DEWESoft X1

Data acquisition, processing, analyzing and storage software

User Manual

(Beta version)

地址：北京市海淀区小营西路27号金领时代大厦12层
电话：136 1171 664；010-5361 2036
传真：010-5635 3026
网站：www.chinaksi.com
电邮：ksi@chinaksi.com

General information

Software version

DEWESoft User Manual corresponds with software version X1.

Printing notice

Specifications subject to change without notice.

Copyright notice

Trademark notice

All trademarks are acknowledged to be the property of their owners, with all rights and privileges thereto. Noinfringement is intended.
Microsoft, Excel and Word are registered trademarks of Microsoft Corporation.
National Instruments is a trademark of National Instruments
Microstar, DAP and iDSC are trademarks of Microstar Labs, Inc.
Data Translation is a trademark of Data Translation, Inc.

Disclaimer

DEWESOFT makes no claim about the efficacy or accuracy of the information contained herein. Use ofthis manual is entirely at the user's own risk. Under no circumstances will DEWESOFT assume any liabilitycaused by the use, proper or improper, of this manual or the information, textual, graphical or otherwise,contained within it.

Table of Contents

科尚仪器
KingSci Instruments
...推动测试创新 创造实在价值
地址：北京市海淀区小营西路27号金领时代大厦12层
电话：136 1171 664；010-5361 2036
传真：010-5635 3026
网站：www.chinaksi.com
电邮：ksi@chinaksi.com

Part I

User Guide

DEWESoft is a measurement software which can **acquire** data from many different measurement hardware and enables the user to do **processing**, **storage** and **analysis** in a simple way.

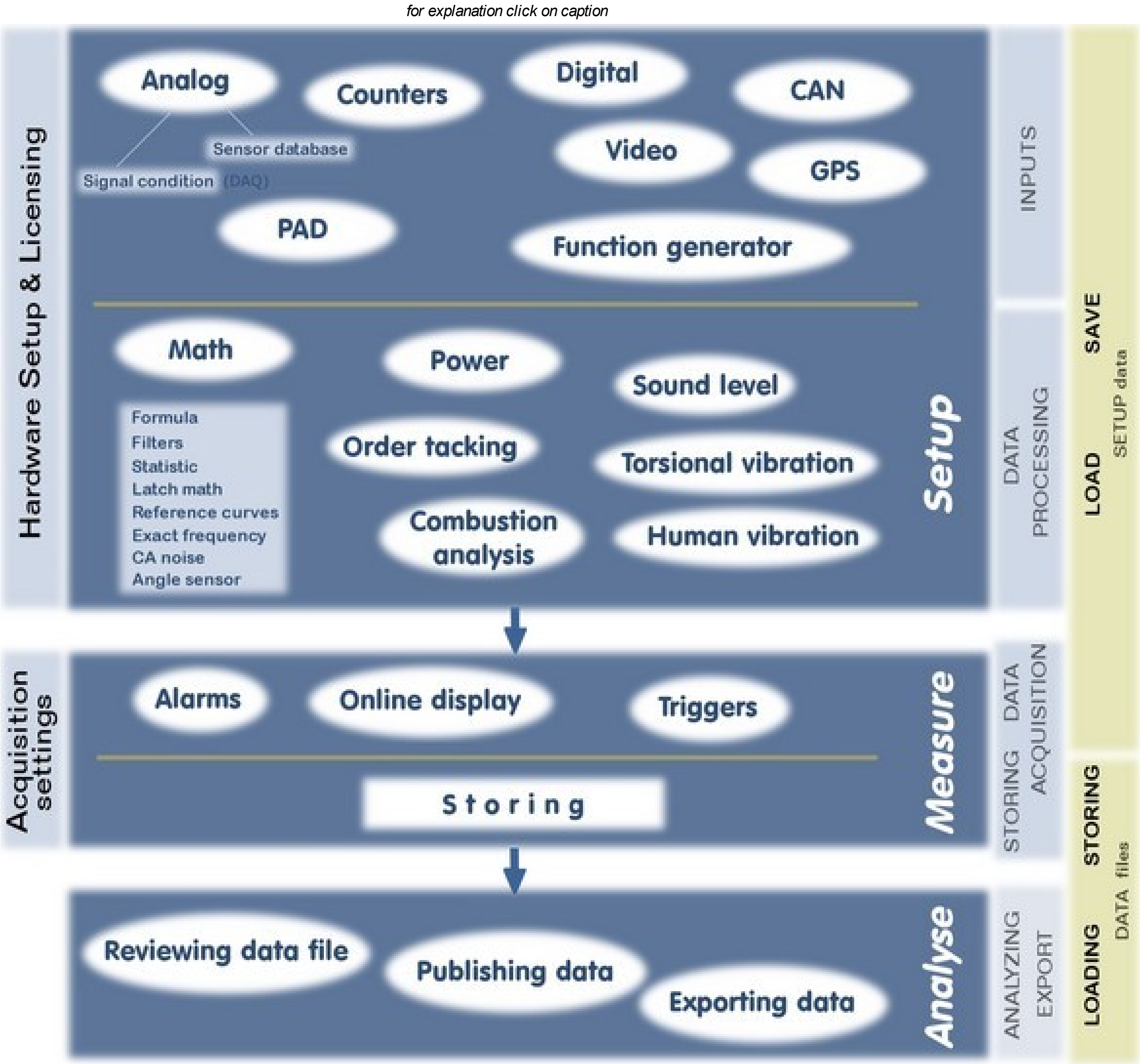
The main idea of DEWESoft is to have two modes of operation: **Acquisition** and **Analysis**. The main difference is that **Acquisition** part works with a real hardware while **Analysis** works with stored file.

But same math processing and visualization can be applied either during measurement or also on stored files. Therefore the parts of the manual describing the Measurement is valid also for analysis.

This manual should give the user an overview on the DEWESoft X1 and its **Application modules** functionality and its configuration possibilities in combination with various *measurement units*.

Manual covers general operating system settings as well as the DEWESoft configurations for various *measurement topologies* → *see also* **About this Manual**

for explanation click on caption



About this Manual

This manual is divided into the following sections:

User Guide

DEWESoft X1 includes **preparatory information** in **Setup files** (how to **create** and **select** the current *setup* or sequence for immediate start of measurement) and **Measurement setup** (important starting-point how to **select** and **set** measurement *channels* - **Analog in/out Channel setup**, how to **set** *data recording settings* (with *triggers*), how to **define** different *processing* (e.g. Math, Power, DSA...), with **General Mathematics module** chapter (how to **calculate** with *signals*, **filter** them or **set** *digital states* according to your acquired data)

Measurement - Data acquisition section explain **Measuring** and **data acquisitions** with DEWESoft X1 and include **Design Display Screens** chapter (how to **define** acquired data *appearance on screen*), and **Storing data** chapter (how to **define** on what conditions data are **stored**, how to **start** *measure* and *store* data)

Analyse - how to **analyse** acquired *measured data* with DEWESoft X1 (Loading data file, Displaying data and Replaying data, Selecting data to analyse, Working with Events), how to **Publishing the data** (printout of instruments, copy Channel setup and Display elements) and how to **Export data** for off-line analysis using other software

Data management - how to **Import** (Open) or **Export** (Save) *setups*, measured and other *data*

Sequencer - how to create and use sequence to automate testing with details about **Sequence building blocks**

System Settings

how DEWESoft X1 must be **set up** to acquiring and processing measured data with installed *hardware*

Installation Guide

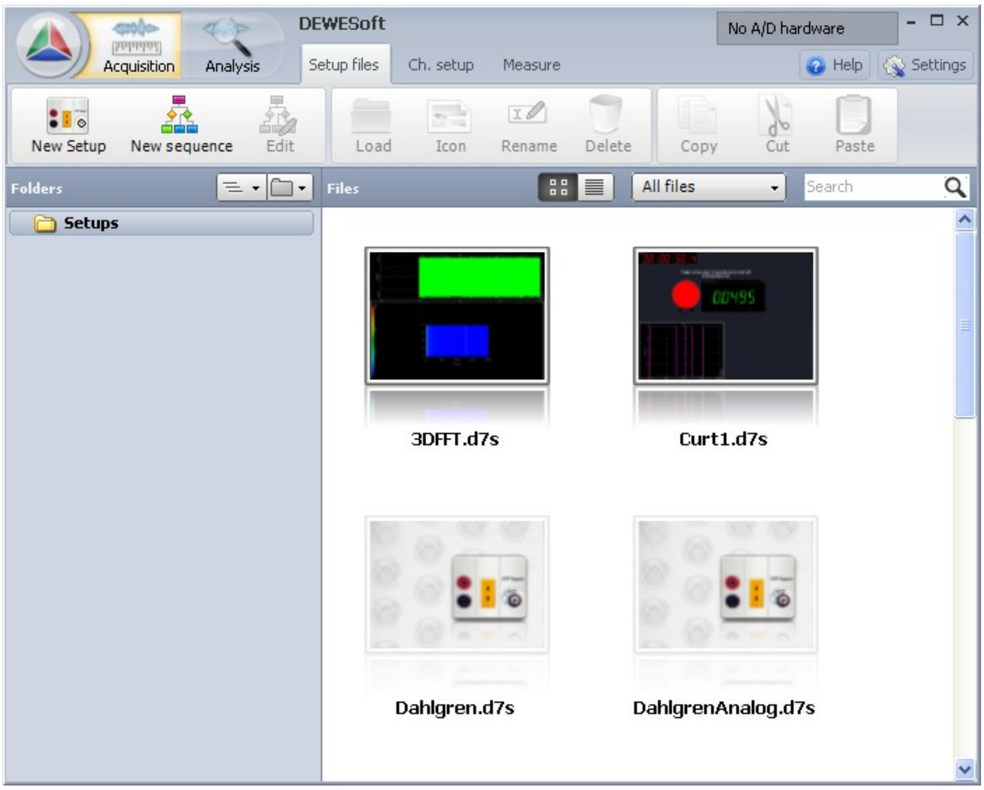
this section describes the **installation procedures** of DEWESoft X1 itself; for effective DEWESoft use all needed hardware devices and their drivers must be properly installed on respective computer

Reference Guide

gives the *background* information and *additional* information about supporting DEWESoft X1 Application

1 Setup files

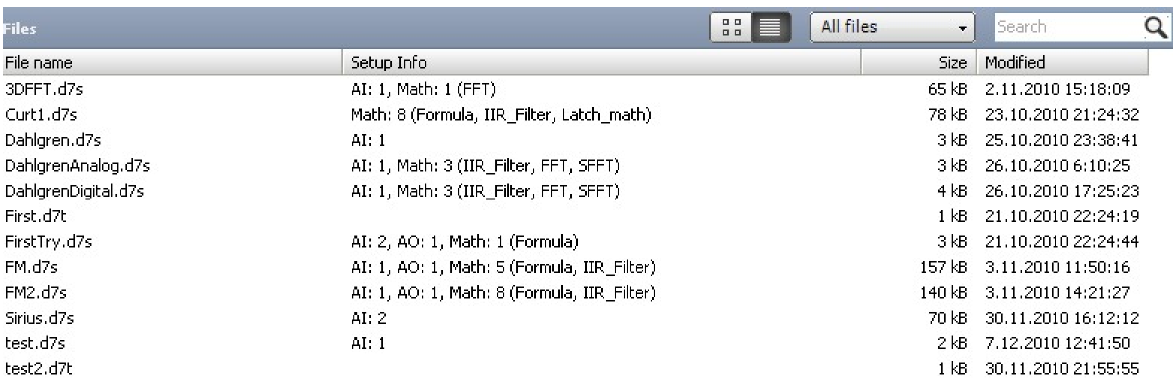
The startup screen of DEWESoft is *Acquistion* - *Setup files*. This will bring up a display where we can select the *current* setup or **sequence** for immediate *start* of measurement.



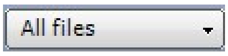
On the left side is the folder view. Folder can be chosen with clicking on the right folder name in the list. We can move on the higher levels of folders with the icon and selecting the right folder. For more information about folder navigation see the **Folder tree view navigation** topic.

On the right side is the file list of sequences and setups. A sequence or setup can be loaded by double clicking on the icon or on the item in the list.

File list can be organized to show icons or to show the list of setup. To switch between those use the toggle. List view will show additional information about the setup file, namely the number of channels in the setup, file size and date when the file was last modified.

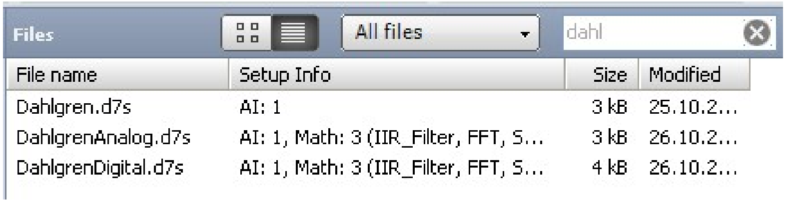


File name	Setup Info	Size	Modified
3DFFT.d7s	AI: 1, Math: 1 (FFT)	65 kB	2.11.2010 15:18:09
Curt1.d7s	Math: 8 (Formula, IIR_Filter, Latch_math)	78 kB	23.10.2010 21:24:32
Dahlgren.d7s	AI: 1	3 kB	25.10.2010 23:38:41
DahlgrenAnalog.d7s	AI: 1, Math: 3 (IIR_Filter, FFT, SFFT)	3 kB	26.10.2010 6:10:25
DahlgrenDigital.d7s	AI: 1, Math: 3 (IIR_Filter, FFT, SFFT)	4 kB	26.10.2010 17:25:23
First.d7t		1 kB	21.10.2010 22:24:19
FirstTry.d7s	AI: 2, AO: 1, Math: 1 (Formula)	3 kB	21.10.2010 22:24:44
FM.d7s	AI: 1, AO: 1, Math: 5 (Formula, IIR_Filter)	157 kB	3.11.2010 11:50:16
FM2.d7s	AI: 1, AO: 1, Math: 8 (Formula, IIR_Filter)	140 kB	3.11.2010 14:21:27
Sirius.d7s	AI: 2	70 kB	30.11.2010 16:12:12
test.d7s	AI: 1	2 kB	7.12.2010 12:41:50
test2.d7t		1 kB	30.11.2010 21:55:55

The 

button has the selection either to display *all* files, sequence files (`d7t`) or only setup files (`d7s`).

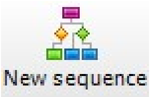
If we have a large list, we can search with entering the keyword in the search entry box.



Search criteria can be *removed* by pressing the  button.

There are several buttons on the top bar which helps us to work with setup and data files.

New setup will *remove* all previous channel, processing and display.



New sequence button will *open* a *Sequence editor* with *empty* sequence.

Edit *sequence* button will *edit* the currently *selected* sequence.

Load button will load the currently selected item. If it is a *setup*, if will show the channel setup. If it is a *sequence*, it will automatically *start* the sequence.

Icon button will open the dialog for *loading* the picture to be shown with setup or sequence in the icon view.

Rename button will open the dialog to *rename* currently *selected setup* or *sequence*.

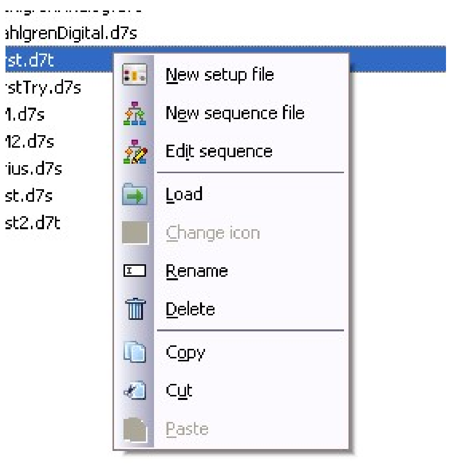
Delete button will *delete* currently selected *setup* or *sequence*.

Copy button will copy currently selected item to clipboard. Please note that the item can be directly pasted to any folder with *Windows Explorer* or it the *email* to send.

Cut will *remove* the item when *pasted* to *another* location.

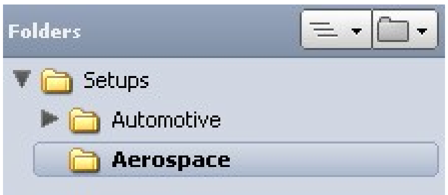
Paste will *add* the file which is on the clipboard *to current folder*.

Same options are available with right click on the file or icon list:



1.1 Folder tree view navigation

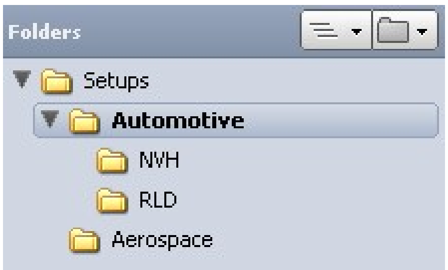
The same folder view is shown for setup, data and export files.



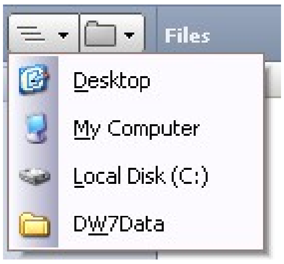
Navigating through folders

Currently selected folder is the one marked with darker color and the border. The current folder can be changed by clicking on the wanted folder.

When the folder has the sign in front, it has sub folders. The sub folders can be opened by double clicking on the main folder folder (in our case Automotive). Then the list of the sub folders will be shown. Folder can be closed by double clicking on it again.

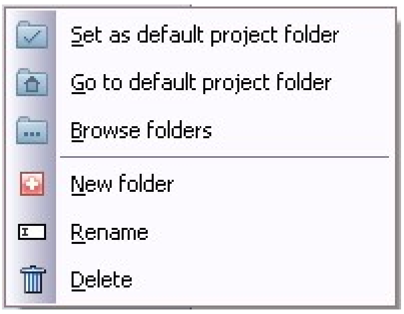


The top level folder is the one selected as default in the project settings. The folders above can be navigated by pressing the button. This will show the folders above currently selected one. We can go in those folders by clicking on the wanted folder from the drop down.



When we do this, the list will show the folders below the currently selected one. So if I select the Local Disk (C:), the list will show all sub folders of the c drive. The navigation works the same.

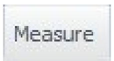
There are several actions which can be performed on the folder list. These action can be selected either by clicking on the  icon or by right clicking on the folder list. The actions are:



Set as default project folder	- this will set the currently selected folder as the default project folder (will be also shown at next restart as default
Go to default project folder	- will set the current top folder from the project file
Browse folders	- opens the file open dialog where the folders can be selected via standard Windows dialog
New folder	- will create a new folder and open a popup to name new folder under currently selected one
Rename	- will open the dialog to rename currently selected folder
Delete	- will delete currently selected folder. Be careful: this will delete also all sub folders and all files in those folders.

2 Measurement setup

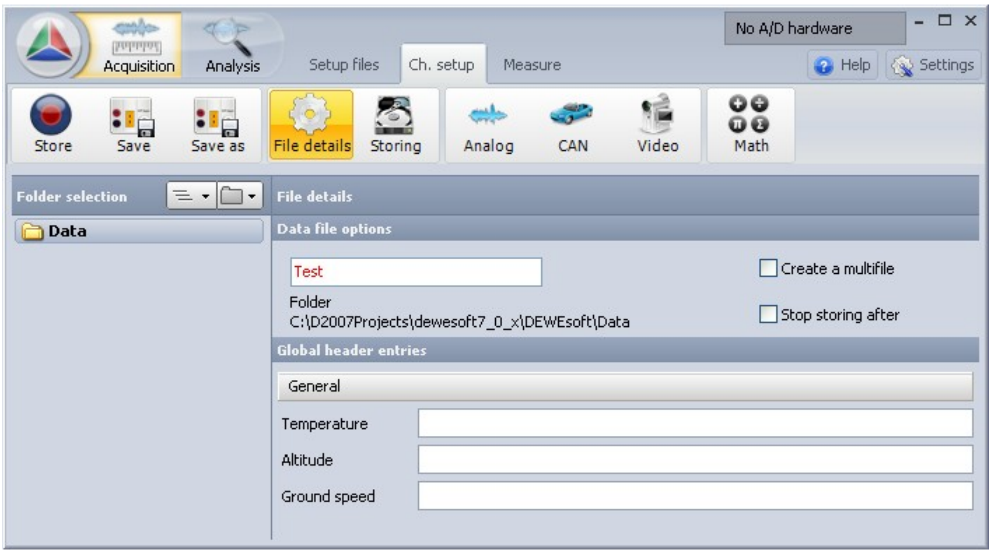
DEWESoft Setup is important starting-point to select and set *measurement channels*, set *Data recording settings (with triggers)*, define different processing (e.g. Math, Power, DSA...).

Press the 

- **Measure** button and then press 

- **Setup** button on DEWESoft tool bar (in the Analyze mode this button have different function: display *Settings*, *Events* and *Data header* and *Define the post processing* math→ *see* → **Analyse mode**).

The setup screen consists of several static tabs, which are always there. Those are **File details** and **Storing**. In those tabs we define *data header* information and *set* the *storing*.



File details

File details are intended to set the file **name** for storing and to enter specific **data header entries** which relates to single measurement which is about to be stored.

In upper part of the DEWESoft *Setup* screen we can define the *file storage options*. For more information, please check next chapter: **Data file options**.

The data header entries can be defined in the *Settings* → *Project settings* → *Data header design*. For more information how to define the data header, please see **Data header** *topic*.

Storing

In the *Storing* tab, the upper part is the same as in the *File details* to set the **file name properties**. The lower part defines the storing strategies, which are explained in **Recording setup** tab.

Displayed tabs right to the file details and storing button of DEWESoft Setup *screen* depend on selection in *System* → *Hardware setup* menu and consecutive on *measurement* mode (**REAL** or **DEMO**) → *see* → System Settings → **Hardware setup**.

Additional screen tabs we can *define* and *set*:

Input channels	
• with **Analog IN Channel Setting** (Analog tab) *all* *analog INPUT channels* and their *sensors* (and *sensors group* from **Sensor database**), which will be *used* during measurement using installed hardware • with **Analog OUT Channel Setting** (Analog out tab) *all* *analog OUTPUT channels* (also *set up* **Function generator**), which will be *used* during measurement using installed hardware • **CAN Acquisition** to *listen* to the *traffic* on CAN bus *interfaces* and to acquire CAN messages	• **Counter** to perform *counting* and *frequency measurements* • **GPS Acquisition** to acquire GPS data • **Video Acquisition** to acquire videos together with other sources and to handles the *relation* between the *analog data* and the video • **Digital input** to measure two ***states of the inputs*** (*low* and *high*); it is useful for measurement of status signals

Supplementary *functions*	
• **Math** to *calculate* with *signals*, *filter* them or *set* *digital states* according to your acquired data	• **Alarms** to *set* the *digital states* according to *acquired data* and *display* alarm state on online screen

Special applications:

- **Torsional vibration** module allows measure *dynamic* and *static* bending and vibration of the shafts
- **Sound level** module allows calculating typical parameters for *sound level* measurements from a *single microphone*
- **Human vibration** allows measurement of *effect of vibrations* and *judge the risk* to human body
- **Combustion analysis** module is used to calculate parameters of *internal combustion engines*
- **Power** module provide all *functions* for a *total* power analysis like calculate *power* with all its *components*
- **Order tracking** method is used to extract the *harmonic components* related to *rotational frequency* of the machine which relates to certain machine *fault*
- **NET Acquisition** to *acquire* data over the network

There are additional tabs possible based on *custom* **plugins** and *custom* **math**, defined as well in the custom settings.

The *standard* **plugins** which are currently present in the installer are:

Custom data acquisition devices (appears under *Analog* tab of *Hardware setup* screen)

BK 4447 - support for 4 channel ICP human vibration meter

Sound card - support for any sound card supported by Multimedia API

RogaDaq2 - support for single or dual channel Roga devices

Data acquisition plugins (appears in Plugins page of *Hardware setup* screen)

Ballard - support for Ballard Arinc 429 and MIL-1553 avionics bus

CANout - output of measured data using CAN bus

CPUUsage - adds channels showing system parameters of the computer (like CPU usage...)

DSNET - support for DEWESoft DS NET modules

ADMA_CAN - support for Genesys ADMA gyro platform using CAN bus

J1587 - support for J1587 truck bus using Vector hardware

RoaDyn2000_LAN - support for Kistler RoaDyn torque wheels using Ethernet interface

NMEA - support for NMEA compliant weather station

OBD2 - support for OBD2 vehicle interface using CAN physical interface

EPAD_BASE2 - plugin to set EPAD_BASE interface for EPAD modules

Tarsus - support for Ulyssix Tarsus PCM card

XCP - support for XCP over Ethernet vehicle bus

XSens MT - support for XSens gyro platform

Processing plugins (appears in Plugins page of *Hardware setup* screen)

Testbed - interface for combustion analysis test beds using AK protocol

AOPlugin - plugin for slow analog output of measured data

AutoExport - plugin to automatically export the data at the end of the measurement

Polygon - plugin for defining the track and calculating distances and positions between objects

Sendmail - plugin to send email on DEWESoft event

Math plugins (appears in Math page of *Hardware setup* screen)

SRS - plugin to calculate shock response spectrum math

Rosettes - plugin to calculate rosette strain gage configuration

BrakeTest - plugin to calculate brake, acceleration, coastdown tests

ACCPerformance - plugin to evaluate active cruise control systems

Psophometer - plugin to calculate the influence of electrical grids on telephone lines

Visual controls (shown on *Design* page of the visual display)

FRF geometry - FRF geometry editor and visualization (useful with transfer function math)

ModalCircle - Display of modal circle plot (useful with transfer function math)

Polygon3D - 3D visualization of vehicles (useful with Polygon math)

RotorBalancer - math for rotor balancing

Cameras (appears in Video page of *Hardware setup* screen)

Dewecam - support for Allied vision cameras based on BCAM driver

Photron - support for Photron high speed video cameras

Exports (appears in Export to file list in Analyse mode)

ATI - native iDEAS file format

BWF - EBU audio broadcast wave format

Google earth - export of GPS data to Google Earth

SDF - export to standard data format defined by HP

TDM - export to NI standard data format

WAV - export to wave file

WFT - Nicolet standard WFT file format

Imports (appears in *data file* list in Analyse mode)

Text import - import of text file format to DEWESoft

DS NET - import of DS NET logged files to DEWESoft

2.1 Data file name options

File name setup is available in Measure → Ch. Setup → File details page and on Measure → Ch. Setup → Storing page.

The folder view on the left side defines the folder where the data will be stored. The folder view is described in the **Folder tree view navigation** chapter.

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

DATA FILE OPTIONS - Setting the filename

On the DEWESoft *Setup* screen we *predefine* the **filename** to be used for *data recording*.

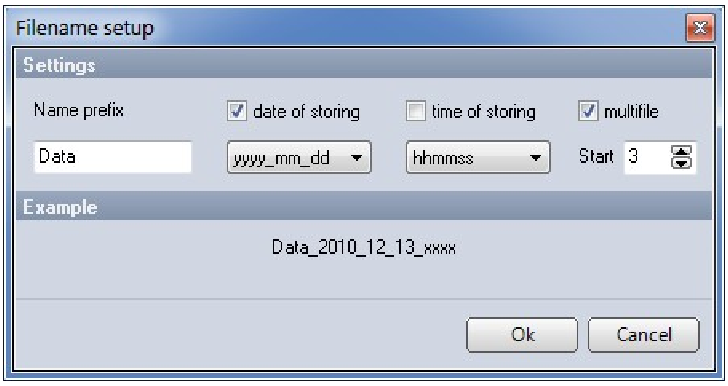
DATA FILE OPTIONS - Create a multifile

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 checkbox called **Create a multifile**. If you check it, then DEWESoft will take your *base* **filename**, like '`Data`' (in example bellow '`Multifile`') and just add underscore and four digits to it *each* time you *start storing* (in example bellow '`_0003`', for 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 checkbox the **Make new file after** checkbox also *appears*.

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



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

If **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 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.

CAUTION: *The multifile storage is independent from 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

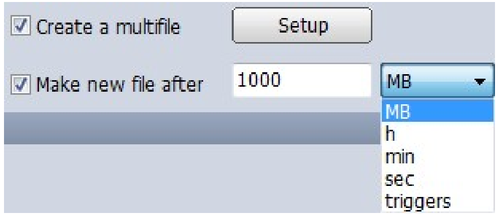
In version 7 all the *file name settings* are stored in *setup file* by default.

Make new file after

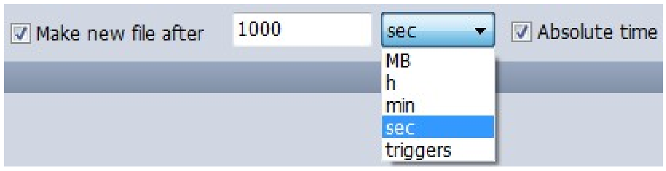
When the Create a multifile box is *checked*, the Setup screen has an <u>additional</u> checkbox called **Make a new file after**. If you check it, you can *enter* a value and the unit. The criterion for switching the files is either the *file size* or *time interval*.

Please *select* between *file size* values like MB (Megabytes) and triggers (*number* of trigger *events*)

in this example DEWESoft will stop storing after 1000 MB



and *time* related values like h (hours), min (minutes), sec (seconds).



If we select *time* relates value, 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 source, if available - defined in hardware setup). The file switching is done in the way that *no data point is lost* in between.

DATA FILE OPTIONS - Stop storing after

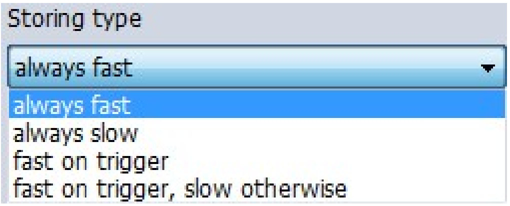
When the Create a multifile box is <u>not</u> checked, the Setup screen has also checkbox called **Stop storing after**. If you check it, you can *enter* a value and the unit. Please *select* between *file size* values and *time* related values - all setting are similar as for **Make a new file after**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

2.2 Recording setup

STORING OPTIONS

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 to the file with *full speed*.

- **always slow** This function **stores** data *at intervals*, set with Static/reduced rate.
  Even thought storing is set to *slow*, DEWESoft will **acquire** the data with 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:

for information about ***start / pause / stop storing*** see → User Guide → **Start / Stop Recording - Manually**

If our data consists of events which can be captured, we can choose to store with two options of **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.

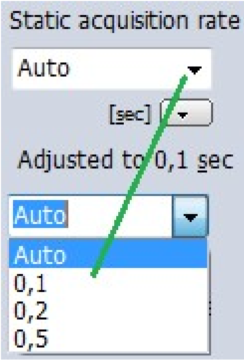
- **fast on trigger** The data will only be **stored** with the *full acquisition rate* when *trigger* condition is *true*.

- **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.

for information about ***triggered storing*** see → **Triggered storing** bellow

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.

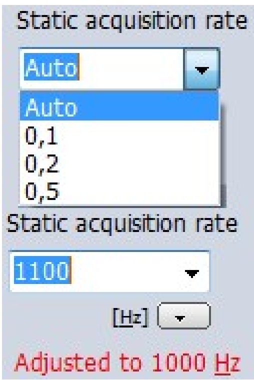
You are also allowed to enter your *own* values, but be aware that not all values will be accepted - if so the *real value* will be mentioned *below* the drop down list.

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, is 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 STATING / 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 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.

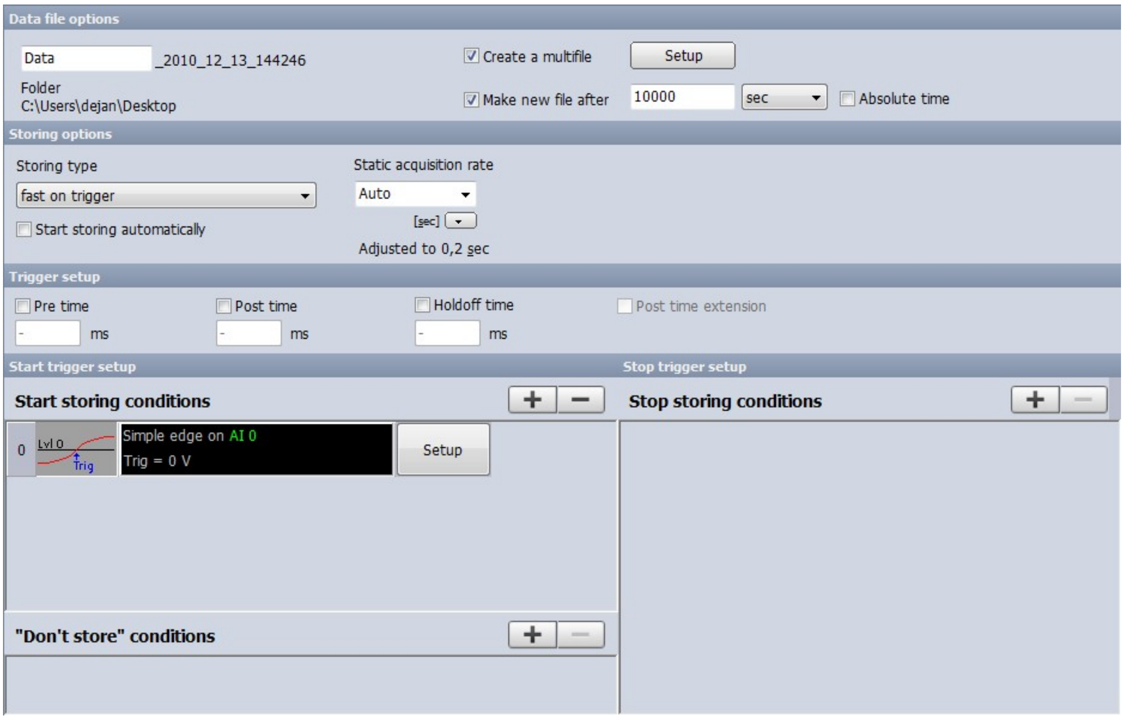
When you switch to the Analyse mode, and *load* one of these files, you can **export** *any set* of values:

min/max, average, or RMS (*see* → User Guide → **Export multiple files**).

Triggered storing

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

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



On this screen can be set *start* and *stop* trigger *conditions*:

- **Trigger setup** — to set the ***storage time*** *before* and *after* the trigger event
- **Start trigger setup** — defines *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 occur → DEWESoft will *keep* the data in the *buffer* until the trigger event occurs and then store also this data to the file.

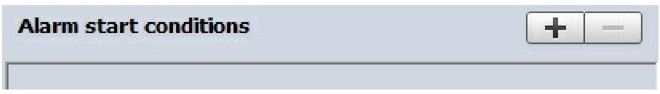
As a standard, this feature is *not* selected and the storage ***starts** with* the trigger event itself.

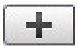
- **Post time** — Post trigger time, defined in milliseconds. This value defines the storage *duration* after the trigger event has been finished → DEWESoft will *continue to store until* we stop it manually or stop condition occurs.
  As a standard, this feature is *not* selected and the storage ***stops** 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 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

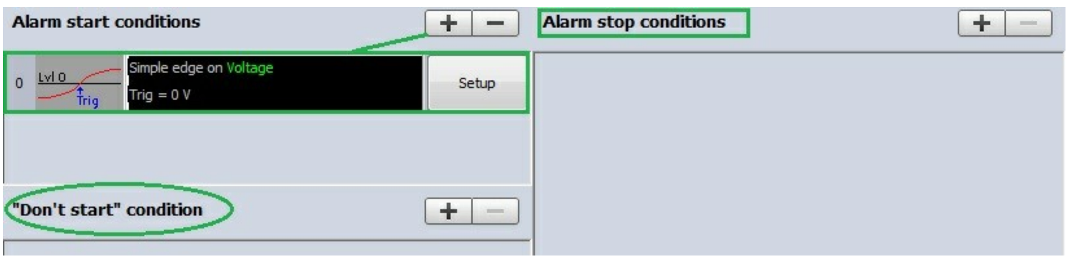
When you select the trigger setup for the first time, there is *no* **start trigger condition** *defined*:



When you press:  - the **Add** button, a new **trigger condition** will appear immediately in the list

 - the (minus) button, *selected* **trigger** condition is *removed* from 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 part of screen
- empty Stop trigger condition in new Stop trigger setup column

As a *standard*, the trigger condition is set to Simple edge trigger with *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 to enter the **trigger condition settings**.

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.

for detailed information about ***Store trigger settings*** *see* → User Guide → **Scope setting**

"Don't store" setup

The **Don't store trigger** condition and Setup works in exactly the *same* way than 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 w hen the pressure grow s above a certain limit. But you only w ant to store the data if there is really a part in the machine. Then you can connect a signal to another channel w hich reports if the unit is present or not and use this result as a Don't store function.

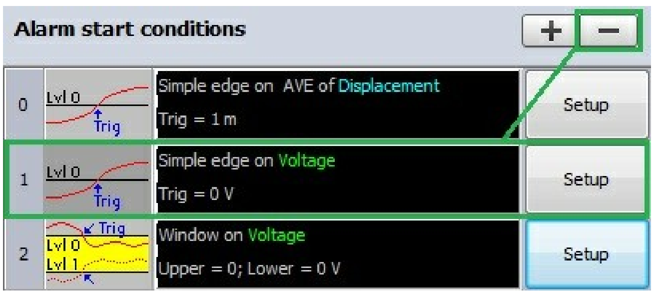
Stop trigger setup

The **Stop trigger** condition and Setup works in exactly the *same* way than 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 the  (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 tool bar or F5 on the keyboard. As soon as the trigger condition *appears* on the *input*, the data will be stored into 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 *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 into the *same* file. DEWESoft offers a special ***analysis*** feature in the Analyse **mode** for an easy data analysis.

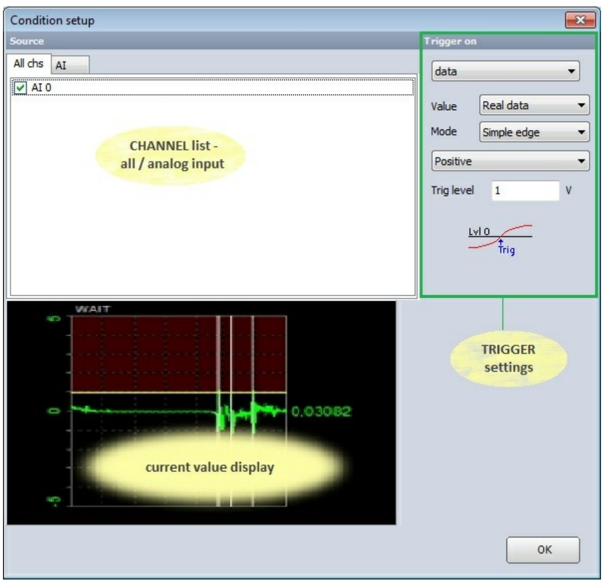
for information about ***start / stop storing*** *see* → User Guide → **Start / Stop Recording - Triggered**

for information about ***triggered data files*** *see* → User Guide → **Reload triggered file**

2.2.1 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:



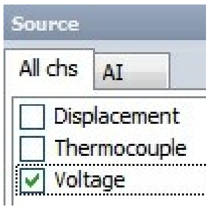
Choose the **trigger condition** according to your requirements and press the **OK** button to *accept* the trigger settings.

In Condition setup window we can *enter all* the **trigger condition settings**:

- Source section on screens left side:
  - 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 lower part *current signal value* is displayed

    The w hite lines displayed indicate w hen the trigger condition *became true*.

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

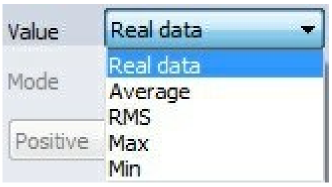
Trigger type

The following trigger types are supported:

- **Data**
- **Time**
- **FFT**

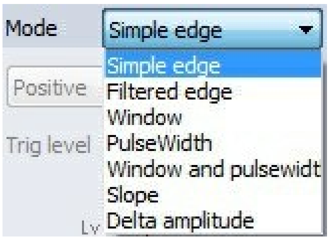
Select *trigger type* from drop down list.

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** or **Slope** from the drop down list.

3. *setup* ***other trigger condition***

These settings (e.g. Slope, Trigger level, Rearm level, Pulse time...) depend on *selected* trigger ***type*** in **Mode** field.

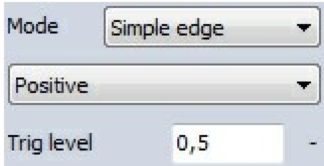
Trigger **Mode and settings**

• 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:

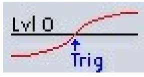


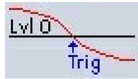
Trigger on *rising* edge w hen signal rises *over* defined **Trig level**.

Negative slope:

Trigger on *falling* edge w hen signal drops *bellow* defined **Trig level**.

symbolic display of trigger condition (below settings)





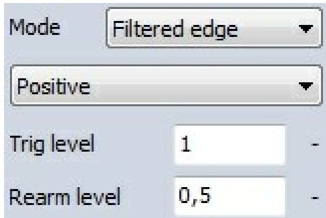
• Filtered edge

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*.

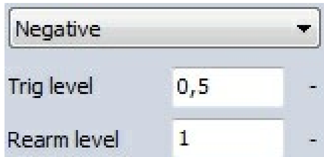
This trigger type is mostly used with very *noisy* signals.

Positive slope:



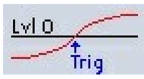
Trigger on *rising* edge w hen signal rises over defined **Trig level**; retriggers *only* w hen **Rearm level** has been *crossed*.

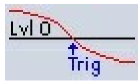
Negative slope:



Trigger on *falling* edge w hen signal drops *bellow* defined **Trig level**; retriggers *only* w hen **Rearm level** has been *crossed*.

symbolic display of trigger condition (below settings)



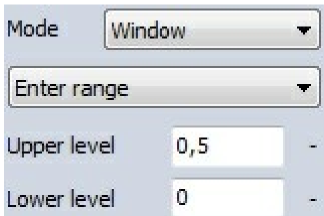


• Window

works with *two independent* levels, which build some kind of window.

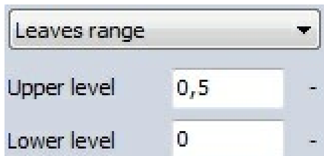
The trigger condition can become true when the signal *enters* or *leaves* the *window*.

Enter ***range***:



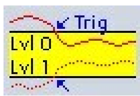
Trigger w hen the signal *enters* the w indow - signal *falls below* **Upper level** or *rises above* **Lower level**.

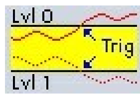
Leaves ***range***:



Trigger w hen the signal *leaves* the w indow - signal *rises above* **Upper level** or *falls below* **Lower level**.

symbolic display of trigger condition (below settings)



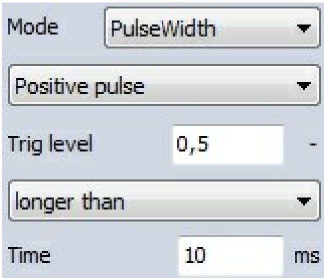


- **Pulse-Width**

Longer than Time

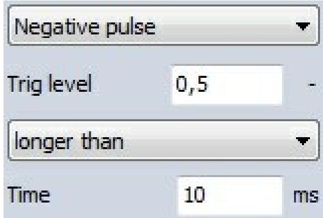
checks in addition to the level (like the simple edge trigger) the *duration* **Time** of the event and triggers only if the event is ***longer*** above the selected *level*

Positive pulse:



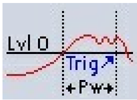
Trigger on *rising* edge when signal rises *over* defined **Trig level** and *stays above* this level longer than selected ***Time***.

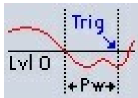
Negative pulse:



Trigger on *falling* edge when signal drops *bellow* defined **Trig level** and *stays bellow* this level longer than selected ***Time***.

symbolic display of trigger condition
(below settings)

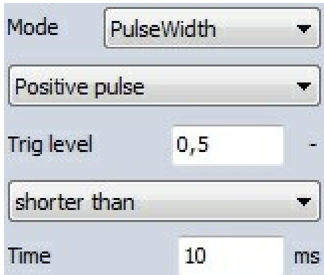




Shorter than Time

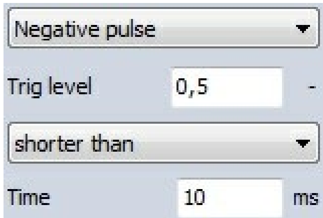
checks in addition to the level (like the simple edge trigger) the *duration* **Time** of the event and triggers only if the event is ***shorter*** above the selected *level*

Positive pulse:



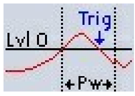
Trigger on *rising* edge when signal rises *over* defined **Trig level**, but *falls below* this level ***earlier*** than selected ***Time***.

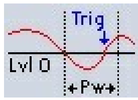
Negative pulse:



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)



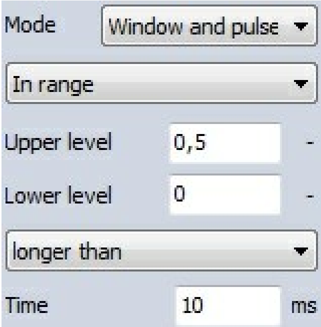


- **Window and Pulsewidth**

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.

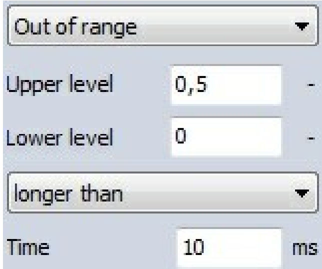
Longer than Time

level **In range**



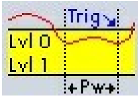
Trigger when the signal *enters* the window (signal *falls below* **Upper level** or *rises above* **Lower level**) and stays *inside* for a longer than defined ***Time***.

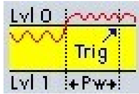
level **Out of range**



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***.

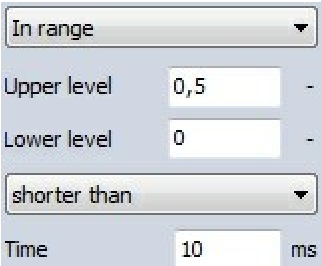
symbolic display of trigger condition (below settings)





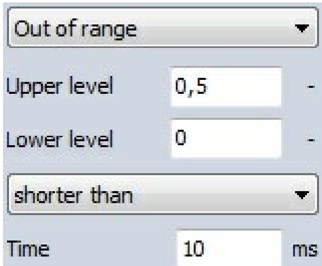
Shorter than Time

level **In range**



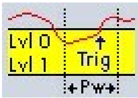
Trigger when the signal *enters* the window (signal *falls below* **Upper level** or *rises above* **Lower level**) but *leaves before* the defined ***Time*** is over.

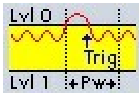
level **Out of range**



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 (below settings)

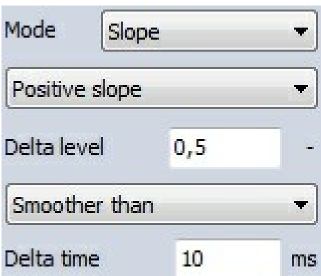




- **Slope**

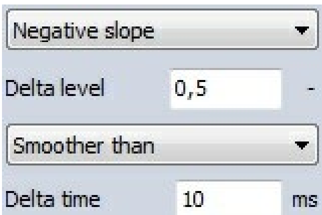
Smoother than Delta time

Positive slope



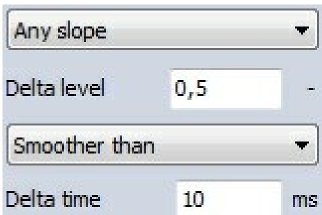
Triggers when signal rises *over* defined **Delta level** ***later*** than defined ***Delta time***.

Negative slope



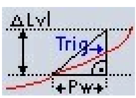
Triggers when signal drops *below* defined **Delta level** ***later*** than defined ***Delta time***.

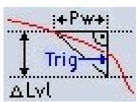
Any slope

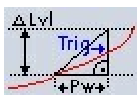


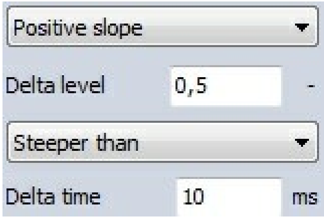
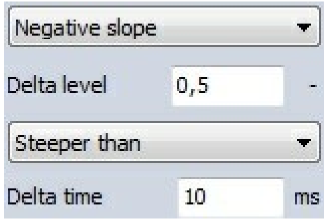
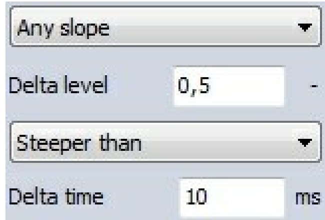
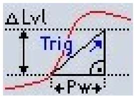
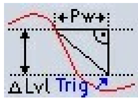
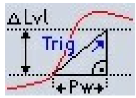
Triggers when signal rises *over* or drops *bellow* defined **Delta level** ***later*** than defined ***Delta time***.

symbolic display of trigger condition (below settings)



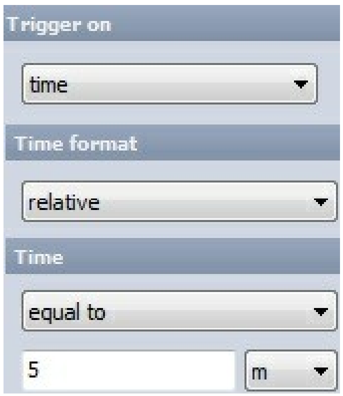
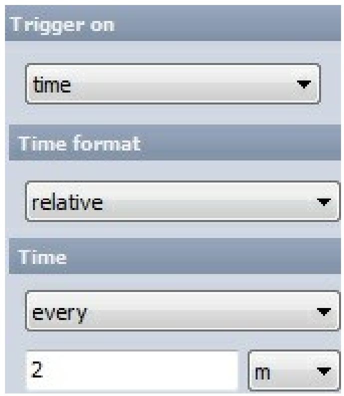




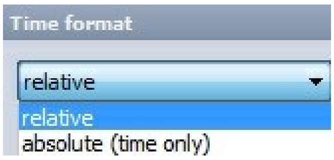
	Positive slope	Negative slope	Any slope
Steeper than Delta time			
	Triggers when signal rises *over* defined **Delta level** ***earlier*** than defined ***Delta time***.	Triggers when signal drops *bellow* defined **Delta level** ***earlier*** than defined ***Delta time***.	Triggers when signal rises *over* or drops *bellow* defined **Delta level** ***earlier*** than defined ***Delta time***.
symbolic display of trigger condition (below settings)			

Time trigger setup

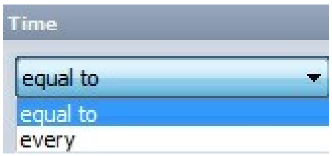
	time **equal to**	level **every**
absolute (time only)		
	The system triggers *exactly* at the defined ***time*** hh:mm:ss.xxx (also *every* day if the time matches).	The system triggers ***every*** defined ***time*** x [unit], the time starts running from the *beginning* of measurement.
displayed time		

	time **equal to**	level **every**
relative		
	The system triggers when the defined ***time*** x [unit] has been *passed* since the measurement has been started.	The system triggers every ***time*** x [unit], the time starts running from the *beginning* of measurement.

Time format select from drop dow n list:



Time select from drop dow n list:

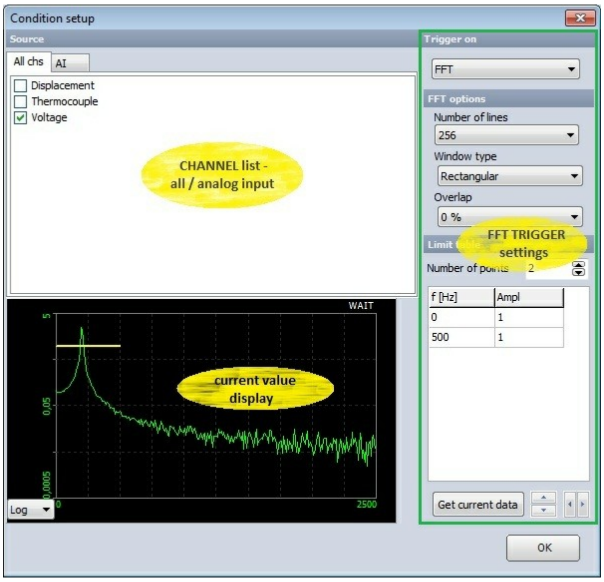


[unit] select from drop dow n list:

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 *frequency* domain!

This type of trigger is very helpful in any kind of *dynamic* applications where you want to *supervise* the *frequency behaviour* 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 detailed information about ***Number of lines***, ***Window type*** and ***Overlap*** *see* → User Guide → **FFT instrument settings**

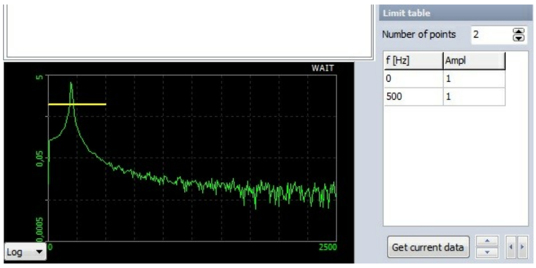
for background information about ***FFT analysis*** → *see* → Reference Guide → **Theory of frequency analysis**

Preview at the left bottom area *shows* the change effects on the FFT *immediately*. On this display can be select beside **Log**arithm (*see* display above) also **Lin**ear display.

Limit table

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

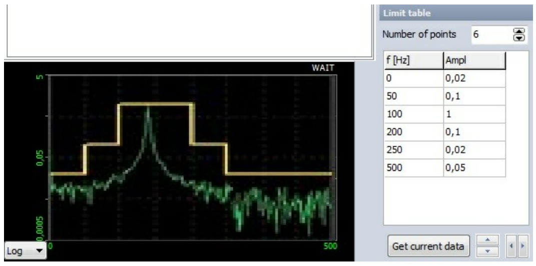
- **Number of points** (limits on Limits table)
- the **limits** - Ampl. **levels** for f [Hz] on Limits table.



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 (e.g. 6 in example on right).

You can now define the limits in two ways:

1. *Enter* the values *manually*

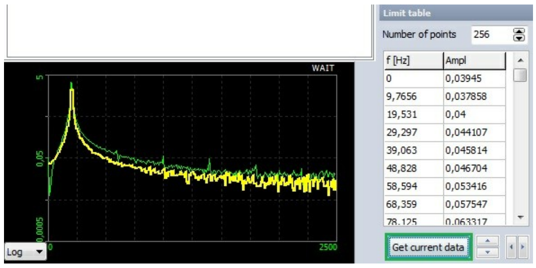
   When you enter the values manually into the table, you normally *take* just a *view points* to define the ***frequency mask***.

2. *Take 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.

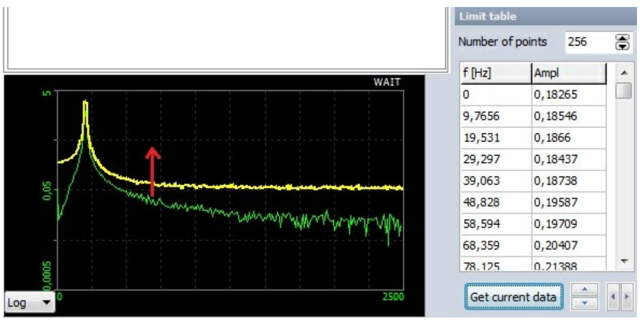
   The *currently* ***calculated FFT*** will be *stored* as a *mask* and displayed both on the *preview display* and in the *table* at the left bottom.

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



Examples:

frequency mask moved *up*



frequency limits widened

To set the ***table*** back to a default state *reduces* the Number of points to 2.

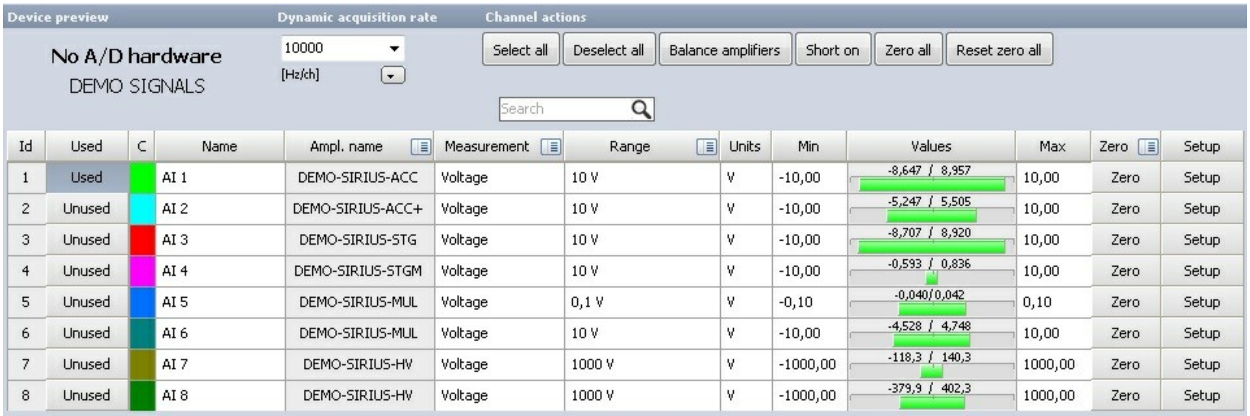
2.3 Analog in

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

After DEWESoft **analog input Hardware setup** (this procedure is to perform *only* by new installation or changing input hardware), the **Analog** tab on DEWESoft *Setup screen* can be selected to *set all* **analog input channels** and this screen displays two main parts:

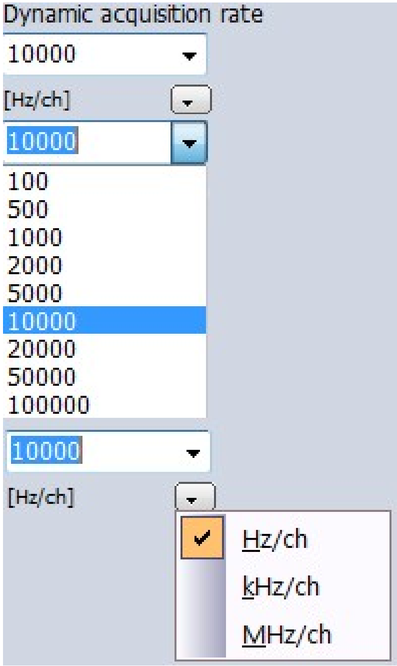
Settings
- **Dynamic acquisition rate** - defines the sample rate of analog, counter and digital channel
- **External Clock** - to use *external clock* as the ***source*** of ***acquisition***
- **Start on external trigger** - to *start the acquisition* on ***external event***

Channel list
- **Analog IN channel list** with information and settings in columns including **Channel setup** window call



Id	Used	C	Name	Ampl. name	Measurement	Range	Units	Min	Values	Max	Zero	Setup
1	Used		AI 1	DEMO-SIRIUS-ACC	Voltage	10 V	V	-10,00	-8,647 / 8,957	10,00	Zero	Setup
2	Unused		AI 2	DEMO-SIRIUS-ACC+	Voltage	10 V	V	-10,00	-5,247 / 5,505	10,00	Zero	Setup
3	Unused		AI 3	DEMO-SIRIUS-STG	Voltage	10 V	V	-10,00	-8,707 / 8,920	10,00	Zero	Setup
4	Unused		AI 4	DEMO-SIRIUS-STGM	Voltage	10 V	V	-10,00	-0,593 / 0,836	10,00	Zero	Setup
5	Unused		AI 5	DEMO-SIRIUS-MUL	Voltage	0,1 V	V	-0,10	-0,040/0,042	0,10	Zero	Setup
6	Unused		AI 6	DEMO-SIRIUS-MUL	Voltage	10 V	V	-10,00	-4,528 / 4,748	10,00	Zero	Setup
7	Unused		AI 7	DEMO-SIRIUS-HV	Voltage	1000 V	V	-1000,00	-118,3 / 140,3	1000,00	Zero	Setup
8	Unused		AI 8	DEMO-SIRIUS-HV	Voltage	1000 V	V	-1000,00	-379,9 / 402,3	1000,00	Zero	Setup

DYNAMIC ACQUISITION RATE



Enter or pull-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 options drop down list to select different storage options - *see* → **Storing 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 to the file.

for details about ***sampling rate*** also *see* → Reference Guide → **Basics of A/D conversion**

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!

