line in the scope graph. Simply click on the line, keep the mouse button pressed and move the line to the desired position.**

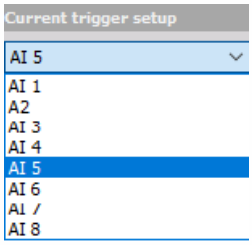
The time window can also be changed using the buttons at the right bottom of each graph.

Current trigger setup

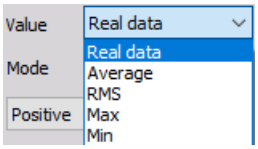
The trigger conditions for Auto, Norm and Single data trigger are the same and work in the same way than described in Using trigger to start and stop recording.

For detailed information about using triggers see -> Using Triggers to start and stop recording.

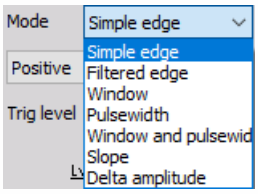
- **Select the desired channel** First of all you have to select the desired channel out of the drop-down list. It displays all available channels.



- **Define the value** Select the Real data, Average or RMS from the drop-down list.



- **Define the mode** Select the trigger type Simple edge, Filtered edge, Window, Pulse-Width, Window and pulse-width or Slope from the drop-down list.



- **Setup other** These settings (e.g. Slope, Trigger level, Rearm level, Pulse time,...) depend on selected trigger type in Mode field.

For detailed information about trigger mode and set up triggers see -> Trigger setup.

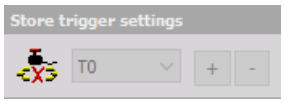
HINT: **The trigger level can also be changed by moving the white vertical line in the scope graph. Simply click on the line, keep mouse button pressed and move the line to the desired position.**

Store trigger settings

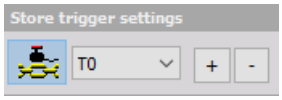
This is a very nice function to define the storing options directly within the scope.

For detailed information about set up trigger see -> Trigger Setup.

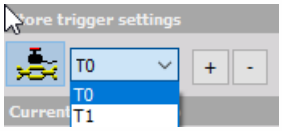
Any changes done here are automatically copied to the system trigger and vice versa. To activate this function press the Link store trigger button.



The drop-down list next to the button shows - if already available - existing triggers conditions or starts with a fresh entry T0.



The buttons can be used to define additional conditions, which can be selected by the drop-down list and changed according to your requirements.

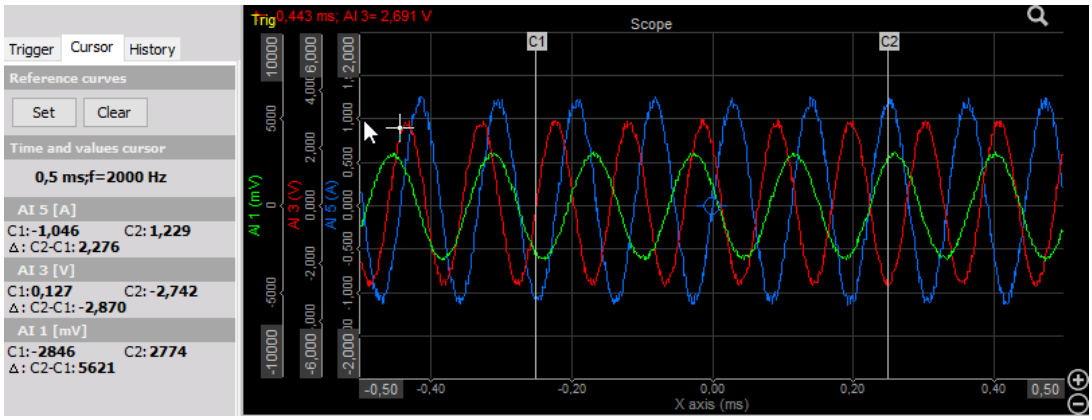


The buttons can be used to delete selected additional conditions.

Cursor

Cursor measurements

Dewesoft offers also Cursor measurements function to show the cursor readouts for each channel within the selected scope. Select the Cursor tab to activate the cursor measurement:



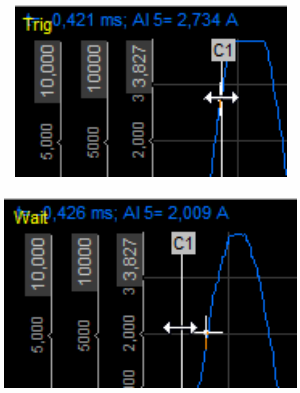
The scope offers two measurement cursors for the active graph, which can be positioned in two ways:

- Move the mouse cursor to the first position of interest. Click the left mouse button - cursor C1 will appear.

- Move the mouse cursor to the second position of interest and click right mouse button for cursor C2 positioning.

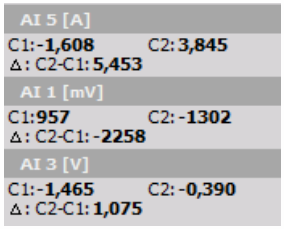
This can be repeated at any other desired position.

- Drag the first cursor from the left side of the graph to the position of interest and the second cursor from the right side of the graph. In both case cursor appearance on measurement cursors .jpg must change to:

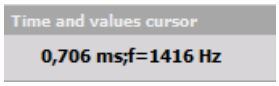


The settings menu at the left side of the screen now displays:

- The cursor readouts for each channel within the selected scope - values at C1, C2 and difference C2-C1



- Time and values cursor between C1 and C2

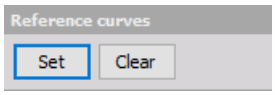


WARNING: **The freeze function is not working in the scope!**

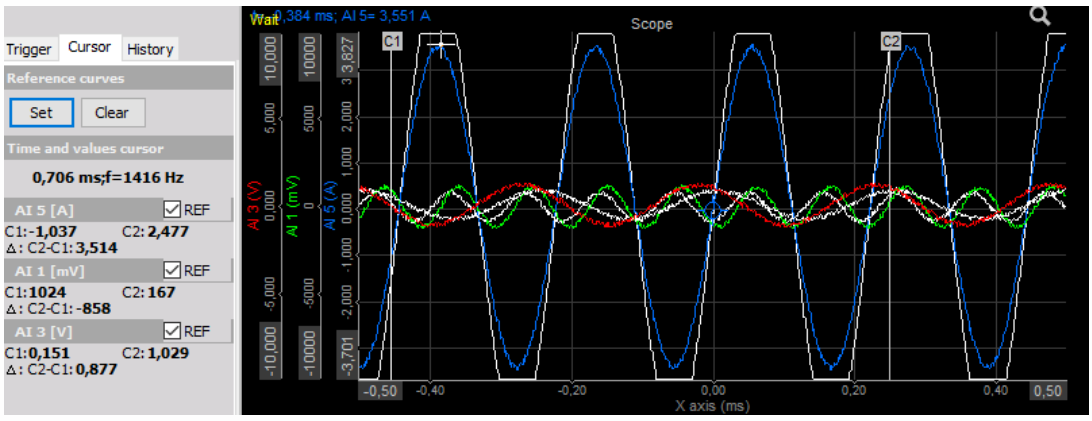
Reference Curves

Dewesoft offers also Reference curves within the scope display.

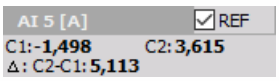
To activate them just press the Set button in the Reference curves Cursor settings.



This will copy the data of all currently displayed channels into a reference curves memory. The reference curve will now be displayed in white color:



In the cursor readouts for each channel REF check-box appear:



To hide the reference curve uncheck the REF check-box for the desired channel(s). This will keep the reference curve within memory. To display it again just check the REF check-box again.

Press the Set button at any time to update the reference curve to the current triggered data.

To remove the reference curve permanently press the Clear button. This will remove the

reference curve data from memory. To define a new reference curve press the Set button again.

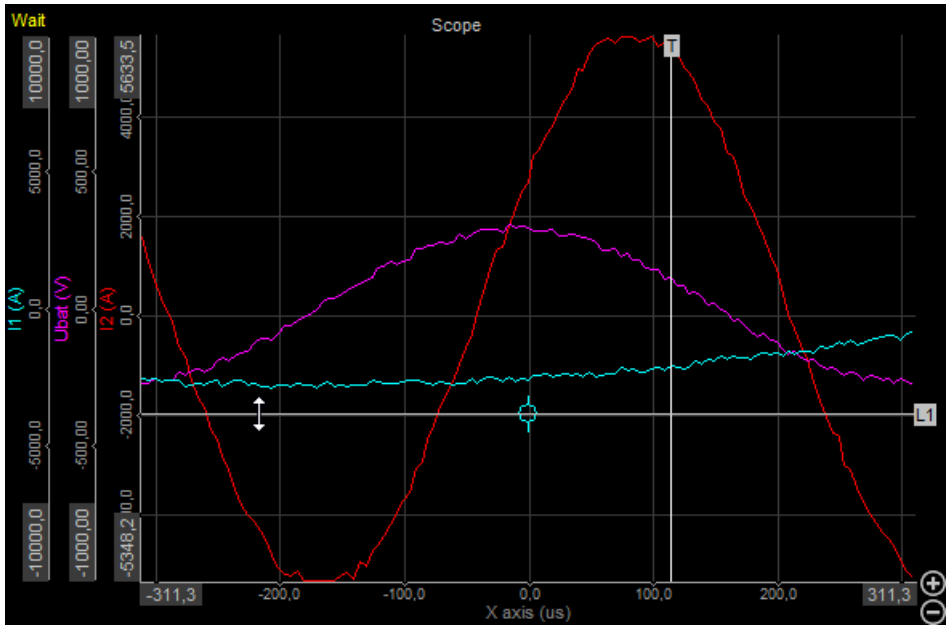
A dashed horizontal line indicates the current offset setting of each channel. It has the same color than the channel itself. You can use this line also to change the channel offset.

Simply move the mouse cursor over the line and the mouse cursor appearance will change to:

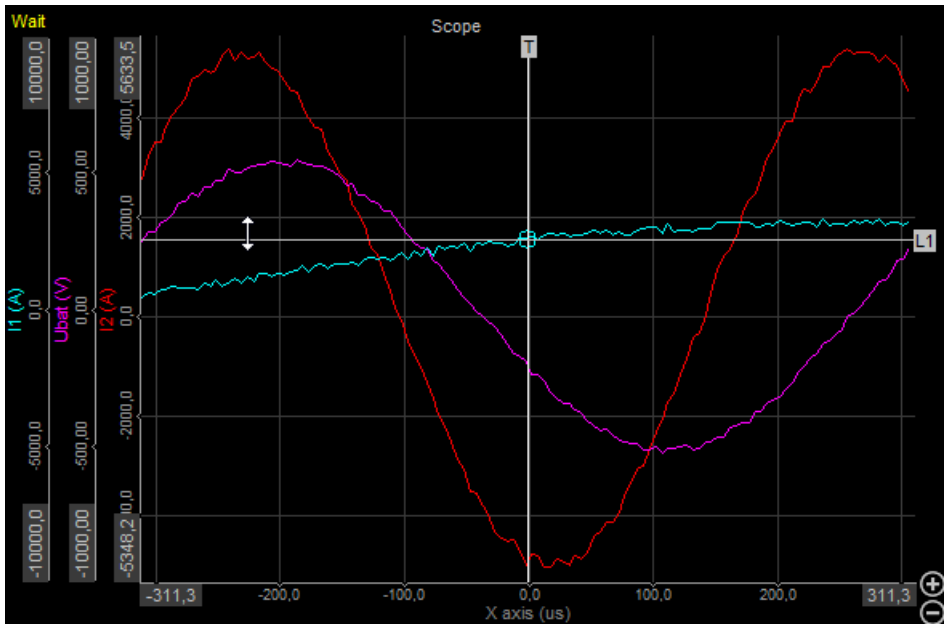
Now click (and keep mouse button pressed) and move the dashed line to the desired offset position.

Examples for scaling and offset functions:

Scope Scale view after scale down Power channel:



And after set negative offset for Power channel:

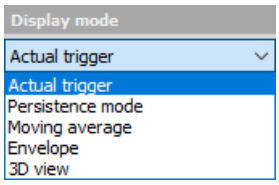

Wait
Scope
T
L1
I1 (A)
Ubat (V)
I2 (A)
10000,0
5000,0
0,0
-5000,0
-10000,0
1000,00
500,00
0,00
-500,00
-1000,00
5633,5
4000,0
2000,0
0,0
-2000,0
-5348,2
-311,3
-200,0
-100,0
0,0
100,0
200,0
311,3
X axis (us)

History

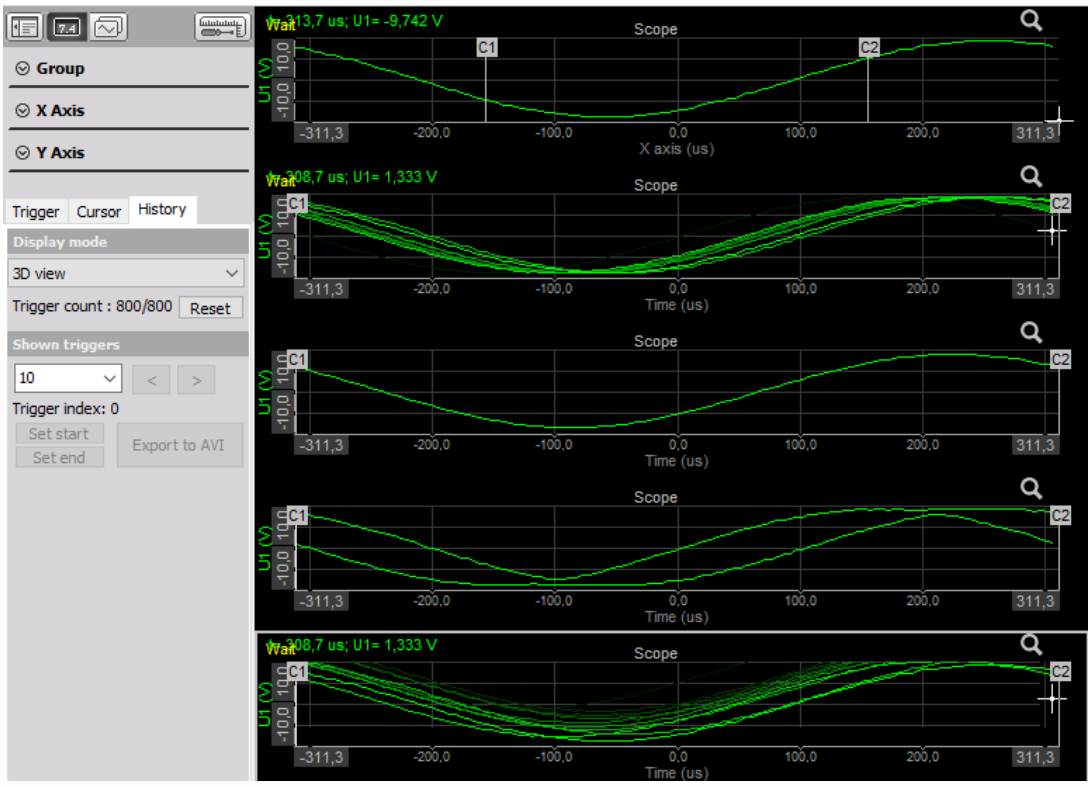
When the Norm or Single trigger is selected, another tab appears: the History. This feature allows you to display the following information - history type:

- **Actual trigger** - Always the latest trigger event is displayed. When you press the Stop button, you can browse through the last recorded trigger events.
- **Persistence mode** - You may still know this feature from analog oscilloscopes: several events will be displayed in an overlay technique, the older the events the brighter their color.
- **Moving average** - Use the moving average function only for repetitive signals; you can increase accuracy and reduce noise on signals using this function.
- **Envelope** - The envelope mode displays two graphs showing the minimum and maximum value of several events.
- **3D view** - Displays the trigger events in a waterfall-like type. Very helpful to show signal abnormalities or changes.

Display mode can be selected from Display mode drop down list:



The following screen should help you to understand the difference between the five history types:



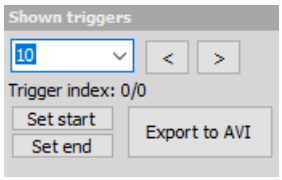
Below the Display mode selection, the Trigger count displays two values: the first mentions the

current number of existing trigger events within the history memory, the second the maximum possible number of events. This value depends dramatically from the displayed time. When the maximum number is reached, both values are the same. The memory is of FIFO type (first in, first out).

If you want to clear the history memory simply press the Reset button.

In the section Shown triggers you can select how many trigger events will be used for the appropriate history type.

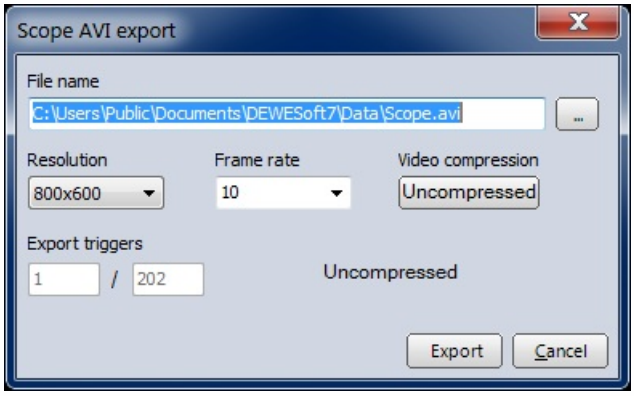
If you are in stopped condition (when you press the Stop butt on tool bar) you can use the arrow buttons to browse through the trigger events.



Keep an eye on the Trigger index: it shows you the current positions within the available events.

In stopped condition it's also possible to export the acquired data by pressing the Export to AVI or Save to file button.

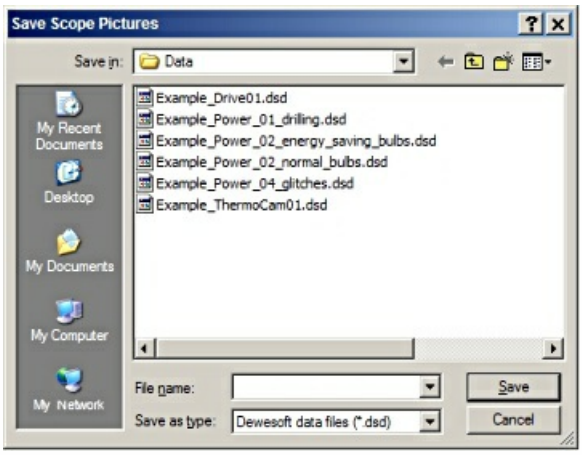
When Export to AVI is selected Scope AVI export window appear to set: File name, Resolution, Frame rate, Video compression and Export triggers field, in which we can define to save only interesting triggered events (to reduce the amount of data).



For information about this settings see -> Export screen to AVI.

Select Export button to export 'avi' data or Cancel button to cancel exporting.

Whit Save to file button we call Save Scope Pictures window to save acquired data from history memory in to Dewesoft *.dsd (Dewesoft data) file:



Choose existing file from list or enter new file name (recommended) and then select Save button to save data or Cancel button to cancel saving.

To reduce the amount of data you can select a certain range within the triggered events by using the Set start and Set end buttons.

FFT ANALYSIS

The Dewesoft FFT (Fast Fourier transformation) instrument shows the frequency components of acquired signals in amplitude and frequency.

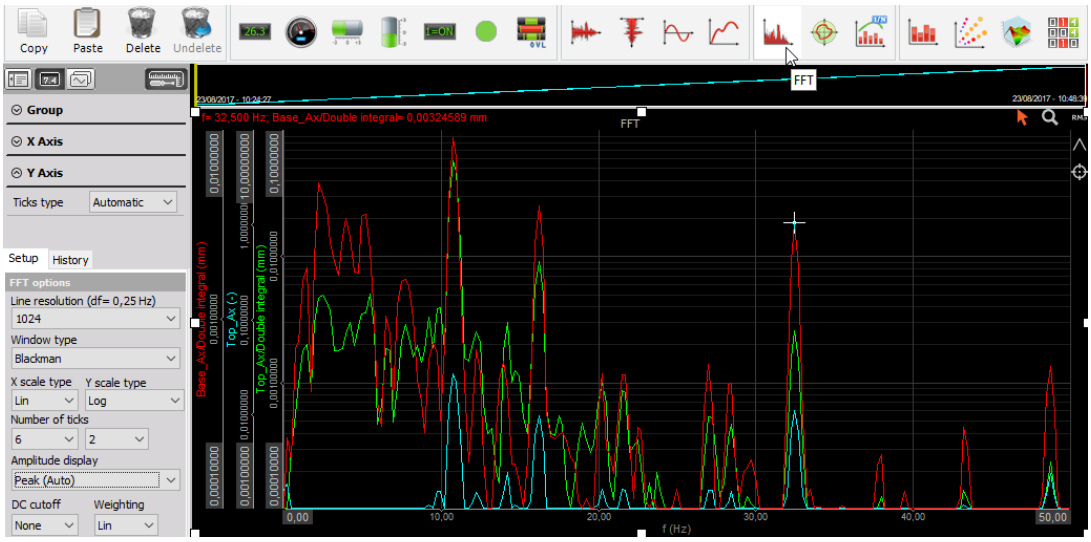
When you select FFT instrument in the design or run mode, following settings will appear on left part of the screen:

- Control properties - For detailed information about FFT instrument Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.
- FFT settings
- Setup with Line resolution, Window type, X/Y scale type, Number of X/Y axis ticks, Amplitude display, DC cutoff and Weighting
- History - display options for Current FFT and Averaged FFT

Available appearance setting for FFT instrument in Run mode are:

- Changing the Frequency axis scale
- Y-axis scale/auto scale (Common instruments tools)
- Cursor Function (change the current position and scroll through the whole acquired data)
- Online calculation
- Channels selector

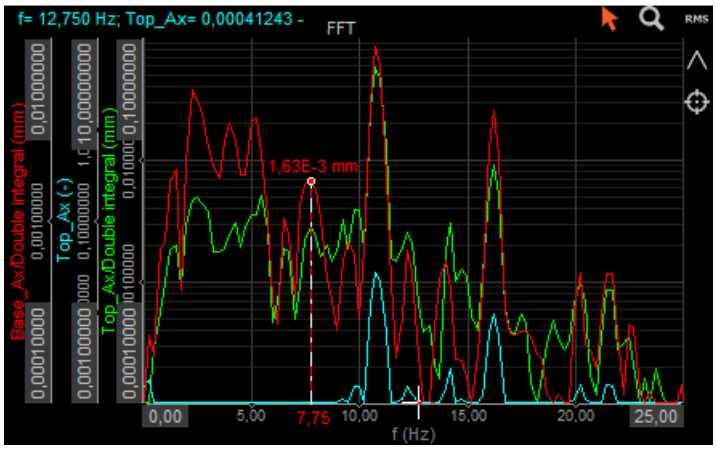
For detailed information about assigning / reassigning channels to/from FFT -> see -> Display settings.



Appearance on screen

The FFT element offers all important information:

- channel name(s)
- unit(s)
- frequency information
- zoom functions…



For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Changing the Frequency axis scale

Run mode Frequency axis scaling

WARNING: **Changing the frequency axis this way can't be done in Design mode -> available only in Run mode.**

Use the blue PLUS and MINUS symbols to expand or compress the Frequency axis. Clicking MINUS will show more Frequency across the graph until the maximum analyzable frequency has been reached.

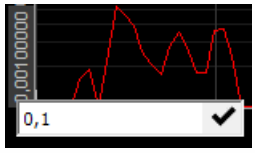
Clicking PLUS will show the frequency more detailed.

Run mode Changing the Frequency axis scale

WARNING: **Changing the frequency axis this way can't be done in Design mode -> available only in Run mode.**

Enter values

You may enter the values by yourself - simply click on the number at either extreme or minimum (see below) and then type in a new number.



This is possible when the cursor has the following appearance:

When clicking on this cursor, enter the field for new value appear (see left). After entering the new min and/or max value press OK button and new Frequency axis scale appear on the graph.

Cursor function

With two buttons in the FFT display, the function of the cursor can be defined.

With this function enabled (= default setting), the cursor shows the current frequency and amplitude values at the cursor position within the FFT signal. To show the current values in the cursor readout, just click on the desired position of the signal. To remove the readout, click again on this button - cursor marker.

With this zoom button enabled, you can zoom into any area - range of the FFT display. Just click into the display, hold the button (on this place vertical white line appears), move the mouse and release the button. To undo the zoom, just press the right mouse button in the graph.

Online calculation

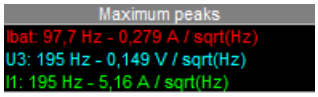
To keep an overview over the FFT, Dewesoft can calculate the RMS values and/or peak values (max) of the displayed signals. To activate the calculation, just click on:

-

  for the RMS values

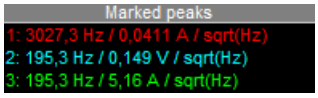
-

  for the peak values

-

  for the marked peaks

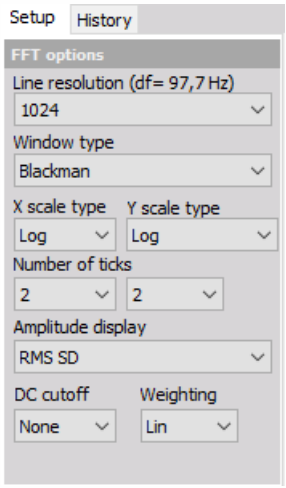
  

FFT Settings

Serious FFT analysis requires several settings to get a useful result. All these settings Dewesoft combine in two tabs:

Setup

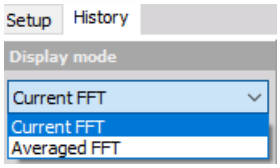
- Line resolution
- Window type
- X scale and Y scale type
- Number of ticks for X and Y axis
- Amplitude display
- DC cutoff
- Weighting



History

This setting defines Display mode for three group displays:

- Current FFT
- Averaged FFT



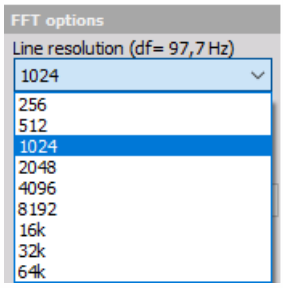
FFT Setup settings

Line resolution

The FFT lines are responsible for the frequency resolution. The higher the FFT lines value, the better the resolution - but also the higher the calculation time.

This line resolution depends on the sampling rate and the number of lines chosen for the FFT. So

if we want to have a fast response on the FFT, we choose less line, but we will have a lower frequency resolution. If we want to see exact frequency, we set higher line resolution. A simple rule is: if it takes 1 second to acquire the data from which the FFT is calculated, the resulting FFT will have 1 Hz line resolution. If we acquire data for 2 seconds, line resolution will be 0.5 Hz.



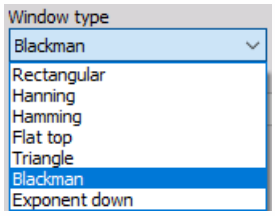
The current frequency resolution is mentioned in the selection line next to the heading (df = n Hz).

Example: The sampling rate has been set to 10000 Samples/sec and the resolution to 1024 FFT lines. These settings allow an FFT analysis up to 5000 Hz (half sampling rate). Now you divide the max analysis frequency by the FFT lines (5000 Hz / 1024 lines). The result is 4.88 Hz per line resolution (mentioned in the selection line).

To change the FFT lines, just click in the Line resolution field and select from the drop-down list.

Window type

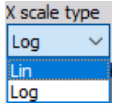
Dewesoft supports the most common Window types for FFT analysis. Select the window from the drop-down list according to your application.



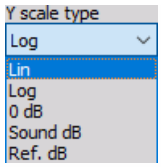
X and Y scale type

Dewesoft allows:

- two different X (frequencies) axis types (Linear and Logarithmic)

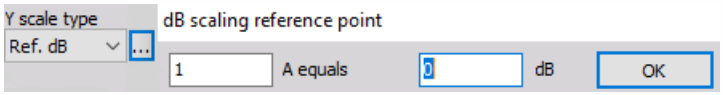


- four Y (amplitudes) axis scaling types (Linear, Logarithmic, 0 dB and Noise dB and Ref. dB scaled)



Select the axis type from the drop-down list according to your application.

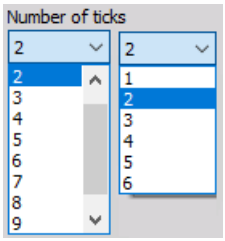
When in Y scale type Ref. dB is selected, 'Db scaling reference point' window appears to enter this reference point and confirming that with OK.



Number of ticks

You can select the Number of ticks from one to six ticks (dashed white line which divides the graph area) for your FFT display, which can differ between X and Y axis.

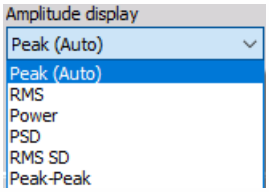
The first field in the row is to select the X axis ticks, the second row for the Y axis ticks.



Amplitude display

The Amplitude display section defines display in Y-amplitude axis.

From Amplitude display drop down list we can select different types of amplitude scaling of the FFT. Basic setting is Amplitude (Auto), which shows for pure sine wave the amplitude of the sine.



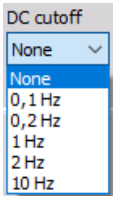
If we have the sine wave with 2 V peak amplitude (4 V peak-peaks), we would have in the FFT 2 V amplitude. For the table let's assume that the original signal unit is Volts. If the units are different, it will appear in the FFT.

Amplitude type	Units	Description
Amplitude (Auto)	V	is the pure signal amplitude

Amplitude type	Units	Description
RMS	Vrms	is the RMS amplitude, calculated as Amplitude/sqrt(2)
Power	V*V	calculated as RMS value squared
PSD	V*V/Hz	calculated as RMS squared, divided by the line resolution and sqrt(2) - used for checking the noise
RMS SD	V/sqrt(Hz)	calculated as RMS value, divided by the square root of line resolution - also used for checking the noise

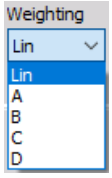
DC cutoff filter

TO remove DC or low-frequency components, select from drop down list the DC cutoff filter - lower limit.



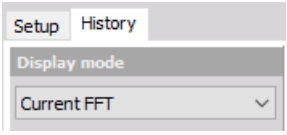
Weighting

As a standard, FFT analysis uses a Linear Weighting. For sound analysis, special FFT weighting can be set. As opposed to the sound module in math, where the weightings will be calculated in time domain, this will calculate the sound weighting in frequency domain.



FFT History settings

Current FFT

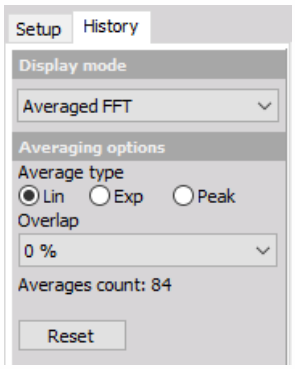


Display current FFT with settings in Setup tab (see above). Just select Current FFT from the Display mode drop-down list. Only this one setting is necessary for this Display mode type.

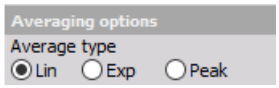
Averaged FFT

Use averaging mode to get a more stable FFT display. To activate the averaging just select Averaged FFT from Display mode drop-down list (see below).

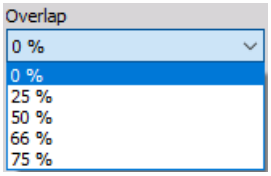
All necessary setting can be done on Averaging options section.



- Average type - From Averaging options select Average type: Linear, Exponential or Peak. As a standard, linear is selected.



- Overlap - Depending on the application, it may be necessary to define a data overlap. When the window type is used, we have to use overlap otherwise some of the data will be ignored. Therefore the use of overlap is highly recommended.



OCTAVE ANALYSIS

The Dewesoft Octave instrument is meant to display the frequency components of acquired signals in amplitude and frequency. The octave analysis is synthesized from the FFT analysis. On Octave instrument only one channel can be displayed.

IMPORTANT: **Octave analysis instruments provides basic options. For advanced CPB analysis, please add CPB math channel in the setup and use 2D graph to visualize the data.**

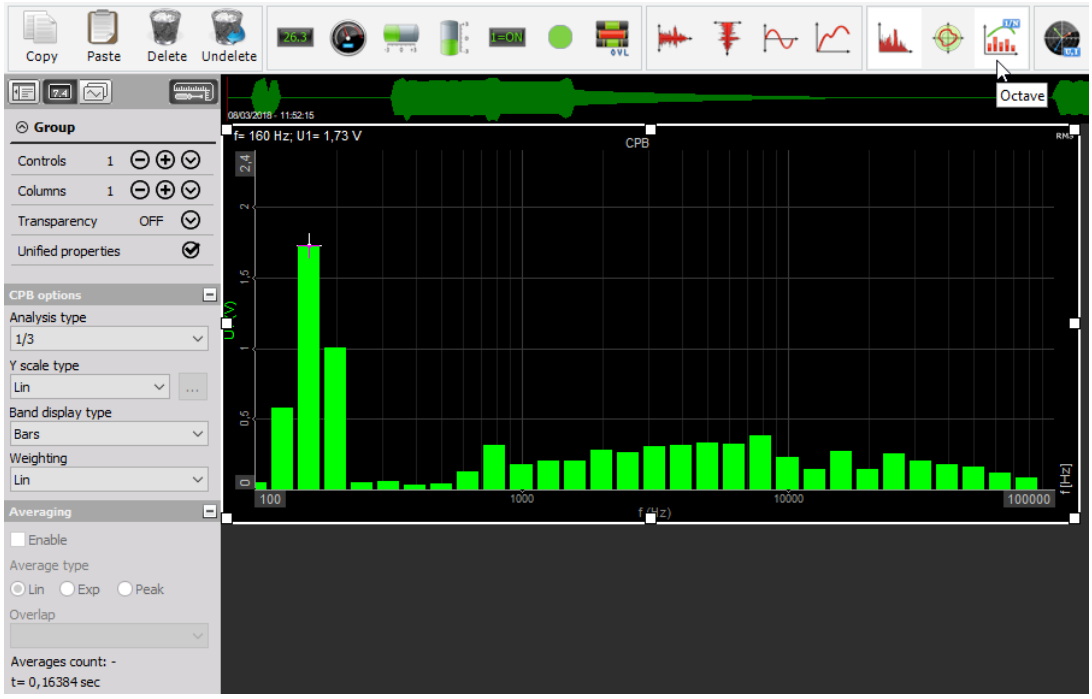
When you select Octave instrument in the design or run mode, following settings will appear on left part of the screen:

- Control properties - For detailed information about Octave instrument Control properties: grouping, number of columns, Add / Remove, transparency,...see -> Control properties.
- Octave settings
- Display settings - with Analysis type, Y scale type, Band display type and Weighting
- Averaging settings - Averaging type, Overlap and Averages number

Available appearance setting for Octave instrument in Run mode are:

- Y-axis scale/auto scale
- Online calculation
- Channels selector

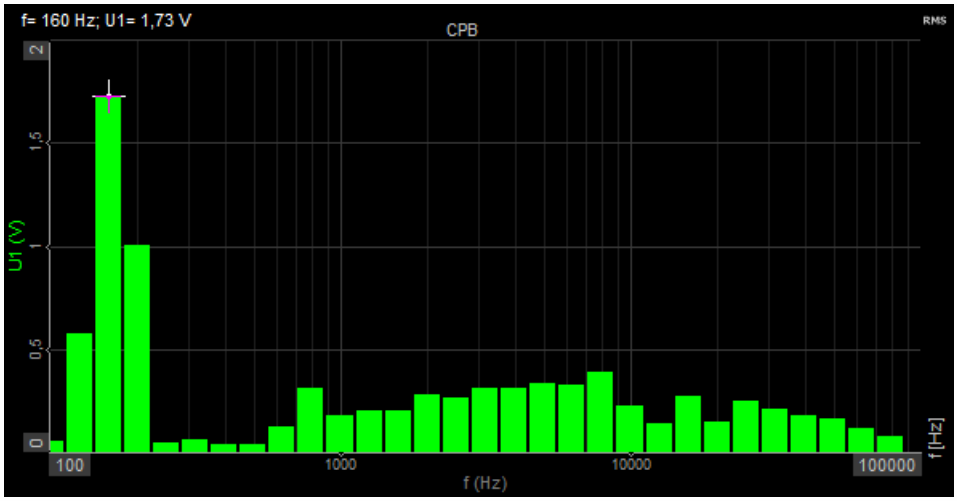
For detailed information about assigning/reassigning channels to/from Octave -> see -> Display settings.


Copy
Paste
Delete
Undelete
Octave
Group
Controls 1
Columns 1
Transparency OFF
Unified properties
CPB options
Analysis type
1/3
Y scale type
Lin
Band display type
Bars
Weighting
Lin
Averaging
Enable
Average type
Lin Exp Peak
Overlap
Averages count: -
t= 0,16384 sec
08/03/2018 - 11:52:15
f= 160 Hz; U1= 1,73 V
CPB
RMS
U [V]
f [Hz]
100
1000
10000
100000

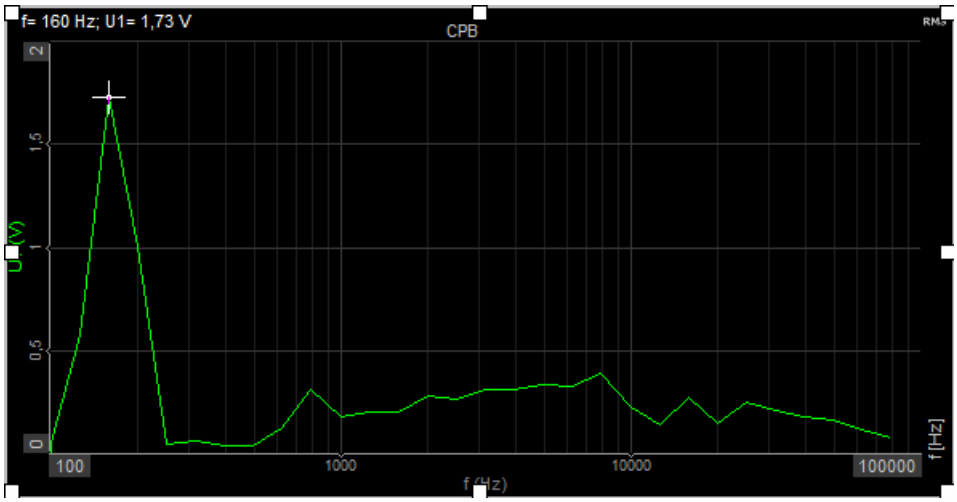
Appearance on screen

The Octave element offers all important information: channel name, unit, frequency information, calculation values…

Bar Band display type



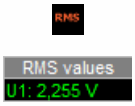
Lines Band display type



For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Online calculation

To keep an overview over the Octave, Dewesoft can calculate the RMS values displayed signals. To activate the calculation, just click on the RMS button to see the RMS values.



Octave display settings - CPB options

Serious Octave analysis requires several settings to get a useful result. All these settings Dewesoft combine the CPB options part. It contains:

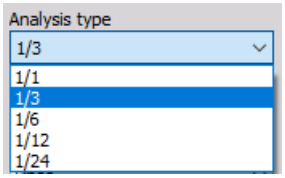
- Analysis type
- Y scale type - For detailed information about Y scale type see -> FFT X and Y scale type.
- Band display type
- Weighting - For information about weighting see -> FFT Weighting.

Analysis type

Dewesoft knows four different Analysis types for FFT analysis. The analysis type defines the width of each band.

The next band is calculated as 2^(AnalysisType) from the previous band, so for 1/1 this is 2^(1/1)=2, for 1/3 analysis this is 2^(1/3)=1,26 and so on.

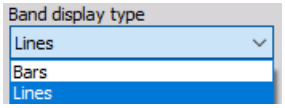
For 1/3 spectrum, there will be 10 bands per decade, for 1/12 there will be 40 and for 1/24 there are 80 values.



Band display type

Dewesoft supports two display types which can be selected from drop Band display type down list according to your application:

- Bars
- Lines

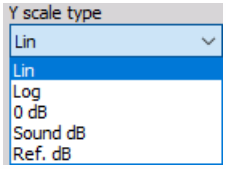


For information about this display types -> see Appearance on screen (above).

Y scale type

Dewesoft allows:

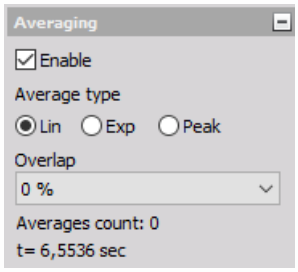
- four Y (amplitudes) axis scaling types (Linear, Logarithmic, 0 dB and Noise dB and Ref. dB scaled). Select the axis type from the drop-down list according to your application.


Y scale type
Lin
Lin
Log
0 dB
Sound dB
Ref. dB

Octave Averaging settings

Use averaging mode to get a more stable Octave display.

To activate the averaging just click the Enable check-box on Averaging section and all control become available.



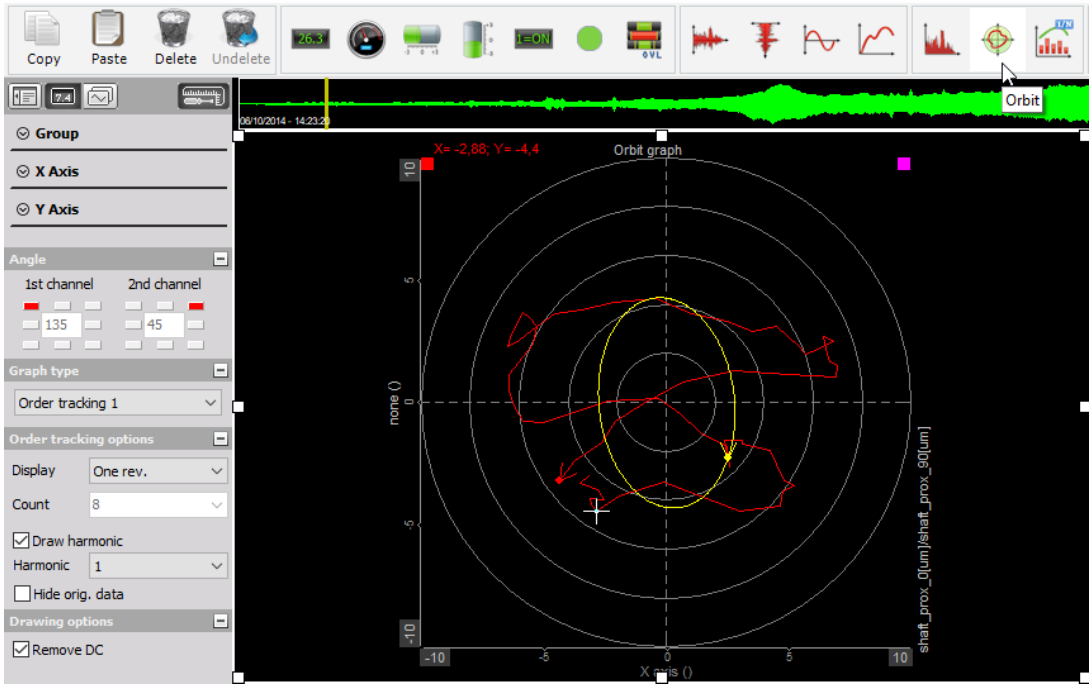
For detailed information about Averaging settings see -> FFT Averaging.

ORBIT PLOT

The Dewesoft orbit plot shows the x-y scope with a chance to rotate x and y axis. It is mainly used for displaying the axis movement in DSA analysis.

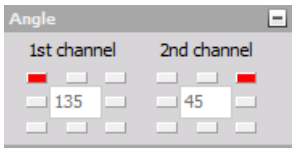
When you select orbit plot in the design or run mode, following settings will appear on left part of the screen:

- Control properties - For detailed information about Control properties: grouping, number of columns, Add / Remove, transparency,... see -> Control properties.
- Mounting angle - Describes the orientation of the sensor.
- Graph type - There are two basic graph types - Normal, where the inputs are any analog channels, or from Order tracking, where the inputs are only the channels which are used in Order tracking math module. Both graph types have common Drawing options. Available appearance setting for Orbit graph in Run mode is:
- Axis scale/auto scale
- Channels selector For detailed information about assigning/reassigning channels see -> Display settings.



Mounting angle

Angle is defined for first and second channel and depends on the mounting of the sensors.

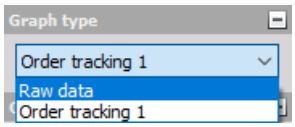


For example if the first channel angle is mounted from the top, we select 90 degrees. If it is from the right side, we select 180 degrees.

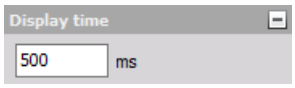
Please note that the sensors should not be mounted in a straight line - there should be an angle offset between them.

Graph type

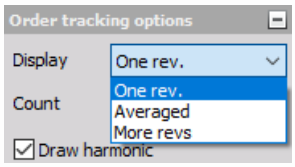
There are two possible graph types - Raw data and Order tracking.



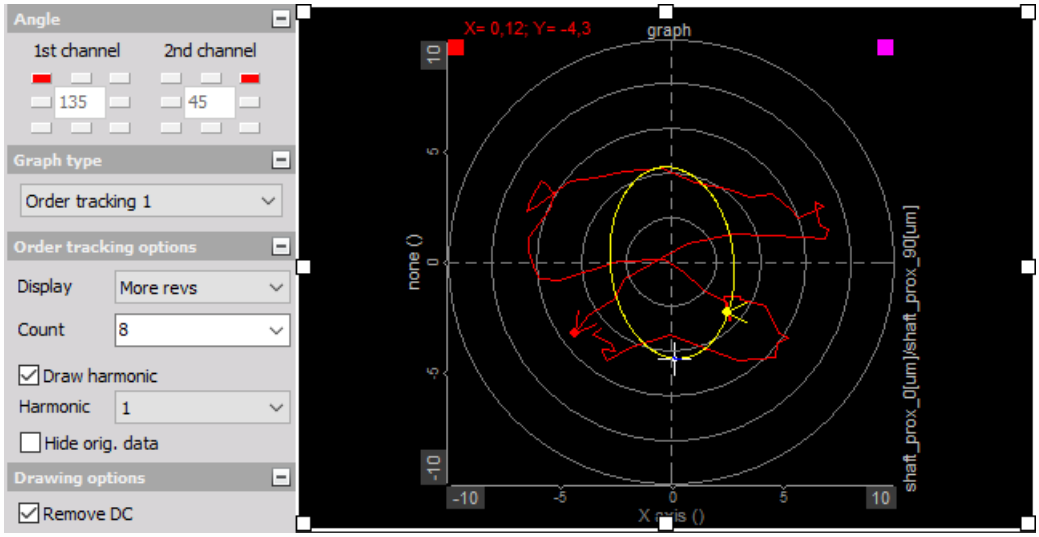
- Raw data mode means that the orbit plot will display the x-y plot from any two measured channels oriented at any angle defined by the Angle orientation. The only special setting is the Display time. This defines the time displayed on the screen.



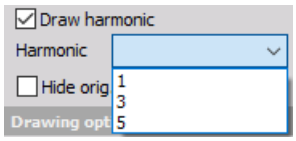
- In Order tracking mode the signal sources can only be channel used in the order tracking. Since order tracking defines the rotation frequency, we can display current rotation - One rev. mode, Averaged number of cycles or More revs (cycles). For last two modes we need to define the number of cycles to average or display. Please note that Order tracking needs to output also Phase angles. In other case the Orbit analysis will display a warning.