2.3.1 Channel list

Id	Used	C	Name	Ampl. name	Measurement	Range	Physical qua.	Units	Min	Values	Max	Z...	Setup
1	Used		AI 1	SIRIUSi-ACC	Voltage	10 V		V	-10,00	0,000	10,00	Zero	Setup
2	Unused		AI 2	SIRIUSi-ACC+	Voltage	10 V		V	-10,00	0,000	10,00	Zero	Setup
3	Unused		AI 3	SIRIUSi-STG	Voltage	10 V		V	-10,00	-0,025	10,00	Zero	Setup

Common channel settings on **Analog in/out** and also other **channel lists** are in columns:

Slot - *input* channel

On/Off - *activate / deactivate* channels

C	- channel *color* selector
Name	- channel *name*

Additional to these on *analog input channel* list the following columns with *information* and *channels settings* appear:

Amplifier	- *amplifier* type and ranges; also set channel for *storing*
Physical values	- current *input values*
CAL	- remove *offset* - *input zeroing*
Setup	- *input settings* and *calibration;* to call **Channel setup** window

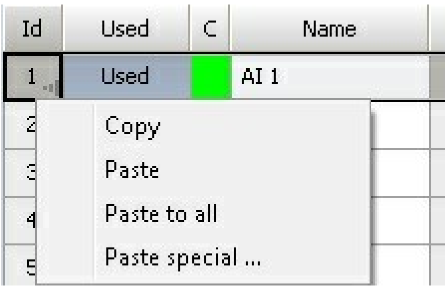
2.3.1.1 Input channel - Slot

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.

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

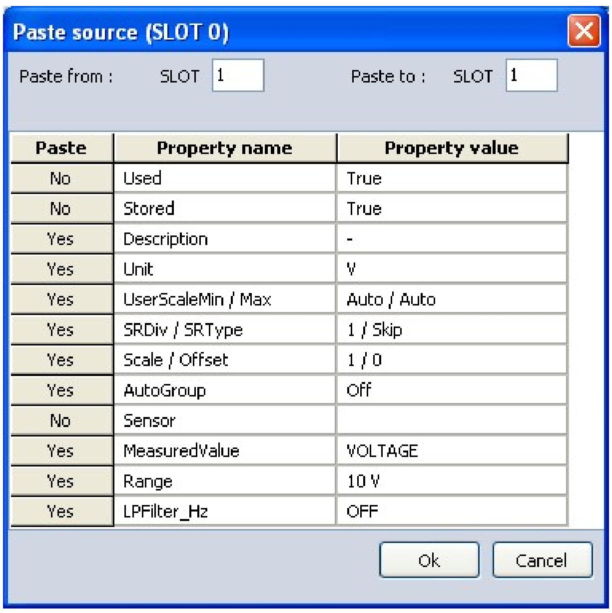
This field has also a copy / paste function. When you right *click* on a *slot number*, a window will appear:



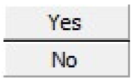
Example: several channel (e.g. 1, 2 and 6) contain the *same* module with the same sensor connected. If you click now on slot 0 and select Copy, the system will remember all the important ***settings*** from channel 0, like amplifier *type*, *input* and *filter ranges*, *units*, *calibration* and *zero* 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 module.

But you can copy the settings also to a channel *selection* or to *all* the channels.

Paste special will be give 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.



ATTENTION: 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.

2.3.1.2 Activating channel - On/Off

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.

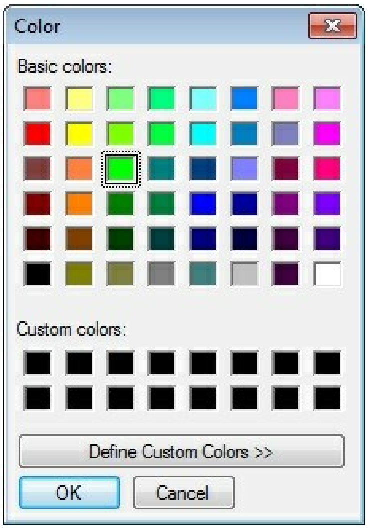
Select input channel for storage

Inside the channel setup **Store** button is available. It is on by default. To disable storing of the channel, unpress 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*).

2.3.1.3 Channel color - C

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.

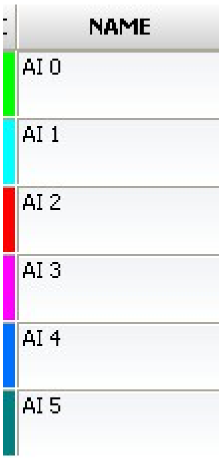
The color of the channel can also be changed on the *Channel setup* screen - General part of:

- *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.

2.3.1.4 Channel name - Name

Free text field for **naming** this channel.



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

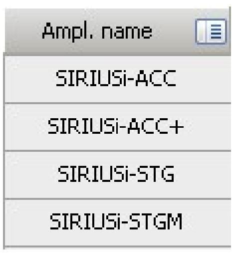
Navigation keys are same as in other *Windows* application.

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**

2.3.1.5 Amplifier type and ranges, sensors

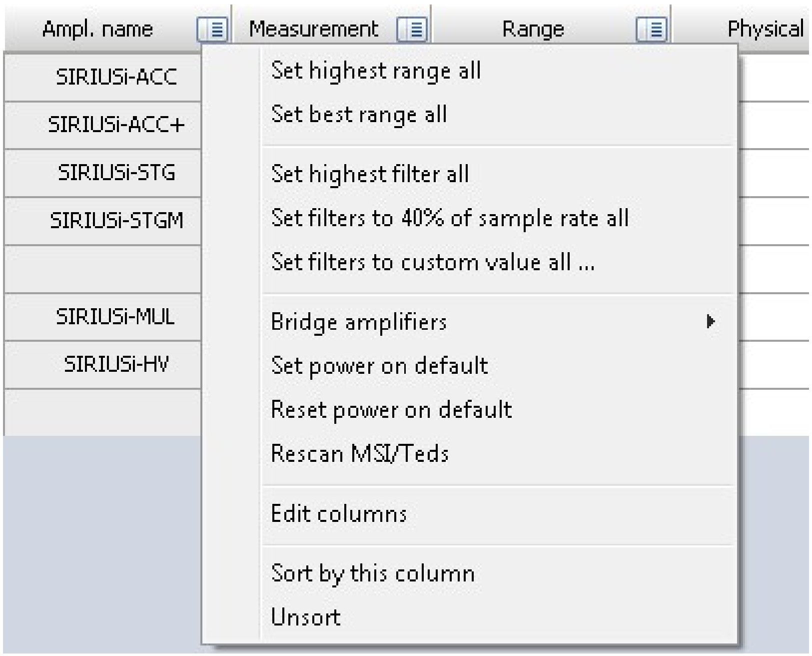
Shows the **Module** currently *installed* in this *slot*:



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* default settings to *factory default*.

If the module is *removed* from the system during operation, it will turn in 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 sample rate ... will *set* the *analog input filter* to the *nearest value* of the Nyqist 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 option to *start adding* modules to the system.

for more information about ***add / replace modules*** *also see* → System settings → **Module Installation tips**

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

If antialiasing 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 filter 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 helps 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* start and end *offset* of amplifiers.

If any Charge modules are installed, the dropdown 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).

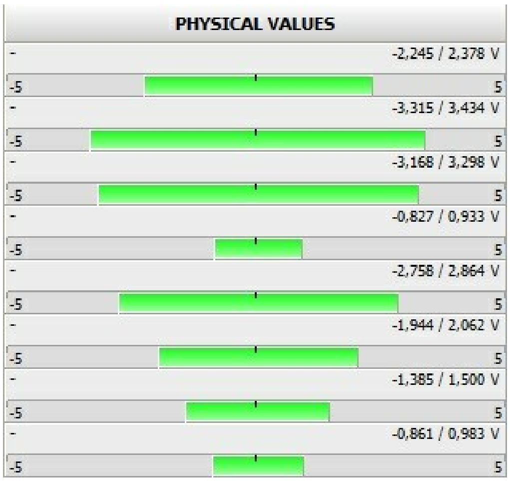
Sensors and TEDS sensors

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

*for more information about **Sensor database** also see →* User Guide → **Sensor database**

The same column shows the **TEDS** sensors, if there are any attached to the amplifier. TEDS sensors are sensors with 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 user defined sensor.

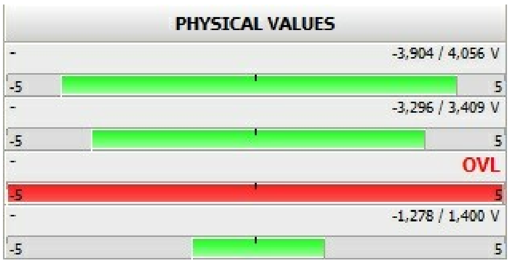
2.3.1.6 Physical values



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.

When the input signal *exceeds* the possible range, a red indicator OVL - *overload* will be displayed:

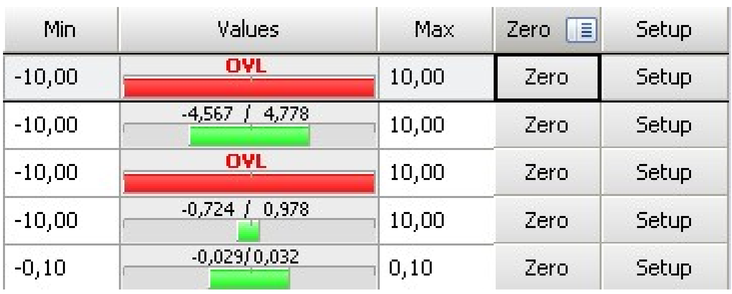


if this happens, *check* your *sensor* and / or *select* another *input range* (in Channel setup).

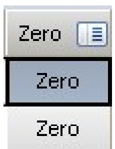
By clicking on the PHYSICAL VALUES caption, a drop down *which values are shown* is displayed. Normally Display measured values is selected. In the case when we have bridge amplifiers, there are additional option to display bridge balance or shunt calibration values.

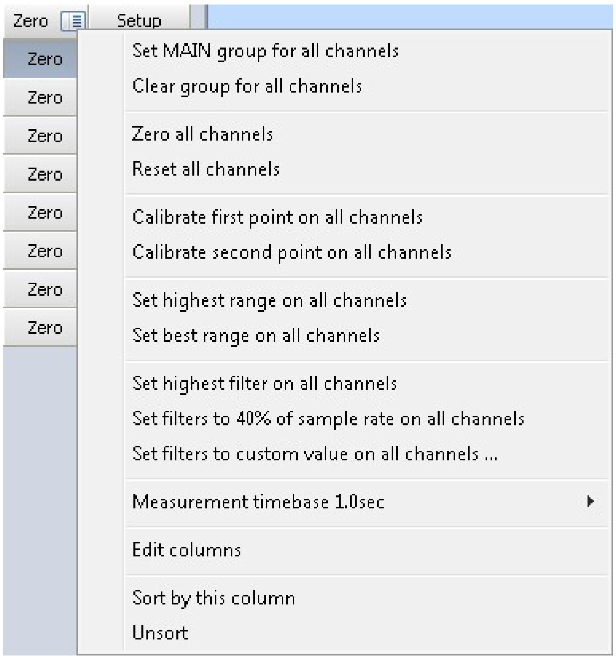
2.3.1.7 Input zeroing

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 offset* to *all* channels.

- **Calibrate first point on all AUTO channels** option allows to *calibrate 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.

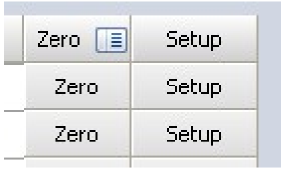
2.3.2 Channel setup form

In *Channel setup* window we can do **settings** and **calibration** for selected input channel.

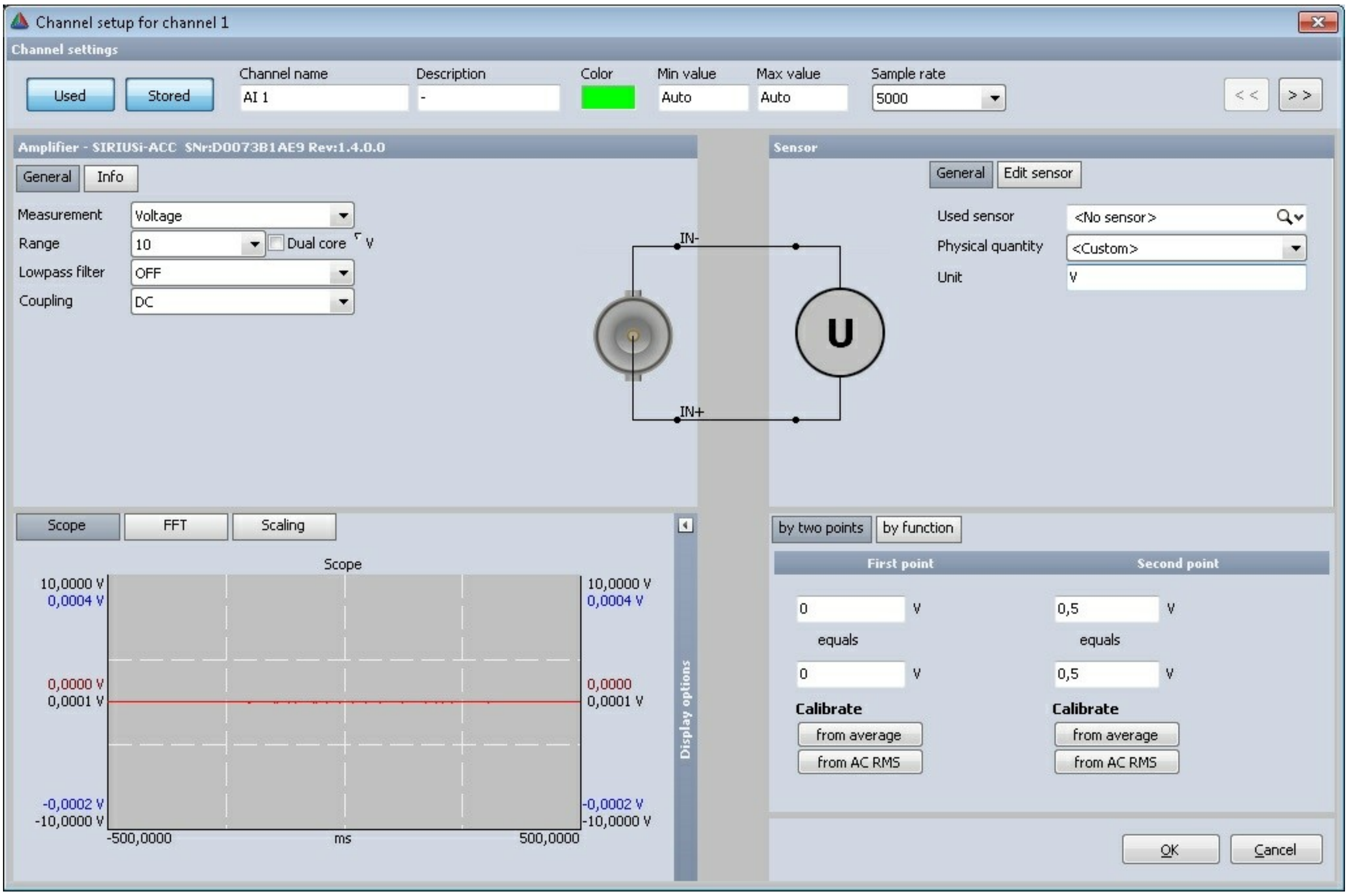
General settings	you can *enter* on the left side of **General** part the **Channel name**, the **measurement value**, **Units** of measurement, *define* **Color** (click on the color bar to change the color for this input), **display range** - Min value, Max value and **Sample rate divider**
Sensors settings	*define* and *set* sensors information and data in *sensor database*
Amplifier settings	*define* and *set* amplifier settings: *Non-programmable*, *series* and *special programmable modules*

Scaling — perform **scaling** and *manual* or *automated* **calibration**; set a **dynamic representation** of your signal

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 *Channel setup for channel n* dialog box for this *input channel* and this setup dialog will appear:



At the top-right corner of the dialog, you can change from **General** information (also *see* → **General setting**) to **Sensors** to *select* pre-defined **sensors** for *scaling*. Please find out more about this function in → **Sensors setting**.

At the top-left part of screen left corner of the dialog always **Amplifier settings** are displayed.

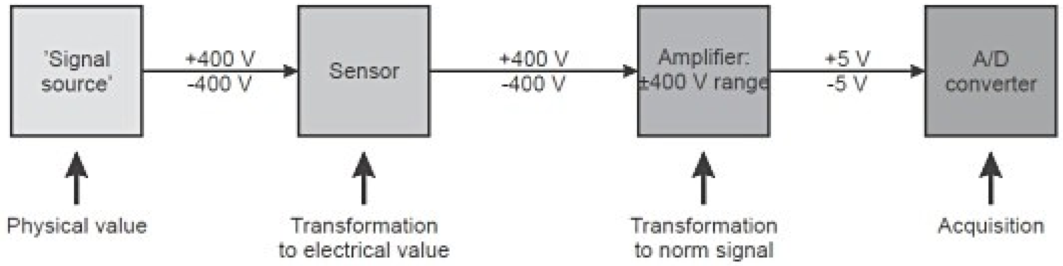
On bottom part of screen **scaling** and **calibration** can be performed.

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

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*) w ill be measured w ith cables (= '*sensor*') connected to a high voltage module (= *amplifier*).



2.3.2.1 General settings

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

- **Channel name** and **measurement value**
- **Units** of measurement
- *define* **Color** (click on the color bar to change the color for this input)
- **display range** - Min value, Max value

Measurement value and unit

Measurement value can be F or Force, Voltage, etc and are entered as *text* in lower field of Channel name.

Unit of measurement value: N (newton), kg, V, etc are entered as *text*.

Navigation keys are same as in other *MS Windows* application.

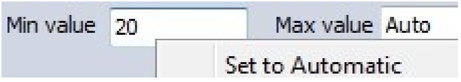
Min / Max value

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 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.*

Sample rate

The **Sample rate** can be used to '*reduce*' the sample rate for *each* channel.

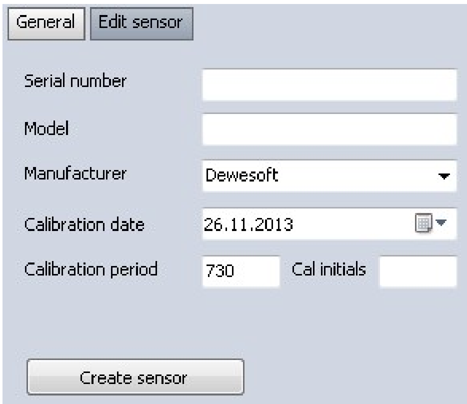
for details about *sampling rate* also *see* → Reference Guide → **Basics of A/D conversion**

2.3.2.2 Sensors setting

In this Sensor part of DEWESoft *Channel settings* screen you can *select* the **Sensor group** and **Sensor** (both from Sensor database which include also sensors ***scaling***, ***offset*** and ***correction***) previously entered in the **sensor editor**.

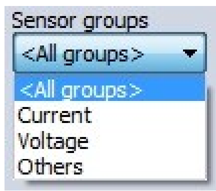
for details about ***Sensors editor*** *see* → User Guide → **Sensor database**

Open the Edit sensors tab and screen like this appear:

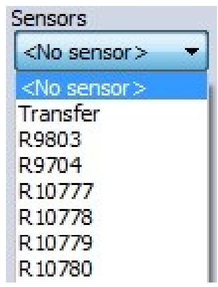


The upper left side shows the Sensor predefined values as well as additional values which can be defined in setup. The *predefined* values are Scale, Offset, Description and Recalibration date. If calibration is *overdue*, the date will be shown in red.

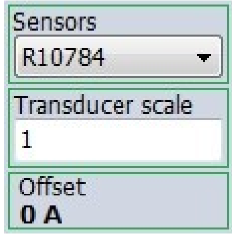
Choose from Sensors group selection list appropriate **sensor group** in sensor database.



Choose from in Sensor field drop down list the *serial number* of the **sensor** in sensor database.



Nothing much happens, but note that we *can't enter the normal* ***scaling*** or ***sensitivity*** anymore. On lower part of *Channel setup for channel n* we can see only *graphical representation* of signal without *scaling* and *calibration* sections.



Additional factors are **Transducer scale** and **Custom offset**. These two factors must be *enabled* in Sensor editor when defining the sensor and *additional* fields and one button appear in *Sensors* tab sections:

- **Transducer scale** is additional scaling factor *multiplied* with sensor scaling. It is used mainly with transducers like voltage probes where additional voltage converter is used in front of the primary transducer. We can use this field for *reversing the polarity* of the sensor by entering a value of -1.
- **Custom offset** can also be defined. For sensors like *quarter bridge strain* sensors the offset can be defined when a part where the sensor is installed is mounted on the vehicle - that means just *before* the test. Therefore only a scaling factor is defined with the sensor and the offset is defined with each *individual setup* or data file. This value appears also on left Offset field.
  As part of Custom offset setting also **Set zero** button appear. Sensor zero is equal to Set zero button in *Scale by function*. It will set the current average value as the offset of the measurement, therefore zeroing the current value.

The normal scaling is not available when the sensors are used, but the *module* **ranges** are available.

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 have 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 Data menu **Sensor editor ...** and the sensor *editor* will open.

for details about ***Sensors editor*** *see* → User Guide → **Sensor database**

2.3.2.3 Scaling

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

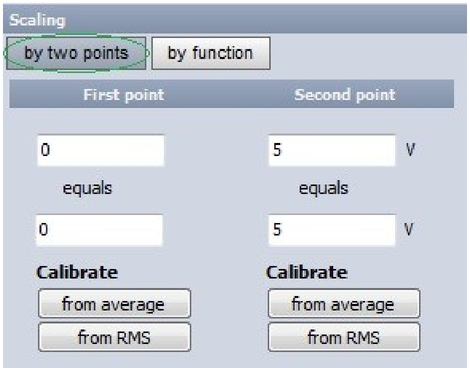
The scaled "*physical-engineering*" input value is needed for *signal display* reason or because to have *different units* as the "*electrical*"input value.

The *bottom-left* 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.

Scaling - Calibrating on input 'by two points'



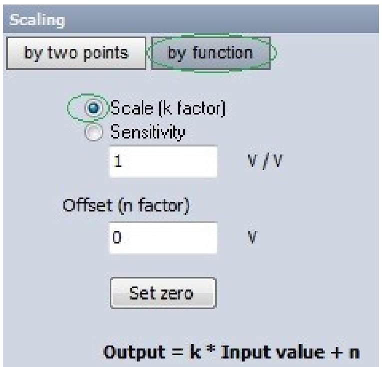
In this example we enter scaling factor so that the input channel on First point: 0 real volts *equals* 0 N, and on Second point: 100 mV (electrical units) *equals* 1 g (engineering units - ***Acceleration***).

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 useful hints about ***scaling factor*** by different *Measurement application see* → DEWESoft Tutorials

Scaling - 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 above example we calibrate the system so that for channel this factor is 0.02.

Above *shows* your basic algebraic formula:

$$y = k * x + n \quad (\textit{physical value} = \textit{scale} * \textit{measured value} + \textit{offset})$$

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

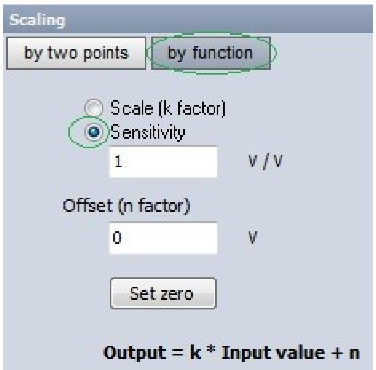
There are *two* ways of entering the ***value***:

- **Scale (k factor)**; example above: k = 0.02 N / mV

  Scale factor is the factor which defines how to *scale* the electrical signal to deduce the physical values.

- **Sensitivity**; same example on right: k = 50 mV / N

  Sensitivity is the factor which is usually used in sensor *calibration sheets* and *defines* the electrical output per physical quantity measured by the **sensor**.



Use the **Set Zero** button 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 $y=k*x+n$, to perform simple *offset*.

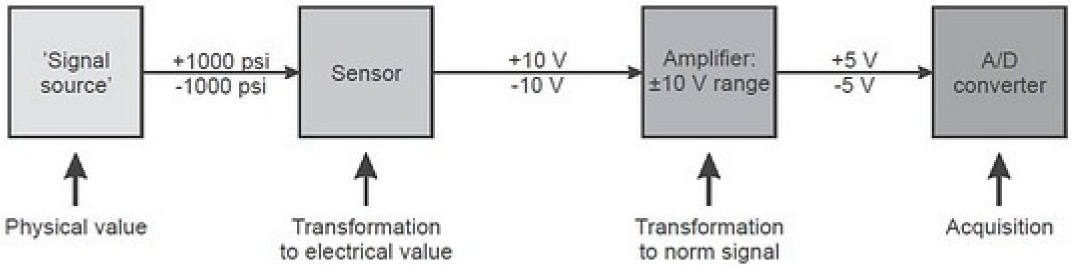
By *multiplying* °C by **1.8** as Scale and then *adding* **32** as an Offset, the same result is achieved.

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 Tutorials

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.

2.3.2.4 Calibrate

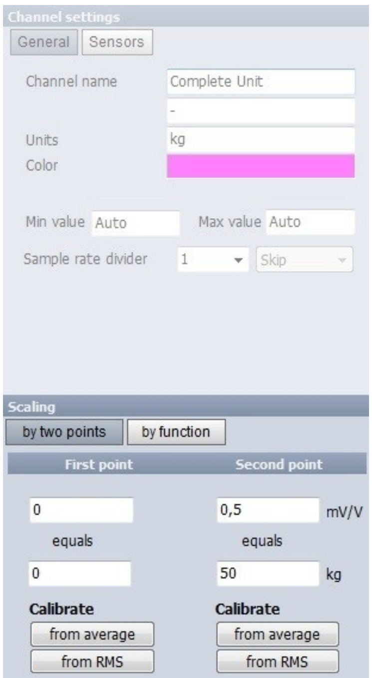
The ability to *quickly* **calibrate** directly to DC and AC *signal sources* is a great advantage of **DEWESoft**, and will save you much time, and ensure better and more reliable *data recording* long into the future.

NOTE: *this procedure is only available with 'by two point' scaling!*

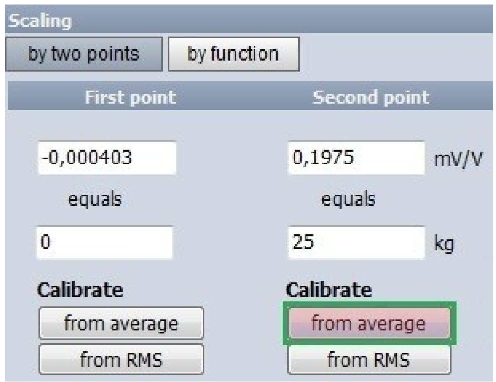
Calibrating to signal sources

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 (25 kg) as our known *second* point.

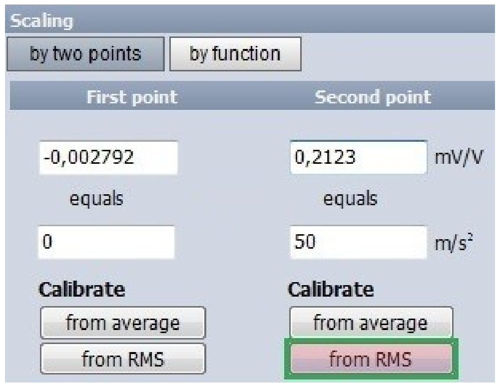
The first thing to do is simple enter those two values in these boxes as shown:



Now, we need to *apply* the known *zero* (0 kg) load. When it stabilizes, *press* the left-most **from 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.



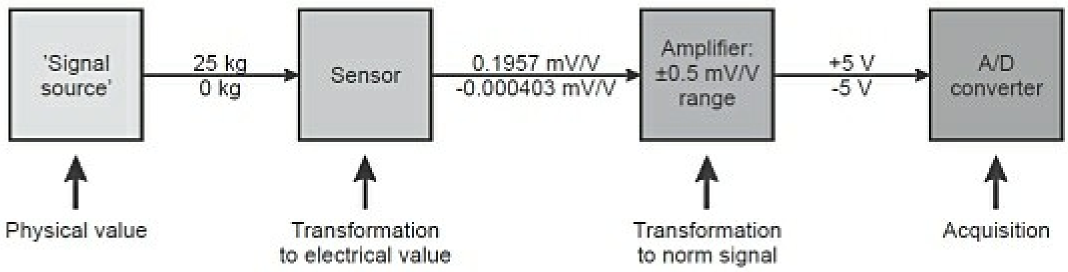
Now *apply* the known 25 kg *load*, let it stabilize, and *press* the right-most **from average** button (*see* picture on right). DEWESoft is now *perfectly* **calibrated** to your *signal* based on the known loads that you *input*. Note in the screen picture above that DEWESoft has *automatically* 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:

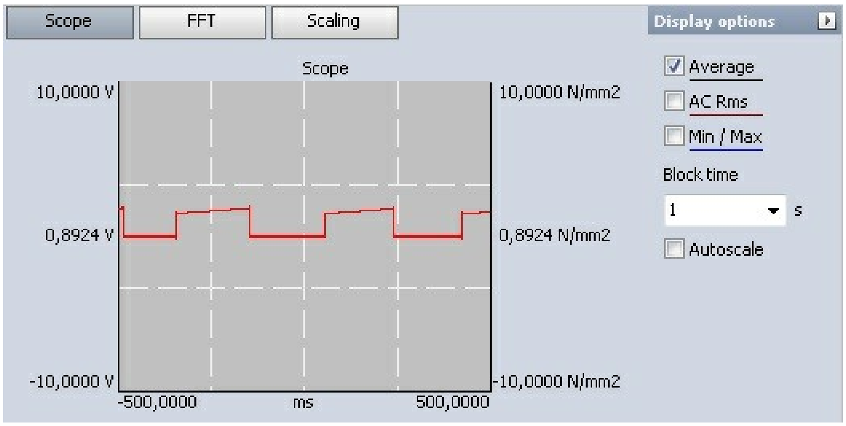
Defined ***weight*** from 0 to 25 kg (= signal *source*) will be measured with an *uncalibrated* **strain gage** sensor (= *sensor*) connected to a DAQP-BRIDGE module (= *amplifier* with 0.5 mV/V *range*).



for useful hints about ***Calibrate*** by different *Measurement application see:* → DEWESoft Tutorials

2.3.2.5 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 **dynamic graph**. On a dynamic graph your *signal* are represented:

- 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 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 shows 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.

2.4 Analog out

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.

Analog output channels, which will be *used* during measurement, are *set* in DEWESoft *Setup screen* on Analog out tab. Any *analog output* card from *National instrument* can be used as the hardware. After DEWESoft **analog output Hardware setup** (this procedure is to perform *only* by installation *new* or *changing* output hardware) the **Analog out** tab on DEWESoft *Setup screen* must be selected to *set all* **analog output channels** - **Function generator**; this screen displays three main part:

Recording setup with ***Data file options*** to set target of data storing *see* → User Guide → **Recording Acquired Data**

for information about ***New***, ***Load*** and ***Save*** the *channel Setup data* *see* → User Guide → **Data management**

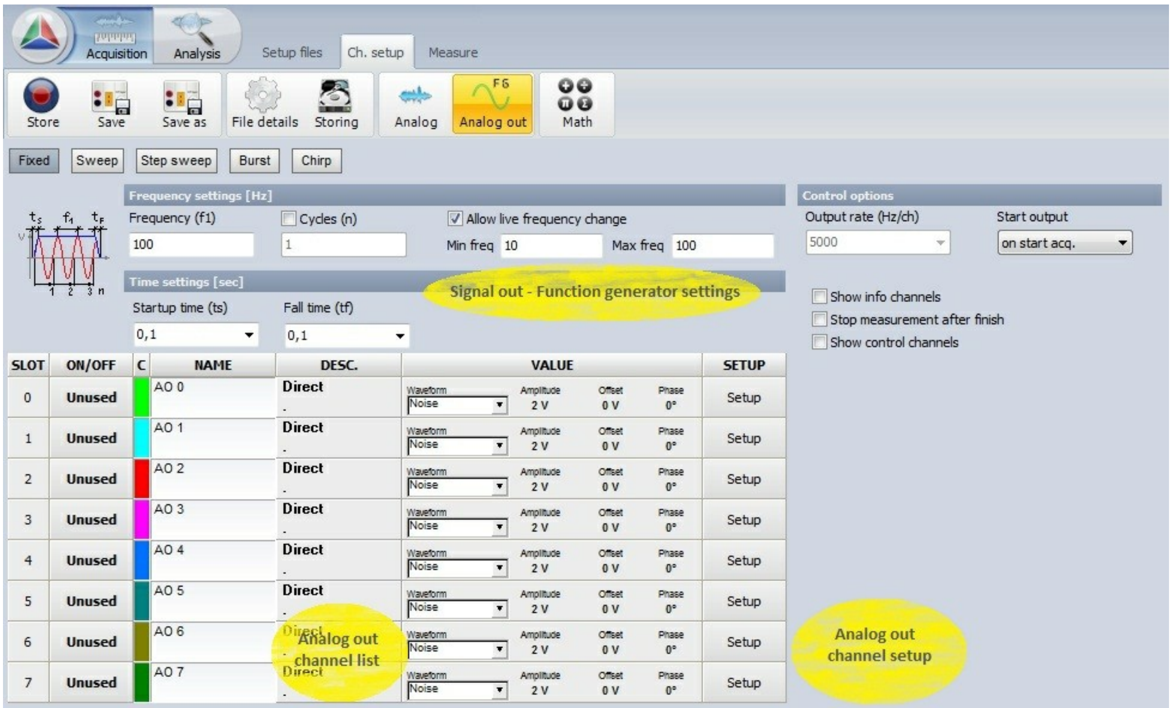
Function generator with all *definitions* and *settings*:

- **Operation modes** - ***fixed*** frequency, ***sweep*** or *stepped sweep*, ***burst*** or ***chirp***; for each mode many functions are available like *frequency*, *timings*...
- **Signals Waveform** - for each channel individually we can decide on ***waveform type*** (*sine*, *triangular*, *rectangular*, *saw*, *white noise* or *trigger*). *Amplitude*, *offset* and *phase* can be defined for each channel as well.
- **Control options** - to define ***Output rate*** and ***Start/Stop*** output condition for the *analog output*

Analog out channel list display *Analog OUT* ***channel list*** with information and settings in columns:

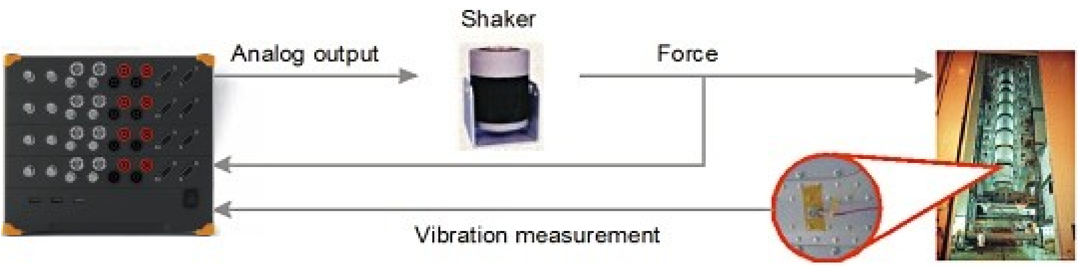
- **Slot** - indicates the *number* of the channel; it always starts with 0 and counts up to the *maximum* number of *available* output channels
- **On/Off** - *activate / deactivate* output channels; *click* on the button to activate or deactivate channels
  - displays the state of the channel: Unused (disabled, **no signal output**) or Used (*active*, **signal output**)
- **C** - channel *color* selector
- **Name** - channel *name*
- **Desc** - *amplifier settings*
- **Value** - output signal *type* and *values*
- **Setup** - *input settings* and *calibration;* to call **Channel setup** window

Setting during Measurement - how to display *Analog OUT* ***channels*** and set their parameters as internal channels



Example:

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.



2.4.1 Operation modes

In **Analog out** tab of DEWESoft *Setup screen* can be *set* for particular **operation mode**:

- **Frequency setting [Hz]** and **Time setting [sec]** - *for explanation click on operations mode name in table below*
- **Control Options**

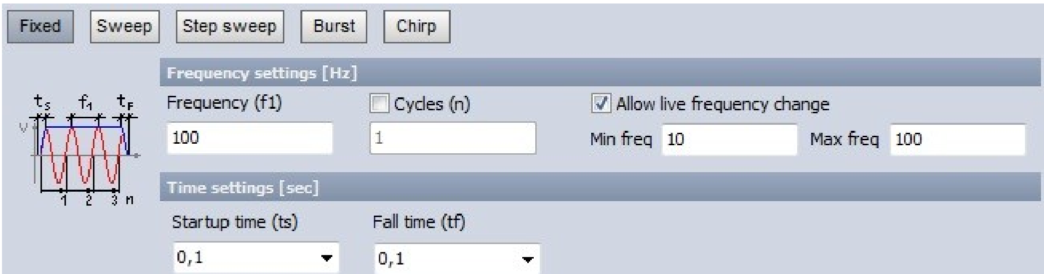
The function generator offers five *different* **operation modes**, which are *set* for all **analog output channels**:

- **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

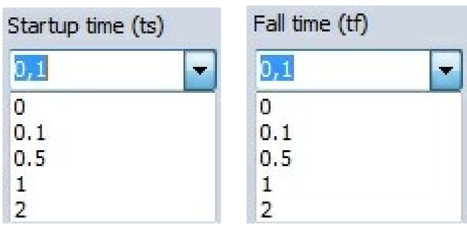
Fixed mode *outputs* the selected **signals** with a constant **Frequency (f1)**. Output frequency is the *same* for *all* the channels and is *defined* in the setup.



It is also possible to define a *number* of **output cycles** if the Cycles (n) checkbox 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* selected frequency *range* (Min frequency and Max frequency).

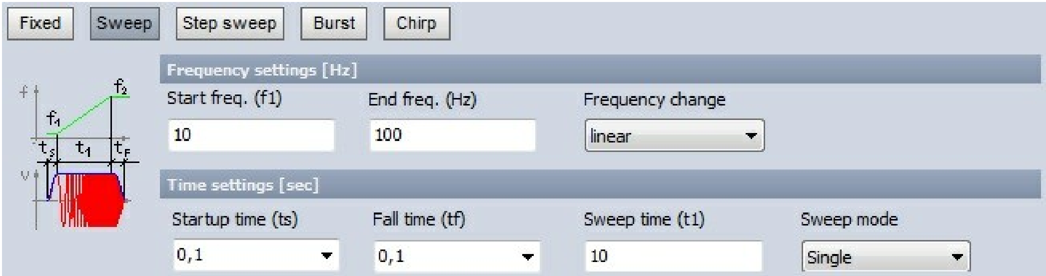
To avoid *jumps* in the signal a Startup time (ts) and a Fall time (tf) at the *beginning* and the *end* of the **signal output** can be *set* with selection from drop down list.



Applications: fixed mode is used if a defined waveform should be *output without changing* the *frequency* or *amplitude automatically*. Such 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), a End freq.(HZ) - f2 and a Sweep time (t1).



Startup time (ts) and a 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 drop down list.

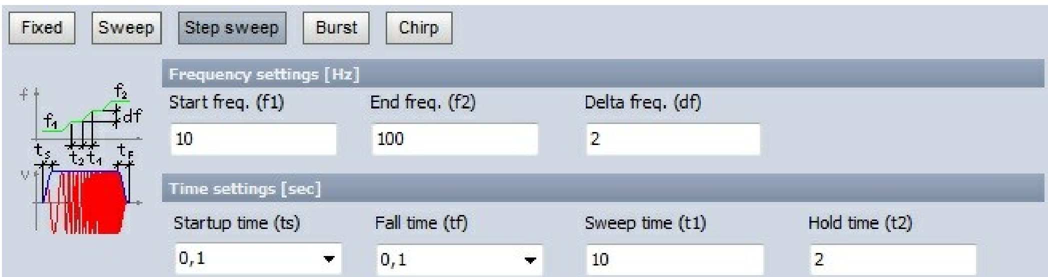
Two Sweep modes are selectable from drop down list, **Single** and **Loop**. In *single* mode you will get a *single* sweep output only, at *loop* mode the sweep is output *continuously*.

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

Step sweet

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 *transfer function* can be achieved with smaller steps. Even though this procedure is the *slowest* to finish the test, it provides *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 take results of modal test.



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

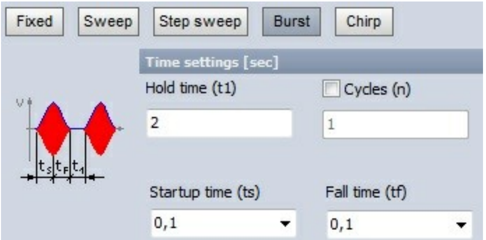
Sweep time (t1) *defines* the *sweep* **duration** to the *next* frequency *step*, Hold time (t2) *sets* the **duration** where the frequency *keeps fixed*.

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

Applications: 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.



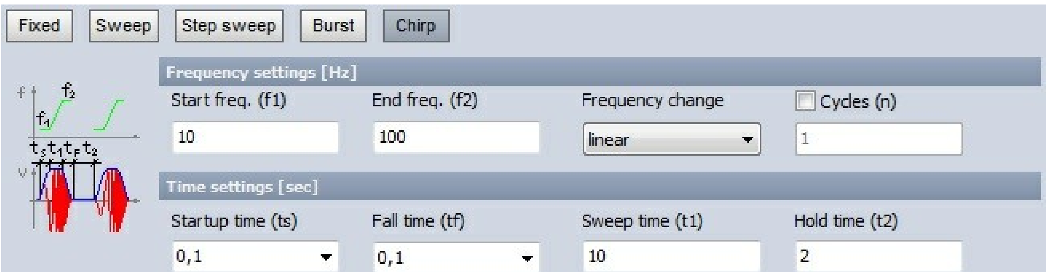
Within the *startup* and *fall* time you get **noise output**.

If the Cycles (n) checkbox is *enabled*, it is possible to set the **number of burst** to *output*.

Applications: 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*.

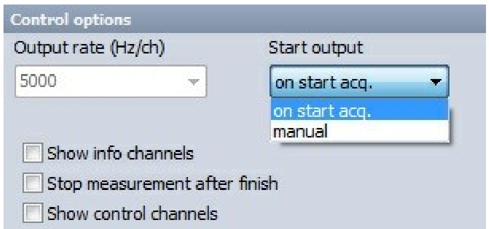
The *time settings* also contain Startup time (ts) and Fall time (tf) which *defines* the **ramp** at the *beginning* and the *end* of the chirp, Sweep time (t1) - the chirp **duration** - and the Hold time (t2) *between* two chirps.

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

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

2.4.2 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, or
- 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* then the highest *output frequency* to *avoid* harmonic distortion.

If Show info channels is *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*.

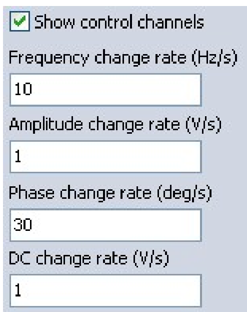
The **start** and **stop** of *generation* can be either **on start acq.**(uisition) or **manual**. Each operation mode carries *different* manual control:

- at **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 *manual* to the *next* frequency

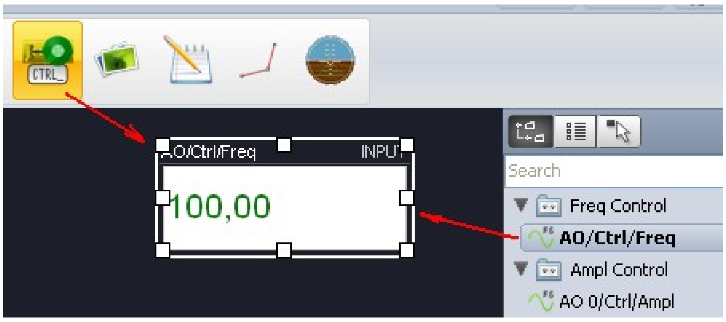
The **start** and **stop** of *generation* can be with a half sine window, so the amplitude will not rise from 0 to full amplitude *immediately*, but within a *certain time* which ensures that the systems under test don't suffer from *impact shock*.

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.

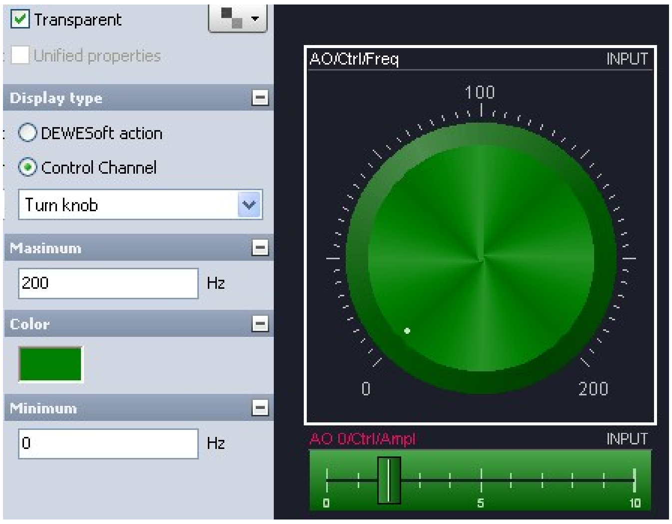
If this option is chosen, then the *rates of change* must be defined first in the channel setup screen to define maximum change when the slider is changed.



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.



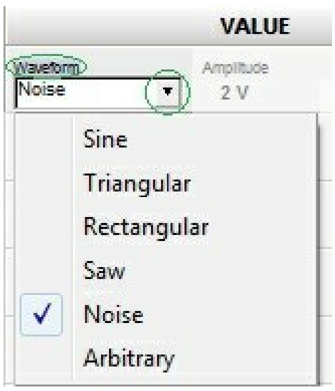
2.4.3 Channel value

In **Value** column you can *set all* **analog output channels** - *signals individually* for each channel:

Waveform type

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

- *Sine*
- *Triangular*
- *Rectangular*
- *Saw*
- *Noise*
- *Arbitrary*



Also Amplitude, Offset and Phase can be set *individually* for *each* channel as well.

If you need *precise* entries move cursor over the numeric field, cursors appearance change to , simply *click* and *enter* the desired *value directly* with your keyboard:

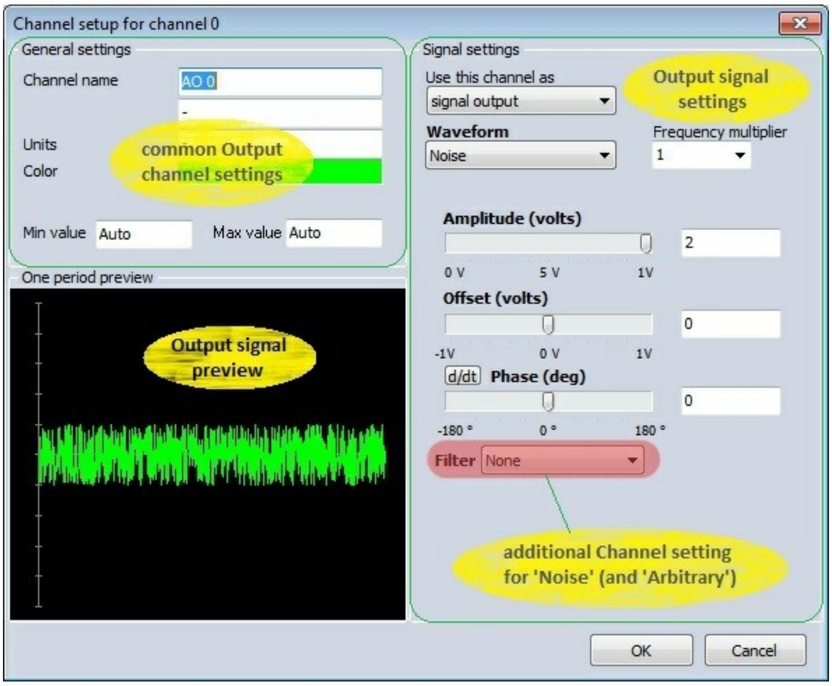
Amplitude, *offset* and *phase* can be set also in **Channel setup** with slider or enter the desired value directly.

2.4.4 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**

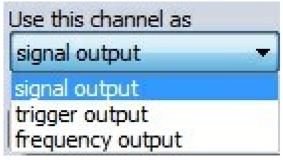


Signal settings

The *upper right* area contains two elements of Signal setting:

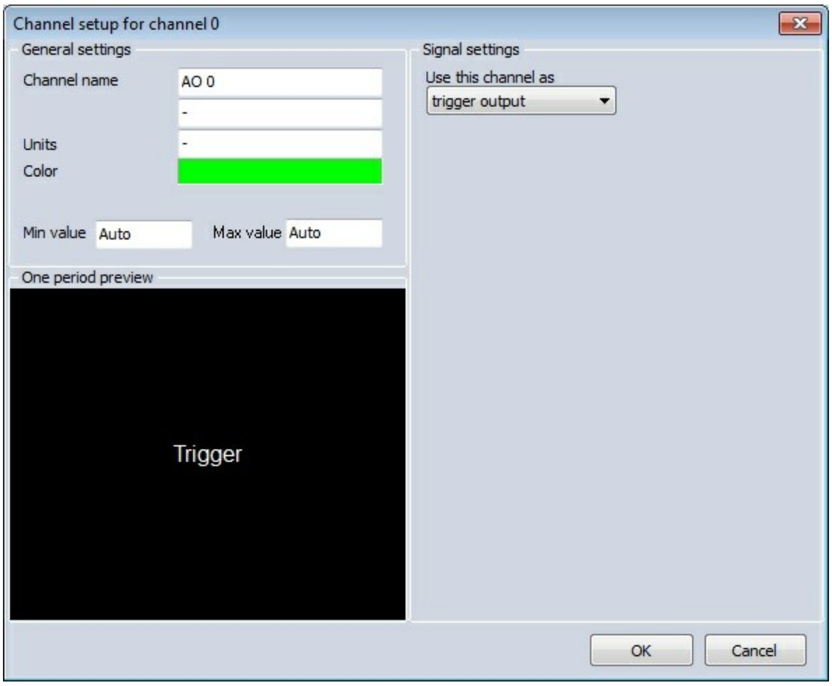
signal usage selection in Use this channel output as.

The *channels* can be used as:



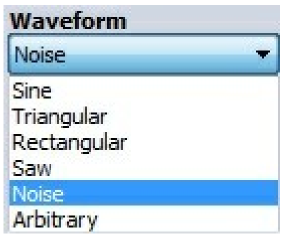
- **signal output** for the selected waveform is *default*, or as
- **trigger output**. *Each* channel can be set to trigger output instead of signal output

Often it is intended to trigger the *acquisition* on *measurement* side when *certain criterion* is *met*. When the sweep is active, the trigger value *goes up*. It is up at step sweep, at 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 finally contains the *sliders* to *change* **Amplitude**, **Offset** and **Phase**. For *input* precise entries each this value is next to the slider also *numeric field* → simply *click* in the field and *enter* the desired value directly with your keyboard.

The phase setting has additional option - not to define fixed phase, but variable phase. This can be achieved with pressing **d/dt** button near **Phase** caption (this caption after press this button change to Phase change (deg/s).

Variable phase can create for example *stroboscope effect*. One *sine* wave with *fixed* phase can drive for example a *shaker*, while we can use the other TTL *rectangle signal* with *variable* phase for *camera trigger* to create a *stroboscopic video*.

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

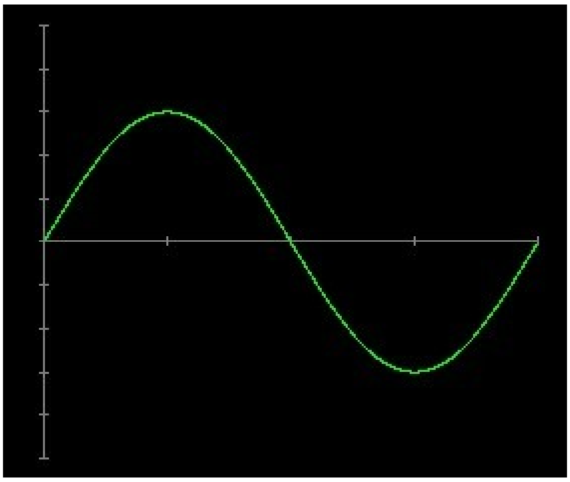
Additional signal setting

For **Arbitrary** and **Noise** type of *waveform* are beside sliders and numeric fields also **additional signal setting**:

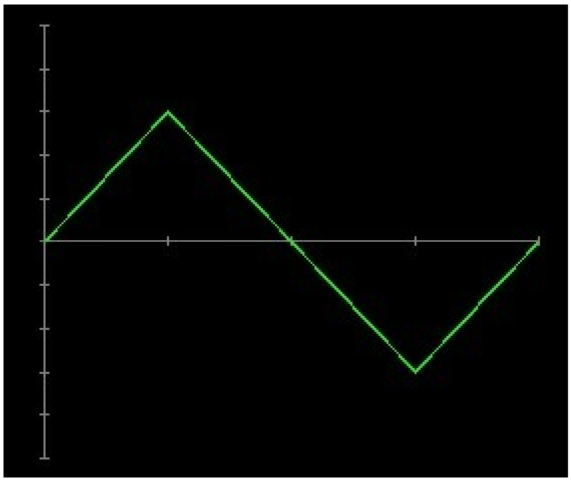
- for **Arbitrary** type of *waveform* there are sliders and numeric fields in Signal tab, *additional* is also **Points** tab for this **signal setting** → *see* → **Arbitrary - Additional settings**
- for **Noise** type of *waveform* bellow the sliders there is *also* an additional **Filter signal setting** → *see* → **Noise - Additional settings**

2.4.4.1 Waveform type

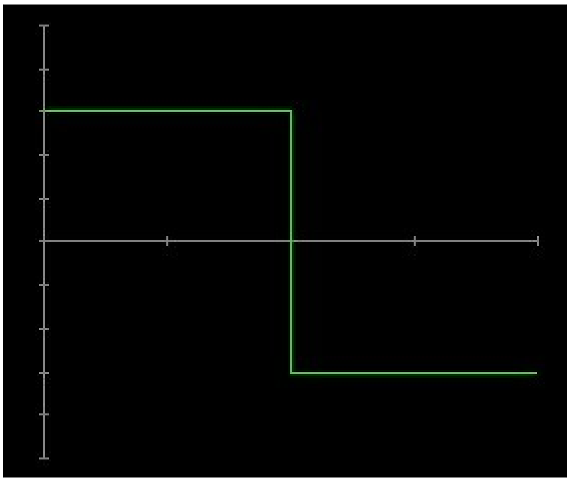
In DEWESoft software the **waveform types** are:



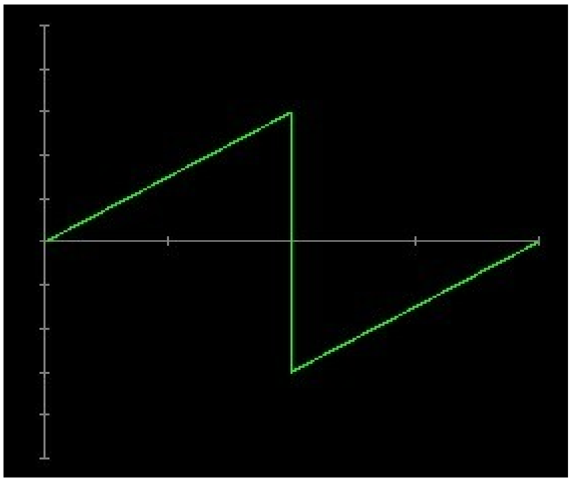
SINEWAVE



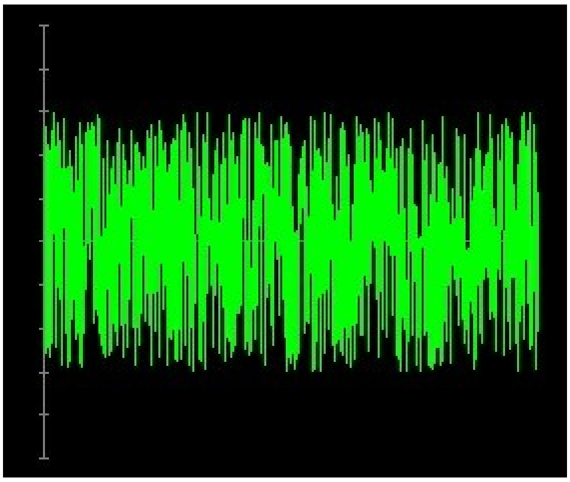
TRIANGLE



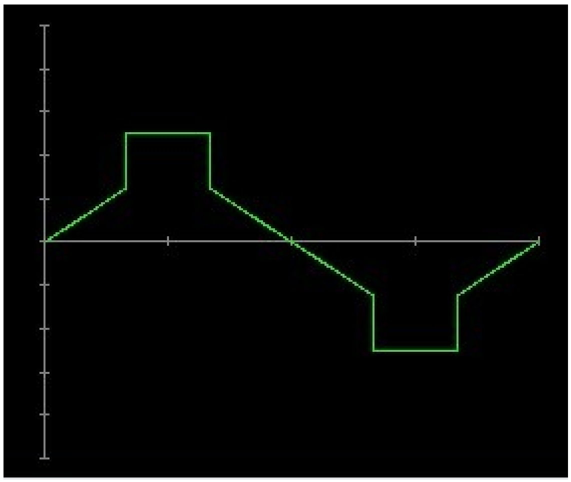
RECTANGLE



SAW



NOISE



ARBITRARY

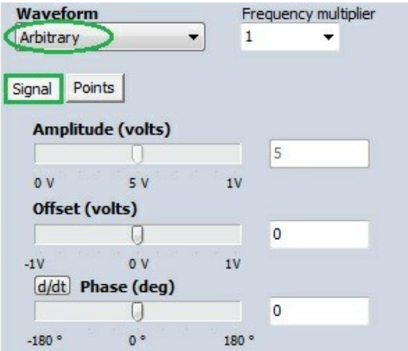
Waveform types can be *set* in:

- **Waveform** of Value column in Analog out tab or
- **Signal setting** of analog *output* DEWESoft *Setup screen* in **Analog out** tab

2.4.4.2 Arbitrary - additional settings

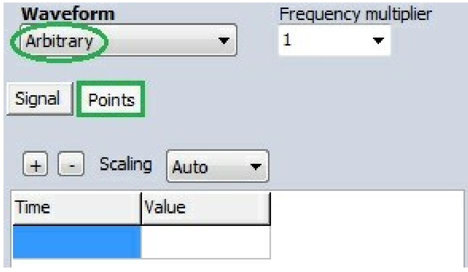
The **arbitrary** waveform makes <u>definition</u> of special waveforms very easy through two tabs:

Signal tab

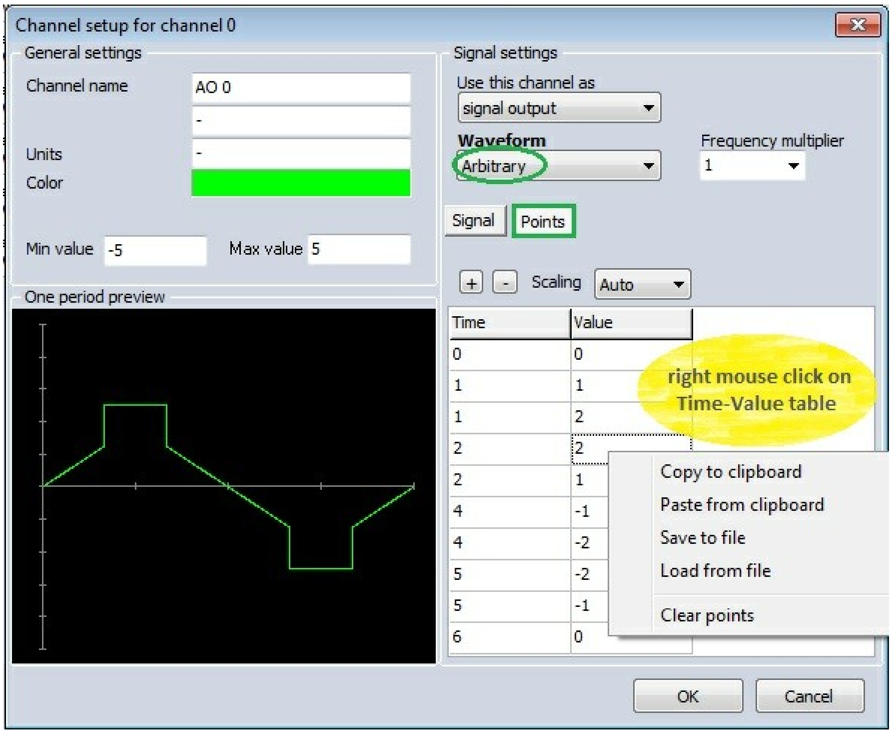


All parameters, like **Amplitude**, **Offset** and **Phase**, can be **set up** as already known.

Points tab



The *waveform* itself is <u>defined</u> in a table, which can be displayed in the Points tab → *see* below



There are *three* ways to *get* a **signal**:

- The signal values can be *entered* <u>directly</u> in the table.

  Press [+] or [-] buttons to <u>add</u> or <u>remove</u> *rows* in the table and *enter values* for Time and Value (*amplitude*).

- Paste from clipboard

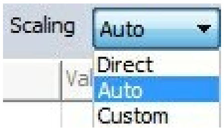
For example from *Excel* to DEWESoft. If you copy from another programs like *Word* or other *Editors* be aware to use the tabulator as *separator* between the columns.

- Load from file

  The data have to be provided as 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:



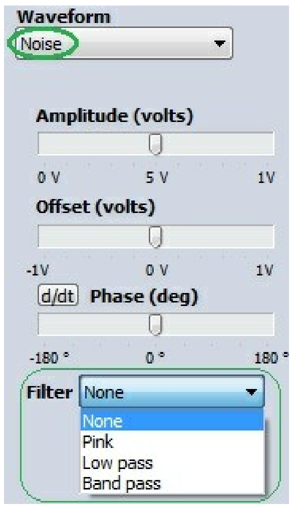
- **Direct** inserted values are taken directly and can *never* be *changed*
- **Auto** inserted values are scaled with the *amplitude value*, like the other waveforms (*even* online)
- **Custom** inserted values are scaled by an *scaling factor*. This factor is not *changeable* during the signal output!

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

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

2.4.4.3 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:



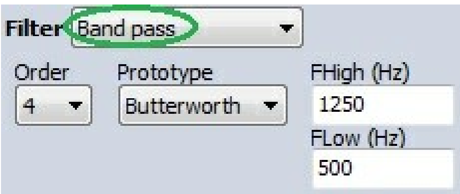
- **None** - no filter is used - the output signal is **white** noise
- **Pink** - a filter is used to obtain **pink** noise

- **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

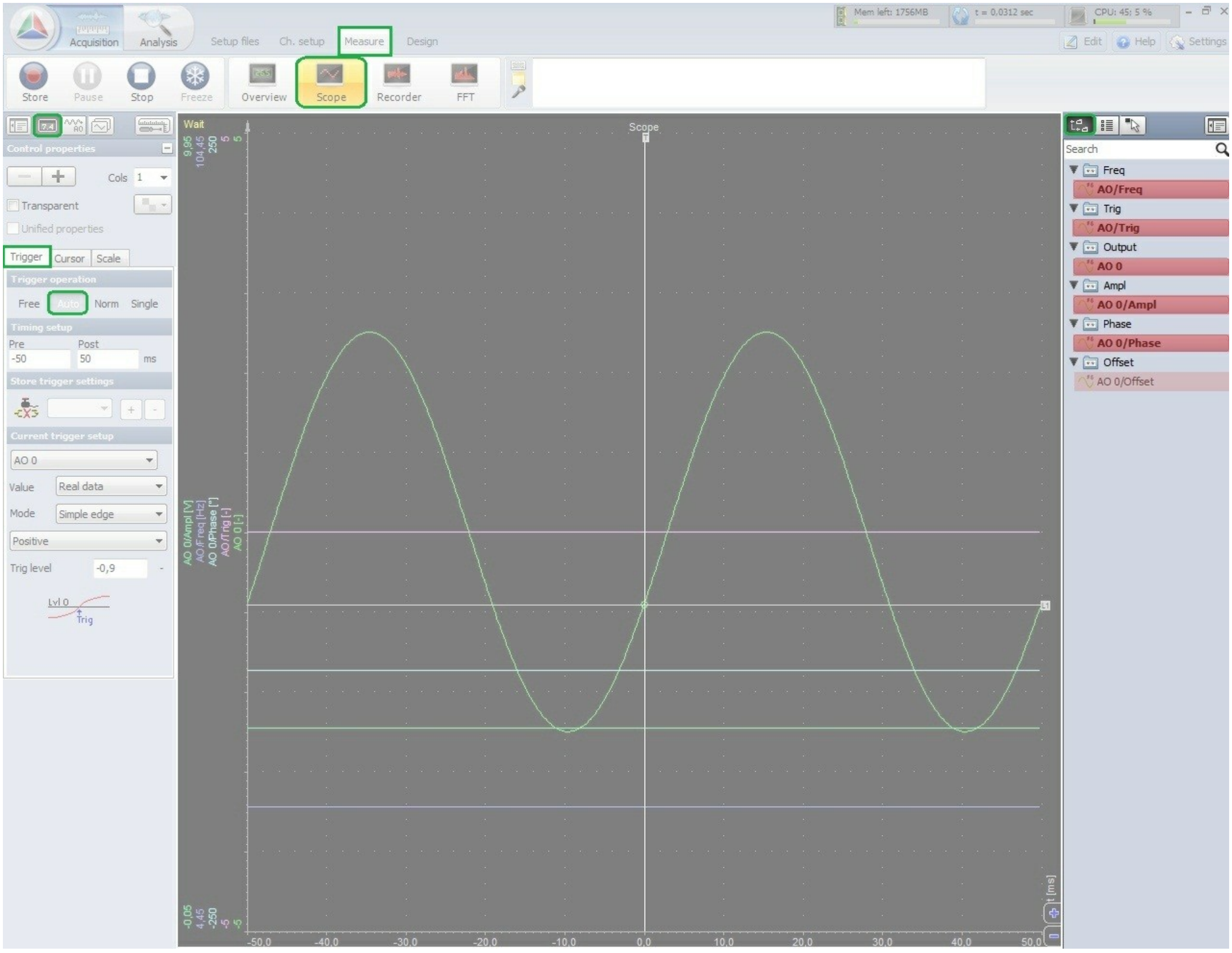
Order and Prototype can be selected from drop down list:

for information about ***Order*** and ***Prototype*** *see* → User Guide → **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.

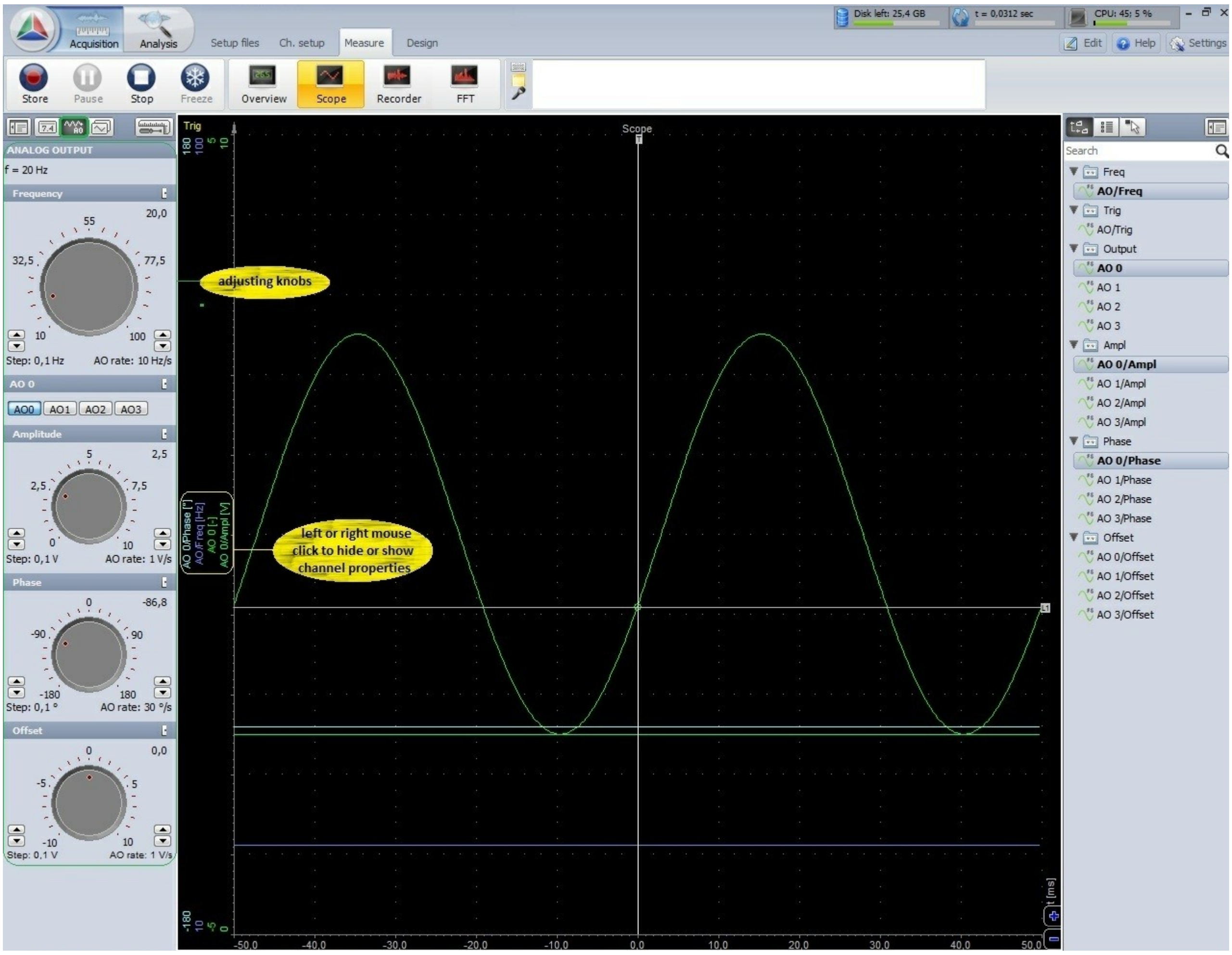
2.4.5 Setting during Measurement

All analog *output channels* (including their parameters) are available in DEWESoft as *internal* channels also (if *hard sync* option is set in *hardware* setup). Therefore you can **display** them together with your *other signal sources*.

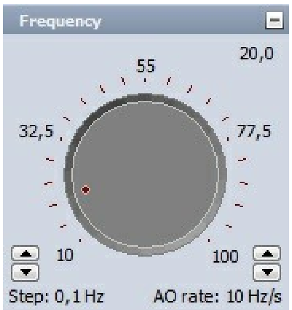


If analog outputs are activated in DEWESoft, you can see a ***channel list*** on the right side. The list contains all the channels that are currently in use and can be displayed or hidden simply by clicking on them.

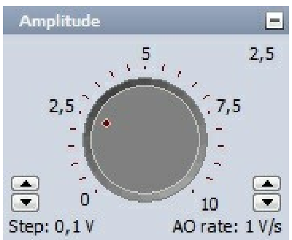
When you press the  button, the instrument will show the following controls:



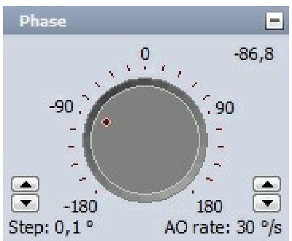
Use the **adjusting knobs** to change the parameters Frequency, Amplitude, Phase and DC offset:



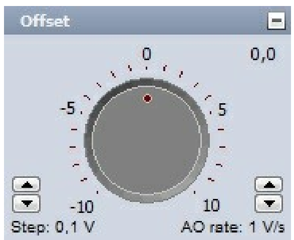
FREQUENCY



AMPLITUDE



PHASE

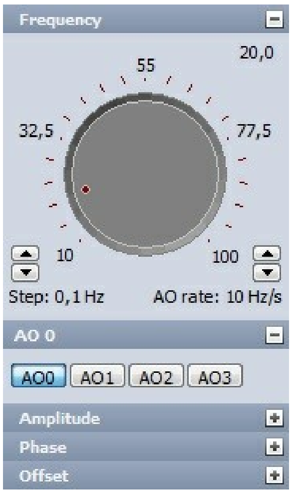


DC OFFSET

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*.

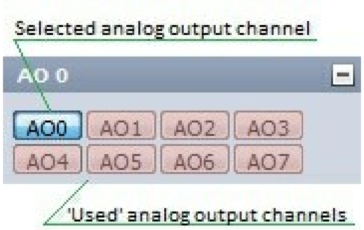


Example: all controls knobs are minimized

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:



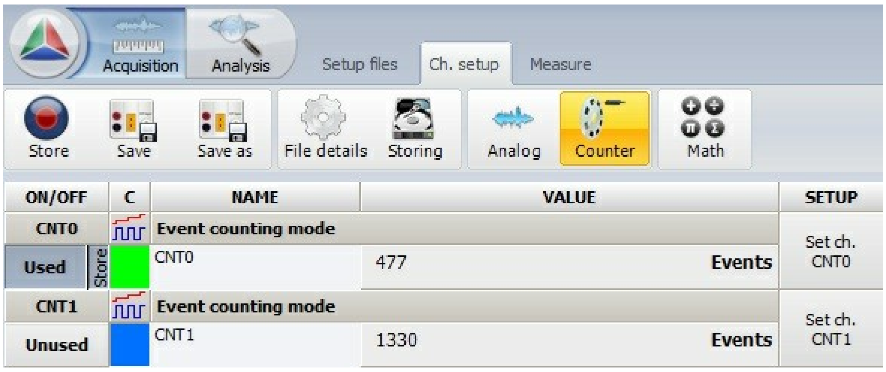
As already described, all parameters are *available* also through *control channels*.

2.5 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.

The Counter hardware can be *activating* on the DEWESoft System menu → Hardware setup... → **Analog** tab by set the required *counter channels* (CNT chnls column) in Device information list.

When you select a **Counter** tab on DEWESoft *Setup* screen, **Counter** *channel line* will appear on lower part of screen:



Counter channel list

First let's look at the columns that are shown in *Counter channel list*, and describe what each is for:

SLOT — counter channel *input* slot - *number* of the counter

ON/OFF — *activate / deactivate* counter channel - press the **Unused** button - it will change now to **Used** - to set the desired counter channel ***active*** for *counter data acquisition* and they are ***available*** in *all instruments*

C — channel *color* selector - this color will carry through the text and graph representations of *this* channel *throughout* all DEWESoft procedure screens

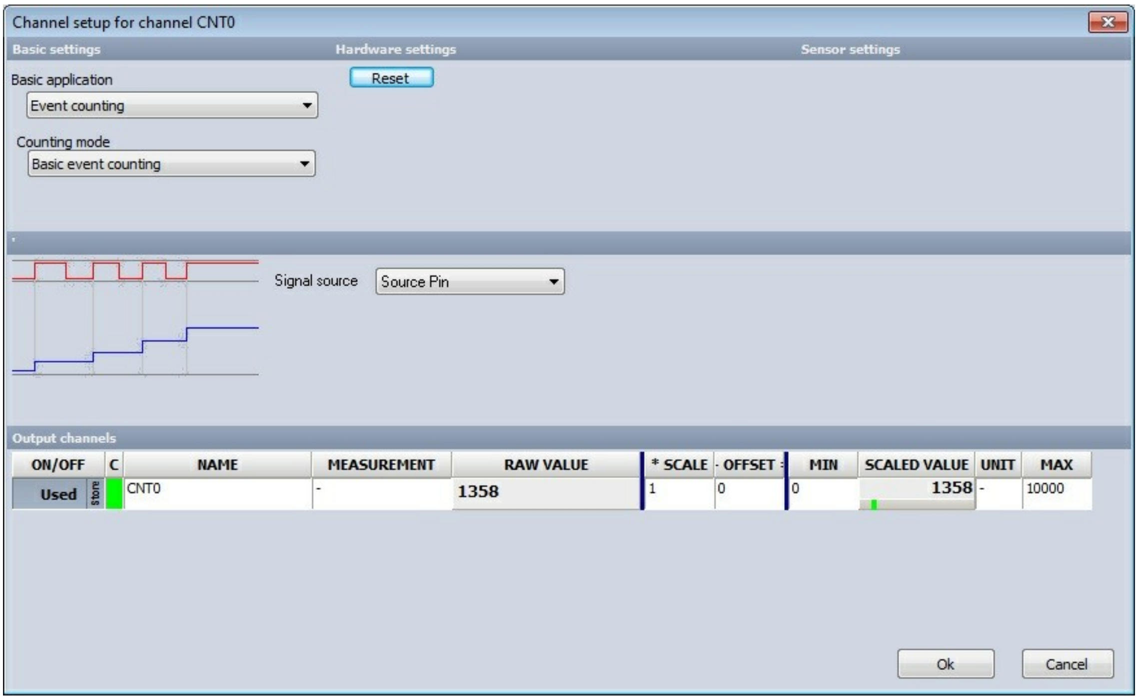
NAME — counter channel *name* - just click into the **NAME** field and you can *enter* the new counter channel ***name*** as usual text

VALUE — *current* counter channel *state*

SETUP — counter channel *settings* (*see* below)

for information about ***Slot***, ***On/Off***, ***C*** and ***Name*** column *see* → User Guide → **Channel setup**

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

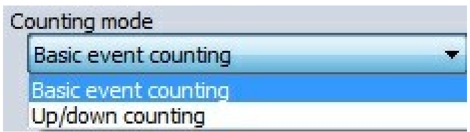
  are similar to the *analog input* settings and contain the Sensor groups, Sensors and Description: → *see* → **Analog (in) Channel sensor setting**

Hardware parameters

Set and number of parameters (settings fields) are dependent on the *used* hardware.

- **Counter mode**

  Option of this field depend 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 parameters for Orion expansion CNT type are:

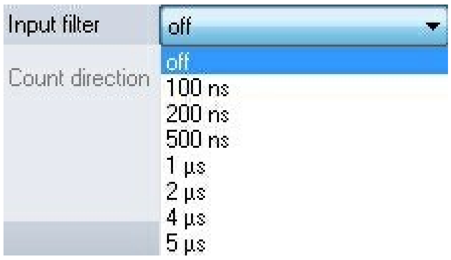
- **Reset on start measure**

  Check this box to counter **reset** on start of measurement.

- **Reset** button

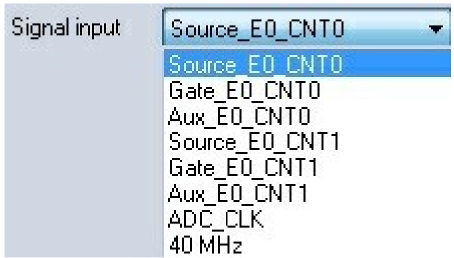
  With press on Reset button we can **reset** actual counter to zero.

- **Input filter** button

  The filter is 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 can be selected in this field from drop down list value of input filter in ns or off for not a filter. 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 drop down list in this field.

- **Count direction**

  We can select **Count direction** either to count up or down. This value can be selected from drop down list in this field.

- Signal **inv**erting

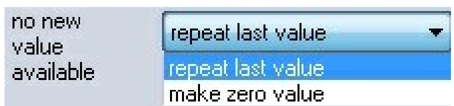
  When the *normal state* is *high*, sometimes is nice to invert the signal by choosing the **inv** check box beside signal field.

  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**

  In 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.

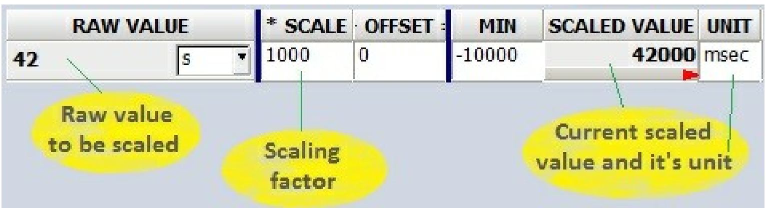
  If we choose to repeat the last value, then the *same* value will be added *until a new transition* is made. Alternately, we can select to output zero value when *no new* value is available, so we will have *only spikes* at the points of *new* data and the rest of data the value will be zero.

Scaling

Finally the Scaling area can be used to *adapt* the *sensor* to your *system* with *enter 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 current** button.



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 lower part actual ***equation*** for Output value are displayed.

2.5.1 Event, gated and up/down counting

Event counting is one of the *simplest counter* operations.

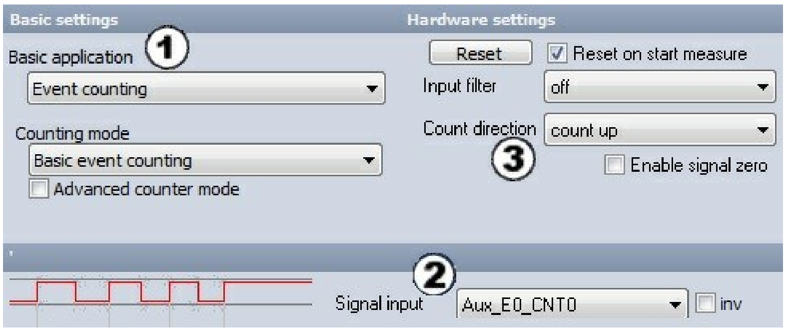
Required hardware	DEWESoft USB
Setup sample rate	At least 1 kHz

There are two special modes of event counting available:

1. gated event counting, where events are counted only when a gate signal is *high* (only available on Orion counters)
2. up/down counting, where the events are counted up when the *gate is high* and down when the *gate is low* (available on Orion, NI MX, NI E series and DT cards).

for hints about ***Event counting setup*** and ***measurement*** see → DEWESoft Tutorials

Simple event counting



1. The **Counter mode** 'Events' ① are 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 ② as the input (e.g. AUX_CNT0 above).

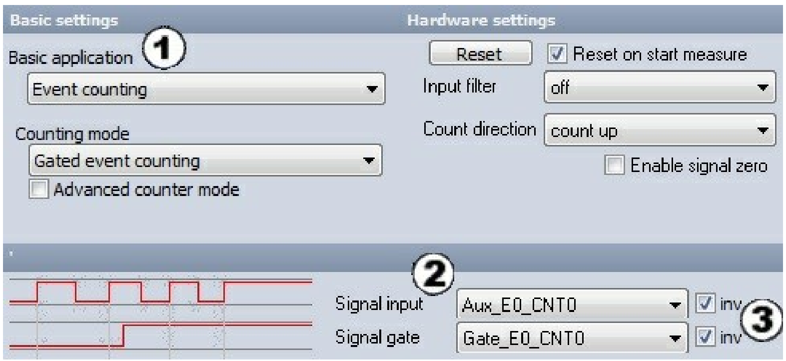
   When the *normal state* is *high, sometimes is nice to* invert the signal by choosing the **inv** check box ②. This has two effects: first is that the levels will *change*, so normal level will be *low* and consequentially the counter will count on *falling edges*.

3. Reasonable is also set the **Input filter** to prevent *double* counts. We can select **Count direction** either to count up or down ③ . This value can be selected from drop down list in this field.

→ *see* → **Counter hardware parameters**

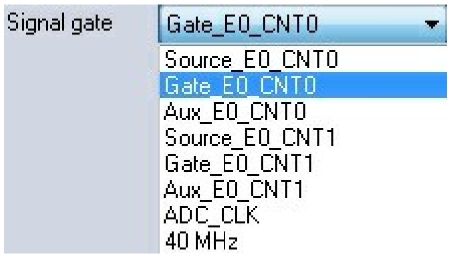
Gated event counting

Gated event counting is the mode where the counter **counts** only when a gate signal is *high*. It is available only with Orion counters. This application is based on the previous '*Simple event counting*' section (*see* above).



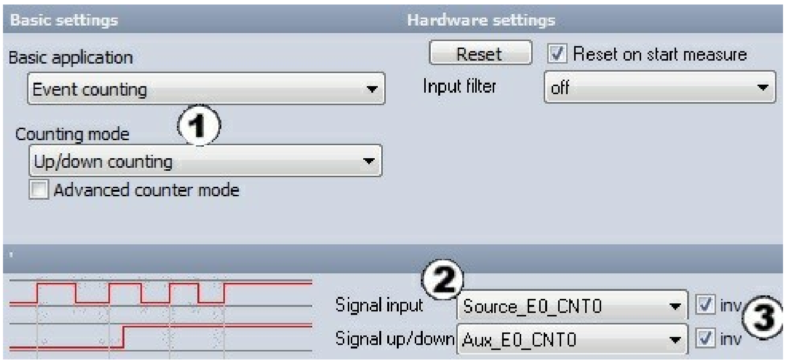
Additional to above '*Simple event counting'* settings (here we choose 'Gated event' as **Counter mode** ①) we must set:

– **Signal gate** (e.g. Gate_CNT0 above) ②. The counter will count the transitions from *low to high* only when a signal gate is *high*. Signal channel can be selected from drop down list.
If we have the Signal gate inverted by choosing the **inv** check box ③ (normally it is *high*), we choose to *invert* also the gate signal, that it will count only when a *gate signal* level is *low*.



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 above '*Gated event counting*' settings:

– we choose 'Up/Down' counting ① as **Counter mode**

– select **Signal input** and **Signal up/down** (e.g. Aux_CNT0 above) ②.
The counter will count the transitions from *low to high* - **counts** up when Signal up/down is *high* and counts down when a this signal is *low*.
Signals channel can be selected from drop down list.
If we have the Signal up/down inverted by choosing the **inv** check box ③ (normally it is *high*), we choose to *invert* also the gate signal, that it will count down only when *gate signal* level is *high*.

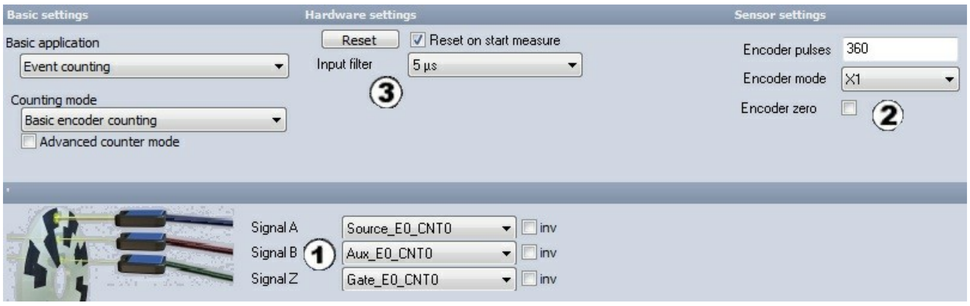
2.5.2 Encoder

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 know also a direction of movement and a *zero pulse* - one pulse per revolution which can tell us the absolute position of the encoder.

Required hardware	DEWESoft USB
Setup sample rate	At least 1 kHz

for hints about ***Encoder setup*** and ***measurement*** see → DEWESoft Tutorials

X1, X2, X4 modes



1. We choose 'Encoder' as the **Counter mode**.

2. Then we choose the Signal inputs: **Signal A**, **Signal B** and **signal Z** (e.g. Source_CNT0, Aux_CNT0 and Gate_CNT0 in above example) ①. Signals can be selected from drop down lists.

3. We set the **Encoder mode** ② 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 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 assumes 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* ③ to prevent *double* counts.

see also → **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*.

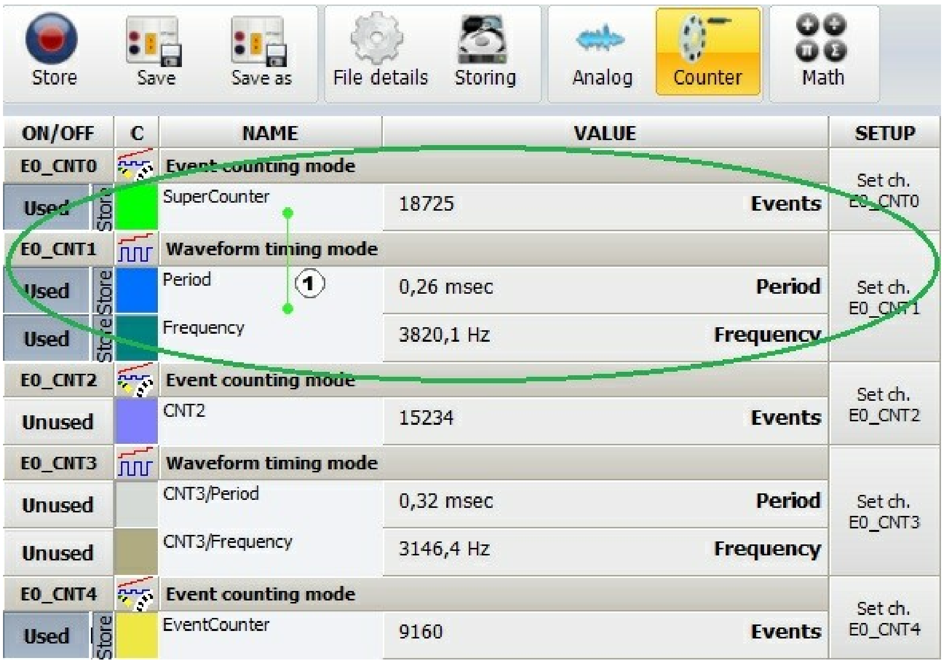
2.5.3 Advanced counter mode

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 discreet* values on *each sample*.

But since the Dewe counters can measure *where EXACTLY the position* of the pulse is in between two samples, we can **calculate** two things out of this: *exact interpolated position* of counter at the sample point as well as *exact frequency* of the pulses.

DEWESoft can perform these measurements on a single counter.



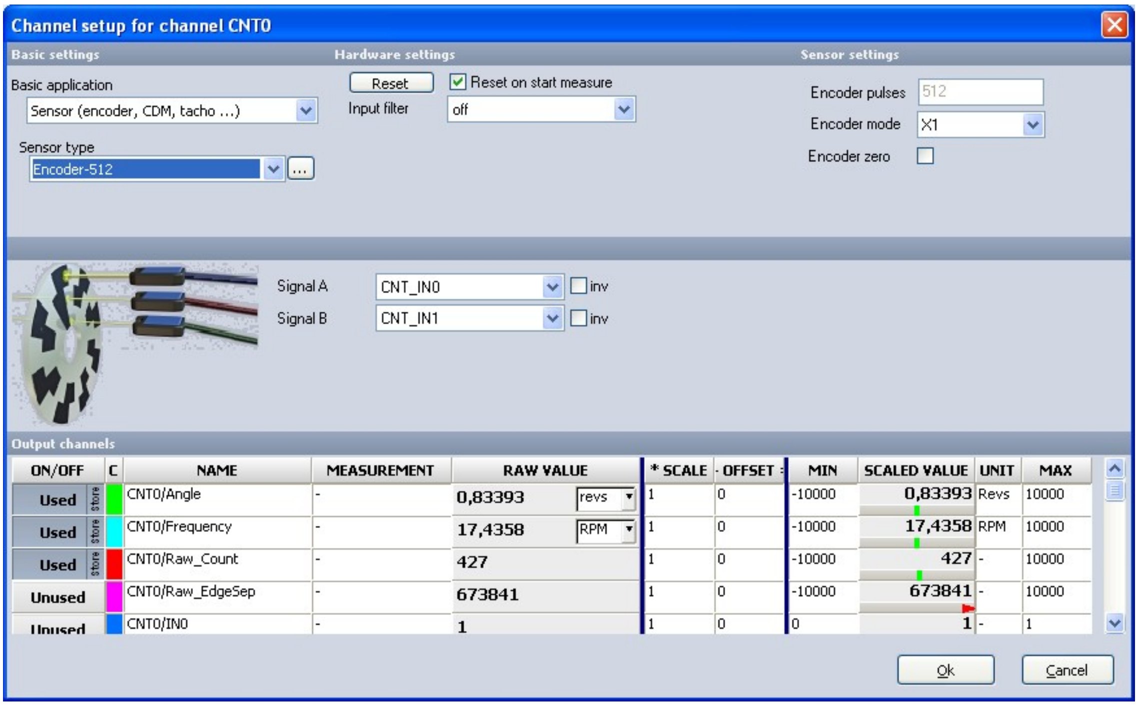
Advanced counters can be used with counting modes by selecting the Advanced counter mode button in *Sensor mode* or in *Timing mode*.

2.5.4 Sensor mode

Required hardware	DEWESoft USB
Setup sample rate	At least 1 kHz

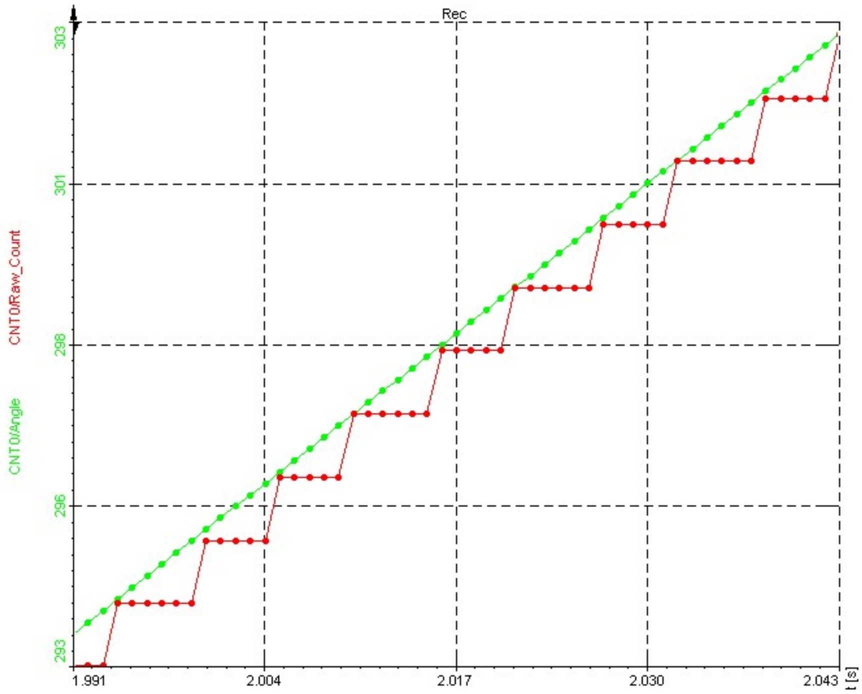
Sensor mode is very easy way to **measure** signals coming from the *real* sensors. By choosing the Sensor mode, DEWESoft offers a list of sensors defined in *Settings → Counter sensor editor*.

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 details and hints about ***Super-counter setup*** and ***measurement*** see → DEWESoft Tutorials

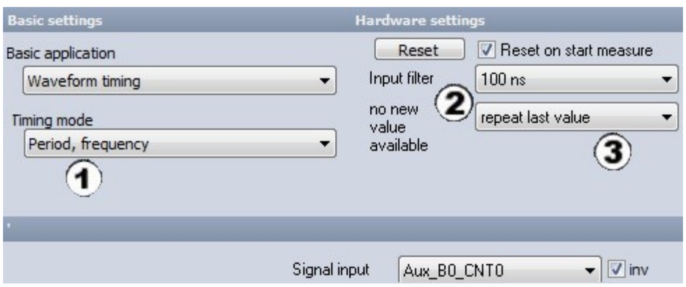
2.5.5 Timing modes

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

for hints about ***Period and pulsewidth setup*** and ***measurement*** see → DEWESoft Tutorials

Period measurement



1. Select the **Counter mode** as 'Period' ①.

2. Choose the **Signal input** (e.g. Aux_CNT0 in above example) and set the signal **Input filter** to *prevent glitches* ②.

see → **Counter hardware parameters**

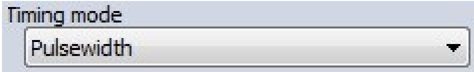
3. In additional field **no new value available** tells the software what to do when *no new value* is available ③.

see → **Counter hardware parameters**

Pulsewidth measurement

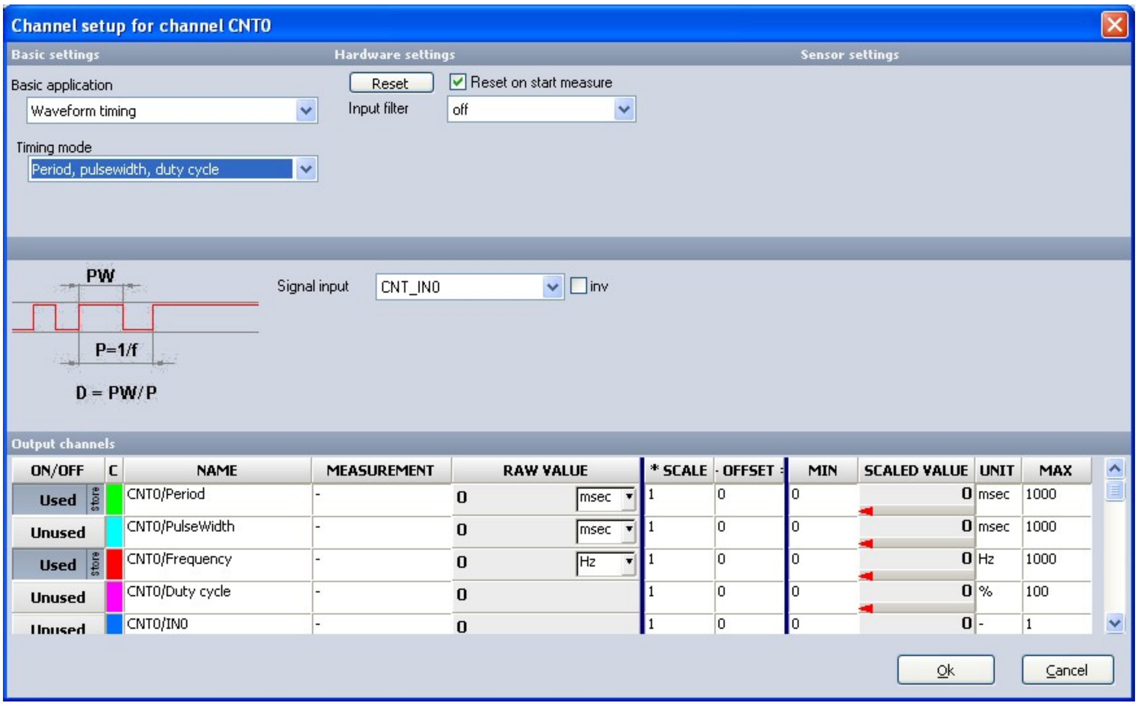
Pulsewidth measurement setup is the same as above in the *period* measurement.

The only change is to select the 'Pulsewidth option' in Timing mode ①.



Duty cycle measurement

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.

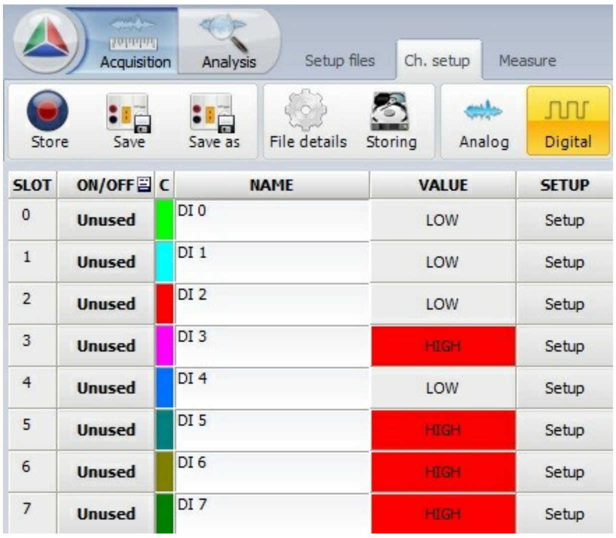
for hints about ***Duty cycle setup*** and ***measurement*** see → DEWESoft Tutorials

2.6 Digital input

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.

The Digital input hardware can be *activating* on the DEWESoft System menu → Hardware setup... → **Analog** tab by set the required *counter channels* (DI chnls column) in Device information list.

When you select a **Digital** input tab on DEWESoft *Setup* screen, on lower part of screen **Digital input** *channel line* will appear:



Digital input channel list

First let's look at the columns that are shown in *Digital input channel list*, and describe what each is for:

SLOT – Digital input channel *input* slot - *number* of the digital input

ON/OFF – *activate / deactivate* Digital input channel - press the **Unused** button - it will change now to **Used** - to set the desired Digital input channel ***active*** for *Digital input data acquisition* and they are ***available*** in *all instruments*

C – channel *color* selector - this color will carry through the text and graph representations of *this* channel *throughout* all DEWESoft procedure screens

NAME – Digital input channel *name* - just click into the **NAME** field and you can *enter* the new Digital input channel ***name*** as usual text

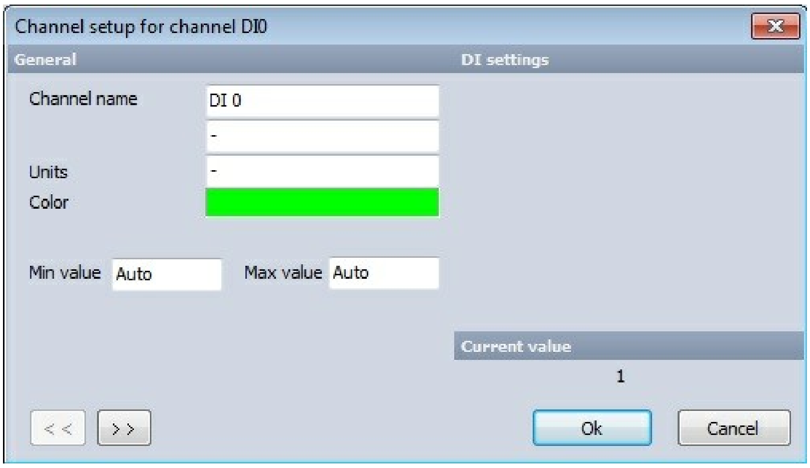
VALUE – current Digital input channel *input state*

SETUP – Digital input channel *settings* (*see* bellow)

for information about ***Slot, On/Off, C*** and ***Name*** column *see* → User Guide → **Channel setup**

Channel setup for input channel

Press **Setup** in **SETUP** column 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 and Min and Max value (*see* → **Analog (in) Channel Setup**)

- DI settings

  We can define the *signal filter* (similar like *counters*) in this section, but only if Orion card is installed. This card also offers a chance to *invert* the *input*.

- Current value

  Displays digital value (0 or 1) of current **Digital input** *channel*.

2.7 CAN bus

DEWESoft **CAN Acquisition** module is able to *listen* to the *traffic* on **CAN** *bus interfaces* from different manufacturers and to *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

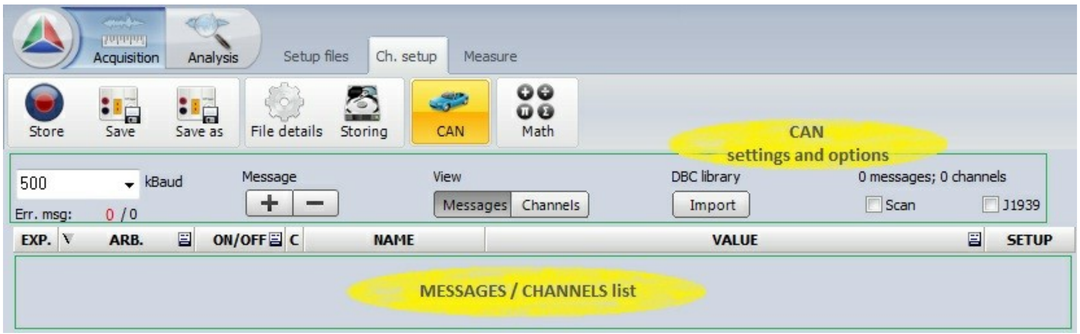
CAN messages/channels must be *defined* and *set up* with:

- *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**
- **Select for storage**

for hints about ***CAN bus Acquisition*** see also → DEWESoft Tutorials

When you select a **CAN** tab on DEWESoft *Setup* screen, on lower 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 *number* of messages and channels in list is displayed on right part 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**

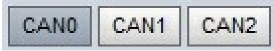
All this work can be done with *settings* and *options* on this part of CAN Setup screen.

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 **CANn** button, select from CAN0 to CAN3 (depending on *available ports*).



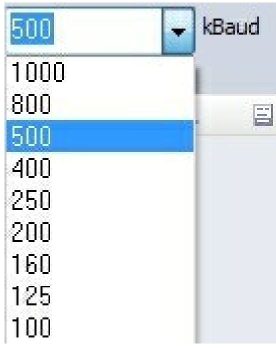
Numbers of buttons depend on selected ports in CAN HW setup.

for detailed information about ***CAN HW setup*** *see* → System Settings → **CAN Hardware setup**

CAN speed- baud rate

The baud rate setting ② is ***very important***. In fact, some vehicle operation can be *interrupted* if we connect to the bus *with wrong* baud rate set.

Select the *CAN speed* from the **kBaud** field drop down list or enter 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 *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* have any *problems* due to bad connection or bus overload.

Import messages / channels from existing .dbc file

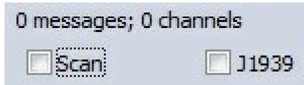
Press the DBC Library  - **Import** button to get a standard *Open* window. Select now the required ***. dbc** file and press the Open button to **import** desired file and a *list* of *all available* *messages* and *channels* will appear.
(*see* also → **Messages/channel list**).

You can *edit* the messages and channels now also *manually*.

Also **export** of CAN messages are possible. For that, please enable the Export DBC option from DEWESoft Tuner. Tuner can be found by opening **System** folder (go to *Settings* → *Global setup* → *Folders*). The export can be enabled in the CAN section of the *Tuner*.

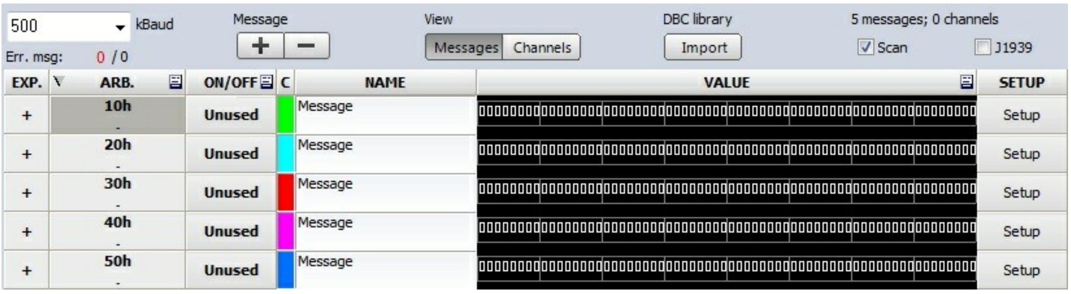
After restart of DEWESoft, export DBC option is also available in CAN channel setup.

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** checkbox.

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, 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 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.

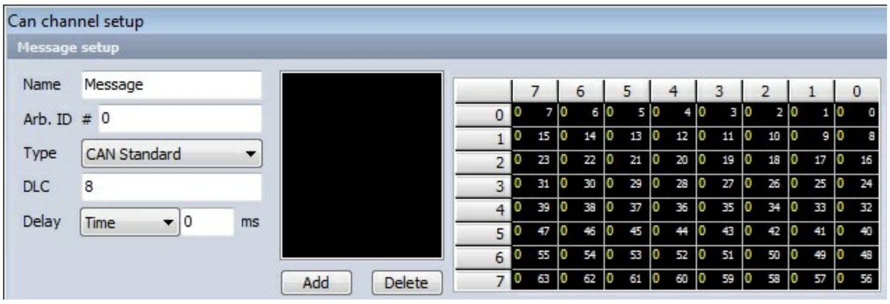
Arb ID is always extended in this case. This is most widely used on trucks. Please make sure that the bus type is really J1939 before enabling this option.

2.7.1 Channel setup

Setup messages / channels

Add messages / channels manually - Can channel setup

To **add** a message *manually* simply click on the Message - **Add** button. The *Can channel 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* for the message.

Arb. ID # — *ID number* of your message on the CAN bus

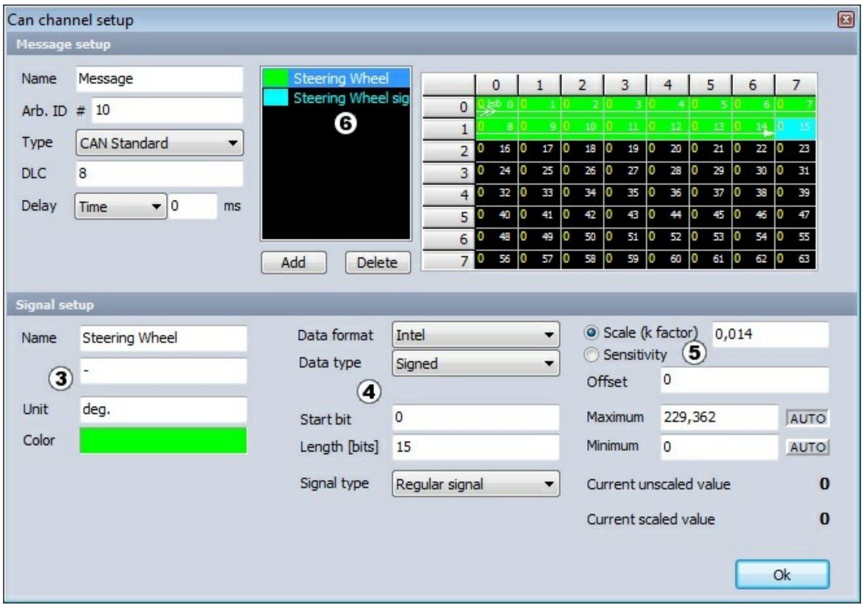
Type — Select between CAN standard and CAN extended from drop down list. Those two differs in *identifier length* - standard length is 11 bits and extended is 29 bits.

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* 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 or Delete - **Delete** to **remove** the *selected* channel.



The Signal setup offers a lot of settings:

Name ③ — *name* of the channel (signal within the message)

Unit ③ — *unit* of the signal

Color ③ — color of the signal on displays and lists - color selector

Data format ④ — select between Intel (little endian) and Motorola (big endian) format from drop down list:

Data type ④ — select between Unsigned (only *positive integer* number), Signed (integer number where the first bit is the *sign*) and IEEE Float (*floating point* number) from drop down list:

Start bit ④ — this value indicates the ***start*** of your *signal* <u>within</u> the *message*

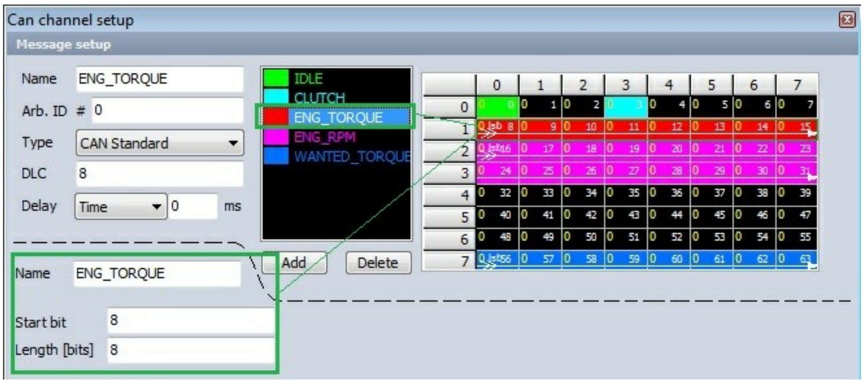
Length [bits] ④ — this value defines the ***number*** of bits <u>used</u> for this *channel*

Factor, Offset ⑤ — 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  button this values can be set automatically.
When you use only a *small range* out of it, you can *decrease* the display range.

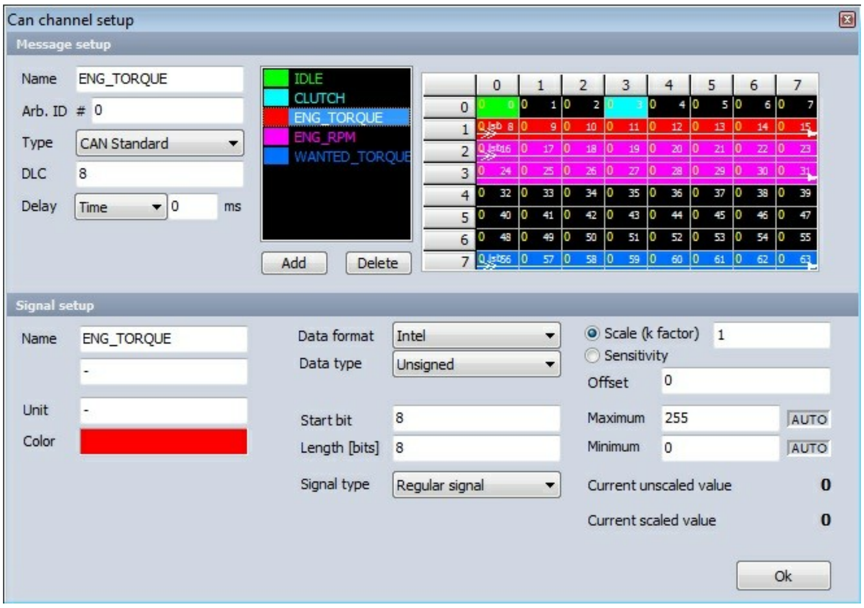
In addition to the signal settings, you have an *online display* of Current unscaled value and Current scaled value in lower right part of Signal setup area.

The currently available *signals* within the *message* are also displayed in the array ⑥ on the upper right area of the window to get a *fast overview* of your message:

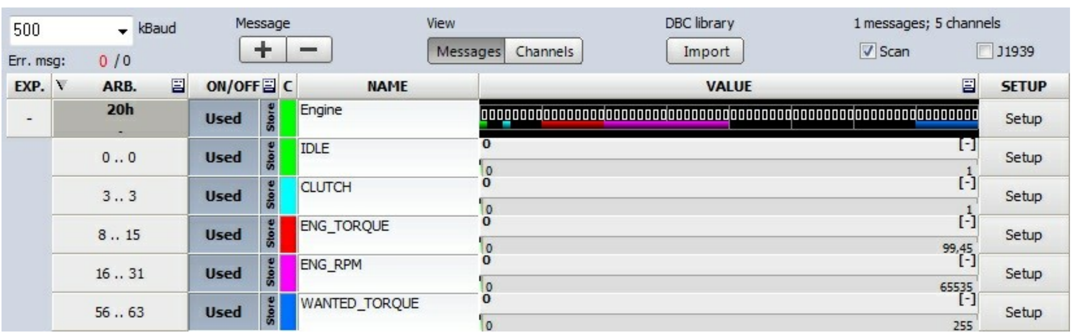


In this fields array the running number of bit is displayed on right side of field. Start bit is designated with *double* arrow, *end* bit with *simple* arrow and both arrows are connected with white line, which represent Length of signal.

Example: *Can channel setup after define 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).



Edit messages / channels

To *edit* CAN *messages* or *channels* simply press the appropriate **Setup** button; you will receive the same *Can channel setup* window than 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

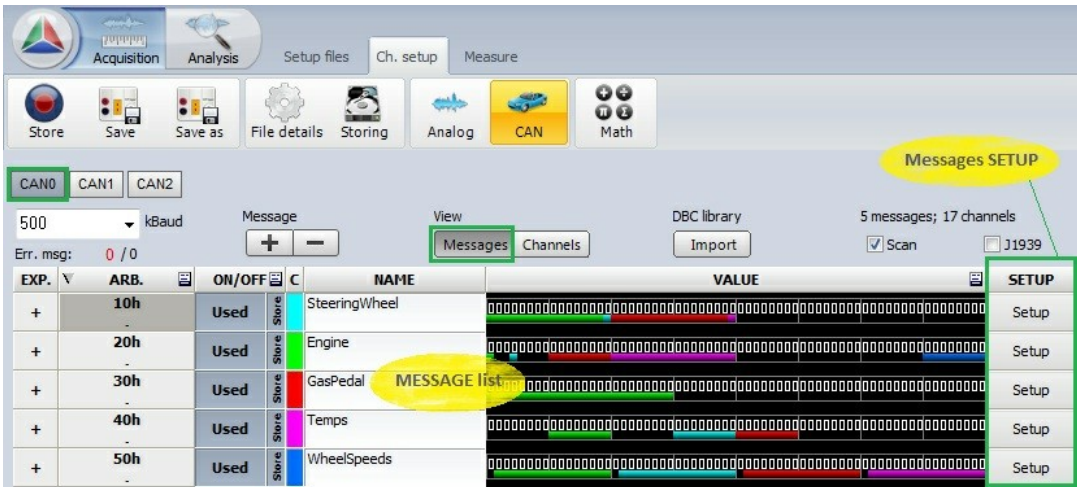
After you have defined *all signals* (*channels*) within *all messages* or you have import messages from `*.dbc` file, all this *messages* appear on **Message list** or **Channel list** (*channel overview*) on lower area of CAN setup window.

View options

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

1. Messages list (view)

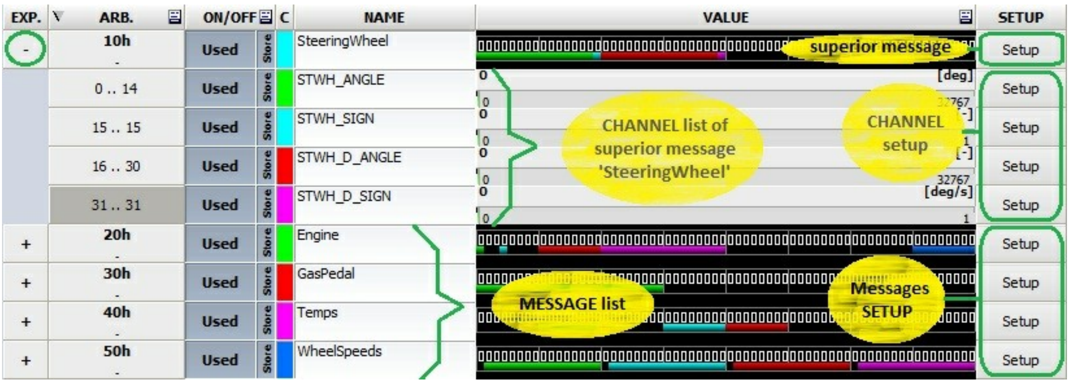
With selecting Messages View button *list* of defined *CAN messages* appear:



Columns that are shown in your *CAN messages list*:

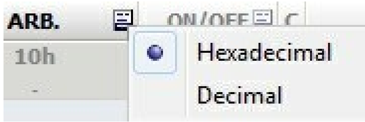
EXP - to *display / hide* channels of particular message

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 specific message. Values can be displayed in Hexadecimal or Decimal form, use icon and select desired form from displayed drop down list.

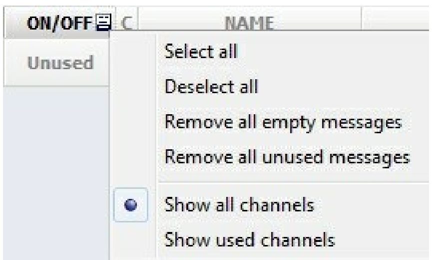


ON/OFF - to *activate / deactivate* CAN messages or channels for online ***display*** and ***storage***.

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.

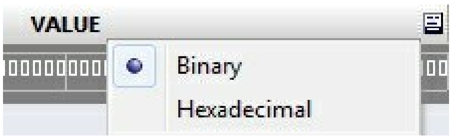
C - channel *color* selector

NAME - CAN message (channels) *name* - just click into the **NAME** field and you can *enter* the new ***name*** as usual text.