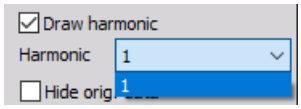
Please note that we display the large point on the graph. This is the position of the zero angle from the angle sensor of order tracking (if we use for example tacho or encoder sensor).



If we extract harmonics from the order tracking, we can also display the orbit of first, third and fifth harmonic on the display.



These harmonics must be defined in list of Output extracted harmonics as channels section of Order tracking module setup screen, otherwise only first harmonics is available on list:



Drawing options

The last option to set in both modes is Use DC, which, if it is checked, will remove the offset from the signal and will display the orbit in the center of the graph.

If this option is not checked Harmonics list is dimmed and values are not available.

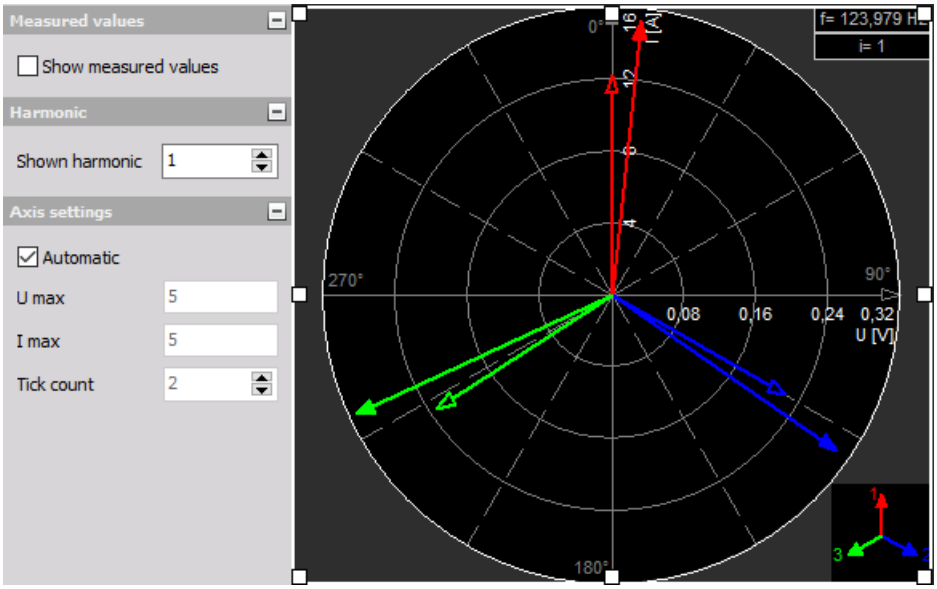
VECTOR SCOPE

The Dewesoft Vector scope instrument is used for displaying the amplitudes and phase angle between the voltage and current power module channels and additionaly to the vector also the most important measurement values for each phase: Ui, Ii, Phi, cos Phi, Pi, Qi and Si.

NOTE: **Vector scope instrument on Design tool bar is available only in case of selection at least one power module in Dewesoft Setup -> Power tab.**

When you select an icon of Vector scope in the design or run mode, following settings will appear on left part of the screen:

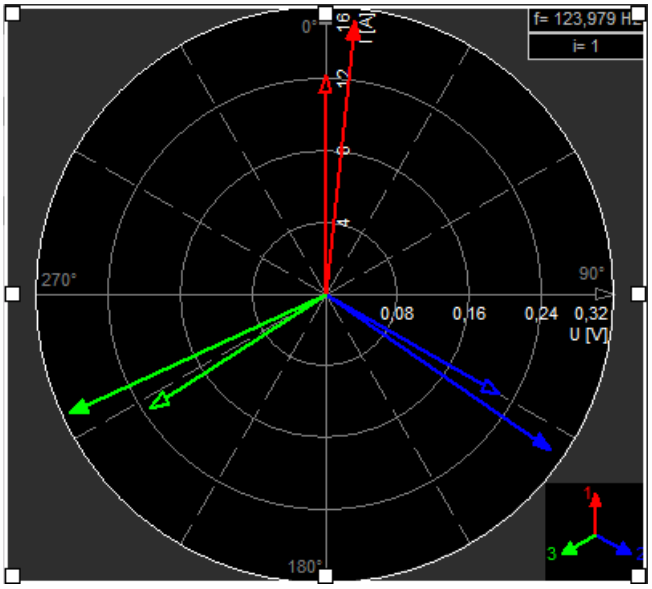
- Control properties - For detailed information about Vector scope Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.
- Vector scope settings - Available appearance setting for Vector scope instrument:
- Measured values
- Harmonic
- Axis setting
- Channels selector - For detailed information about assigning / reassigning channels to/from Vector scope -> see -> Display settings.



Appearance on screen

The Vector scope displays the phase angle between the channels and:

- channel names
- unit(s)
- frequency information
- typical values

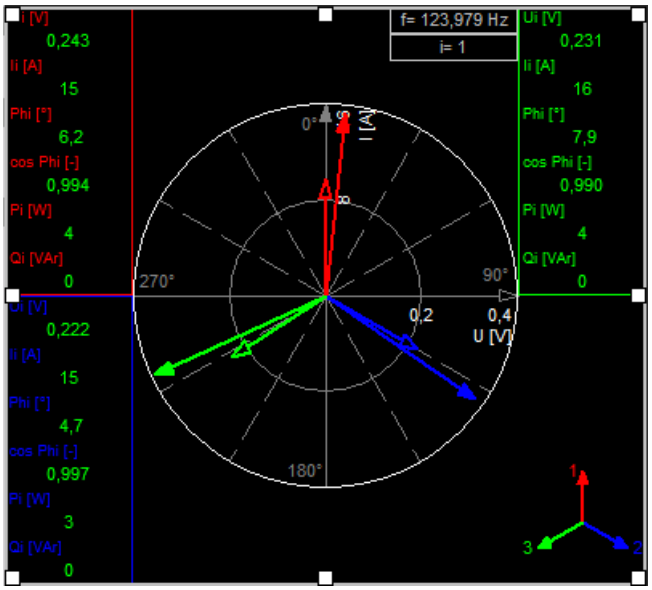


For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

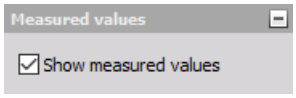
Measured values

Show measured values

This function shows in addition to the vectors also the most important measurement values for each phase: Ui, Ii, Phi, cos Phi, Pi, Qi and Si (where i is the number of the selected harmonic).



Use the Show measured values check-box to show important measurement values (see examples on the right and above).

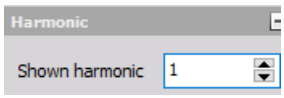


Harmonic selection

Shown harmonic

The Harmonic shown selection allows changing the displayed harmonic. You can choose from 1st to 50th harmonic.

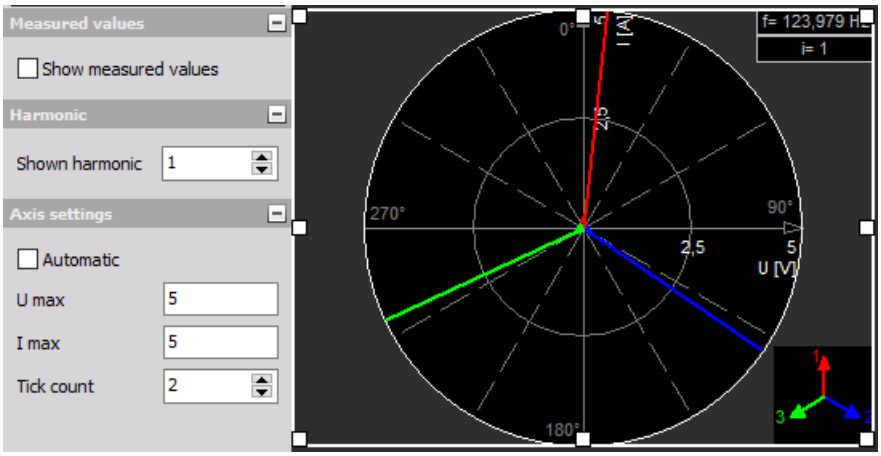
Use the Up / Down arrow to increase /decrease the displayed harmonic.



Axis settings

- **Automatic** - With Automatic enabled, the vector scope always scales to the maximum of all displayed channels.
- **Manual** - When Automatic is disabled, you can enter value for:
- U_max and
- I_max

Use the Up / Down arrow to increase /decrease number of ticks -Tick count.



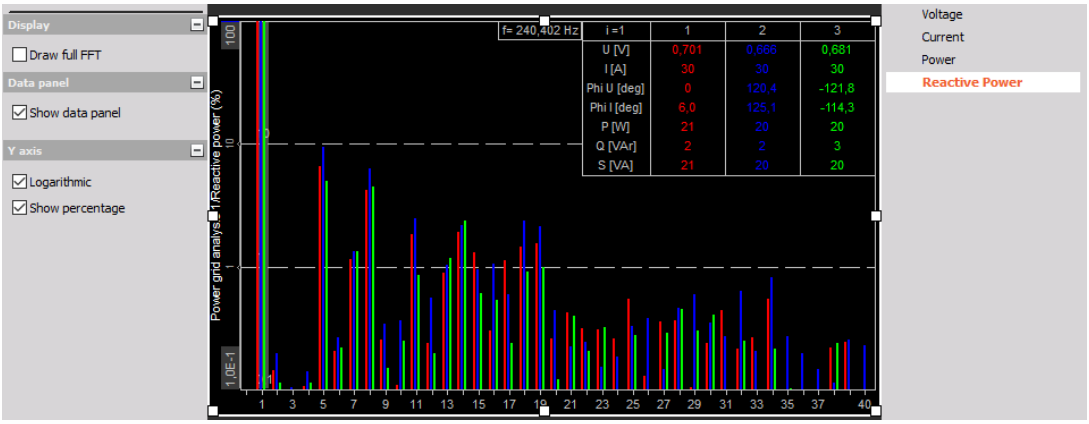
HARMONICS ANALYSIS

The Dewesoft Harmonics display shows frequency components of the input signals. Harmonics analysis is similar to the FFT analysis, the main difference is the type of calculation and displaying: the harmonics display refers to a base frequency (e.g. 50 or 60 Hz) and displays its harmonics.

NOTE: **Harmonics instrument on Design tool bar is available only in case of selection at least one power module in Dewesoft Setup -> Power tab.**

When you select Harmonics display in the design or run mode, following settings will appear on left part of the screen:

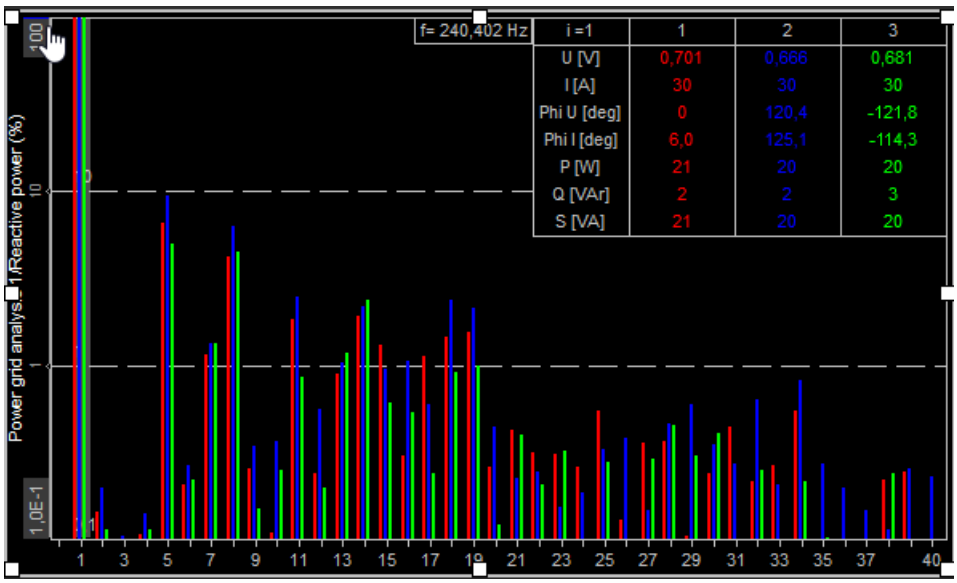
- Control properties - For detailed information about Harmonics display Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.
- Harmonics settings:
- Display value
- Draw full FFT
- Y axis display
- Logarithmic
- Show percentage
- Channels selector - For detailed information about assigning / reassigning channels to/from Harmonics display -> see -> Display settings.



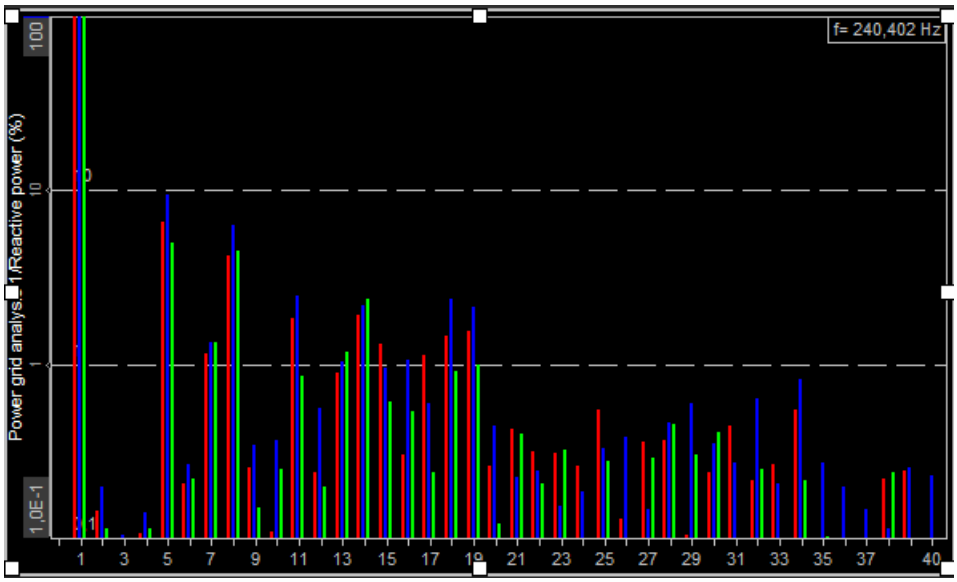
Appearance on screen

The Harmonics displays show: base and harmonic frequencies, channel name(s), unit(s)...

Harmonics display with Data panel and Logarithmic Y axis



Harmonics display with Logarithmic Y axis

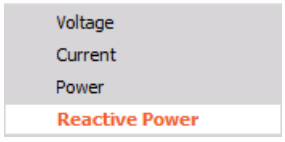


For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Display value

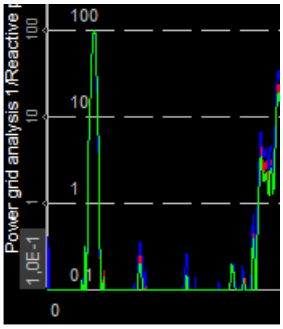
Dewesoft Harmonics display always shows all channels from one module at the same time. The Display value defines what the content should be from the channel selector on the right side:

- Voltage
- Current
- Power - active power
- Reactive power - this is wasted energy
- Line voltage



Draw full FFT

With check Draw full FFT check-box in Display value section instead harmonics bars full FFT spectrum can be displayed (see right).



Data panel

This function shows in addition to the bars also the most important measurement values for each phase: Ui, Ii, Phi, cos Phi, Pi, Qi and Si (where i is the number of the selected harmonic).

To display these values check the Show data panel check-box (example picture see above) in Data panel section.

To select a Harmonics, which values will be displayed in data panel, simply move the mouse cursor over the bars, a gray harmonics cursor (rectangle) will follow and indicates your selection.

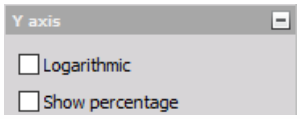
When you want to fix your selection press the left mouse button (on example above 1st harmonic is selected). To select another harmonic move the mouse to its position and left-click again.

If you want to release the harmonics cursor move the mouse to its position and left-click again. Now the harmonics cursor is 'free' again.

NOTE: **Harmonics cursor works only by checked Show data panel check-box.**

Y axis display

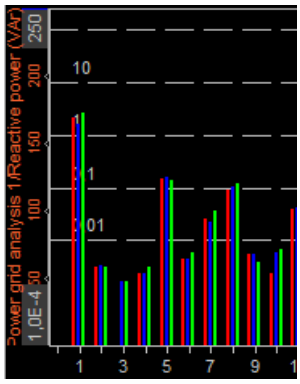
In this part of Harmonics display settings you can choose with check /uncheck appropriate box different Y axis scaling:



Logarithmic

Logarithmic check-box in Y axis section is:

- Selected -> logarithmic Y axis scaling

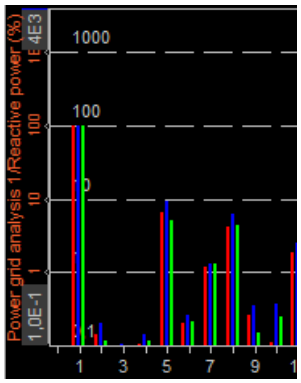


- Unselected -> linear Y axis scaling

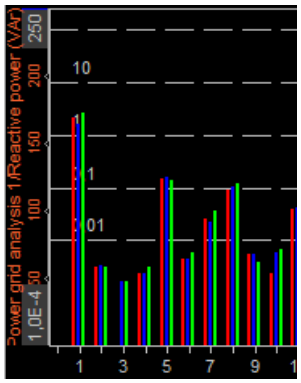
Show percentage

Show percentage check-box in Y axis section is:

- Selected -> the Y axis is scaled in %



- Unselected -> the Y axis is scaled in it’s units (V, A, W or Var)



VIDEO DISPLAY

Dewesoft Video display is provided to show content sensitive acquired videos together with other different data of measurement in various instruments. This video information can help to interpret and to document these data and measurement.

NOTE: **ideo display possibility on Design toolbar is available only in case of physical connection and set up of camera in Dewesoft Setup -> Video tab.**

When you select Video display in the design or run mode, following settings will appear on left part of the screen:

Appearance on screen

The Video display present content sensitive acquired video information.

- Control properties - For detailed information about Video display Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.
- Camera settings - Scaling setting and information about selected camera for Video display.
- Cameras selector - Assigning/reassigning cameras to Video display.

For detailed information about instruments positioning, size and transparency see > Screen edit functions.

Camera (Video) setting

Camera information

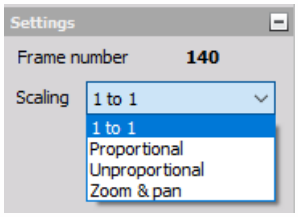
In the first part of this information screen area is displayed Frame number of selected camera.

Scaling

Dewesoft allows select Scaling of displayed video on Video display:

- 1 to 1
- Proportional
- Unproportional
- Zoom&pan

Select the scaling type from drop down list according to your requirements.



- Proportional to available display width

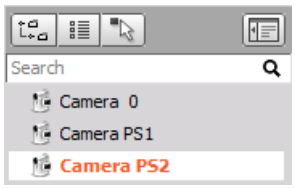
- 1 to 1 - display to acquired video)

- Unproportional - fill available display

- Zoom&pan - zoom or pan

Cameras selector

It will display the available cameras to select the right one.



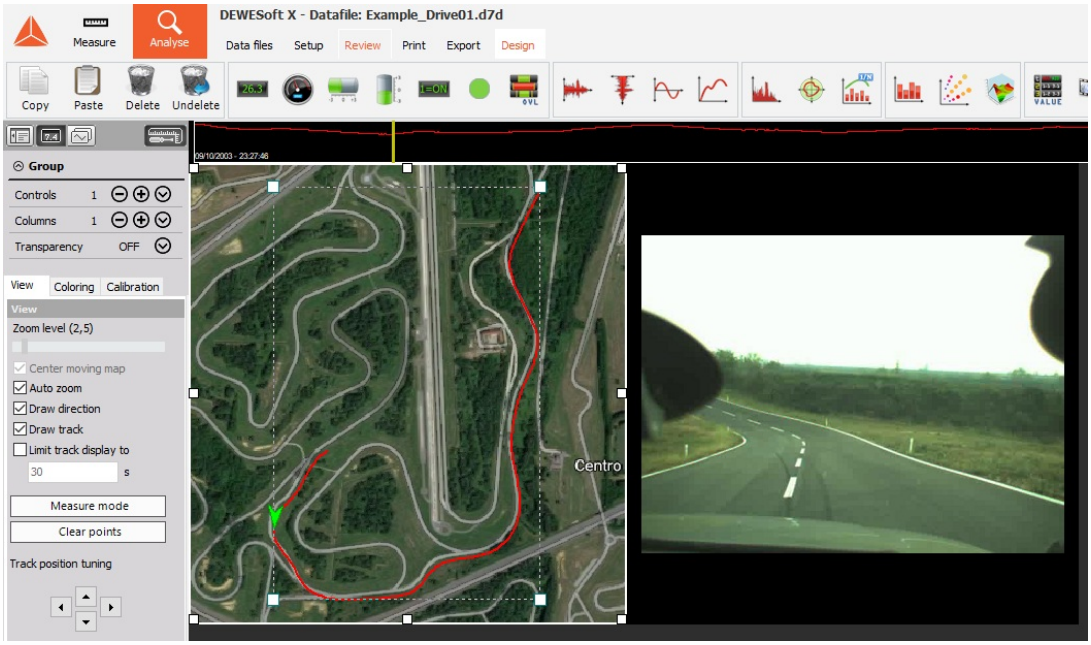
2D MAP

The GPS display is provided by Dewesoft to show different acquired data from GPS position and heading information. Different data can be calculated from the GPS channels.

NOTE: **GPS display possibility on Design tool bar is available only in case of physical connection and set up of GPS in Dewesoft Setup -> GPS tab.**

When you select GPS display in the design or run mode, following settings will appear on left part of the screen:

- Control properties - For detailed information about GPS display Control properties: grouping, number of column, Add / Remove, transparency,... see -> Control properties.
- GPS settings Typical setting for GPS instrument are divided in three modes:
- View
- Coloring
- Calibrate map
- Channels selector - For detailed information about assigning / reassigning channels to/from GPS display -> see -> Display settings.



Appearance on screen

The GPS instrument consists of three parts of display:

- the course
- an arrow displaying the direction
- a scale indicator at the right bottom area of the instrument

For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

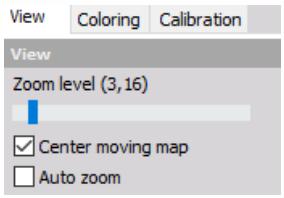
Settings

The GPS instrument can be switched in two modes using the tabs:

- View
- Calibrate map

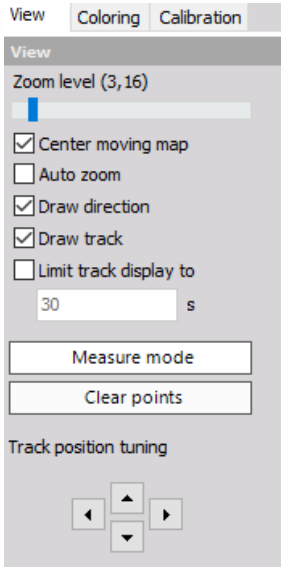
View setting

- Zoom - Dewesoft offers two ways to scale the track: Auto zoom or zoom manually.

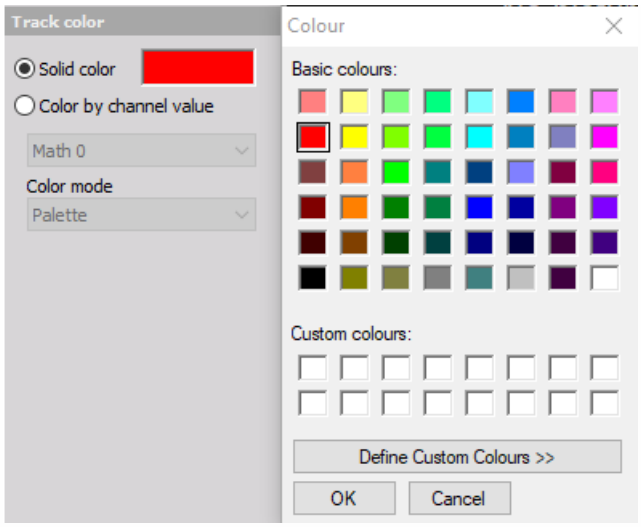


Use the Zoom level slider to zoom manually; above the slider after caption, you can see the zoom factor.

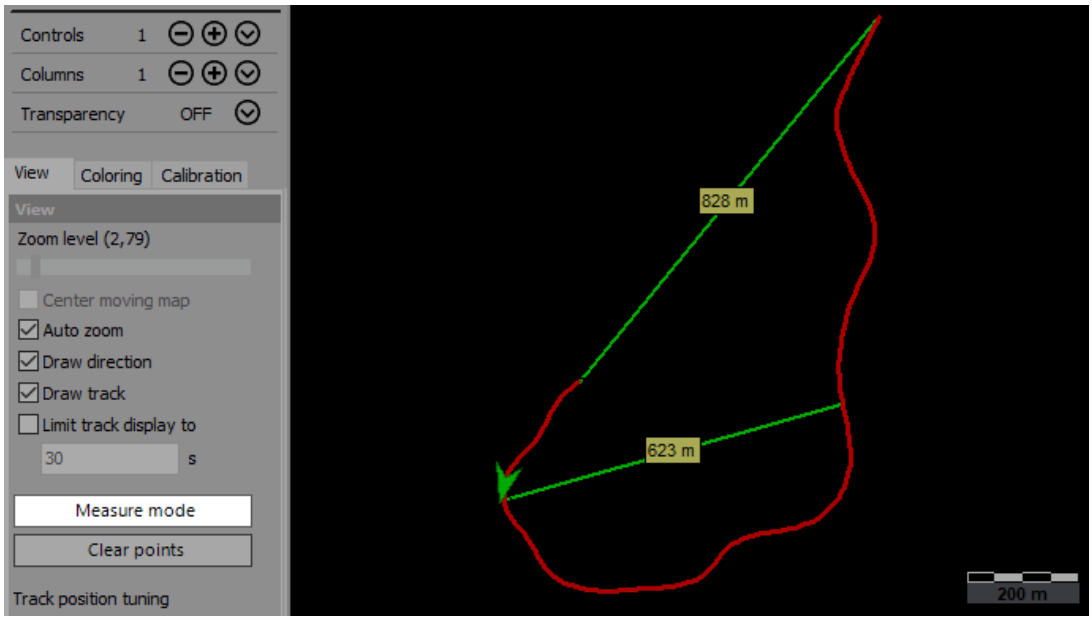
- Use the Center moving map option to keep the track centered on the screen.
- When you select the Auto zoom option, the track will be centered automatically in the same way than described above.



- Use the Track color to change the color of the displayed track. This feature is very helpful to achieve a good contrast to background maps.

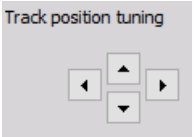


- When you select the Measure mode you can measure distances within the map. Simply click a point within the map for the starting point. Move the cursor to the second desired position and click again - a line will be drawn with the distance labeled:



You can make as many measurements as you want. To remove the measurements, press the Clear points button.

- Track position tuning - When the Auto zoom is deselected, you can use the arrow buttons to move the track within the map. You can also use the mouse: click on the track, keep the mouse button pressed and move the track to the desired position.



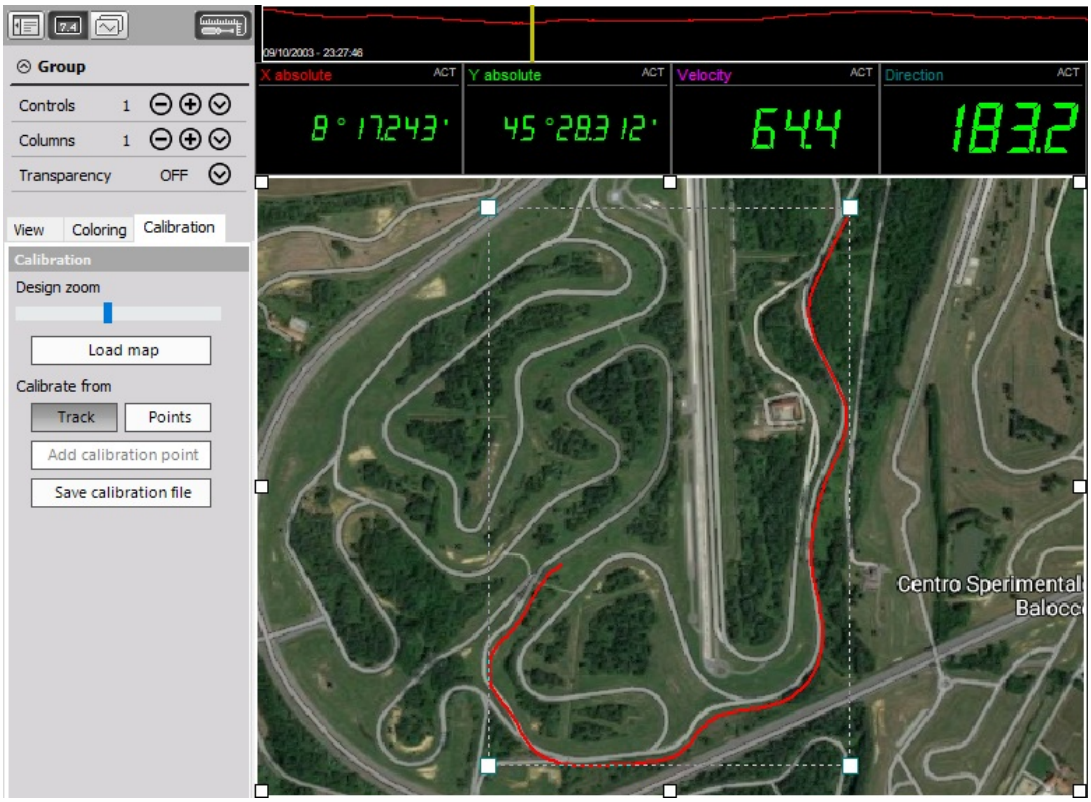
Calibrate Map

Dewesoft offers the possibility to display a background image behind the track. As a standard, the image will be a road map. To calibrate the map you have to perform several steps.

Calibrate from track

First press the Load map button to load the map from your system - maps directory. Accepted file formats are *.bmp or jpg.

Now you can define the map position. When Calibrate from -> track is selected, click on the map image, keep the mouse button pressed and move the map image as required. To re-size the map image, use the Design zoom slider.



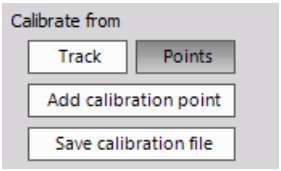
If we don't know the coordinates of the map, we can take a short tour in the area storing the measurement. Then we go to GPS screen, go to Calibration and Load map. The map is shown in behind and the traveled path gets four handles around it. Now we can drag and adjust those handles to match the position and map.

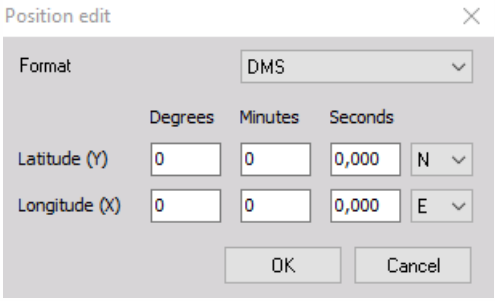
If we know the exact coordinates on the map, we can also enter them by switching to Calibrate from -> points mode. Once this is adjusted, we can Save calibration file. This map will be from that point on always shown behind each measurement. We can also have multiple files for different zoom levels. We can have an overview file like above and when we zoom in, it will switch to the photography.

Calibrate from points

If Calibrate from -> points is selected, you can add calibration points. Press the Add calibration point button first and then the point within the map. A Position edit new window will appear where

you can enter the GPS position.





Add several points to complete the calibration by points and make it more accurate - at least two points are required.

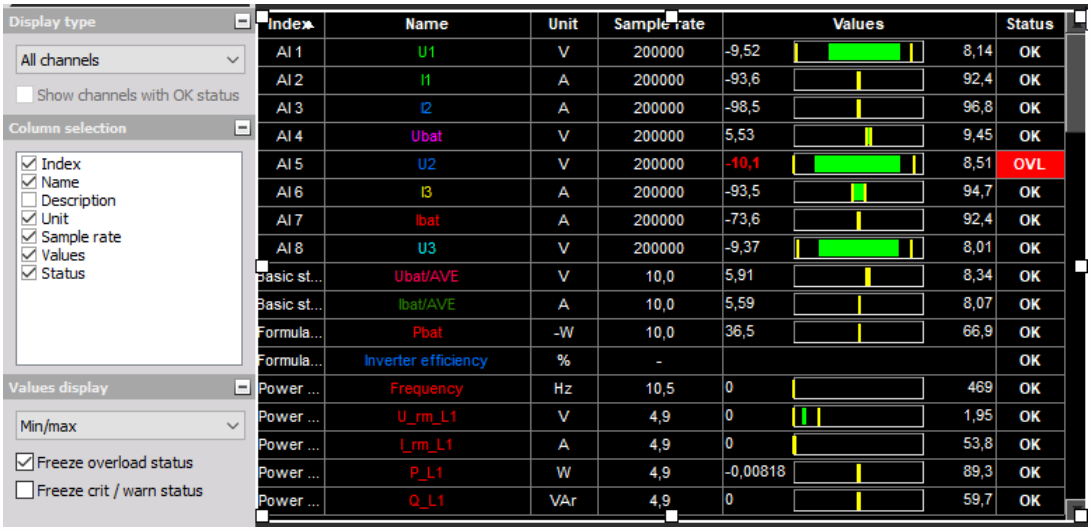
After you have done these settings, press the Save calibration button to store the calibration settings. These values will be loaded automatically when you analyse the recorded data.

OVERLOAD INDICATOR

Together with other different data of measurement in various instruments Dewesoft provide Overload indicator display to show list information about overloaded signals.

When you select Overload indicator display in the design or run mode, following settings will appear on left part of the screen:

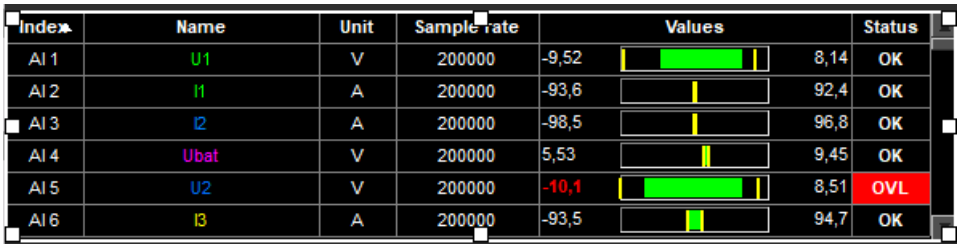
- Control properties - For detailed information about Overload indicator display Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.
- Overload indicator settings - Typical setting for Overload indicator instrument are: Display type , Column section.
- Channels selector - For detailed information about assigning / reassigning channels to/from Overload indicator display -> see -> Display settings.



Appearance on screen

The Overload indicator display in tabular form channels information about:

- Index
- Name
- Description
- Unit
- Sample rate
- Values
- Overload



Index	Name	Unit	Sample rate	Values			Status
AI 1	U1	V	200000	-9,52		8,14	OK
AI 2	I1	A	200000	-93,6		92,4	OK
AI 3	I2	A	200000	-98,5		96,8	OK
AI 4	Ubat	V	200000	5,53		9,45	OK
AI 5	U2	V	200000	-10,1		8,51	OVL
AI 6	I3	A	200000	-93,5		94,7	OK

In Values column also channel min and max values is displayed, bar graphically shows the current levels of the signal with their limits. In Overload column overloaded signals are designated graphically.

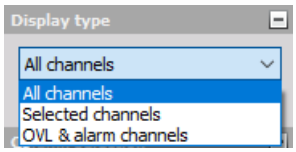
For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Overload indicator settings

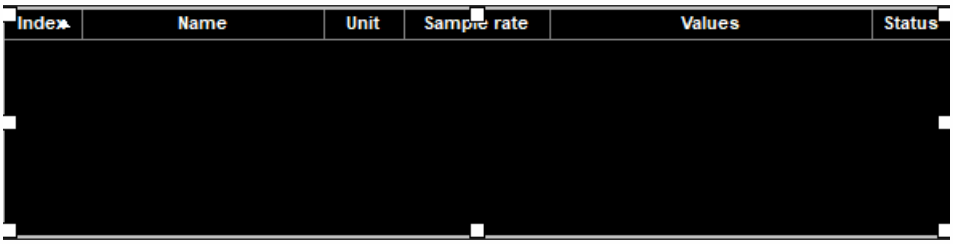
Display type

The Overload indicator display can show channel information for:

- All channels
- Selected channels



Select the Display type from the drop-down list according to your requirements. When we select All channels option then display the Overload indicator information for all channels. When we select Selected channels then empty Overload indicator appear:

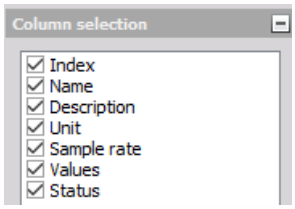


and in Channels selector list of all available channels appear. From this list we can choose channels to display it in Overload indicator table.

Column selection

Dewesoft allows select columns which are displayed on Overload indicator list.

Simply checkbox in Column selection in front of the desired column name to show this column on Overload indicator (see below).

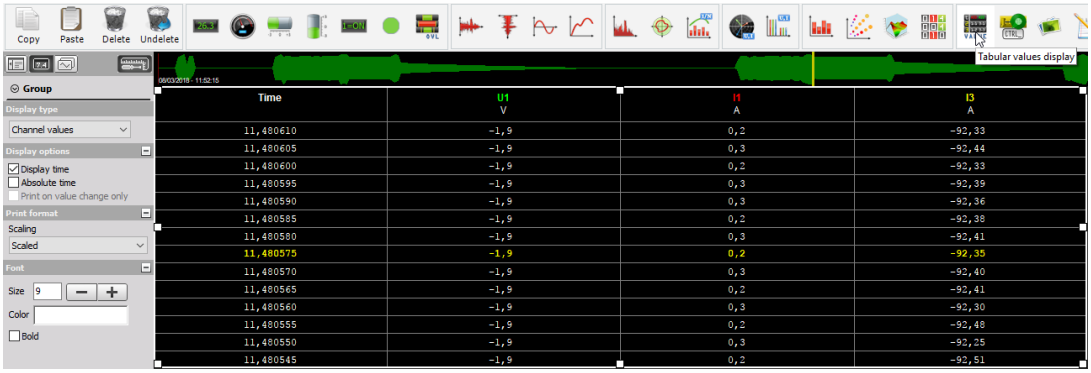


TABULAR DISPLAY

Dewesoft provides together with other different instruments also Tabular values display to show list of measured data and related time.

When you select Tabular values display in the design or run mode, following settings will appear on left part of the screen:

- Control properties - For detailed information about Tabular display Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.
- Tabular value settings - Typical setting for Tabular values display are: Display options , Print format.
- Channels selector - Please note that the tabular display is the only display which can also show CAN messages as a hexadecimal value. For detailed information about assigning / reassigning channels to/from Tabular display -> see -> Display settings.



Appearance on screen

The Tabular values table display in separate columns:

- Time data
- Values of channels

For every in Channel selector selected channel singly column with values is displayed (see picture above). When Absolute time is unchecked, the Tabular values table is displayed without Time column.

For detailed information about instruments positioning, size and transparency see -> Screen edit functions.

Tabular values display settings

Display options

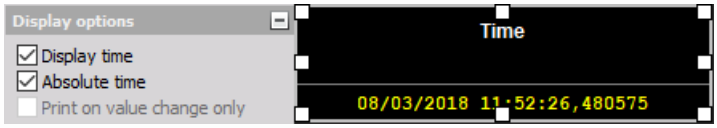
The Tabular values display offer three possibilities:

- Display time
- Absolute time
- Print on value change only

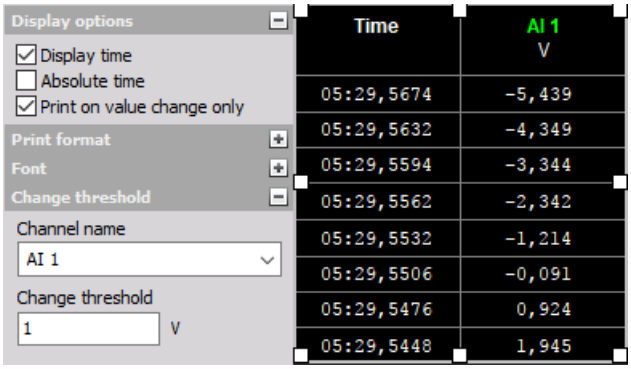
Simply checkbox in the Display option to:

- **Display time** - When we select this field check box, time data from measurement start in predefined format is displayed, if it is not checked, the Time column is hidden. All values of the selected channel are displayed.

- **Absolute time** -When we select this field check box, the date is displayed additional in absolute time format. All values of selected channels are displayed.



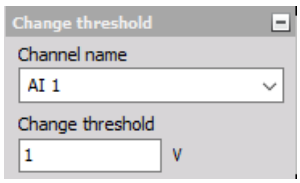
- **Print on value change only** - When we select this check box, additional part of Tabular values setting - Change threshold section is displayed (description see below). Only selected channel values which meet condition defined in Change threshold field are displayed.



Change threshold

Dewesoft provide on this part of Tabular values display settings:

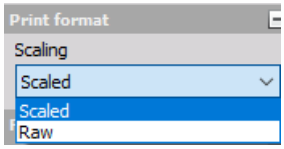
- Channel name - In this combo box the channel tow which the threshold value will be assigned to is displayed.
- Change threshold - Field to enter limit difference of channels values when this value will be updated displayed in list.



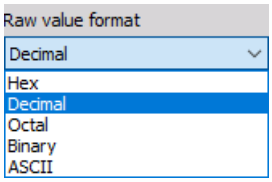
Print format

The Tabular values display offer settings for displayed channels values:

- **Scaling** Select the Scaled or Raw type from drop down list according to your requirements. Usually the scaled value is displayed, but sometimes it is nice to see the raw channel values, especially when data is transmitted digitally (CAN, PCM or other digital buses).



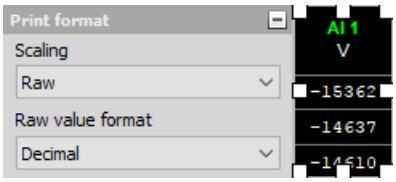
- **Raw value format** Select the Hex, Decimal, Octal or Binary format type from drop down list according to your requirements. This is valid only when raw values will be displayed.



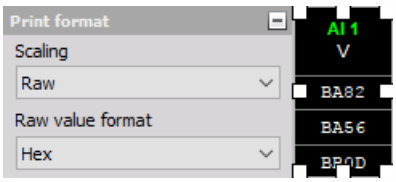
Examples different Tabular values formats:

- Scaled Raw Scaling

- Decimal Raw Scaling



- Hex Raw Scaling

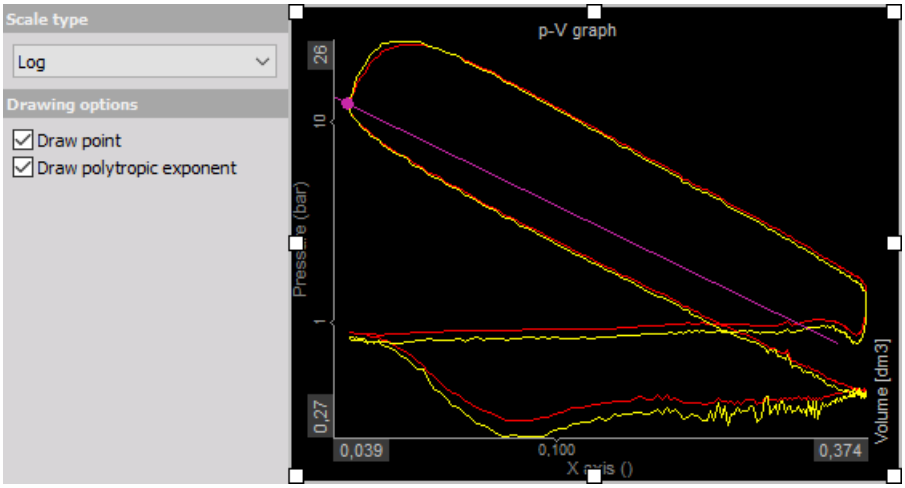


CA P-V

The Dewesoft Combustion analysis p-v plot shows the x-y scope volume vs. pressure in the cylinder. The combustion math module should be used that this graph can be chosen.

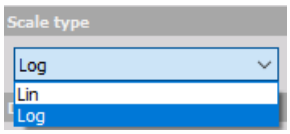
When you select p-v plot in the design or run mode, following settings will appear on left part of the screen:

- Control properties - For detailed information about Control properties: grouping, number of column, Add / Remove, transparency,...see -> Control properties.
- Scale type - Describes the orientation of the sensor
- Drawing options - There are two basic graph types - Normal, where the inputs are any analog channels, or from Order tracking, where the inputs are only the channels which are used in Order tracking math module. Available appearance setting for CA p-v graph in Run mode are: Axis scale/auto scale (Common instruments tools), Online calculation
- Channels selector - For detailed information about assigning/reassigning channels -> see -> Display settings.



Scale type

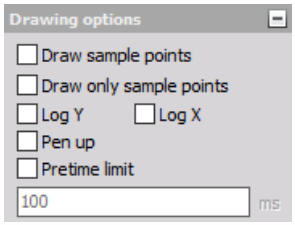
The scale can be displayed either as linear or logarithmic (for volume and pressure). Linear axis is common used type for seeing the real value of the pressure, while logarithmic have two advantages: the pumping cycle can be seen very nicely and also the polytropic expansion and compression are linear in log-log scale.



Drawing options

We have also an option to draw start of combustion and end of combustion points. This shows the points where start of combustion and end of combustion happens (calculated from the heat release).

Additional option with logarithmic scale is to display the Polytropic exponent. This is used to see how good the entered polytropic coefficient fits to the measurement data.



CONTROL ELEMENT

Dewesoft provide new control elements, allowing users to control Dewesoft actions like start and stop or to directly influence outputs, like function generator parameters.

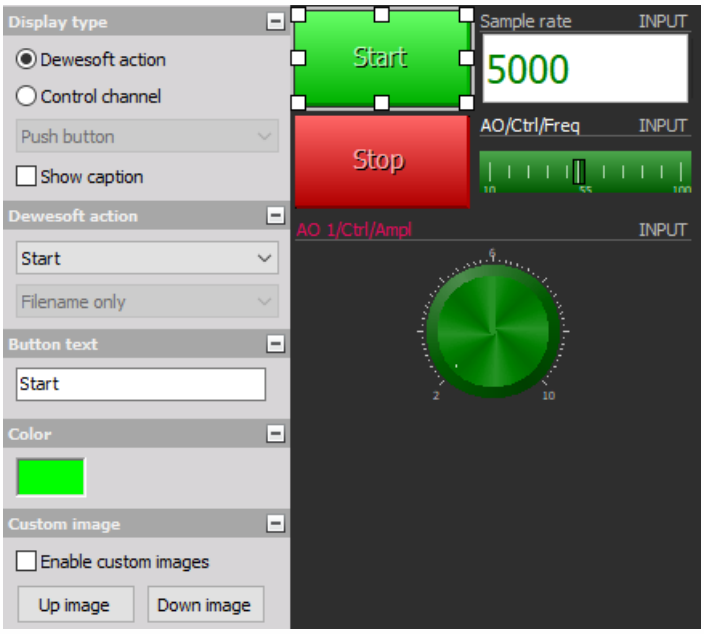
Control element has two basic modes of operations:

- Dewesoft action to control Dewesoft for example in full screen.
- Control channel operation to manually control function generator or for example some of the digital outputs.