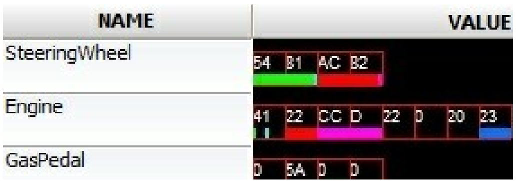
VALUE - current *CAN messages / channels value*

As a standard the setup displays the *current data transfer* on the CAN bus in a ***binary*** format.

Click on the 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 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 ***message setup*** *see* above → **Message setup**

for information about ***C*** and ***Name*** column also *see* → User Guide → **Common channel setup**

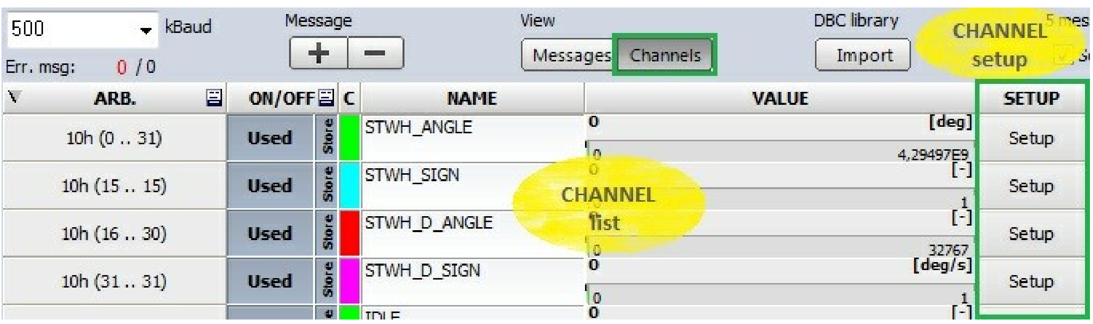
Select messages / channels for storage

When you *activate* channels for display in On/off column, the **Used** button offers also a small - **Store** button. As a standard, this button is set ***active*** - the message or channel will be *stored*.

When you press the **Store** button, his appearance change to and you can ***avoid*** the *storage* of this message or channel.

2. **Channels list** (view)

With selecting Channels View button *list* of defined *CAN channels* appear:



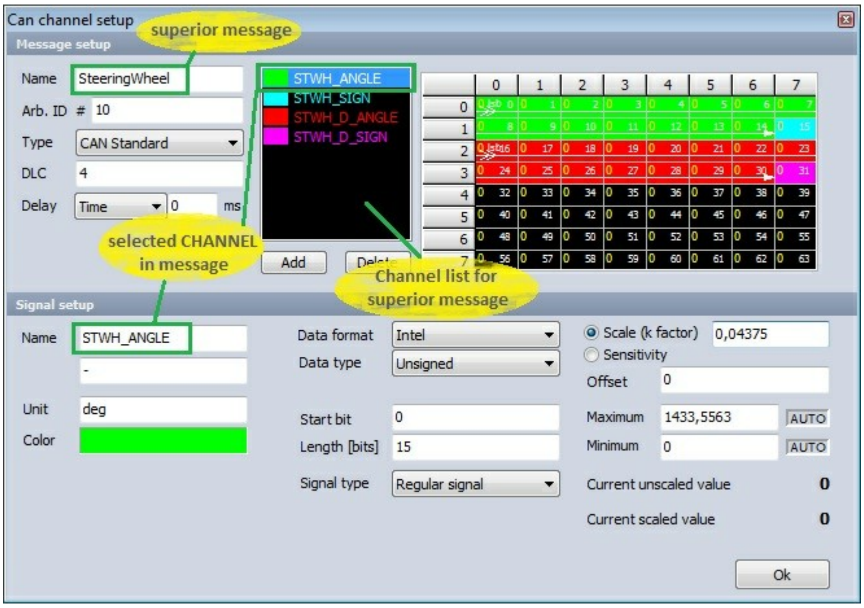
Columns that are shown in your *CAN channels list* are similar as in Messages view, except:

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 bellow)

After selecting **Setup** button Can channel setup window will automatically open for *enter* required ***channel*** *setting changes*:



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

for information about ***channels setup*** *see* above → **Signal setup**

2.8 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

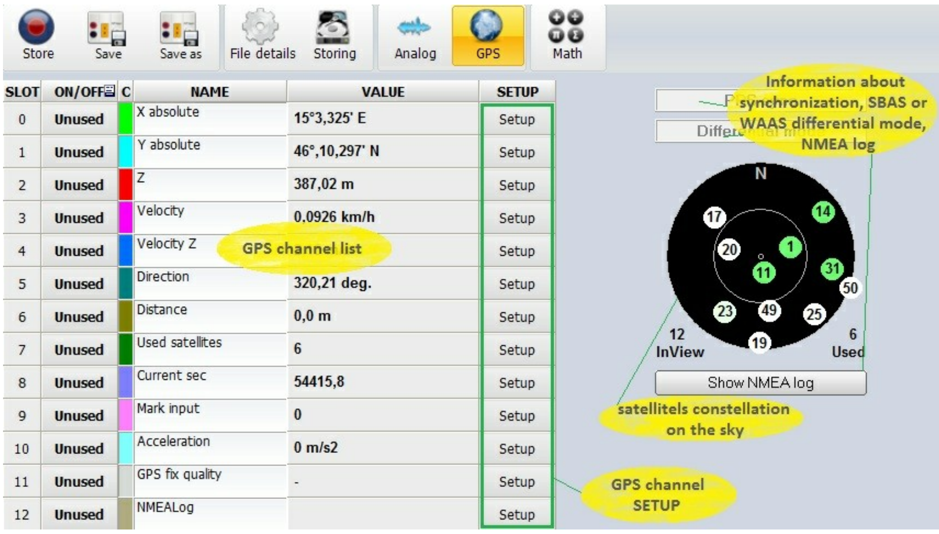
for hints about ***GPS Acquisition*** see → DEWESoft Tutorials

for detailed information about ***GPS HW (sensor) setup*** *see* → System Settings → **GPS Hardware setup**

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



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 (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 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 *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

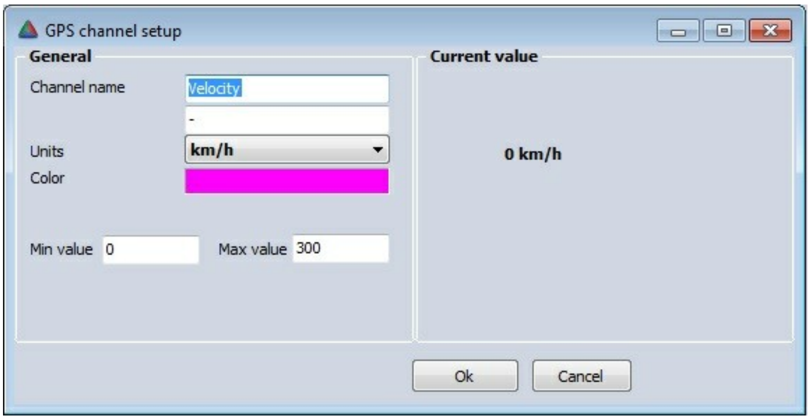
VALUE – current GPS channel *value*

SETUP – GPS channel *settings* (*see* bellow)

for information about ***Slot***, ***On/Off***, ***C*** and ***Name*** column *see* → User Guide → **Channel setup**

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*

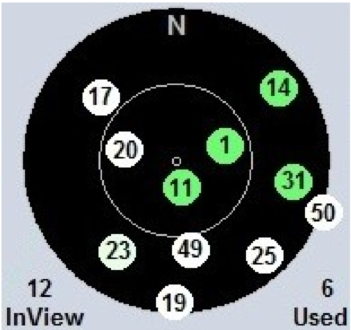
Edit and navigation keys are same as in other *Windows* application.

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.



Above InView field is number of *all* satellites and above Used is number of used satellites.

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* a synchronization to other data source a lot. If the PPS sync is not there, we need to switch it off in the DEWESoft Tuner utility under **GPS section** to be *able to receive* the data from such GPS. If this indicator is dark green, PPS signal is *present*. This is important that GPS data is perfectly synchronized with analog data.

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.

2.9 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 hints about ***Video Acquisition*** see → DEWESoft Tutorials

for detailed information about ***Video HW setup*** *see* → System Settings → **Video Hardware setup**

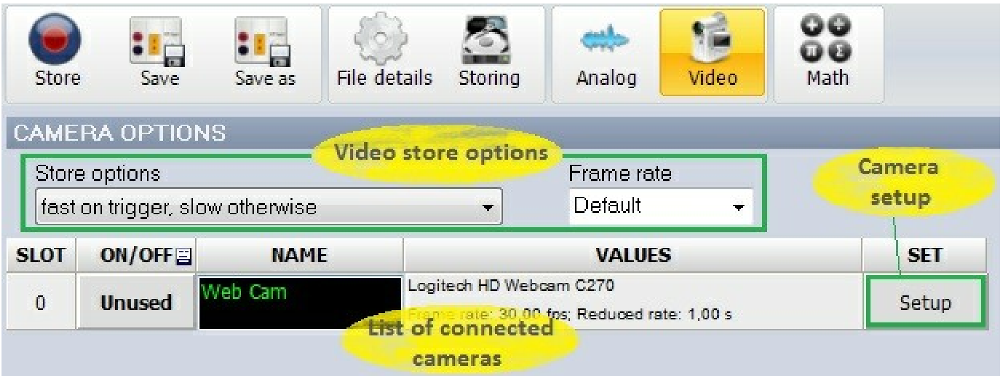
for detailed information about ***High speed Photron cameras setup*** *see* → **HS Video**

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



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 *camera list*, and describe what each is for:

SLOT – camera channel *input* slot

ON/OFF – *activate* / *deactivate* camera - press the Unused button - it will change now to **Used** - to set the desired camera ***active*** for *video data acquisition*

NAME – camera *name* - just click into the **NAME** field and you can *enter* the new ***name*** as usual text

SETTINGS – current *camera settings - manufacturer* and *name, information about frame rate*

SETUP – *camera settings* and *calibration* (*see* bellow)

for information about ***Slot***, ***On/Off*** and ***Name*** column *see* → User Guide → **Channel setup**

Camera setup

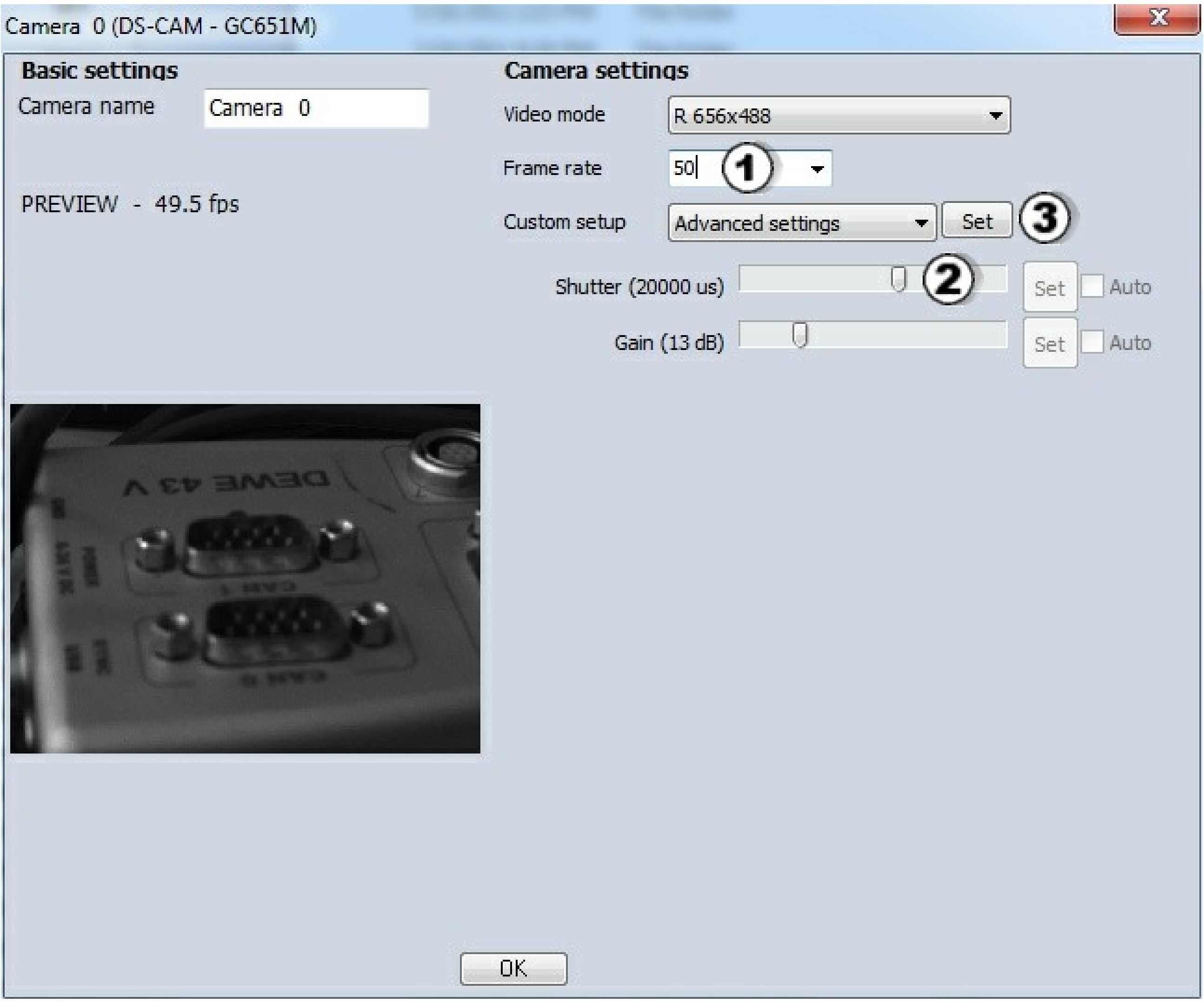
Press the **Setup** button of the camera on list of connected cameras you want to *change* and the *Camera n[type]* **setup** window will open. This window consists of three main part:

Basics settings Just click into the **Camera name** field and you can *enter* the new camera ***name*** as usual text. Navigation keys are same as in other Windows application. You can also enter this text on the Video setup screen.

Preview Preview of captured picture.

Cameras settings *Depend* on used camera (HW) - see below . He can contain: Mode setting, Frame cut settings, Compression and Resolution setting, Picture setup settings...

Camera setup:



Mode to set the **Frame rate** to fps ①; select value from drop down list.

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 ② depend on 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** ② (on the other hand this will *increase the noise* on the picture and will reduce the picture quality).

– **Auto WB** button sets the color balance of the picture. This function assumes that the *color average* of the picture is white, therefore we should press this button with holding the *white piece* of paper in front of the camera for the best effect.

The **settings** of the web and handy cam depend on the capabilities of the camera. There is a huge difference between one web cam 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* type like YUV or I420. This means that the 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 uses 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 camera and when **Direct setup** is selected you can now change on upper part of this settings: **Compression** (select from drop down list uncompressed or compression mode), **Resolution** and **Frame rate** (both from drop down list).

– When **Dialog setup** is selected on little different screen:

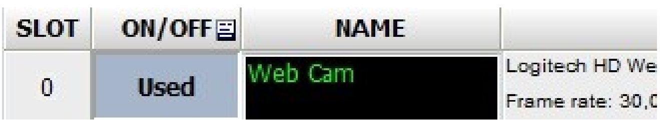
you can call with **Output** button cameras Properties window with *own* output settings. This SW is created by camera producer.

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 also **Custom settings** ③, which will show in cameras Properties window all the *special functions* of the specific cameras like flipping or rotation the picture.

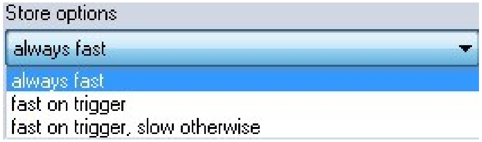
After you have done your settings press **OK** to leave the camera setup.

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 drop-down list:

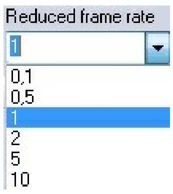


- **always fast** the data will be continuously *stored all the time* to the file with the *selected full* image acquisition rate - *full speed*.

If our data consists of events which can be captured, we can choose to store with two options of **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.

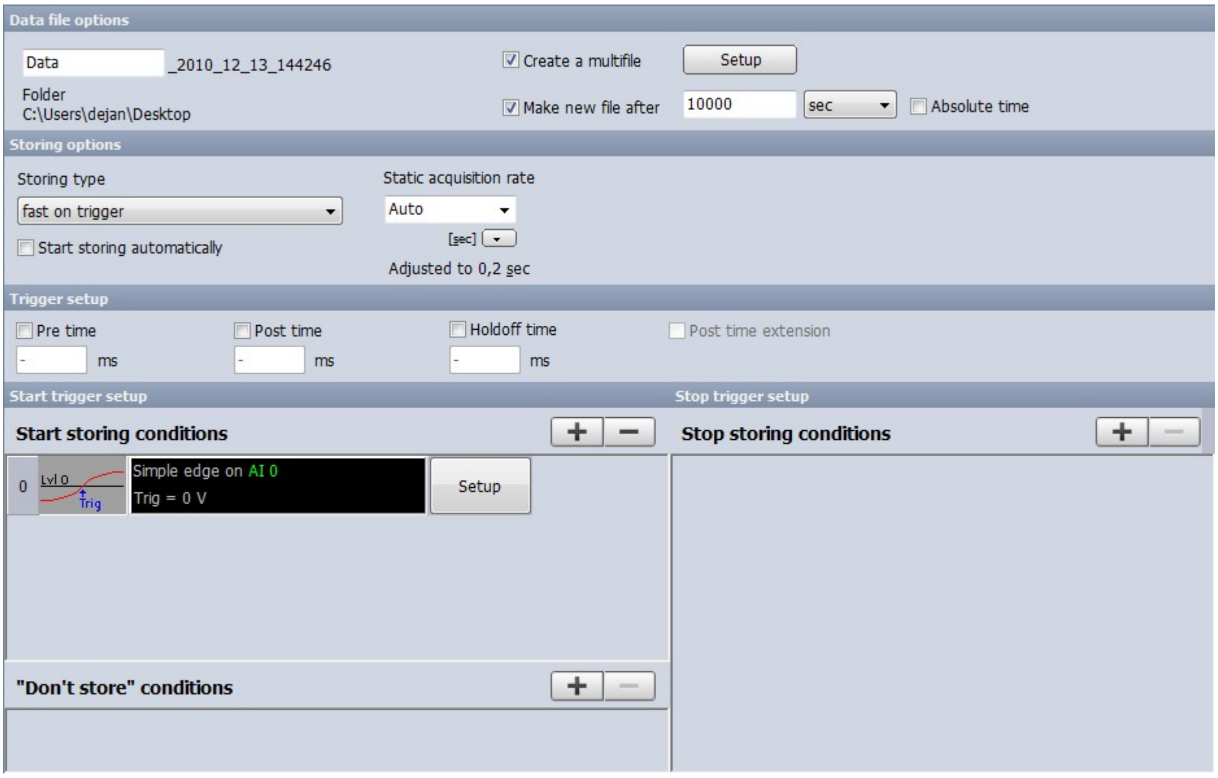
- **fast on trigger** ... the video data will only be *stored* with the *full* image acquisition rate when ***trigger*** condition is *true*

- **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 *value* from drop-down list.



Example:
if we select 0.1 seconds, this means that video picture will be *stored* every 0.1 seconds

When you select one of **trigger** storing option, on DEWESoft *Setup* screen will *automatically* appear *new* Trigger tab to *set up* trigger.



for detailed information about ***set up trigger*** *see* → User Guide → **Setup - Trigger**

2.9.1 HS Video

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

Please refer to the **Photron Installation guide**.

Hardware setup

Please refer to the **Photron Hardware setup guide**.

Setup

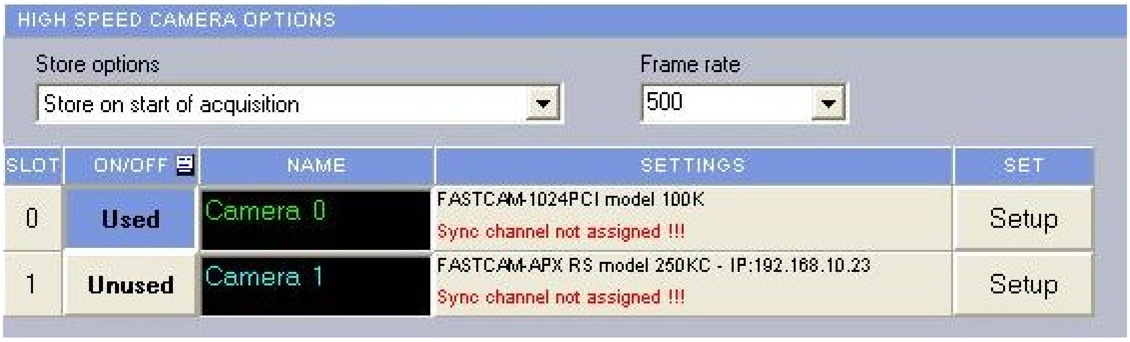


Image - Video channel setup

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 following options:

Store on start of acquisition ... camera will *start* at the beginning of *recording*

Store on DEWEsoft trigger ... 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 ... camera will be triggered with external trigger. Refer to Photron Hardware 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 selected frame rate, the closest one will be used. You can set frame rate for each 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.

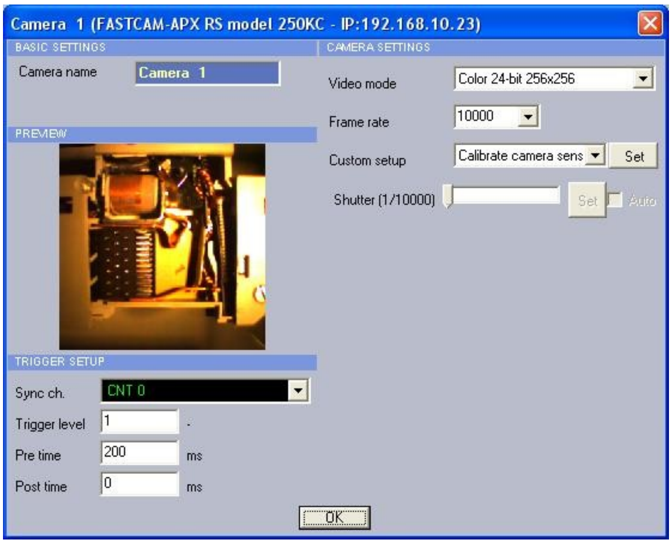


Image - Camera setup

Camera settings

Here you can set Resolution, Frame rate, Shutter speed and Sensor gain (if camera supports hardware sensor gain) parameters. You can also calibrate the camera sensor here. The options are depending on 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 parameter is not supported in DEWESoft, DEWESoft will leave it as it was).

The camera settings are applied to 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 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. 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* trigger and *after* 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 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 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 sensor to get 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 sensor will get calibrated.

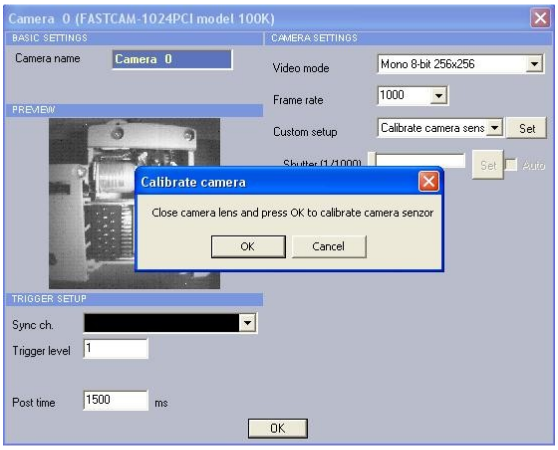


Image - Camera sensor calibration

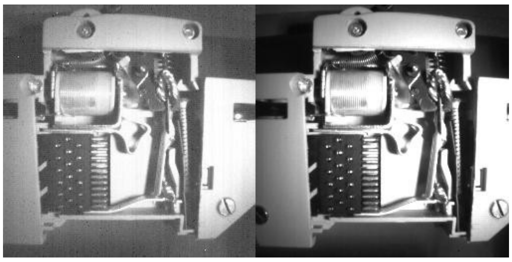


Image – Picture before and after calibration

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 transfer window. Pleas wait while all data is transferred from the camera. For situations, where it is obvious that test was not successful, there is also **Cancel** button which will stop the transfer process.

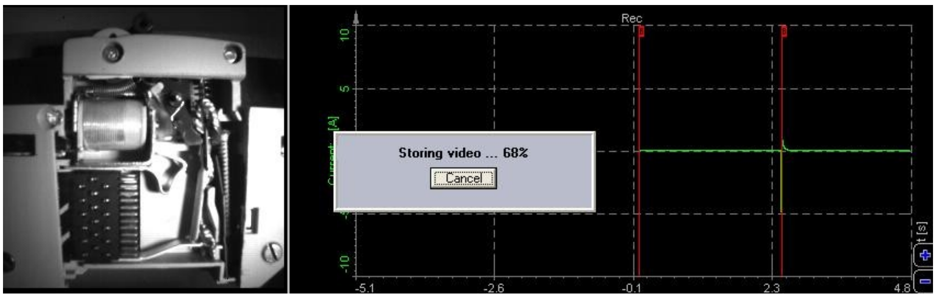


Image – Frames data transfer

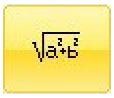
2.10 General mathematics modules

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 **mathematic modules**.

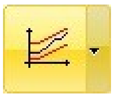
Enabling Mathematics and Filter

First you have to make sure that the **mathematics** 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 System settings → **Mathematics Setup** to enable the **mathematics functions**.

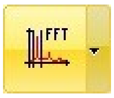
On the DEWESoft Math setup screen, you will get basic icons to *add new* **mathematics** module:



– **Formula module**

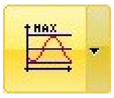


– **Curve modules** →
- **Reference curve**
- **XY reference curve**
- **FFT reference curve**
- **Constant**

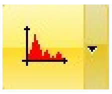


– **Filters** →
- **IIR filter**
- **FIR filter**
- **FFT Filter**
- **Envelope**

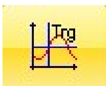
– **Dynamic signal analysis section** →
- **CA Noise**
- **Exact frequency**
- **Angle sensor**



– **Statistics** →
- **Basic statistics**
- **Array statistics**
- **Latch math**
- **Classification**
- **Counting**



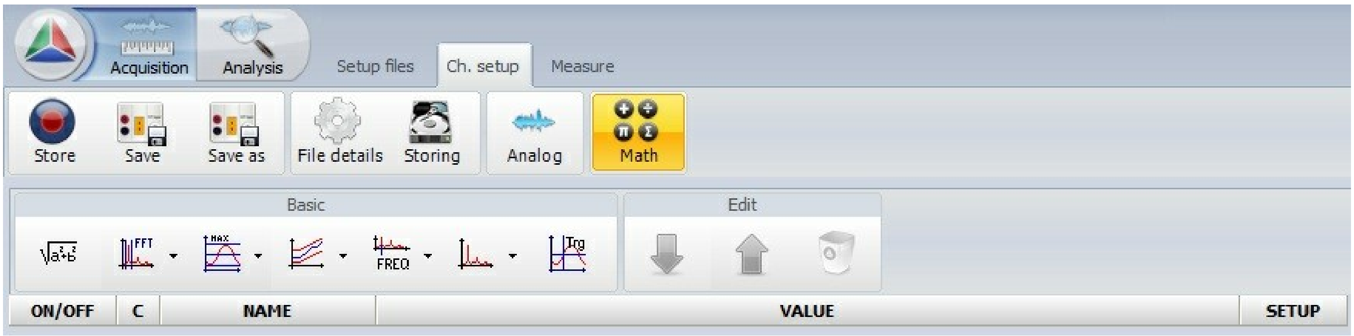
– **Spectrum analysis section** →
- **FFT analysis**
- **STFT analysis**
- **CPB analysis**



– **Scope trigger** →
- **Scope trigger**

Some icons like *filters* have a drop down button which reveals more icons. The icons are grouped according to the *application* area (*filtering*, *statistics*, *DSA*).

Please select the **Math** tab and then will *automatically* appear:



For more details on general information how to *add* and *setup* the modules, *see* **Setup screen and basic operation** page.

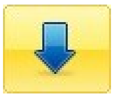
There are also three ***function keys***:

Remove selected mathematic function or filter.

If you want to **delete** the channel, simply select desired filter or function with mouse *click* and then press this minus button above the channels.

Move selected function *up* within the list.

If you want to **move** the channel *upwards*, simply select desired filter or function with mouse *click* and then press this button above the channels. For first channel in the list this button is not available.



Move selected function *down* within the list.

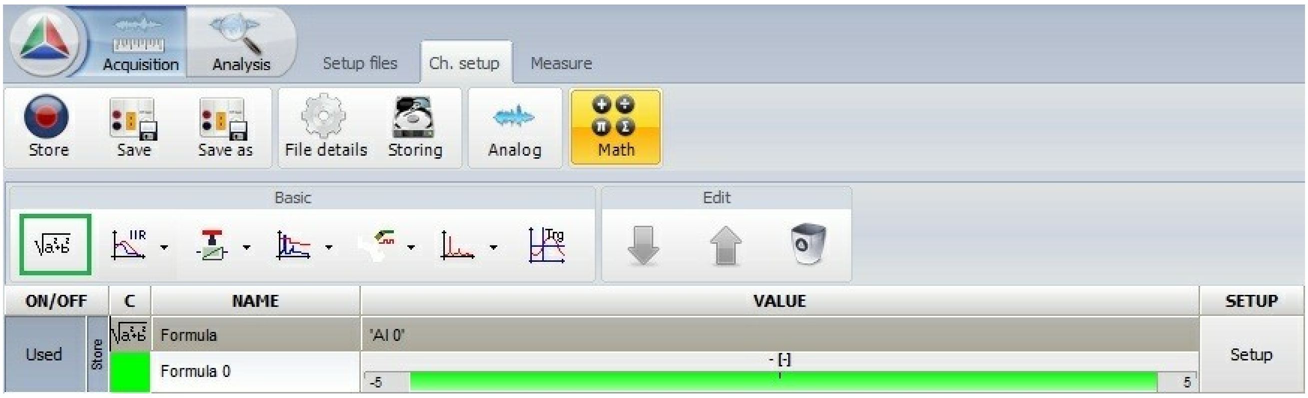
If you want to **move** the channel *downwards*, simply select desired filter or function with mouse *click* and then press this button above the channels. For last channel in the list this button is not available.

Moving the modules up and down *doesn't change* the *calculation order*. DEWESoft automatically *detects* the ***references*** and ***orders*** the mathematic 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 CPUs*. This is available only on multi core processors and can be switched on in DEWESoft **Tuner** in **Add-on options** tab with Number of calculation threads to use option. It will work best if the math modules are *not referencing* on *each other*, for example that we calculate the *IIR filter* of several input channels. Then the calculation can be nicely split on several cores. If the channels are referencing each other, this is *not possible* since the order of calculation is strict and well known.

2.10.1 Setup screen and basic operation

When you choose on the DEWESoft Math *setup* screen any button with specific Math function (e.g. formula  button), a *new* **Math** *channel* is displayed:



The columns in the new Math *line* look like similar to the ***analog channel*** setup:

- **On/Off** - *activate* / *deactivate* the channels
- **C** - *channel color* selector
- **NAME** - formula / output channel *name*
- **VALUE** - formula *preview (symbolic description)* and the *calculated values*
- **SETUP** - button to *enter the formula* in **Formula setup** window

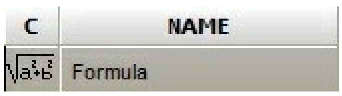
C - channel color selector

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.

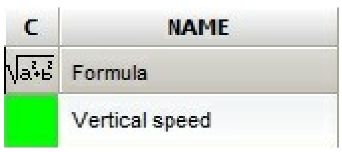
for detailed information about ***C*** *color selector see* → User Guide → **Channel Setup - Channel color**

NAME

In first channel row of Name column the *symbol* with title Formula is displayed.

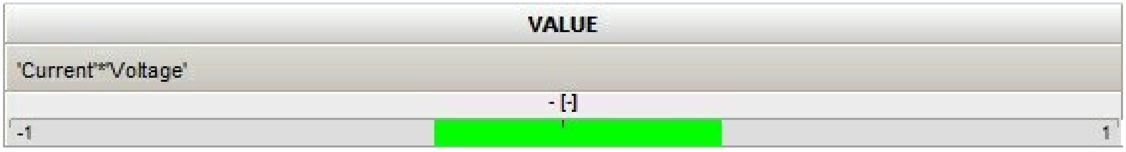


In second channel row of Name column the ***name*** of the output channel created with the *formula* is displayed.



VALUE

In first channel row of Value column the *description* of math module is display. For *formula* a ***symbolic record*** of formula is displayed:

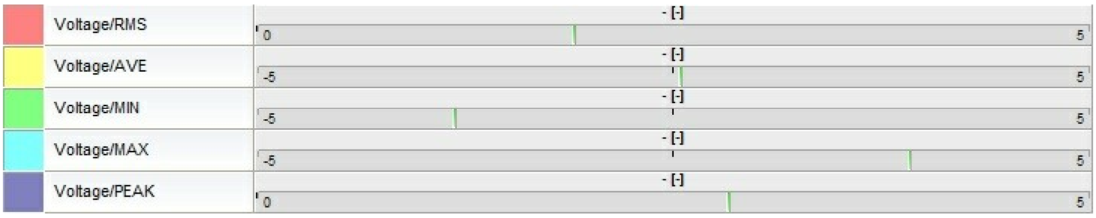


for filter, the *filter* ***settings*** are shown:

In second channel row of Value column the ***live preview*** of calculated value is displayed.

For the math modules with possibility of selecting *several input* channels (like filters) or having *several output* channels for each input (like statistics), channel output section looks little bit different - it shows *one* line for *each output* channel.

The math can have also *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:

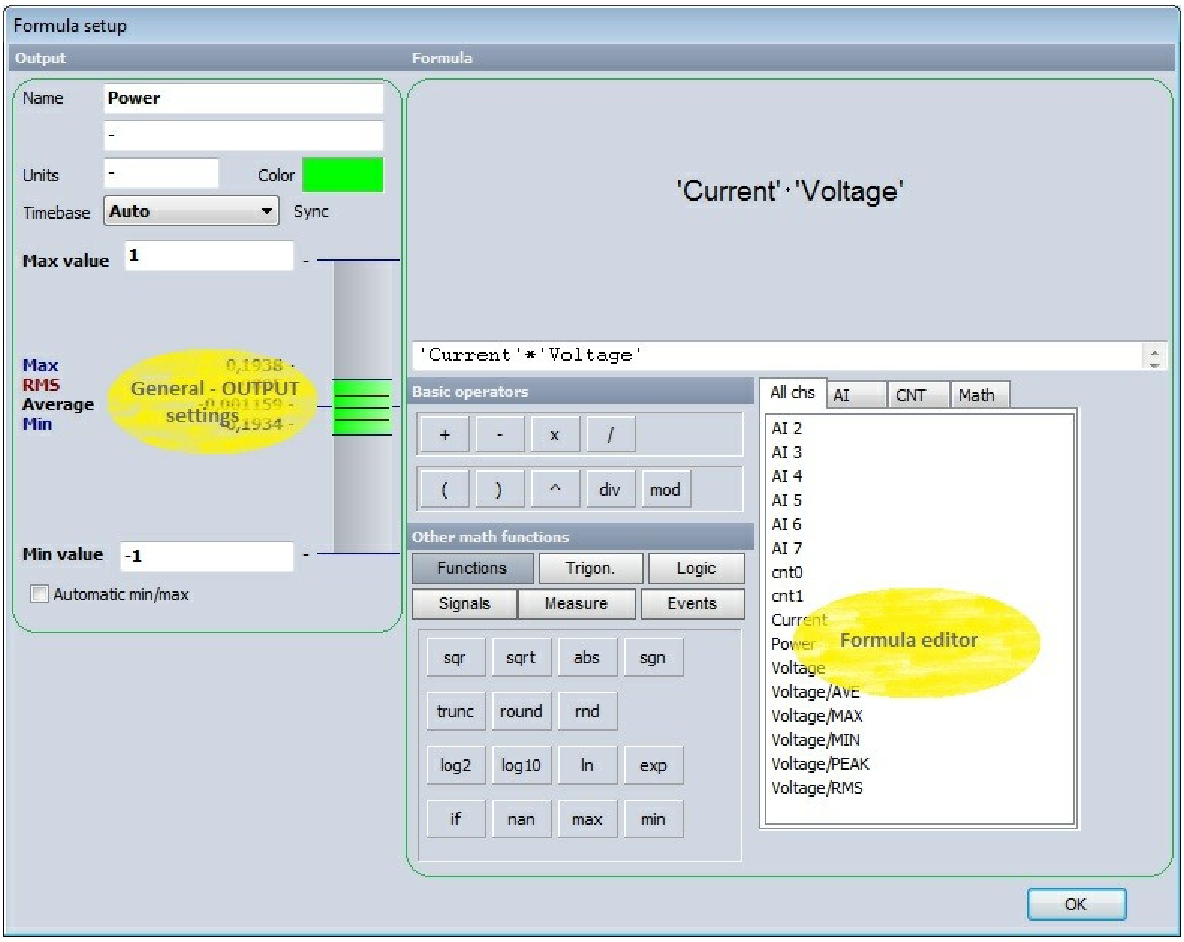
There are several possible **errors**:

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)

Math 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 :



INPUT / OUTPUT setup

There are two basic options. The formula parser or **reference curve** can have *several input* channels, but just *one output* channel. In this case the left side of the display shows the setting of the output channel while the right side of the setup screen shows the specific editor for the currently selected module.

On the other sides modules like **filters** or **statistics** can have *more input* channels and then apply the *same* settings (same filters, for example) to *all* chosen *input* channels. The result is that we have also *more output* channels from the mathematic module.

In this case the upper left section shows chosen `input channels` (they can be *selected* or *deselected* by clicking on them), lower left part shows the *general* `output channel settings`, where the only difference is that we have also two more buttons <- and -> to *choose* the output channel to set.

The currently selected MODULE is *selected* by selecting the *input channel* in the upper right section.

General output setup

The lower-left area *displays* the output channel settings like in the analog setup, like the channel **Name**, the **Units**, and also the **Color**.

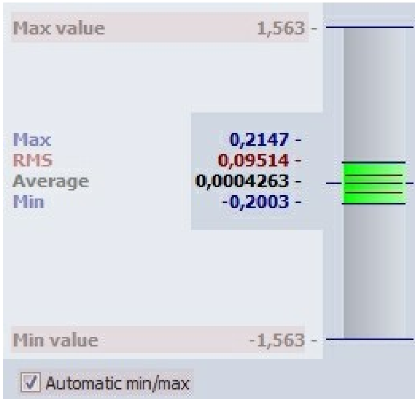
for detailed information about this **General** *setting see* also → User Guide → **Channel Setup**

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. If we multiply voltage and current in math formula, the resulting unit is $V*A=W$. These units **must** be *defined* by the *user*.

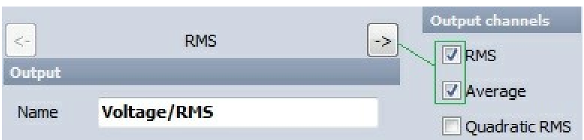
Min val

Max val 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 entering the value in the fields. Then the max value is not shown in the bar graph since it is displayed in edit box.



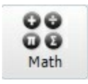
Automatic min/max calculates minimum and maximum automatically from input channels and math type. In this case min and max can't be freely defined.

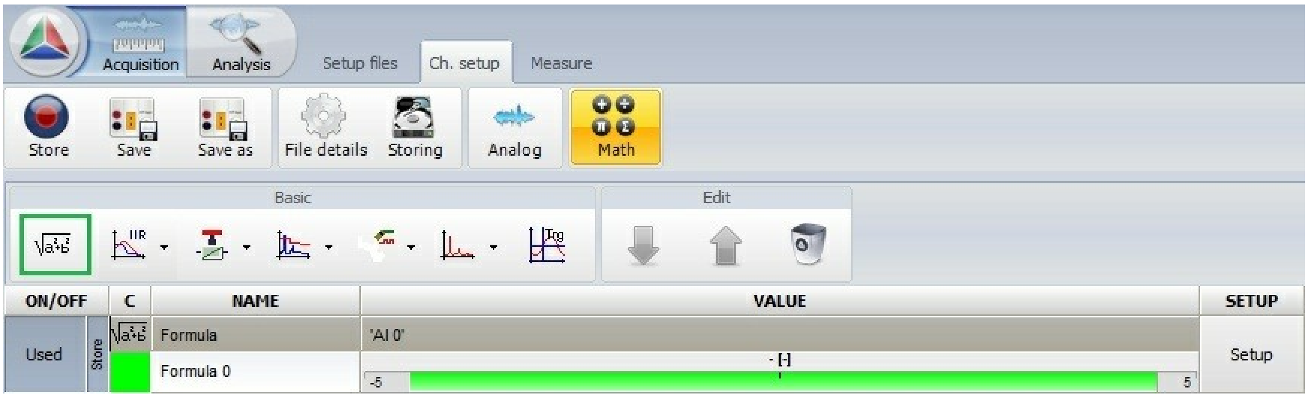
Modules like **filters** (with *more input* channels) or **statistics** can have *more* output channels from the mathematic module. In this case the upper left section shows chosen input channels, lower left part shows the *general* output channel settings, where the only difference is that we have also two more buttons <- and -> to *choose* the *output channel* ***to set***.



2.10.2 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*.

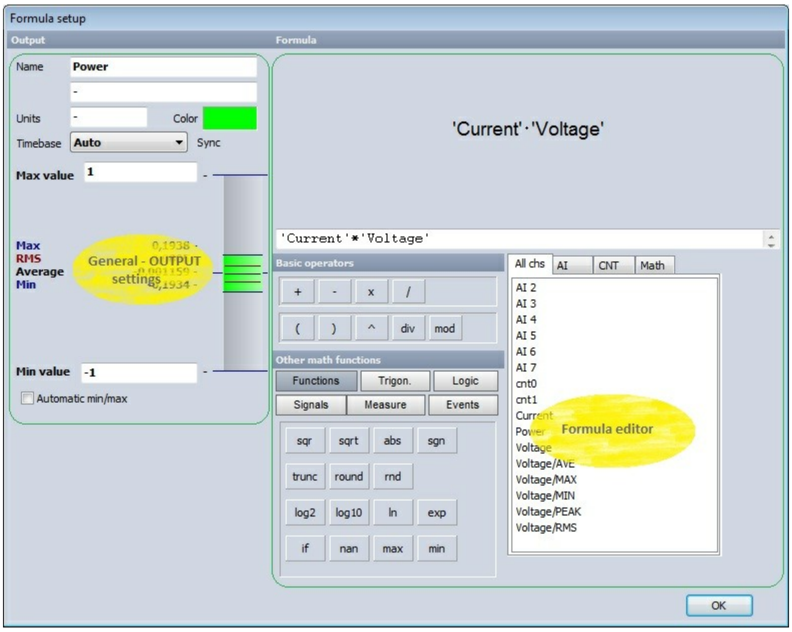
When you choose on the DEWESoft Math  *setup* screen button with  - formula icon, a *formula* **Math** *channel* is displayed:



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*.

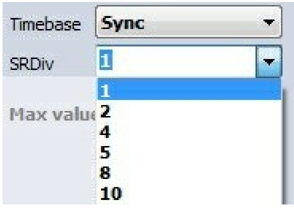
For example:
- If the input is synchronous channel with *Sr div* of 4, the resulting channel will also have a sample rate divider of four.
- If the input is asynchronous channel, the output will also be asynchronous channel.
- If there are more input channels, output will be matched to the input channel with the highest rate.

Manual — 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.

Single value — The result of mathematic is *only* a single value. Single value is a special type of DEWESoft channel where the result is *not a set* of values with time stamp, but always just one value. This is useful for calculating *cumulative values*.

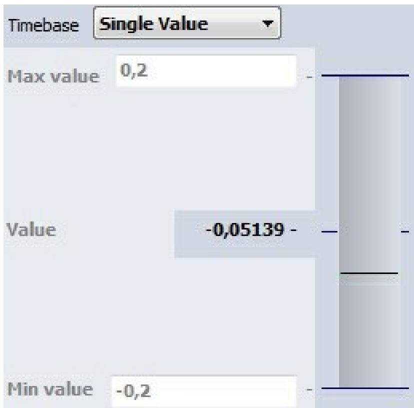
Auto

Manual



From list we can select Manual values from 1 to 500.

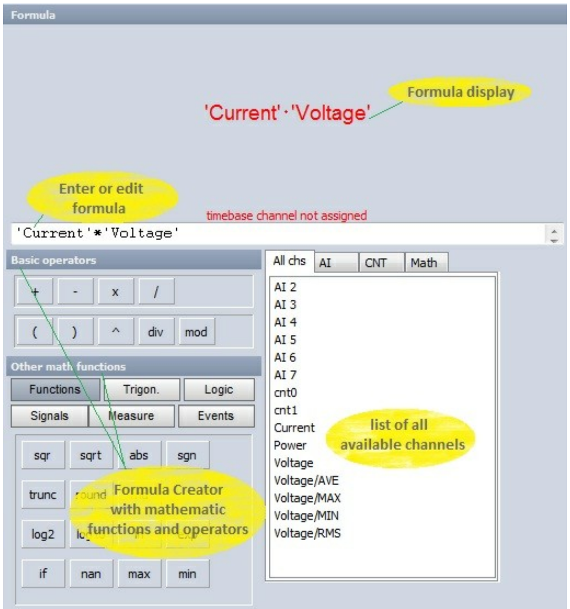
Single value



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 ***mathematic 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:

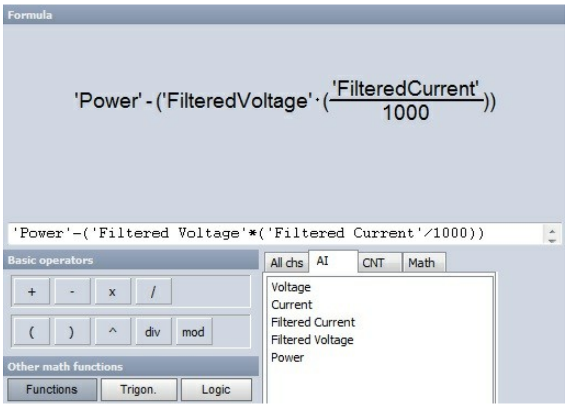
– *double-click* on the **channel** (in list of all channels) or

– *single-click* on the **function** (in 'Other math functions') or

– *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 (*see* bellow).

If you do a mathematic operation on *two* channels with **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.

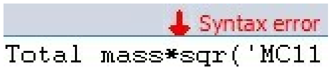
In list of all *available* channels we can see:

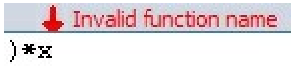
- *all* channels → select All chs tab
- *analog input* channels → select AI tab
- *power* channels → select Power tab
- *mathematical* channels → select Math tab

DEWESoft allows *different types* of mathematics, which are grouped in six tabs. Please learn more about these functions in → **Operators and mathematical function**.

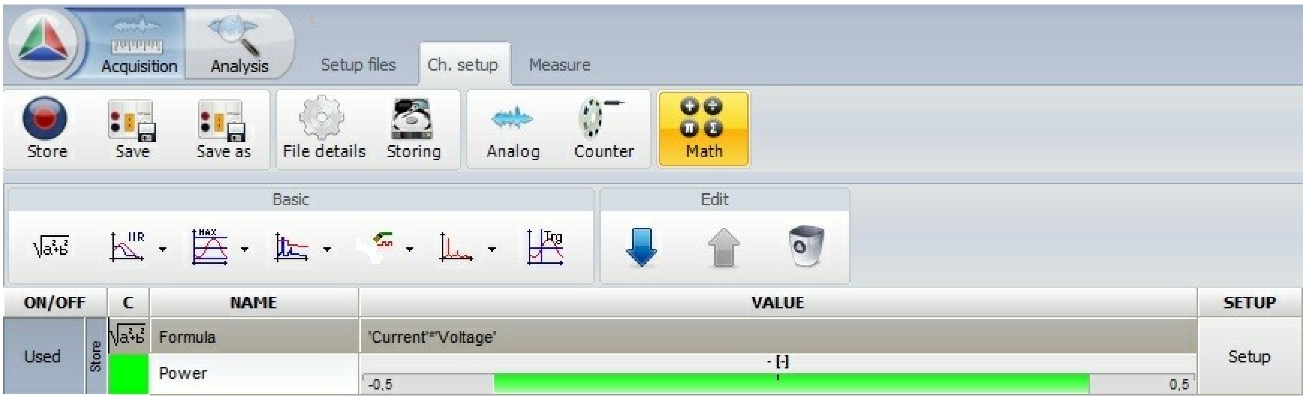
If you make an error, you will get an ***indicator*** which shows you immediately where the error is located and error *description*.

Some error examples:





When you have entered your formula, press the **OK** button to leave the setup. Now a *new* line has been added in the mathematics *overview screen*:



See next topic to have a complete list of available **operators and mathematical functions**.

2.10.2.1 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**
- **Trigonometry**
- **Signals**
- **Events**

DEWESoft X1 introduces ***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 pow er 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 MOD 720 = 0
740 MOD 720 = 1

MOD

Modulus Delivers the rest of a division.

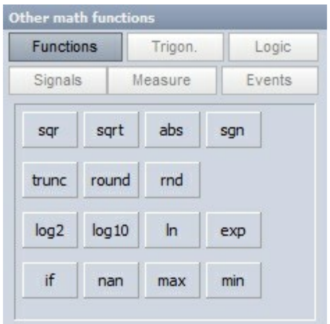
420 MOD 720 = 420
740 MOD 720 = 20

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* mathematic functions.



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 w hole number
RND	RND	random number betw een 0 and 1
LOG2	LOG2(*expression*)	logarithm (base 2)
LOG10	LOG10(*expression*)	logarithm (base 10)
LN	LN(*expression*)	natural logarithm
EXP	EXP(*expression*)	exponent ($e^{expression}$)

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) w ill alw ays deliver 0 instead of a complex number in Dew eSoft.

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* Extracts the sign of a channel or a number.

sgn(0) = 0 Sign function delivers 0 if input channel or number is 0

TRUNC

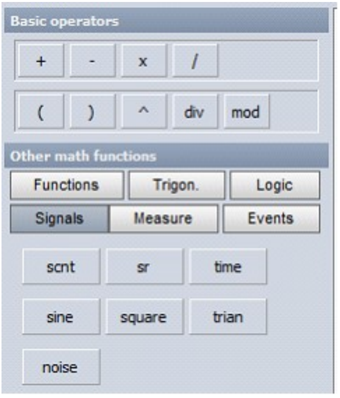
Truncate function

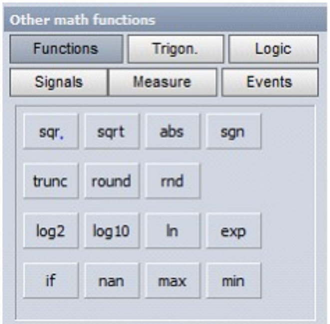
Trunc(1452.457) = 1452
Trunc(-1452.457) = -1452

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.

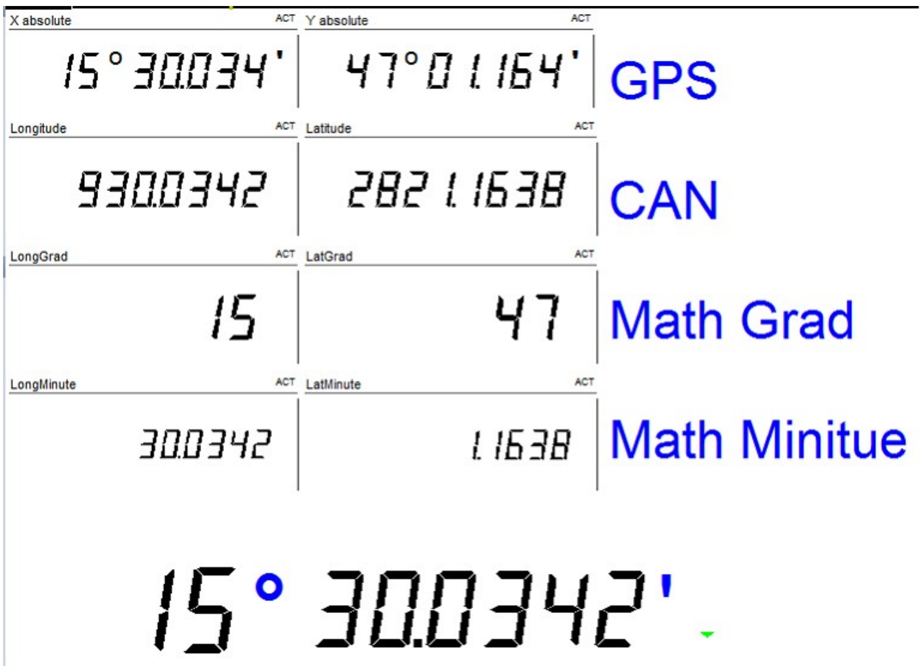
It is not rounded so either: *Trunc(86.248) = 86* and also *Trunc(86.848)* will give *86*

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*	Rounds a number or channel depending on the digits after the
	round(14.501) = 15	comma to an integer value.
	round(-14.492) = -14	3.xxx if xxx is bigger than or equals 0,5 it will be rounded up to
	round(-14.51) = -15	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 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*	Calculates the logarithm (base 2) of a number or an input channel.
	Log2(a) = b	The logarithm extracts b from an equation 2^b = a.

LOG10

Logarithm base 10	*Log10(100) = 2*	Calculates the logarithm (base 10) of a number or an input channel.
	Log2(a) = b	The logarithm extracts b from an equation 10^b = a.

LN

Natural logarithm base e	*LN(100) = 2*	Calculates the natural logarithm (base e=2.71828…) of a number or an input channel.
	LN(a) = b	The logarithm extracts b from an equation 2.71828^b = a

EXP

Expotential function of e	*EXP(1) = 2.71828*	Calculates the exponential function of e from a number or an input channel.
	EXP(b) = a	The logarithm extracts b from an equation 2.71828^b = a

IF / NAN

If, 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 channel, could be a number or any DEWESoft channel, is equal or bigger that 12, the Condition becomes true and result1 will become true. Otherwise condition is false and result2 will become true.

Example: ***IF('ID'>=1,'velocity','displacement')***

'*ID*', '*velocity*', and '*displacement*' are analogue 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 are used for output depending on the condition, both at the same time are not possible.

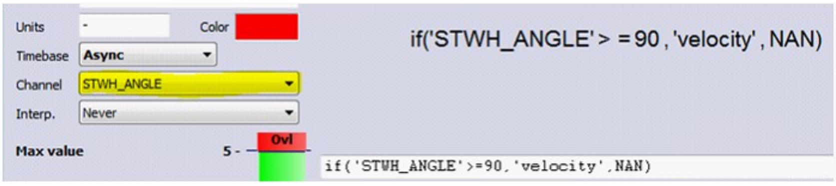
NAN(not a number)

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 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

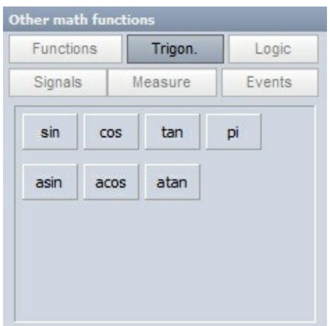
The higher value if both channels will be output.

INFO: Also *multiple* max could be used in one formula.

max(max('pressure1','pressure2'),'pressure3')

Trigonometry

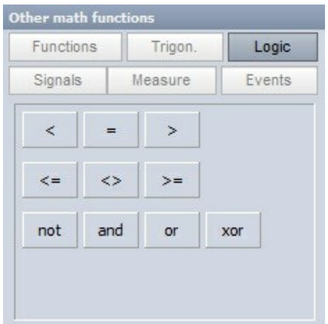
The **Trigonometry** tab contains all required ***angle*** based *functions*.



Trigonometric 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 pi
ASIN	ASIN(*expression*)	arcsine of *expression*
ACOS	ACOS(*expression*)	arccosine of *expression*
ATAN	ATAN(*expression*) or ATAN(x,y)	arctangent of *expression* or arctangent of x/y

Logic

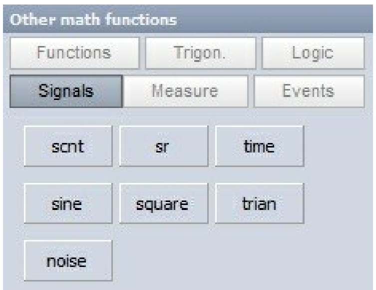
The **Logic** tab contains required ***logic** functions*. Please be aware that the logic operators OR, AND, NOT and XOR works only with PURE digital values - 0 and 1. Therefore all analog channel must be first digitized before used on this operators (like *'velocity'<100*).



for examples *see* → **Example of logic functions**

Logic functions		
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
>	*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
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.

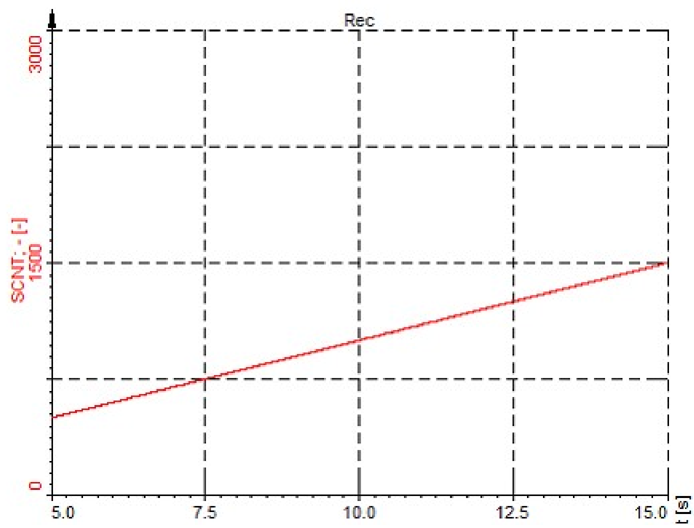
for examples *see* → **Example of signals**

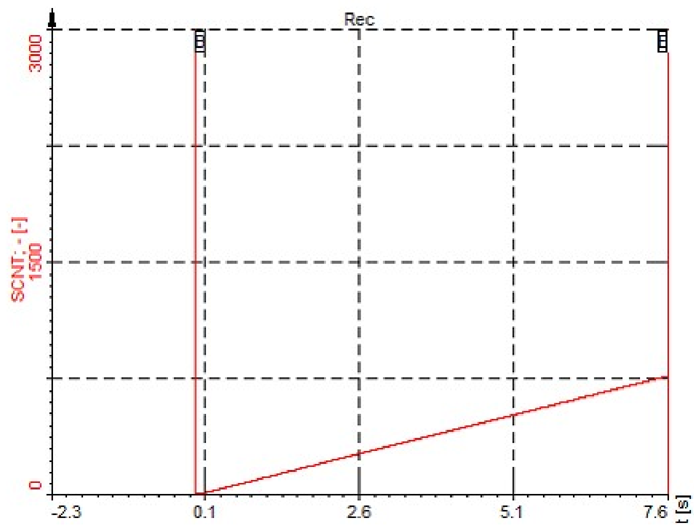
Signal functions and generation		
Function	**Syntax**	**Description**
SCNT	SCNT	Number of samples acquired
SR	SR	Sample rate of acquisition
TIME	TIME	Time elapsed (in seconds)
SINE	SINE(f [,ps])	Generates a *sine* w ave w ith frequency f [Hz] optionally w ith phase shift ps [radian]
SQUARE	SQUARE(f [,ps])	Generates a *square* w ave w ith frequency f [Hz] optionally w ith phase shift ps [radian]
TRIAN	TRIAN(f [,ps])	Generates a *triangle* w ave w ith frequency f [Hz] optionally w ith phase shift ps [radian]
NOISE	NOISE	Generates *noise* (random number betw een ±1)

Signals examples and their explanations:

SCNT

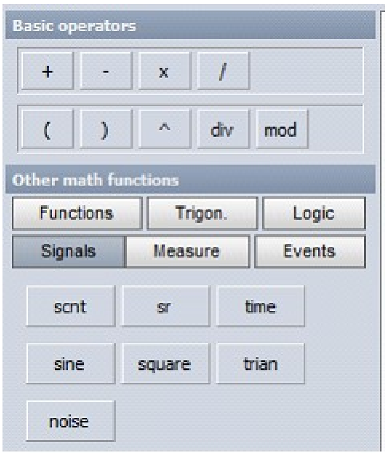
Sample counter Delivers the samples acquired from start of the measurement.
The counter will be rest at start storing.