In the Dewesoft action mode, only push buttons are allowed. There are several actions possible:

- Start - will start the measurement from Stop mode
- Stop - will stop the measurement (and storing)
- Pause - will pause recording, it is actually pause/resume toggle - if paused, it will resume measurement
- Freeze - will freeze the recorders if Freeze buffer is enabled in Project setup
- Store - will start storing
- Trigger - will issue manual trigger in Store mode
- Screen select - will select the screen named the same as Action string
- Keyboard event - will make the Keyboard event
- Set channel - will set channel
- File name - will show file name
- Sample rate - will show sample rate
- Exit Dewesoft - will exit the Dewesoft

Action string defines the name of the button.



When using Control channels, we can display Control element as:

- Input field - to manually enter a new control value
- Push button - to allow short on/off event
- Switch - to switch between two or more states (could be defined by the control channel)
- Turn knob - to allow smooth transition between values defined by min and max
- Horizontal / vertical slider - same as turn knob, but linear
- Check box
- Drop-down list
- Up down list

For turn knob, horizontal and vertical slider we can also define minimum and maximum limit. In analyse mode Control element has no function.

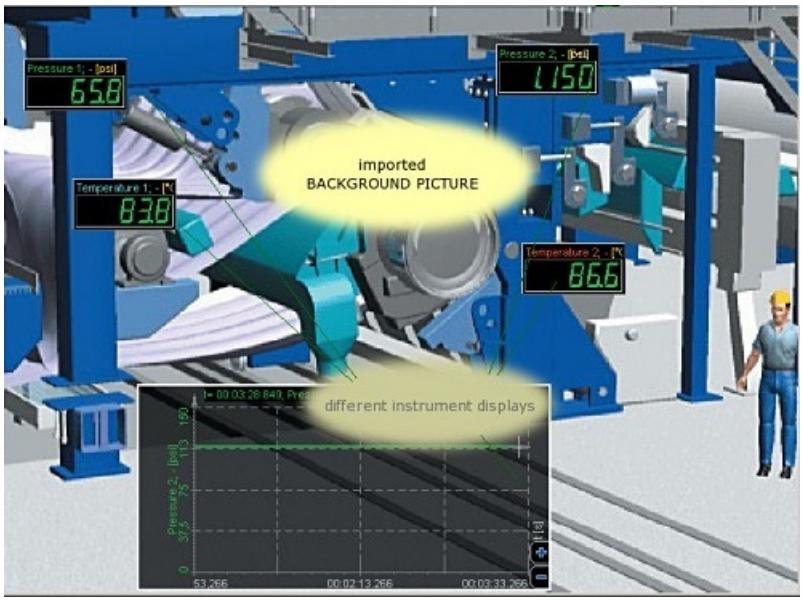
ADDITIONAL CONTROLS

Dewesoft offers an additional data displays and controls to create instrument appearance on the online display screen.

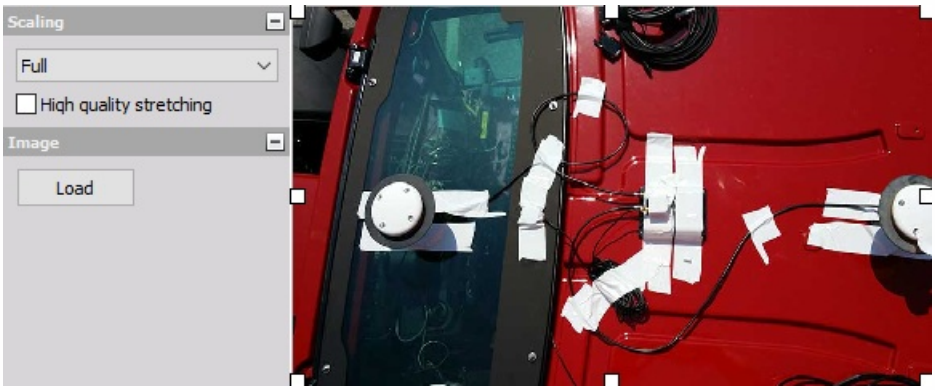
- Background picture - used to illustrate your measurement
- Text element - used to write text on the screens
- Line element - used to draw lines and shapes, connect different elements,...

Static image

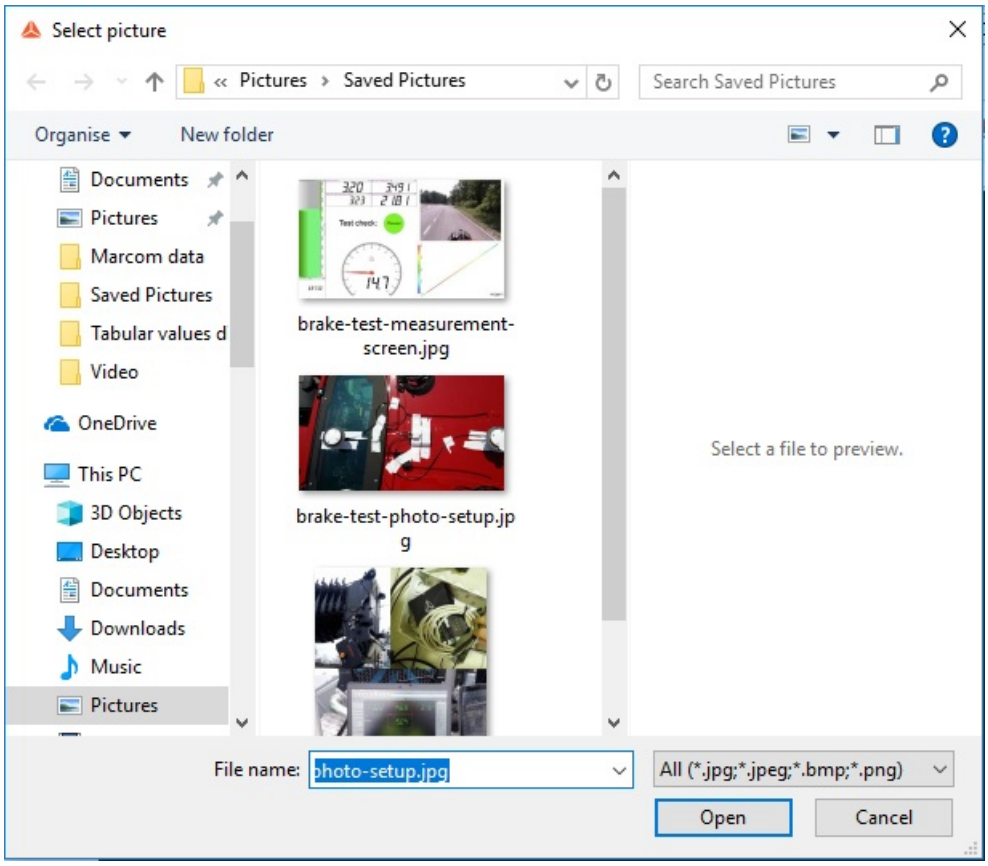
The static image can be used to illustrate your measurement or as base picture to place different instrument displays with measured data on it.



When you select the Static image control in the design mode, the empty placeholder for the picture is placed on display. We can resize it and place the image onto the placeholder by pressing the Load button in the control setting on the left side.



A standard OS dialog will open allowing to load any jpg, jpeg, bmp, ico, emf and wmf image. Pressing Open will load the image.



We have an option to Scale the image proportionally to the size of the placeholder with Full option, we can scale it to proportionally or keep the original size.

Text element

The Text element can be used to write any text at any position of the screens like caption, comments and reminder.

When you select the Text notice in the design or run mode, on left part of the overview screen a text controls will appear:

Note that this the Text element don't have any Channel selector.

Text element appearance

The Text element is very simple: just a field where you can enter your text.

Positioning and size

Are same as by other elements (instruments) see -> Instruments appearance.

Font and paragraph properties

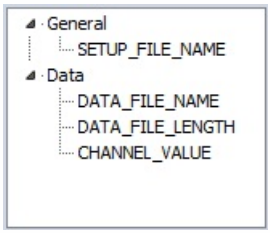
In the Font section, you can define the text styling in same way as by other Windows text editors. Choose from all installed fonts on your Windows system, define the font Size, Color and appearance like Bold, Italic or Underlined.

The Paragraph section contains two features: the Alignment of your text (Left, Centered or Right) and the Word wrap (active by default).

Text editor

Press the Edit button on Text editor section or simply double-click on the text field to enter or change the text.

In Edit mode we can also enter some variables, like setup file name, data file name, data file length or global header entry. For the global header, we need to write the name of the header entry, for example .

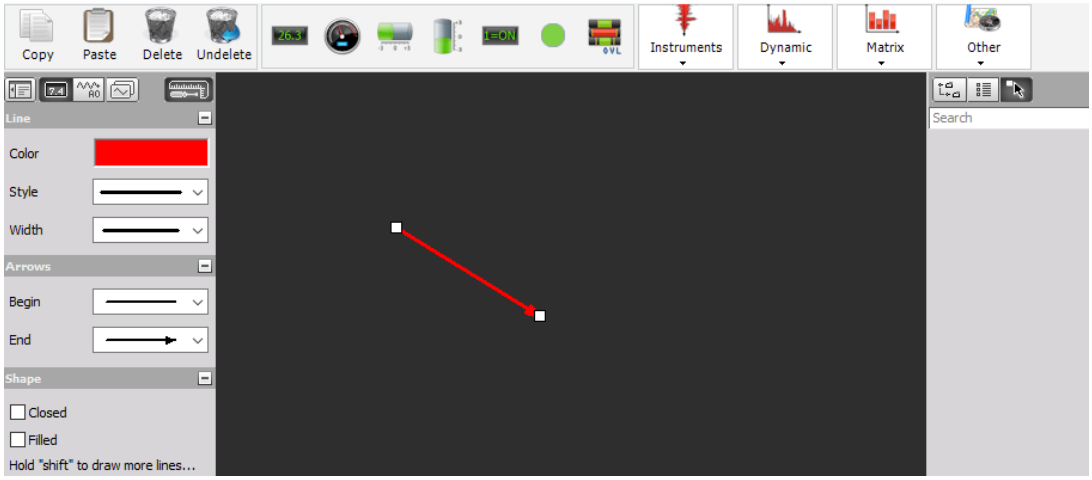


When you have finished, simply click anywhere outside of the text element or press once again Edit button to confirm the changes.

Line element

The line control can be used to draw lines, connect different elements, mark something,...

When you select the Line control in the design or run mode, on left part of the overview screen a line controls will appear:

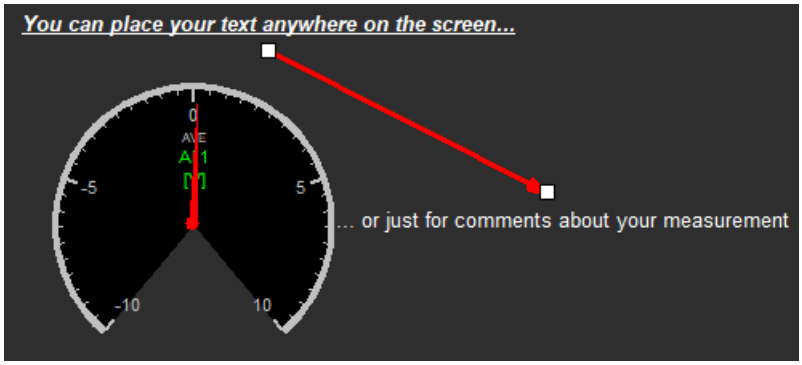


NOTE: **This the Line element don't have any Channel selector.**

Line element appearance

The Line element is very simple: just click once where you want to start your line and the second time where it should end.

Example: Line appearance between two Text elements



Draw shapes

In Shape section you must first check Closed (and in case of need Filled) field, then:

1. you draw line with two points (first two corners of shape) as described above 2. move mouse cursor to third corner of shape, press and hold Shift key on keyboard and when you left click the shape appear; while pressed left mouse button, you can move this corner on desired location; with release left mouse button (and consecutive Shift key) shape is drawn 3. on this way you can add fourth, fifth,... corner

Positioning and size

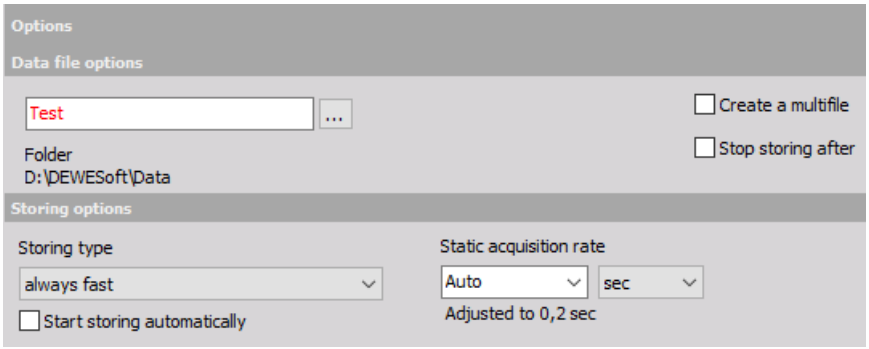
- To modify a line, simply click once on it to select element and then move the end points to the desired new location. In the same way you can modify shape -> change position whichever corner. When cursor is over start/end point of line or over corner of shape, change to 'hand' and modifying is possible.
- To move whole line or shape, simply click once on it to select it and then move (with cursor on line/shape and with pressed left mouse button) whole line or shape to the desired new location.

Line, arrows and shape properties

Use the Line, Arrows and Shape sections to style up your line with different colors, widths, arrows,... - please try out the functions to find the best for your requirements.

RECORDING ACQUIRED DATA

Measured data are stored in Dewesoft *.d7d (Dewesoft data file). We predefine the filename to be used for data recording and some other recording features on the Dewesoft File details and Storing setup screen:



Data file options

- Setting the filename - setting filename and folder
- Create a multifile - setting filename of multifile and conditions for make new file after
- Stop storing after - setting “after” condition for stop storing
- Save multifile option - save multifile settings in setup file

Static/Reduced rate - Selecting and setting reduced rate data storing.

Storing options

Types of storing data - selecting storing strategies that relate to the basic sample rates Start storing automatically - automatically activate data recording Start / Stop Recording

How data recording can be activated:

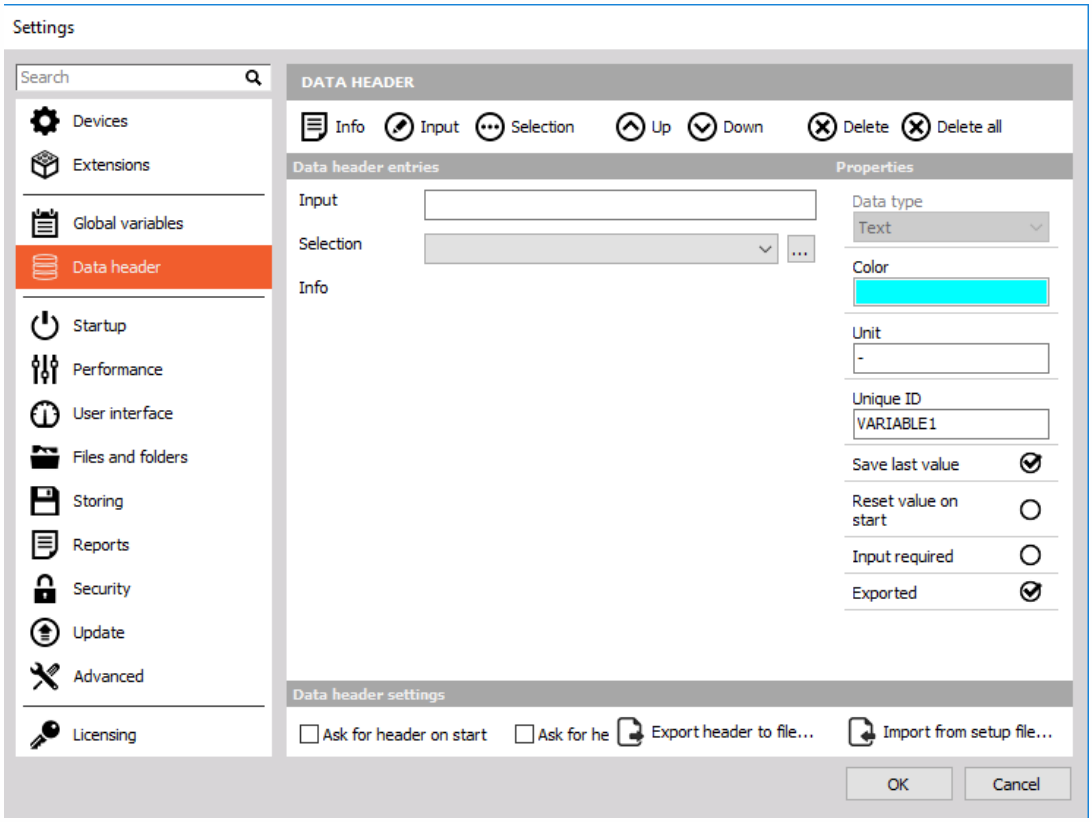
- Manually
- Triggered
- Automatically

With Event markers, which are used to mark areas of interest in the data record for later review

Special Dewesoft Measure procedure while recording data is Freeze mode - only displays are stopped - frozen with the last data.

HEADER INFORMATION

Additional information about the files can be defined in Settings -> Project options as header information. The header is usually shown and entered in the Ch. setup -> File details, but we can define in the header settings to show the header either/or at the start and at the end of the measurement. When we do so, a pop-up window will appear and allow us to define the header data.



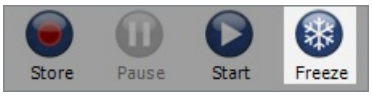
When all the values are entered, leave the window with the OK button.

For more information please see Data Header.

FREEZE MODE

Sometimes it is necessary to make a fast data analysis also during measurement. For this case, Dewesoft offers a freeze mode, where most displays (except of FFT, Octave, Video, GPS) are stopped - frozen with the last data.

Press the Freeze button to get the same analysis functions like in Analyse mode, including zoom and window zoom functions.



NOTE: **You only get a limited amount of data (smaller analysis time) restricted by a buffer memory in Dewesoft.**

During the freeze mode, all functions like storing or trigger are still working. You only freeze the current displays.

To run the displays again simply click the Freeze button again.

START AND STOP RECORDING

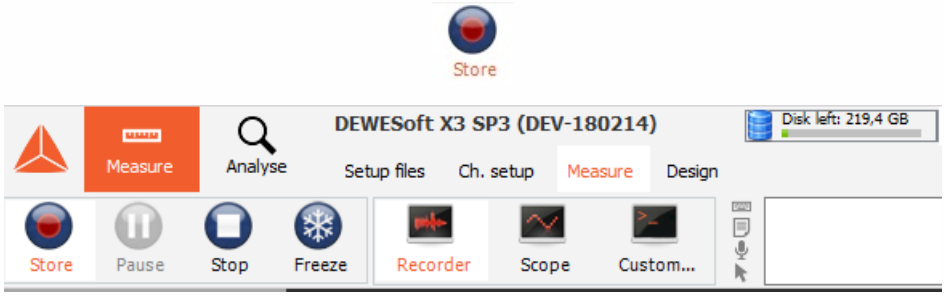
With Dewesoft you can activate/deactivate recording in three ways:

- Manually - using the Store, Pause, Resume and Stop buttons on the screen; with Event markers, which are used to mark areas of interest in the data record for later review
- Triggered - by setting any channel(s) to start and stop the recording according to levels
- Automatically- start storing automatically

Drawing options Manually

Start data recording

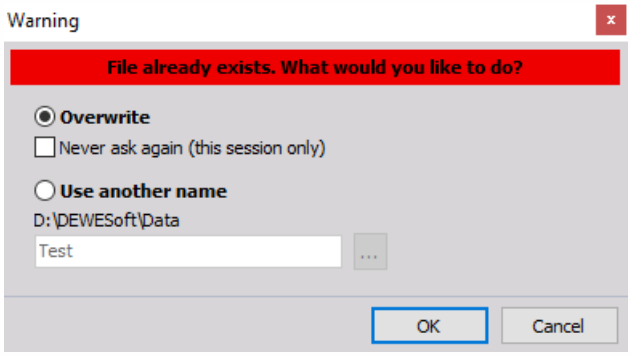
It is as simple as pressing the Store button on the main toolbar or press F5 on the keyboard.



On the right top part of this screen, you will see the currently available space on your hard disk drive.

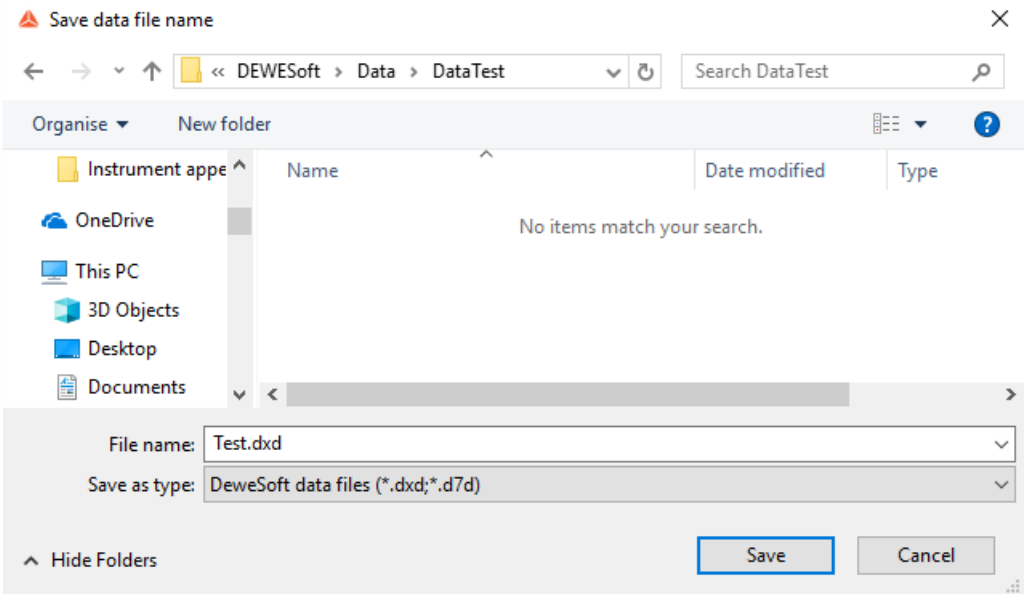
As you can see in the example screen, the hard disk is nearly full - better free some hard disk memory before you start the acquisition or select another file location.

If a file already exists of the same name as the one that you are trying to record, the following Warning window will appear (the Data header windows appear if on the Global header design window Ask for header on start option is selected:

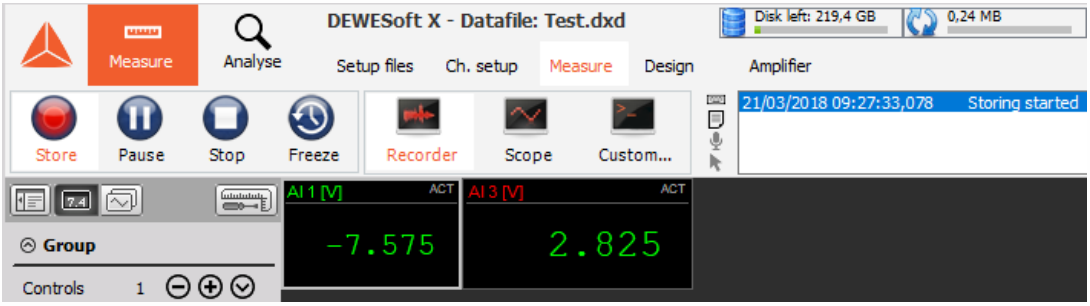


On this window you can:

- overwrite the existing file with selecting Overwrite option; there is an additional option - Never ask again to always override the file without asking; so this form will not be shown anymore until Dewesoft is restarted
- change the name here with selecting Use another name option and enters new name or selecting the " … " button to call standard Save data file name window to select new name).



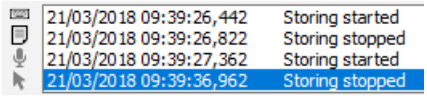
- Cancel recording altogether with selecting Cancel button. Make your choice and then hit OK to begin recording. The top of the screen will show you the current filename, how large the file is at all times, and other important parameters. The Store button itself changes color and is labeled with Pause to indicate that recording is activated:



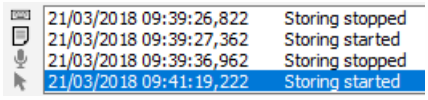
If data was already shown on the screen it is cleared and a red vertical line indicates where recording began. In addition, the scrolling EVENT box at the top-right of the screen tells us that "storing started at [date] [time]". The Store button on the main toolbar now changes to Pause button.

- Pause recording - the Pause function gives you the possibility to stop data storage for some time.

After pause the Resume button appears and when pressed storing is resumed within the same data file.

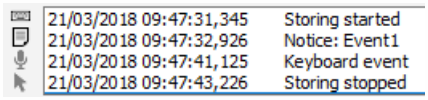


- Stop recoring - To stop recording click the Stop button or press F7 on the keyboard.



Marking your data with events

Conventional chart recorders have event markers, which are used to mark areas of interest in the data for later review. Dewesoft builds on these capabilities by providing three ways of marking your data:

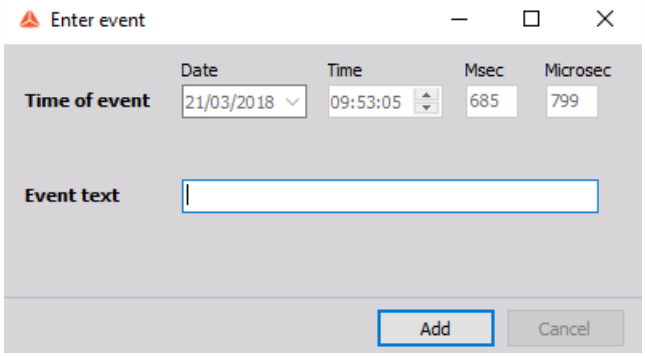


- **Keyboard events** - Press the spacebar or the icon to mark an area of interest.

A thin gray vertical line will appear on the data at the location, marked with a " K " symbol, and a notice is added to the scrolling EVENT box.

- **Notice events** - Press < n > key on your keyboard or the icon and then type some text in.

The text input dialog box will appear:



When you press the Enter key, the text is added to the record at that point. You can also click the OK icon on the screen to enter a notice event. The event is also marked with thin gray vertical line and an " N " icon on data at location when we started entering the event.

- **Voice events** - If you have a DirectX sound card installed, and it has been set up under the General Setup section of the System menu, you can use a microphone to make spoken notices to your recorded data. Simply press and hold down the < v > key on your keyboard, or click the microphone icon and then speak into the microphone.

Release the key when done. These voice events can be replayed through your speakers later on, and are a great way to add a rich layer of information to your data. This event is marked with the " V " icon and a thin vertical gray line.

V

Events can be added, edited or deleted also in Analyse mode -> see -> User Guide -> Working with events.

Internal markers

Similar to the user created markers, Dewesoft itself have two markers:

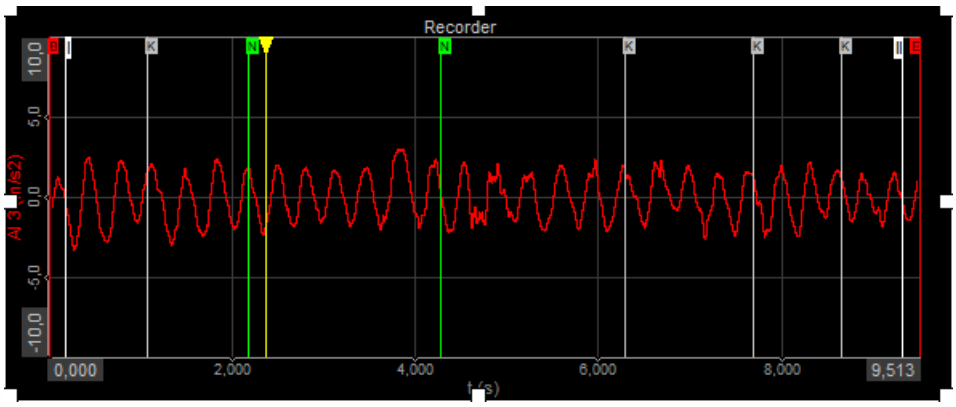
- the begin of recording marked with the " B " icon and a thin vertical red line (information about this is also automatically noted as event - see above).

B

- the end of recording marked with the " E " icon and a thin vertical red line (information about this is also automatically noted as event - see above).

E

Example of several marked events in on stored signal:

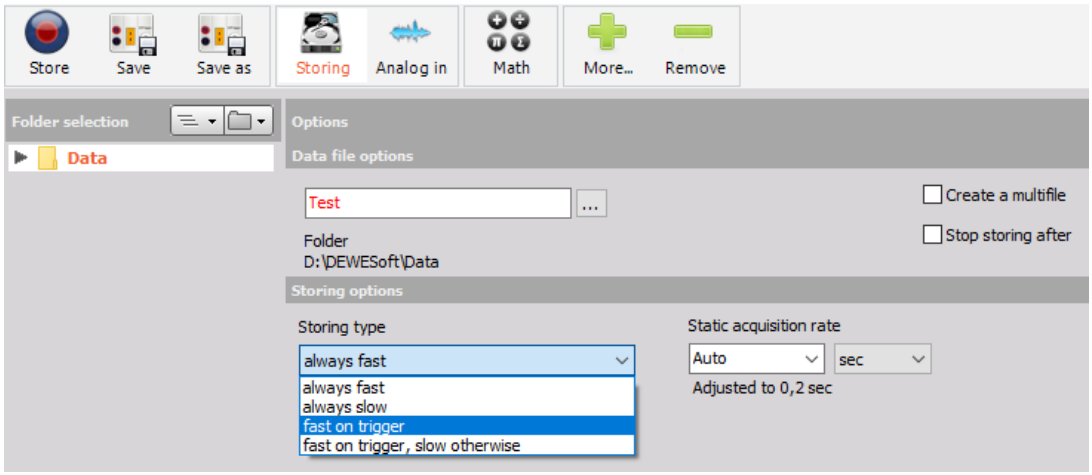


Triggered

You can trigger from your signals to start and stop recording.

You can select two different types of trigger storing from the storing options drop-down list on the Dewesoft Setup screen:

- fast on trigger
- fast on trigger, slow otherwise



When we select one from trigger selection, a tab labeled Trigger will appear and when we select it, lower part of screen will automatically open channel list for triggers set up. In trigger setup we define Start storing condition and we define pre-time and post time. Pre-time is the time that the data will be stored before the trigger occurred while the post time is the time after the event.

For detailed information about Triggers setup see -> User Guide -> Setup - Triggers.

After we have set our trigger criteria (manually or also achieve automatically) we can store data.

When we press Store button, this button changes the name to Arm button and we see additional Trig button, which tells us that we are using triggered storing.

We can also press this button to issue manual trigger to trigger even without an event.

If the post time is not defined, data will be stored until a manual Stop button is pressed or Stop storing condition occurs.

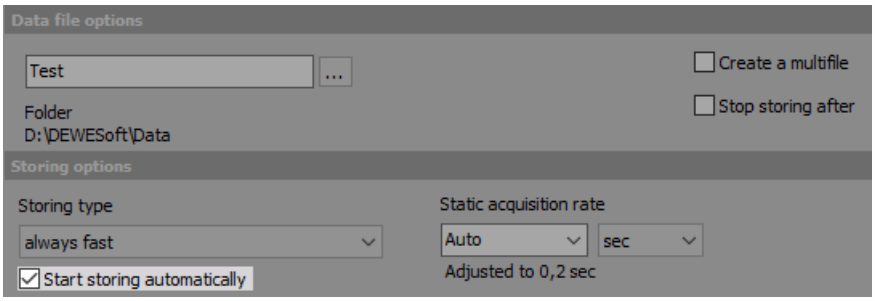
Automatically

If the Start storage automatically check-box in the main Dewesoft Setup screen is selected, the data storage will be automatically started as soon as you change to any online display (scope, recorder,...).

Just press the Stop button or F7 on the keyboard to stop the storage. This function is independent from the storing type (fast, reduced, triggered,...).

This function can be used together with the auto-load function of Dewesoft for highest automation.

For information about auto-start of the measurement see -> System Settings -> Recording setup.



NET ACQUISITION DATA

Dewesoft NET application module provides nice way to acquire data over the network. This allows to use multiple systems as one instrument or to acquire data from different locations.

Dewesoft NET application module allows one or more measurement units (named MU's) to be under the control of other computers, named CLIENTS. The MU(s) and CLIENT(S) must be connected via TCP/IP.

Work with Dewesoft-NET composes three basic procedures (steps):

- NET Setup Network configurations, appropriate hardware and Dewesoft-NET setup
- Measurement - Manage measurement units with NET menu option including Measure transfer speed; Creating a display, measure-acquire data and store this data on net
- Analyse Analyse acquired and stored data on net, export measured data

Modes of operation

Dewesoft NET offers three modes of operation. With these three modes, almost any application can be covered. From single channel expansion over remote control to distributed measurements over hundreds of kilometers - everything is possible.

1:1 mode - Single measurement unit and single client

1:1 mode works with single measurement system and a single client. In this mode, there are two types of operation: full remote control and data view only. In full remote control, the client computer acts like a master of the measurement system. When controlling client changes to the setup screen, the measurement system also changes the setup screen.

However, in the "view only" mode, the measurement system acquires data, while the client computer can connect to it and view the "live" data ... But it can't control the measuring system. A "view client" can look, but not touch.

x:1 mode - Multiple measurement units and single client

Multiple measurement systems and single client are used in the case of distributed measurements or too high acquisition rates to be managed by a single measurement unit. The measurement systems have to be synchronized either with hardware clock (one unit is the clock master, the others are slaves) or with the external clock source that is either IRIG or GPS.

All measurement systems have to run with the same acquisition rate, in this case, only one connection option is possible - the client is always the master. It starts and stops the measurement on all units in the measurement network. At any time, the client has access to view mode - but only to one measurement system (one-to-one connection like in single measurement system & single client configuration). Additional view devices are possible, but they can access

only a single measurement system.

1:x mode - Single measurement unit and multiple clients

The third network configuration is to have a single measurement system controlled by one 'master' client and additional 'view' clients. The master client is able to change the measurement system setup, storing strategy, start and stop measurements, and much more. The view clients are only allowed to take a few channels from the measurement unit (up to the bandwidth limitation) and view and store the data on their local hard disk.

Types of connections

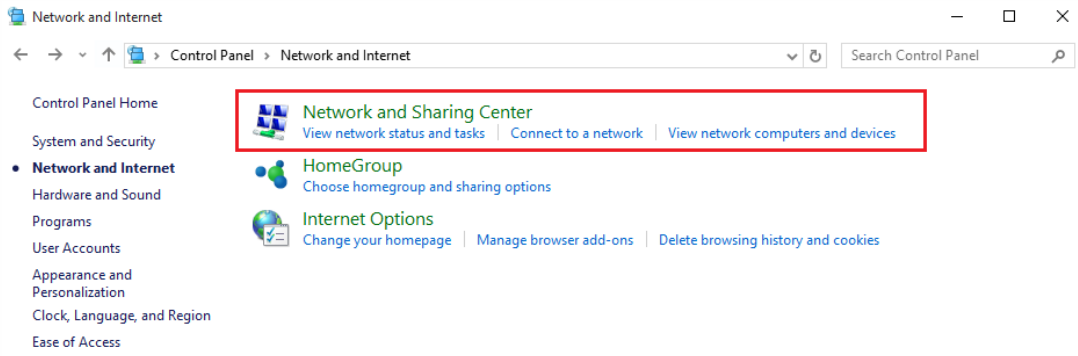
The NET system is always connected via Ethernet protocol. This can be obtained in several different ways:

- point to point connection (It is the easiest and most simple configuration of the NET system. On either side, you have a computer (S-box, Minitaur, PC, laptop, ...) with a measurement device (DEWE-43, Sirius, ...) or without it. Point to point connection means, that we have an Ethernet cable between those two devices. With this connection, you have to manually set the IP addresses on both sides.)
- through router
- through local network LAN (the limitation is the connection speed)
- through the internet (dedicated TCP/IP ports needs to be opened):
- wired
- wireless

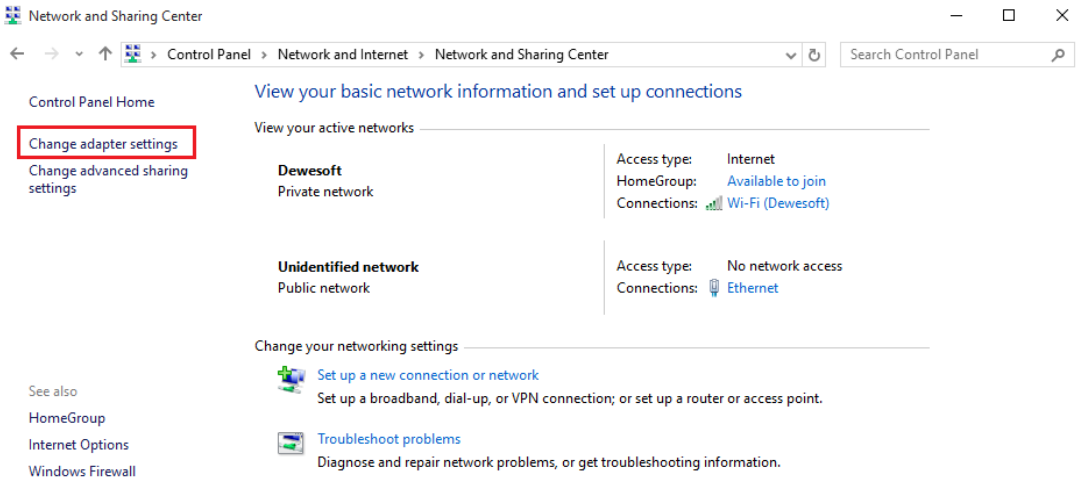
Point to point connection

With the point to point connection, we connect two devices with the Ethernet cable. We just have to manually set the IP addresses on both sides.

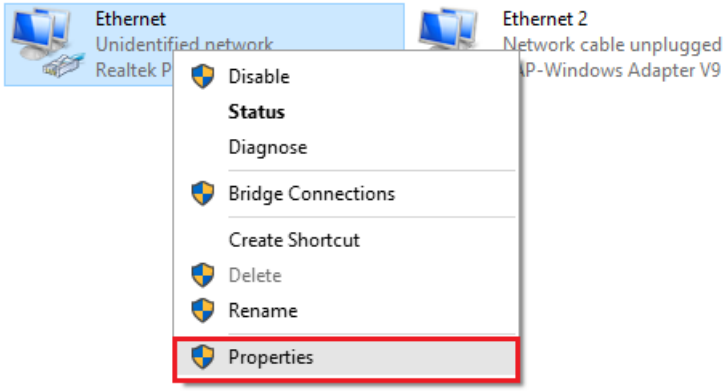
In order to do that, we have to go to Control panel - Network and Internet - Network and sharing center.



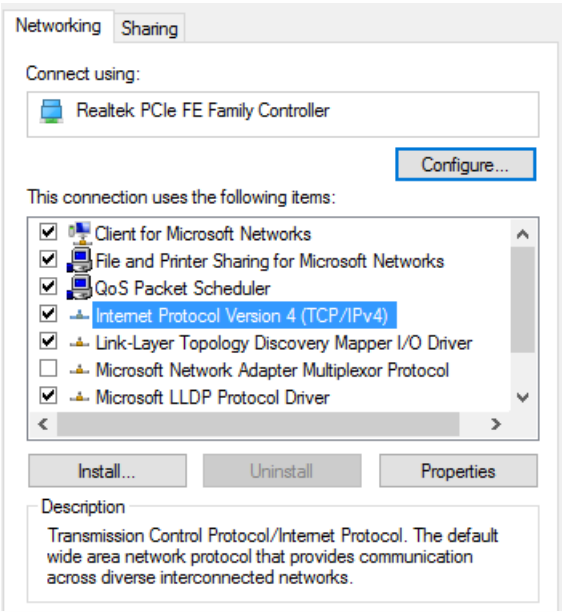
We need to select Change adapter settings.



Right click on Local connections and select the Properties

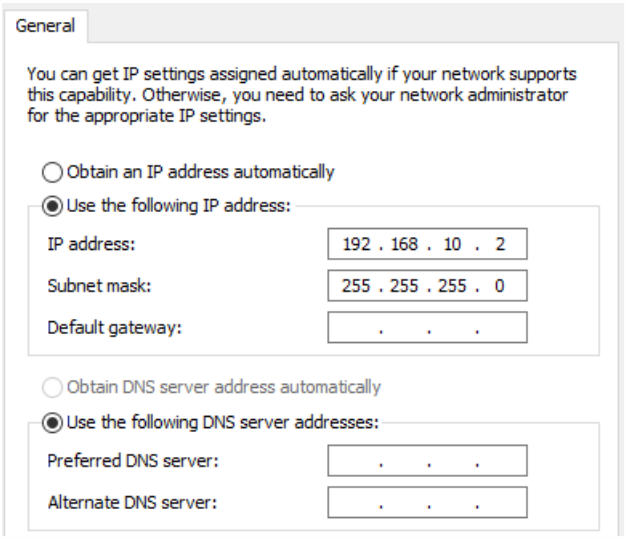


In Properties, we choose Internet protocol Version 4 (TCP/IPv4) and type in the IP address.

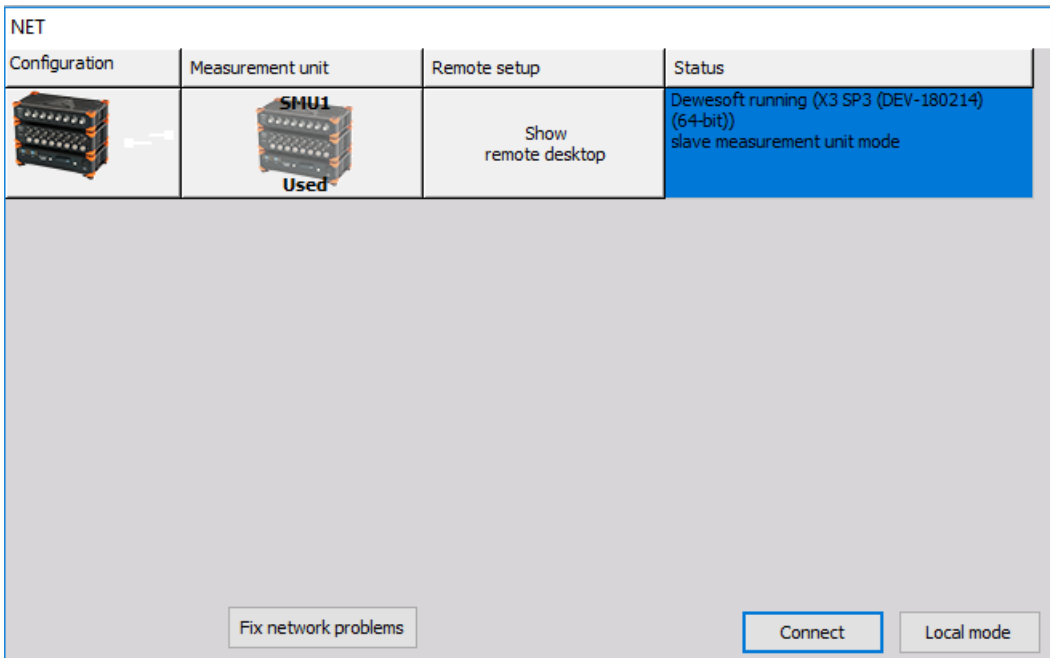


IP addresses on devices must be the same, except for the last number, which must be different (ex. IP address on the first device 192.168.10.1 and the IP address on the second device is

192.168.10.2)



Now we can connect with the device, measure and collect data from it.



Measurement unit configuration

There are several possible configurations and setups:

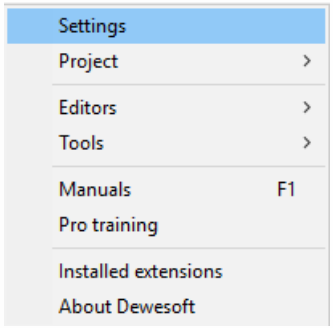
- Standalone unit – not networked to anything (the default setting of Dewesoft upon installation)
- Slave measuring unit – can measure data under either local control or under the control of a master client
- Master measuring unit – can both measure data and control other measurement units (optional)
- View client – can view data being recorded on the measurement units, but cannot control them
- Master client – can control the measurement unit(s) and view their data

Each measurement unit must be configured as a slave measuring unit in order to utilize the Dewesoft NET software – however, if you are going to use one of your measurement units as the controller for the others, then you should configure that one unit as a master measuring unit.

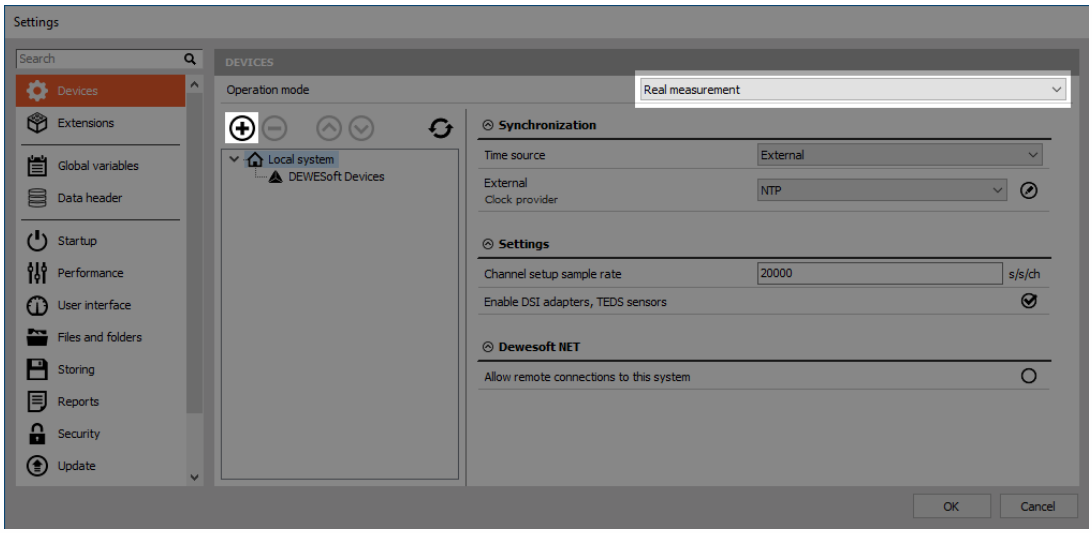
It is not possible to have more than one "master" within a single Dewesoft NET system, in order to avoid confusion and conflicts.

Setup

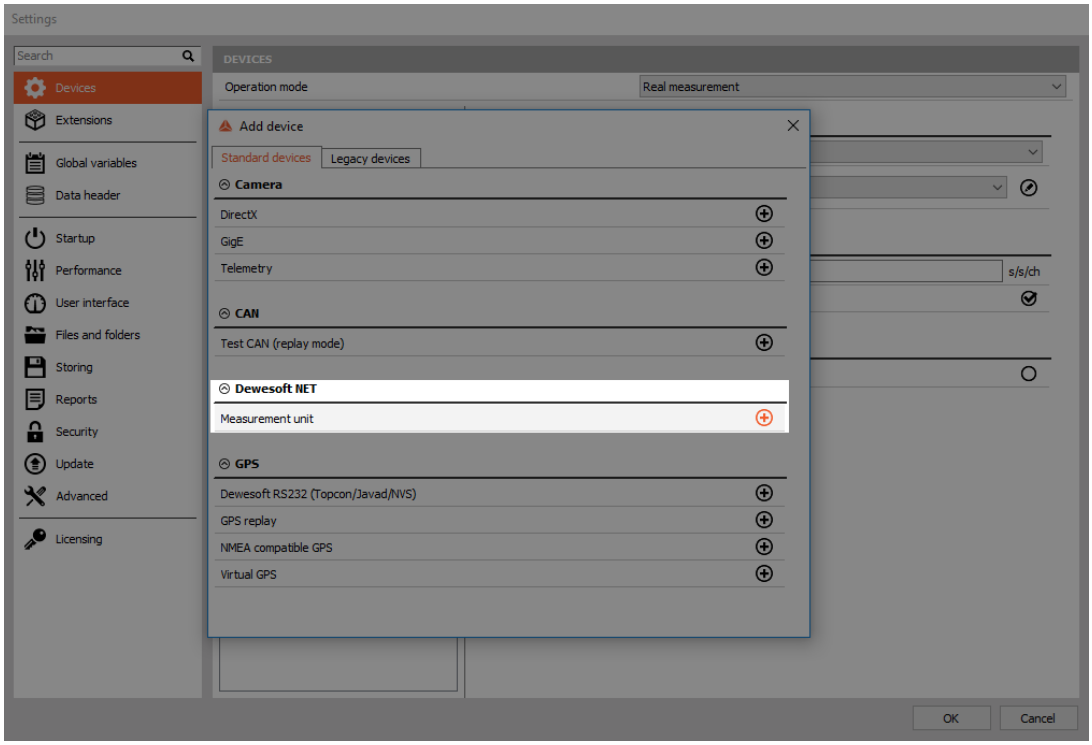
To activate the appropriate mode for each system within the Dewesoft NET, first run Dewesoft. Now click on the Settings menu.



Operation mode must be set to Real measurement. To add new devices click the plus button.

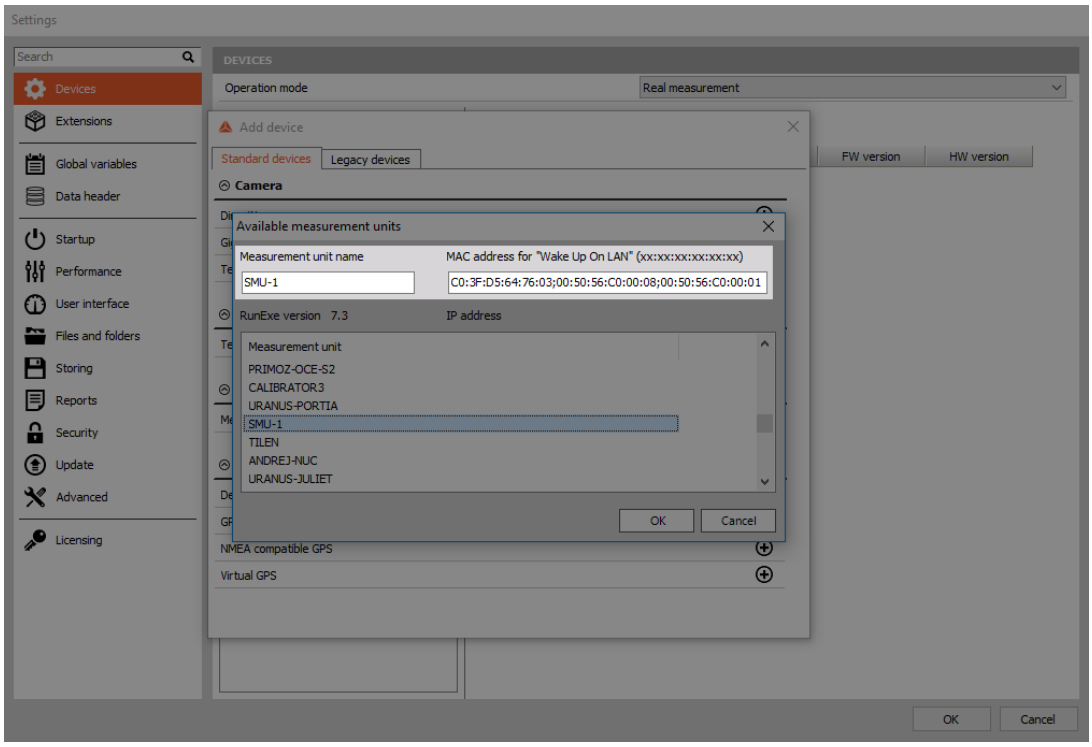


Now you can add new devices. Under Dewesoft NET select to add new Measurement unit.



You will see all available measurement units (names, MAC addresses, ...).

Dewesoft Launcher must run on measurement devices so that they can be seen as available units.

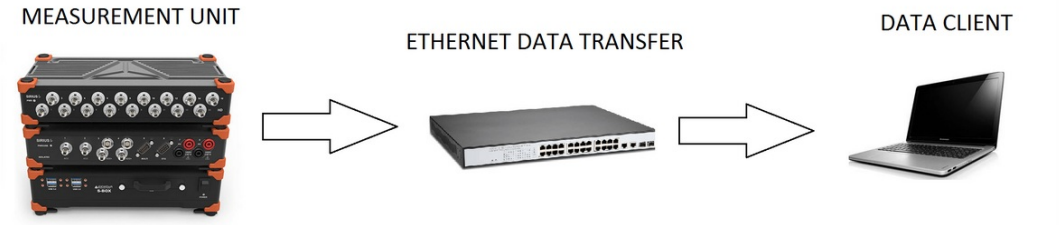

Settings
Search
Devices
Extensions
Global variables
Data header
Startup
Performance
User interface
Files and folders
Storing
Reports
Security
Update
Advanced
Licensing
DEVICES
Operation mode
Real measurement
Add device
Standard devices
Legacy devices
FW version
HW version
Camera
Available measurement units
Measurement unit name
MAC address for "Wake Up On LAN" (xx:xx:xx:xx:xx:xx)
SMU-1
C0:3F:D5:64:76:03;00:50:56:C0:00:08;00:50:56:C0:00:01
RunExe version 7.3
IP address
Measurement unit
PRIMOZ-OCE-S2
CALIBRATOR3
URANUS-PORTIA
SMU-1
TILEN
ANDREJ-NUC
URANUS-JULIET
OK
Cancel
NMEA compatible GPS
Virtual GPS
OK
Cancel

Principle of Dewesoft NET application module

It is important to note that by default actual data is stored on the measurement units, even though it can be viewed by the clients. This is critical to protect yourself against data loss which might occur by the network going down or transmission being interrupted. Even if this happens, the data is safely stored on the measurement units. When the network connection is reestablished, it is possible to reconnect automatically.

Even if the network going down or transmission was being interrupted, the data is safely stored on the measurement units. The idea of Dewesoft NET technology is to have a distributed system when:

- the required computing power is too high for a single measurement unit (many channels sampled with a high sample rate),
- there is too much distance between the units for analog data transfer,
- the measurement unit is not accessible (dangerous measurements, test rig measurements, ...),
- data from measurement units shall be displayed on several client computers,
- measurements have to be remotely controlled or supervised.



Within Dewesoft NET the master client completely controls the slave measurement unit - when the master unit switches to the setup screen, also the slave unit switches to the setup screen.

It is often not possible to transfer all possible channels in real time to the client for storage, even using a gigabit Ethernet interface.

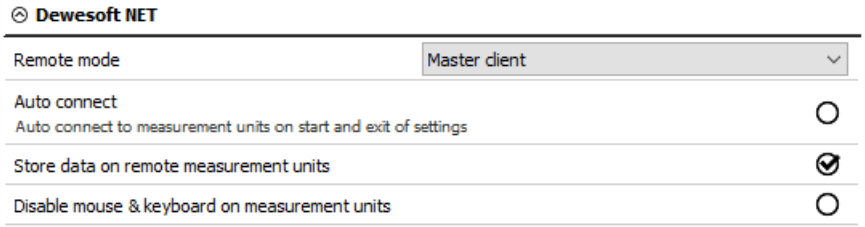
Imagine even one Dewesoft system with 32 channels, being sampled at 200 kB/s each at 24-bit mode. This is already 25.6 MB/sec, which is more than 200 Mb per second (where each Byte = 8 bits). It does not take long for the network to be completely overloaded with data and be overwhelmed with packet loss.

Immediately after the acquisition is stopped, a button appears on the controlling client allowing the data file(s) to be uploaded from the measurement units for viewing on the client computer.

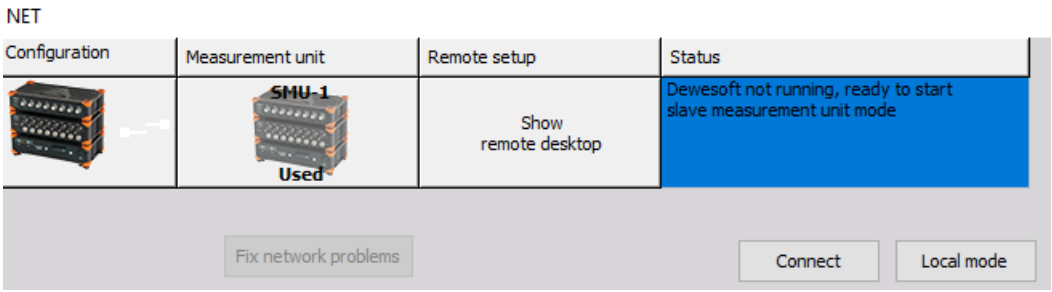
Setting up a master client

In the case where your Dewesoft systems are all slave measurement units, you will need one master client to control them. Let's assume that we are now sitting at this computer and have Dewesoft properly installed and ready. Click on the Settings menu.

Now use the selector to assign this computer to be the Master client, as shown below:

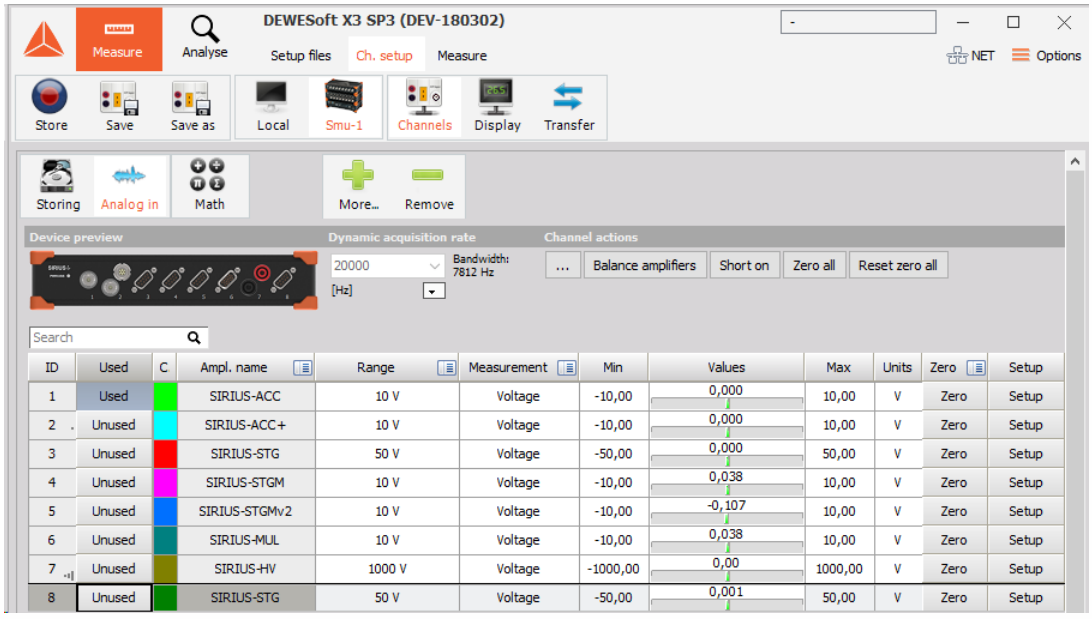


After confirming the Dewesoft NET setup, the following window will appear:



Select the Connect button to connect with the measurement unit.

After successfully connecting, the measurement unit will be seen like in the picture below.

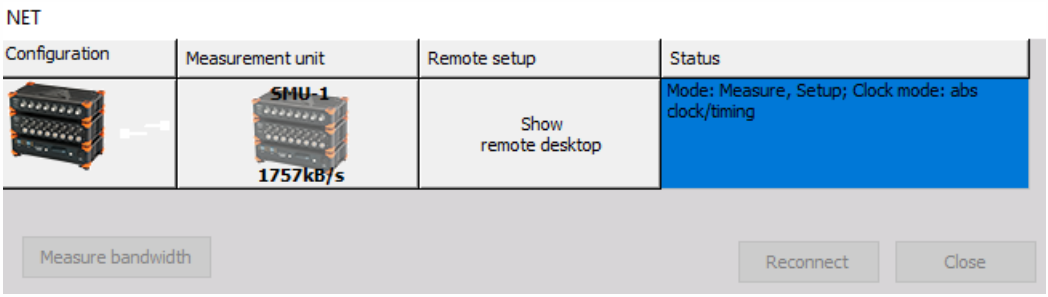


Note that the slave measurement unit called Smu-1 was already found in our example. But if you have not already configured and connected to a measurement unit, just click the Add button and

select one or more measurement units in order to add them to the system.

Now click OK to close this dialog and you will notice that the basic Dewesoft screen has a new addition to the top bar – a NET icon:

Click on NET icon to show the connection screen. Clicking the Measure bandwidth button in the system will check network performance. In a 100 Mbit network the transfer speed is about 10 MB/second while Gigabit LAN offers speeds close to 100 MB/second. Please note that the real bandwidth is also limited by system performance.



Measurement

In this section we can see the procedures to control the acquisition by the client:

- creating a display on the client
- storing data on the measurement unit
- transfer stored data to the client
- NET menu option

Creating a display on client

Before you begin storing, you may want to set up the local display. In previous steps, you may have configured the display of one or more measurement units, but you probably want to see data here, too!

You certainly know how to do this. Use the Overview, Scope, Recorder buttons at the top of your own screen here on the Client to create displays with any combination of channels from any and all measurement units!

As mentioned previously, all measurement units must have a SYNC method in place in order to ensure these three things:

- truly synchronized data files from multiple measurement units
- ability to display channels from more than one measurement unit on the client
- ability to create math channels on the client with channels from more than one measurement unit

Note that the CHANNELS list is now showing channels with the "name" of the MU that they come from automatically. This is so that you know the source of every channel in an easy and convenient way.

In our example, the name of the MU, "TESTING-PC" is added in front of the transfer channels from that MU by default.

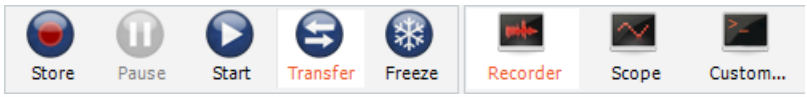
The channels from each measurement unit will be shown this way automatically. This is the only thing that differs from setting up a screen in the standalone mode of Dewesoft.

Storing data on the measurement unit

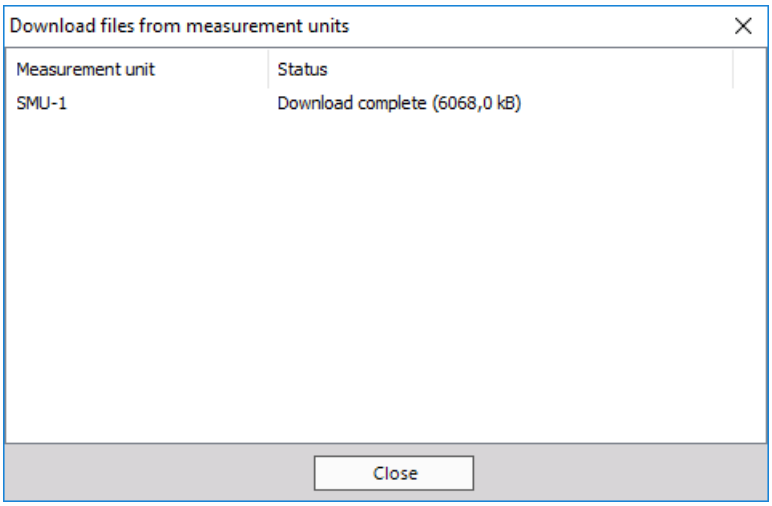
With the client and measurement unit properly configured, we can now store data. Just click Store in the toolbar in the normal way.

Transfer stored data to the client

As soon as the storing is stopped, an important button appears automatically, called Transfer:



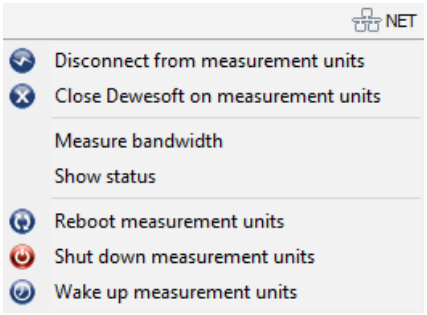
Please click it, and the data file(s) from all measurement units that we just used will be downloaded to the client for you. A "transfer box" appears to show the progress and eventual completion of the download:



In this case we only had one measurement unit, so only one file needed to be downloaded.

NET menu options

Click the NET menu option to see the list of options:

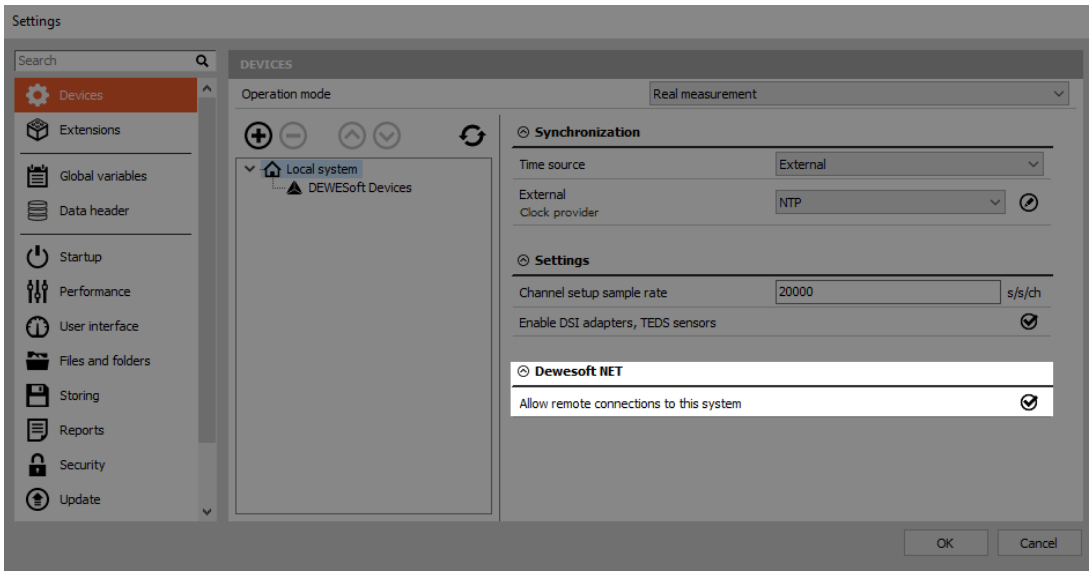


Notice from the menu that you have several useful capabilities:

- Connect/Disconnect from measurement units - connect to all measurement units/releases the connection
- Close Dewesoft on measurement units - closes the Dewesoft application on all measurement units
- Measure transfer speed - measuring the bandwidth (transfer speed) between the measurement units and this client
- Show status - (displays window with the current status of all measurement units)
- Reboot measurement units - reboots the measurement unit computers (useful if they have crashed or hung up)
- Shut down measurement units - shuts down the measurement units (requires ACPI power system on the measurement units)
- Wake up measurement units - starts measurement units (requires ‘Wake-up on LAN’ option enabled on the measurement units)

Setting up a slave client

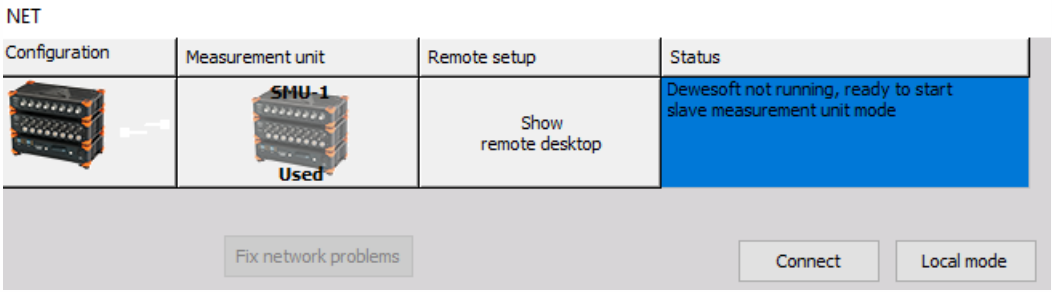
If you want your unit to be set as a slave client, you must first enable remote connections.



Note that a graphic named Configuration appears which indicates the topology type in which the measurement unit is represented by a picture of a data acquisition system (the picture for reference – it is not meant to look exactly like your particular model).

The white lines represent the TCP/IP network, which must be connected among all MU's and the client (represented here as a notebook computer, again only as a reference).

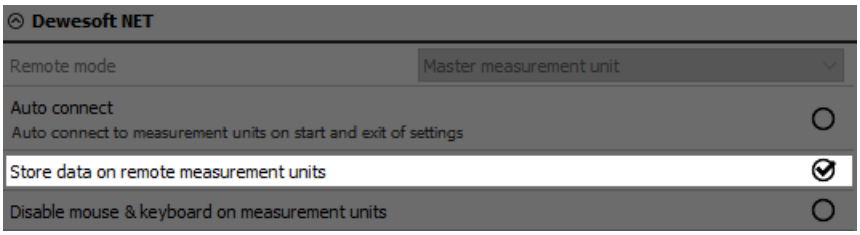
The red lines represent the SYNC which must be present if there is more than one measurement unit and you want to show channels from more than one measurement unit on the client at the same time. It also ensures that the data files recorded on each measurement unit are totally synchronized. Unless you have a hard sync, it is not possible to create totally synchronized data files.



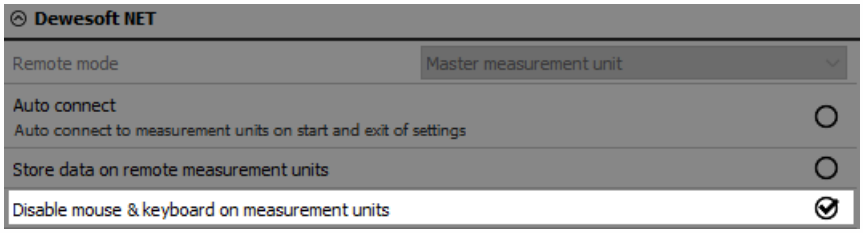
In the example above, the system sees one measurement unit on the network, called SMU-1. Select it and click OK at the bottom of the dialog box.

Further setup can be done in Settings.

- Store data on measurement units (checked by default, and highly recommended!)



- Lock mouse and keyboard on measurement units

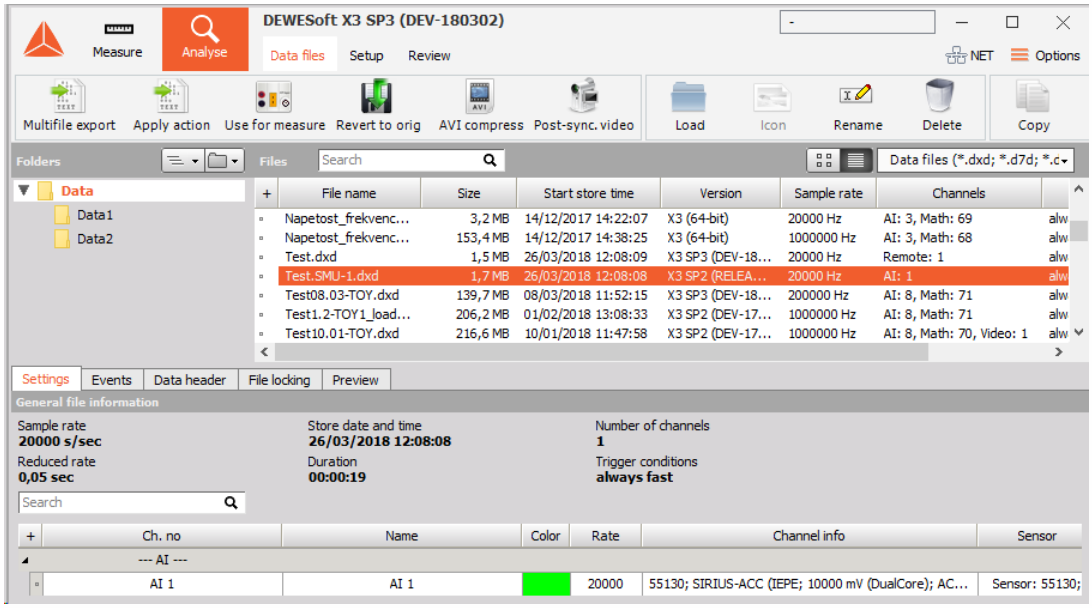


Use the "lock mouse and keyboard" check-box if you want to prevent a local operator from changing any settings on the measurement unit or interfering with the test. With this checked, the measurement unit cannot be operated locally – you will have complete control over the client.

Analysis

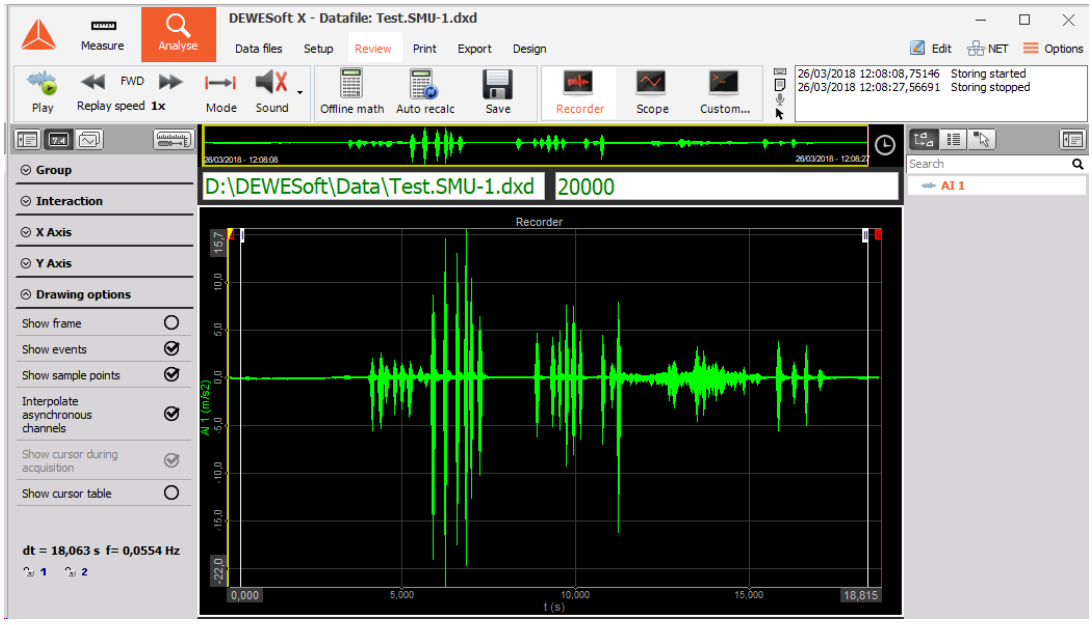
Once the captured data files are downloaded to the client, you can replay them there.

Click the analysis button and locate any transferred files that you have. Notice that we also put the name of the measurement unit into the filename by default, so that you can see that this file came from a measurement unit called SMU-1.



The filename was set to Test. So the name shown here is Test.SMU-1.

Double-click it to open and use the normal tools for analyzing, reviewing, printing and more.



Transfer limitations

USB bandwidth limitation

One reason why we use NET configurations is that USB bandwidth is limited. USB port has a lower bandwidth limitation than an Ethernet port.

USB 2.0 limitation:

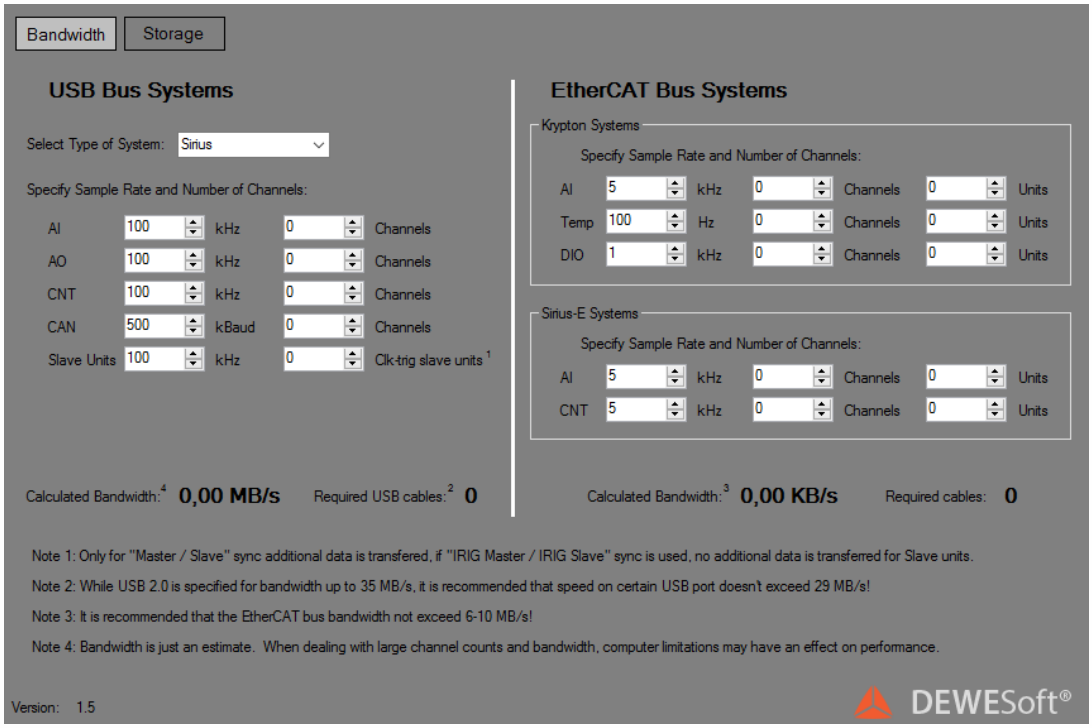
- 35 MB/s (in practice ~ 30 MB/s)
- Dewesoft USB limit notice at 26 MB/s
- USB 3.0 does not help upstream

General formula for calculation AI and AO channels:

- Number of channels * Sample rate * 4 bytes/sample (/2 if we use HighSpeed)
- CNT channels: Number of channels * Sample rate * 8 bytes/sample
- CAN channels: Number of CAN ports * baudrate/8
- Slave units (only for clock/trigger sync): Sample rate * 8 bytes/sample

It is very important how the USB port is internally wired in the computer - 30 MB/s per single root hub.

Dewesoft calculator is a simple tool that calculates USB transfer (MB/s) for various Dewesoft devices. You can download it here or by clicking on the picture below.



NET transfer limitations

The limitations of NET transfer are dependent on:

- the speed of Ethernet link in case of large bandwidth
- the write speed of hard drive in case of large data storage
- CPU performance in case of advanced math

We are limited with data transfer because all the PCs are connected together with 1Gb LAN cable.

Next table shows upper limits of transferred data (in samples/second) per 1 PC.

Each PC can store approximately 25.600.000 samples per second and that number of samples can be distributed randomly between measurement units.

With full this sample rate, PC stores approximately 100 MB/s of data:

- 10s for 1 GB
- 1min 40s for 10 GB
- 16min 40 for 100 GB
- 2h 46min 40s for 1 TB

Synchronization

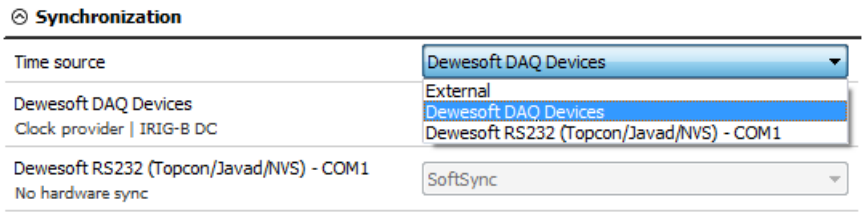
The basic idea of synchronization is to provide a clock signal from a time source. Clock slave receives the signal from the clock provider and the devices are synchronized.

Devices can be synchronized in two different ways:

- Software synchronization - The software synchronization accuracy is around 2-10 ms, which is enough for a simple temperature measurement. This synchronization solution requires no additional hardware.
- Hardware synchronization - This is a hardware solution which can synchronize all USB devices (SIRIUS, DEWE-43, ...) and EtherCAT devices (KRYPTON).

In the example below we have connected one Sirius, one DEWE-43 and one Krypton measurement unit to the S-BOX.

Time source



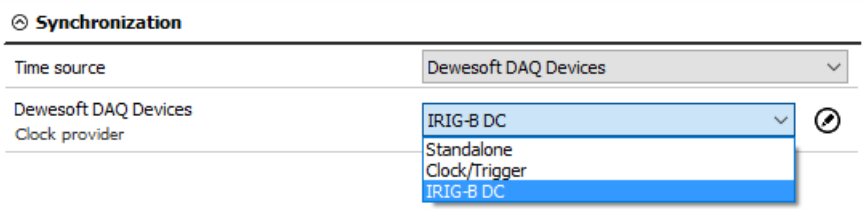
Time source provides a clock for synchronization. It can be selected from:

- Dewesoft DAQ devices (when we have a Dewesoft measurement device connected to our computer),
- External (Clock/Trigger, IRIG-B DC, NTP, GPS PPS),
- PC clock (when we have only a computer, without a measurement device),
- GPS devices (Dewesoft RS232 (Topcon/Javad/NVS), NMEA compatible GPS, ...).

Synchronization types are dependent and automatically adjusted from the DAQ devices connected to our system.

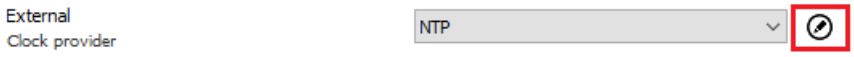
Type of synchronization

If Dewesoft DAQ device is selected as a time source, clock provider must be selected from:

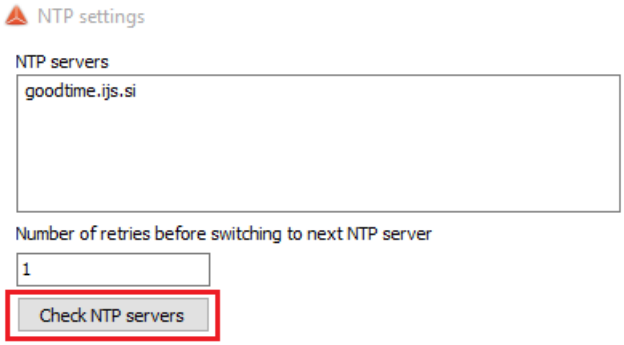


- Automatic - this option automatically selects the best option for synchronization regarding the hardware connected to the system.

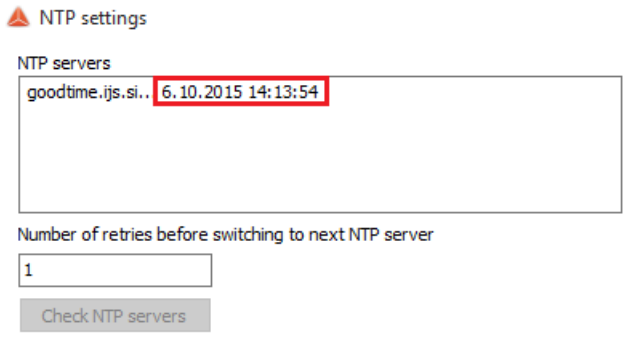
- Standalone - only one device, there is no synchronization between devices needed.
- SoftSync - this synchronization solution requires no additional hardware. The accuracy is > 10 ms.
- Clock/Trigger - a clock and a trigger signal are used. With each trigger signal a sample is acquired.
- IRIG-B DC - It contains time-of-year and year information in a BCD format (it contains the information about the absolute time). This is the best way to synchronize the devices because it is the most exact one.
- GPS PPS - Since the satellites are transmitting exact absolute time and better receivers usually output this pulse with a high precision (below one microsecond), we can use this technology to synchronize remote systems - and there is no distance limit.
- NTP - Network Time Protocol (NTP) is a networking protocol for clock synchronization between computer systems. It is less accurate than other methods.



To enter the addresses of the NTP server click the edit button near the drop-down menu. If you want to have more devices synchronized via NTP protocol, the same server address must be entered in all of them.



When you check NTP servers, time and date of the server will be displayed. If the server address is wrong, the check will fail.



Synchronization between Dewesoft USB devices

	Accuracy	When to use	Device
Clock/trigger	~ 1 us	stationary	Dewesoft, RoaDyn…
IRIG-B DC	~ 1 us	stationary	Dewesoft, Meinberg,…

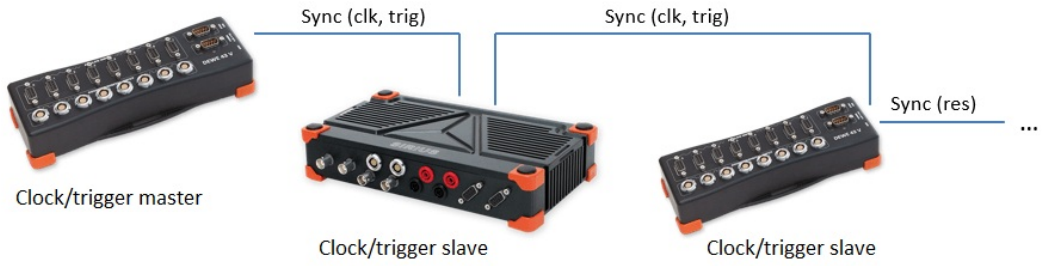
	Accuracy	When to use	Device
GPS PPS	~ 1 us	mobile	GPS receiver
NTP	+ 10 ms	Ethernet	NTP server
NTP	+ 10 ms	when there is no external time source	

Any Dewesoft device can be precisely synchronized by hardware (Sirius, Dewe-43, Minitaur, DS-CAN2).

When using the NET system, there are several possibilities that can be used for synchronization:

- Clock/trigger - relative time
- IRIG-B DC Master / IRIG-B DC Slave - absolute time
- GPS - absolute time
- NTP - absolute time

Clock/trigger connection example



IRIG-B DC connection example of one unit:

- Dewesoft device as IRIG-B DC generator

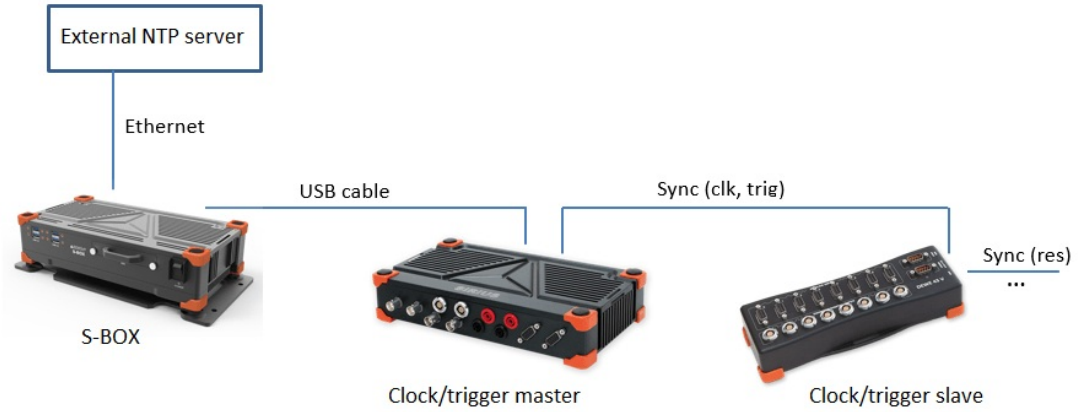
- External IRIG-B DC provider

GPS PPS connection example of one unit:

- Dewesoft device has a GPS receiver

- External GPS receiver

NTP connection example of one unit:

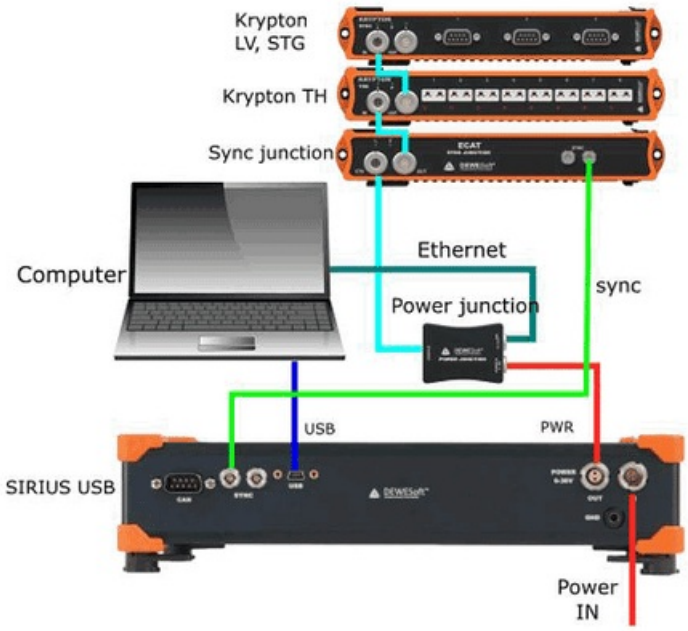


Synchronization with ECAT-SYNC-JUNCTION

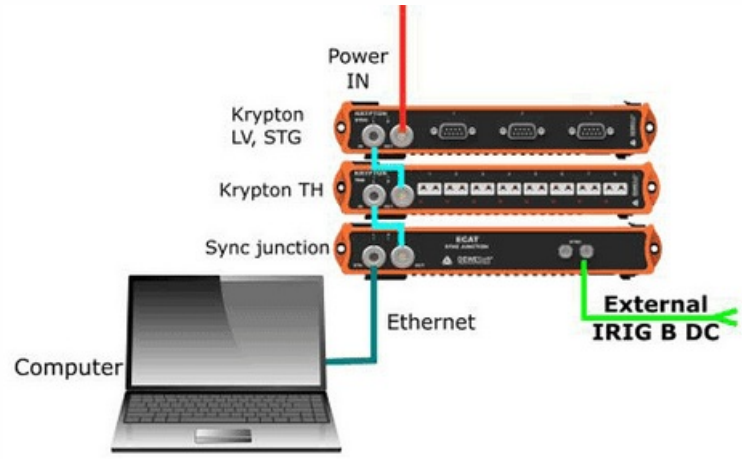
ECAT-SYNC-JUNCTION works in the same way as other Dewesoft devices. It is automatically recognized within Dewesoft X software (supported in version X2 SP4). By default, ECAT-SYNC-JUNCTION will be set up to synchronize between KRYPTON EtherCAT® and SIRIUS USB.

With ECAT-SYNC-JUNCTION several connection options are possible:

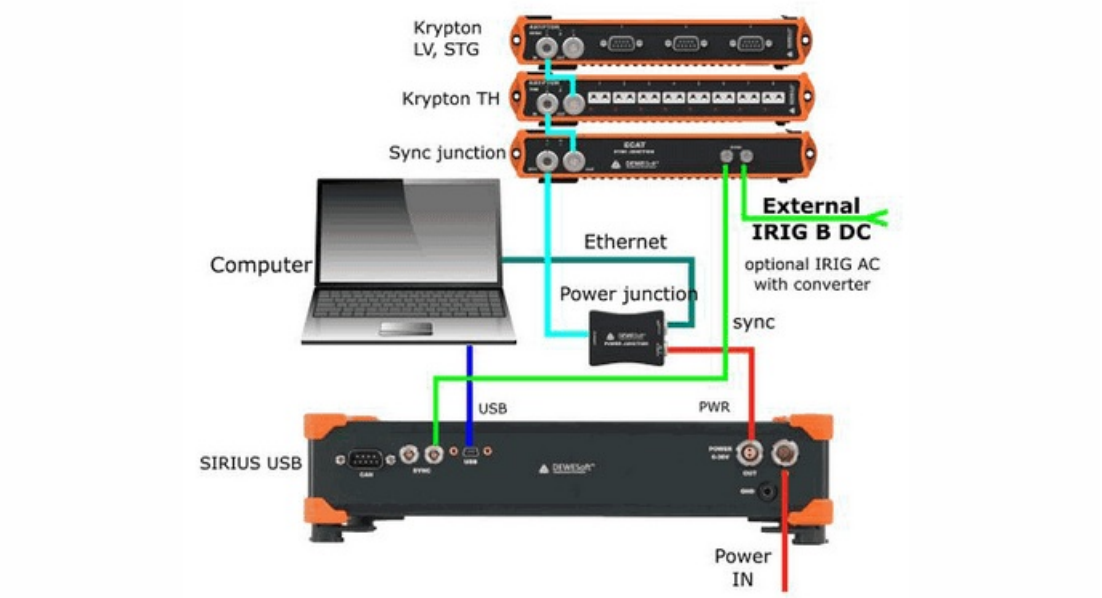
Synchronization of SIRIUS/DEWE-43 USB with KRYPTON/SIRIUSiwe EtherCAT® devices, where the accuracy of synchronization is microsecond



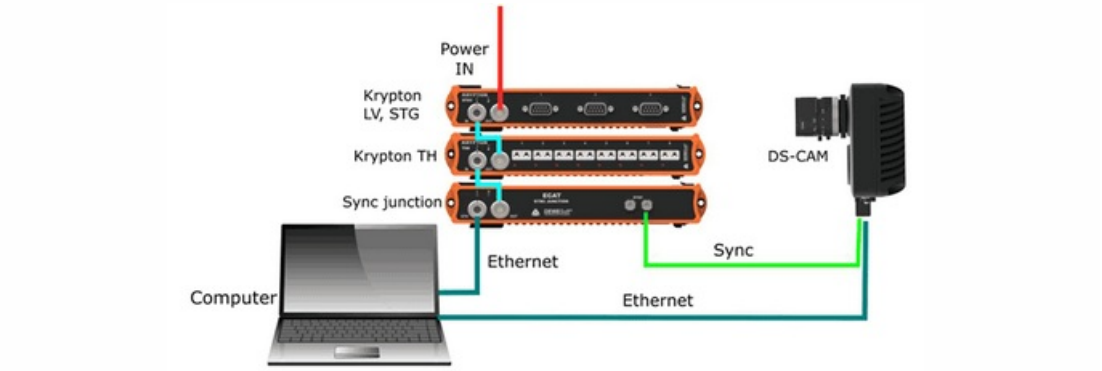
Synchronization of KRYPTON module with an external IRIG B DC triggering source



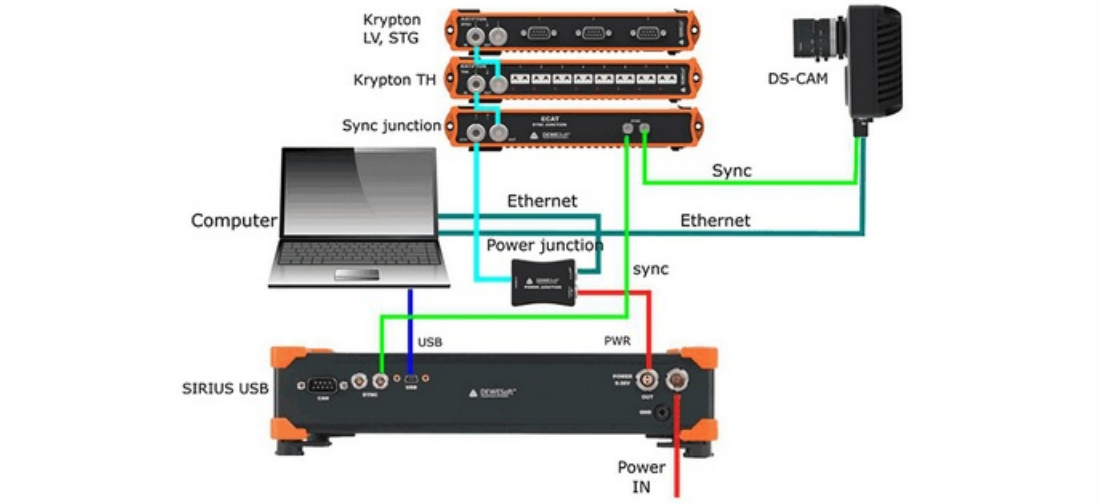
Synchronization of KRYPTON and SIRIUS USB with an external IRIG B DC triggering source



Synchronization of KRYPTON/SIRIUSiwe with triggered cameras



Synchronization of SIRIUS/DEWE-43 USB with KRYPTON/SIRIUSiwe and triggered cameras



NOTE: **Dewesoft EtherCAT® devices (KRYPTON) are already synchronized between**

each other by the daisy chain cable, therefore no additional cables are necessary between them.