The above picture is showing the result, while at the bellow picture the reset after start storing could be seen.

Example: **Create Angle Signal**



Example: If external clocking is used and the signal should be shown in a 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.

MATH: ***SCNT MOD 720*** will deliver a saw tooth 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 a XY diagram to show the result angle based.

The only disadvantage will be, if we get 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 an Orion Counter, where this is eliminated.

Because the TRG PULS will reset the counter every revolution.

TIME

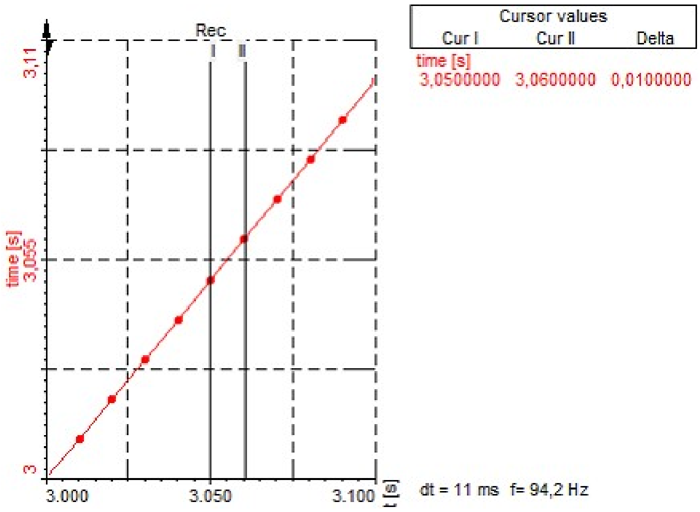
Time function Is providing the *elapsed time* of the measurement in *seconds*.

This is a similar function like *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.

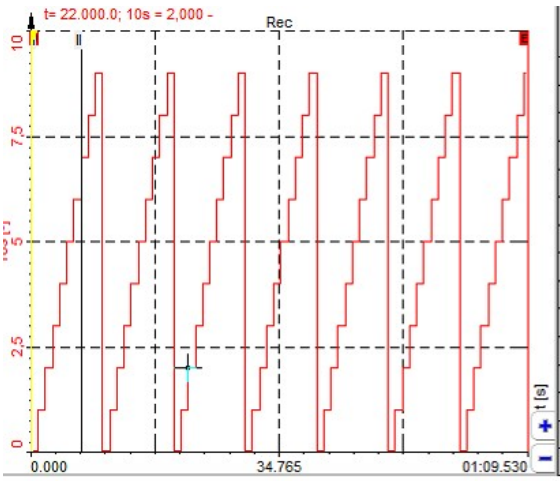
The screen shoot below is showing the time function in a recorder. We could see that the resolution is 0,01s and also dt=10ms which results in a sampling rate of 100Hz. So the time will count up 0,01s after every sample @ sampling rate of 100Hz or 0,001s @ sampling rate 1kHz.



Example: **Show actual value averaged every 10s in a list and export it to Excel.**

MATH: ***(Time MOD 60)***

This will crate a saw tooth with a period time of 60s. Look to the picture below. This is channel will be use as 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 @ Export only the asynchrony 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

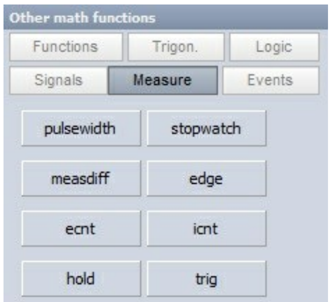
Exported	Index	Type	Acq. rate	Dimension	Name
No	0	AI 0	10000	Scalar	pressrue
No	1	Math 0 (Formula)	single	Scalar	Frm0/Formula 0
No	2	Math 1 (Formula)	10000	Scalar	Frm1/10s
Yes	3	Math 2 (Latch math)	0,1	Scalar	Latch0/Latch index
Yes	4	Math 2 (Latch math)	0,1	Scalar	Latch0/pressrue/Latch

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**

Measurement functions		
Function	**Syntax**	**Description**
PULSEWIDTH	PULSEWIDTH(*cond* [, *rearm*])	Measures time [s] betw een tw o *cond* edges (*cond* jumps from 0 to 1), rearm edge optional
STOPWATCH	STOPWATCH(*cond1*, *cond2*)	Measures time [s] betw een *cond1* and *cond2* edges (*cond* jumps from 0 to 1)
MEASDIFF	MEASDIFF(*value*, *cond1*, *cond2*)	Measures *value* difference betw een *cond1* and *cond2* edges (*cond* jumps from 0 to 1)
EDGE	EDGE(*cond* [, *rearm*])	Returns 1 w hen 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 w here the *cond* have a logical value 1
HOLD	HOLD(*value, cond* [, *rearm*])	show s and holds value, w hen *cond* goes from 0 to 1 w ith optional reams condition
TRIG	TRIG	has a value of 1 w hen a store condition appears

Measure examples and their explanations:

HOLD

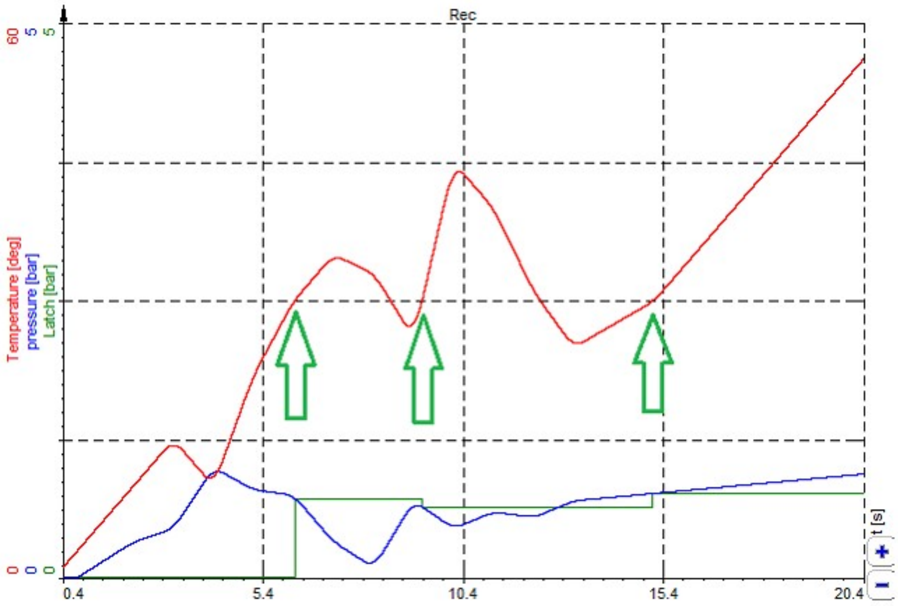
Hold function

Hold(value,latchcondition,[rearmcondition])

The hold function is used to latch or hold a single value if a condition is met.

Example1: ***Hold('pressure', 'Temperature'> 30)***
Hold(value,latchcondition)

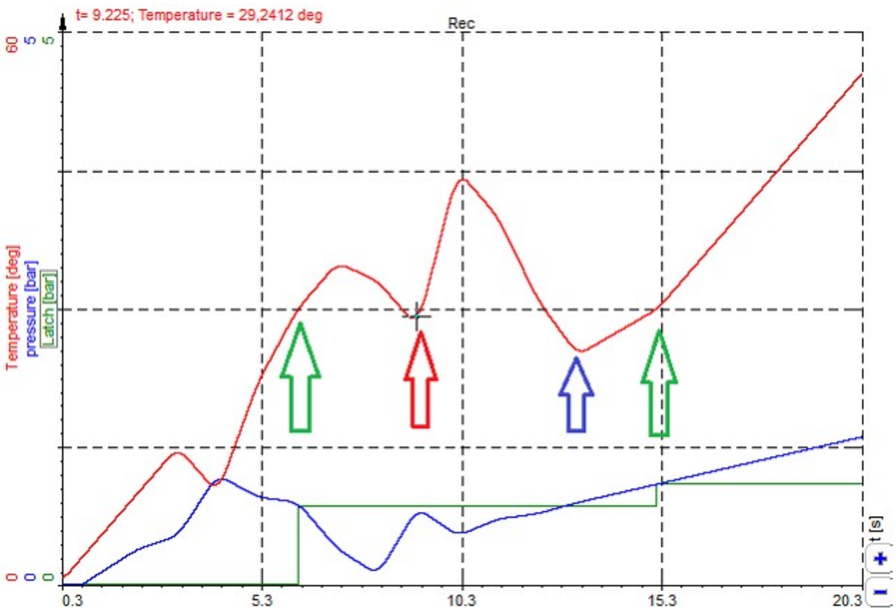
In the example above the function will hold the actual pressure if the temperature is higher or equal 45deg.



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.

Example2: ***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 a 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 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

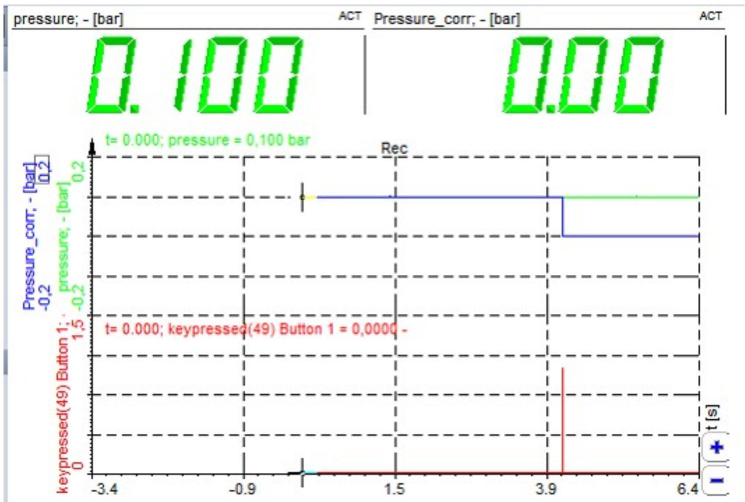
Example3: **Remove offset from a Channel**

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.

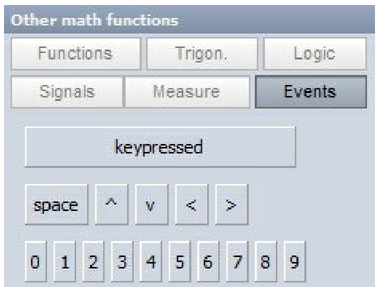
So even during the measurement a offset compensation could be done by pressing a specific key. The picture below is showing an example. The '*keypressed(49)*' channel is indicating the pressed key in an additional math channel to make it more obvious.



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**



Event functions		
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.

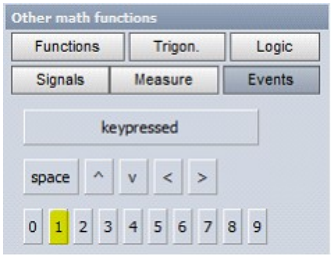
Events examples and their explanations:

Keypressed

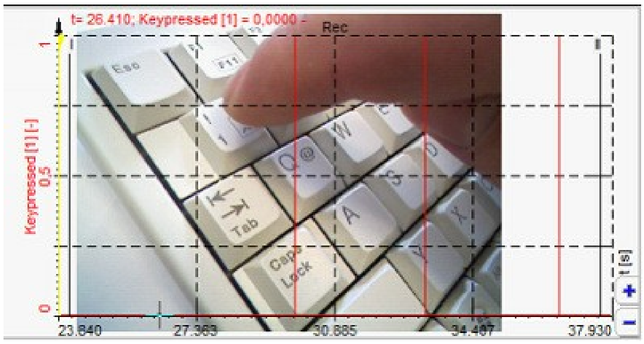
Keypressed function

keypressed(49) .... produces signal from 0 to 1 with a duration of 1 sample if [1] button is pressed.

A few predefined key are already prepared like shown in the picture below. So if you press the [1] button below the line '*keypressed(49)*' will be automatically added to your formula.



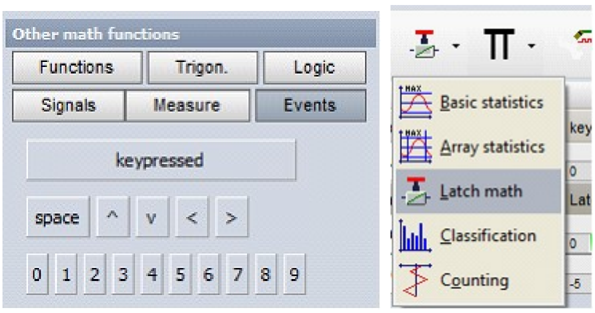
Every time the [KEY 1] is pressed on your keyboard a signal is produces.

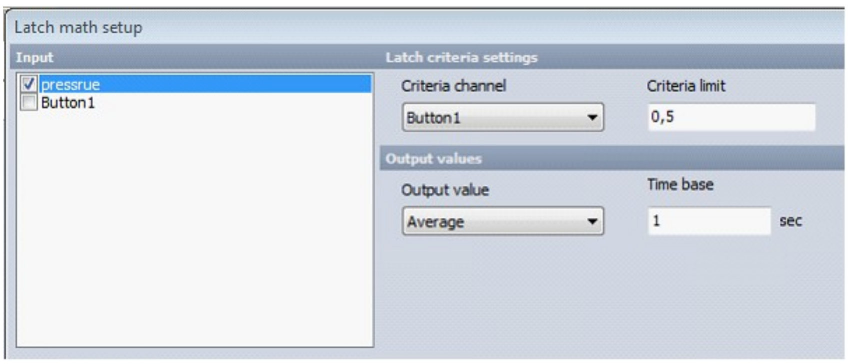


Almost *any* key could be used in the '*keypressed*' function. The value in the brackets is reflecting the virtual key code in decimal.

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)*.

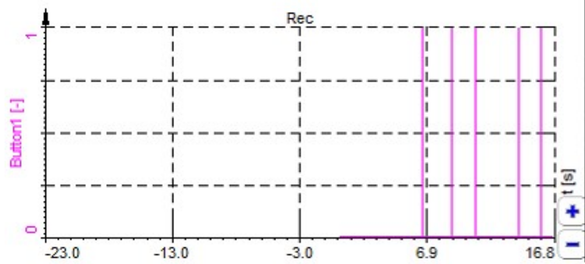
Example: **Latch Value into List**





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

2.10.2.2 Array operations

DEWESoft v7 introduces **array channels**. Those channels can come *from* scope trigger, FFT, CPB, STFT, classification, counting and other new procedures. Formula has operations which works on those array channels.

Extracting subset of the array

To *extract* a value from array, use brackets with index. Let' s say we want to extract one single value of the FFT:

'AI0/AmplFFT'[10]

For extracting *sub set* (like 10..20) please use version 7.1.

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 *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 .

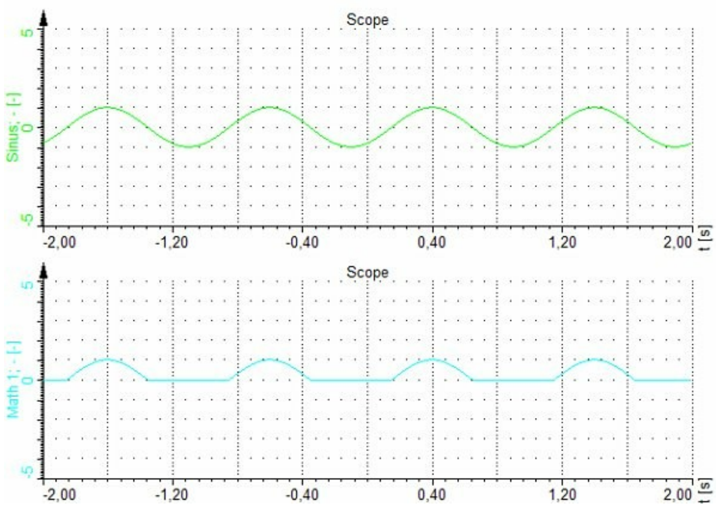
IMPORTANT: *arrays with different sizes can't be combined into single formula*

2.10.2.3 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 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 original signal will result in *cutting* the *negative* part of the signal.

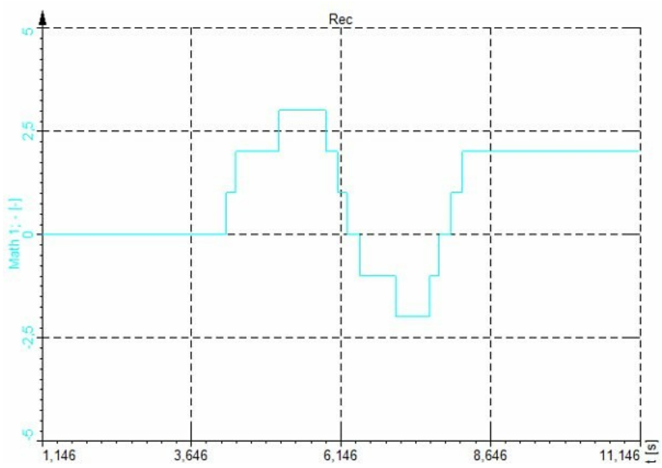


2.10.2.4 Example of input events

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.

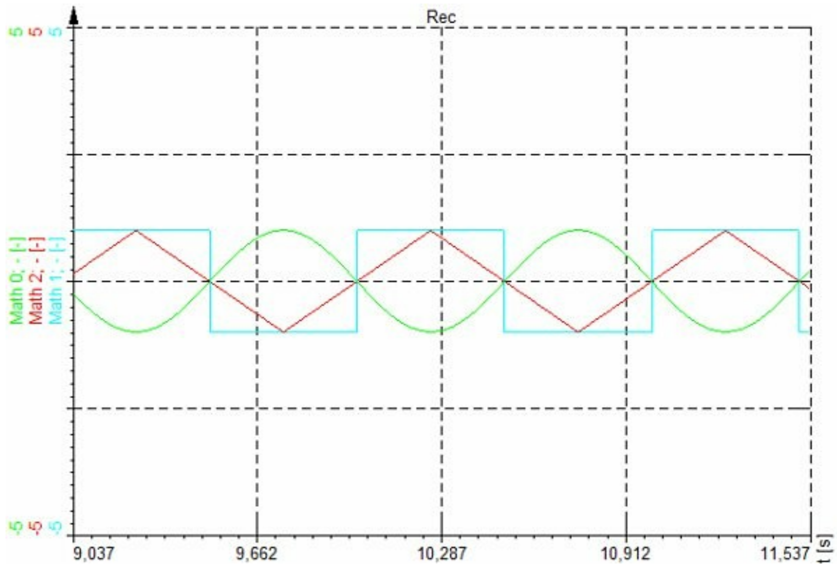


2.10.2.5 Example of signals

The **signal generation** is useful for making *test signal* for testing. We have basic functions like *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)

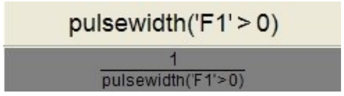
The example above results in the following signals:



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.

2.10.2.6 Example of measure functions

The following example shows the **measurement** of *pulse width*. 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 *input signal*. The pulsewidth function will than measure the *time* between each transition. If we want to measure the *frequency*, we can calculate *inverse* of this function as seen below.



The graph shows a *pulse width* measurement (*Math0*) and *frequency* measurement (*Math1*).

'pulsewidth' function is useful for *digital signal*. The resolution of 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.

2.10.2.7 Example of stopwatch

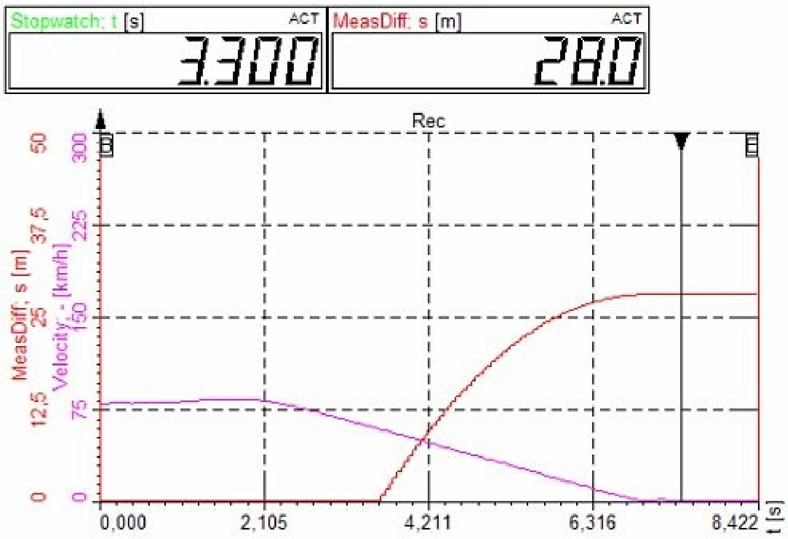
There are two functions in **Measure** section of math which helps to measure *time* and *value* between *events*. The ***Stopwatch*** functions enables us to measure time between a *start* and *stop* event. The 'measdiff' function makes same measurement, but we can define the measurement *channel* instead of time.

In the example shown we measure the *brake time* and *brake distance*. 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)

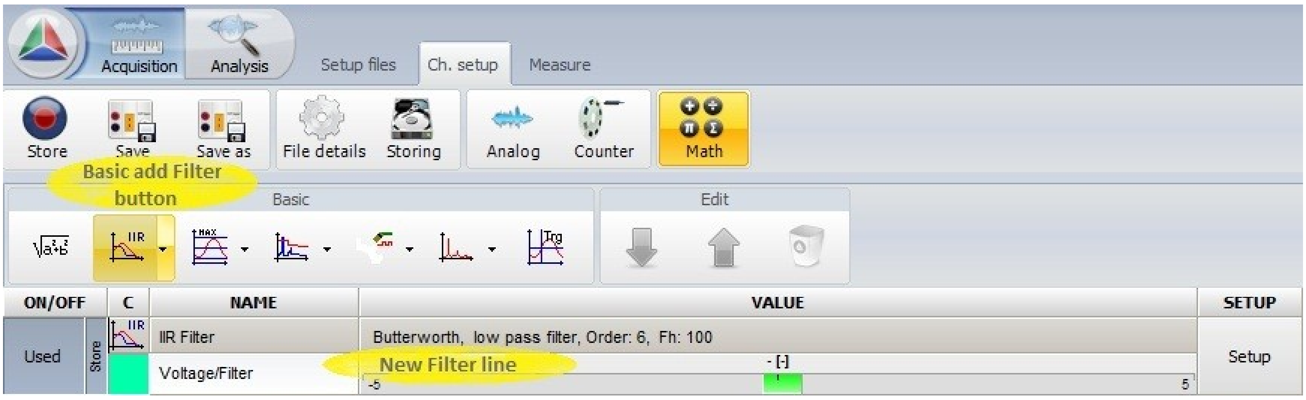
The graph below shows the example of measured data - *input* channels of *velocity* and *distance*. The *meter* shows the time and distance of braking.



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.

2.10.3 Filters

New **Filter** can be *add* on the DEWESoft Math *setup* screen by selecting *basic add* **Filter** button or from the **Filter** *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**

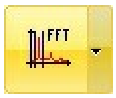
When you press the **Setup** button on activated *new Filter line*, the **Filter setup** window will open. This window depends on selected filter *type* and his name appears on title line of window:

- **IIR Filter setup**
- **FIR Filter setup**
- **FFT Filter setup**
- **Envelope setup**

Add new Filter

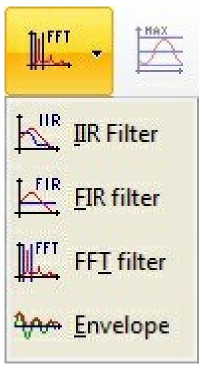
Available **Filters** on Math *setup* screen are: **IIR Filter**, **FIR Filter**, **FFT Filter** and **Envelope**. New *Filter* can be **added** by selecting:

- *basic add* **Filter** button (when appropriate filter is shown on icon)



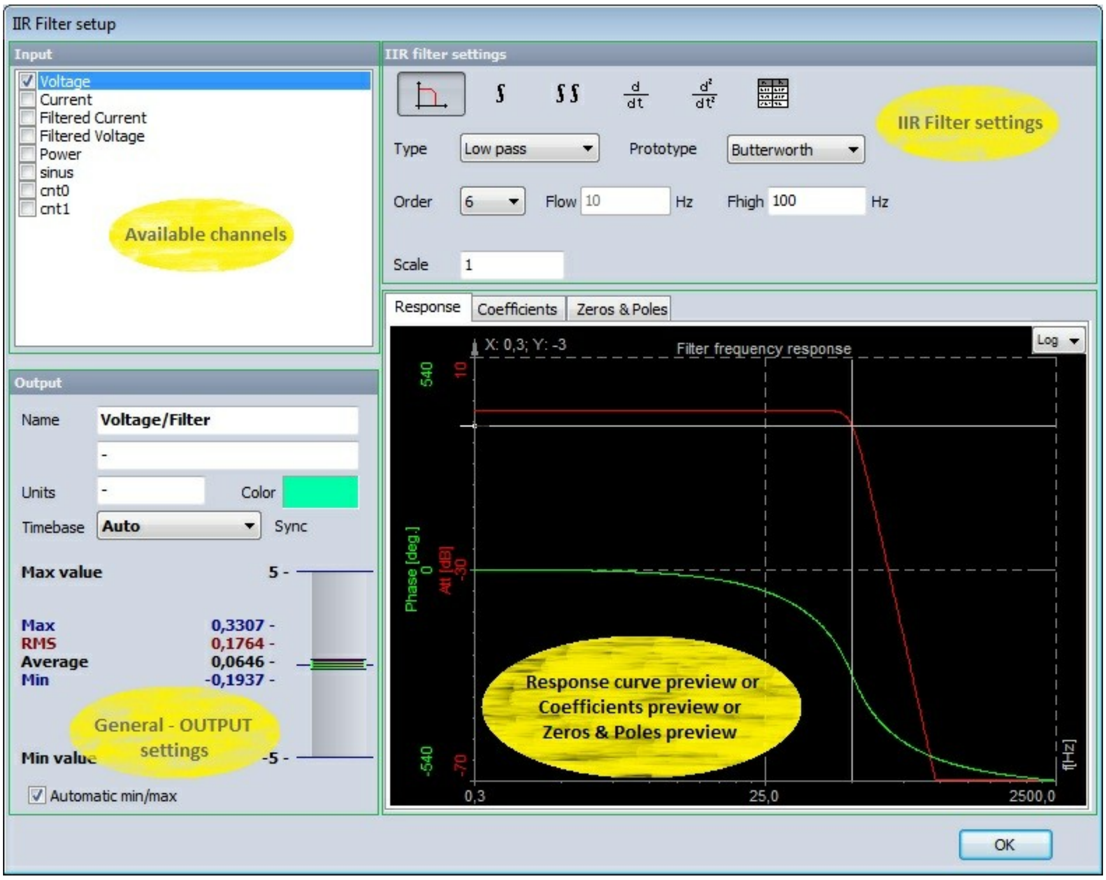
- from the **Filter** *list* → for *filters list* display select ▾ *triangle sign* on basic button

  When we select *Filter* from *list*, icon changes to *selected* filter, so we can choose the same filter again by simply clicking on the basic *add Filter* button.



2.10.3.1 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 abbreviation for ***infinite impulse response***. It means that the response to the impulse will be *non zero* over *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 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:

Standard **Integration** **Double integration** **Derivation** **Double derivation** **Custom**

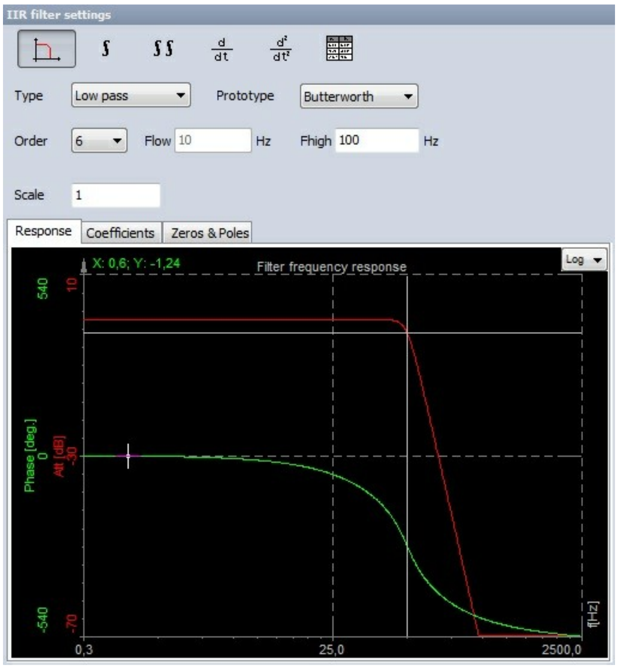
2.10.3.1.1 Standard filter

Standard filter allows basically doing high, low, band pass and band reject filters.

With ***Filter*** option on IIR Filter settings section you can *set*:

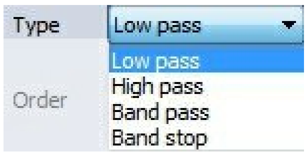
- **Type** and **Prototype**
- **Order**
- *Cut-off Frequency* - **FLow** and **FHigh**
- **Ripple**
- **Scale**

You can see *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 **Type** of Filter from list between:



- Low pass

  Low pass filters cuts 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, for example band around 50 Hz to cancel the supply voltage effects.

Prototype of Filter

You can select **Prototype** of Filter from list between:

- 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*.

- Butterworth

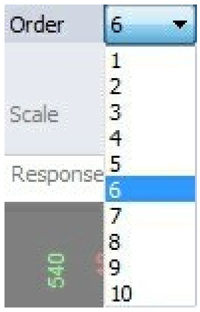
  Butterworth is *without the ripple* and *maintains* the shape with *higher orders*.
  Roll-off is defined with *(-20 db/decade)*order*. It is also known as maximally flat magnitude, suggesting that the filter response is *really flat* in the pass band.

- 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 list. The order of the filter defines the steepness of the filter. For the Butterworth the roll-off is *(-20 db/decade)*order*, so for the sixth order the roll-off would be -120 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 *each two* ***orders*** of the filter.

Cut-off frequency

You can enter Cut-off frequency:

- **FHigh** - *High* frequency

  You can enter *FHigh* for Low pass, Band pass and Band stop filter.

- **FLow** - *Low* frequency

  You can enter *FLow* for High pass, Band pass and Band stop filter.

- Both *High* and *Low* frequency

  You can enter **FHigh** and **FLow** for Band pass and Band stop filter.

FLow value must be always *lower* than FHigh. 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.

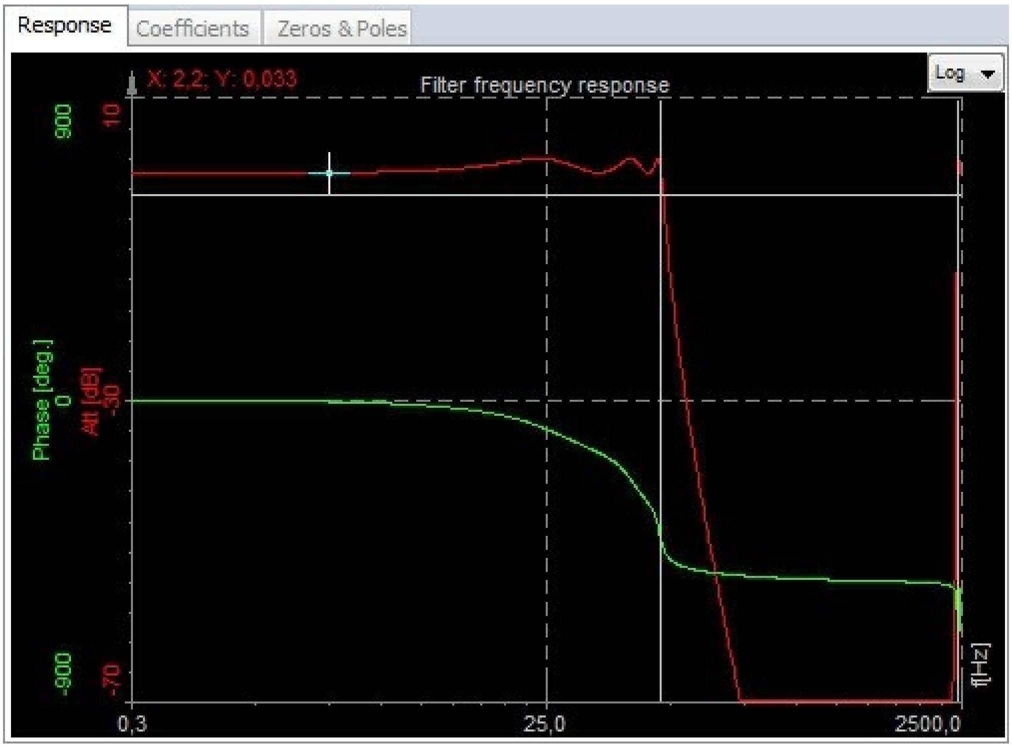
NOTE: *This field appears only for Chebyshev filter Prototype*

Scale

For filters you can enter also **Scale**. Scale factor means the *final multiplication factor* before the value is written to output channel. It helps us to change the unit, for example. A good example of using the Scale is shown in 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.

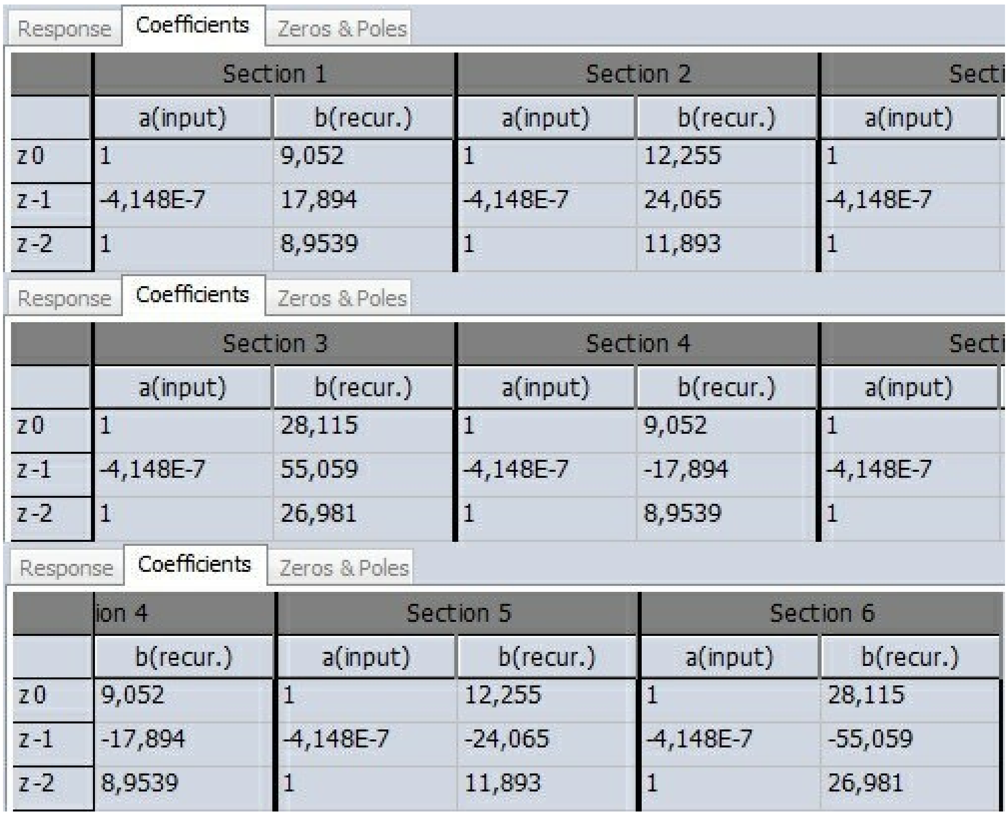


The red graph shows the amplification / attenuation of the filter in dB related to the frequency. To refresh the memory, dB scaling is calculated with equation $a\ [\mathrm{dB}] = 20 * \log10(A)$, so the attenuation ratio is calculated with $A=10\hat{}(a/20)$.

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 in 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 right mouse click on the table to be use from / in other calculation programs.


Response | Coefficients | Zeros & Poles

	Section 1		Section 2		Secti
	a(input)	b(recur.)	a(input)	b(recur.)	a(input)
z 0	1	9,052	1	12,255	1
z-1	-4,148E-7	17,894	-4,148E-7	24,065	-4,148E-7
z-2	1	8,9539	1	11,893	1

Response | Coefficients | Zeros & Poles

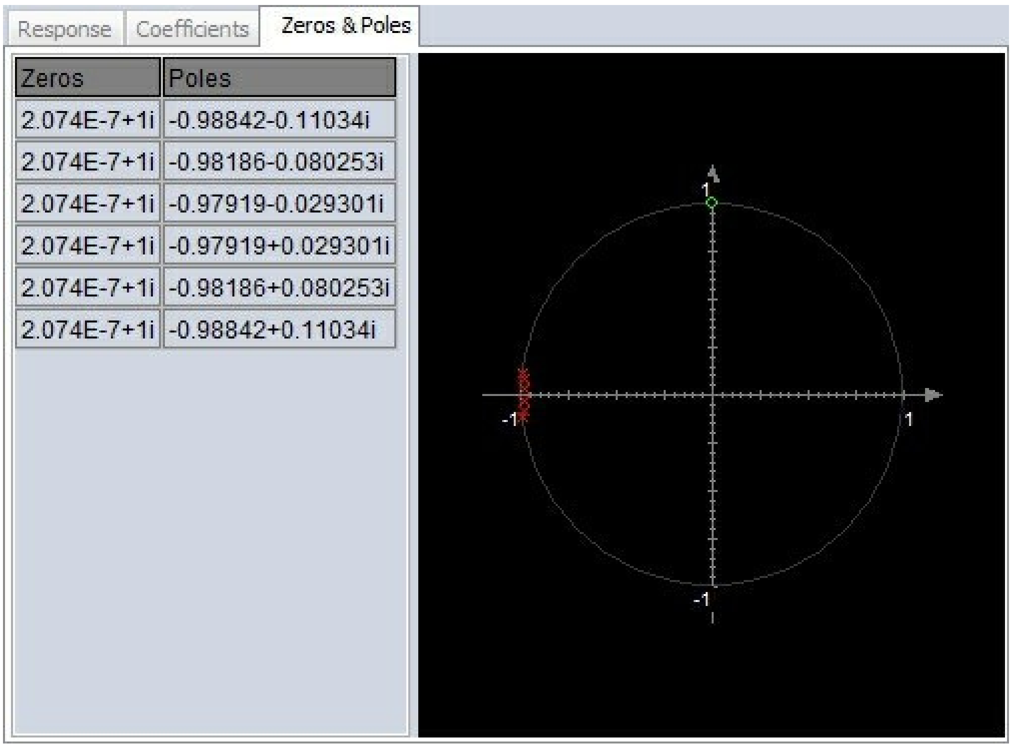
	Section 3		Section 4		Secti
	a(input)	b(recur.)	a(input)	b(recur.)	a(input)
z 0	1	28,115	1	9,052	1
z-1	-4,148E-7	55,059	-4,148E-7	-17,894	-4,148E-7
z-2	1	26,981	1	8,9539	1

Response | Coefficients | Zeros & Poles

	ion 4	Section 5		Section 6	
	b(recur.)	a(input)	b(recur.)	a(input)	b(recur.)
z 0	9,052	1	12,255	1	28,115
z-1	-17,894	-4,148E-7	-24,065	-4,148E-7	-55,059
z-2	8,9539	1	11,893	1	26,981

On **Response** curve preview you can choose between Log*arithmic* and Lin*ear* 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

Zeros	Poles
2.074E-7+1i	-0.98842-0.11034i
2.074E-7+1i	-0.98186-0.080253i
2.074E-7+1i	-0.97919-0.029301i
2.074E-7+1i	-0.97919+0.029301i
2.074E-7+1i	-0.98186+0.080253i
2.074E-7+1i	-0.98842+0.11034i

2.10.3.1.2 Integration and double integration

Integration

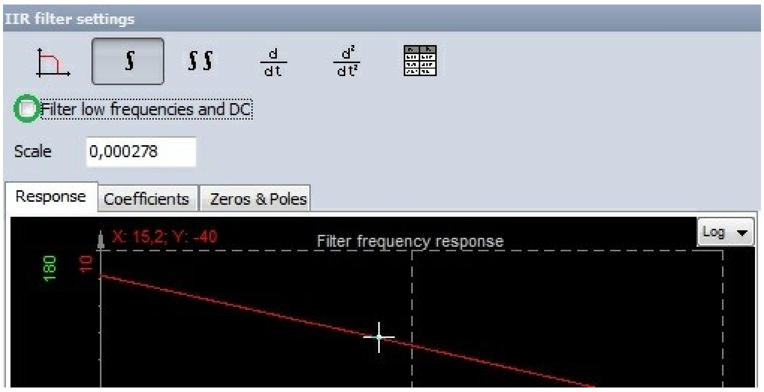
There are two main purpose of **integration**: first is to *get* for example traveled *distance* from *measured velocity* or to get the *energy* from *measured power*. Another type of integration is to *calculate* the *velocity vibration* from the *measured acceleration*.

A clear difference between two basic functions is that in 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.

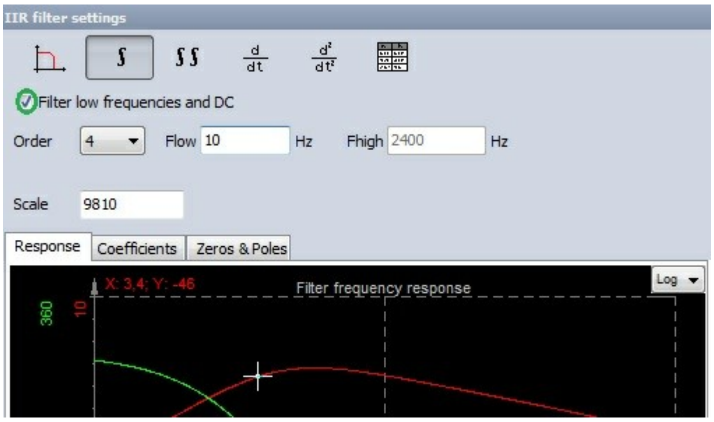
1. Let's first take a look how to integrate distance from velocity. If we have for example velocity in km/h as an input channel, we choose a filter, integration, and switch OFF the option to 'Filter low frequencies and DC'. Now the only thing left is to define the Scale. We have to know that integration 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 power in kW, we have at output:
   $kW * sec$=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 amplifier and AD converter offset. This offset will result in 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 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 virtually entire frequency range.

For ***vibration*** measurement, usual value is from 3 to 10 Hz for *general* purpose measurement. For *low frequency* vibration like *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

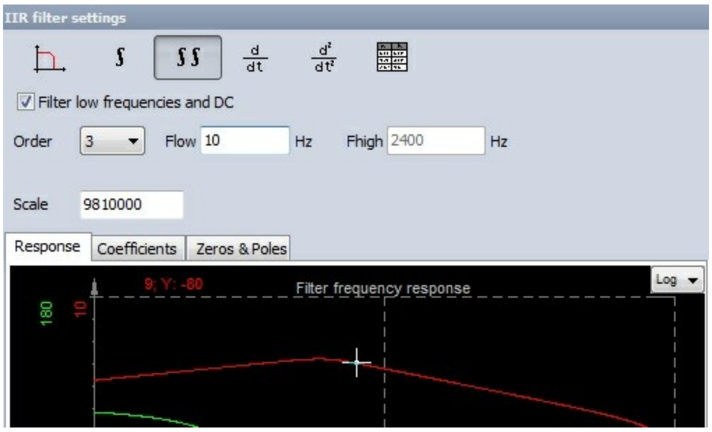
So we need to enter 9810 in the field scaling.

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



2.10.3.1.3 Derivation and double derivation

Derivation and **double derivation** is used, as the name already suggest, to *calculation* **derivation** of chosen input signals. Here the application range is not divided in 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*.

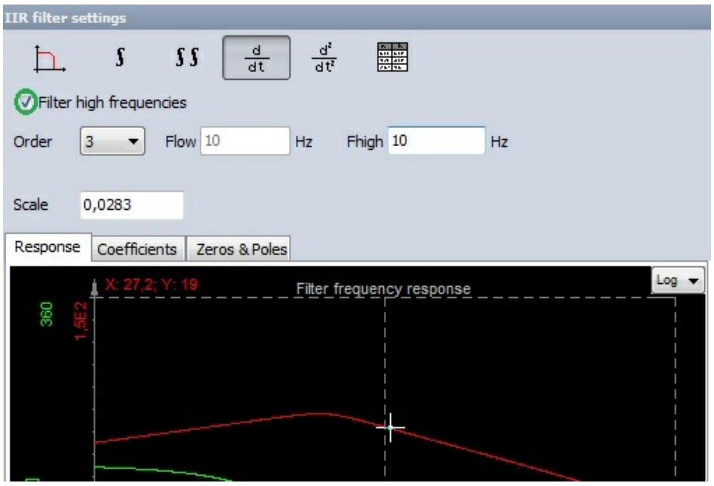
However this might produce very noisy signals, especially with high sampling frequency. We can look at the derivation as the 20 db / decade *growing filter* in the *frequency* domain. Sometimes it is therefore nice to cut the high frequency contents. We can choose an option to Filter high frequencies. 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 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.



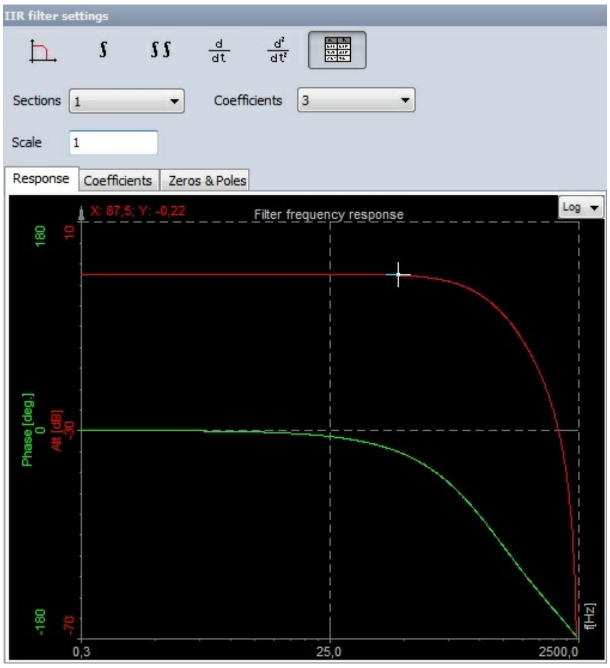
2.10.3.1.4 Custom filter

The **Custom defined filter** setup requires *setting*:

- number of Sections and Coefficients
- 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 *number of rows* in the table. This basically defines the filter *order*.



The last think 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* 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 general formula for second order filter:

$$H(s) = \left(g_0 + g_1 \cdot s + s^2\right) / \left(h_0 + h_1 \cdot s + h_2 \cdot s^2\right) \qquad (1)$$

To get the filter coefficients in the **z**-*plane* (*time domain* coefficients) we need to use the bilinear transformation:

$$s = 2 \cdot f_s \cdot \left(1 + z^{-1}\right) / \left(1 - z^{-1}\right) \qquad (2)$$

where f_s is the *sample frequency*. The upper equation reveals 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:

$$\begin{aligned} & g \cdot \left(1 + z^{-1}\right)^2 + 2 \cdot g_1 \cdot f_s \cdot \left(1 - z^{-1}\right) \cdot \left(1 + z^{-1}\right) + 4 \cdot g_2 \cdot f_s^2 \cdot \left(1 - z^{-1}\right)^2 = \\ & \left(g_0 + 2 \cdot g_1 \cdot f_s + 4 \cdot g_2 \cdot f_s^2\right) + \left(2 \cdot g_0 - 8 \cdot g_2 \cdot f_s^2\right) \cdot z^{-1} + \left(g_0 - 2 \cdot g_1 \cdot f_s + 4 \cdot g_2 \cdot f_s^2\right) \cdot z^{-2} \end{aligned} \qquad (3)$$

The first third of the equation is valid for z^0 coefficient, second for z^{-1} and 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 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*:

$$H(s) = 1/\left(1 + sqrt(2) \cdot s/\omega_c + (s/\omega_c)^2\right) \tag{4}$$

where the ω_c is the *cutoff frequency* in rd/s. We have to adapt the cutoff frequency to the sample rate with prewarping:

$$\omega_c = 2 \cdot f_s \cdot \tan(\pi \cdot f_c / f_s) \tag{5}$$

If we write out the factors for this filter:

$$g_0 = 1; \quad g_1 = 0; \quad g_2 = 0; \quad h_0 = 1; \quad h_1 = sqrt(2)/f_c; \quad h_2 = (1/f_c)^2 \tag{6}$$

Now let's make the following filter:

cutoff frequency	f_c=100 Hz
sampling rate	f_s=1000 Hz

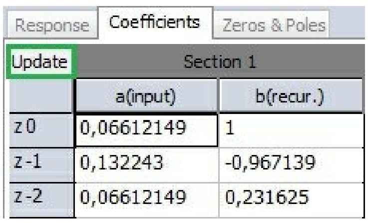
First we do the pre warping:

f_{cp} = 2 * 1000 * tan(π * 100/1000) = 649,8 rd/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	b(z-2) = 1 - 2 * sqrt(2) * 1000/649,8 + 4 * 1000²/649,8² = 6,12
a(z-1) = 2*1-0 = 2	b(z-1) = 2 * 1 - 8 * 1000²/649,8² = -16,944
a(z0) = 1+0+0 = 1	b(z0) = 1 + 2 * sqrt(2) * 1000/649,8 + 4 * 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.



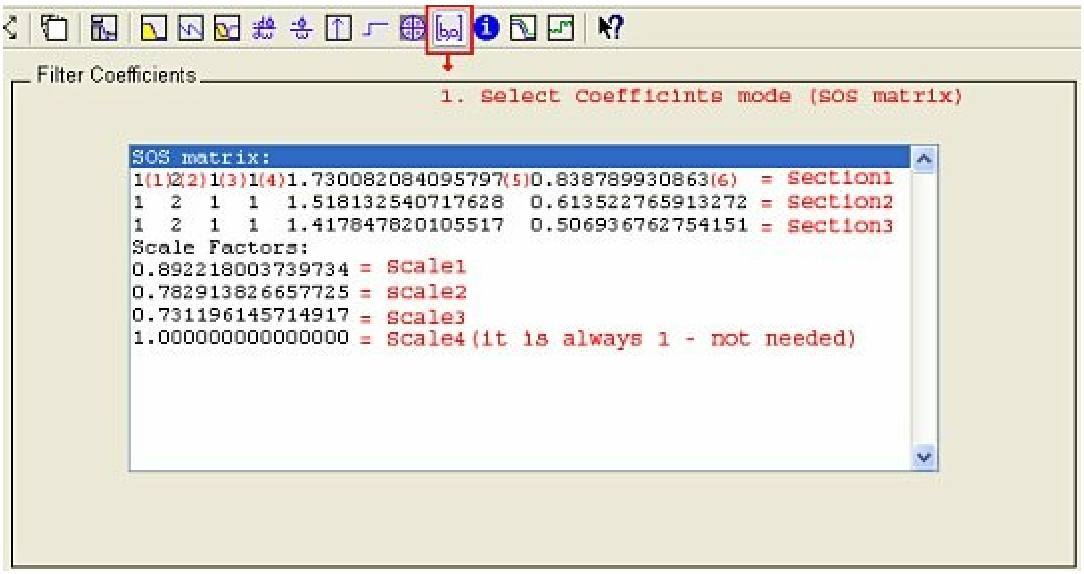
Remember, this is *valid only* for a sampling rate of 1000 Hz. For others, we need to *recalculate* the f_{cp} and coefficients.

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 from Matlab**.

2.10.3.1.5 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* 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*:

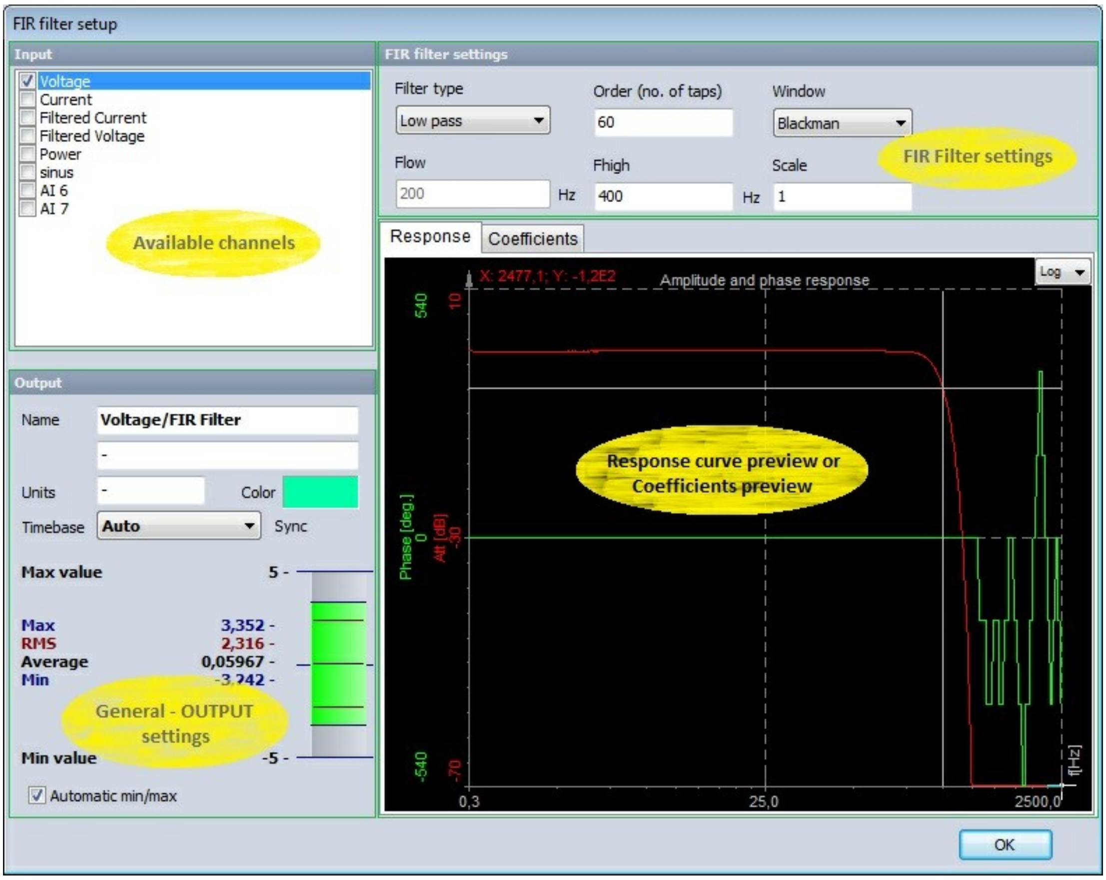
	a(input)	***b(recur.)***
z0	*Scale i * Section i*(1)	*Section i*(4)
z-1	*Scale i * Section i*(2)	*Section i*(5)
z-2	*Scale i * Section i*(3)	*Section i*(6)

The coefficients for our example are calculated below:

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

2.10.3.2 FIR Filter setup

When you press the **Setup** button on *new* 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 have the calculation delay, we can use the 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 phase shift. The drawback of **FIR** filters are that they will take *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

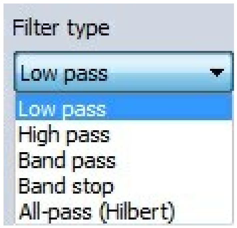
For ***FIR Filter*** you can *set*:

- **Filter type** and **Window type**
- **Order**
- *Cut-off Frequency* - **FLow** and **FHigh**
- **Scale**
- Kaiser window type - **Ripple**

You can see *effect* of this settings 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 cuts 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.

Window type

You can *select* **Window** type from the list.

The window *defines* the *behaviour* of the filter in the transition and the *stop band* (the height of the side bands and the width of the main band).
For common usage Blackman window is quite good choice, because the side bands are extremely low.

for hints about recommended using ***Window type*** → *see* → Reference Guide → **Theory of frequency analysis**

Order

In this field you can *enter* **Order**. The order of the filter defines the *number of coefficients* of the filter and that will directly affect the slope of the transition band. The filter order is not directly comparable with FIR filter.

Kaiser window type - **Ripple**

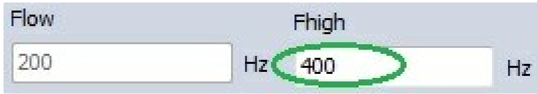
When a Kaiser Window type is selected, new **Ripple** field appears on the right side of the Window type field. In this filed you can enter ripple value in dB. It tells the maximum allowed pass band ripple of the filter. The more this value is, the bigger will be non-linearity in the pass band, but the filter will be stepper.

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:

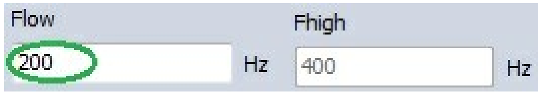
- **FHigh** - *High* frequency

  You can enter *FHigh* for **High pass**, **Band pass** and **Band stop** filter.



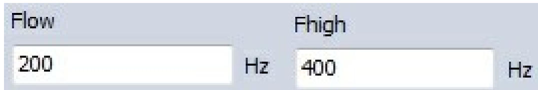
- **FLow** - *Low* frequency

  You can enter *FLow* for **Low pass**, **Band pass** and **Band stop** filter.



- Both *High* and *Low* frequency

  You can enter FHigh and FLow for **Band pass** and **Band stop** filter.



FLow value must be always *lower* than FHigh. 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

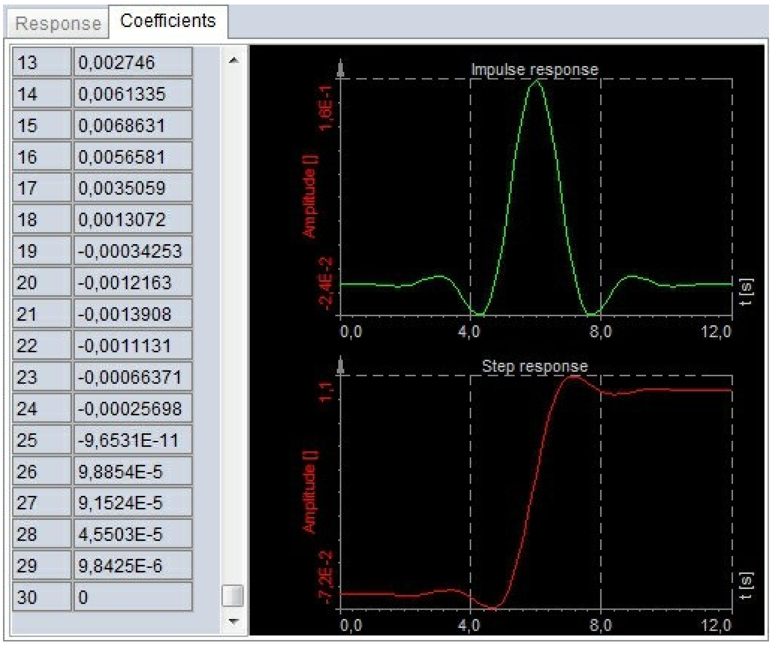
For filters you can enter also **Scale**. Scale factor means the *final multiplication factor* before the value is written to output channel. It helps us to change the unit, for example. A good example of using the Scale is shown in Integration section.

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 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.



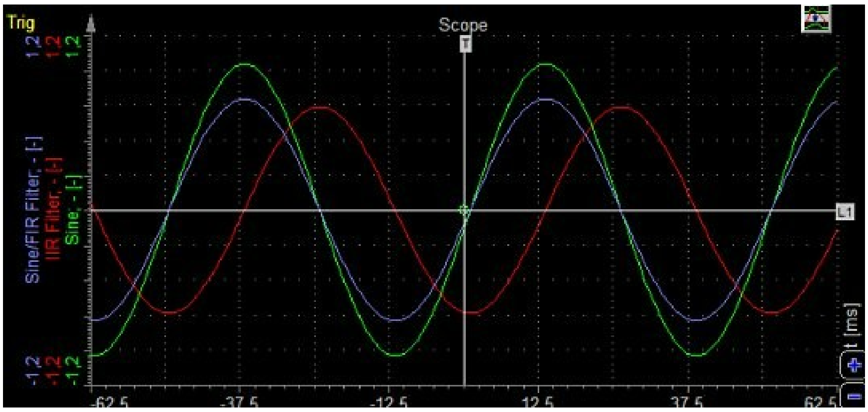
On *Response* curve preview you can choose between Log*arithmic* and Lin*ear* display, you can also *edit* coordinates value and *auto scale* Y axis.

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 higher number of coefficients.

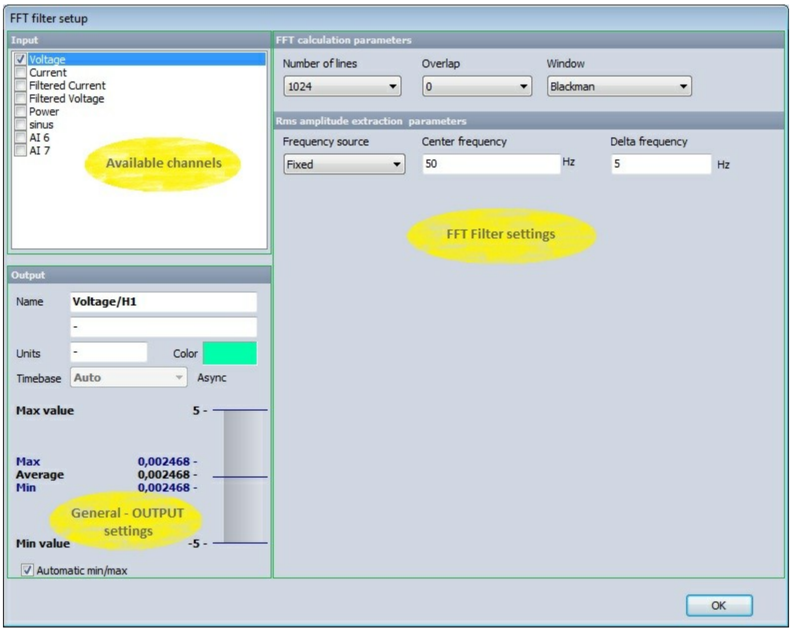


Enough of the down sides, let's look at the response graph at 20 Hz (exactly at the limit). The green curve is the *original sine* wave while the red one is *calculated* with IIR filter. We can clearly see the *phase delay* of the output.

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*.

2.10.3.3 FFT Filter setup

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.

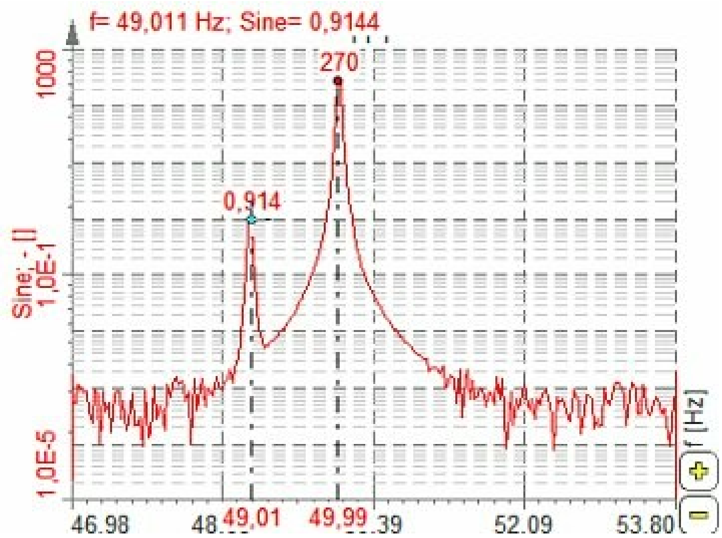
for detailed information about ***basic settings*** of the *input* and *output channels see* → **Setup screen and basic operation**

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 near by where it wouldn't be possible to choose IIR filter which would extract this low amplitude.

The example below shows the electro motor 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*:

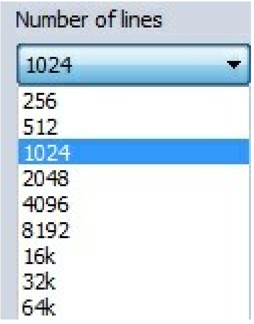
FFT calculation parameters

- **Number of lines**
- **Overlap**
- **Window**

Amplitude extraction parameters

- **Frequency source**
- **Center frequency**
  only for Fixed *Frequency source*
- **Frequency channel**
  only for Tracking *Frequency source*
- **Number of harmonics**
  only for Tracking *Frequency source*
- **Delta frequency**

Number of lines



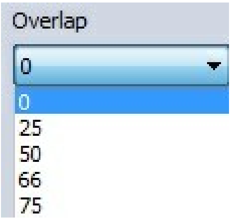
You can select **Number of lines** from list.

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 higher result update.

for detailed information about ***Line resolution*** →

see → Reference Guide → **Properties of the Fourier transform**

Overlap



You can select **Overlap** from list. Overlap defines (same for *FFT averaging*) how many 'old' data is taken for next calculation. This increases the result update rate with same number of lines.

for detailed information and hints about ***Overlap*** →

see → Reference Guide → **Theory of frequency analysis**

Window type

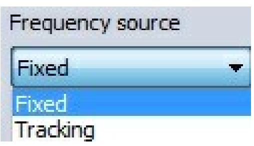
You can select **Window** type from the list.

The window *defines* the *behaviour* of the filter in the transition and the *stop band* (the height of the side bands and the width of the main band).

for hints about recommended using ***Window type*** →

see → Reference Guide → **Theory of frequency analysis**

Frequency source



You can select Frequency source from list:

- **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

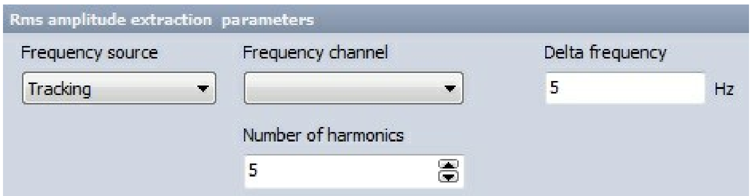
In this field you can *enter* **Delta frequency** in Hz. This value depends on the wanted frequency band. It also *depends* on the *window* and *number of lines* (line resolution). In our example we would choose 0.5 Hz since we don't want that 50 Hz value will appear in the result.

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

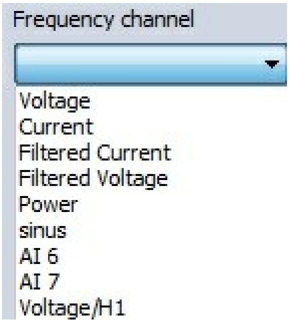
If we choose Tracking Frequency source, we need to enter the **Frequency channel** and **Number of harmonics** instead of *Center frequency*:



Frequency channel

Frequency channel is the channel with *current frequency* which needs to be *extracted*. The unit of this channel must be in Hz.

Frequency channel can be selected from list:

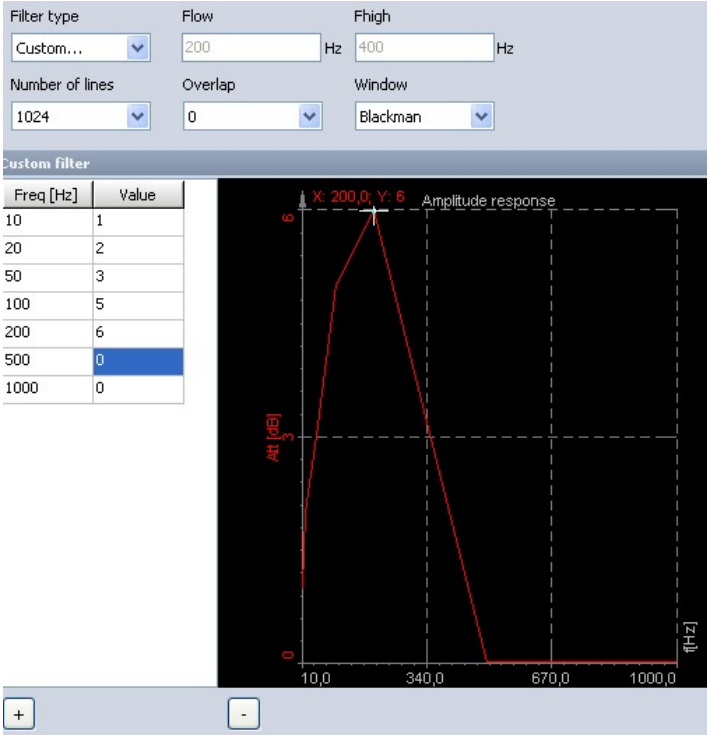


Number of harmonics

Number of harmonics describes how many harmonics needs *to be extracted* from the spectrum. If we enter a value of 5, there will be 5 channels created for *each input channel*. *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

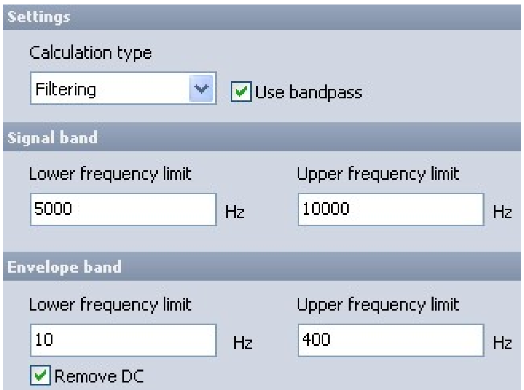
If Custom filter is selected from the Filter type, then we can design our *own* filter. With this option we can create any type of filter curve in frequency domain and calculate RMS value. Sometimes it is not easy to define filter characteristics in the time domain, but we have it defined in the frequency 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 *external clock*.

2.10.3.4 Envelope setup

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.

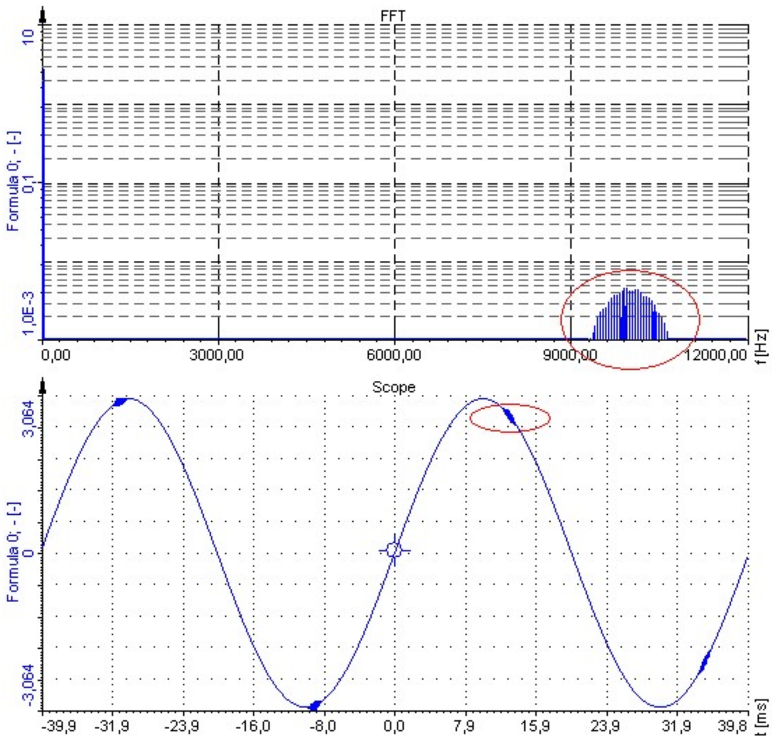
Filtering uses *filter* procedure for envelope calculation. Filtering is a standard procedure for calculating envelope used also in other implementations.

Peak detection uses the procedure of *detecting peak values* in the signal. Peak detection is a procedure which calculates amplitudes more exact than filtering.

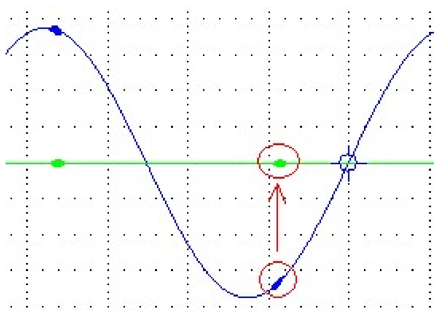
Use Bandpass check box enables or disables the *first* stage of calculation - *band pass* filtering. Acceleration sensor measures entire frequency range and acquires unbalance, misalignment and other faults on the machine. Ball bearing errors have very low energy and therefore is a small contribution in 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 have to know also that inner ring, outer ring, cage and balls have different typical repeating frequency depending on the geometry of the bearing and the rotational frequency.

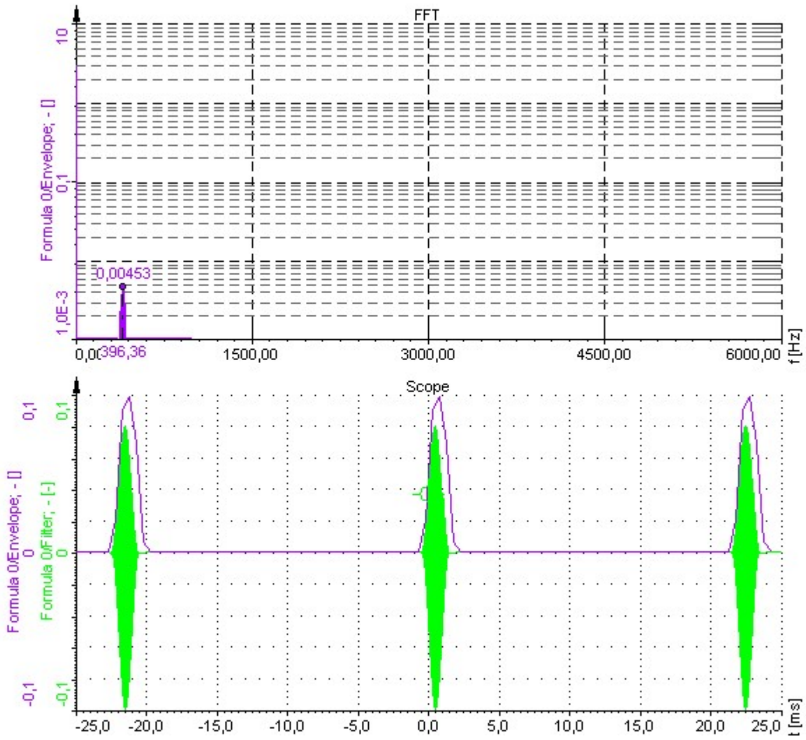
To only focus on these high frequencies of the ringing, we have to look at the original frequency spectrum. I have generated a sine wave which have a small 10 kHz rings on top. In frequency domain we don't see at all the frequency that the ringing repeats, but only a major sine wave (could come from unbalance) and very high frequency coming from the bearing.



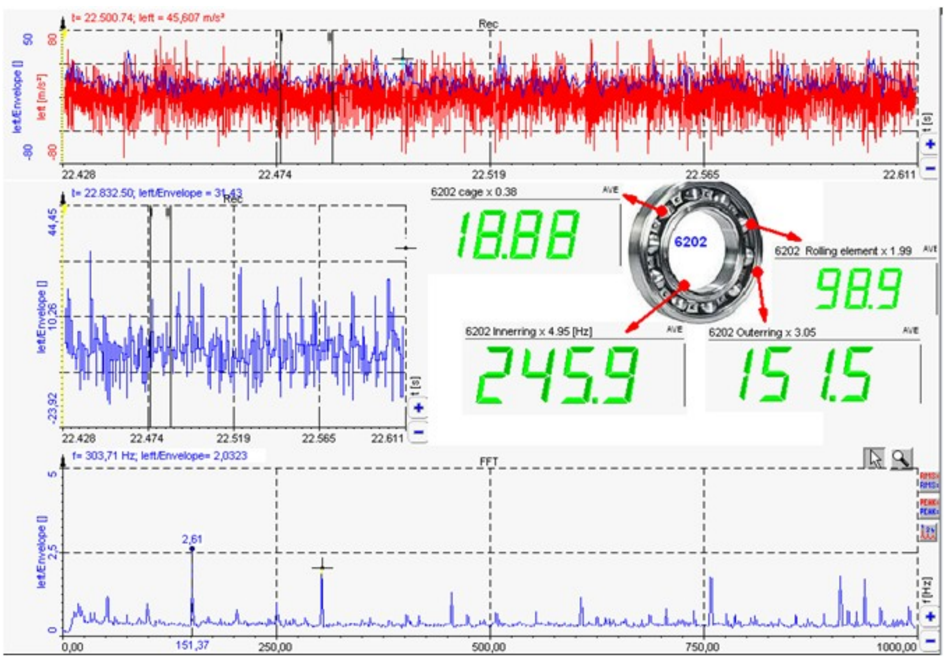
Bandpass filtering in the envelope detector must be set to *remove all components except* ringing of the ball bearing. This can be usually found around 10 kHz. In my example I have set lower frequency limit to 6 kHz and upper limit to 12 kHz to get all the energy. Signal after filtering would look like this:



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 do 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 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 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*.



Following picture shows the typical damage of the outer ring of the large bearing (courtesy of Kalmer d.o.o. Trbovlje).

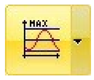
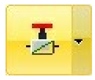
2.10.4 Statistic group

New **Statistics** can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Basic Statistics** button.

Add new Statistics

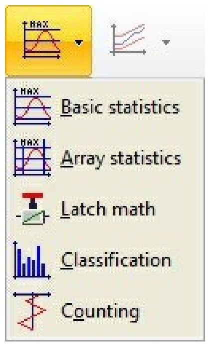
Available **Statistics** on Math *setup* screen are: **Basic Statistics**, **Array Statistics**, **Latch math**, **Classification** and **Counting.** New *Statistics* line can be *added* by selecting:

- *current* **Statistics** button (when appropriate *Statistics* is shown on icon)


or


- from the **Statistics** *list* → for statistics *list* display select *triangle sign* on basic button

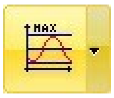
When we select *Statistics* from *list*, icon changes to *selected* one, so we can choose the *same* Statistics again by simply *clicking* on the basic *add Statistics* button.



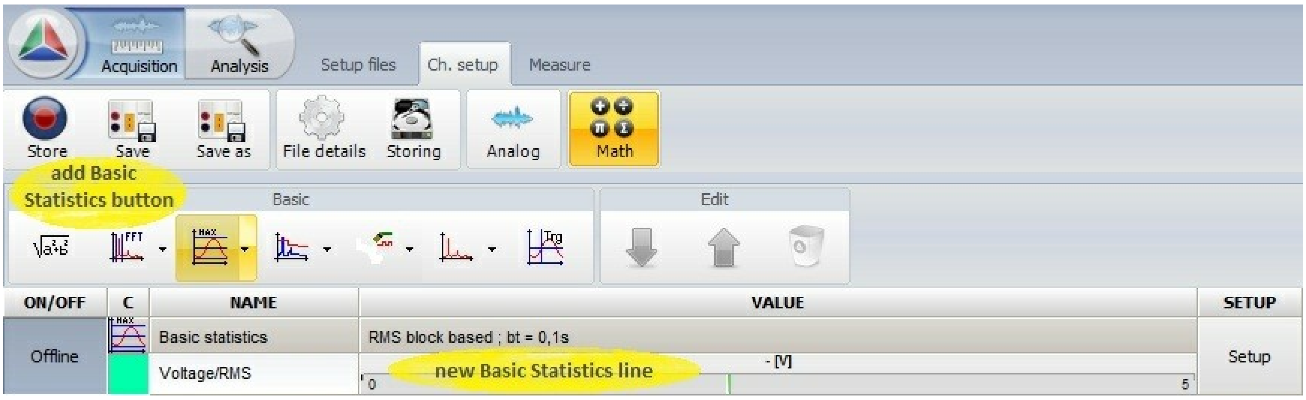
When you press the **Setup** button on activated *new Statistics line*, the Statistics setup window will open. This window depends on selected *Statistics type* and his *name* appears on title line of window:

- **Basic Statistics**
- **Array statistics**
- **Latch math**
- **Classification**
- **Counting**

2.10.4.1 Basic statistics

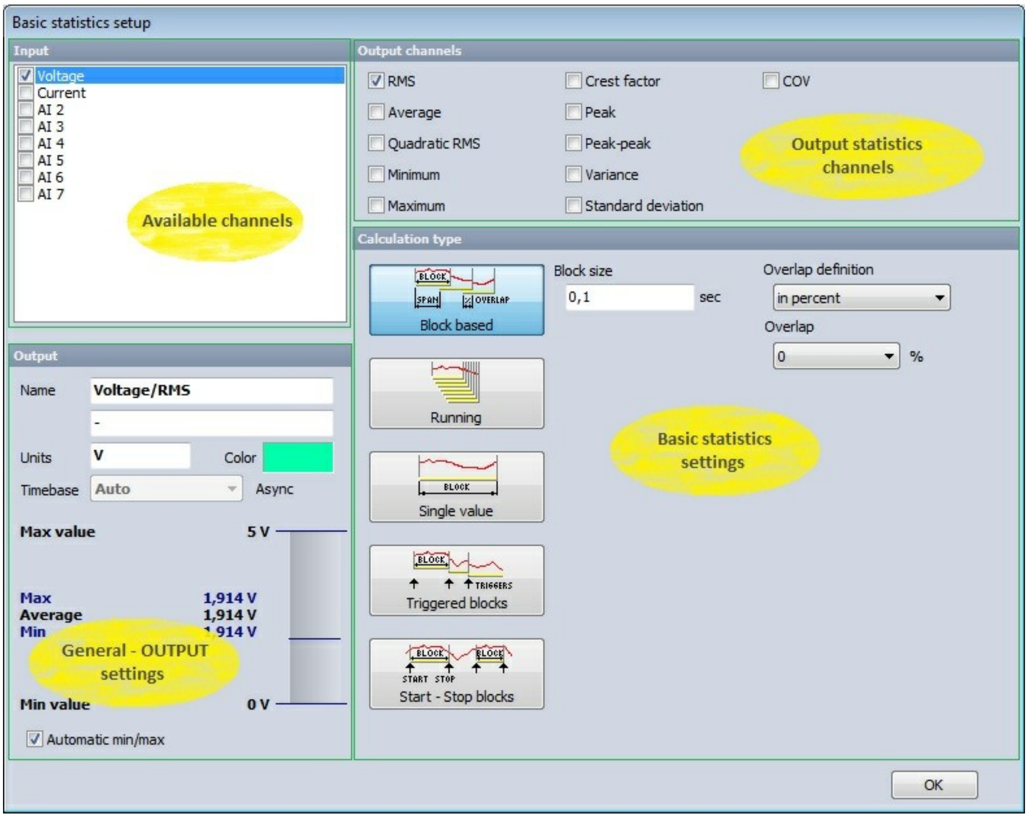


New **Basic statistics** can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Basic Statistics** button from Statistic group.



for detailed information about ***basic settings*** *see* → **Setup screen and basic operation**

When you press the **Setup** button on *new* activated **Basic statistics** *line*, the following Basic statistics setup window will open:



The statistics supports *multiple* input channels.

for detailed information about ***basic settings*** of the *input* and *output channels see* → **Setup screen and basic operation**

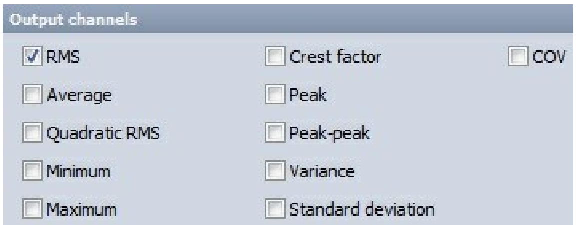
The statistic module provides basic ***statistical quantities*** of the *signal*. There are five basic calculation modes:

- **Block based**
- **Running**
- **total - 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 box beside its name on OUTPUT CHANNEL section:



RMS

Will calculate the root mean square value of the signal.

Average

Average will calculate the average or middle point.

Quadratic RMS

This is similar to the RMS, except all the values are double squared and summed.

Minimum

The minimum will calculate a minimum value of the signal for the specified period.

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.

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.

Peak

Peak is the maximum deviation of the signal from the average value.

Peak-peak

Is the difference between the minimum and maximum.

Variance

Indicating how possible values of a signal are spread around the expected value.

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).

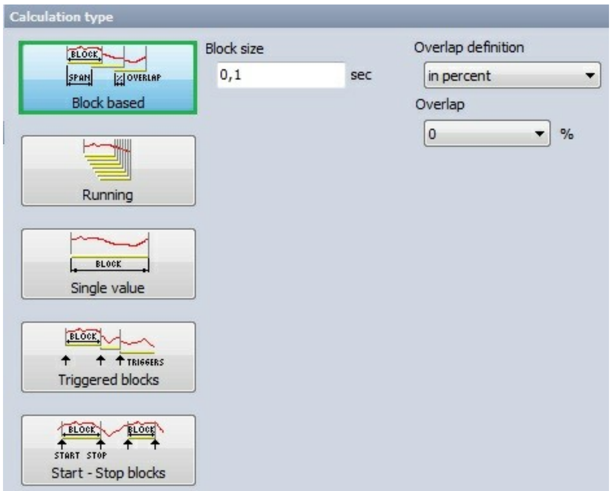
COV

Coefficient of variation is normalized measure of dispersion of probability distribution. It is calculated as ration between standard deviation and the mean.

Calculation type

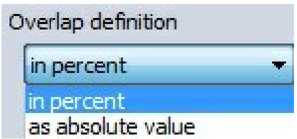
Block based

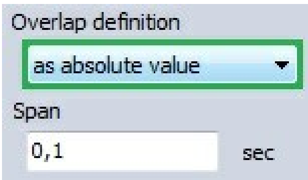
Block based calculation calculates the statistical quantity based on ***specific time interval*** defined by the ***block size***.



- **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 *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.





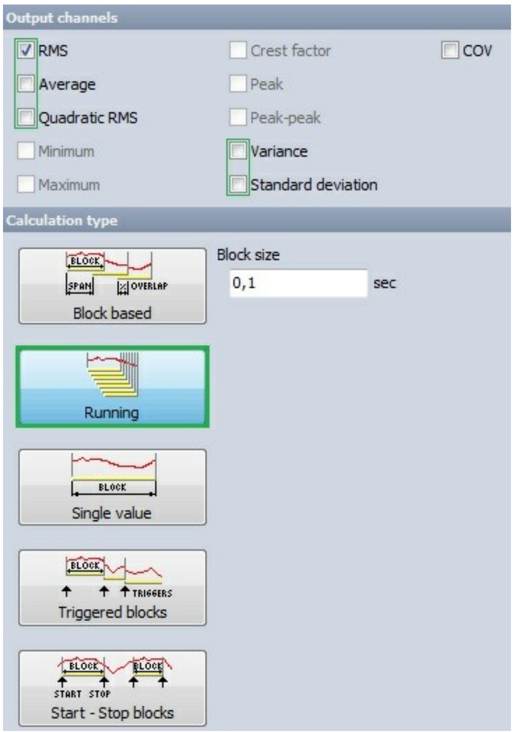
Overlap can be defined:

- in percent → *see* above left and also **FFT Filter setup - Overlap**
- as absolute value - **Span** in sec (*see* right)

Running

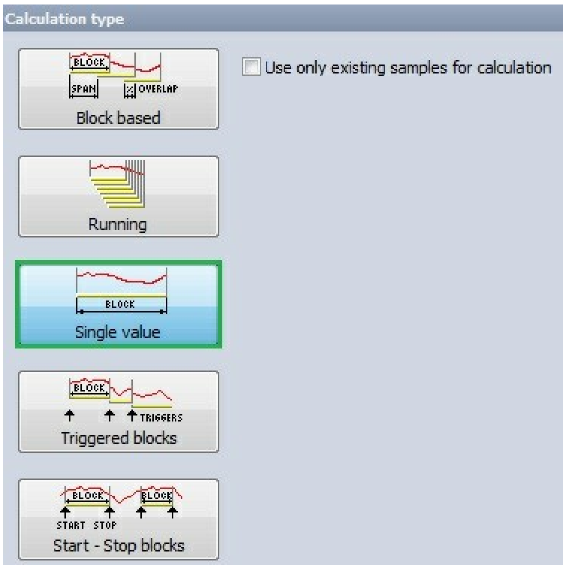
Running calculation is an '*extreme*' version of *overlapping*. The second block is calculated *after one sample* after the first block. Block size has the same meaning as for *block based* calculation.

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).



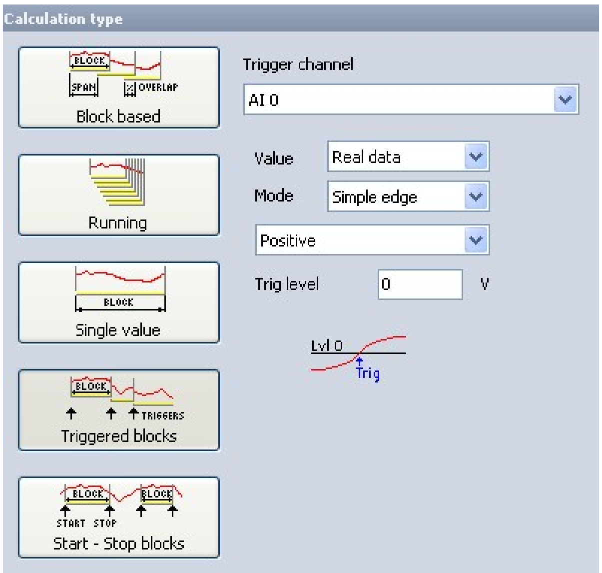
Single value

Single value is the *simplest* calculation and has *not* setting. It will produce *only one value* at the *end* of the measurement. Result will be *updated* also *during* the measurement, but *only* the ***final value*** will be *stored* to the data file.



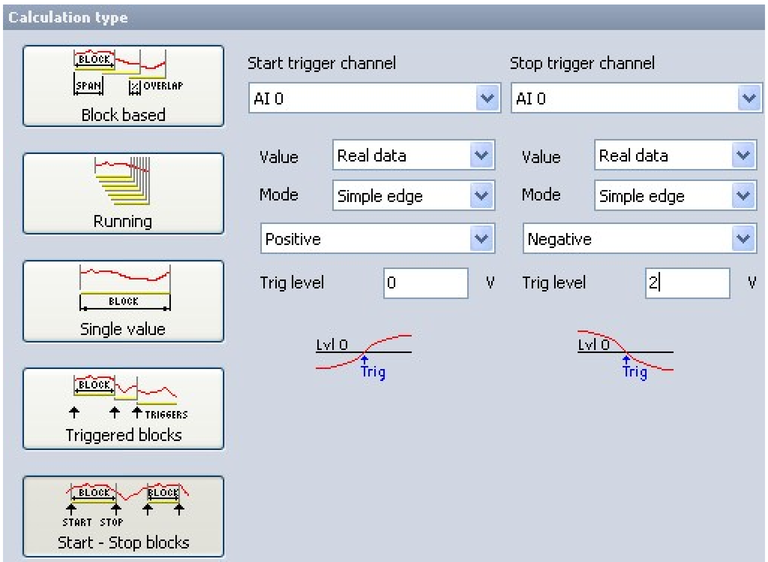
Triggered blocks

Triggered blocks option calculates the statistical value *based on* specific **trigger event**. When event is recognized, it start to calculate. When a second event is recognized, it stops the first calculation, writes the statistical value with its time stamp 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.

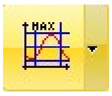


Start/stop blocks

Start/stop blocks option calculates the statistical value *starting on* specific **trigger event**. When 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 new start event is recognized. The start and stop channel can be any channel, also different one and the trigger condition have same options as the alarm or storage triggers.



2.10.4.2 Array statistics



New **Array statistics** module can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Array statistic math** button from *Statistic* group.

The array statistics can calculate the *statistical value* from the *array*.

There are several options which can be chosen:

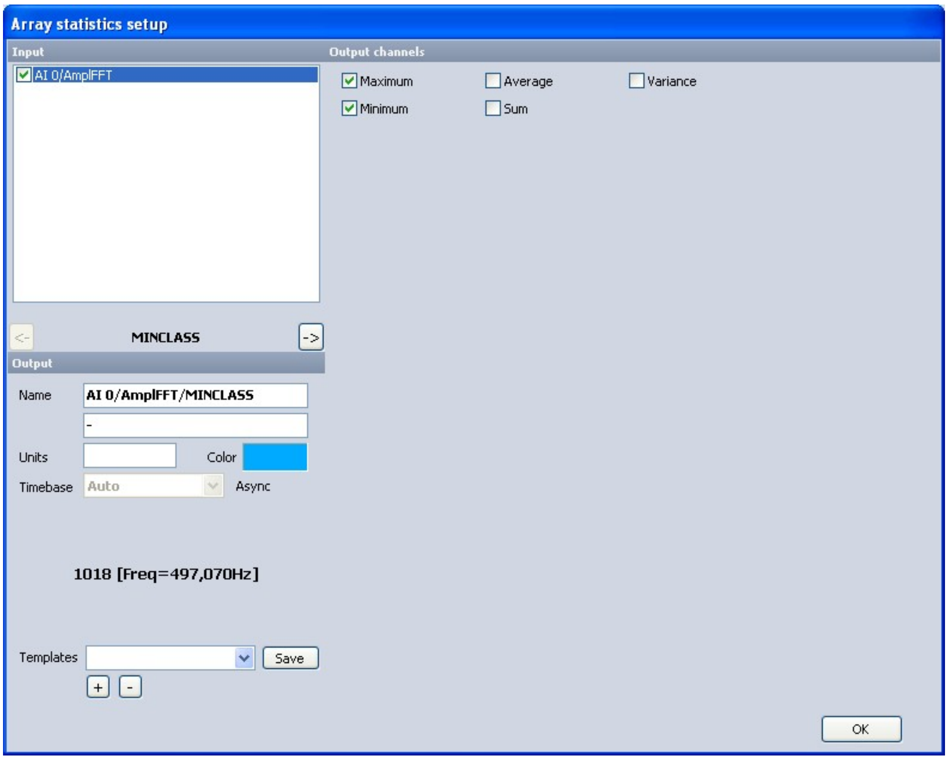
Minimum ... finds *minimum* value from the array. There are two output channels created: class and and value. Class will describe which index of the array holds the parameter and the value will be the minimum value itself.

Maximum ... finds *maximum* value from the array. There are two output channels created: class and and value. Class will describe which index of the array holds the parameter and the value will be the maximum value itself.

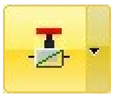
Average ... calculates *average* value of all elements from the array.

Sum ... calculates *sum* of all elements from the array.

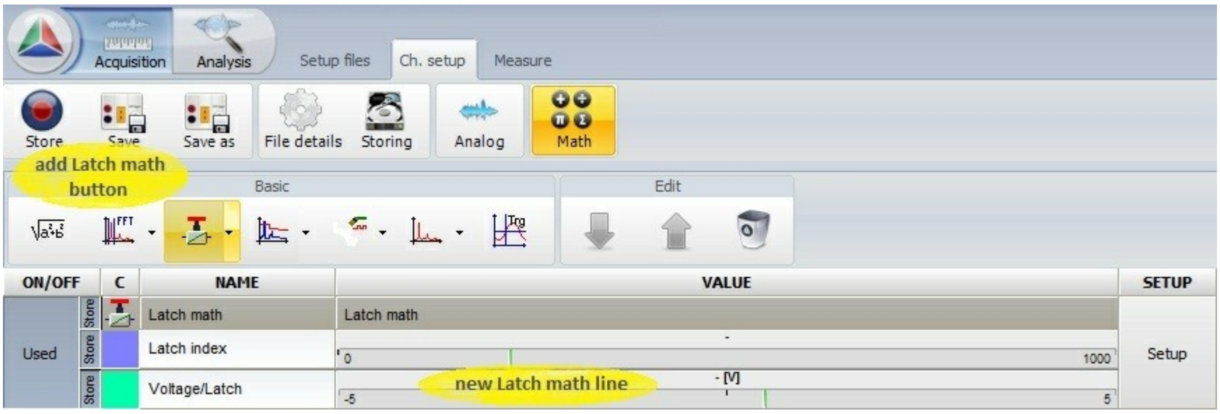
Variance ... calculates the *variance* of all elements from the array.



2.10.4.3 Latch math

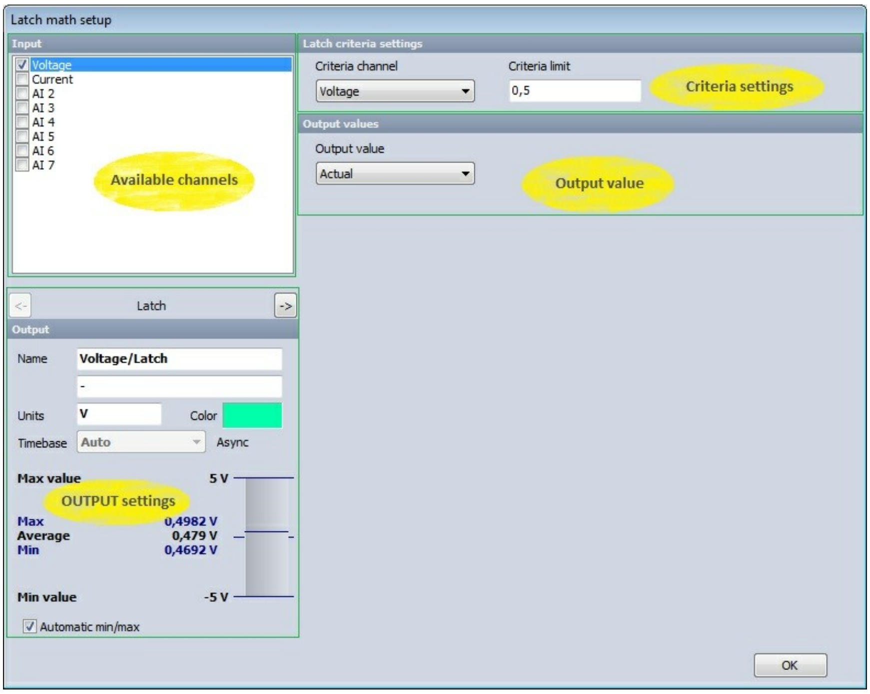


New **Latch math** module can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Latch math** button from *Statistic* group.



for detailed information about ***basic settings*** *see* → **Setup screen and basic operation**

When you press the **Setup** button on *new* activated **Latch math** *line*, the following *Latch math setup* window will open:



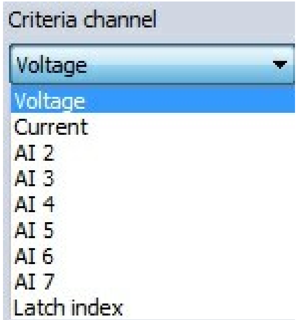
for detailed information about ***basic settings*** of the *input* and *output channels see* → **Setup screen and basic operation**

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

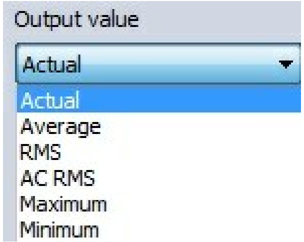
Select Criteria channel from drop down list of available channels.



After that enter Criteria limit ***value*** in corresponding field.

Output value

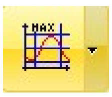
Select **Output value** from drop down list of available ***Actual*** and ***statistical*** *values*.



For all ***statistical*** *values* additional field appear to enter Time base in seconds:

Lower left part of Latch math setup window shows the *selected output* / **Latch** *channel settings* (*see* picture above) and predefined **Index** channel. We have also two more buttons  and  to *choose between* this output channels.

2.10.4.4 Classification



New **Classification** module can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Classification math** button from Statistic group.

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.

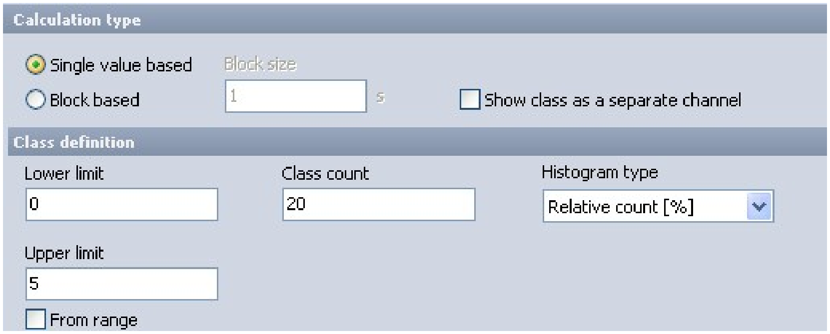
Classification in measurement field is used for various applications, for example to find the distribution of power grid frequencies with time or to find the distribution of sound levels to which certain area or working place is exposed to.

First of all we need to define what will be the result of classification. There are two 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 array* each one added at the end of 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.

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 multi meter.



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 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 from 0 to 0.125. 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 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 total number of samples and divided by class width. In this case the value is not depending on number of classes within a range

density [%] ... same as density, but multiplied with 100

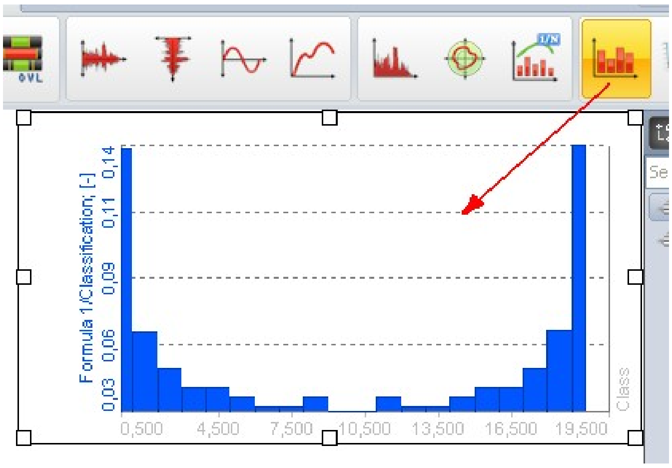
distribution ... provides empirical probability distribution, each class value has the sum of all lower classes and the number of current samples, normalized to total number of samples. The highest class has the value of 1.

distribution [%] ... same as distribution, but expressed in percent. The highest class has the value of 100.

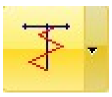
There are also several special output channels available. Two of them are **Skewness** (asymmetry of probability distribution) and **Kurtosis** (measure of "peakedness" of distribution). Additionally we can output a list of Distribution point values. Distribution points are the class values at which distribution reaches entered value.

For the moment the distribution points works only if 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 use also 3D graph to display the history of classifications.



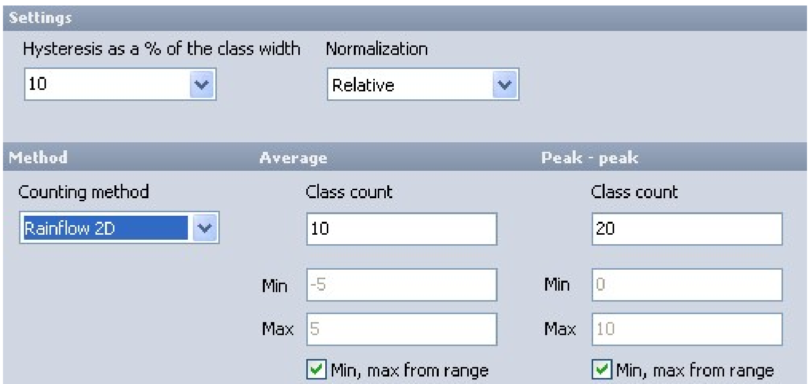
2.10.4.5 Counting



New **Counting** module can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Counting math** button from Statistic group.

Counting is the standard procedure to ***reduce amount of data*** for analysis. It is for example used in 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 rainflow analysis is made.



The counting procedures counts the *peaks* and *valley* of the *signal*. The Hysteresis is defined in *percentage* of class width. This prevents too much false counts if the signal is noisy.

There are three possible output values (Normalization):

Absolute ... it outputs *number of cycles* as a *value* - values will increase with time

Relative ... it outputs the *number of cycles normalized to absolute number of cycles* - sum of all values will be always 1

Relative [%] ... it outputs the number of cycles normalized to absolute number of cycles multiplied with 100 - 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 slope between peaks and valleys is positive. We can choose either to count positive, negative directions or both).

Level crossing ... counts number of times when that signal crosses various levels.

Rainflow 1D ... counts range pairs. Rather than counting the peaks it splits the signal variations to smaller and larger pairs of values and counts all of them independently.

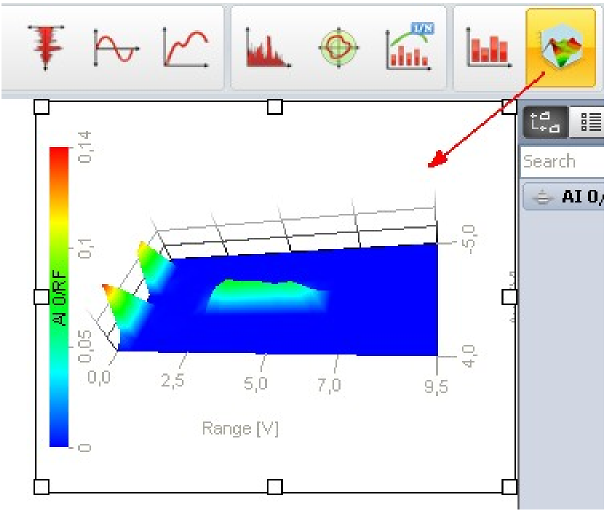
Rainflow 2D ... same procedure as above, but it also calculates the average of each cycle and creates matrix of average vs. peak-peak value vs. number of cycles.

First four options outputs *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 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*.



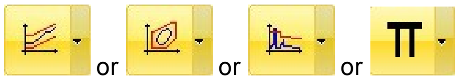
2.10.5 Reference curves

New **Reference curve** can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Reference curve** button.

Add new Reference curve

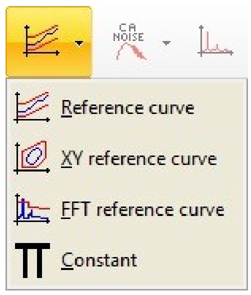
Available **Reference curve** on Math *setup* screen are: **Reference curve**, **XY reference curve** and **FFT reference curve**. New *Statistics* line can be *added* by selecting:

- current **Reference curve** button (when appropriate *Reference curve* is shown on icon)



- from the **Reference curve** *list* → for reference curve *list* display select triangle sign on main button

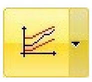
When we select *Reference curve* from *list*, icon changes to *selected* one, so we can choose the *same* Reference curve again by simply *clicking* on the basic *add Reference curve* button.



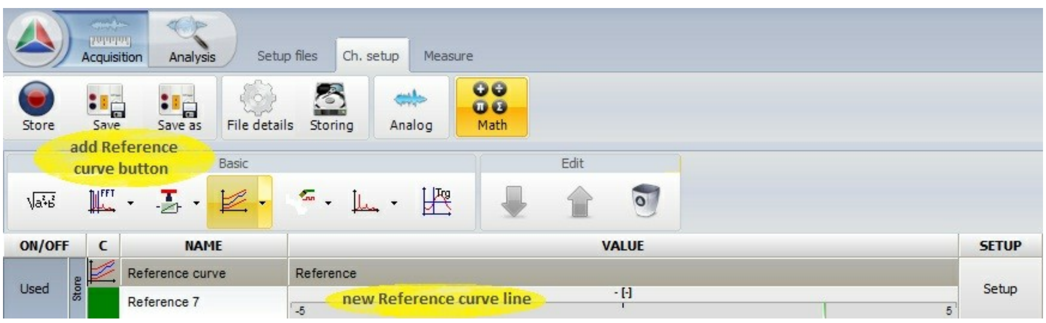
When you press the **Setup** button on activated *new Reference curve line*, the **Reference curve setup** window will open. This window depends on selected *Reference curve type* and his *name* appears on title line of window:

- **Reference curve**
- **XY reference curve**
- **FFT reference curve**
- **Constant**

2.10.5.1 Reference curve

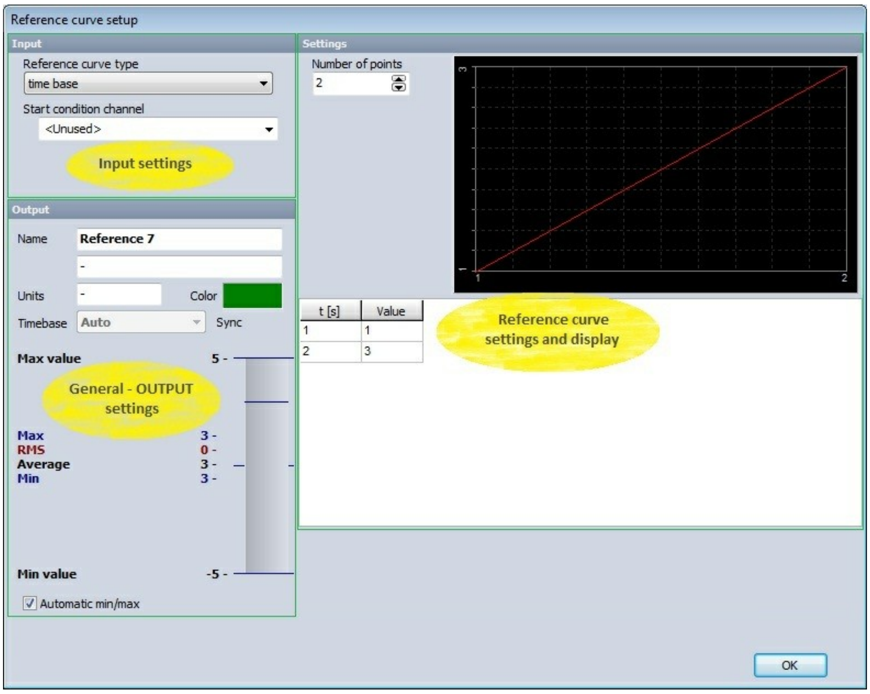


New **Reference curve** can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Reference curve** button



for detailed information about ***basic settings*** *see* → **Setup screen and basic operation**

When you press the **Setup** button on *new* activated **Reference curve** *line*, the following Reference curve setup window will open:

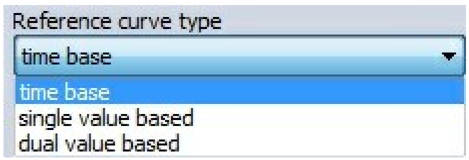


for detailed information about ***basic settings*** of the *output channels* *see* → **Setup screen and basic operation**

Input settings

Reference curve type

In this field we can select *Reference curve type* from drop down list:



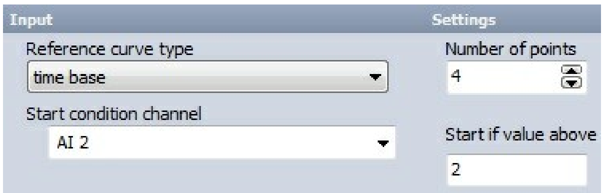
- **time base**
- **single value based** or
- **dual value based**

Time base reference curve

Time base reference curve is useful for *defining* a curve *in time* as a reference during 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 second and decelerate to 0 km/h in 10 second.

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 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.

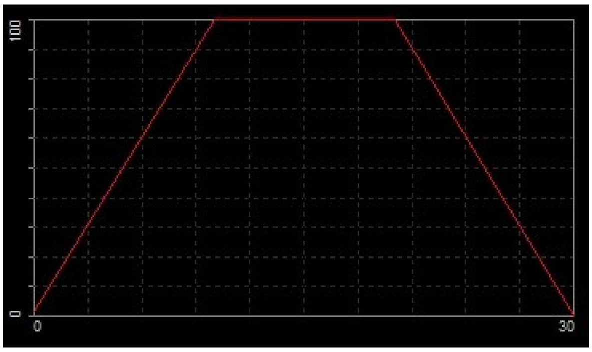


A table will show four empty rows. Now we enter the *time points* and the *values*. In our case it will be:

first point:	0 s	*2 km/h*
second point:	10 s	*100 km/h*
third point:	20 s	*100 km/h*
fourth point:	30 s	*0 km/h*

t [s]	Value
0	2
10	100
20	100
30	0

Display on right upper part Reference curve setup window now shows this curve:



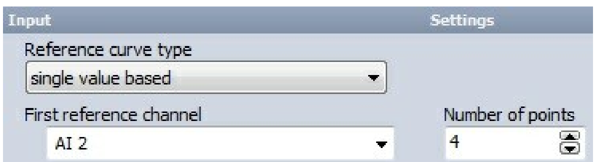
Then we can use the **formula math** channel to see if we are in between or out of the limit. We can for example calculate:

abs('REFCURVE'-VELOCITY')>5

This will give a number of 1 if 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

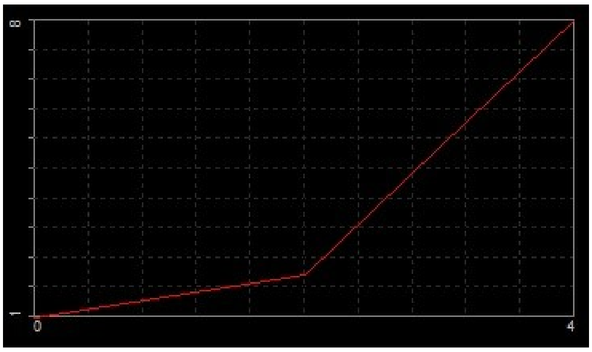
A single value based reference curve can be understood as a sort of *non-linear scaling*. We need to define a **First reference channel**, **Number of points** and a *table* with the *values of reference* and output channel.



So in this case on right, if the input channel will be 3, 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 output. If we have 5 at the input, we will have 11 at 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:



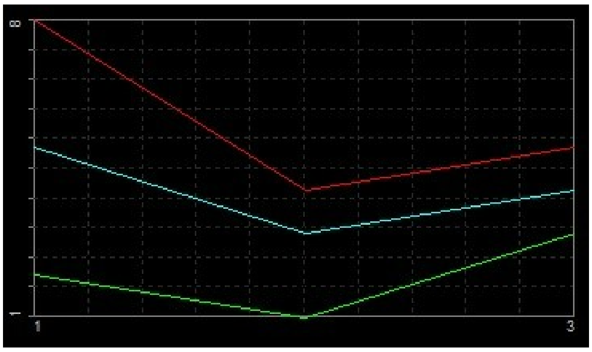
Dual value based reference curve

Dual value based reference curve have *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.

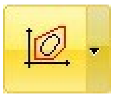
Ch. 1 \ Ch. 2	1	2	3
2	2	1	3
3	5	3	4
4	8	4	5

Display on right upper part *Reference curve setup* window now shows this curve:

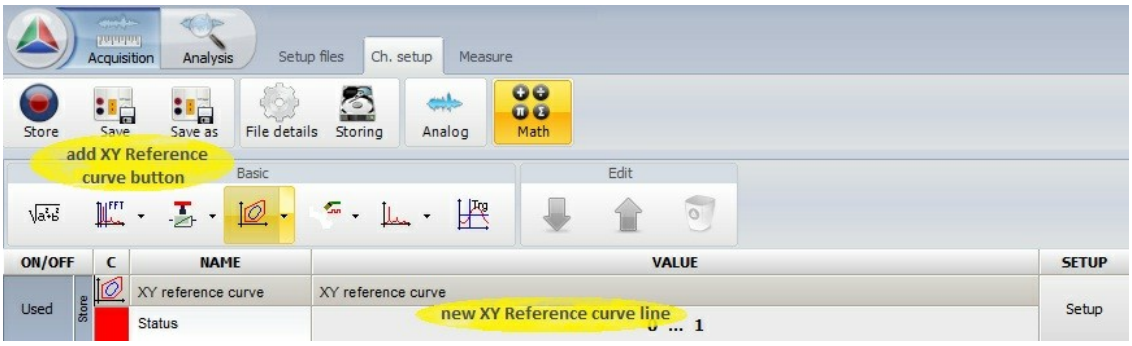


A good example of using *dual* reference curve is to define the limit oil pressure referenced to *RPM* and the *oil temperature*.

2.10.5.2 XY reference curve



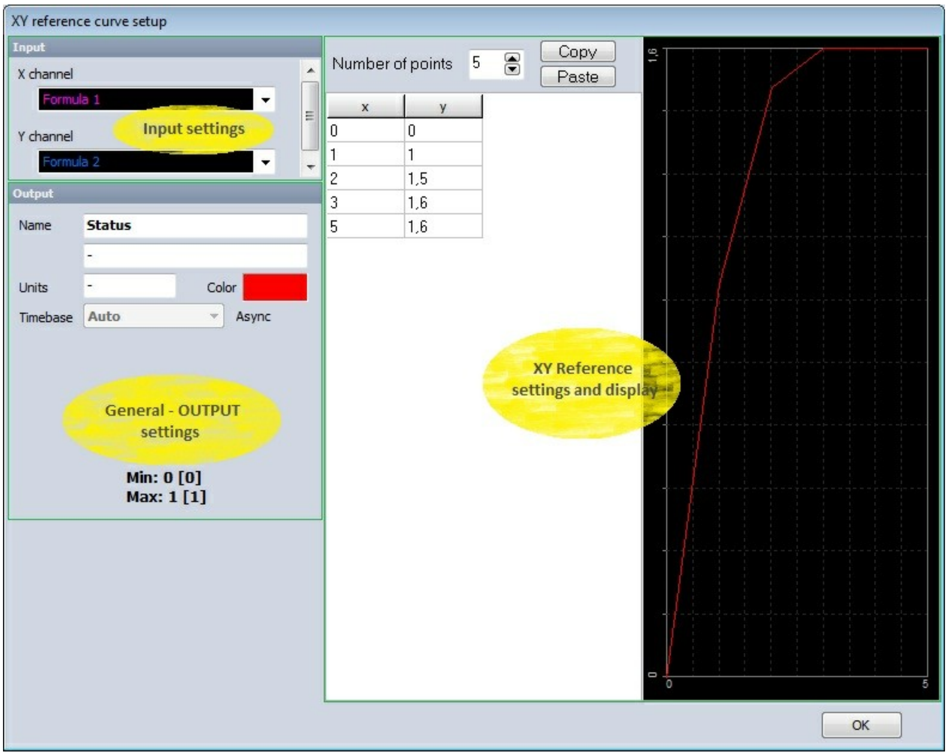
New **XY reference curve** module can be *added* on the DEWESoft Math *setup* screen by selecting *add* **XY reference curve** button.



for detailed information about ***basic settings*** *see* → **Setup screen and basic operation**

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 *output channels* *see* → **Setup screen and basic operation**

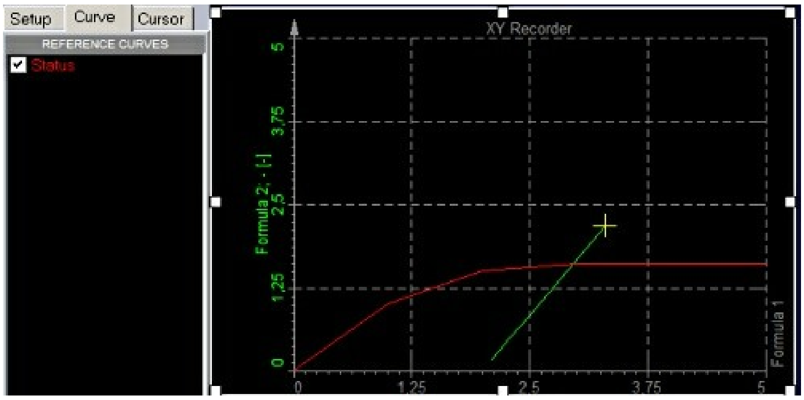
After choosing INPUT **X channel** and **Y channel**, we need to *set* **Number of points** for **XY reference curve**. The reference points can be entered value by value. We have also another option for easier transfer of points - *copy* and *paste* function. **Copy** button copies all the points to the clipboard in *tab delimited* style, so it can be easily pasted to any spreadsheet software. The values can be also copied from other 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 be pasted in DEWESoft.

x	y
0	0
1	1
2	1,5
3	1,6
5	1,6

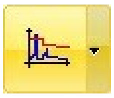
Display on right upper part *XY reference curve setup* window shows defined curve. Lower left part of *XY reference curve setup* window shows the *selected* OUTPUT *channel settings*.