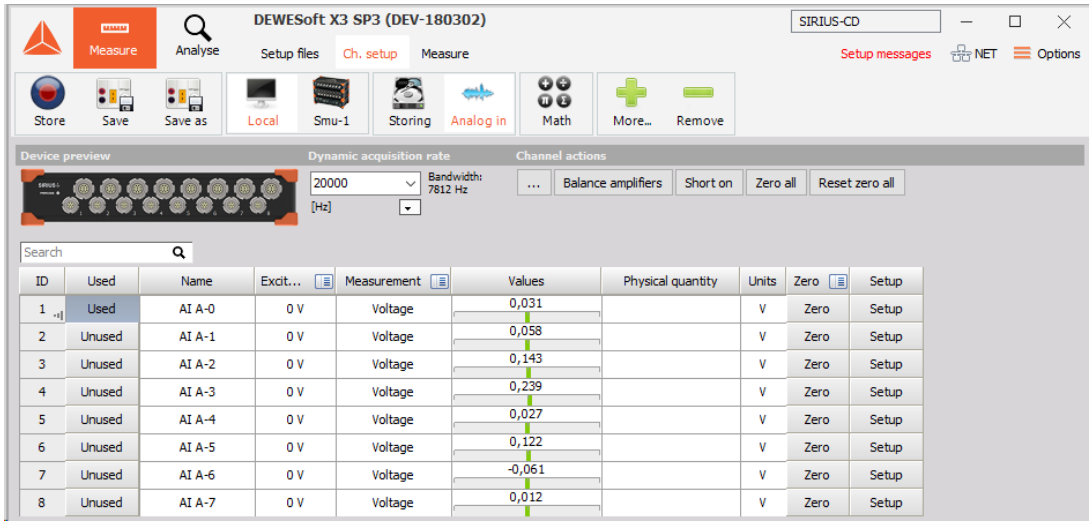
Remotely controlling a measuring unit

In this section, we are using only the master client computer to remotely configure and control a measurement unit from this master client. The measurement unit is already connected to it using the steps from the preceding section. All steps are done on the client.

We are not touching the measurement unit at all. It could be a few feet away, on the other side of the building, or miles away. As long as it has a reliable network connection to the client, we can control it from this client!

Local setup

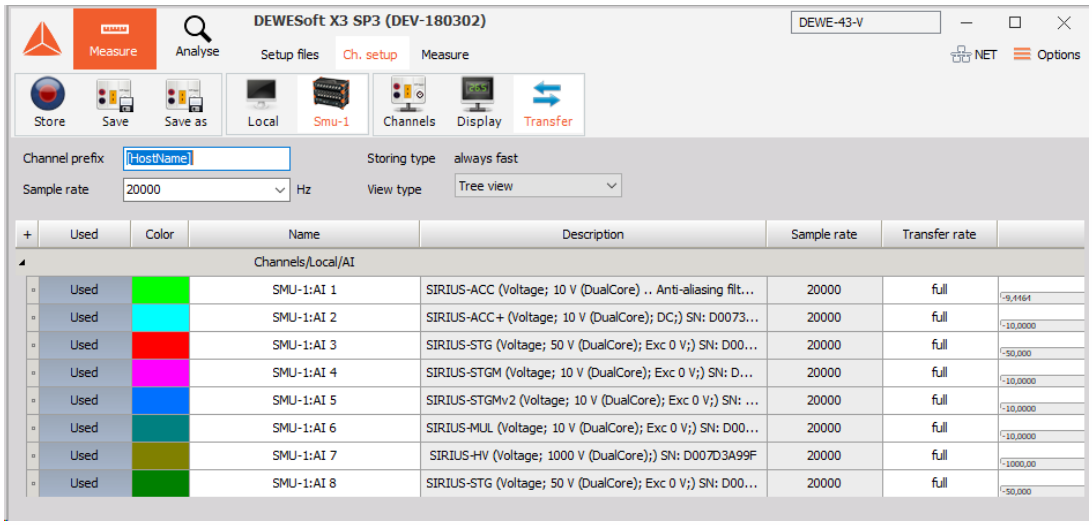
Now click the Ch. setup button. If you know what the Dewesoft Setup screen normally looks like, you will notice a subtle but important difference:



Note the buttons for Local and SMU-1. If you have more than one measurement unit, their names will be shown on this bar as well. The local computer is our master client, so it does not have any real measurement channels of its own. However, it still has Math buttons. It is interesting to note that you can perform math functions in real time on this client using any channels that are transferred from the measurement units! You can even combine channels from more than one measurement unit here in math channels – as long as the measurement units are synchronized!

SMU-1 setup

Click on the SMU-1 button so that we can proceed to set this measurement unit up.



You can see that SMU-1 has all eight channels set up to be transferred in real time to the client. Of course, they will also be stored on the local measurement unit, because this has been selected by default on the hardware setup screen, NET page.

Note the three buttons on NET toolbar beside selected WEBSTER:

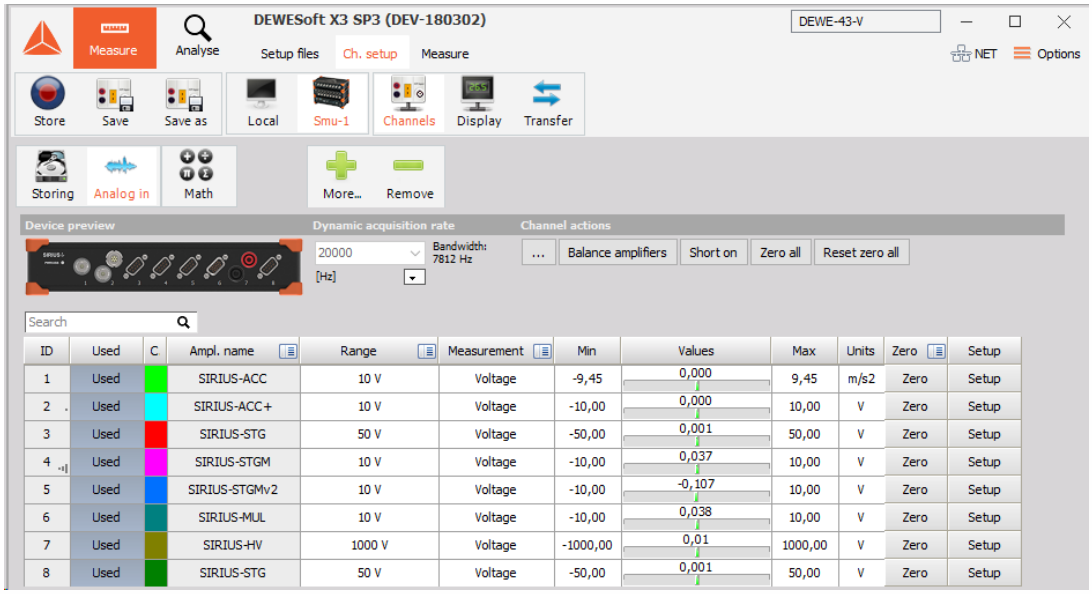
Channels - setup, activation, and scaling of the actual channels that will be stored

Display - to have a display screen on the measurement unit for local observers

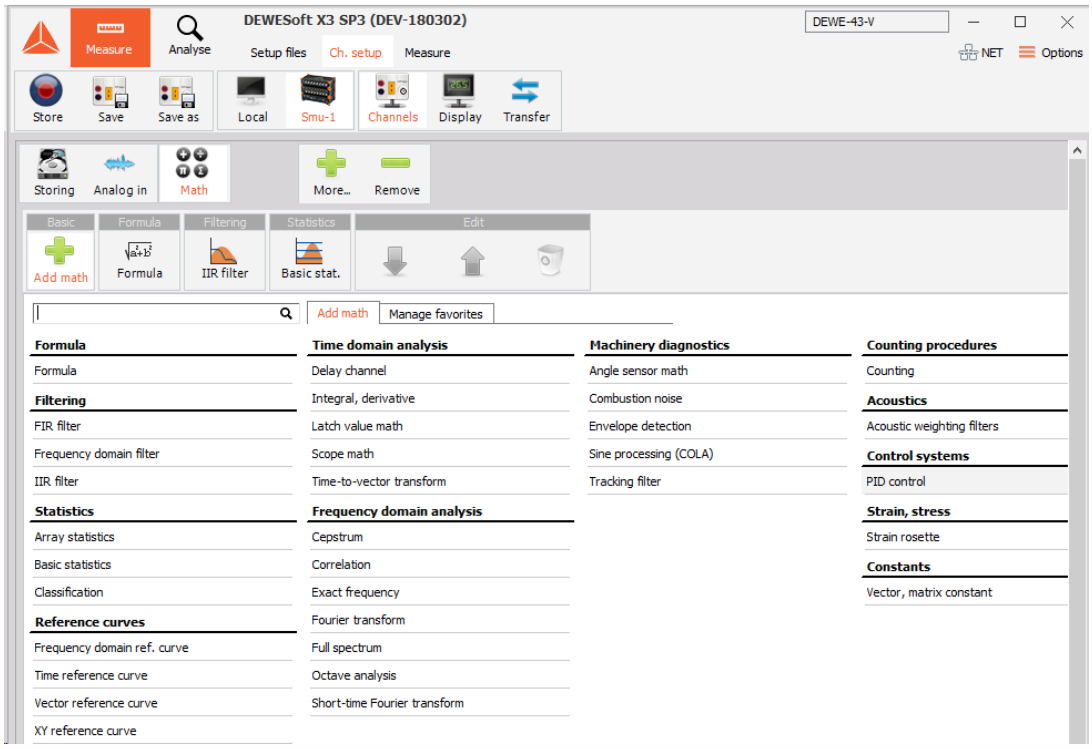
Transfer - to set up the transfer from the measurement

Chanel setup on the measurement unit

This is perhaps the most critical step of the three because it is the setup, activation, and scaling of the actual channels that will be stored on SMU-1. So let's start there. Click the Channels button on NET toolbar beside selected SMU-1.



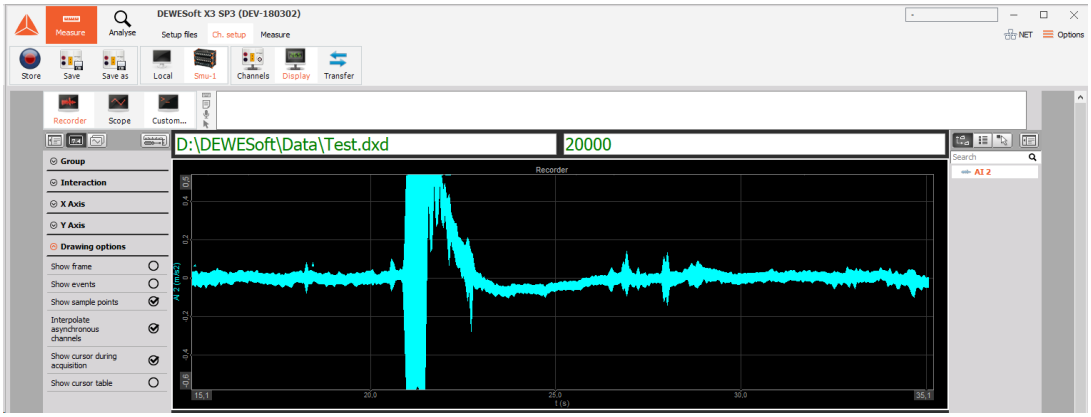
On this screen you are doing exactly what you would be doing if SMU-1 was a stand-alone measurement unit activating channels with the Used / Unused buttons, scaling them using the Set ch. # buttons, and so on. You can also set the dynamic and reduced sample rates, choose a filename, and more. In this example, SMU-1 is a computer with 8 analog input channels. In addition, the Math channels can also be created:


Measure
Analyse
DEWESoft X3 SP3 (DEV-180302)
Setup files
Ch. setup
Measure
DEWE-43-V
NET
Options
Store
Save
Save as
Local
Smu-1
Channels
Display
Transfer
Storing
Analog in
Math
More...
Remove
Basic
Formula
Filtering
Statistics
Edit
Add math
Formula
IIR filter
Basic stat.
Add math
Manage favorites
Formula
Formula
Filtering
FIR filter
Frequency domain filter
IIR filter
Statistics
Array statistics
Basic statistics
Classification
Reference curves
Frequency domain ref. curve
Time reference curve
Vector reference curve
XY reference curve
Time domain analysis
Delay channel
Integral, derivative
Latch value math
Scope math
Time-to-vector transform
Frequency domain analysis
Cepstrum
Correlation
Exact frequency
Fourier transform
Full spectrum
Octave analysis
Short-time Fourier transform
Machinery diagnostics
Angle sensor math
Combustion noise
Envelope detection
Sine processing (COLA)
Tracking filter
Counting procedures
Counting
Acoustics
Acoustic weighting filters
Control systems
PID control
Strain, stress
Strain rosette
Constants
Vector, matrix constant

Display setup on the measurement unit

This is only important if you want to have a display screen on the measurement unit for local observers to see. If there is no one looking at the local display on the measurement unit (perhaps it is in a remote location without any people near it), then you can skip this step.

But if you want a local display on the measurement unit, click the Display button on NET toolbar beside selected Testing-PC and then set up the screen as you desire, using the normal Dewesoft methods and conventions for screen design.



It is really important that the client computer has a display which has more resolution than the measurement units! If your measurement units have 1024x768 screens, your client should have the next size up or greater, else you may run into trouble seeing some of the screen objects near the bottom when remotely controlling measurement units from the client.

The display above is the one that will appear on the screen of the remote measurement unit called TESTING-PC! It is not the display that you will see here on the client.

Transfer setup of a measurement unit

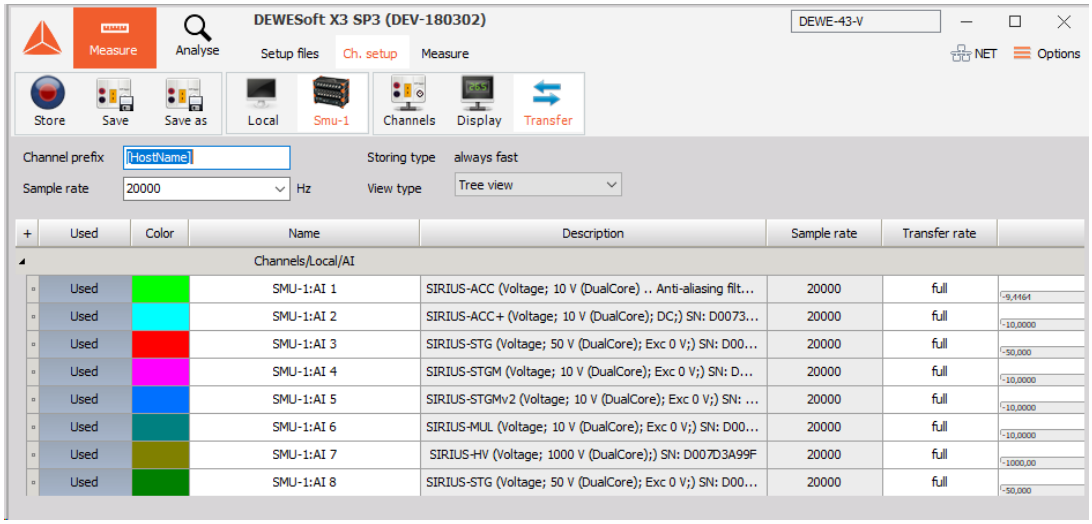
Finally, we will set up the transfer from the measurement unit. What does this mean? Transfer: "which channels will be sent across the network during recording, for display on the client".

That is the entire scope of what transfer means. It has no effect on the actual Storage of data on the client (assuming that local storage is enabled – the default and highly recommended setting).

Therefore, you can have multiple measurement units, each with dozens or even hundreds of channels, and transfer only a few channels – or even no channels – to the client. It will have no effect on the storage of all channels on the measurement units! This is important to understand.

Due to bandwidth limitations of any network, we recommend being prudent about transferring channels – keep the bandwidth in mind and select only those channels that you really need to see on the client in order to monitor and control the test.

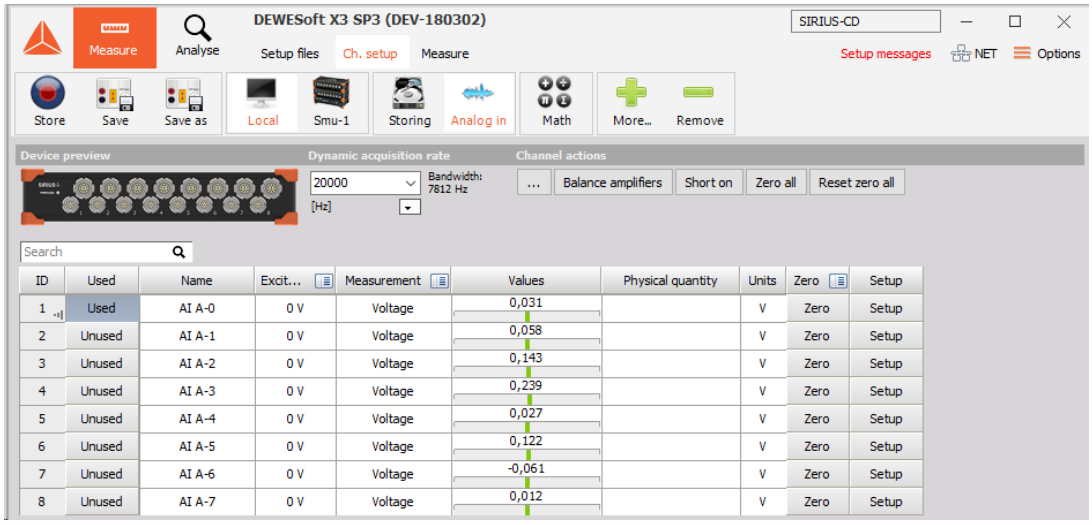
In this case, we will transfer all seven channels. Note that all four channels are set as Yes in the Transfer setup screen:



Local Net setup

When the connection was successful, the NET button turns green and the Setup screen appears. Now click the Setup button. If you know what the Dewesoft Setup screen normally looks like, you will notice a small but important difference.

Local setup screen with buttons to the remote setup screens:

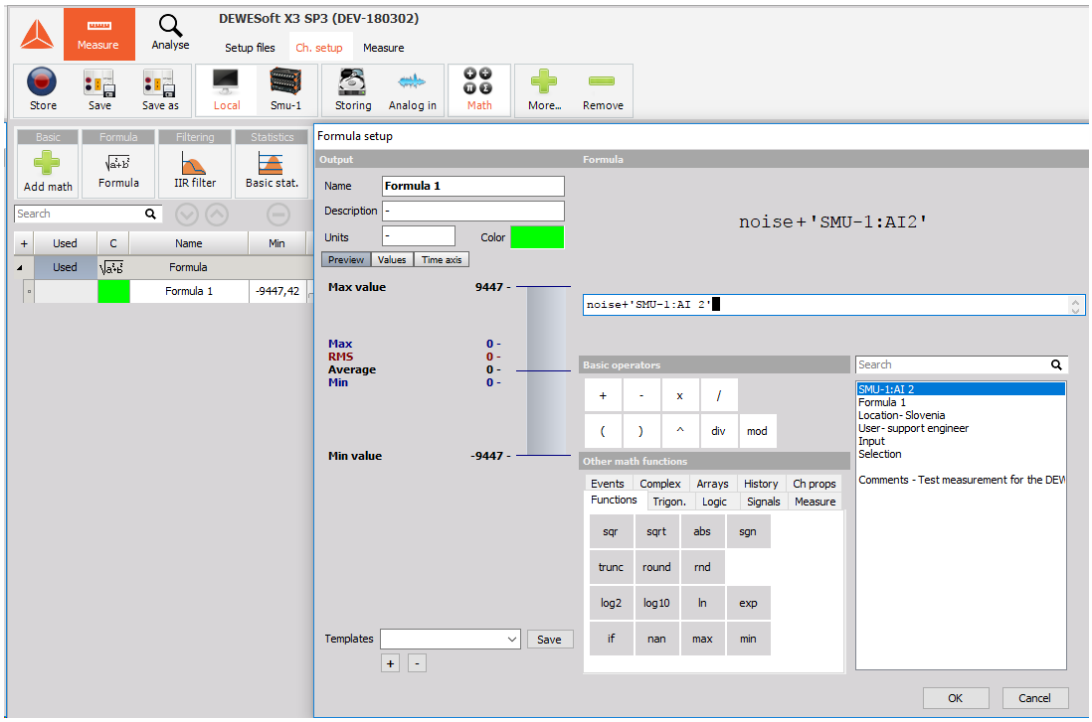


Each system setup (local and remote units) can now be accessed via the button bar (yellow indicates the selected unit).

Note the buttons for Local and Testing-PC. If you have more than one measurement unit, their names will be shown on this bar as well. The Local computer is our master client, so it does not have any channels of its own

Math tab

However, the Local computer still has a Math tab. It is interesting to note that you can perform math functions in real time on this client using any channels that are transferred from the measurement units! You can even combine channels from more than one measurement unit here in math channels – as long as the measurement units are synchronized!



Note the buttons for Local and Testrig. If you have more than one measurement unit, their names will be shown on this bar as well. The local computer is our master client, so it does not have any real measurement channels of its own.

Example of large NET system configuration

An example of a large system connected with the NET system.

Channels:

- Analog, 19 Sirius slices
- XSENS
- GPS
- ARINC
- CPAD
- Power module
- GigE cams

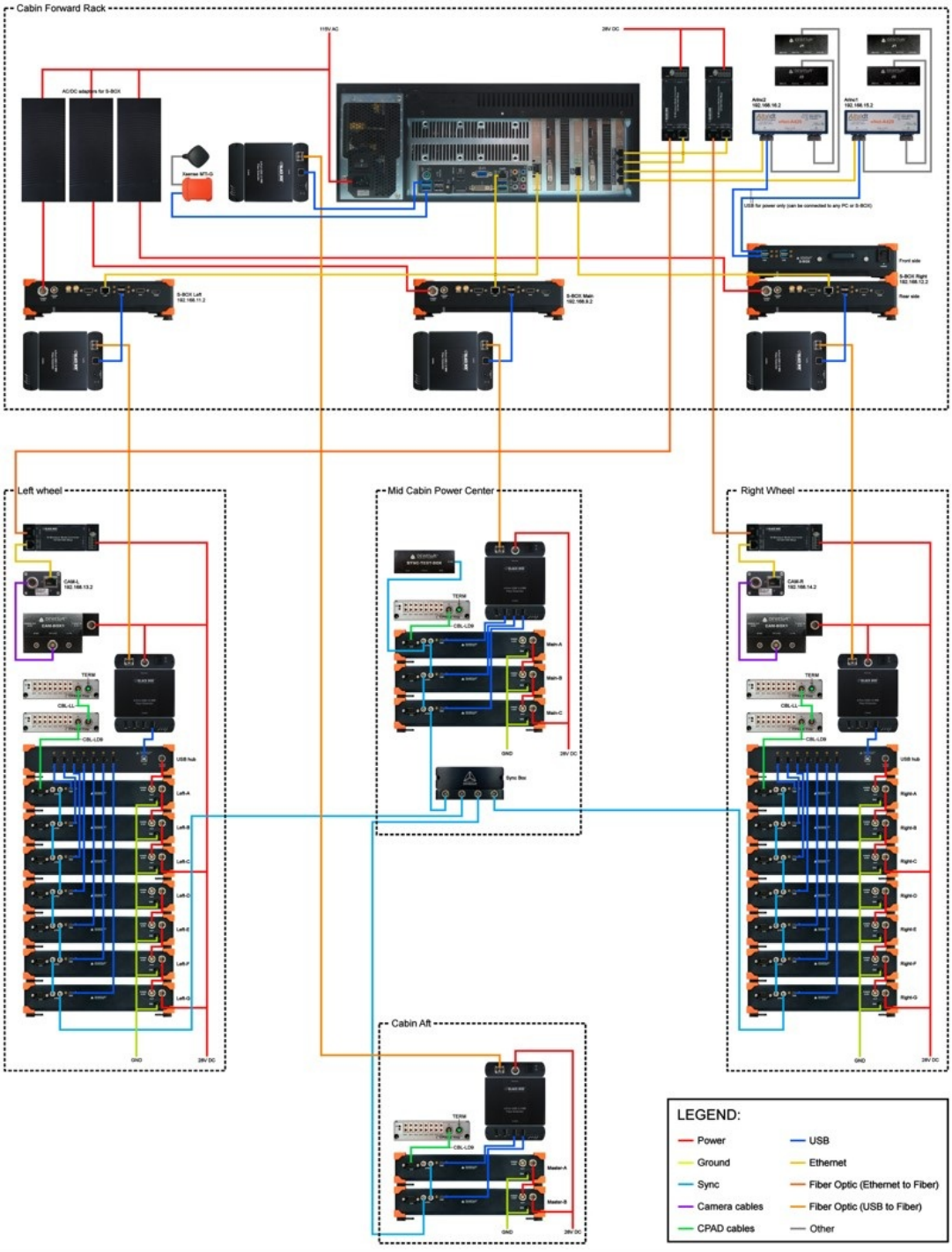
Controlled via Master computer

- Sync: IRIG Master / IRIG Slave
- Three Slave measuring units
- Different SR

Ethernet to optics extenders

- USB to optics extenders

System Wiring Diagram



MEASUREMENT

In this section we can see procedures to control the Acquisition (from the Client) and storing data on net:

- Creating a display on the client
- Remote display setup
- Storing data
- Uploading stored data to the client

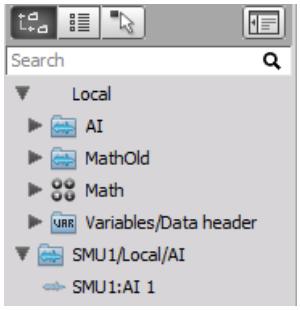
Creating a display on the client

Before you start storing, you may want to set up the local display. In (Remote) Display setup procedure you may have configured the display of one or more MU's, but you probably want to see data here, too!

Use the Overview, Scope, Recorder (et al) buttons at the top of your own screen here on the Client to create displays with any combination of channels from any and all MU's!

As mentioned previously, all MU's must have a SYNC method in place in order to ensure these three things:

- Truly synchronized data files from multiple MU's
- Ability to display channels from more than one MU on the client
- Ability to create math channels on the client with channels from more than one MU



Note that the CHANNELS list is now showing channels with the name of the MU that they come from automatically. This is so that you know the source of every channel in an easy and convenient way.

So in our example, the name of the MU, 'MINITAUR' is added in front of the transfer channels from that MU by default, as you can see above.

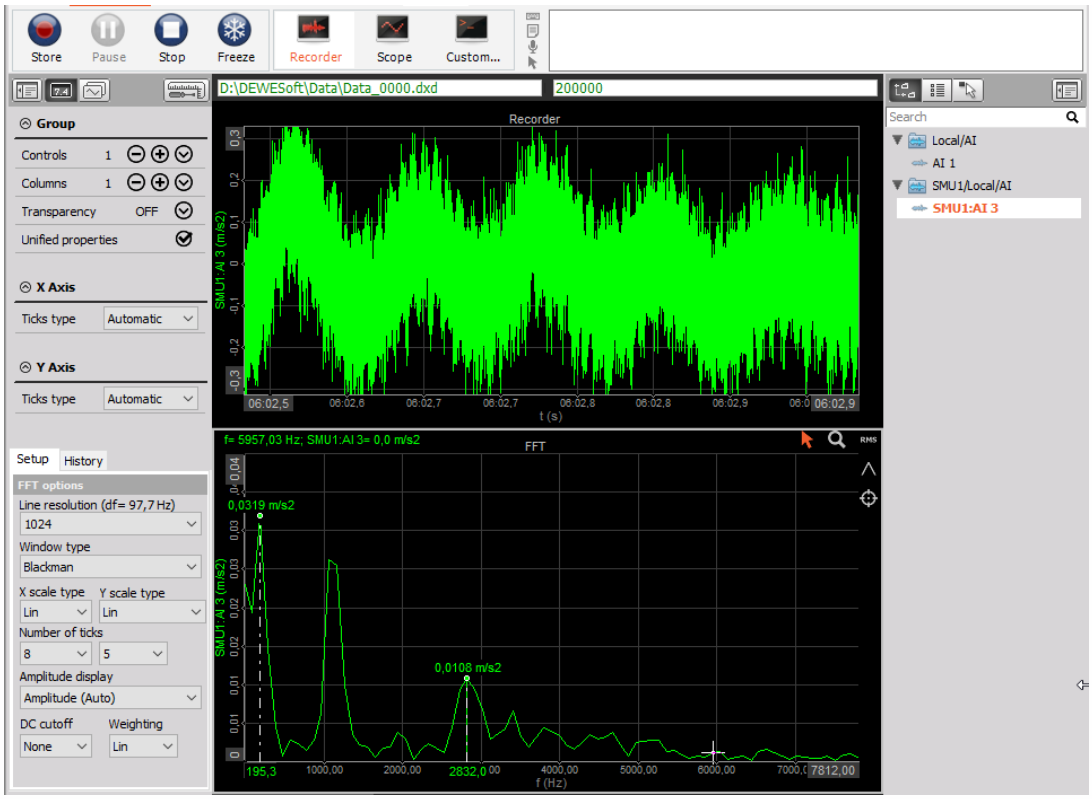
The channels from each MU will be shown this way automatically. This is the only thing that is different from setting up a screen in the standalone mode of Dewesoft, for years now.

Please see the User Guide -> Screen Design if you need further help in setting up a display.

Remote display setup

The Remote Display setup screen configures the displays (Recorder, Scope, analog meters, digital meters etc.) which are shown on the remote (Slave MU) unit(s) when the measurement is started. This feature is useful if no peripherals (keyboard or mouse) are connected on the remote unit or if these have been deactivated during Dewesoft NET setup (Lock mouse and keyboard on measurement units is checked).

This is only important if you want to have a display screen on the MU for local observers to see. If there is no one looking at the local display on the MU (perhaps it is in a remote location without any people near it), then you can skip this procedure. But if you want a local display on the MU, click the Display setup tab and then set up the screen as you desire, using the normal Dewesoft methods and conventions for Screen Design.



For details see User Guide -> Screen Design.

NOTE: **It is really important that the CLIENT computer have a display which has MORE RESOLUTION than the MU's! If your MU's have 1024x768 screens, your client should have the next size up or greater. Otherwise you may run into trouble seeing some of the screen objects near the bottom when remotely controlling MU's from the client.**

The display above is the one that will appear on the screen of the remote MU called MINITAUR! It is not the display that you will see here on the client.

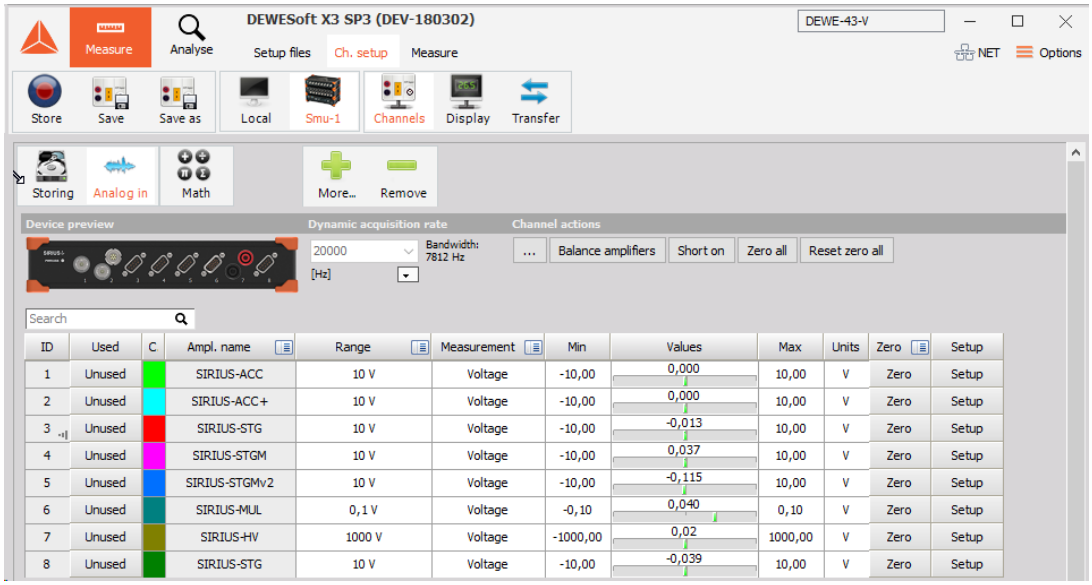
Storing data

The live channel data is transferred via Ethernet to the client(s) with the drawback that the Ethernet interface is the bottleneck for the transfer. Therefore the live channel data can be stored in different ways to avoid excessive network traffic and optimize the calculation power of unit:

- **Local** - Measurement data is stored locally (i.e. on the client side). The live data of the selected channels is transferred to the client where it is stored. The maximum throughput for local storage is 12 MByte/sec.
- **Remote** - Measurement data is stored on the measurement unit (i.e. on the server side). The data is transferred to the client manually (on user request), when the measurement is stopped. The maximum throughput for remote storage is 8 MByte/sec.
- **Local & Remote** - It is often not possible to transfer all possible channels in real time to the client for storage even with a gigabit Ethernet interface. Imagine even one DEWE system with 32 channels, being sampled at 200 kB/s each at 24-bit mode. This is already 25.6 MB/sec, which is more than 200 Mb per second (where each Byte = 8 bits). It does not take long for the network to be completely overloaded with data and be overwhelmed with packet loss. As it is mentioned in System settings -> NET Configuration and Setup is highly recommended to store all measurement data on the measurement unit (Remote storage).

Remote storing setup

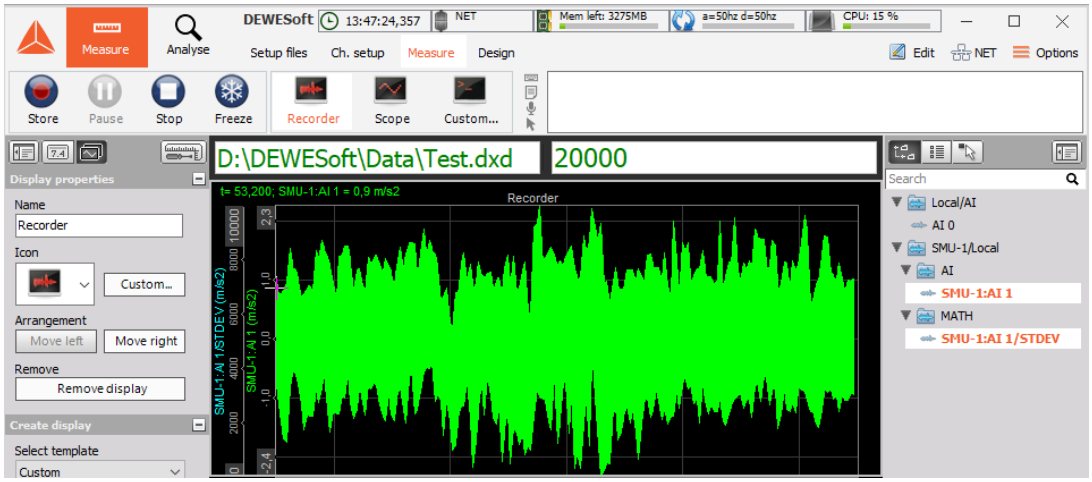
When the connection was successful, the NET button turns green and the Dewesoft Setup screen appears. Each remote unit's system setup can be accessed via the button bar (green indicates the selected unit), therefore click on the name of remote unit's button (in example below MINITAUR) so that we can proceed to set up this Slave MU:



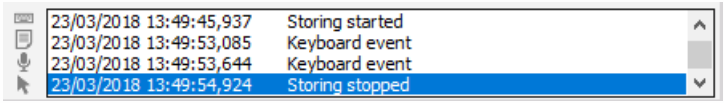
Now click the Setup button and on central part of this screen you are doing exactly what you would be doing by stand-alone MU: set the dynamic and reduced sample rates, choose a filename, set storing option,...

Storing data on the MU(s)

With the client and MU properly configured, we can now store data. Just press Store from the toolbar, in the normal way.



You can see from the event log (see right) that we started storing, added a notice event by pressing the spacebar during the recording, then stopped it by press Stop button from the toolbar.

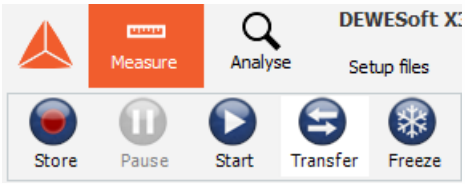


For details see -> User Guide -> Recording Data.

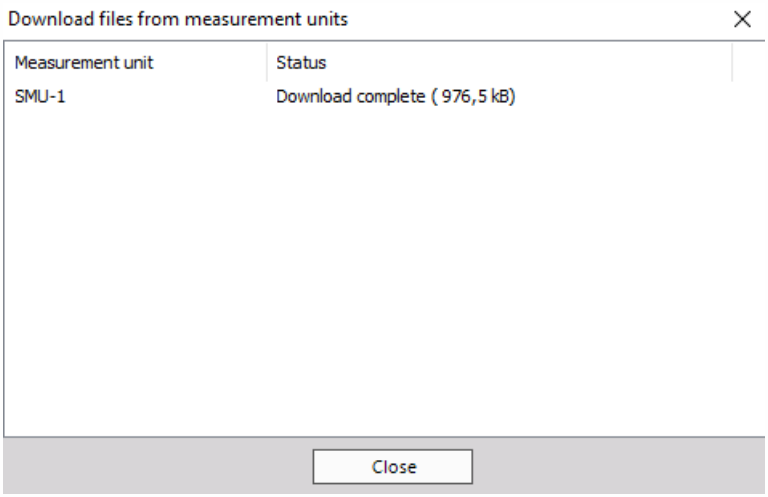
Uploading stored data to the client

Immediately after the acquisition is stopped, a button called Transfer appears on the controlling client allowing the data file(s) to be immediately uploaded from the MU's for viewing on the client computer.

When Store data on measurement units has been checked during Dewesoft NET configuration (System menu -> Hardware setup... -> NET tab), the data is transferred from the local MU for viewing on the client computer by clicking the Transfer button after the measurement.



Please click it, and the data file(s) from all MU(s) that we just used, will be downloaded to the client for you. A transfer box appears to show the progress, and eventual completion of the download:



In this case, we only had one MU named "MINITAUR", so only one file needed to be downloaded.

NOTE: **This is a software option, which must meet Dewesoft NET Requirements.**

ANALYSE

The Analyse procedure is an important part of Dewesoft for analyzing acquired data. To enter the Dewesoft Analyse mode press the Analyse button on the main toolbar.

Once data has been acquired, there are a number of things that you can do with it: review data, display data in the selected instrument display with a signal overview, replay data, start/stop replay and sound output, arranging instruments, (re-)assigning channels, watch events, signal analysis, reload the triggered file, a printout of instruments, store settings, copy Channel setup and display elements to Clipboard, exporting data,...

For additional help visit Dewesoft PRO training ->Dewesoft Web page -> PRO Training.

To enable efficient analyzing process for acquired measured data you must leave the Dewesoft Measure mode and enter the Analyse mode to perform following procedures:

1. STEP: Reviewing data files

- Loading data file to load acquired measured data stored in the data file; display data file information: Settings, Events or Data header
- Displaying data and Replaying data to display data in selected instrument display with a signal overview, replay data
- Selecting data to analyse acquired measured and video data; Selecting triggered data
- Working with Events - keyboard, notice, and voice type

- Storing settings and events
- Video post synchronisation
- Text import

2. STEP: Postprocessing Add the ability to define and recalculate additional math channels in analyse mode.

3. STEP: Publishing the data Printout of instruments, copy Channel setup and Display elements to Clipboard

4. STEP: Exporting data Exporting data for off-line analysis using other software with possibilities to export several files at once - Export multi files and to export any instrument screen to a video - Export instrument display to Video

REVIEWING DATA FILES

Reviewing data files is very helpful to analyse measured data and include the following procedures:

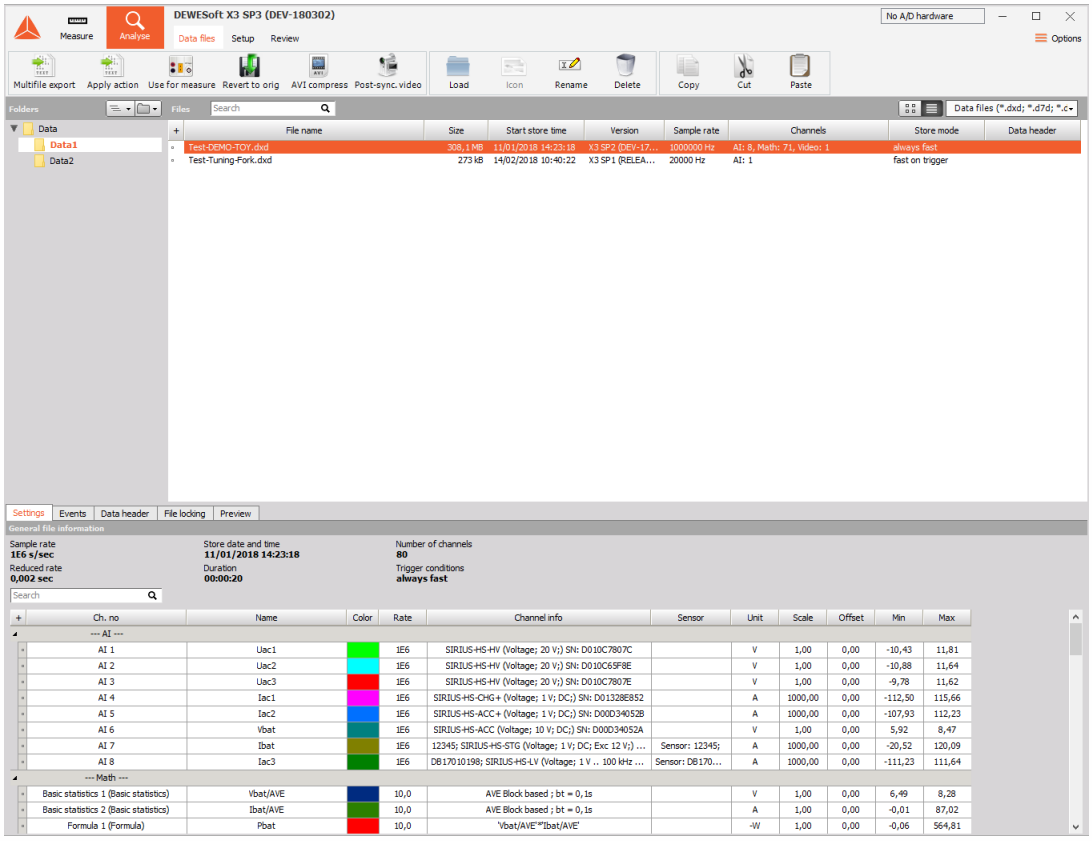
1. Loading data file: to load acquired measured data stored in data file; display data file information: Settings, Events or Data header
2. Displaying data: display data in selected instrument display with signal overview, selecting a channel for overview window, Time selector
3. Include Video file: Video data can be included and synchronized in Dewesoft, zoomed, scrolled, scaled, played, resynchronized, removed from data file
4. Replaying data: start/stop, Sound output selection, Replay mode, Replay direction and Replay speedin different Display Screens
5. Selecting data:

- to analyse acquired measured and video data with functions to have a more detailed look on the recorded data with Selection range of stored data
- to help measure a precise value over a long time
- to Selecting triggered data (to reload multiple trigger events within one file)
- Store settings and events

6. Working with events: to add, display and save Keyboard, Notice and Voice events

LOADING DATA FILES

If you have just captured some data, and you press Analyse button, Dewesoft will automatically load the recent data file. Otherwise (when starting Dewesoft or from the Setup screen), it will present you a selection window (file explorer) where you can select any data file to load:



The window shown above offers subfolders of our main data folder, a lot of information in file list about the existing files and the currently selected file.

Folders structure

The upper left section shows the folders structure. In this area, we can select where the data file should be loaded from. If we have subfolders, we can choose them by double-clicking on the subfolder.

The first level data folder can be changed by clicking on:

- ‘up’ button, which brings us one level higher in the folder structure:

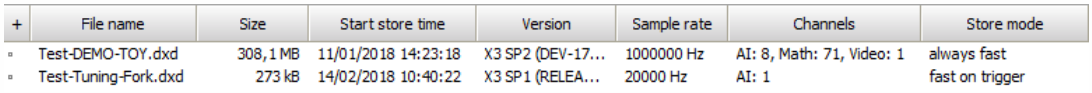
- ‘folder’ button to browse for the new folder.

The default folder can be remembered by right-clicking on the folder list and selecting 'Set by default' choice.

File selection list

In the upper right section, you can select the file which should be loaded for analysis with double click on it. It provides you also with plenty of information about all files available in the selected folder:

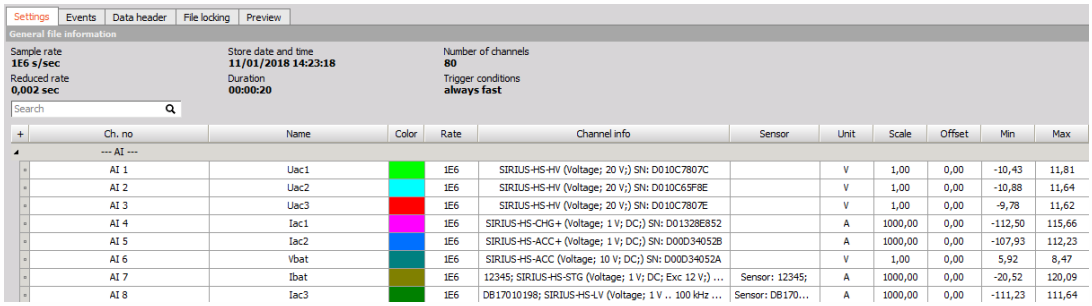
- **Name** - Name of the file
- **Size** - Size of the file in kB (= 1024 Bytes)
- **Start store time** - Date and time when the file has been modified
- **Version** - The version of Dewesoft used to acquire the data file
- **Sample rate** - The used sampling rate; it will also show the reduced rate if the data was stored slow and fast on trigger
- Channels - Number of active channels
- **Store mode** - One of the four storage modes: always fast, always reduced, fast on trigger or always reduced and fast on trigger
- **Video** - Video type and file size (available only when a video has been recorded)



+	File name	Size	Start store time	Version	Sample rate	Channels	Store mode
▫	Test-DEMO-TOY.dxd	308,1 MB	11/01/2018 14:23:18	X3 SP2 (DEV-17...	1000000 Hz	AI: 8, Math: 71, Video: 1	always fast
▫	Test-Tuning-Fork.dxd	273 kB	14/02/2018 10:40:22	X3 SP1 (RELEA...	20000 Hz	AI: 1	fast on trigger

File Settings information

The bottom of the window displays the Settings, Events and Data header of the selected file. As a standard, Settings is selected.