The *XY display* can show the reference curve as seen of the picture below.

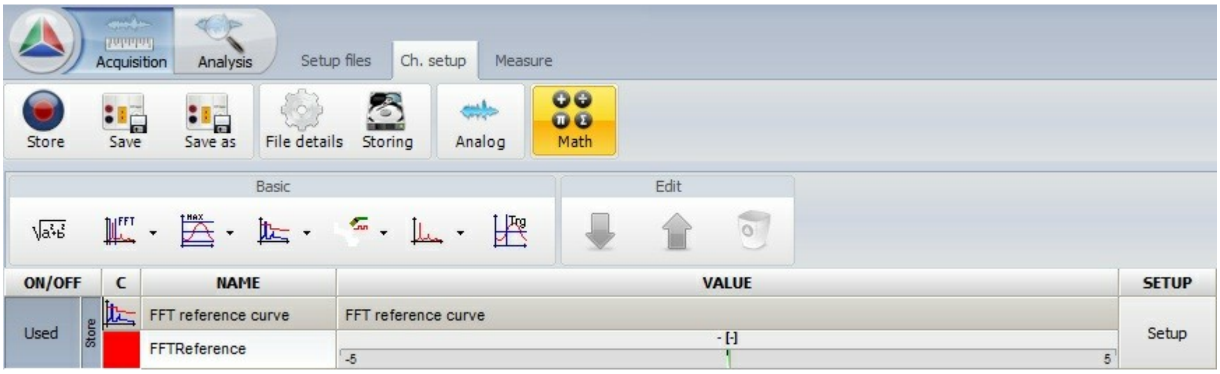


There is another advantage of the reference curve. The *output channel* (Status in our case) gives the value of 1 when the xy curve crosses the **reference curve**. This can be used for *trigger* criteria or for counting of events (with ECNT function in *formula editor*).

2.10.5.3 FFT reference curve

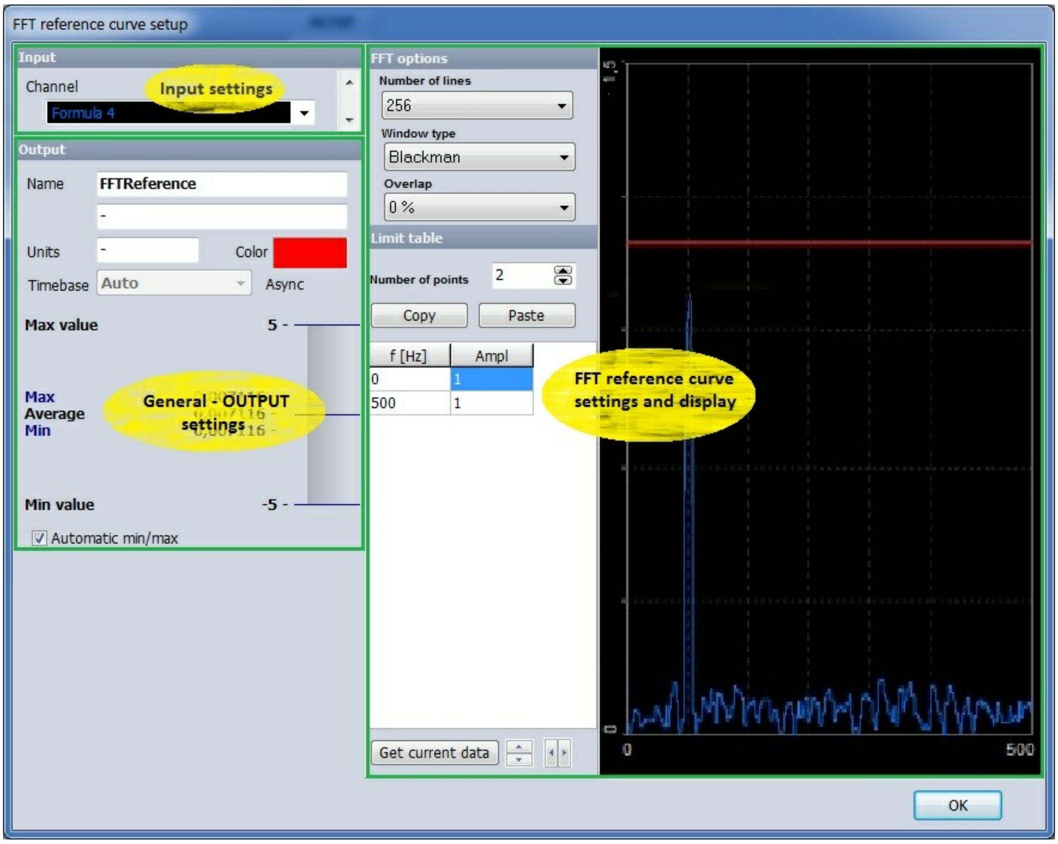


New FFT **reference curve** module can be *added* on the DEWESoft Math *setup* screen by selecting *add* **FFT reference curve** button.



for detailed information about ***basic settings*** *see* → **Setup screen and basic operation**

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*.

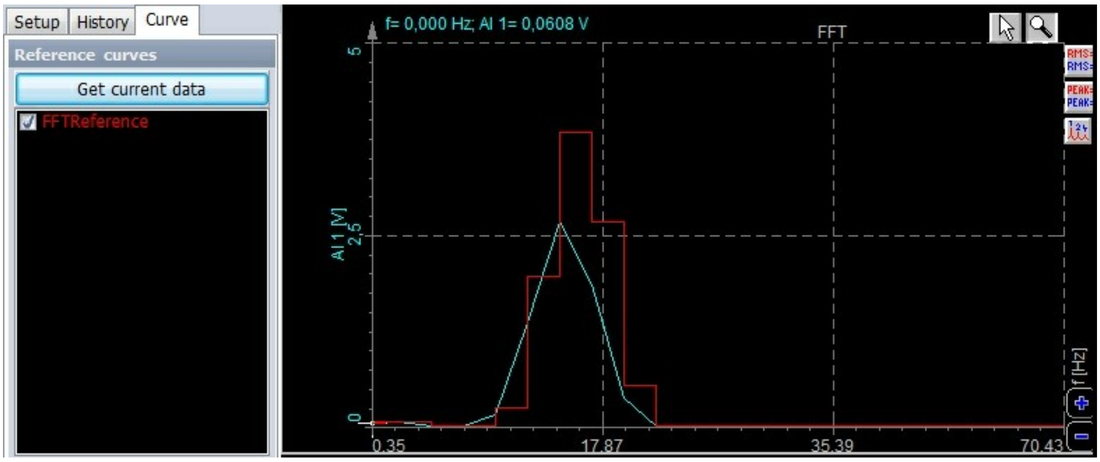


for detailed information about ***basic settings*** of the *input* and *output channels* *see* → **Setup screen and basic operation**

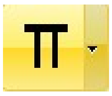
We define in the *channel setup* the maximum Number of lines, Window type and Overlap. These values will be used for calculation of FFT spectrum. Then we define the limits - Limit table. They can be *entered* value by value or from *current data* - Get current data button. This means that current FFT (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.



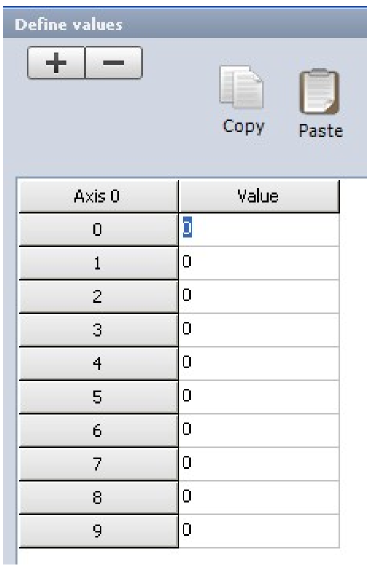
2.10.5.4 Constant



New **Constant** module can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Constant** button.

The **constant** can define a *set of values* which will be as a result an array channel. We can add points by pressing ⊞, remove points by pressing ⊟ button 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*.



2.10.6 Dynamic signal analysis section

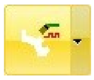
New **CA NOISE** and/or **Exact frequency** and/or **Angle sensor** *channel* can be *added* on the DEWESoft Math *setup* screen by selecting *basic add* **CA NOISE** / **FREQ** / **Angle sensor** button or from the button *list*.

Add new CA noise / Exact frequency / Angle sensor channel

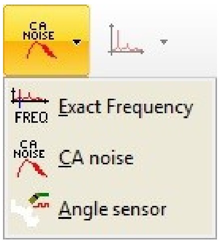
Available DSA *function* on Math *setup* screen are: **CA NOISE**, **Exact frequency** and **Angle sensor**.
New DSA *function* can be *added*:

- by selecting current **DSA function** button (when appropriate option is shown on icon)

or


or


- from the **DSA function** *list* → for DSA *function list* display select *triangle sign* on basic button



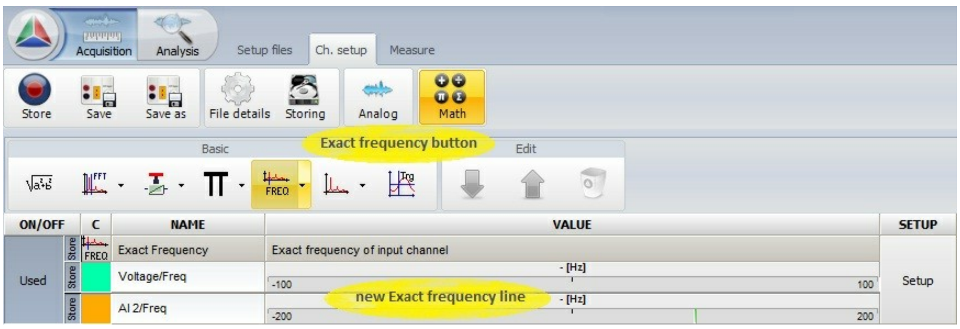
When we select **CA NOISE** or **Exact frequency** or **Angle sensor** from *list*, icon of basic add *DSA* function button changes to *selected* function, so we can add *same DSA* function again by simply *clicking* on the basic *add* **DSA function** button.

When you press the **Setup** button on activated *new CA noise / Exact frequency line*, the *setup* window will open. This window depends on selected *type* of function and his name appears on title line of *setup* window:

- **Exact frequency**
- **CA noise setup**
- **Angle sensor**

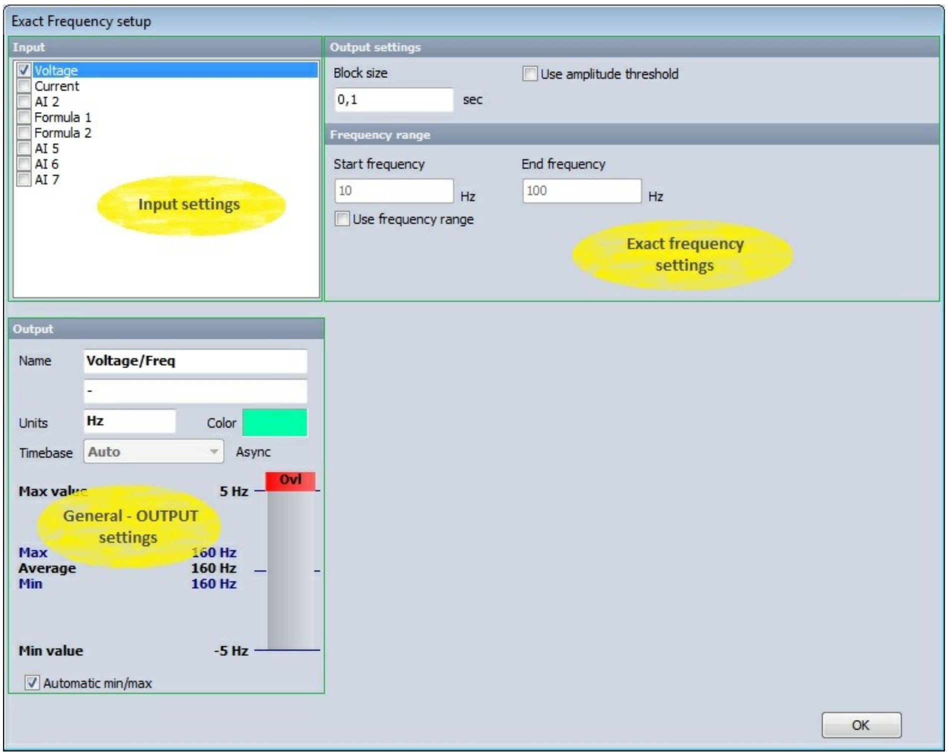
2.10.6.1 Exact frequency

New **Exact frequency** function can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Exact frequency** button.



for detailed information about ***basic settings*** *see* → **Setup screen and basic operation**

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 exact* frequency of the signals. Even with *lower* sampling rates which doesn't allow methods like period measurement and even with signals with not that clear level crossing this method will work very nice.

The method is based on finding *best fit* to the theoretical *sine wave*. It can measure millihertz accurate with sampling rates only 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.

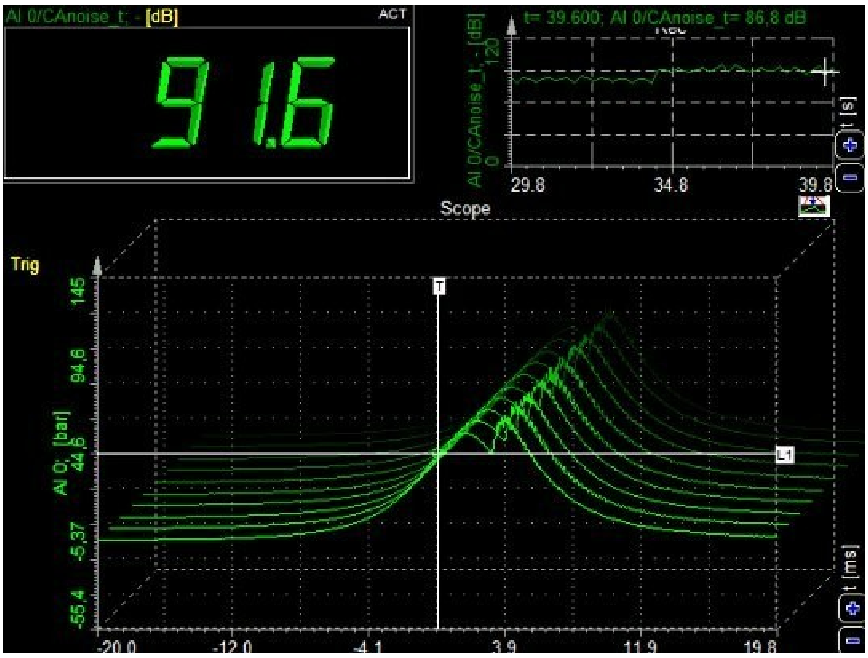
In the right area of the window are - *Output settings* - *Calculation parameters*- with:

- **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.*

2.10.6.2 CA noise

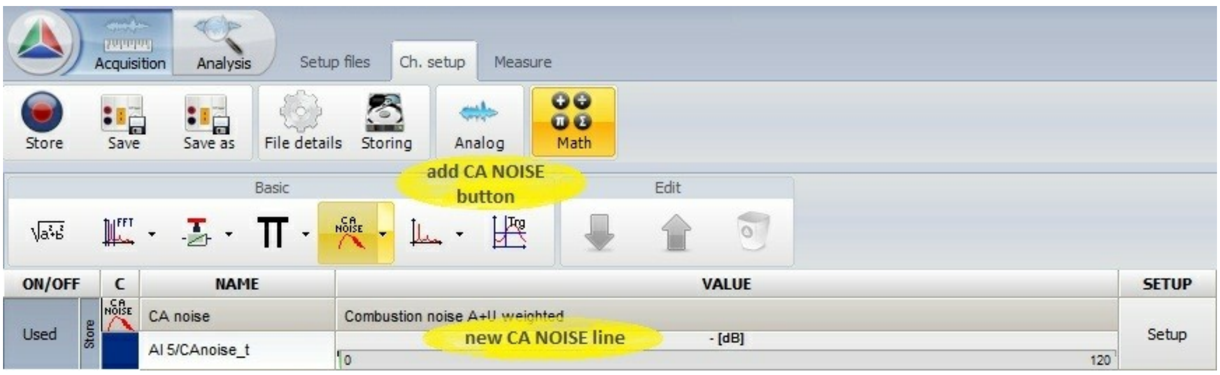
CA noise description



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 approximate *value* of what *kind of noise* this engine will produce.

Setup

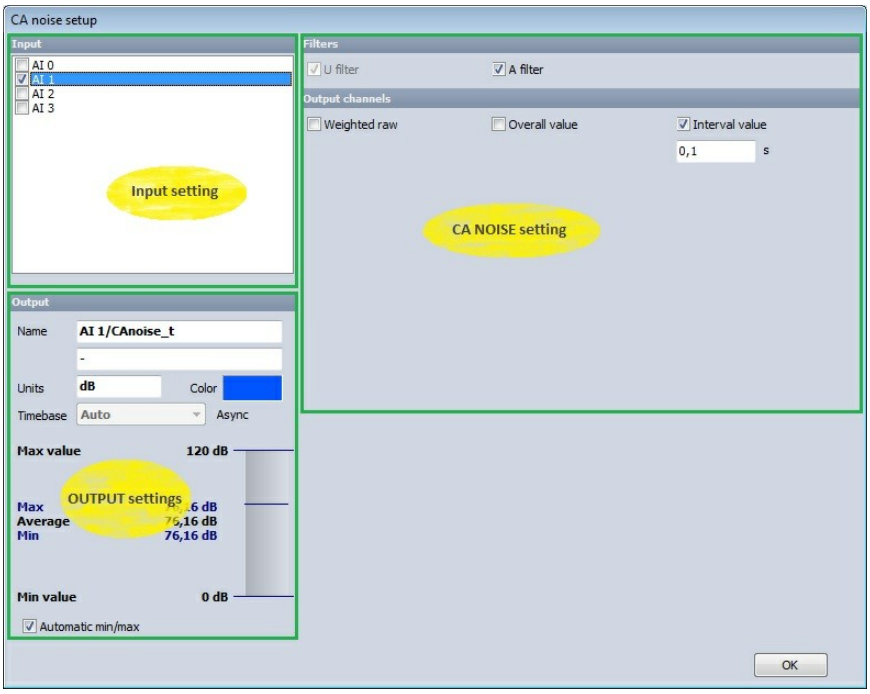
New **CA NOISE** function can be *added* on the DEWESoft Math *setup* screen by selecting *add* **CA NOISE** button.



for detailed information about ***basic settings*** *see* → **Setup screen and basic operation**

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 afterwards, otherwise error message appear on *CA NOISE line* in *Value* column:

There are three types of output channels available:

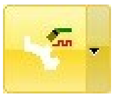
Weighted raw ... the output is the *time curve* with applied U (and optionally A) weighting; this channel might be used in the **Sound Level** mathematics for advanced sound analysis

Overall value ... this channel will have *one* value (single value) at the *end* of the measurement

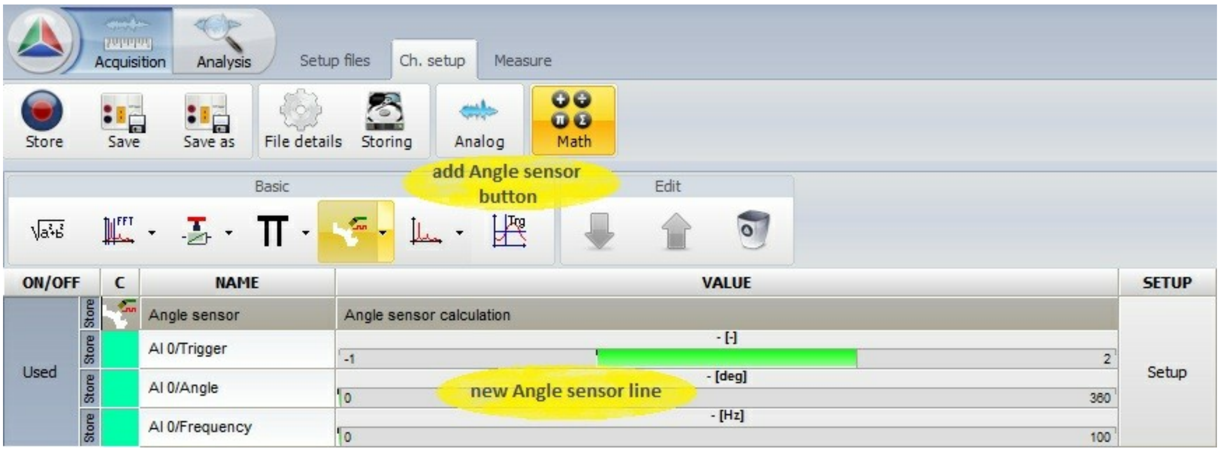
Interval value ... this channel will have a value for *each* interval, which we can *freely* define in seconds.

Please note that custom CA noise or CA noise on *external clock* can be also calculated *using* FFT filter.

2.10.6.3 Angle sensor

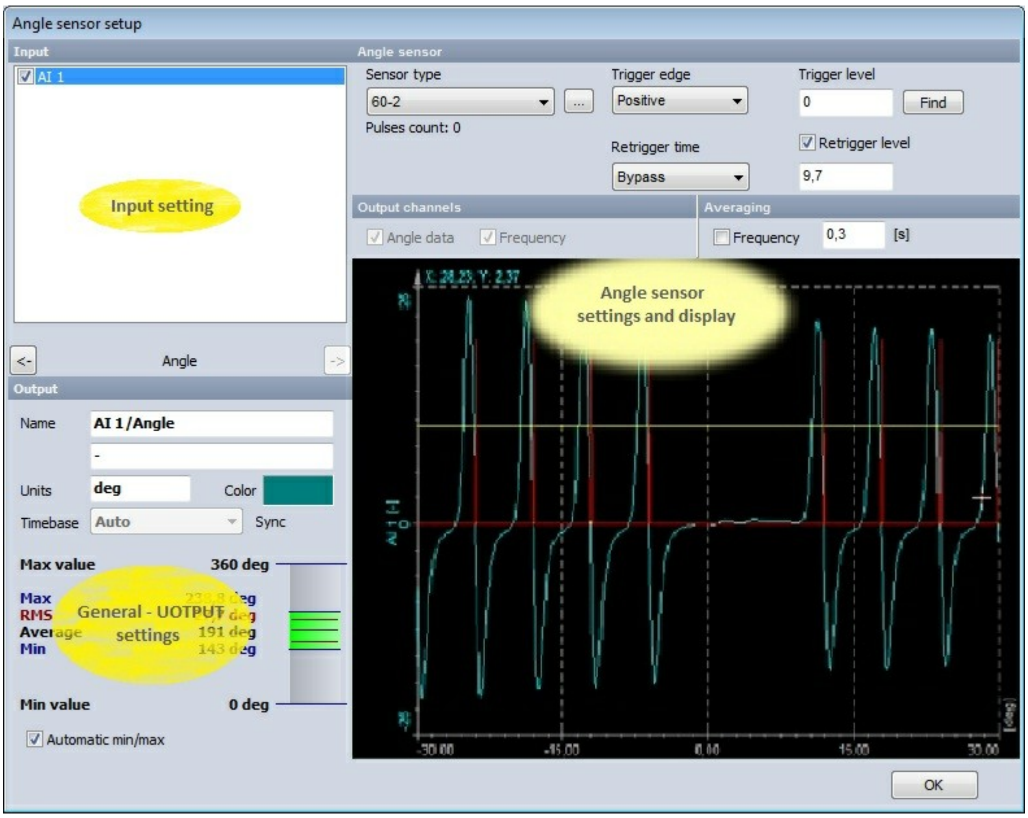


New **Angle sensor** function can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Angle sensor** button.



for detailed information about ***basic settings*** *see* → **Setup screen and basic operation**

When you press the **Setup** button on *new* activated **Angle sensor** *line*, the following *Angle sensor setup* window will open:

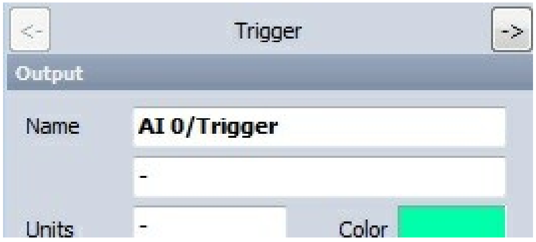


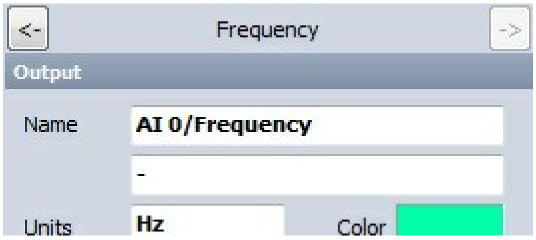
for detailed information about ***basic settings*** of the *input* and *output channels see* → **Setup screen and basic operation**

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 mathematic to calculate the *current* *angle* and the *frequency* of incoming signal. In both cases the signal needs to be *connected* to *analog input* of the instrument. Then we need to *select* this signal, select the Sensor type from the list of 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 Retrigger level input field. Next we can define the Retrigger 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:

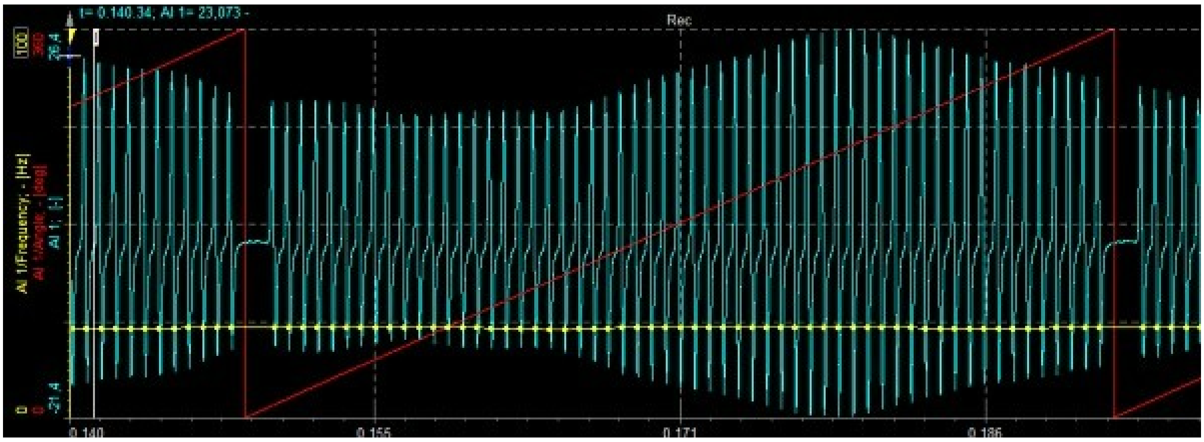
- *angle*: Angle channel as we can see from the picture above and Trigger channel
- *frequency*: Frequency channel





These *channels* can be *selected* by checking appropriated box in 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*.



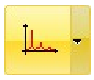
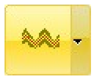
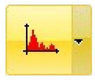
The nice feature of this module is that it *implements* calculation **delay**. For sensor like 60-2 it is not possible to determine the angle until the first gap is recognized. For *that time* the **angle sensor math** *holds all* the calculations and *with* the first gap also calculates the data for 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 *analog* sample rate. If 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 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.

2.10.7 Spectrum analysis section

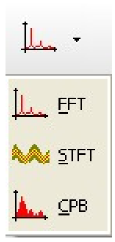
Available **Spectrum analysis** on Math *setup* screen are: **FFT**, **STFT** and **CPB analysis**. New item can be *added* by selecting:

- *current* **Spectrum analysis** button (when appropriate function is shown on icon)

 or  or 

- from the **Spectrum analysis** *list* → select *triangle sign* on the button

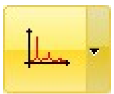
  When we select Spectrum type from *list*, icon changes to *selected* one, so we can choose the *same function* again by simply *clicking* on the main button.



When you press the **Setup** button on activated Spectrum analysis function, the *Spectrum analysis setup* window will open. This window depends on selected *type* and his *name* appears on title line of window:

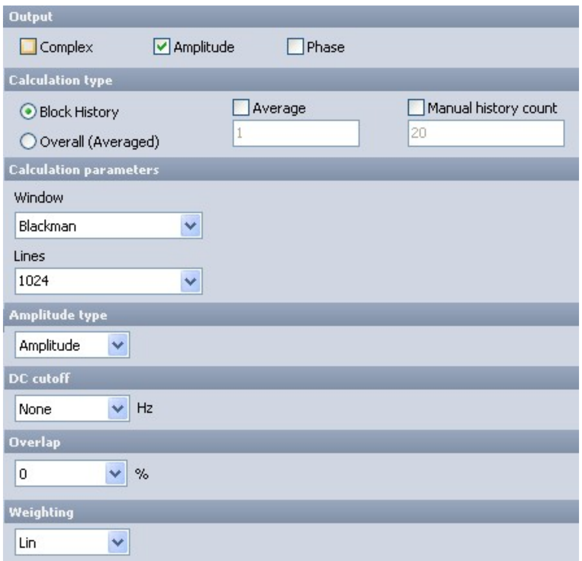
- **FFT analysis**
- **STFT analysis**
- **CPB analysis**

2.10.7.1 FFT analysis



New **FFT analysis** function can be *added* on the DEWESoft Math *setup* screen by selecting *add* **FFT analysis** button.

When you press the **Setup** button on FFT analysis math item, the following *setup* window will open:



Output of the FFT analysis could be **Complex** (real, imaginary), **Amplitude** or **Phase** or any combination of those.

Calculation type can be **Overall (Averaged)**, where the result is one spectrum for entire record. Second option is **Block history**, where the FFTs are acquired shot by shot and put into the buffer. In this case they can be observed on 3D graph. Number of **Averages** can be also set. When we enter a value of n, then it will calculate average of n spectrum and put 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 **Window** type, which describes the FFT window. There is a good description of usage of different window functions in the tutorial. By default we use Blackman, because it is a good compromise between amplitude error and width of side bands. Number of **Lines** defines the size of the FFT. More lines we choose, more accurate the frequency will be, but also longer time is needed for calculation. 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 *noise*

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 same weight on the result independant of the window which is used.

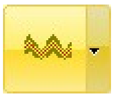
Weighting defines which *sound weighting* should be used on the resulting FFT.

for more information about ***FFT properties***, please look at

→ *see* → Reference Guide → **Theory of single channel frequency analysis**

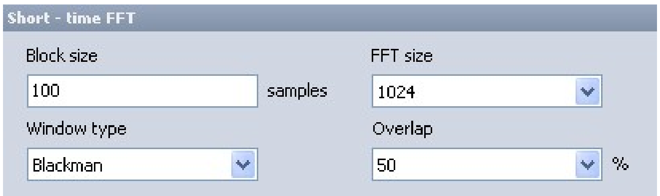
As all *matrix channels* also FFT can be best seen on 2D graph. *Block based FFT* can be also put in the 3D graph for viewing the *history*.

2.10.7.2 STFT analysis



New **STFT analysis** function can be *added* on the DEWESoft Math *setup* screen by selecting *add* **STFT analysis** button.

Short time Fourier transformation is the procedure which *calculates more lines* than the normal FFT. This is achieved by having 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 FFTs for the same time base. It is nice to be used 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 usage of different window functions in the tutorial. By default we use Blackman, because it is a good compromise between amplitude error and width of side bands.

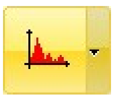
Overlap defines how much will two FFT shots *overlap* between each other. 50% is enough that all the samples will have same weight on the result independent of the window which is used.

for more information about ***FFT properties***, please look at

→ *see* → Reference Guide → **Theory of single channel frequency analysis**

STFT as can be viewed in 2D and 3D graph.

2.10.7.3 CPB analysis

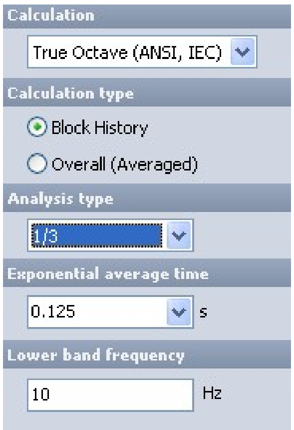


New **CPB analysis** function can be *added* on the DEWESoft Math *setup* screen by selecting *add* **CPB analysis** button.

As *opposed* to *FFT analysis*, which has specific number of lines per *linear* frequency (x axis), **CPB** (**constant percentage bandwidth**, called also *octave*) has specific number of lines if *logarithmic* frequency x axis is used. Therefore *lower* frequencies has *more* number of 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 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 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 entire record. Second option is **Block history**, where the PCBs are acquired *shot by shot* and put into the buffer. In this case they can be observed on 3D graph. Number of averages can be also set. When we enter a value of n, then it will calculate average of *n spectrum* and put 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.

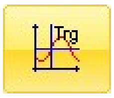
Analysis type defines *number of bands* within *one* octave. One octave means that the next center frequency of the band is *twice* the value of the current one. If we have 100 Hz, next octave band will be at 200 Hz. Then these octaves are further divided by a number defined in the analysis type field. 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 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 corresponds 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.

Second field is the *Lower band frequency*, which defines what will be the *lowest value* for calculation.

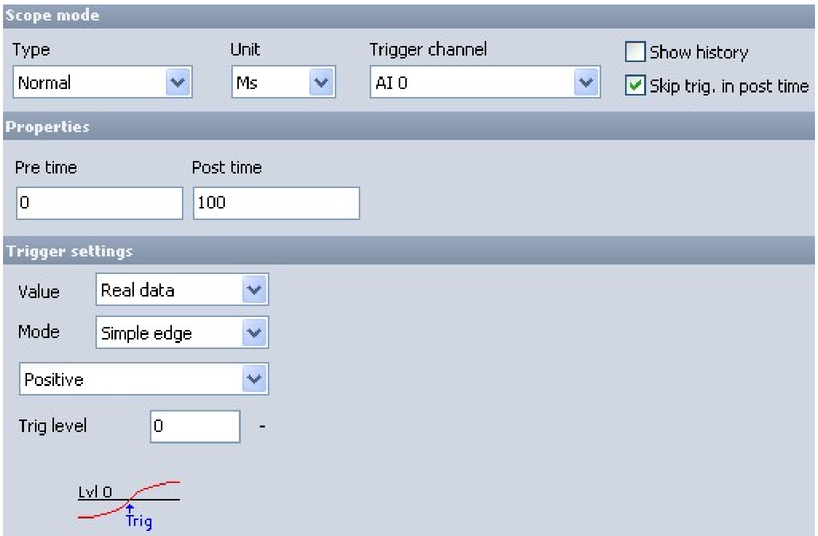
As all matrix channels also *CPB* can be best seen on 2D graph. *Block based CPB* can be also put in the 3D graph for viewing the *history*.

2.10.8 Scope trigger



New **Scope trigger** function can be *added* on the DEWESoft Math *setup* screen by selecting *add* **Scope trigger** button.

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 a 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 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 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*.

Then we define **Pre time** and **Post time** in the chosen units.

Trigger settings are the same as in normal scope, alarms or storing triggers.

2.11 Power module

DEWESoft Power is one of the most complex **mathematic module** 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 (1/2 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.

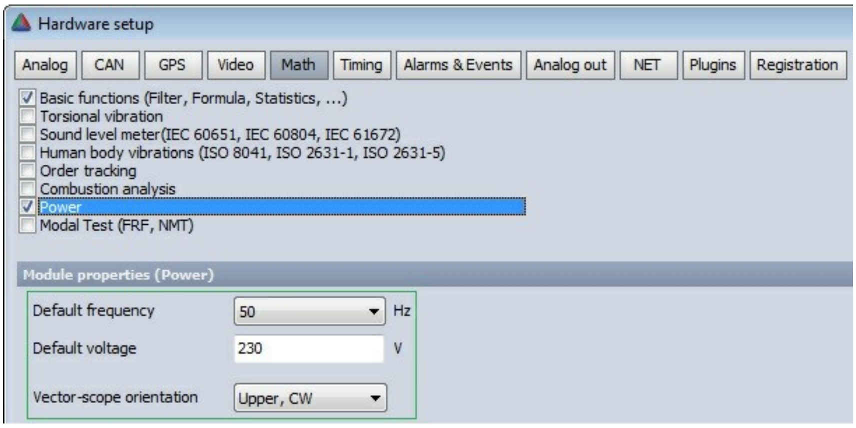
To measure **Power**, we need at least two *input channels* connected to your instrument for measuring voltage and current.

Required hardware	Dewesoft Sirius
Required software	SE or higher + POWER option or EE
Setup sample rate	At least 5 kHz

for hints about ***Power module*** application see → DEWESoft Tutorials

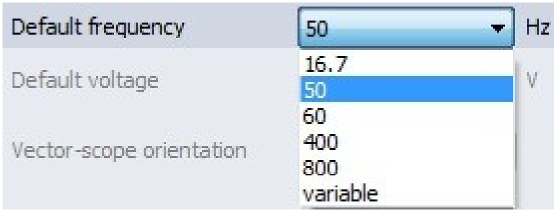
for detailed information how to ***initialize Power module*** *see* → System Settings → **Math & Applications**

After *required hardware installation* and *setup* according to manufacturer instructions (this procedure is to perform *only* by installation new or changing hardware), DEWESoft **Power module** measurement is allowed by *selecting* the Power module checkbox in *System* → *Hardware setup* → **Math** tab.



On this tab we can for the Power module also:

- select **Default frequency** (in Hz) from drop down list
- enter **Default voltage** (default= 230 V)



After selecting this options, a tab labeled Power appears in the DEWESoft *Setup* screen (*see* picture in *Add Power module* chapter below) to *define* and *setup* *Power module*(s).

Setup input channels

First we will take the **setup** for *analog input channels*. So the *first* channel is the voltage while the *second* channel is the current.

To use **Power** module, please *select* and *set up* **voltage** and **current** *analog channel(s)* in Analog tab

see → User Guide → **Analog in Channel Setup**

NOTE:	Note that 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 enter 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 measurable range - *up to one* kilovolt.

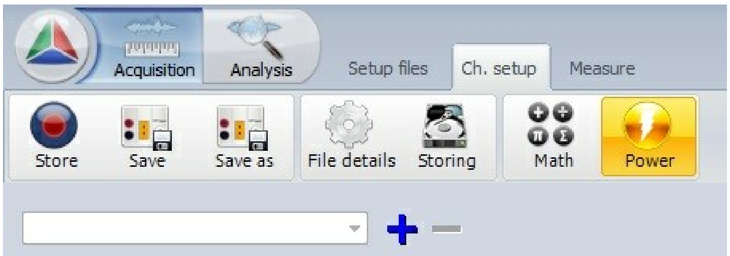
It is very important that the *measurement* Range is chosen *according* to the expected signal.
We need to take special care for the settings of the Lowpass filter. If this setting is *lower* than half of the sample rate, it will *cut* the signals already in the range of the measurements. Sometimes this 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

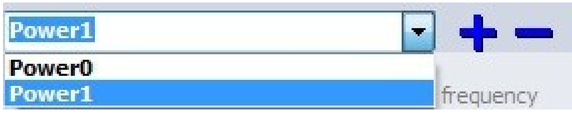
for information about ***Current measurement setup*** *see* → **Calibration current**

Add / New Power module



When we *switch* first time to Power tab, this tab area is empty (also *Order tracking module list*), except there only - **Add** button is active.

After *selecting* a Power tab, press button to add *new* **Power module**. *Several* modules can be used (added) within a session. All new *Power modules* are named automatically with Module n, where n= *running number* and appear on *Power module* drop down list, from which can be selected to set up.



Delete Power module

Each module can be deleted by *selecting* him and then using minus button.

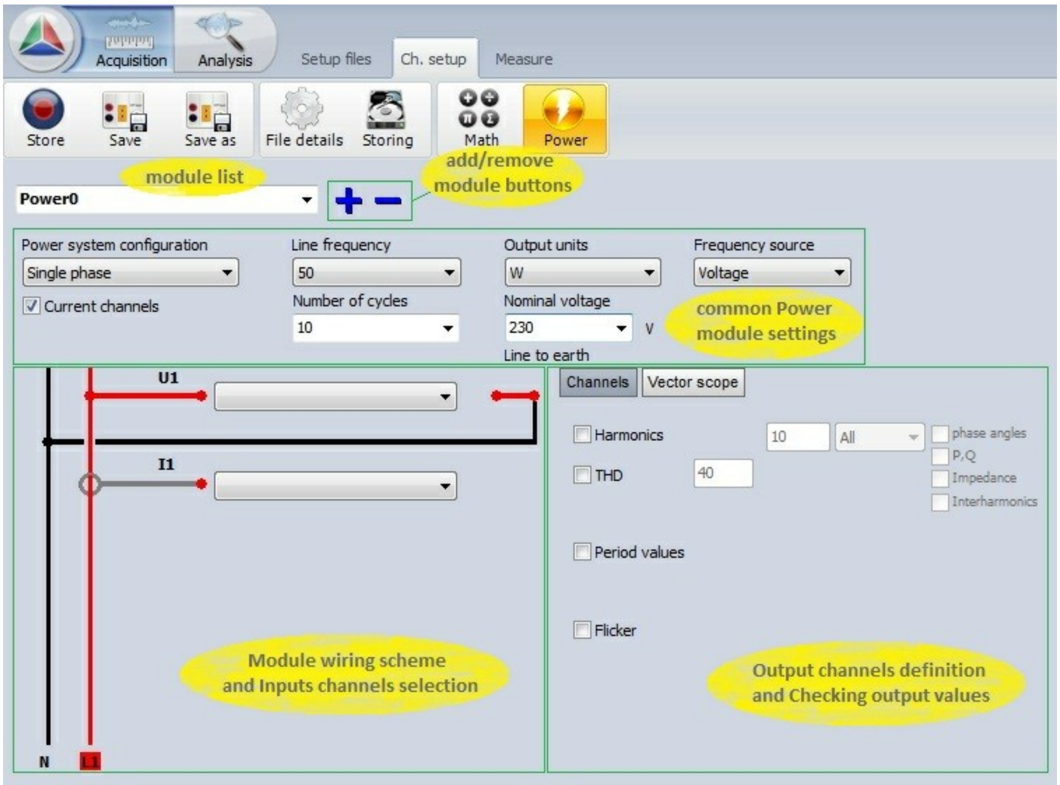
WARNING:	*Be careful - there is no UNDO function for this command. If you delete a power module, it cannot be recovered.*

Basic procedures of **Power** application setup are:

- **Power module setup** for applied hardware
- **Calibration current**

Power module setup

After adding *new* **Power module** screen like this will appear:



This screen has the following main sections:

- **Power system configuration**
- **Common Power module settings**
- **Wiring scheme / Input** channels selection
- **Output channels** definition
- **Checking output values - vector scope**
- **Current calibration**

Power system configuration

We can define *type* of power system from Power system configuration drop down list.

DEWESoft enable following *types* of power modules:

- one type of **Single phase** with *enabled / disabled* Current channels
- four types of 3 phase systems:
  - **3 - Phase star**
  - **3 - Phase Aron**
  - **3 - Phase delta**
  - **3 - Phase V**

All 3 phase types can have *enabled / disabled* Current channels and Calculate line voltages possibility.

for closed description of these ***types*** *see* → **Wiring scheme / Input channels**

for information about ***Current channels*** and ***Calculate line voltages*** *see* → **Output channels**

Common Power module settings

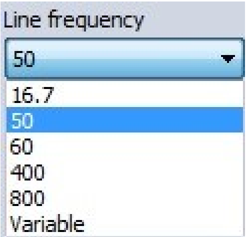
On this section can be defined settings:

- **Power system configuration** → *see* → *above* and **Wiring scheme / Input channels**

- **Current channels** and **Calculate line voltages**

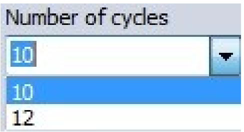
  for information about ***Current channels*** and ***Calculate line voltages*** *see* → **Output channels**

- **Line frequency**



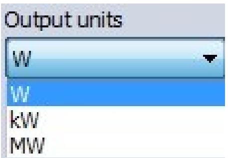
Line frequency depends on the grid frequency. We can have 16 2/3 (in trains), 50 Hz (Europe), 60 Hz (USA); 400 and 800 Hz (vehicle power systems) or Variable (for frequency *converters*).

- **Number of cycles**



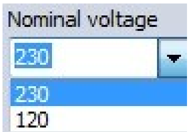
Choose from drop down list or freely enter any value. Standard values are 10 for 50 Hz grids and 12 for 60 Hz grids. This basically *defines* the *measurement interval*.

- **Output units**



Choose *power unit* for your system from drop down list.

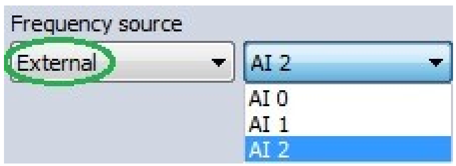
- **Nominal voltage**



Choose *nominal voltage* of your system from 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 drop down list. When we select External as source, *input channel* for this source can be selected from drop down list of available channels (*see* right).



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.

Wiring scheme / Input channels selection

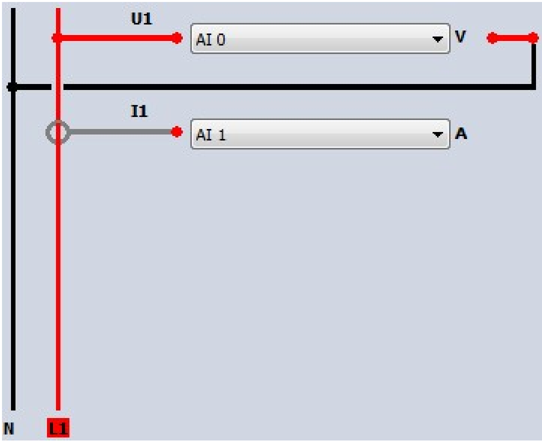
On this left lower part of the DEWESoft **Power** *Setup* screen ***principle module wiring scheme*** with ***input channels*** selection fields are displayed.

Module wiring scheme

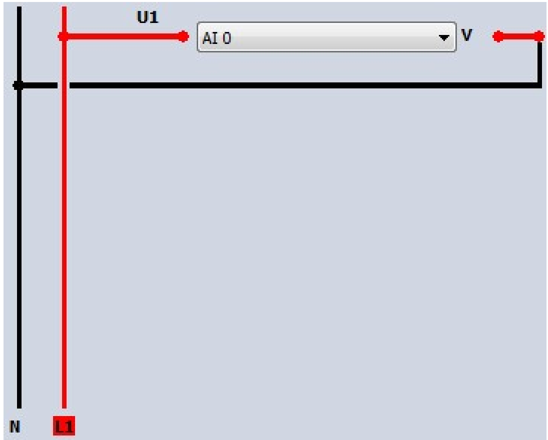
This scheme depends from selection in Power system configuration and Current channels field:

- **Single phase**

Current channels choice = *enabled* (default)

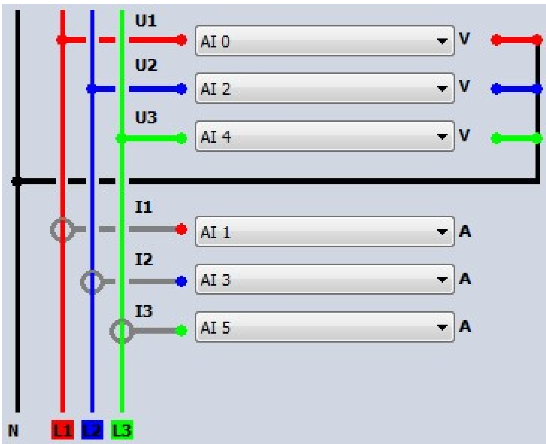


Current channels choice = *disabled*

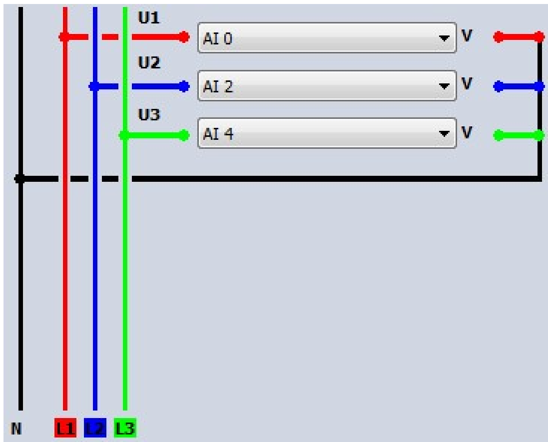


- **3 - Phase star**

Current channels choice = *enabled* (default)

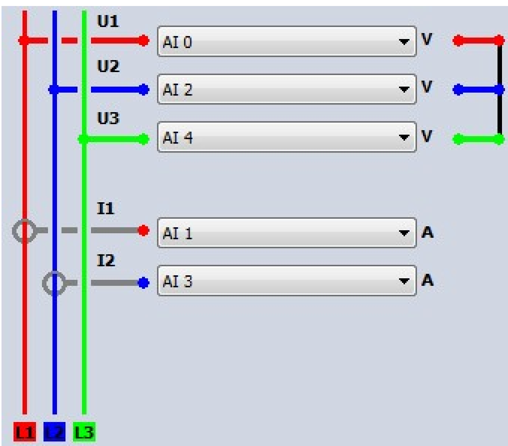


Current channels choice = *disabled*

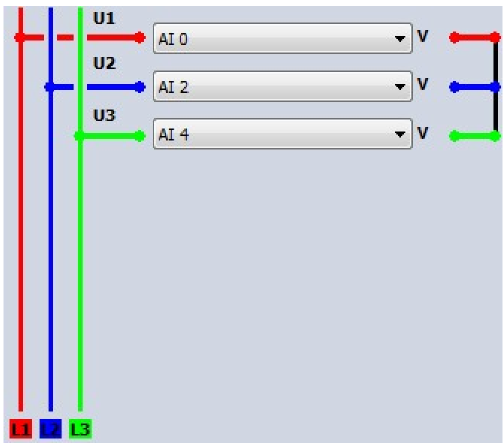


- **3 - phase Aron**

Current channels choice = *enabled* (default)

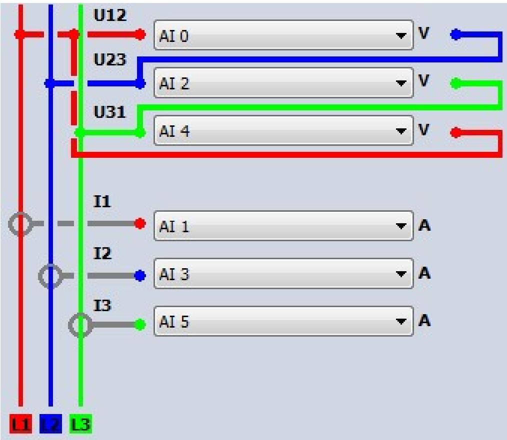


Current channels choice = *disabled*

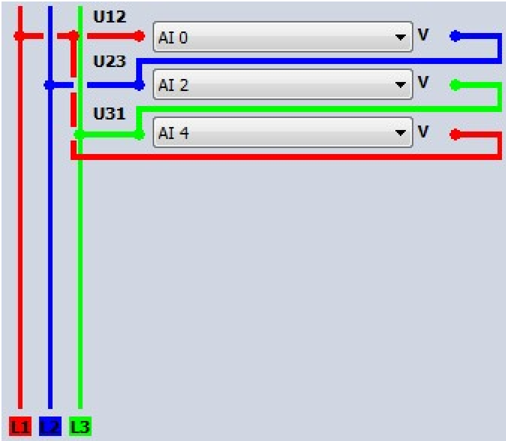


- **3 - phase delta**

Current channels choice = *enabled* (default)

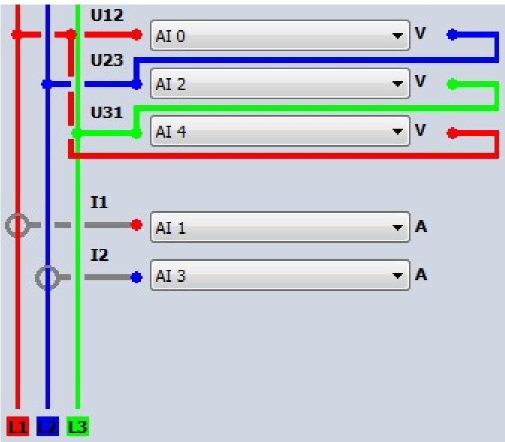


Current channels choice = *disabled*

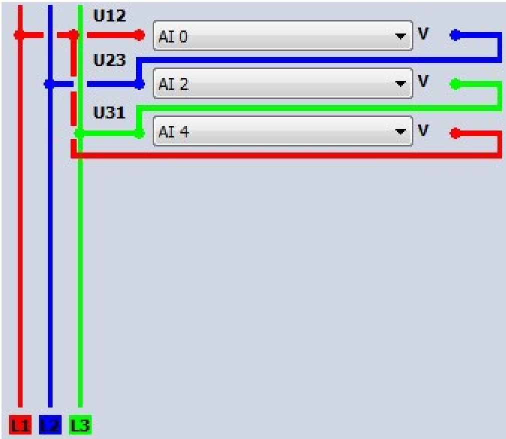


- **3 - phase V**

Current channels choice = *enabled* (default)



Current channels choice = *disabled*



Input channels selection

DEWESoft provide on module wiring scheme also all 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:

Output channels definition

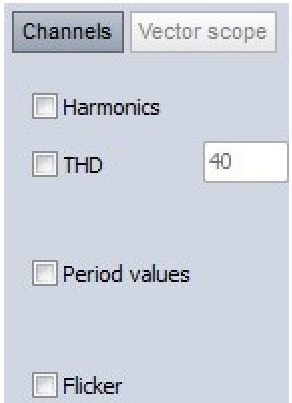
When 

button is selected, on right left lower part of the DEWESoft **Power** *Setup* screen we can *define **output channels***. Settings of output channels depend on module type selection in Power system configuration, Current channels and Calculate line voltages fields:

- **Single phase**

In this section we can select for this power module *type* following additional *output channels*:

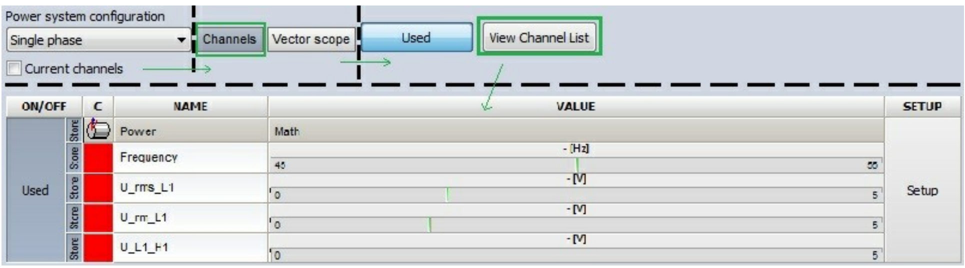
- **Harmonics**
- **THD**
- **Period values**
- **Flicker**



1. **basic output channels** → **no** option is checked

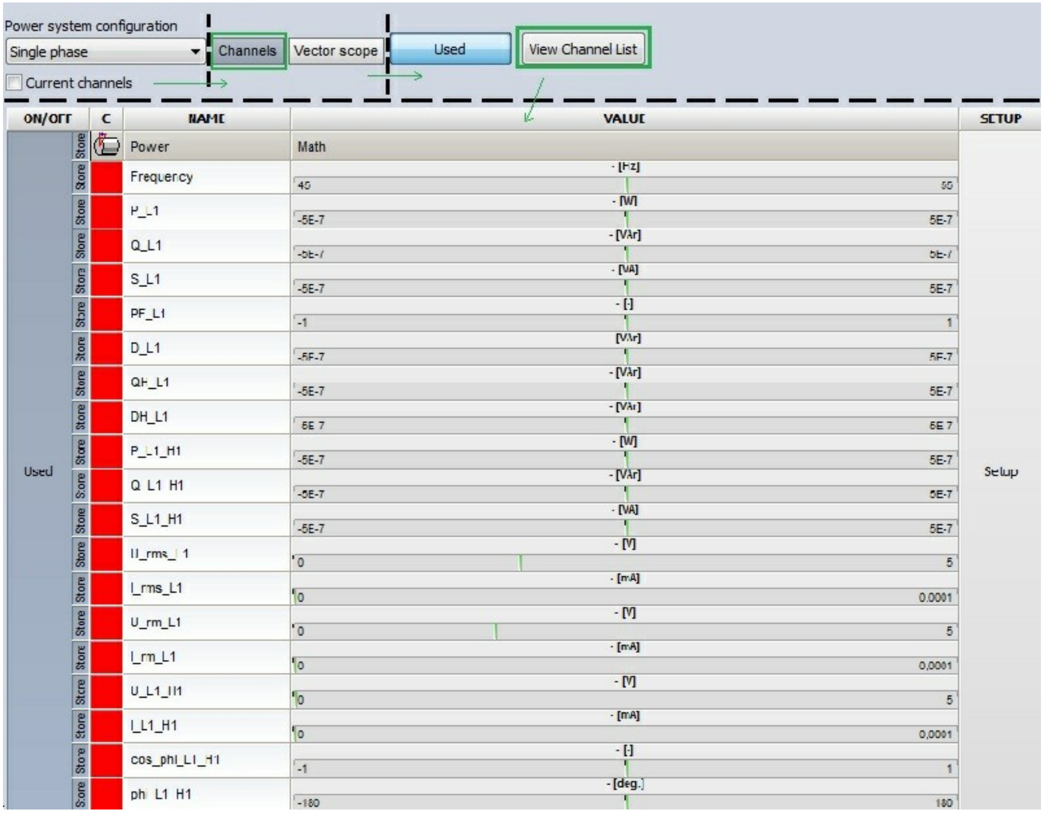
DEWESoft automatically create following *output channels*:

Current channels choice = *disabled*



To 'Used' analog channels *voltage* (U_H1 and U_rms) and *frequency* channels are added.

Current channels choice = *enabled*



To basic channels (*see* on left) *emphasized* channels are added → *current* and *power* channels.

NOTE: H1 *harmonics channels* are created anyway for:

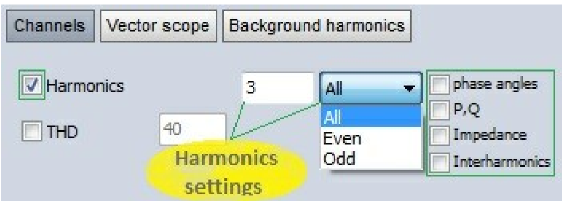
- *voltage* and *current channels* (U and I)
- *power channels* (P, Q, S, phi, cos phi)

2. <u>Harmonics</u> output channels → Harmonics option is checked

The *line voltage* isn't perfect and can have *distortions*, which are nicely shown as *harmonics* in the *frequency spectrum*.

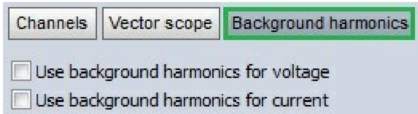
When the Harmonics option is checked we can:

- 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 (*see* **additional harmonics option** below)



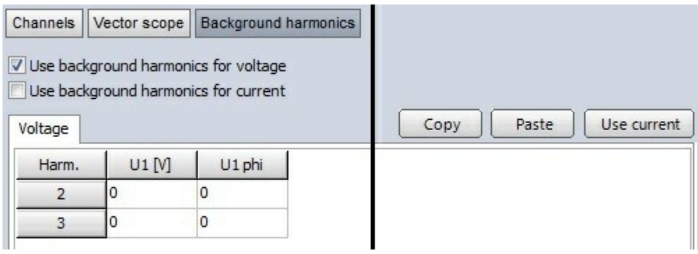
Background harmonics

On upper picture you can see additional Background harmonics button, which appear by checked Harmonics option. The background harmonics are used to *compensate* normal load and to see the *difference* from this normal conditions.