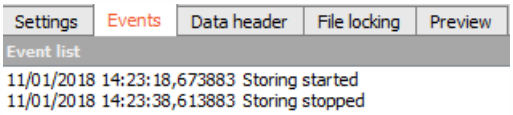
+	Ch. no	Name	Color	Rate	Channel info	Sensor	Unit	Scale	Offset	Min	Max
	--- AI ---										
	AI 1	Uac1		1E6	SIRIUS-HS-HV (Voltage; 20 V;) SN: D010C7807C		V	1,00	0,00	-10,43	11,81
	AI 2	Uac2		1E6	SIRIUS-HS-HV (Voltage; 20 V;) SN: D010C65F8E		V	1,00	0,00	-10,88	11,64
	AI 3	Uac3		1E6	SIRIUS-HS-HV (Voltage; 20 V;) SN: D010C7807E		V	1,00	0,00	-9,78	11,62
	AI 4	Iac1		1E6	SIRIUS-HS-CHG+ (Voltage; 1 V; DC;) SN: D01328E852		A	1000,00	0,00	-112,50	115,66
	AI 5	Iac2		1E6	SIRIUS-HS-ACC+ (Voltage; 1 V; DC;) SN: D00D34052B		A	1000,00	0,00	-107,93	112,23
	AI 6	Vbat		1E6	SIRIUS-HS-ACC (Voltage; 10 V; DC;) SN: D00D34052A		V	1,00	0,00	5,92	8,47
	AI 7	Ibat		1E6	12345; SIRIUS-HS-STG (Voltage; 1 V; DC; Exc 12 V;) ...	Sensor: 12345;	A	1000,00	0,00	-20,52	120,09
	AI 8	Iac3		1E6	DB17010198; SIRIUS-HS-LV (Voltage; 1 V .. 100 kHz ...	Sensor: DB170...	A	1000,00	0,00	-111,23	111,64

This part shows now information about the selected file on General file information section (Sample rate, Store date and time, Number of channels and Trigger condition) and more detailed information on Channel info section:

- **Ch No.** - Channel number
- **Acq. rate** - Acquisition rate will show the acquisition rate of each channel. This is important if sample rate divider is used or if the channel is asynchronous (CAN, GPS, PAD,...). In this case, it will show approximate sample rate (or keyword ASYNC with old file versions)
- **Name** - Channel name and color of the channel
- **Settings** - Type, input range and filter range of the amplifier. For newer amplifiers also the serial number is mentioned
- **Scale (k)** - Scaling factor k (y = kx + n)
- **Offset (n)** - Scaling offset n (y = kx + n)
- **Range (from..to)** - Scaled input range (min. to max. value)
- **Min** - Minimum value within the whole file for the channel
- **Max** - Maximum value within the whole file for the channel

File Events information

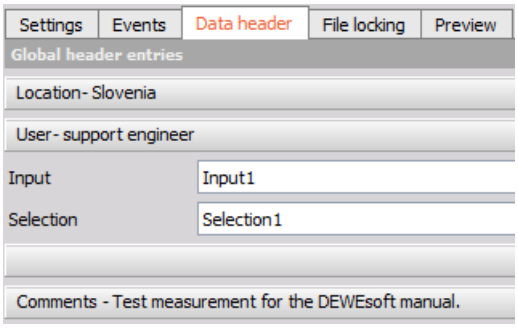
When you change to Events, all happened events will be displayed. Events are start and stop of measurements, keyboard events, notice events and also voice events.



For information about enter Events for the measurement see -> User Guide -> Marking your data with events.

File Data header information

The Data header finally displays all the information you have entered in the data header window at the beginning (or the end) of a measurement. The exact content depends also on the fields you may have changed in the global header design.



Loading a data file

Select any data file in the file list and double click on the entry to open the file in the preferred instrument or select the desired display, for example the recorder or the FFT.

All of the channel scaling, name, and units info that were in effect at the time of recording are restored so that your data can be properly interpreted, and the Recorder display screen will be shown.

For information about display data on screen see -> Displaying data and Replaying data.

To load any existing data file, you can also select Load Data File from the main Dewesoft Data menu (requires that you be in the Analyse mode).

Display a data file information

When during Analyse mode Setup button on Dewesoft toolbar is selected, Settings, Events and Data header information are displayed on whole screen and Analyse mode is interrupted.

To continue with Analyse mode the Analysis button on main toolbar must be selected again.

DISPLAYING DATA

As soon as you have reloaded the measurement file in Dewesoft Analyse mode, the recorded data will be displayed. Independent from the selected instrument type, there are several elements which are the same for all instruments.

Signal overview window - on the top of display screen is small bar with one channel as an overview of the whole measurement with possibility of Select a channel for overview and display the signal only within the selected Time overview box using Time selector.

Let's take as an example the Overview screen:

NOTE: **All section of this chapter refers to the file Example_Drive01.dsd, which will be installed automatically on your system with the standard installation of Dewesoft. You can load this file to try out the described functions.**

All of the channel scaling, name, and units info that were in effect at the time of recording are restored so that your data can be properly interpreted.

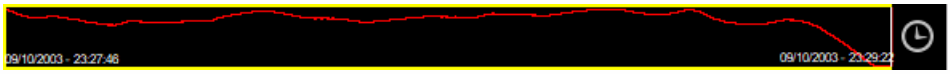
The display settings can be done in Measure and Analyse mode. You can use in Analyse mode all of the features available in the Measure mode: to add/delete graphs and inputs from each graph, change the T-axis and Y-axis scales, and more. If we alter the display settings in the Analyse mode we need to manually store those settings with choosing Data -> Store settings and event and we can also use these settings for next measurement with Data -> Use setup for the measure.

For information about Store settings see -> User Guide -> Settings and Events.

The interesting area for us is now the top area of the screen. You will see several elements which are not available in the Measure mode. Let's go into detail now.

Signal overview window

Each instrument offers at the top of the screen a small bar with one channel as an overview of the whole measurement:



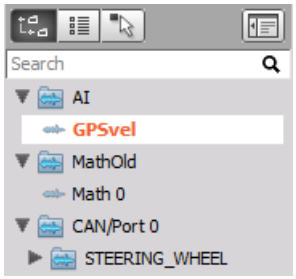
Stored data offers at the top of the screen a small bar - Signal overview window / Time overview bar with:

- symbolic display of signals in white bordered rectangular with yellow cursor as an overview of the whole stored data
- start date and time of stored data
- end date and time of stored data
- Time selector and Time overview box

Within the Time overview bar there is also a cursor available and as a standard it is located at the left side of white bordered rectangular. If you look carefully at this area, you will see a yellow vertical line and this is a cursor.

Select a channel for overview window

Click on the overview bar to select it. Now you can change the displayed channel - as a standard, it is the first acquired channel. To do that simply clicks on the name of the channel in CHANNELS selector as we have already done before in the Measure mode.



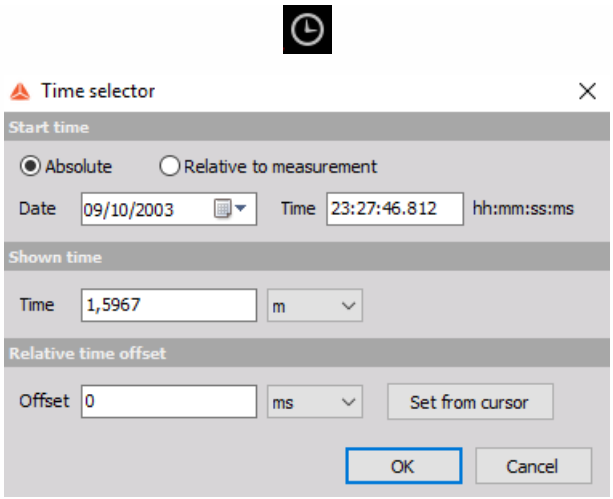
For information about Channels selector see -> Channels selector.

Now in overview window symbolic display of new selected channel is displayed:

Whatever is displayed in the instruments below, the Time overview bar will always show you the whole signal. For example you have selected just a small time area in the recorder within the whole signal; you get this area marked in the overview bar.

Time selector

Click Time selector icon on the right side of the overview bar, the following Time selector window will pop-up on Analyse screen below Time overview bar:



Now you can enter:

- Shown time; from drop down list in right field day, hour, min, sec or millisecond time unit can be selected.
- exact Start time position within the file:
- Absolute - in left field date is displayed, which can be selected from the drop-down list (calendar is displayed); in right field, start time to show measurement is displayed in form h:mm:sec - we can direct enter values or select group h, min or sec
- Relative to measurement - in left field days are displayed, which you can enter direct into field); in right field start time to show measurement is displayed in form h:mm:sec - we can direct enter values or select group h, min or sec.

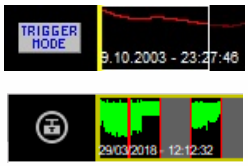
The result depends on the used screen instrument: the single value elements like digital or analog meter, bargraph,... will show the current values the beginning of the selected time window, multiple value elements like the recorder, scope, FFT,... will display the signal only within the selected Time overview box (= like a zoom function).

We can select another stored data within the whole signal with movement Time overview box on Time overview bar but with same range of stored data. You can move the selection window - Time overview box in three ways:

- enter a new start time in the Start show time window as described above (by this option we can also change size of range - Showtime)
- click on the selection window - Time overview box and move it by mouse
- simply use the cursor keys (this will move the selection window each time by the half size of the window itself)

Time overview box

Selected range of stored data within the whole signal is marked in the Time overview bar with white bordered box - Time overview box (Time window). Symbolic display of signals outside of this box is dimmed.



In case of selecting only range of stored data on Time overview bar Trigger switch button changes

and instead appear button.

Within the Time overview box there is also a cursor available and as a standard it is located at the left side of box. If you look carefully at this area, you will see a yellow vertical line and this is a cursor.

With mouse pointed on yellow cursor mouse cursor changes and then you can can drag this cursor within box.

At the same time move also yellow cursor relatively proportional to stored data in the Instrument Display on central part of screen.

After analyse range data in Time overview box with the press on this switch we enlarge selection

to initial - whole range of stored data and Trigger mode button return to:

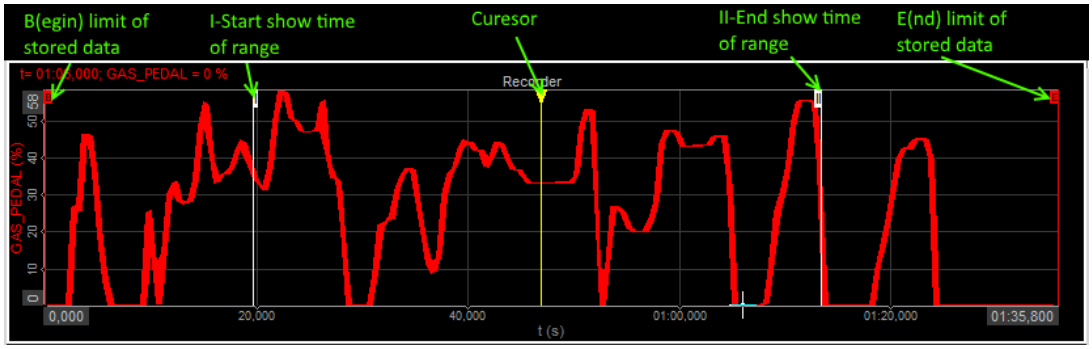
SELECTING DATA

In addition to the online visualization, the Dewesoft Analyse mode offers several functions to have a more detailed look on the recorded data:

- Selection range of stored data we have options to narrow range of displayed data and select only part - smaller range of stored data:
- Measurement cursor
- Zoom in time axis
- Window zoom
- Lock Cursor 1 to selected position to help measure a precise value over a long time

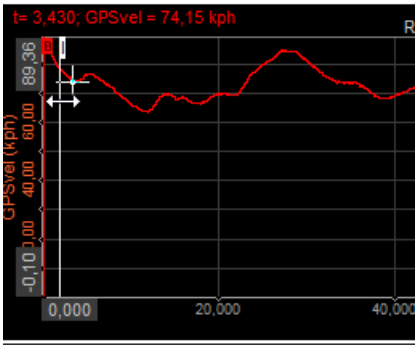
Selection range of stored data

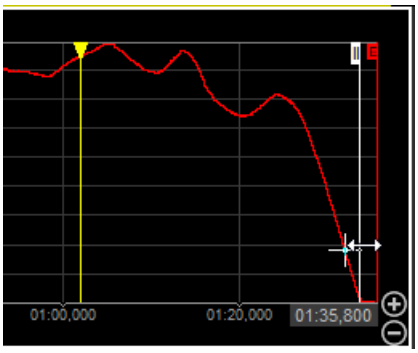
Measurement cursor



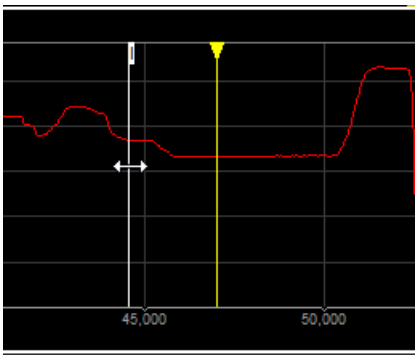
The recorder and vertical recorder offer two measurement cursors for the active graph on top labeled with " I " for Start show time and " II " for End show time. You can drag these cursors to select a certain region in two ways:

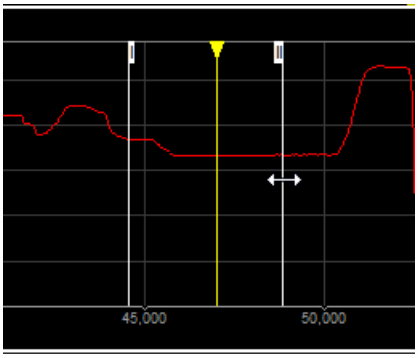
- move mouse cursor to the first position of interest; click (measurement cursor ” I ” appear on this position) and hold left mouse button and move (on mouse cursor appear measurement cursor ” II ” and move whit him) to the second position of interest





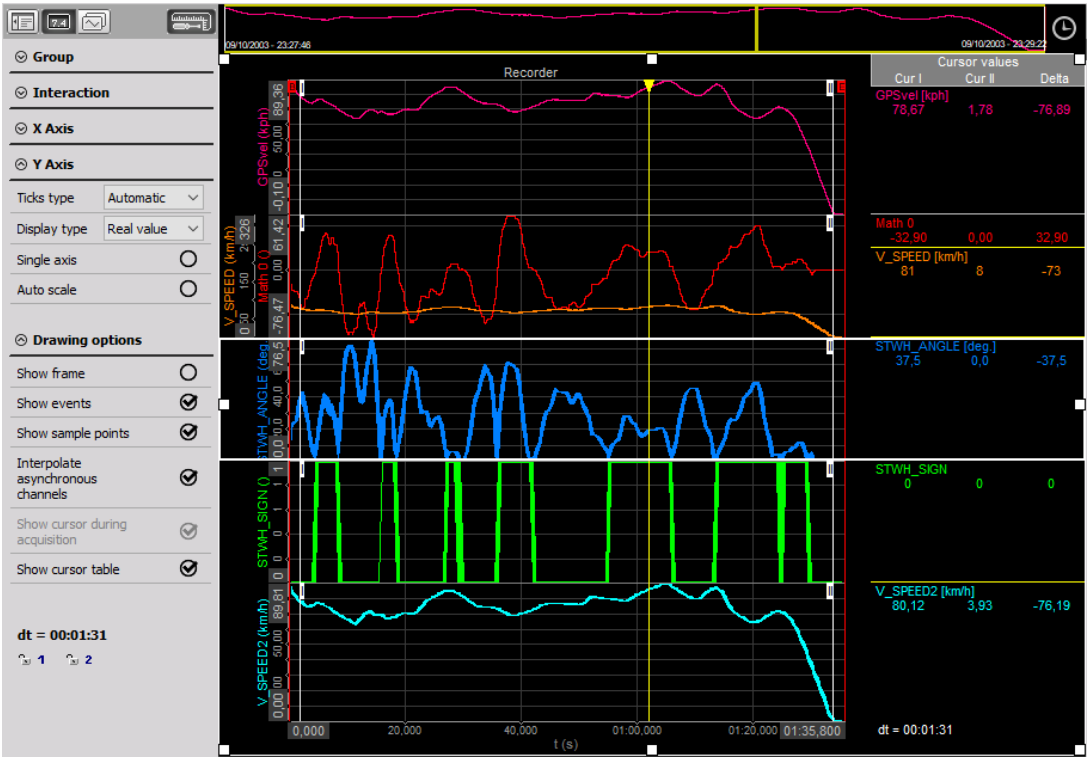
- drag the first cursor ” I ” from leftmost side of the graph to the position of interest and the second cursor ” II ” from the rightmost side of the graph (when mouse point on vertical line of measurement cursor mouse cursor change to two-sided arrow which indicate feasibility to move measurement cursor)



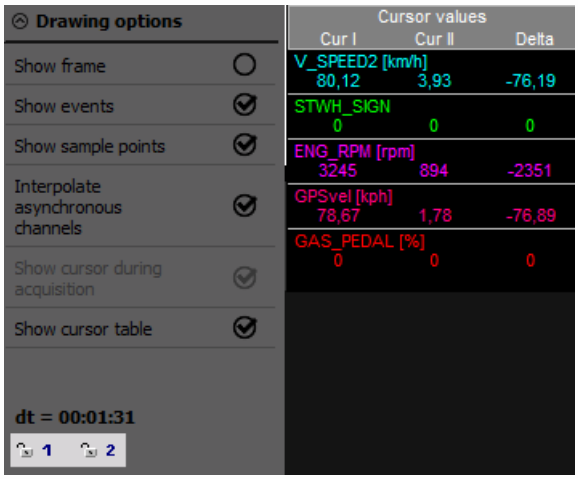


It’s no matter in which recorder you use the cursor; it will automatically move in all displayed recorders and vertical recorders simultaneously.

On the left part of the display, the readout values of the currently selected graph at cursor position will be displayed. To show values from another graph, simply click on it and the values will change. As there is only one-time base available, the cursor position is the same for all displayed graphs:



As we can see above, in Analyse mode on left part of screen below recorder settings (same also by vertical recorder) appear new section to display readout values of signals and time at cursor position - Time and values cursor with icon to lock cursor (see also Lock Cursor 1 to selected position).



Zoom in time axis

After loading the data, you will always see the signal over the complete storage time. But you can also see details - zoom into the area with:

- Just use both measurement cursors as described above to select an area of interest (see also -> Signal overview).

Move the mouse between these two measurement cursors - it will change the appearance immediately to zoom icon.

Then simply do a left mouse click to zoom into the selected area. We can do this several times to come to the region of interest. We can also drag the x scale left and right to position the data exactly. To undo the zoom just use the right mouse button and you will zoom out step by step to the full scale.

Do a left mouse click on the Plus button on right side of graph to zoom into the selected area.

To undo the zoom use the Minus button and you will zoom out step by step.

As there is only one time base available all graph are zoomed in (or out) together, independent which button is selected.

You can do that action several times to zoom into the selected area step by step (in each step it is decreased by the range of about 1/25 of the prior range) and is meaningless in what place is the cursor.

By first zoom out, graph returns to previous zoom (range) and after that, step by step. If we use

before multiple zoom (with + button or measurement cursor), in the first step displayed range returns to about 3/4 of its whole range and after that step by step.

After zoom into the selected area, you will see now more details.

The signal overview bar at the top of the recorder display shows always the current position within the signal. You can also click on the marked area and move the marker - this will also move the displayed signal.

You can do that action several times to see the information of real interest.

Lock Cursor 1 to selected position

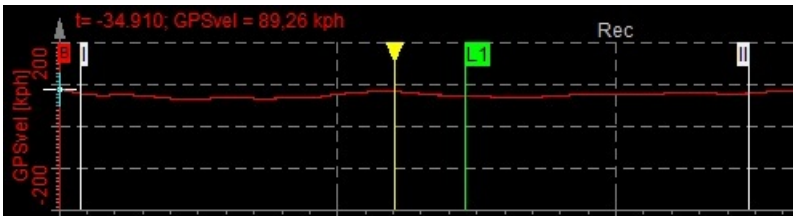
We can position the measurement cursors anywhere within the acquired signal. But what to do if you have to measure a precise value over a long time? The resolution may not be enough to set the measurement cursors at the exact positions.

The solution is quite simple: Zoom into your signal at the point where you want to place the first cursor and position cursor 1 there. Now press the

icon to fix cursor 1 (" I ") to its current position. Lock cursor icon change to

and the cursor will now change to green color and the number at its top changes to locked.

Now you can zoom out and in again to the desired second position. During that you will see a new cursor 1, but this one is only used for zoom issues - not for measurement. The 'original' cursor 1 is still locked at its position. Use cursor 1 and 2 as described before to define the second area of interest. Now move the cursor 2 to the point of interest and you can readout the desired value from the Time and cursor values readout section.



To unlock cursor 1 position simply press

icon again - the cursor

disappears and cursor 1 is again available for new locks. We can also lock cursor 2.

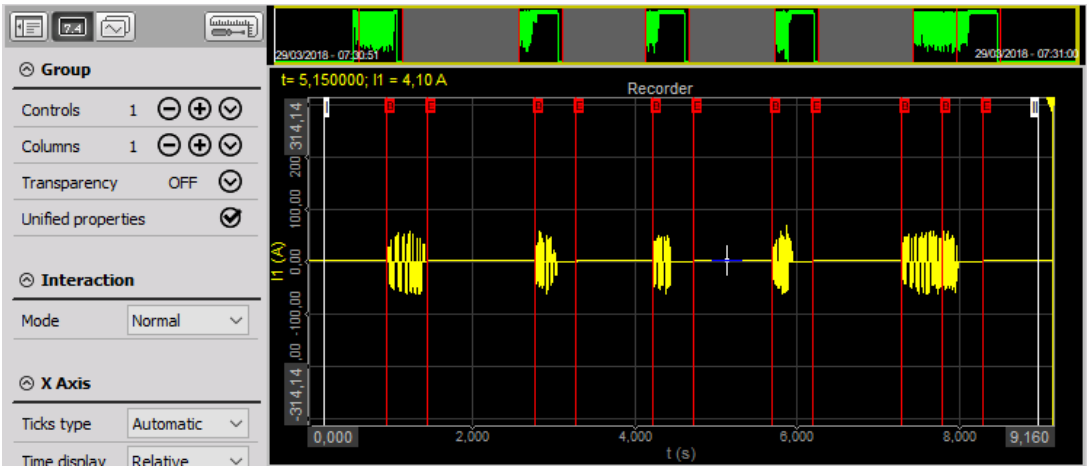
SELECTING TRIGGERED DATA

If our data consists of events which can be captured, we can choose to store fast on trigger. The trigger event can be defined in the software and then Dewesoft will wait for this event and:

- store only the portion of interest by choosing “fast on trigger” storing option
- acquire data with two speeds (to have reduced data also for the regions without trigger event) - we need to use a different strategy Fast on trigger, slow otherwise.

'Fast on trigger' triggered data file

The reload of multiple trigger events within one file shows a different display. We can see that only the trigger events are stored and for the rest of the time the data is blank.

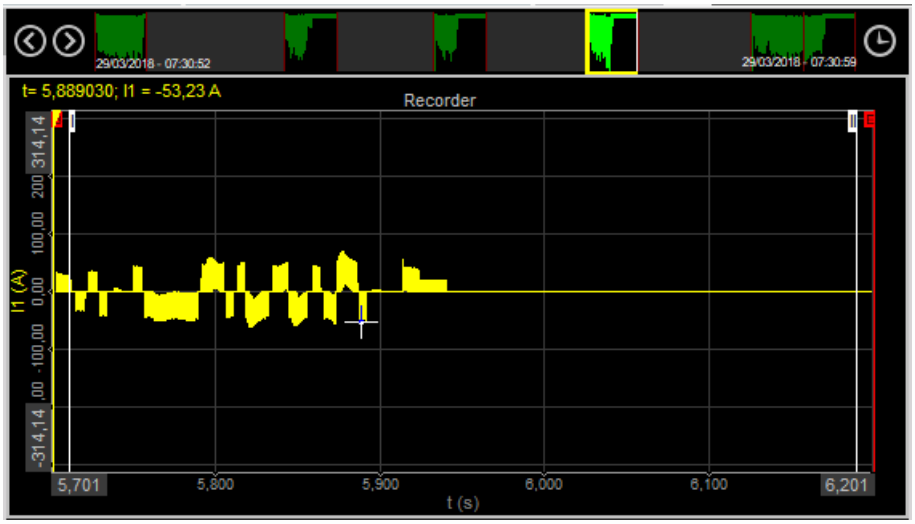


Note that there is a new 'trigger mode' button in data preview.

This gives us a chance to review the trigger events without zooming in the data. If we press it, first trigger event is automatically zoomed in.

The trigger mode button changes to arrows button where we can browse between the events.

If we press those two buttons, the recorder shows the trigger events one by one. On the data preview we see the currently selected trigger event.



To display each event separately, just press the

button and after that use the

arrows to move forward and backward between the events - select event. On first trigger only

arrow is available - to navigate forward to second trigger and on last trigger only

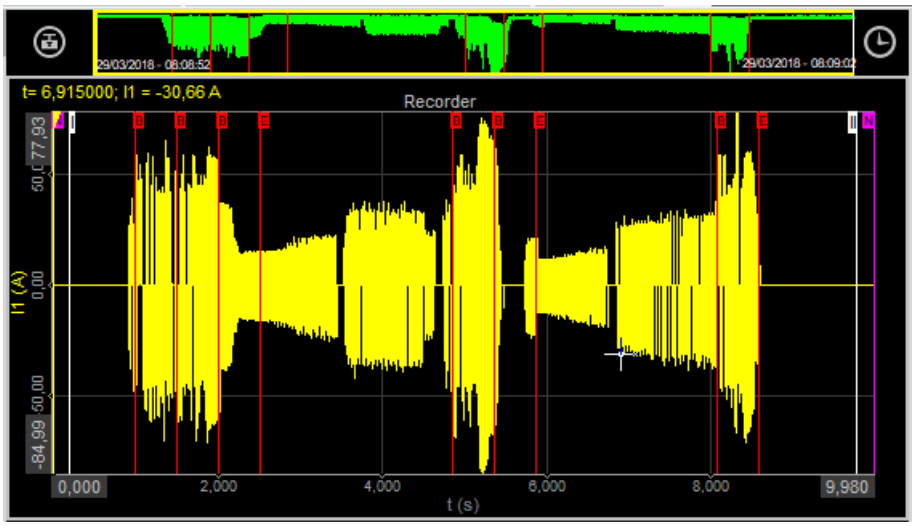
arrow is available - to navigate backwards.

For leaving the trigger mode we can right click on the recorder to zoom out to full region.

All other Analyse action and procedure on triggered signals are same as described -> Reviewing data files.

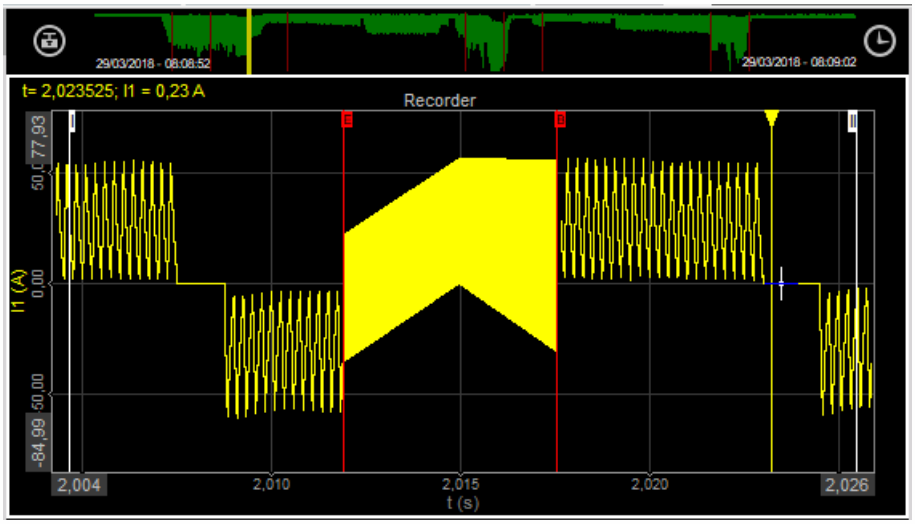
'Fast on trigger, slow otherwise' triggered data file

If we acquire similar data with this strategy and reload it, we can see from the picture below that we have reduced data also for the regions without trigger event.



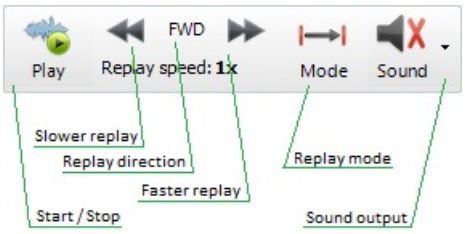
All other Analyse action and procedure on triggered signals are same as described -> Reviewing data files.

If we zoom in the data, we can see reduced stored data before the trigger, where we can only see the maximum and minimum of the signals and then for a region with trigger we can see the full speed data.



REPLAYING DATA

In any instrument it is possible to replay the data in real-time, faster or slower. Therefore Dewesoft offers special control buttons: Start/Stop, Sound output selection, Replay mode for selection different ways of replay, Replay direction and Replay speed in different Display Screens, which you can find at the top right part of the window:



Whit these six buttons you can control the replay process:

- start / stop the replay
- define channel for sound output
- change the replay mode
- change the replay speed
- change replay direction

Start / Stop replay

To start the replay simply press the Play button:

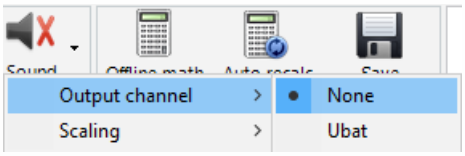
Dewesoft will immediately start to move through the acquired data: the FFT is calculated, scope shows the current data, the video file is replayed, digital or analog meters, bar graphs,... will change their value continuously and in recorders or scopes you will see the yellow cursor moving to indicate the current position within the file.

When the replay is started, Play button change to Stop button, to stop the replay simply press this button.

After pressing the Stop button don't forget to select None at the loudspeaker icon to switch off audio replay (see below).

Sound output

We can even hear the sounds we have stored. Next by the Play button there is a loudspeaker button, but with the red cross. If we click on it, the channel list will appear to select analog input channels for output to sound card. Choose the only channel we have stored and the loudspeaker will not have a red cross anymore. Now press again a Play button and whatever we have recorded will be heard from the loudspeakers.



After pressing the Stop button to stop the replay select None at the loudspeaker icon to switch off audio replay.

Replay mode

You can select between three different replay modes (simply click on the third button from left to change it):

- **Normal** - Replays data once from the beginning to the end of the file or the selected time window.

- **Loop** - Replays data continuously from the beginning to the end of the file or the selected time window.

- **Scroll** - Replays data once from the beginning of the selected time window to the end, then it continues to replay by moving the selected window until the end of the file is reached.

The replay mode can be changed also during a running replay process.

Replay speed

Use the buttons to change the replay speed down to: 1/2x, 1/4x, 1/8x, 1/20x, 1/50x, 1/100x, 1/200x, 1/500x, 1/1000x or 1/2000x real time.

Use the buttons to change the replay speed up to: 1x, 2x, 4x, 8x, 10x, 100x, 1000x, 2000x or 5000x real time.

The replay speed can be changed also during a running replay process.

Replay direction

Change between forward and rewind.

When the FWD button is displayed, with a click on it this button changes to

REW button and data replays backwards.

The replay direction can be changed also during a running replay process.

WORKING WITH EVENTS

Events

You can also see all your events - keyboard, notice, and voice types - on the event list at the top right, directly below the replay control buttons.

Select any event either by clicking on its white or gray vertical line on any recorder graph, or by scrolling through the EVENT selector at the top right of the screen. This is a scrolling list that shows:

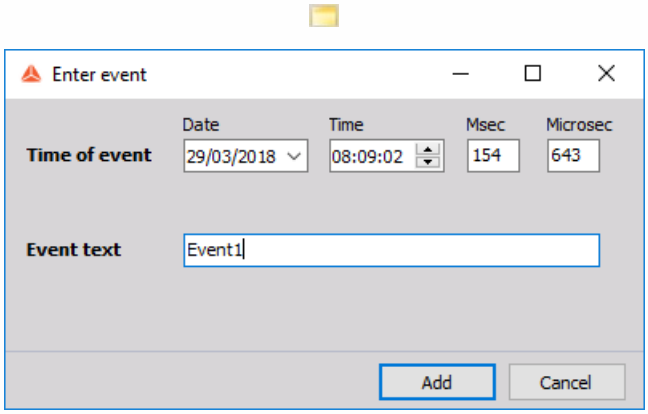
- when data recording began - indicated by a vertical red line on the graph with a ” B ” (beginning) at the top, and each type of event, listed in chronological order, and marked with the exact time that they occurred
- the end of the recording is also marked with a vertical red line and an ” E ” (end) at the top
- if you click on a voice event, it will replay using the speakers in your computer (assuming that you have this DirectX sound recording/replay capability)
- if you click on any notice event, it will show in the EVENT section, and just by hovering the mouse over the gray vertical line, the text that you typed in at the time of recording
- keyboard events show the exact time that they occurred

For information about marking recording data with events see -> User Guide -> Manually Start / Stop Recording.

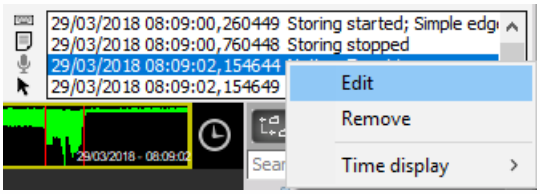
Manage events

As mentioned before, all events are displayed in the recorder.

To add an event just clicks on the notice icon or press the shortcut key, for example for a notice event and the Enter event window will pop up.



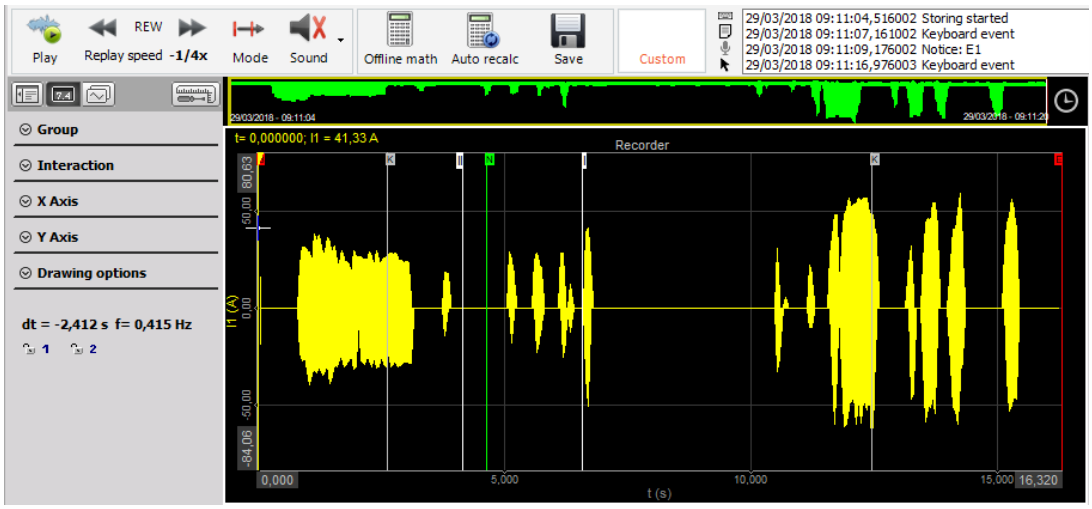
Time of event fields are already filled, enter Event text and with press Add button accept the new event.



To change previous entered notice right-click on this notice in the list, from displayed menu choose Edit and Enter event window will pop up. In this window now Add change to Update button, with which we accept changes to carry out in Event text field.

To delete notice select Remove from displayed menu.

Enter as many events as you need; the types Keyboard event, Notice event and Voice event can be mixed.



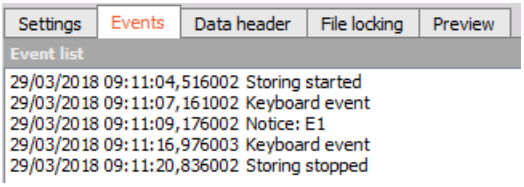
After all your new events are defined, select Store settings and events from the Data main menu. All events will now be stored into the already open data file.

NOTE: **This function stores also the current display settings into the file!**

If you have changed anything in your display appearance, it will now be stored also.

Click again on the Analyse button and have a look on the file explorer. It shows now not any older software versions for this file, it has changed to the current version. So we also did a file version update!

When you select the Events tab, you see the entries for all old and new events:



STORING SETTINGS AND EVENTS

As described in the Working with events section, you can store beside events also changes in the display appearance of the current data file in this file.

Store Setup File

In addition, you can also save the whole settings into a completely new setup file. This file can be used for example for another measurement. Select Save Setup to File from the File menu. A window will open where you can enter the file name of the setup. Press OK to confirm and you have your new setup file stored.

For information about Save settings see -> Setup data.

Load Setup File

Sometimes it is necessary to compare data from different measurements. But the display settings are different: The files are hard to compare. Dewesoft offers the possibility to take any setup file and load its display settings into the currently opened analysis file.

For information about how load setup file see -> Load Display Setup.

WARNING: **Be careful to have the same channels in source and destination data files to avoid any errors or misinterpretations.**

VIDEO POST SYNCHRONIZATION

Dewesoft supports many different video cameras online. But there are cases (for example with very fast video cameras) where we have data file and the video file coming from the frame grabber. It is possible to synchronize those two files in Analyse mode with two simple steps:

1. **Include Video file** - to copy video file and rename it according File name conventions.
2. **Video synchronization** - to synchronize video file with analog data in the Dewesoft data file with:

- Synchronization settings
- Resynchronize files
- Storing post synchronization information

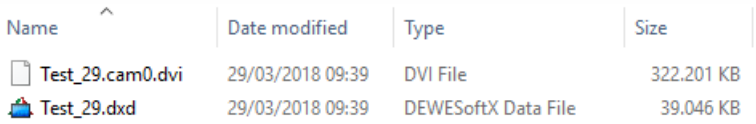
The Video file can be also removed from data file -> see -> Remove video from data file.

File name conventions

To include the high speed video in Dewesoft, there are only few steps necessary.

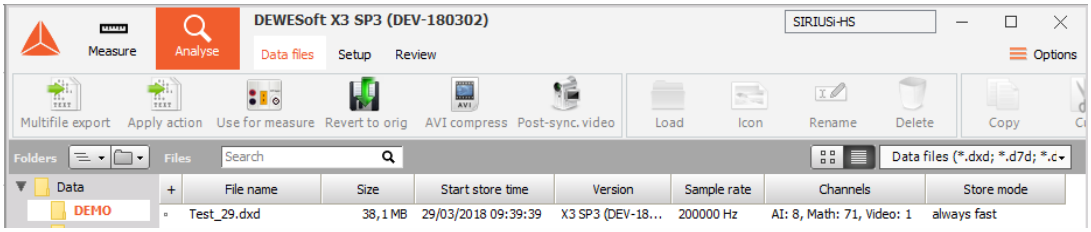
1.After acquisition of video file and the Dewesoft measurement data, please copy *.avi high speed camera file via Windows Explorer to the directory where the Dewesoft data to be synchronized is located.

2.Please rename video file to this specification: xxxxx.cam0.avi,
where 'xxxxx' is name of Dewesoft file to be synchronized. If we have more video files, we can name then xxxx.cam0.avi, xxxx.cam1.avi and so on.



The picture above shows the example how this looks like. Original Dewesoft file is test.dat and therefore a video clip should be named test.cam0.avi.

3.After this open Dewesoft, press the Analyse button to enter the Analyse mode and you should see already video file as part of Dewesoft data file.



Synchronization settings

After including video in data file double click the file to open it. Dewesoft will recognize that this file has no synchronization information included and will ask to synchronize it manually.

After that, specify the Pre trigger: How many pictures have been taken before the video trigger occurred (choose between frames, seconds and milliseconds).

Trigger time finally defines the start position of the video: from trigger means first appearance of a trigger event (or start of measurement - storing of analog data). In this case, both the measurement system and video camera require the same trigger source.

The second way is to define video start from relative time. In this case, the mentioned time counts from the beginning of measurement in seconds.

After you have done all settings press OK button.

The files will now be synchronized, but the synchronization information is not stored at this time! Dewesoft will now display both files in the Video display.

Resynchronize files

The Video display can be handled as any other instrument within Dewesoft: zoom, scroll and scale, play,... - whatever is required. If the synchronization is not perfect, it can still be adjusted by selecting: Data menu -> Video post synchronization.

NOTE: **This option is only available if the synchronization information hasn't been stored!**

Now you get the same Video post synchronization window as described in Synchronization setting (see above). With an additional feature: Trigger time - from position. This is really a great feature, because you can zoom into your display, move the yellow cursor in the recorder to exactly defined position and select Video post synchronization from the Data menu. If you do so, the video will be placed to the position of the yellow cursor. This will also work if pretrigger is required.

Using this function, you can move position of the video file within the data file forward and backward as required.

Storing post synchronization information

Currently there is no separate storage button available. To avoid doing all the synchronization setup every time, simply click on the Analyse button to go back to the file selection list. A warning message will now appear asking if the synchronization information should be stored now or not.

If you select Yes, the information is stored directly into the video file (not the data file!) and is now available at any time, no further synchronizations are required.

WARNING: **If you have stored post synchronization information once, it can't be changed any more!**

Therefore you should always keep a copy of the original video file, which doesn't contain this synchronization information.

Remove video from data file

To remove the assigned video from the data file, simple rename (or delete) the video file. In this case, the video file is not visible any more in Dewesoft. As the synchronization information is stored in the video file, there isn't even an error message, because the data file doesn't miss the video.

Make a short test:

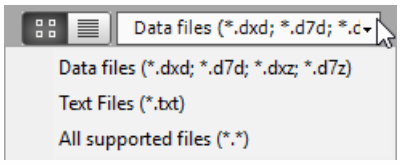
Rename the video back to the data file name and open it in Dewesoft - the video is back and still synchronized.

TEXT IMPORT

With the Text import plugin you can easily import data in *.txt format to Dewesoft.

Enable the plugin

You will find both the 64-bit and the 32-bit version of the plugin on our web-page. The procedure to enable the plugin is the same as for any other, you just need to copy the *.imp file to the “Addons” or “Addons64” folder. Be careful that you download the correct version. If the plugin is correctly enabled the option to view text files will be added in Analyse mode.

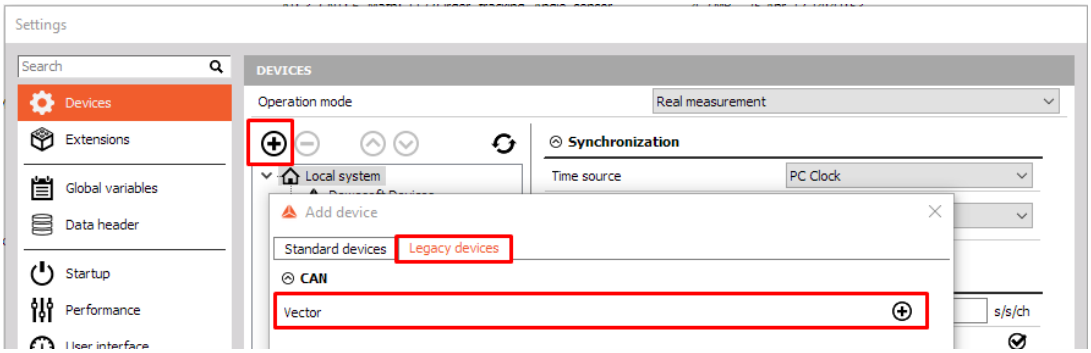


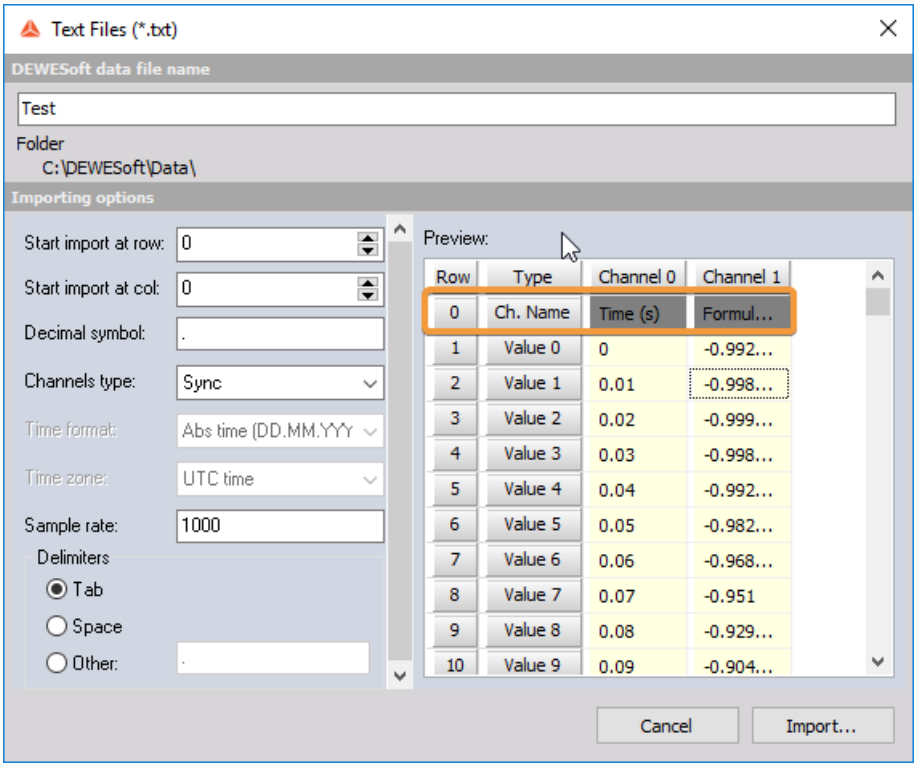
Licensing

You need to have a valid license in order to import text files. You can have any license, other than DEMO or an evaluation one. If you have a DEMO or evaluation license you can only import 1 second of data. If your license is written on a device you need to have the device connected while you’re importing the data.

Settings

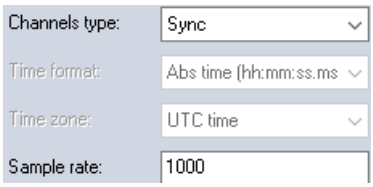
When you load the .txt file the “Text Files (*.txt)” setting will appear. On the top you can select the file name and also see the path to the folder where the file will be created in .dxd format. You have a lot of “Importing options” which you have to carefully set, so the data is correctly imported. On the right site there is also the preview of the data, where you can see if settings were chosen correctly. You can see that the first row of the *.txt data represents the Channel name.





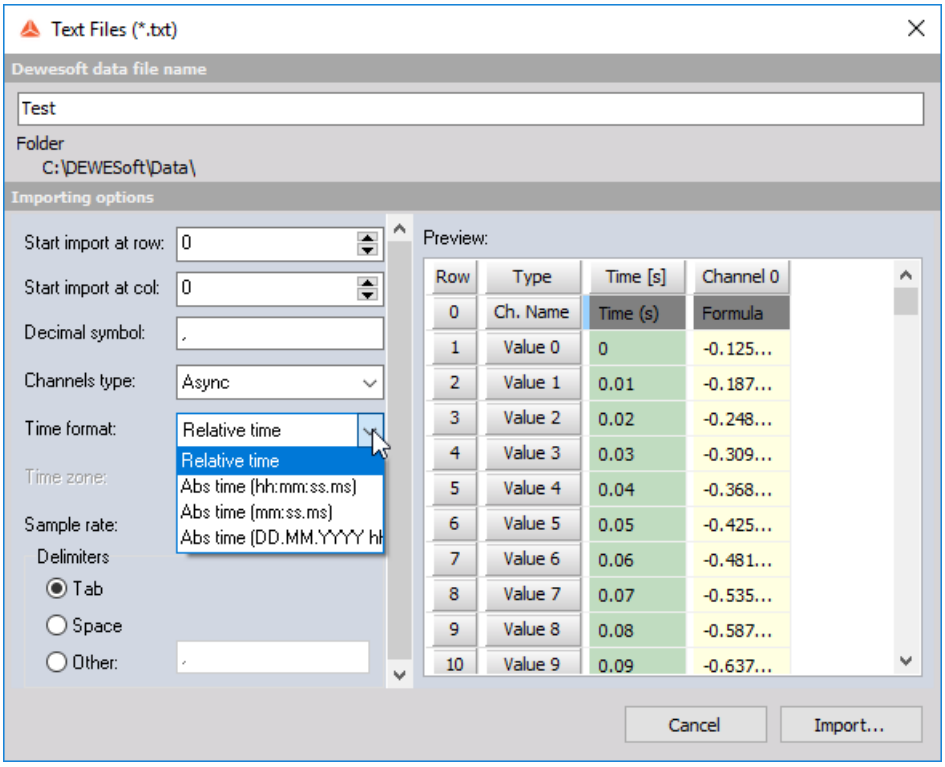
1. You can select at which row/column you want to start importing data from the *.txt file. This is useful when you have some additional descriptions which you don’t want to include.
2. Correctly enter the symbol which represent the decimal symbol in a *.txt file.
3. You can select between the “Sync” and “Async” “Channels type” options:

- If you select the “Sync” option the time stamps will be created according to the sample rate you enter,



- If you select the “Async” option, you need to have the time stamp information in any of four supported time formats:
- relative time,
- absolute time (hh:mm:ss:ms),
- absolute time (mm:ss:ms),
- absolute time (DD.MM.YYYY hh:mm:ss:ms),

In the preview you will see that the "Time stamp" data is presented on green background. You can also select between UTC time and Local time zone. The option “Sample rate” is ignored, when importing async channels.



- The last setting you need to configure is the “Delimiters”, which represent which symbol or character that separates the data in the .txt file. You can choose between two predefined options, or you can enter any other symbol.

If the setting are not set correctly the file won’t be imported and the notification "Can not import *.txt " will appear.

If the data is imported correctly you will see the imported channels in the channel list.

Frequently asked questions

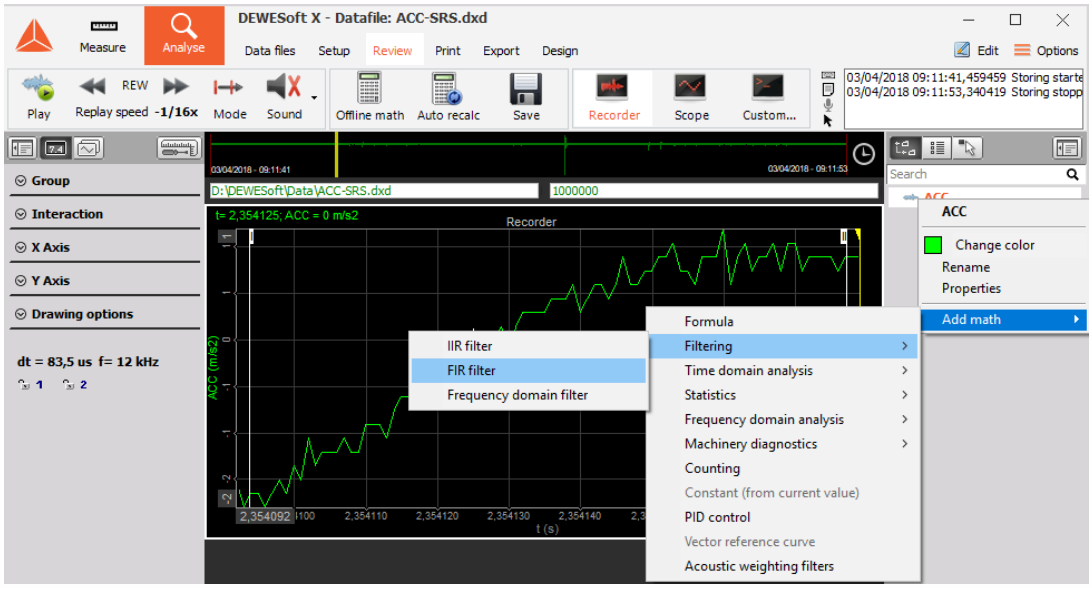
1. Can the “string” data be imported? Files that include string data can be imported but the string values will be ignored if async channel type is selected or replaced by zeros if sync channel type is selected, while all the numerical values will be correctly imported.
2. Why are the last channels missing? Why are my channel name and data mixed? It is important that all the channels have Channel names (Row 0), otherwise the “data” will be assigned to the next available “Channel name”. In this case the number of imported channels will be the same as the number of “Channel names”.

POSTPROCESSING

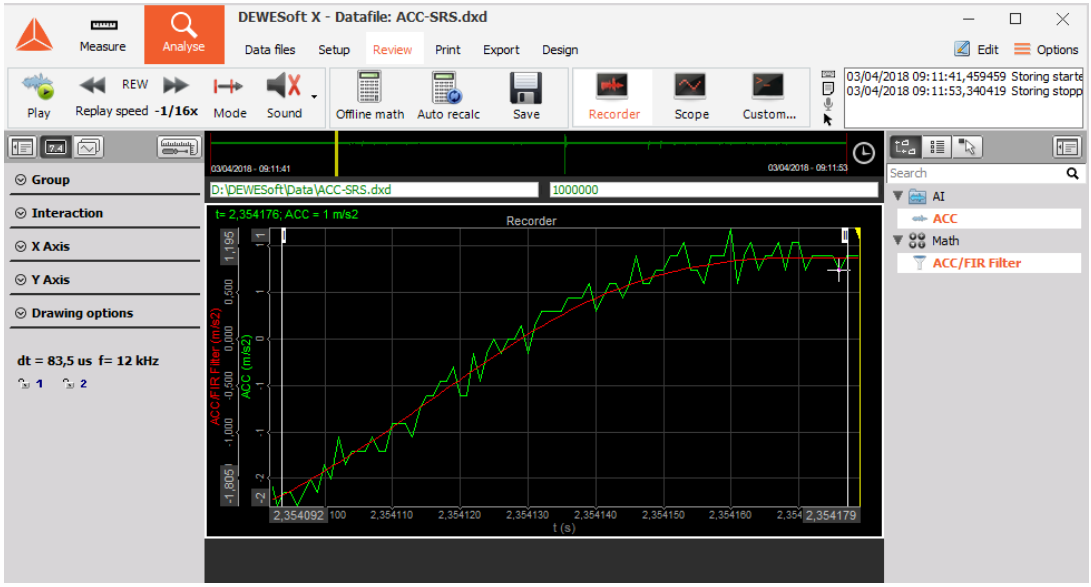
Post processing is a great feature which allows you to:

- add new math channels in analyse mode
- modify existing math channels
- change the scaling of online or offline channels.

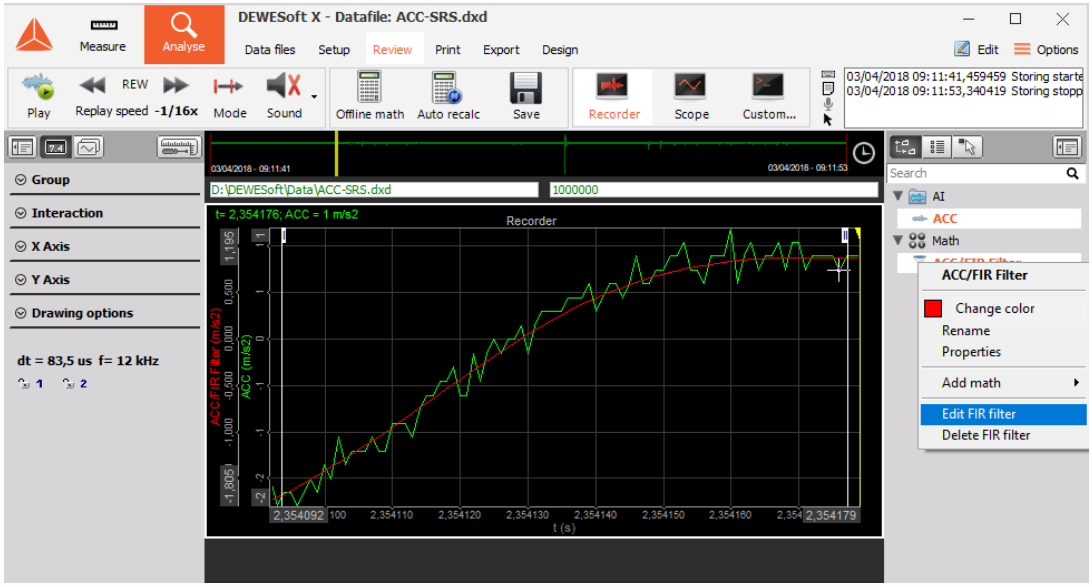
So you could store only the raw data and then in analyse perform all needed mathematics. There are several ways to perform math analysis on already acquired data. Let's start with a simple one. To add a simple math channel like filter you select the channel from channel selector, right click and select Add math and then choose appropriate math.



In this example, an FIR filter was added. When a filter is set, data is automatically recalculated and put into the current display.



Next, you can edit any math (either added during measurement or offline) and modify or delete it by clicking on the math channel and choosing Edit xxx math or Delete xxx math (in this case Edit FIR filter or Delete FIR filter).

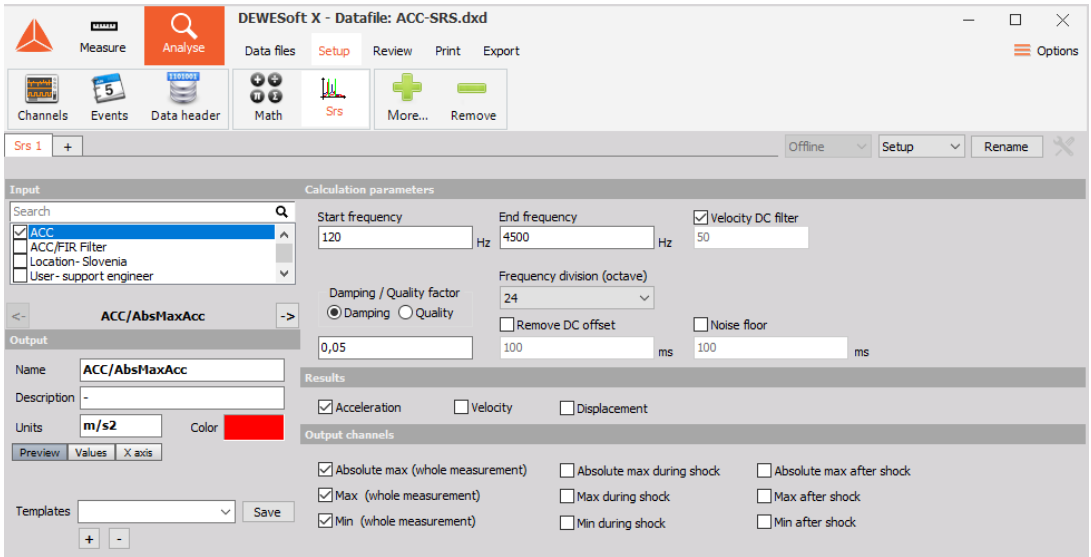


A window of chosen math will pop up, allowing you to change calculation parameters. Upon closing the channels will be automatically recalculated.

NOTE: **Data file will be recalculated only on the chosen time interval. To reduce the time needed to recalculate huge data files, you can zoom in and define all needed calculations on the small area and then zoom out, press Recalculate and get the cup of coffee when Dewesoft recalculates the whole file.**

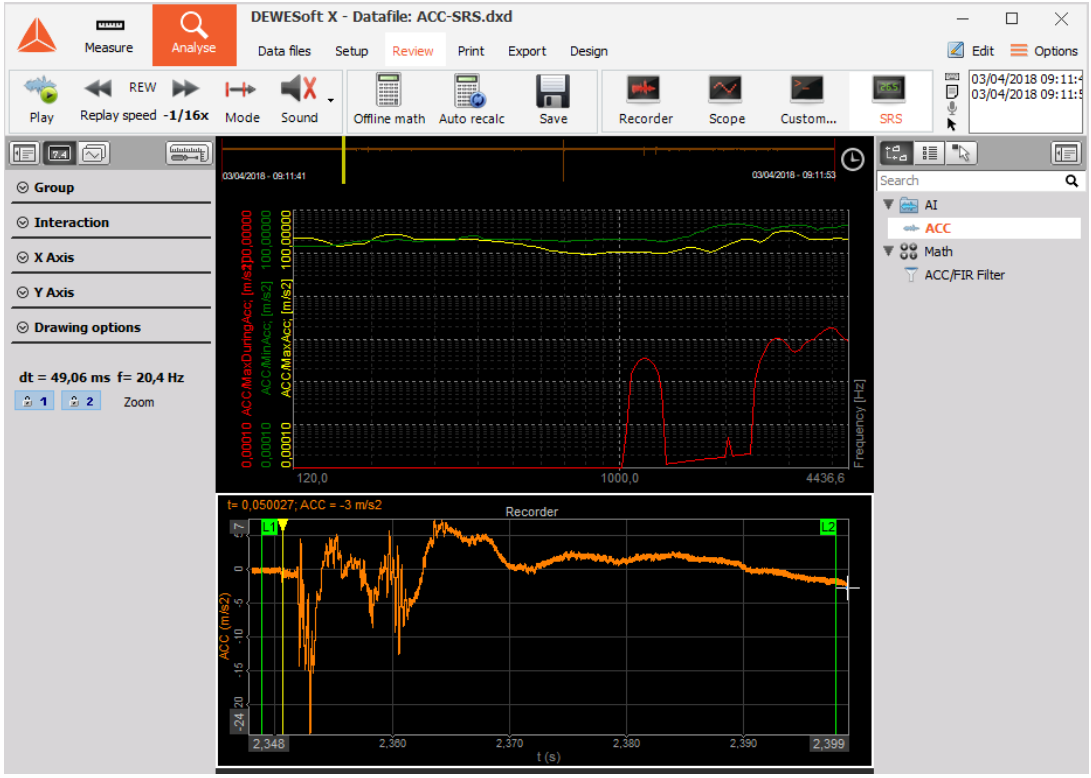
More complicated math can be defined in the math section by pressing Offline math button or Setup button from the top menu. Now you prepare math channels like in Measure -> Setup mode and add advanced analysis like Power, Order tracking, Order analysis and all others.

In our example let's continue to add SRS (shock response spectrum) from the Setup menu.



When finished, go back to review. Please note that Offline math icon changed to Recalculate. SRS is typical offline math (since it requires double pass). Therefore select start and end for recalculation and press Recalculate.

SRS will be calculated. When recalculated, the button changes again to Offline math, notifying that we can go back to math and modify parameters to get perfect results. Please look also at the channel selector topic. You can change the colour, scale and offset of any channel.

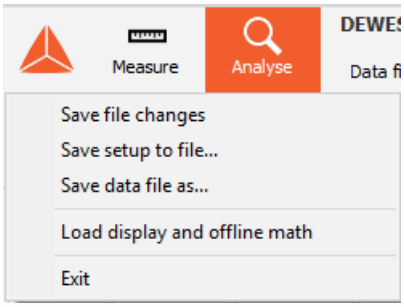


NOTE: **Lengthy calculations can be interrupted with Escape key on the keyboard.**
You can also choose that Dewesoft automatically recalculates the data when zooming in or out. This is, for example, useful when searching for the absolute maximum for a specific time slice. You can save the results of post-processing by pressing Save button. If the data is not calculated or not calculated for the full time of the file, only the setup of all math and display channels will be stored. If the offline math is recalculated for the entire time of the file, also the recalculated results will be stored in the data file.

APPLY ACTION

Apply action on single data file

When you are analyzing the data file, it is simple to apply offline math and displays. In the opened data file click on the Dewesoft logo in the top left corner and use the option Load display and offline math.

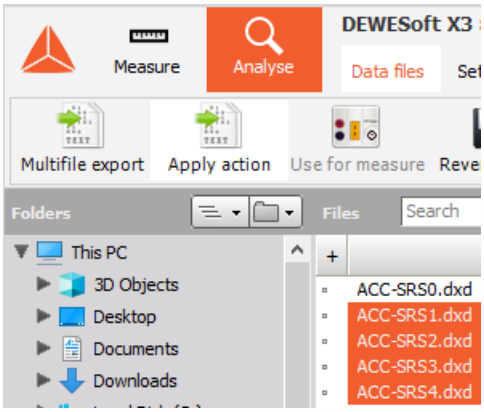


Choose the file from which the information will be imported. After the process is finished recalculate the file.

NOTE: **Dewesoft will import only offline mathematics.**

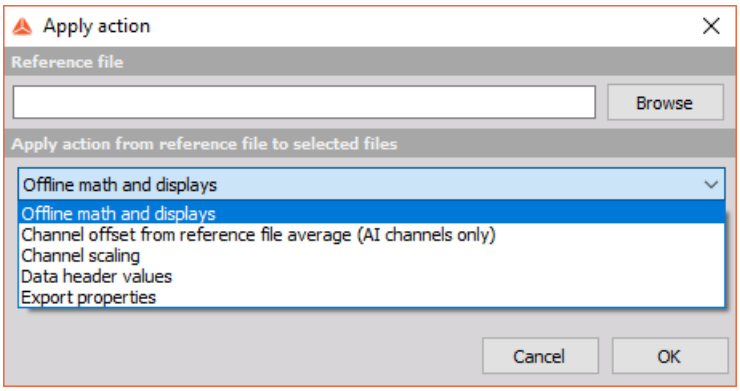
Apply action on multiple files

To use the apply action on multiple data files first select the files on which the change will be applied and click on the apply action button.

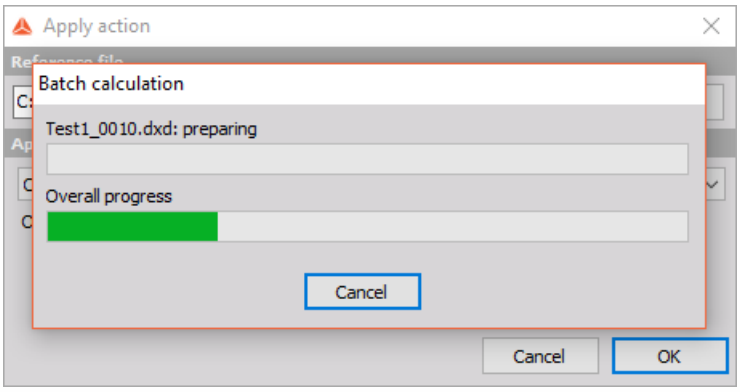


In the pop-up window, first, select the reference file (this is the file from which the information will be imported) and then select the action which you would like to apply.

- **Offline math and displays** - this option will import all the displays and mathematics which have the status set to offline.
- **Channel offset from reference file average** (AI channels only) - Analog input channel offsets in target files will be set to average values of same channels from the reference data file. If reference file holds initial (zero) values, this can be used to zero channels in target files.
- **Channel scaling** - Channel offsets and factors from a reference file will be applied to same channels in target files.
- **Data header values** - Data header variables in target files that are empty or zero, will be overwritten with values from the reference file.
- **Export properties** - Export channel selection and order will be copied from reference file to target files.

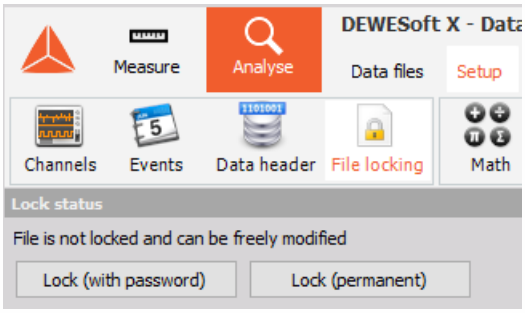


After everything is set, press OK and Dewesoft will start the process.


Apply action
Batch calculation
Test1_0010.dxd: preparing
Overall progress
Cancel
Cancel
OK

FILE LOCKING

You have seen in the Postprocessing that it is possible to alter the data files as we like. Quite often it is required to keep the files intact and lock them for further processing. This can be achieved by opening the file, going to Setup and then choosing File locking. We can lock the files with a password which must be entered next time the file is opened or we can also lock the file permanently.



The file locking can be also enabled in the security tab in settings by default.

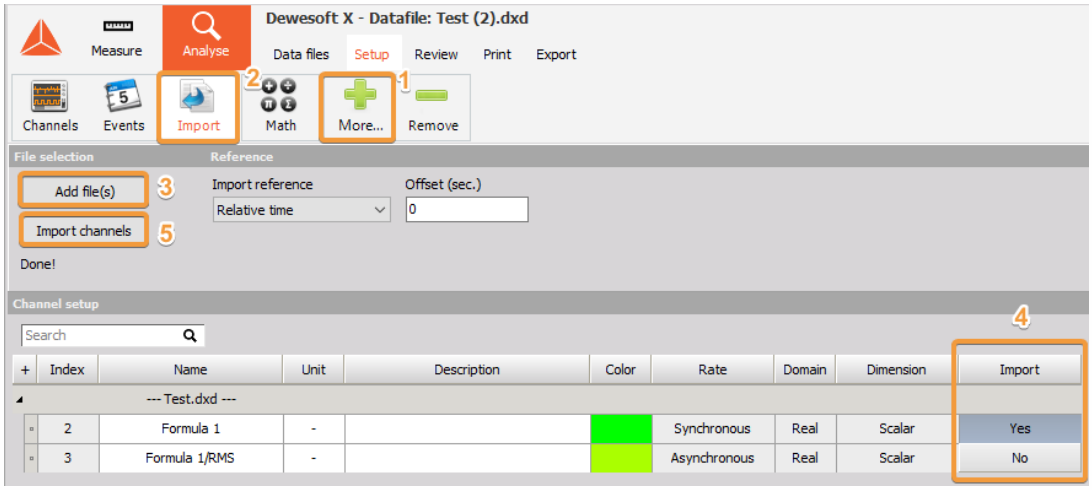
IMPORTING DATA FROM DATA FILES

When channels from one data file have to be merged/compared to another data file, you can use the import functionality, with which we can import selected channels into the current file.

How to import channels

Open a destination data file, then go to Setup and add the Import option by clicking more. click on the Import and all the settings will show up. To add data files, from which you want to import the channels, just click "Add file(s)".

After you add the files, channel selection screen will appear, where you can select the channels you want to import. To select the wanted channels, you have to change the status of "import" to yes. When all desired channels are selected click "Import Channels".



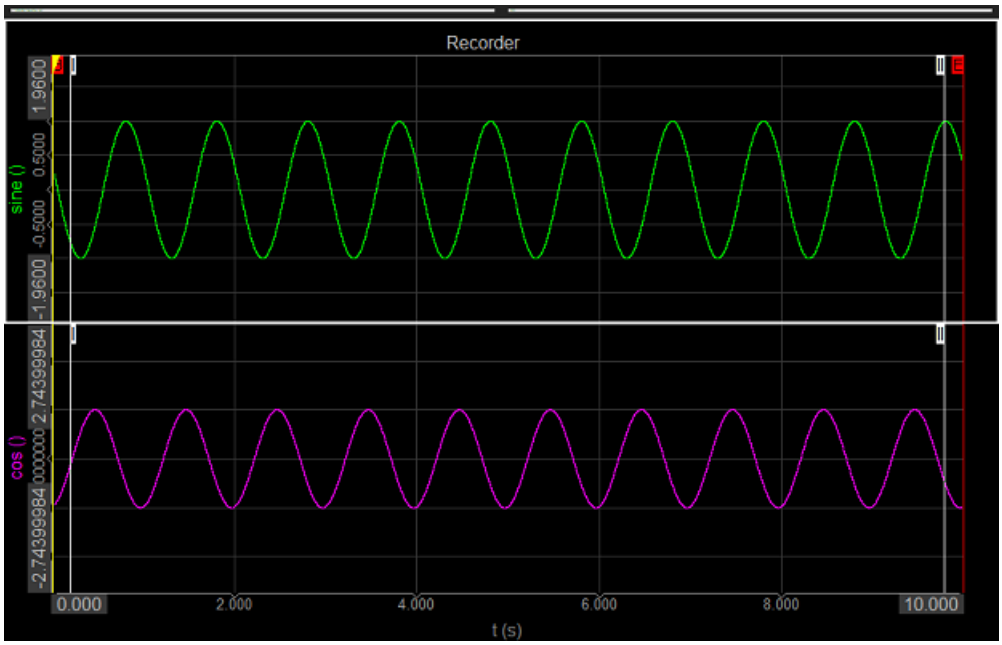
If the import was successful, you will see a message saying "Done!".

In the "Review" tab the imported channels will be added to the channel list into the newly created "Import" folder.

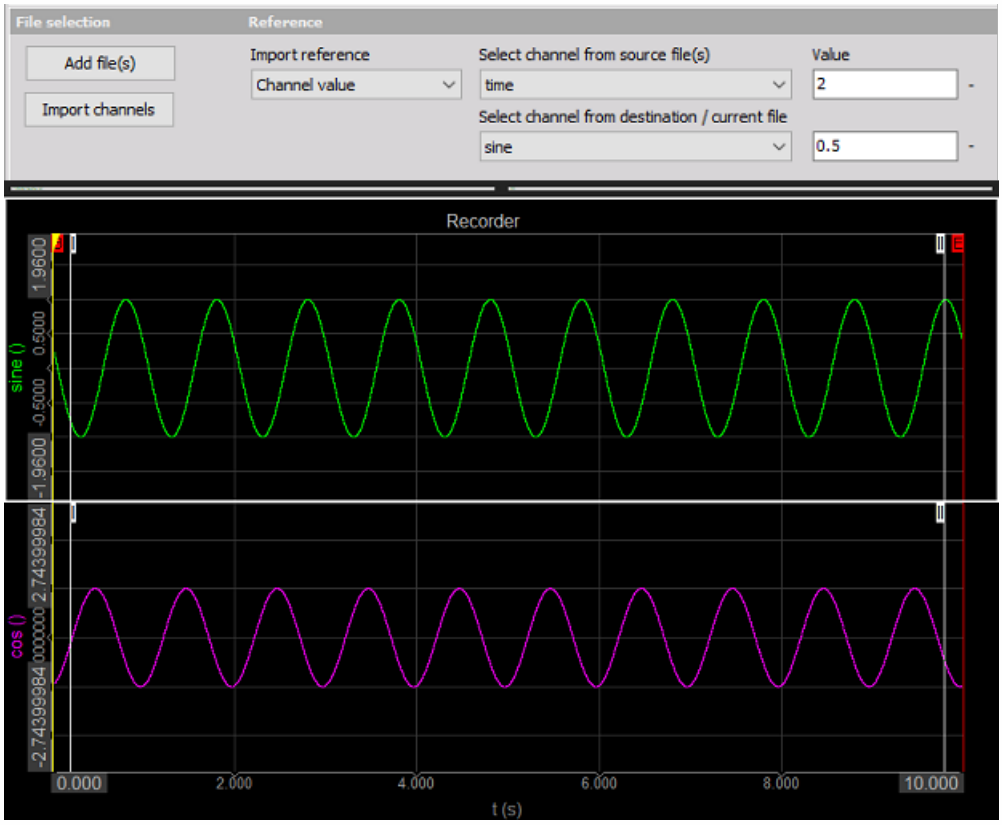
Settings

You can select from 4 different import references:

1. **Relative time** - the reference time for the imported channels will be the start of each data file, starting with 0. If the imported channels don't start at the same time as the original data file, the offset has to be added in order for the signals to be aligned.



2. **Absolute time** - the reference time for the imported channels will be the start of each data file in absolute time format.
3. **First trigger event** - the imported channel will be shown only after or in between trigger events in the original data file.
4. **Channel value** - select the Channel and a value from the source file(s), where you want the imported channel to be aligned with a certain value of the channel from destination/current file.



PUBLISHING DATA

Dewesoft Analyse mode offers following options to publish the data:

- Screen printout: Printing measured data for instrument on all available printers with possibility to set:
- Page orientation
- enter Notes on measurement
- Multipage printout

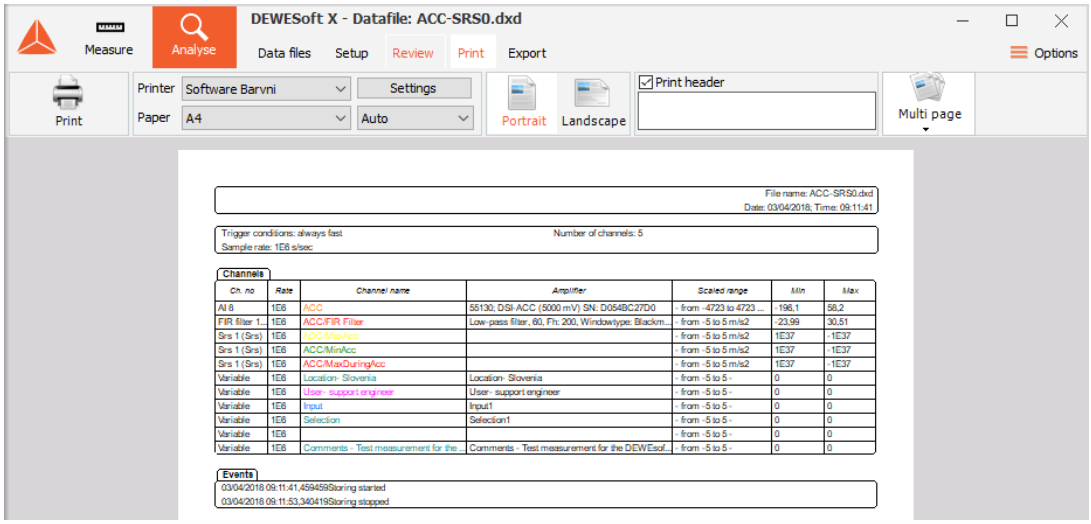
- Copy channel setup: copy to clipboard
- Copy Visual elements: copy to clipboard, than paste this data in other applications.

SCREEN PRINTOUT

In the Analyse mode, Dewesoft offers a printing function.

It is working with any instrument (Overview, Recorder, Scope,...) and also with the setup. To do a printout, just select one instrument, change the appearance as desired and press the Print button.

The following screen will appear:

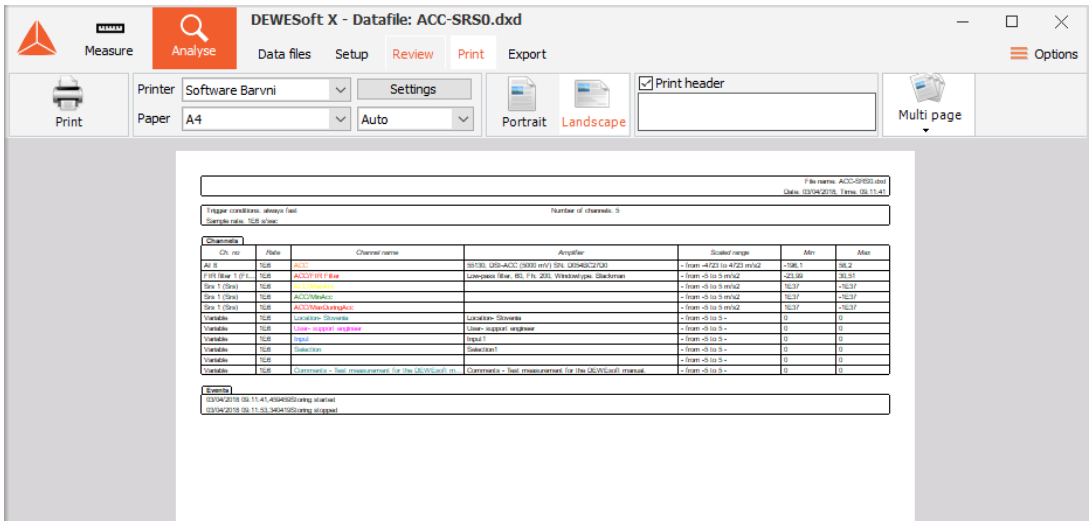


To leave the printout window simply click on any other instrument to change it directly, for example, the recorder.

In PRINT MENU you can set different Print properties:

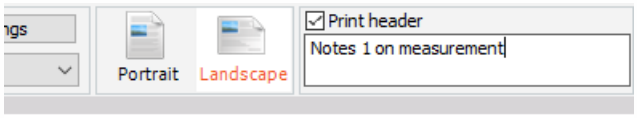
Page orientation

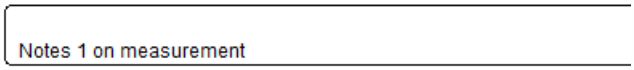
Select between Landscape and Portrait format:



Notes on measurement and page header

In Notes on measurement field you can enter a comment to print with your generated report:



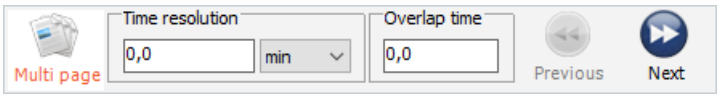


Besides this comment on your page header there will also appear:

- license information,
- file name,
- page/pages number,
- date and time of measurement,
- show time.

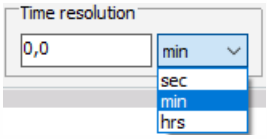
Multi page printout

You can make a printout, spanning over multiple pages by clicking the Multi page icon:



Time resolution

Set **time unit**: sec, min or hrs from the drop-down list and enter **Time resolution**:



Example: **when for measurement with duration 1min 35sec the Time resolution=40sec is entered, multi page printout with 3 pages in all is defined -> the first page for measurement time 0 ÷ 40sec, 2nd for 40 ÷ 80sec and 3rd for 80 ÷ 120sec.**

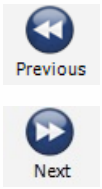
Overlap

The overlap time is the time which will be displayed twice in the printout - on the end of the first page and at the beginning of the next page.

Example: **when for example above the Time scale=15 is entered, multi page printout with 4 pages in all is defined -> the first page for measurement time 0 ÷ 40sec, 2nd for 25 ÷ 65sec, 3rd for 50 ÷ 90sec and 4th for 75 ÷ 115sec.**

Preview

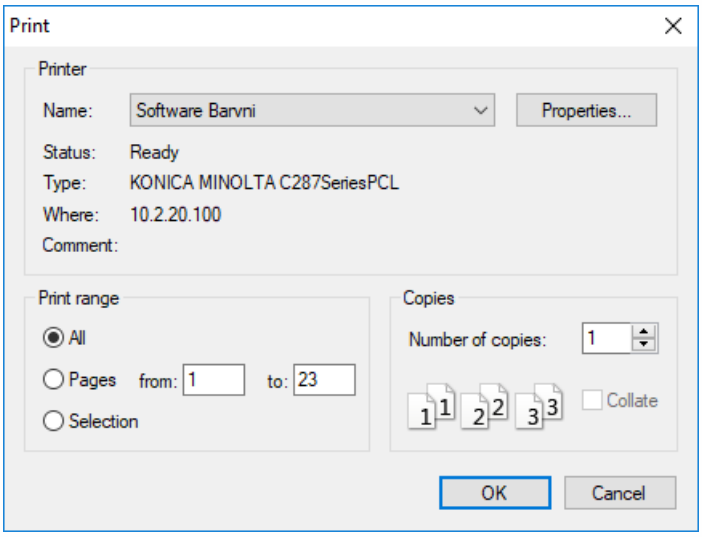
You can display a preview of every page in multi page printout by using the **Next** button for navigation up and the **Previous** button for navigation down between pages.



Printing

After you set all of the desired properties, press the **Print** command button below print menu settings. This will open the standard Windows-> Print window, where you may define your printer properties, print range and copies as in other programs and start the printing job.

Dewesoft uses the complete Windows-> printer support. As a standard, Dewesoft will suggest using your default printer, but you can select between all available printers - even network printer - and also use the printer spooler.

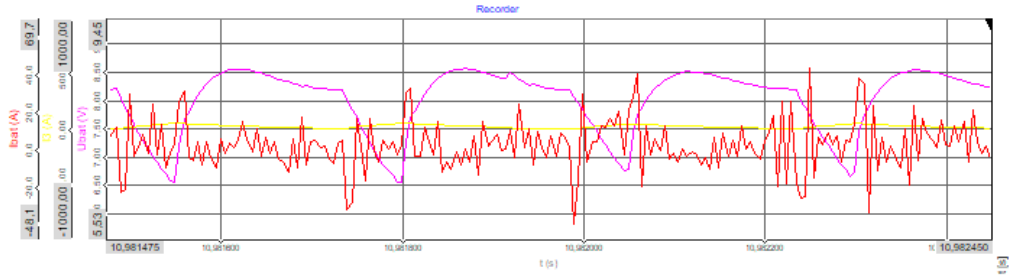


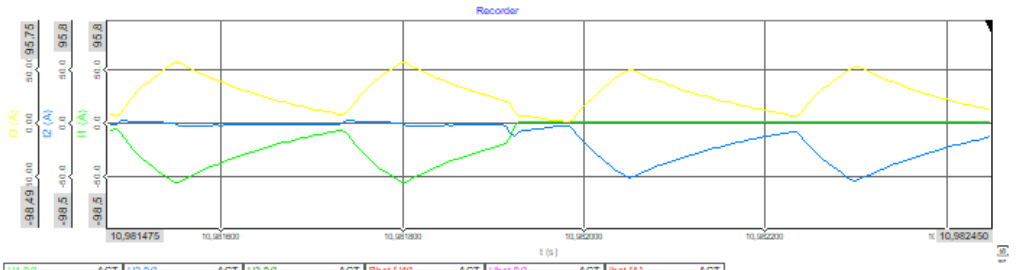
After starting the printout, you can immediately continue working with Dewesoft, even if the printer is still running.

Example: **Typical Setup and Recorder printouts in landscape format:**

EVALUATION LICENSE
Notes 1 on measurement

File name: test3.8..dxd
Date: 08/03/2018; Time: 11:52:15; Show time = 975,0 us f= 1,03 kHz





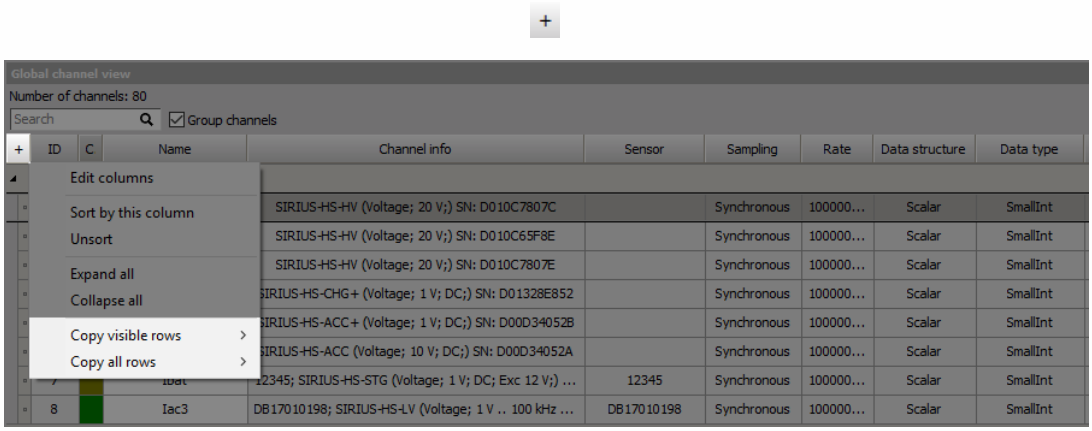
U1 [V] ACT	U2 [V] ACT	U3 [V] ACT	Pbat [-W] ACT	Ubat [V] ACT	Ibat [A] ACT
-0.3	0.5291	-0.3995	57	8.24	-4.9

Unassigned ACT	Inverter efficiency [%]

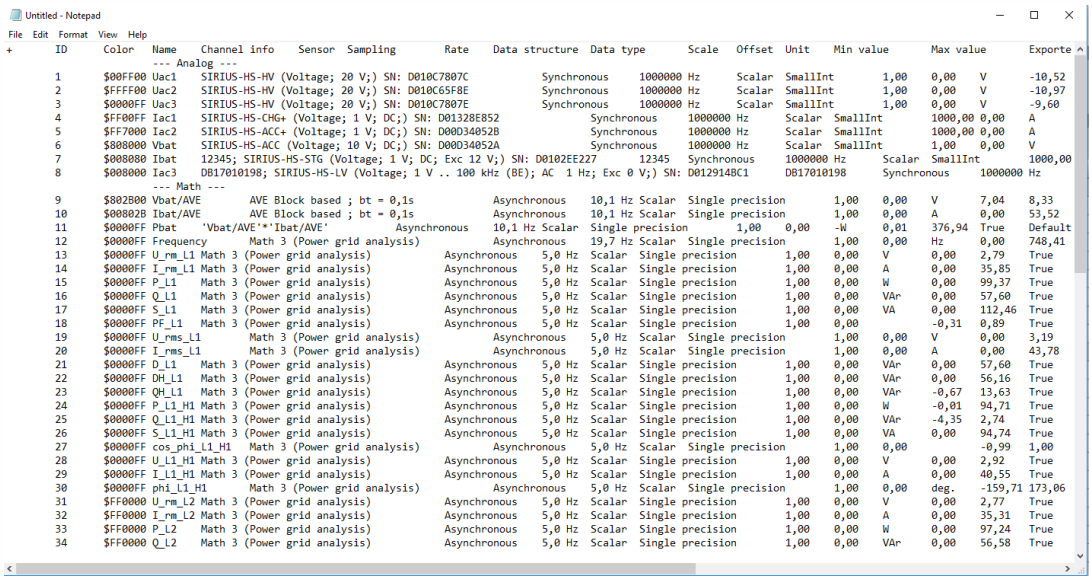
COPY CHANNEL SETUP

If you work with plenty of channels, it would be nice to have them available in other applications, like MS Excel.

In Dewesoft simply change to the Setup and select Settings to get the channel list. Now right-click on the Plus button and select Copy visible rows or Copy all rows from the appearing menu.



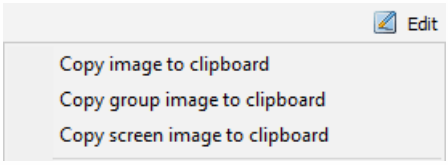
Now open and change to the other application and paste it.



NOTE: **This feature is also available in the settings during file selection.**

COPY VISUAL ELEMENTS

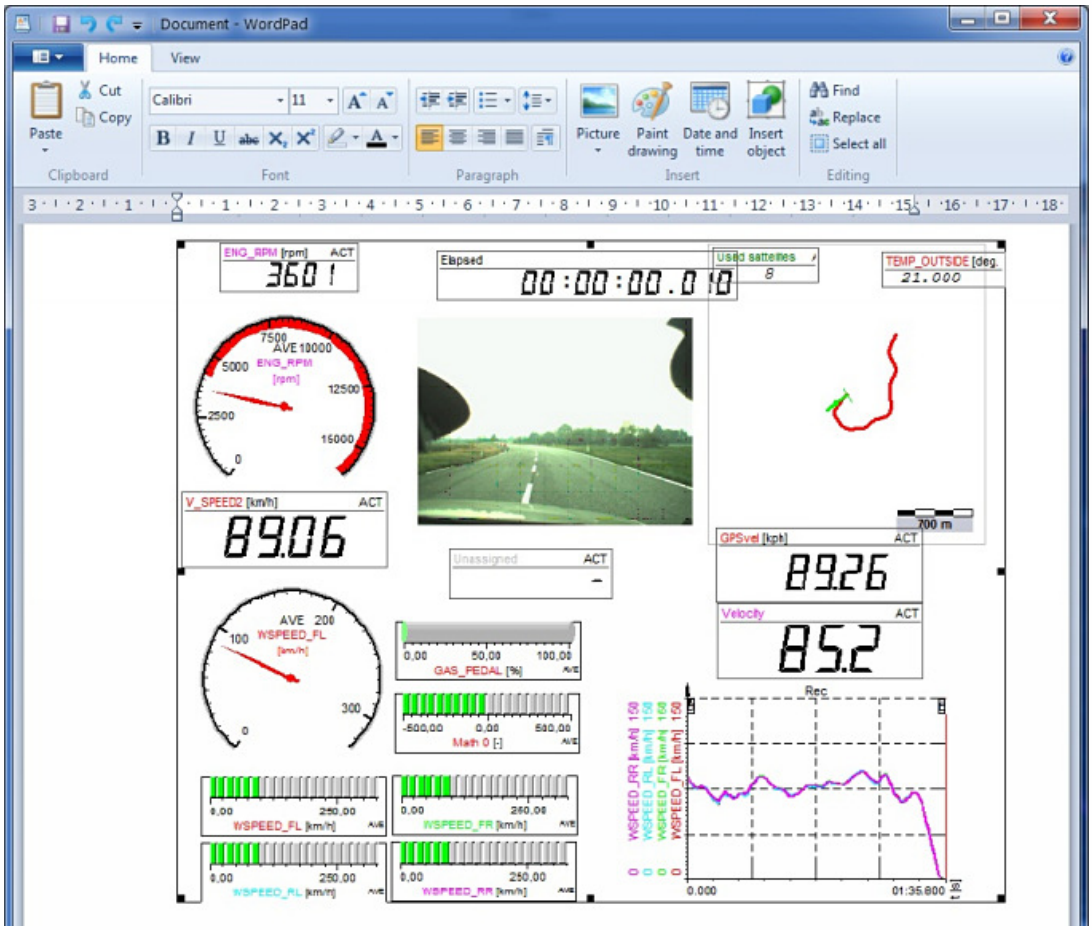
For documentation in external applications, you can also copy the display elements into the clipboard.



Select the desired instrument and select from the Edit menu:

- **Copy image to clipboard** - copies only the selected instrument
- **Copy group image to clipboard** - copies the whole group of selected instrument
- **Copy screen image to clipboard** - copies the whole screen

Example: Screen image copied to the external application



EXPORTING DATA

The important purpose of the Dewesoft Analyse mode is also to export acquired data for off-line analysis using other software.