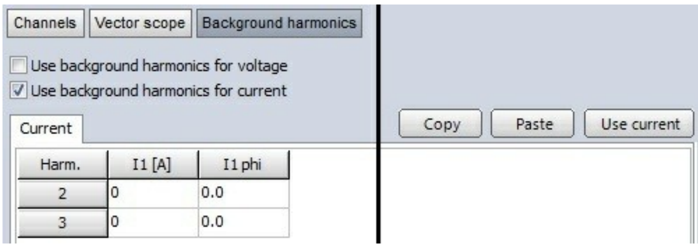


When we select this button two option can be selected to *use* and *define* background harmonics:

- Use background harmonics for voltage



- Use background harmonics for current

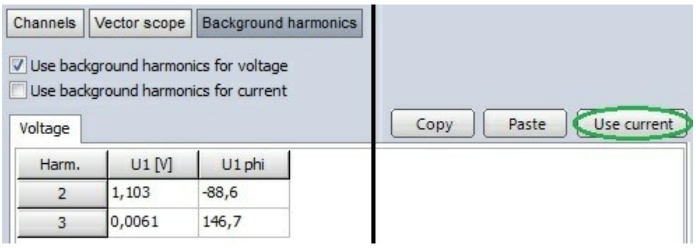


When one from this options (or both) is checked, table with voltage (U1 in V unit and U1 phi - *phase angle*) / current values (I1 in A unit and I1 phi - *phase angle*) for defined number - 1 background harmonics appear.

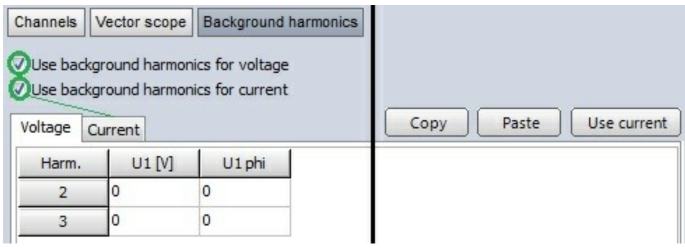
In example above we have selected 3 harmonics, therefore voltage / current value for *second* and *3rd* harmonics can be defined.

Table Voltage / current values can be defined with:

- *selecting* cell and *enter* value
- Copy and Paste function
- selecting Use current button to enter current value in table cells (*see* right)

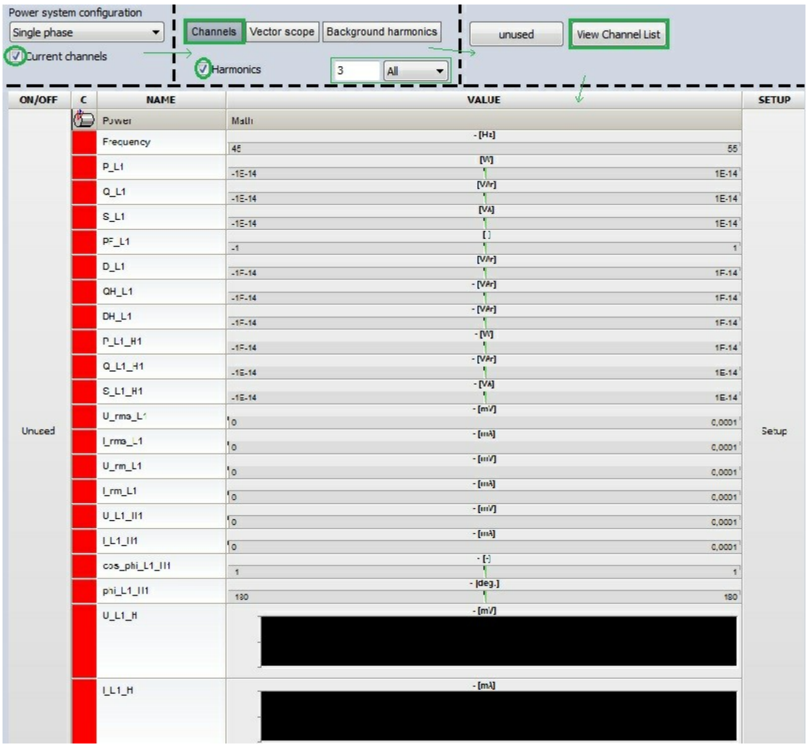


When both options for background harmonics are checked, table with current values appear in *separate tab* (*see* right).



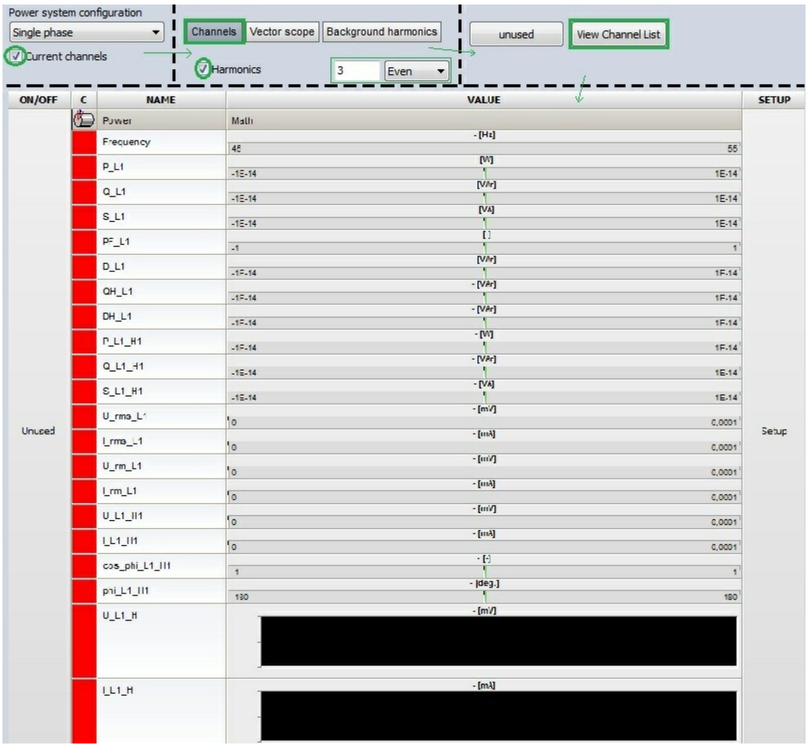
DEWESoft create following *harmonics output channels* for Single phase Power module (example for 3 harmonics and Current channels choice = *enabled*):

To basic channels for **All** Harmonics (*see* **above**) *emphasized* channels are added → *current* and *power harmonics* channels.



When Current channels choice = *disabled*, *current* channels are not created (e.g. above only U_... channels).

To basic channels for **Even** Harmonics (*see* **above**) *emphasized* channels are added → *current* and *power harmonics* channels.

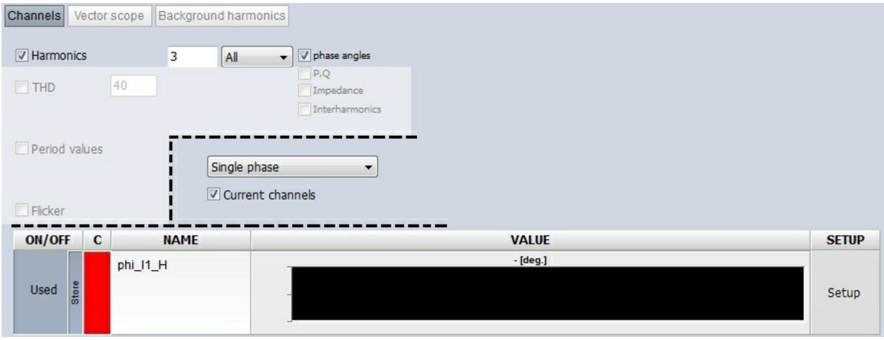


In above example *odd* channels are not created (except …_H1 → see note above).

When phase angles, P,Q, Impedance and Interharmonics option is selected, DEWESoft create following additional *harmonics output channels*:

a) phase angles option is checked

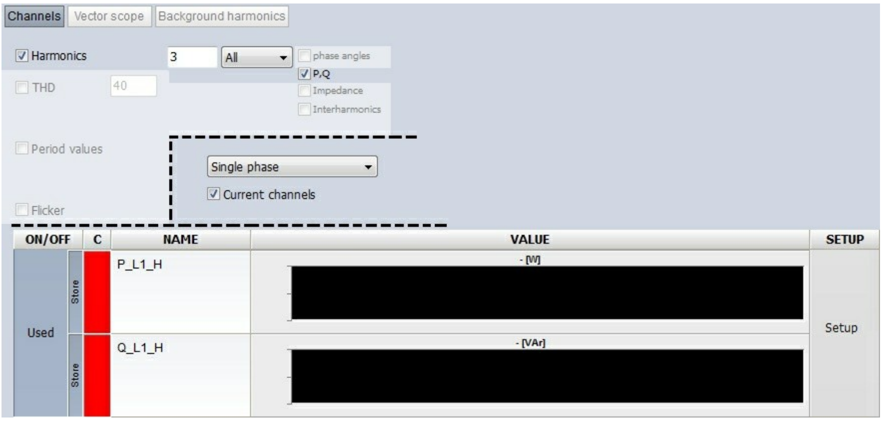
To basic channels for **All** Harmonics (*see* above left) *emphasized* *phase angles* pfi channels for each *harmonic* are added:



For **Even** Harmonics *odd* channels are not created (except …_H1 → see note above).

b) P,Q option is checked

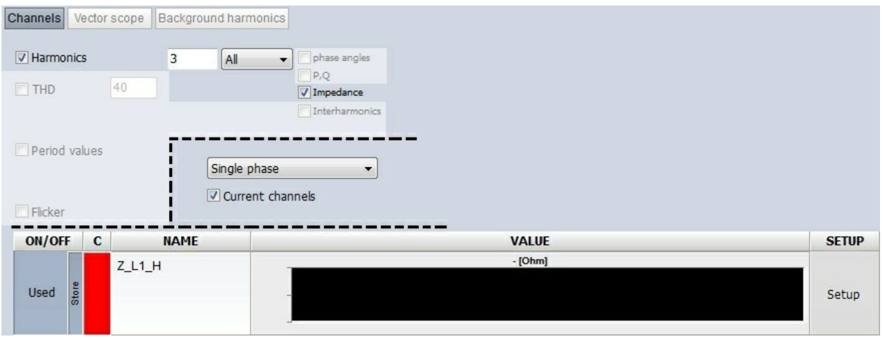
To basic channels for **All** Harmonics (*see* above left) *emphasized* P - *active power* and Q - *reactive power* channels for each *harmonic* are added:



For **Even** Harmonics *odd* channels are not created (except …_H1 → see note above).

c) Impedance option is checked

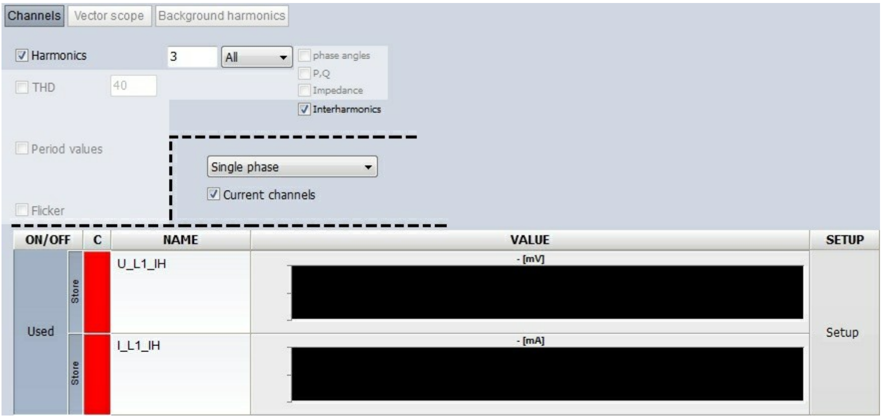
To basic channels for **All** Harmonics (*see* **above** left) *emphasized* *Impedance* Z channels are added



For **Even** Harmonics *odd* channels are not created.

d) Interharmonics option is checked

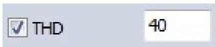
To basic channels for **All** Harmonics (*see* **above** left) *emphasized* Interharmonics ..._IH channels are added

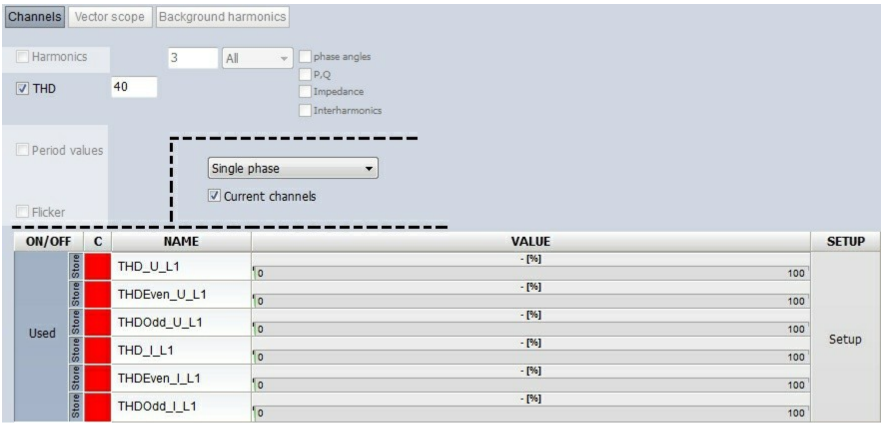


For **Even** Harmonics *odd* channels are not created.

3. THD option

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).



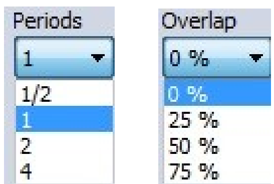
To basic channels (*see* **above**) *emphasized* THD channels are added (not only for *voltage* and *current* but also their *odd* and *even harmonics*.

4. Period values option

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 and with

– **Overlap** = 0%, 25%, 50% or 75%, which can be selected from drop down list

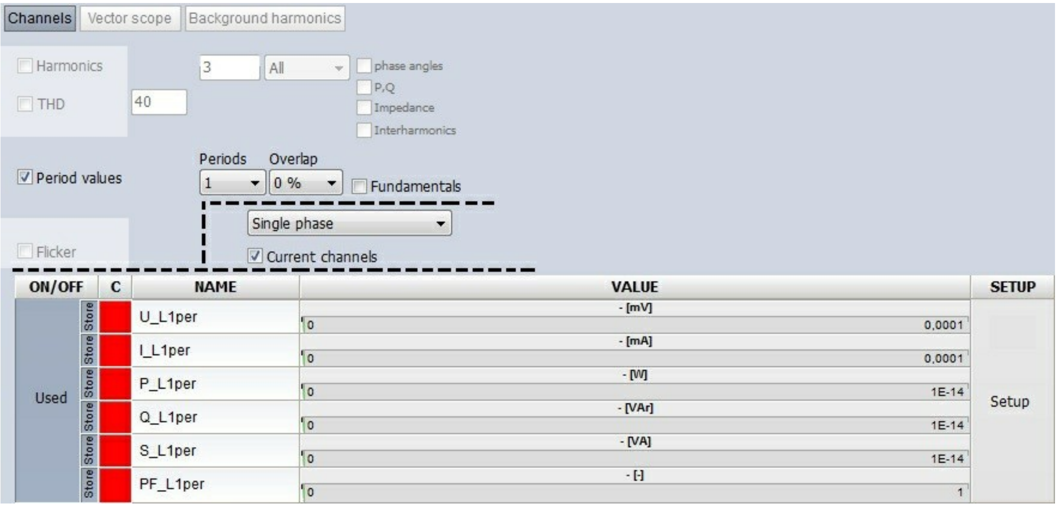


This is very helpful for *triggering*.

for information about ***Overlap*** *see* → User Guide → **FFT instrument settings**

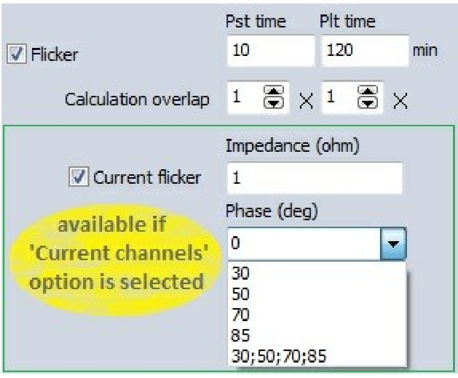
To create *period values output channels* this option must be checked.

To basic channels (*see* **above**) *emphasized* period values ...per channels are added (not only for *voltage* and *current* but also for all *powers*: S - *apparent*, P - *active*, Q - *reactive*).



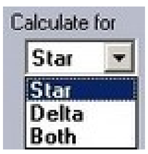
5. Flicker option

The flicker is actually *power quality parameters* measuring *low frequency* distortions of the *voltage*.



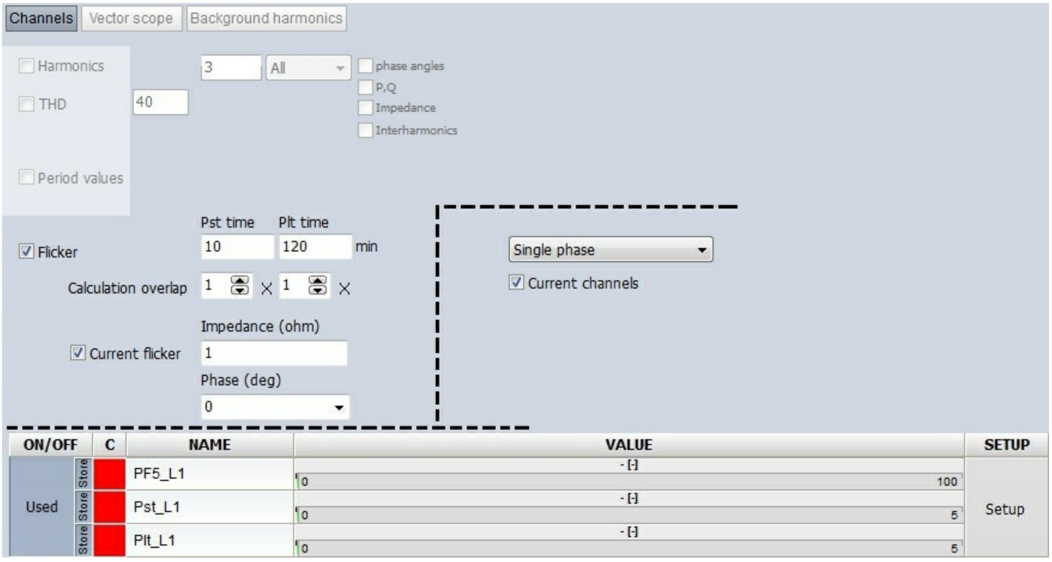
For this option we can set:

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:



To create *flicker output channels* this option must be checked.

To basic channels (*see* **above**) *emphasized* flicker (*powers*) P... channels are added.



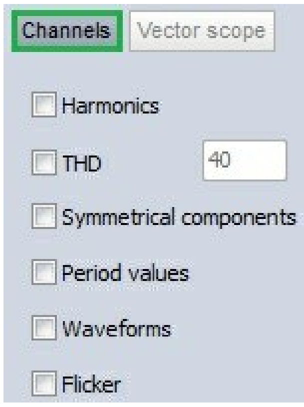
Current flicker calculates the influence of the current on the flicker (contribution of the device being measured on the flicker).

We need to enter impedance in Ohm and Phase as two additional parameters for calculation.

- **3 phase** systems

In this section we can select for this power module *type* following additional *output channels*:

- **Harmonics**
- **THD**
- **Symmetrical components**
- **Period values**
- **Waveforms**
- **Flicker**



NOTE:	Following description and examples to be valid for all types 3 phase systems: 3-phase star, 3-phase Aron, 3-phase delta and 3-phase V.

1. **basic output channels** → **no** option are checked

DEWESoft automatically create following *output channels*:

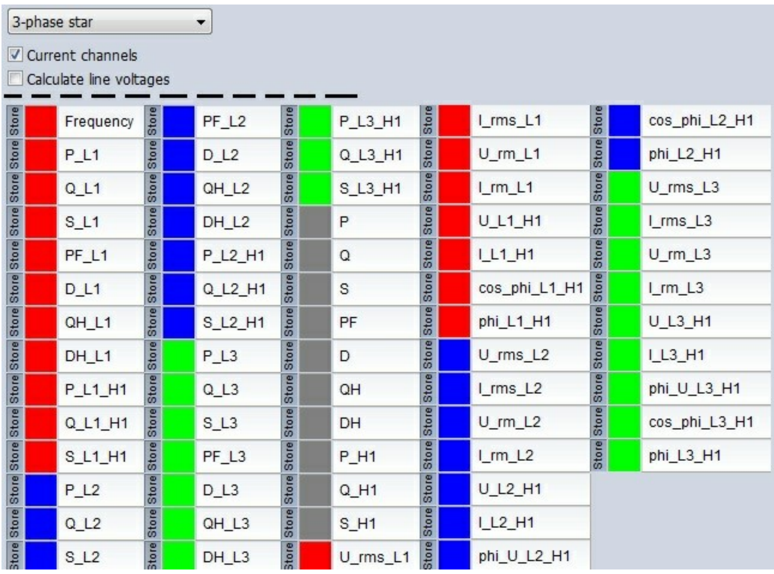
Current channels choice = *disabled*
Calculate line voltages choice = *disabled*

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***

To basic channels (*see* on left) *emphasized* channels are added → between lines *voltages* channels.

Current channels choice = ***enabled***
Calculate line voltages choice = *disabled*



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)

2. Harmonics output channels → Harmonics option is checked

For information about this option *settings* and *output channels* → *see* above → **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.

3. THD output channels → THD option is checked

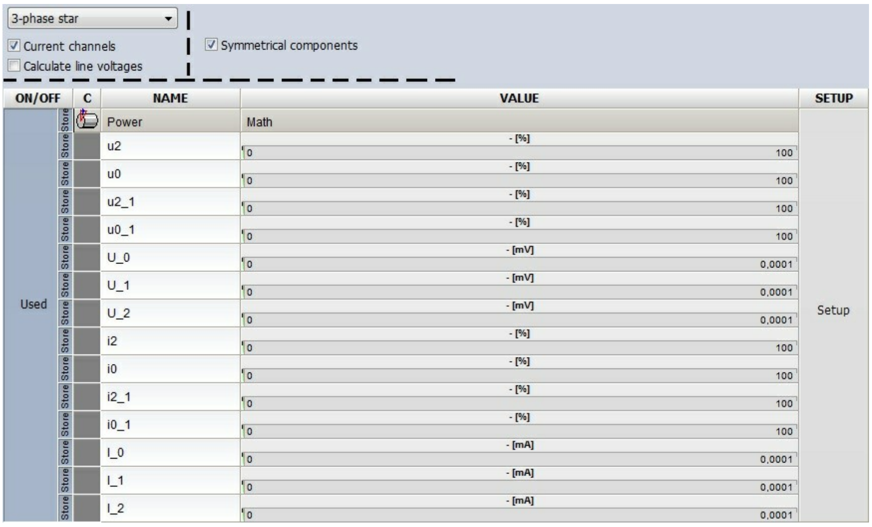
For information about this option *settings* and *output channels* → *see* above → **Single phase THD option**

NOTE: Additional to *Single phase THD channels* only the same kind **output channels** for second - L2 and 3rd - L3phase line are created.

4. Symmetrical components → Symmetrical components option is checked

We can calculate Symmetrical components by selecting 

check box. Symmetrical components are available *only on three phase* systems and tell us the *unbalance* of the grid.



5. **Period values output channels** → Period values option is checked

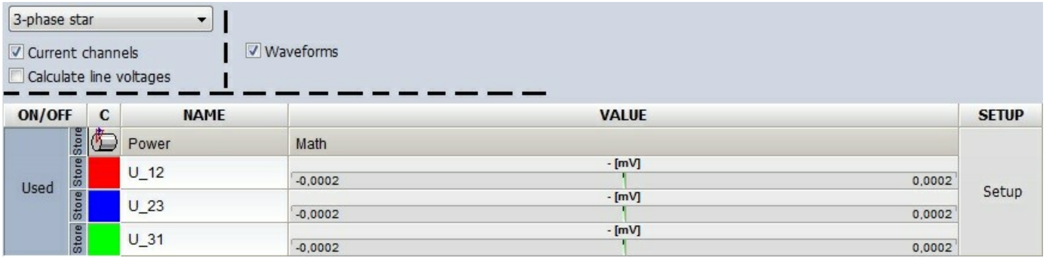
For information about this option *settings* and *output channels* → *see* above → **Single phase Period values option**

NOTE: Additional to *Single phase Period values channels* only the same kind **output channels** for second - L2 and 3rd - L3 phase line are created.

6. **Waveforms** → Waveforms option is checked

If we measure *line to ground* voltages, we can calculate Waveforms for the *line to line* voltages by selecting 

option.



7. **Flicker output channels** → Flicker option is checked

For information about this option *settings* and *output channels* → *see* above → **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.

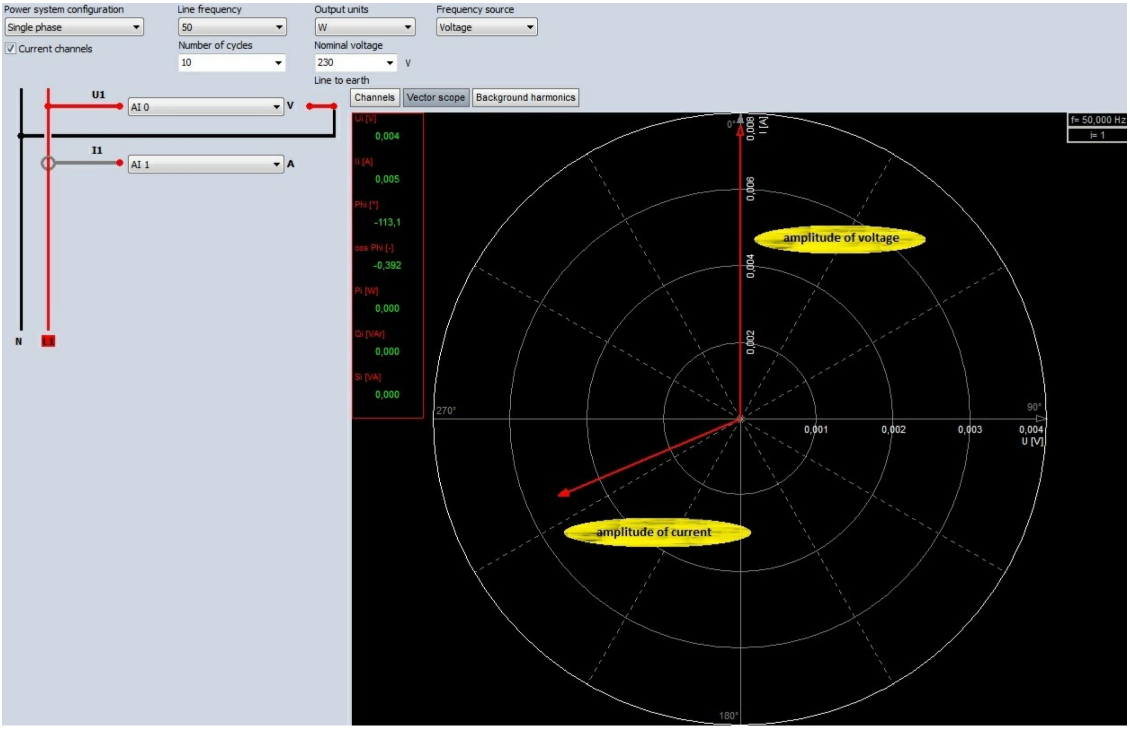
Checking output values - vector scope

When 

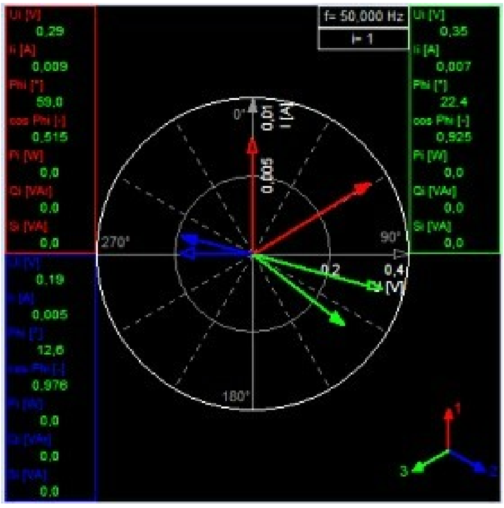
- **Vector scope** button is selected, on right lower part of the DEWESoft **Power** *Setup* screen the *vector scope* instrument is displayed to check if everything is ***connected correctly***.

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.

Example for *Single phase* system:



Principle example for *Three phase* system (single phase line is displayed in different color):



for information about ***Vector scope*** see → User Guide → **Vector scope**

Calibration current input

The **currents** will be measured:

- 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.

  Both principles include a *phase shift* of the *output*.

- 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 also very careful *not to exceed* maximum current of the shunt.

1. 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* → User Guide → **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.

2. Scale the shunt resistor

Module setup can be done from DEWESoft *Setup* → Analog tab **Channel setup**.

for information about ***Modules shunt resistor settings*** *see* → User Guide → **Channel setup - Module**

3. 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**.

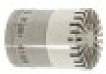
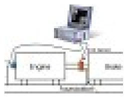
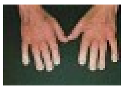
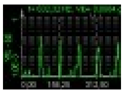
for information about ***transfer curves*** and ***sensor correction*** *see* → User Guide → **Sensor database**

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*.

2.12 Dynamic signal analysis

DEWESoft Dynamic signal analysis covers a wide range of measurements in the field of **structural dynamics**, **industrial acoustics** and **machine diagnostics**.

	Sound level	explains procedures for working with **Sound level module,** which is used to calculate levels of sound with time and frequency weighting
	Torsional vibration	explains procedures for working with **Torsional vibration module** used to measure dynamic and static bending and vibration of the *shafts*
	Human vibration	explains procedures for working with **Human vibration module**, used to evaluate effects of vibration on *human body*
	Order tracking	explains procedures for working with **Order tracking module** used to extract the harmonics during *machine run up* and *run downs*
	Combustion analysis	explains procedures for working with **Combustion analysis module** used to calculate parameters of internal combustion engines
	Frequency response	explains procedures for working with **Frequency response analysis** used to determine natural frequencies of system under test.

for information how to ***initialize above special DEWESoft modules*** *see* → System Settings → **Math & Applications**

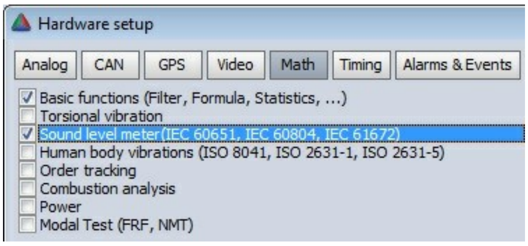
2.12.1 Sound level

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 hints about ***Sound Level Application*** see → DEWESoft Tutorials

After *required hardware - microphone installation* and *setup* (this procedure is to perform *only* by installation new or changing hardware) according to manufacturer instructions, DEWESoft **Sound level** measurement is allowed by *selecting* the Sound level meter checkbox in *System* → Hardware setup → **Math** tab.

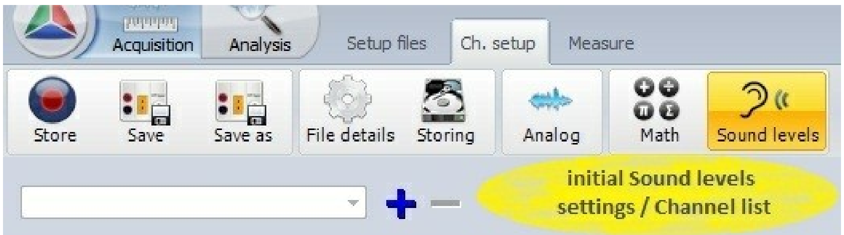


After selecting this option, a tab labeled Sound levels appears in the DEWESoft *Setup* screen (*see* picture below) to *define* and *setup sound levels module*(s).

To use sound level module, please *select* and *set up* first one or few *analog channel* in Analog tab to *measure* the sound

see → User Guide → **Analog in Channel Setup**

Add / New Sound level module



When we *switch* first time to Sound levels tab, this tab area is empty (also *Sound level module list*), except there only + - **Add** button is active.

After *selecting* the Sound levels tab, press + button to add *new* **Sound level module**. *Several* modules can be used (added) within a session. All new *Sound level modules* are named automatically with Sound levels n, where n= *running number* and appear on *Sound level module* drop down list, from which can be selected to set up.

Delete Sound level module

Each module can be <u>deleted</u> by *selecting* him and then using [−] - minus button.

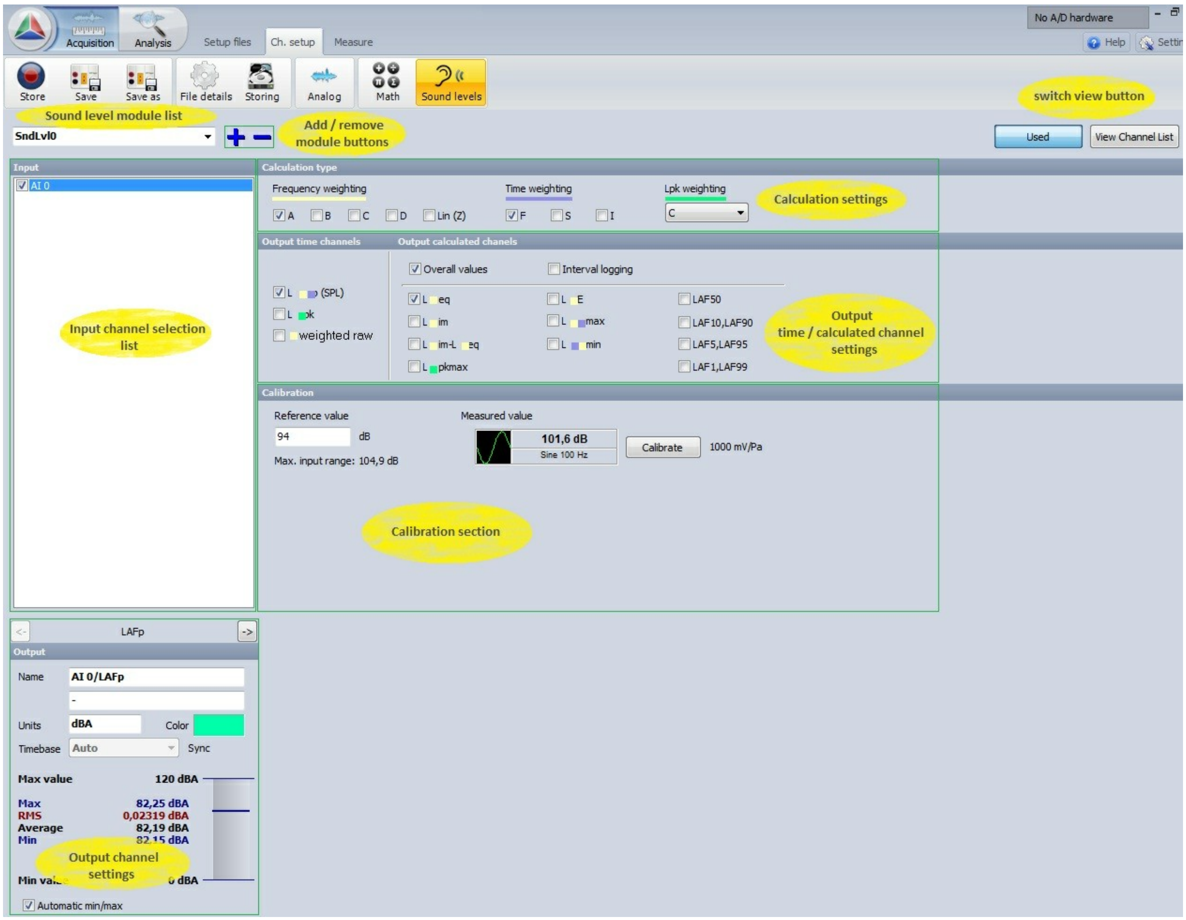
WARNING:	*Be careful - there is no UNDO function for this command. If you delete a Sound module, it cannot be recovered.*

Basic procedures of **Sound level** application setup are:

- **Sound level / channel setup** for applied hardware
- **Microphone calibration**

Sound level / channel setup

After 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**

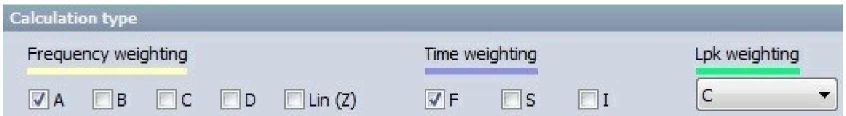
With using **View Channel List** button we can show different view on *Sound levels module* setup → **Channel list** with all *defined output channels* appear.

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

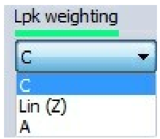
We have several options to choose in Calculation type section:



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 in the analog setup, 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* → User Guide → **Channel Setup**

Min val
Max val

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 entering the value in the fields. Then the max value is not shown in the bar graph since it is displayed in edit box.

Right clicking on the edit box for *min* and *max values* gives us few more options:

Apply to this channel in all modules — Applies current setting to the *specific* channel in *all* modules (like for all RMS values in statistic module).

Apply to all channels in this module — Applies the settings for *all* the channels in this module (for example for MIN, MAX and AVE values of certain input channel).

Apply to all channels in all module — Applies the *current* setting for *all* output channels.

Set to Auto — Sets back the value to *Automatic*.

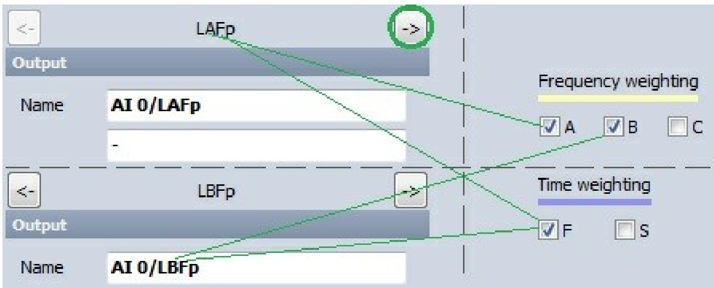
Sound level module (also with only *one input* channel) can have more **output** channels according selection in:

- **Output time channels**

Select desired option with checking box beside caption.

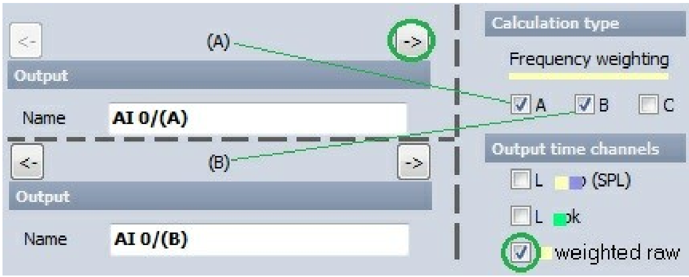
We have three *types* of Output time channels:

1. $L_{FT}p$ - *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:

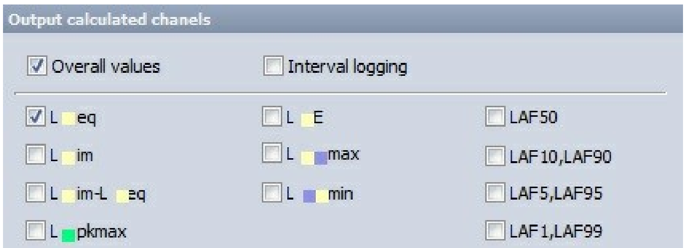


2. L_{Lpk}pk value, which shows the *current maximum value* of the sound levels and depends on Calculation type selection → **Lpk weighting**. For selected choice in this section <u>one</u> *separate* **output channel** is created.

3. $_F$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 <u>one</u> *separate* **output channel** is created:

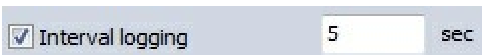


- **<u>Output calculated channels</u>**



Select desired option with checking box beside caption. First choose upper calculation parameter (at least one must be selected for calculated channels):

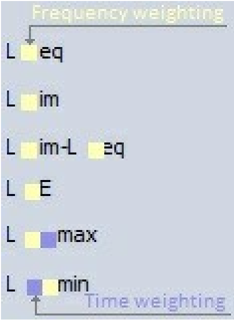
- **Overall values** - we have only *one* value at the end of the measurement
- **Interval logging** - the *time interval* for logging is defined.



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.*

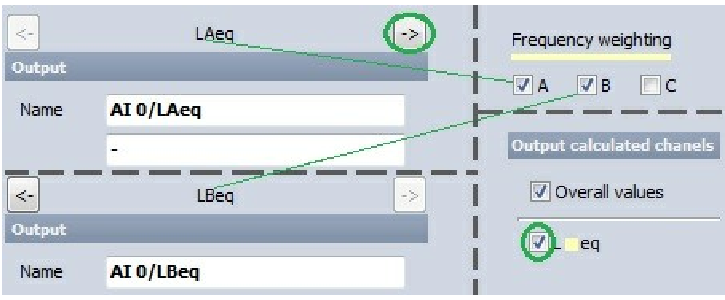
- or both

The *values* which can be **calculated** are:



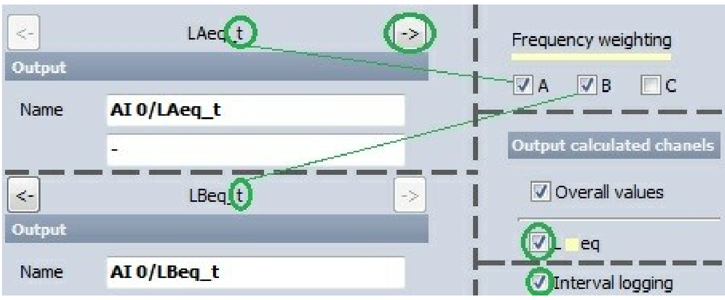
- *frequency* weighted $L_F$eq value, which is equivalent sound *level*
- $L_F$im tells the *impulsivity* of sound and is *impulse* weighted equivalent; the difference between those two values is *calculated* as $L_F$im-$L_F$eq
- L_FE is *frequency* weighted sound *energy*
- L_{FT}max and L_{FT}min is *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 <u>one</u> *separate* **output channel** is created.



When **Interval logging** is selected and defined:

also for these combination of selection in this sections <u>one</u> *separate* **output channel** is created:

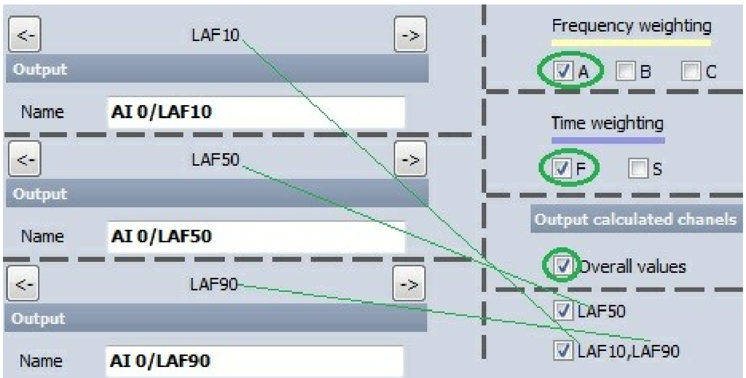


NOTE: Interval logging is designated with _t characters on end of channel name.

- L_{Lpk}pkmax is wither *C* or *linear* weighted maximum *peak* value of sound; depends on Calculation type selection → **Lpk weighting**. For selected choice <u>one</u> *separate* **output channel** is created.

- *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 <- and -> to navigate between the *output channels* (*see* pictures above).

Channel list view

After adding *new* **Sound level module** on upper right part of setup screen appear new button to switch between *module setup* and **channel list** view.

When this button is selected, Channel list screen with all *defined* output channels appear:

First let's look at the columns that are shown in *Sound Level channel list*, and describe what each is for:

ON/OFF – *activate / deactivate* Sound Level channel - press the Unused button - it will change now to **Used** - to set the desired Sound Level channel ***active*** for *Sound Level 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 – Sound Level channel *name* with all defined *output channels*- just click into the **NAME** field of output channel and you can *enter* the new ***name*** as usual text

VALUE – current Sound Level channel *value*

SETUP – return to Sound Level module setup view (*see* above)

for information about ***On/Off***, ***C*** and ***Name*** column *see* → User Guide → **Channel setup**

View Channel List button now change to 

and we can call back *module setup* view with selecting this button or with selecting **Setup** button in channel list.

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*.

1. 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* → User Guide → **Channel setup - Calibration**

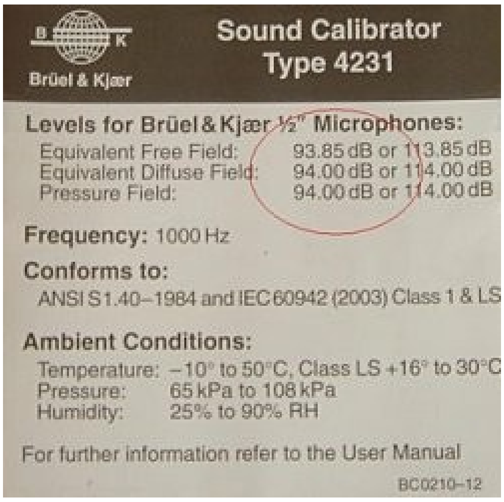
NOTE: First, as usual, we define and enter the *physical* '*Units of measurement*' - Pa is *defined* as the **Units**.
Therefore we need to ***scale*** it to *physical quantity*.
The *Reference sensitivity* is the key value to enter in the DEWESoft setup. Then it is the best to go to scaling '**by function**' tab, check the **Sensitivity** and then enter in V/Pa value from ***calibration data sheet***.

2. 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 on right it is 94 dB.

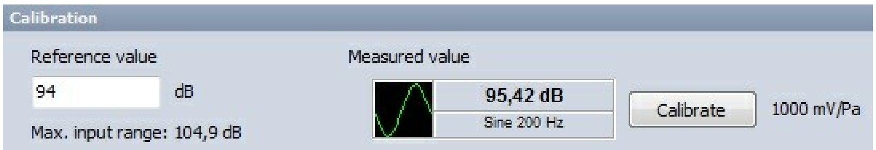
This value is *directly* entered in the **Reference value** field of *Calibration* section in the **sound level** module:



Then we *connect* the **calibrator** to the *microphone* and switch it on. We *see directly* 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.

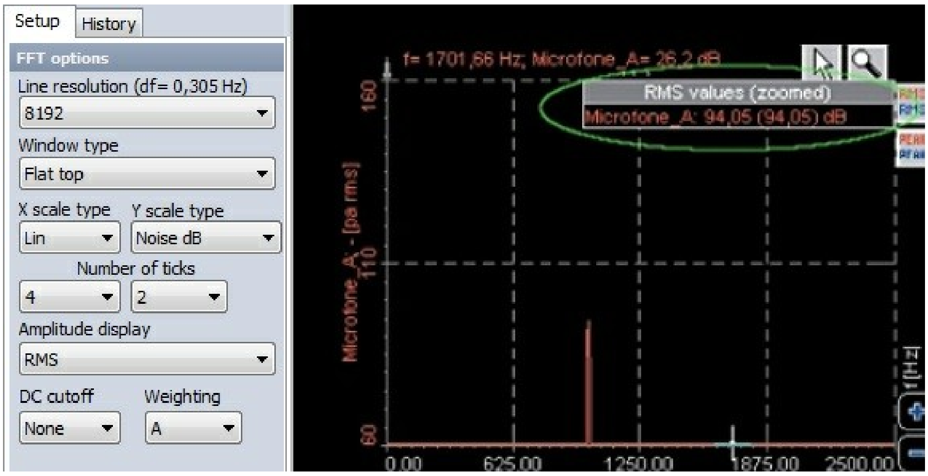
NOTE:	*We need to use setup sample rate in DEWESoft at least 5 kHz or higher to make successful sound calibration. This can be changed in DEWESoft Tuner Utility.*

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* → User Guide → **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*.

2.12.2 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**

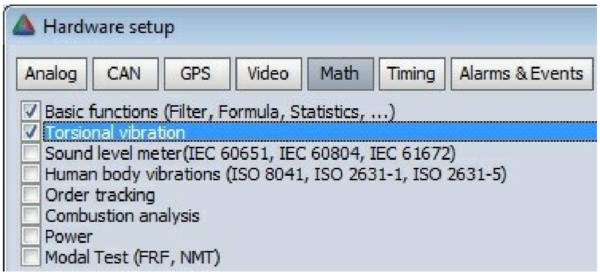
To measure *torsional* or *rotational vibration*, we need Orion card with **counter** expansion, because all other methods do not have precision needed to do this.

Required hardware	Dewe43, Sirius with counter expansion
Required software	SE or higher + TORVIB option (if order extraction is required, also ORDTR option is needed), DSA or EE
Setup sample rate	At least 10 kHz

for hints about ***Torsional vibration Application*** see → DEWESoft Tutorials

The Counter hardware can be *activating* on the DEWESoft System menu → Hardware setup... → **Analog** tab by set the required *counter channels* (CNT chnls column) in Device information list.

After activating the Counter hardware, DEWESoft **Torsional vibration** measurement is allowed by *selecting* the Torsional vibration calculation checkbox in System → **Hardware setup** → **Math** tab.

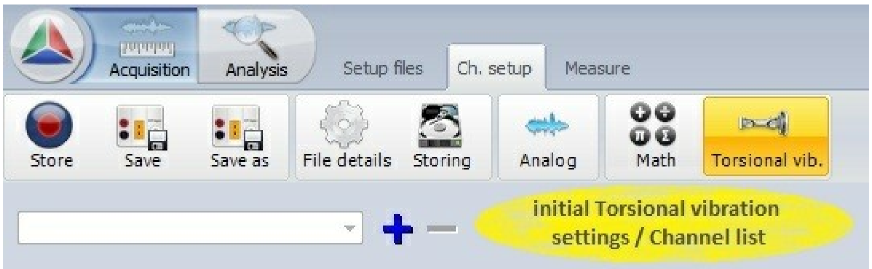


After selecting this option, a tab labeled Torsional vibration appears in the DEWESoft *Setup* screen (*see* picture below) to *define* and *setup* *Torsional vibration module*(s).

To use *Torsional vibration* module we <u>don't need</u> 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.

<u>Add / New Torsional vibration module</u>



When we *switch* first time to Torsional vibration tab, this tab area is empty (also *Torsional vibration module list*), except there only + - **Add** button is active.

After *selecting* the Torsional vibration tab, press + button to add *new* **Torsional vibration module**. *Several* modules can be used (added) within a session. All new *Torsional vibration modules* are named automatically with Torsional vibration n, where n= *running number* and appear on *Torsional vibration module* drop down list, from which can be selected to set up.

Delete Torsional vibration module

Each module can be deleted by *selecting* him and then using button.

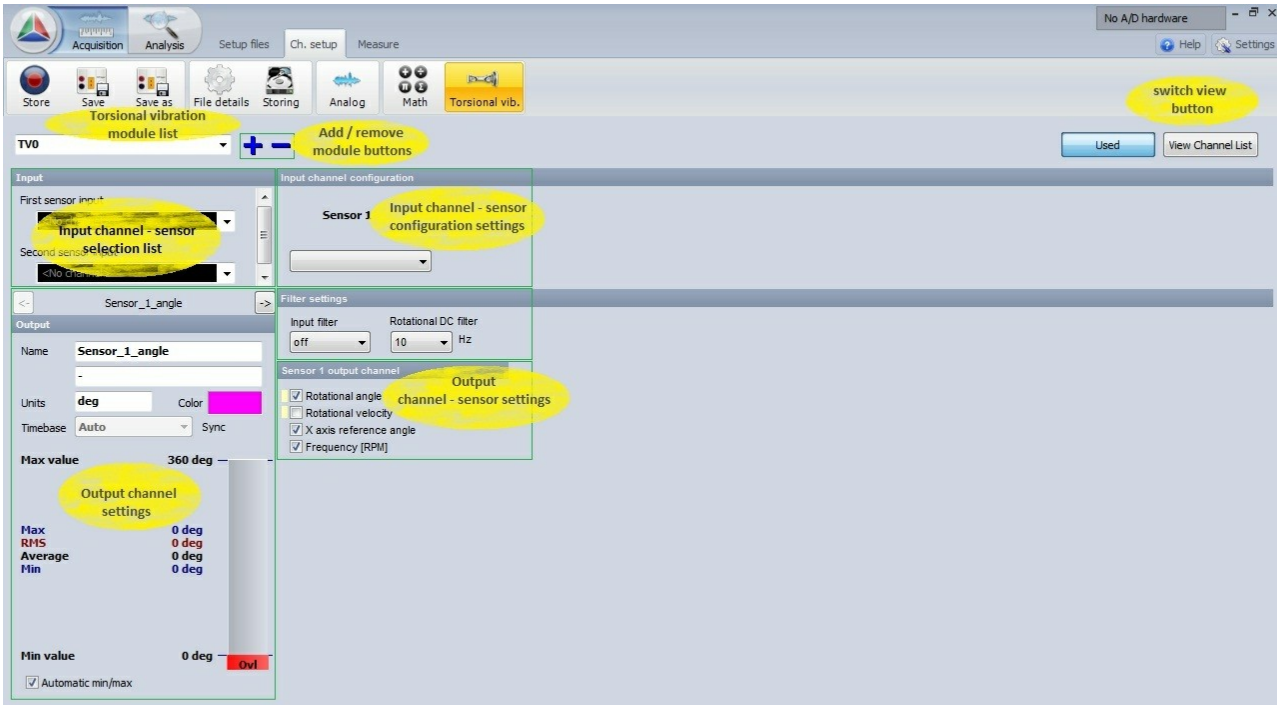
WARNING	*Be careful - there is no UNDO function for this command. If you delete a Torsional vibration module, it cannot be recovered.*

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 four main sections:

- **Input** *channel - sensor* selection
- **Output** *channel* settings
- **Input channel configuration** - for *Sensor 1*
- **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

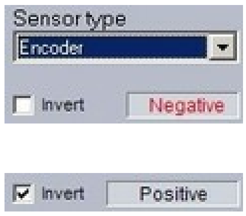
Since we have by Rotational vibration only <u>one</u> (first) sensor connected, on *Rotational vibration setup* screen only <u>one</u> for *sensor 1 Input channel configuration* section appear. On this section we can set:

- **Pulses per revolution**

enter in this field actual value pulses per revolution:

- **Sensor type**

select Geartooth or Encoder sensor type from drop down list:

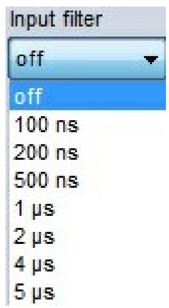


Both sensors must *count* in positive direction.

If we use **encoders**, there is a chance that the counting direction of a sensor is Negative (indicated). In this case the inputs have to be ***inverted*** by checking the Invert checkbox for this sensor.

for information about ***signal inverting*** *see* → User Guide → **Counter setup**

- **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:

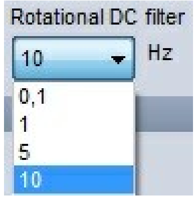
$$InputFilter\ r[s] \leq \frac{10}{RPM_{MAX} \cdot PPR}$$

RPM_{MAX} ..... max. revolution s per minute [s^{-1}]

PPR .... pulses per revolution [-]

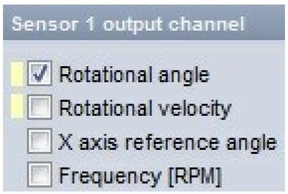
for information about ***Input filter*** *see* → User Guide →**Counter setup**

- **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*.

for detailed information about this ***Output channel*** *setting see* → User Guide → **Sound Level**

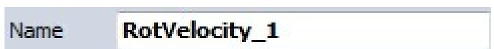
Torsional vibration module can have more **output** channels according selection in Sensor 1 output channel. For *each* selection in this section one *separate* **output channel** with default *name* (this can be changed in Name field → *see* above) is created:

For on Sensor 1 output channel section selected:

Rotation angle choice →

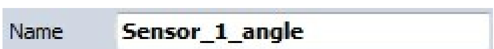
output channel is created

Rotation velocity choice →



output channel is created

X axis reference angle choice →

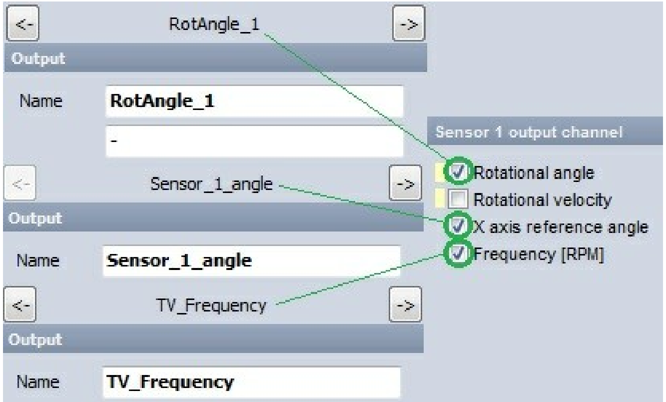


output channel is created

Frequency [RPM] choice →

output channel is created

Example:



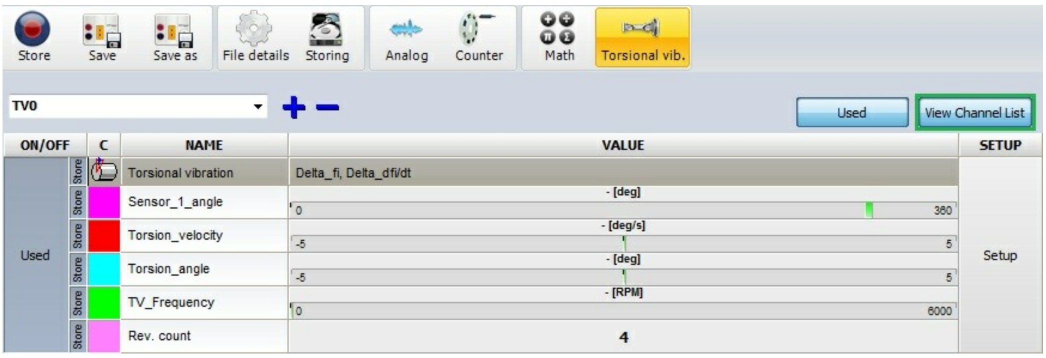
At least one **output** channels must be checked.

Use (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** on upper right part of setup screen appear new button to switch between *module setup* and **channel list** view.

When this button is selected on *Rotational vibration module* setup, *Channel list* screen with all *defined* *output channels* appear:



First let's look at the columns that are shown in Torsional vibration *channel list*, and describe what each is for:

- **ON/OFF** – *activate / deactivate* Torsional vibration channel - press the Unused button - it will change now to **Used** - to set the desired Torsional vibration channel ***active*** for *Torsional vibration 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** – Torsional vibration channel *name* with all defined *output channels*- just click into the **NAME** field of output channel and you can *enter* the new ***name*** as usual text
- **VALUE** – current Torsional vibration channel *value*
- **SETUP** – return to Torsional vibration module setup view (*see* above)

for information about ***On/Off***, ***C*** and ***Name*** column *see* → User Guide → **Channel setup**

View Channel List button now change to 

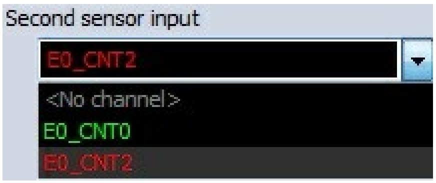
and we can call back *module setup* view with selecting this button or with selecting **Setup** button in channel list.

Torsional vibration setup

Input channel - sensor

To **measure** the **torsional** vibration we need to *select* both input channels in torsional vibration setup.