Export

Dewesoft is intended to be an acquisition package. You can use other post-processing packages for advanced analysis. Data Export is easy, fast, flexible and dynamic with Dewesoft, supporting a wide variety of popular formats which makes data files highly transportable that can be imported into virtually any analysis program. To enable efficient export data process perform the following procedures:

1. STEP

Set export data properties: to define the type of data with Data export and determine Time axis; Time range of data can be selected for export.

2. STEP

Exported channels: select the channels to export from Channels list; not available for Dewesoft export option.

3. STEP

Export option: selects another software application as the target for data export and off-line analysis:

- Microsoft Excel
- FlexPro
- File Export
- Dewesoft
- Clipboard

4. STEP

Template management: Creating and changing template - scripts which are listed on central Postprocessing in FlexPro / MS Excel part of screen (only available for Microsoft Excel and FlexPro export option)

5. STEP

Perform export data: after all settings are done, select Export data button to export data

You also have two special procedures to:

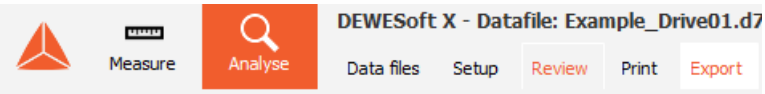
- Export multiple files
- Export instrument display to Video

Time range of data

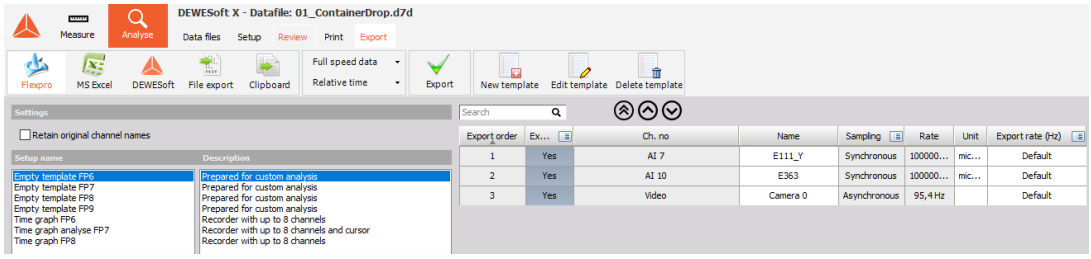
To export only part of acquired data select time range on the recorder or vertical recorder with zooming in the portion of the data.

HELP: **For detailed information about Time range selection see -> User Guide ->Selecting data)**
NOTE: **Be aware that only selected portion of data will be exported. To export full range of data, zoom out first to entire length of the data.**

To export data, press the Export button on main Dewesoft toolbar, which is visible only when you are in the Analyse mode:



Once you press Export, the following screen will appear, ready as standard to export data directly into FlexPro:



Set export data properties

Data export

Select type of data from Data export the drop-down list:

Real data

- Full speed data
- Reduced - When using reduced data, you can select diferent parameters by checking the appropriate box: Min, Max, RMS and/or Average type.

Time axis

Select type of time scale from the Time axis drop-down list:

- Relative, Absolute or Trigger type.
- Relative axis will start with time 0 at the beginning of the measurement.
- Absolute axis will export the absolute date and time of measurement as the time axis.

Trigger time axis will start with time 0 at the trigger point (pre-trigger will show negative time values).

Exported channels

Next you have to select the channels to export from displayed Channels list (at the bottom of the window).

Selection can be made in two ways:

1. Click on the icon in the Exported column to select from the displayed menu:

- Select all channels
- Deselect all channels
- Invert selection

2. Click Yes / No button in the Exported column to toggle between this choices to export / don't export a particular channel.

As a standard, all channels will be exported.

Channel can be moved with click on button up and on button down within Channels list. This will change the export order.

The export list options will be stored when we choose Store settings and events from the main Data menu.

Export option - formats

Dewesoft supports the following data export:

- Microsoft Excel - direct data transfer via ActiveX
- FlexPro - direct data transfer via ActiveX
- File Export - several file formats
- Dewesoft - stores selected area in new Dewesoft file; used to cut files
- Clipboard - copy data to Windows Clipboard, allows pasting into another running application

Simply click on the desired application to select it.

We strongly recommend FlexPro, a third-party application that is easy to learn and use, and yet extremely powerful. Dewesoft offers easy export to this format, and advanced automation when data is exported to this program, including the ability to run custom macro 'scripts' inside.

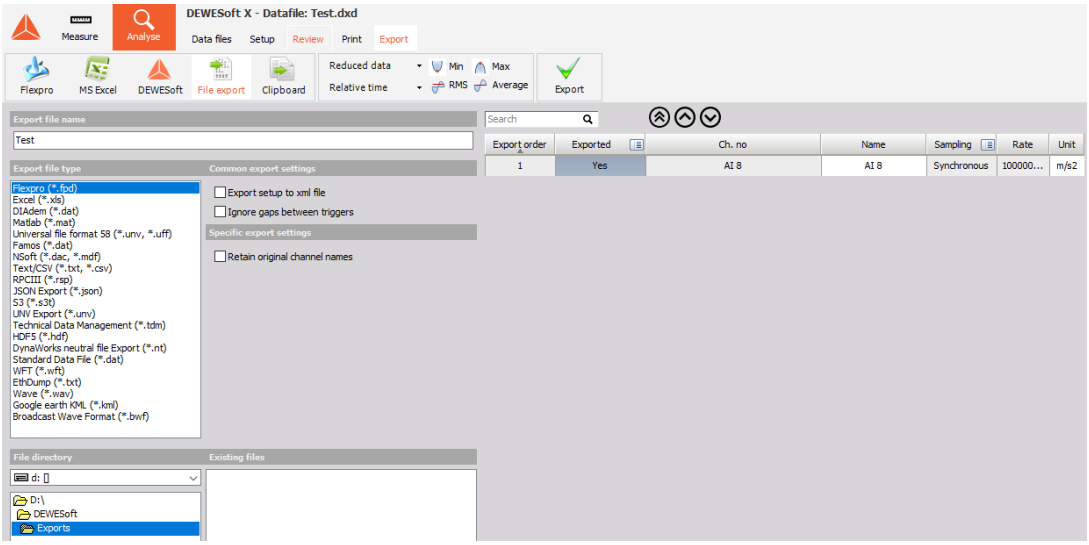
The other file formats except FlexPro and MS Excel just prompt you for a filename, which you supply using a standard Windows file dialog box. Click OK and the software will do the rest. Be sure to supply the proper filename extension.

File export

The supported data file formats are:

- FlexPro [*.fpd] - Powerful, easy-to-use data analysis software
- Microsoft Excel [*.xls] - Standard spreadsheet software (not useful for large amounts of data)
- DIAdem [*.dat] - Powerful data analysis package for automotive industry
- Matlab [*.mat] - Common analysis and mathematics package
- **Universal file format 58** [*.unv] - For import to 3D modeling and structural analysis software from different vendors
- FAMOS [*.dat] - FAMOS file format export
- **NSoft time series** [*.dac] NSoft file format
- Text File [*.txt] - Delimited ASCII text file
- **Sony** [*.log] - Sony DAT recorder data format
- RPCIII [*.rsp] - RPC III data format used for road load data analysis
- **Comtrade** [*.cfg] - Comtrade data format for power analysis
- **Wave** [*.wav] - Wave audio data format (available as custom export)
- **BWF** [*.bwf] - Multi channel audio data format (available as custom export)
- **ATI** [*.ati] - For direct import in iDEAS analysis package (available as custom export)
- **SDF** [*.dat] - For direct import in Prosig analysis package (available as custom export)
- **WFT** [*.wft] Nicolet file format (available as custom export)
- **Google earth** [*.kml] Export of GPS path to Google earth (available as custom export)
- **CAN messages** [*.csv] - tab delimited export of CAN messages for replay in demo mode

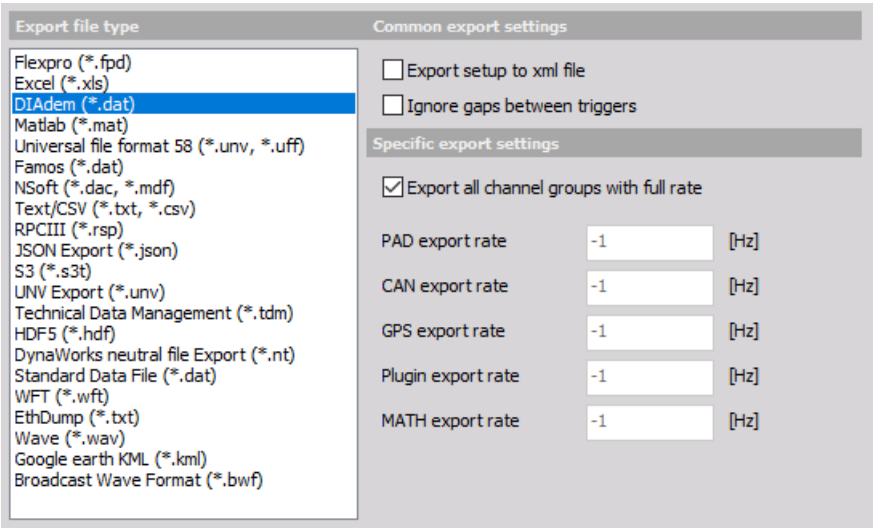
If you select the File Export, following display appears:



On central part following Export to different file formats sections is displayed:

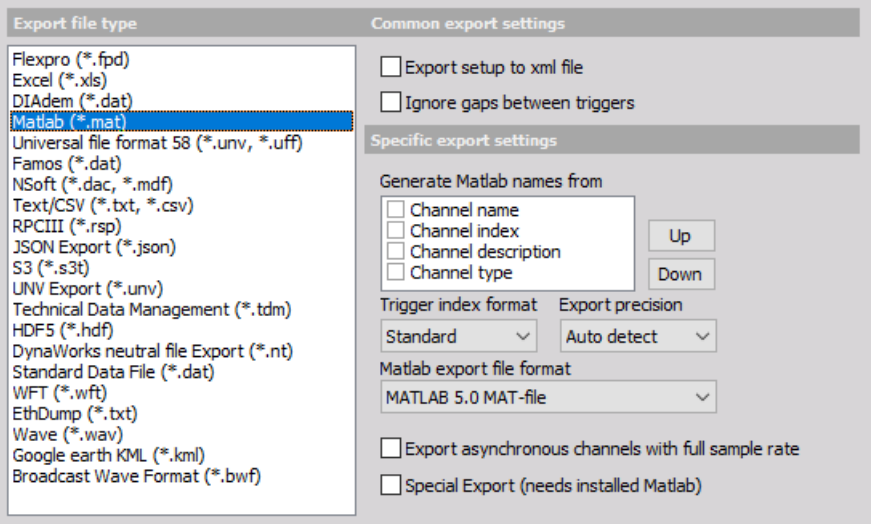
- **Export file name** - name of exported file; enter name according to OS name convention
- **File directory** - name of directory where exported file to be saved; select with double click from drop down list
- **Existing files** - list of existing files same type in selected directory
- **Export file type** - list of available type exported file with optional file settings for:

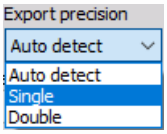
DIAdem *.dat file type



Diadem has an option to export certain groups of channels at specific sample rate, so the resulting data file is shorter. You can achieve this by deselecting Export all channel groups with full rate and defining the rate for each channel group (PAD, CAN, GPS, Plugin and MATH). If -1 is entered, the data will be exported at full rate.

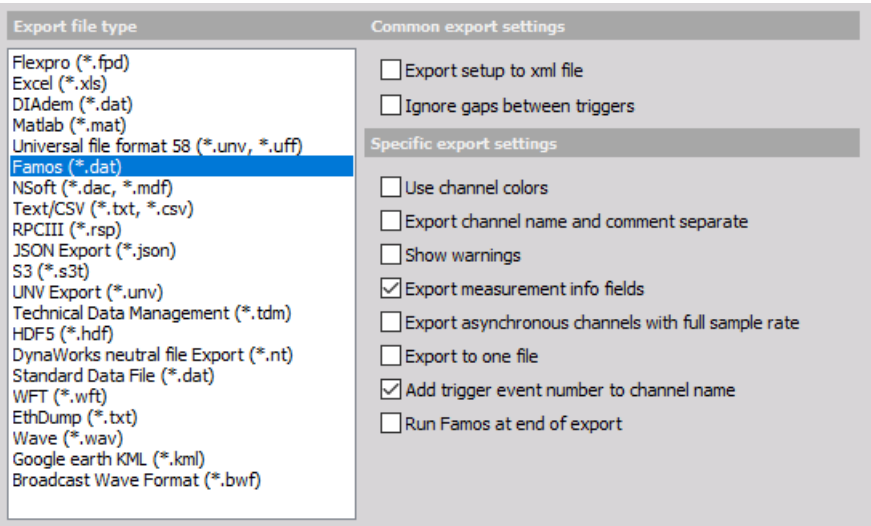
Matlab *.mat file type





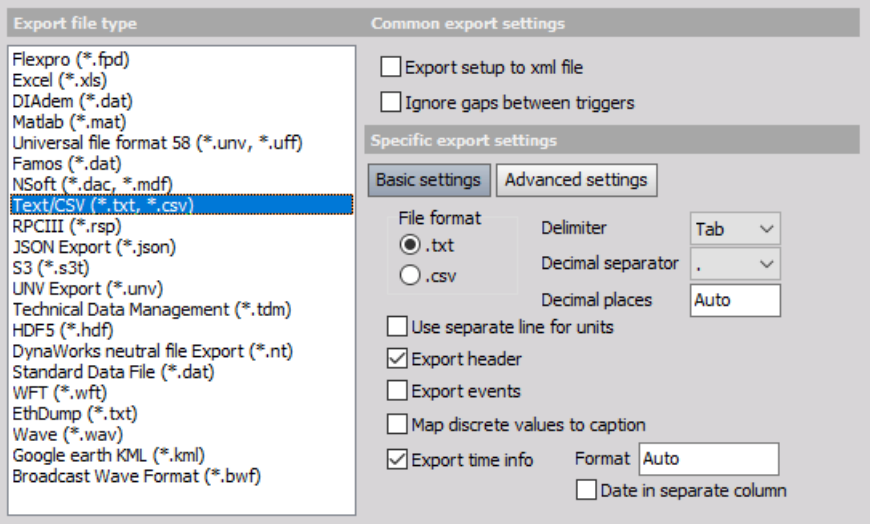
You can select Export precision from drop down list between Single and Double. Usually single precision is enough, but sometimes Matlab functions expect data in double precision. Please be aware that the amount of exported data will be doubled if this option is chosen and Matlab files can't exceed 2 GB except if you chose MATLAB 7.3 MAT-file format. Matlab can also export each channel with separate time channel or exports all data with full speed if Export synchronous channels with full sample rate check box is checked.

FAMOS *.dat file type



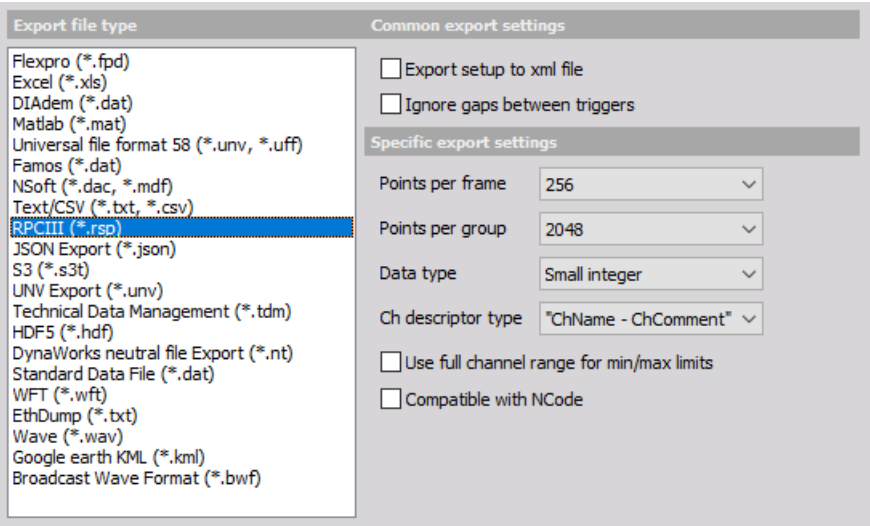
Use channel colors can be checked to use Dewesoft channel colors otherwise default Famos color is chosen.

Text *.txt file type



With checked Export events also events will be exported in text file.

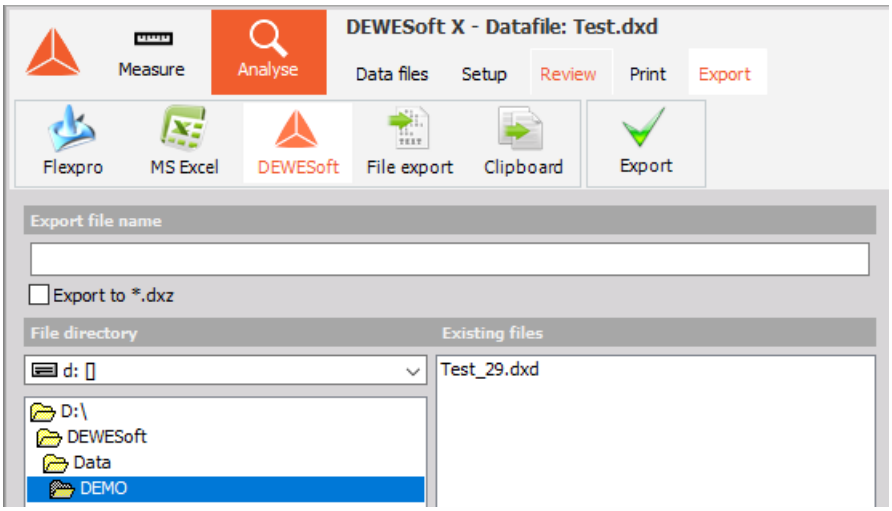
RPCIII *.rsp file type



Buffer length field can be set to match the analysis capabilities. With large files larger buffer length is recommended. The buffer length should be multiple of 2, i.e. 256, 512, 1024 and so on.

Dewesoft export

Stores selected time range in new Dewesoft file; this procedure is primary used to cut files.



By this option on central Export to Dewesoft part of screen are displayed:

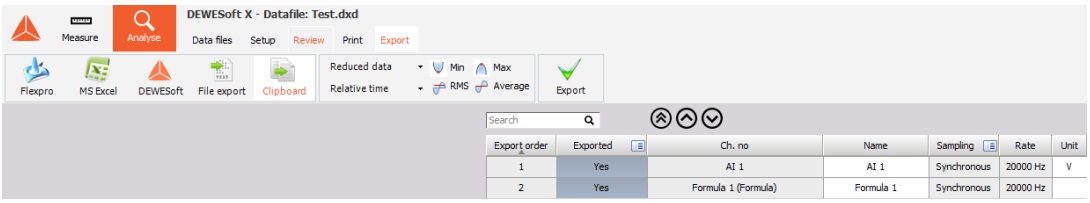
- **Export file name** - name of exported file; enter new file name according to OS name convention
- **File directory** - name of directory where exported file to be saved; select with double click from drop down list
- **Existing files** - list of existing files same type in selected directory

When Export to Dewesoft is selected, Channels list disappear, therefore all channels will be exported.

After you select time range on the recorder or vertical recorder display, enter Export file name and selecExport data button, only acquired data for selected range (and for all channels) will be exported to new file.

Copy to Clipboard export

Copy data to Windows Clipboard and with them allows pasting data into another running application.



By this option the channels to export can be selected from displayed Channels list (at the bottom of the window).

TO FLEXPRO AND MS EXCEL

To enhance export data to Microsoft Excel and FlexPro, the following procedures can be used:

1. STEP:

Select application - click on either FlexPro (ActiveX) or MS Excel (ActiveX) on Export option

2. STEP:

Template Management - Template Management controls are only available for Microsoft Excel and FlexPro export option to manage template - scripts, which are listed on central Post-processing in FlexPro / MS Excel part of screen; templates can be created and changed or deleted |

3. STEP:

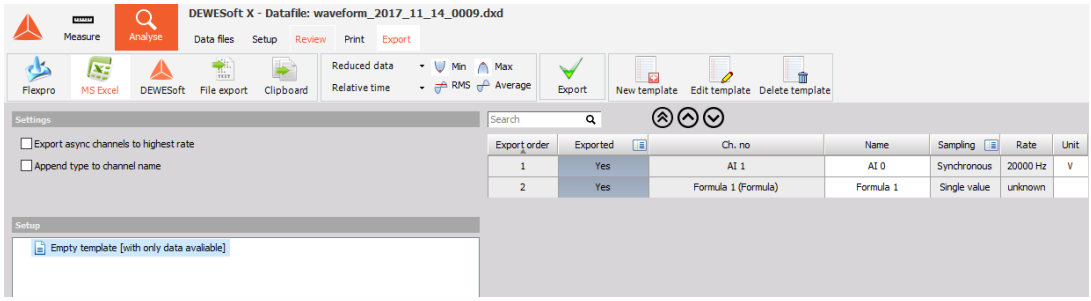
Select template (script) - appropriate script should be selected from the list of available scripts

4. STEP:

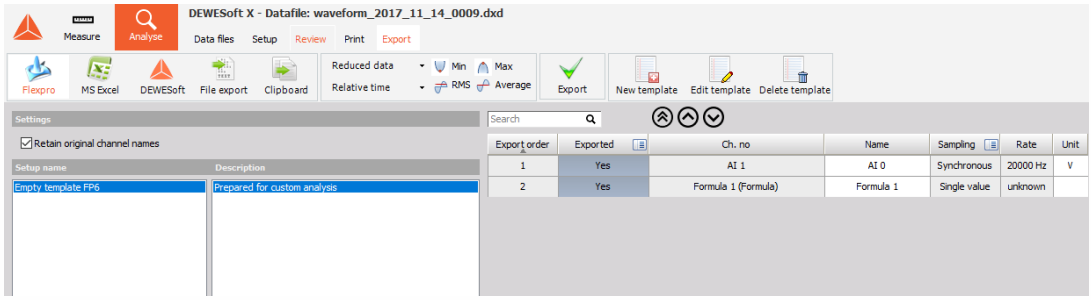
Import FlexPro-Databases - MergeDB script allows the user to import existing FlexPro-Databases to the exported measurement data

If you click on either FlexPro (ActiveX) or MS Excel (ActiveX) on Export option, additional Template management controls in left bottom part of screen and list of available templates - scripts on central Post-processing in FlexPro / MS Excel part of screen are displayed:

Microsoft Excel



FlexPro



For both Microsoft Excel and FlexPro export option you will see that a list of pre-formatted scripts appears in the center of the screen with two columns Setup name and Description of script. Now you should select from the list of available scripts. You can select any one of them and then click

the Export Data button below the format icons to perform the exporting. Don't forget to select the channels to be exported before you press the Export Data button.

Transportable data file

To export to a transportable file which can be moved to another computer running MS Excel and FlexPro, or your favorite other analysis software, use the File export button and from list select Text *.txt file type to make a delimited ASCII text file.

Template Management

FlexPro and Excel scripts for Data export

Experienced MS Excel and FlexPro users can examine the preformatted scripts and macros that are loaded in the <Dewesoft app folder>/Scripts subdirectory, and learn how to create their own scripts for export.

NOTE: **The available scripts depend on the Dewesoft version and may change.**
NOTE: **Please note that FlexPro and Excel export REQUIRE that these applications be installed properly already on this computer, or else the exporting will fail.**

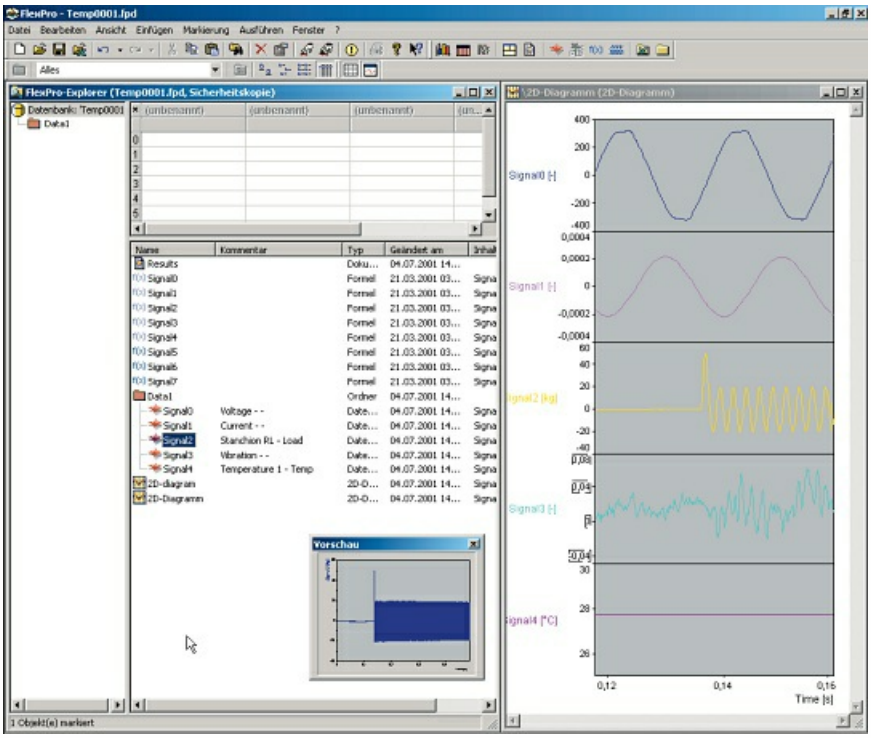
The concept is a nice one: once the script is created, just load any data file and run the script. What happens next is a big time-saver. Dewesoft runs this script:

- automatically starting the other application (you will see a notice on the screen that it is starting the other application, and there will be a slight delay as it does so and establishes communication)
- creates the data-set and
- runs the script.

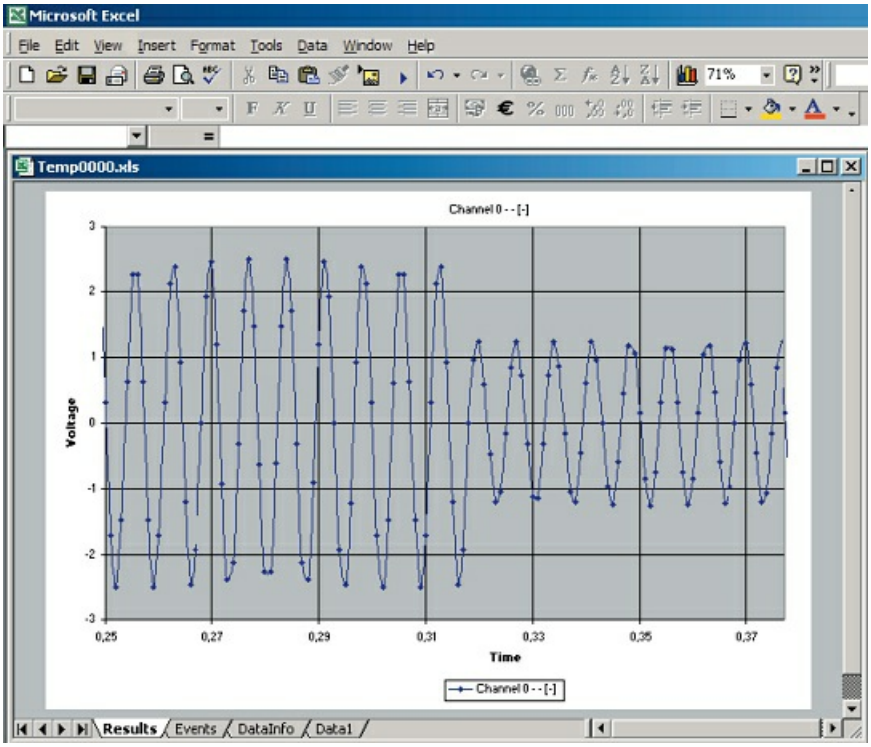
Let‘s say that you have a script that takes four incoming signals, runs 3rd octave analysis on them, creates a polished looking report, complete with your company‘s title bar and logo, automatically puts in the data and other variables from the data file, and formats it for your color printer. After you click the Export button and this process runs, you will see your final report on the screen, and just need to click print to complete your report!

This is the power of "hot script" ActiveX data exporting from Dewesoft to either MS Excel or FlexPro.

Typical FlexPro screen:



Typical MS Excel screen:



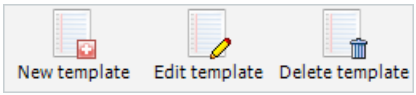
NOTE: **Our MS Excel export function knows about Excel's built-in limitation of 65536 rows per worksheet. If your data is longer than that, it simply creates multiple-worksheets within the workbook that it creates! It also makes a simple chart with the first several thousand data points for each input channel shown. You can use all of MS Excel built-in tools to change the chart format and make additional calculations**

and graphical outputs. But please be aware that MS Excel is not intended to handle million of data points.

Template management controls

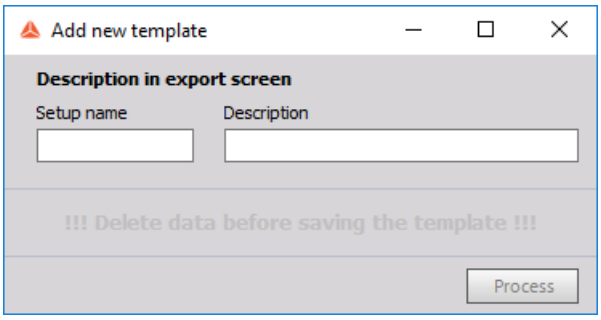
You can easily create your own scripts in **MS Excel** or **FlexPro**, or edit the existing ones.

Just press the New template button to create a new one or Edit template to change an existing script.



To delete an existing script simply select it and press Delete template button.

When you press the New template button, you have to enter a Setup name, Description and a comment for the new script:



When you confirm your entries click on Process button or select Edit button to change an existing script, MS Excel or FlexPro will automatically start with currently selected template.

NOTE: **For more details about creating scripts please refer to the original MS Excel or FlexPro documentation.**

MERGEDB - FLEXPRO

The MergeDB script allows you to import existing FlexPro Databases - (for example from previous Dewesoft-to FlexPro exports) to the exported measurement data.

NOTE: MergeDB requires either FlexPro-Standard or FlexPro-Professional!

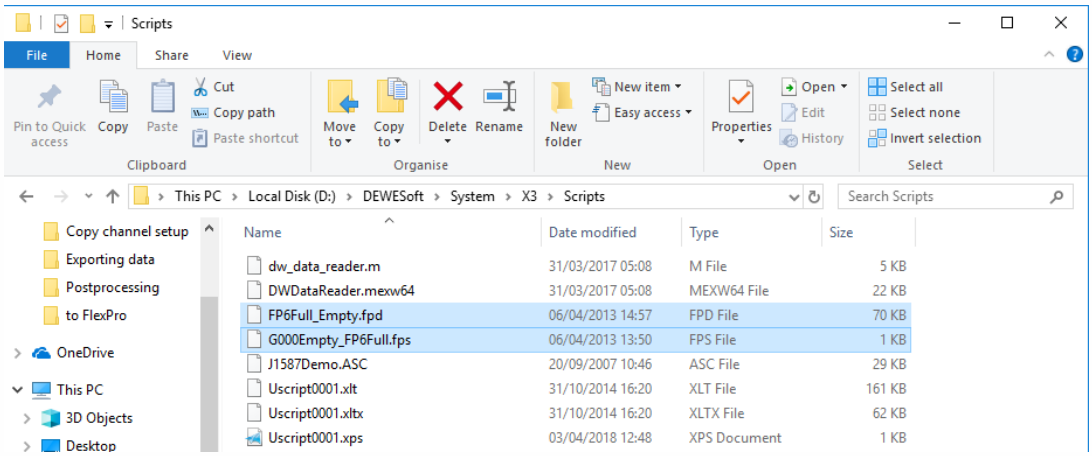
Installation

To make the MergeDB-Script available in the Dewesoft export screen, two files must be in the Dewesoft script folder (Dewesoft addp folder>\Scripts\):

- Copy the file FP7Full_MergeDB_v1.fpd to the Dewesoft script folder
- Copy the file Uscript000X.fps to the Dewesoft script folder

NOTE: There are several Uscript-files in the Dewesoft script folder (e.g.: Uscript0001.fps, Uscript0002.fps ...). The X in the filename Uscript000X.fps must be replaced by a number that is not assigned yet! For example if the last file is called Uscript0002.fps, name the created file Uscript0003.fps!

Dewesoft script folder with the files needed for MergeDB:



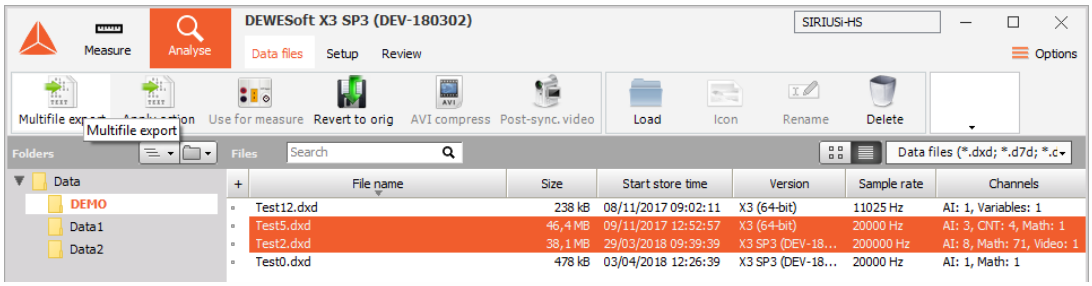
Usage of MergeDB

After you select MergeDB-Script on the Dewesoft export screen the Merge Database window will appear:

- Click on the Browse button to select the FlexPro-Database to be imported.
- Optionally a folder name (Destination folder) can be specified in which the imported files are copied.
- Click on Import to import the selected Database to the Destination folder.
- Click on Done when all Databases have been imported.

EXPORT MULTIPLE FILES

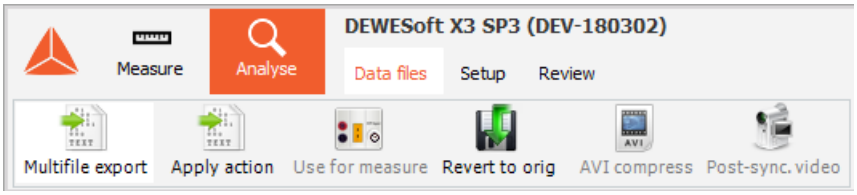
If you want to export several files at once, there is a nice function available. Press the Analyse button to enter the file explorer. Select all files you want to export.



How can we mark several files? There are two possibilities:

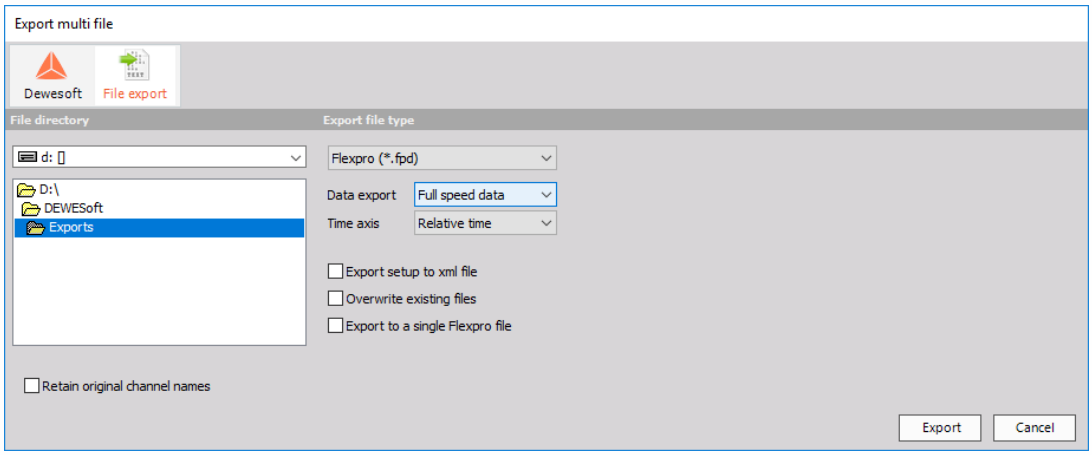
- to mark several files which are listed near to each other, keep the SHIFT button pressed during selection
- to mark several files picked out of the list, keep the Ctrl button pressed during selection

After all files to export are marked, select Multifile export from the Data menu.

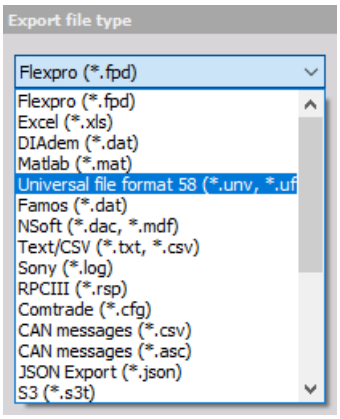


Now the Export multifile window appears. This window allows you to select:

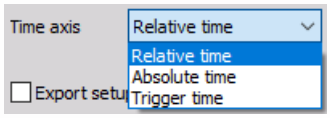
- device and the directory - on File directory part of windows select destination of exported data



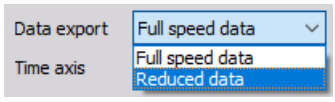
- export format (any already described format) from Export file type list:

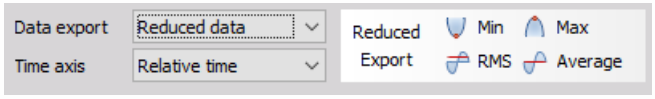


- time format: Relative, Absolute or Trigger type from the Time axis drop-down list:



- export type: Real data, Reduced from Data export list; for Reduced type of data can be selected Min, Max, RMS and/or Average type by checking appropriate box:





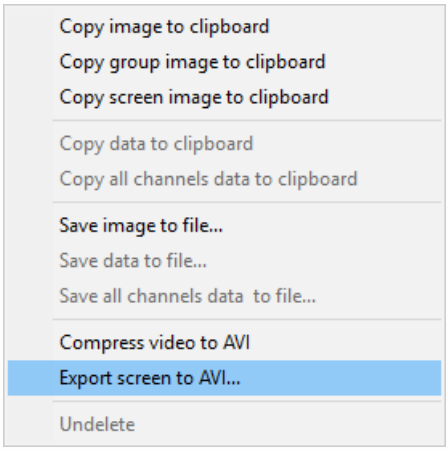
Press the Export button to start the export process or Cancel to leave without exporting the files.

NOTE: **The export keeps the original file names for the exported files.**

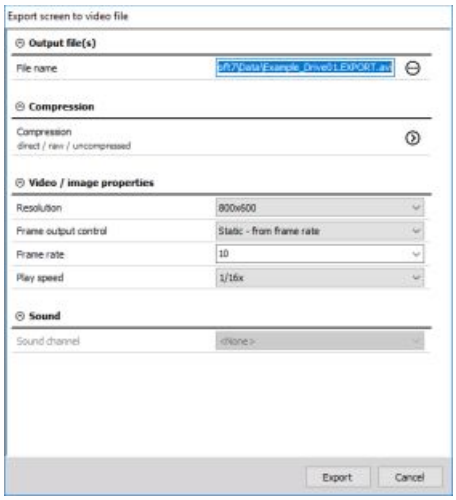
EXPORT INSTRUMENT DISPLAY TO VIDEO

A new way of presenting data is the multimedia document. Dewesoft offers the possibility to export any instrument screen like Overview, Scope, Recorder... to a video, which can be replayed in any video player like for example the Microsoft Media Player.

Select Export screen to AVI from the Edit menu



and the following Export screen to AVI file window will appear:

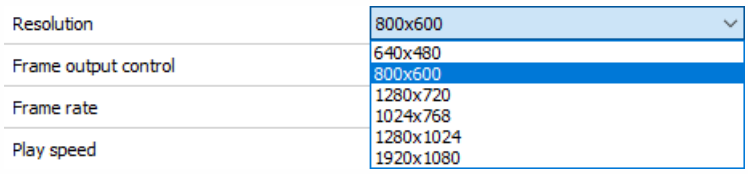


Please change the export settings on this window according to your requirements. The following settings can be changed:

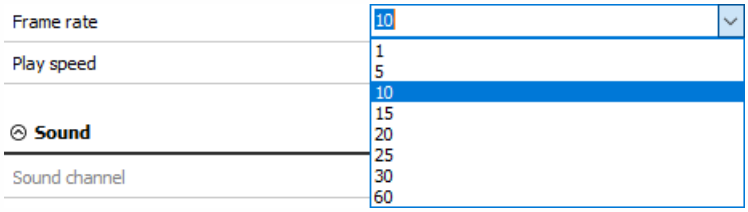
File name and storage path: after pressing the **Browse button**

Save As standard window appear to let you select folder and file name.

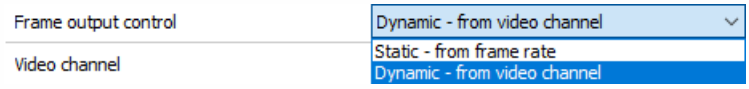
Resolution: from 640 x 480 to 1280 x 1024 selectable from the drop-down list - high resolutions will create larger files!



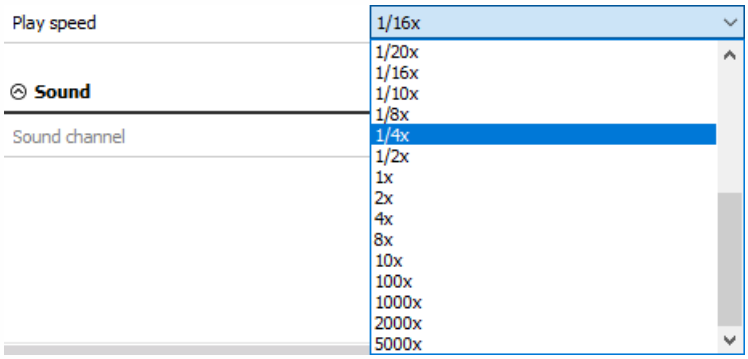
Frame rate: from 1 to 30 - standard video player cannot handle faster data; select from the drop-down list.



Frame output control: select from drop down list Static - form frame rate or Dynamic - from video channel.

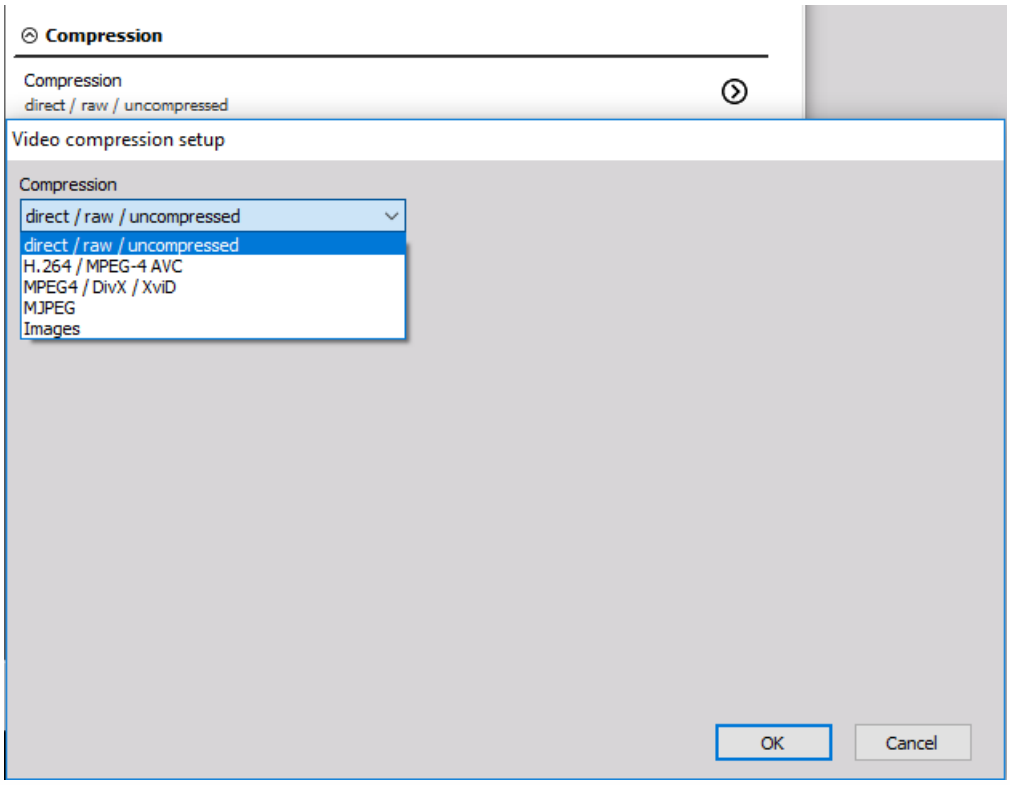


Play speed: perfect for slow motion (from 1/2x to 1/2000x real time) or fast overview (from 1x to 5000x real time); select from the drop-down list.

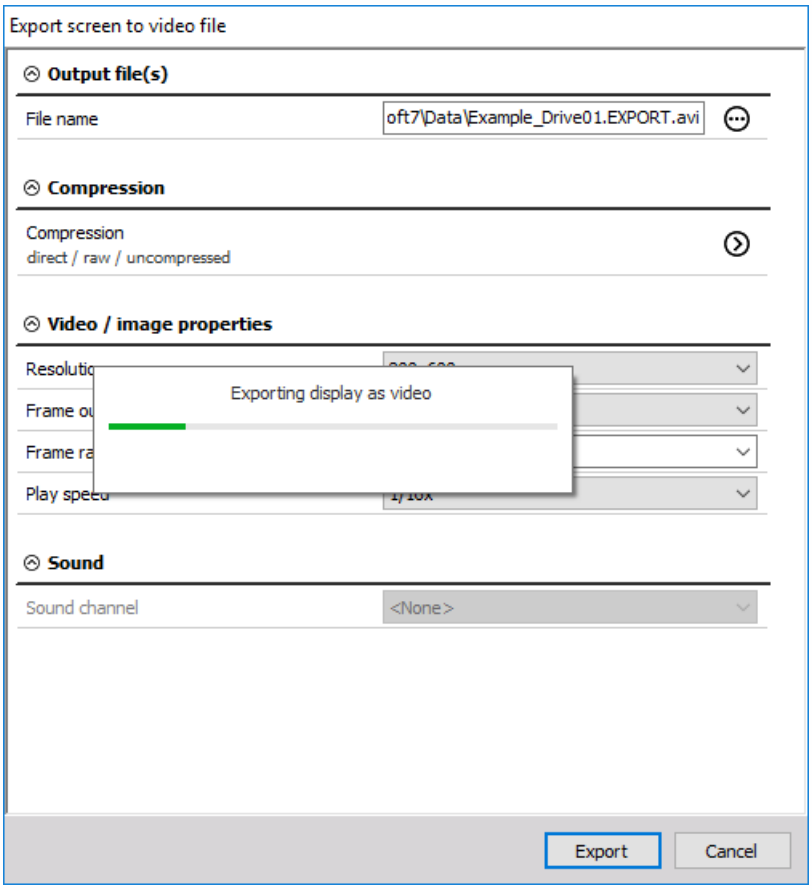


Sound channel: select from drop down list one of the available analog channels to be your audio channel.

Video compression: Press the " > " button and choose from drop down list the appropriate standard compression or decompression, after that you have to confirm you choice.



When you have done all changes press the Export button to start the export.



When Dewesoft has completed the export (depending on data and compression, this can take several minutes) you can open the exported file in a standard video player.

Example exported AVI, displayed with Microsoft Media Player: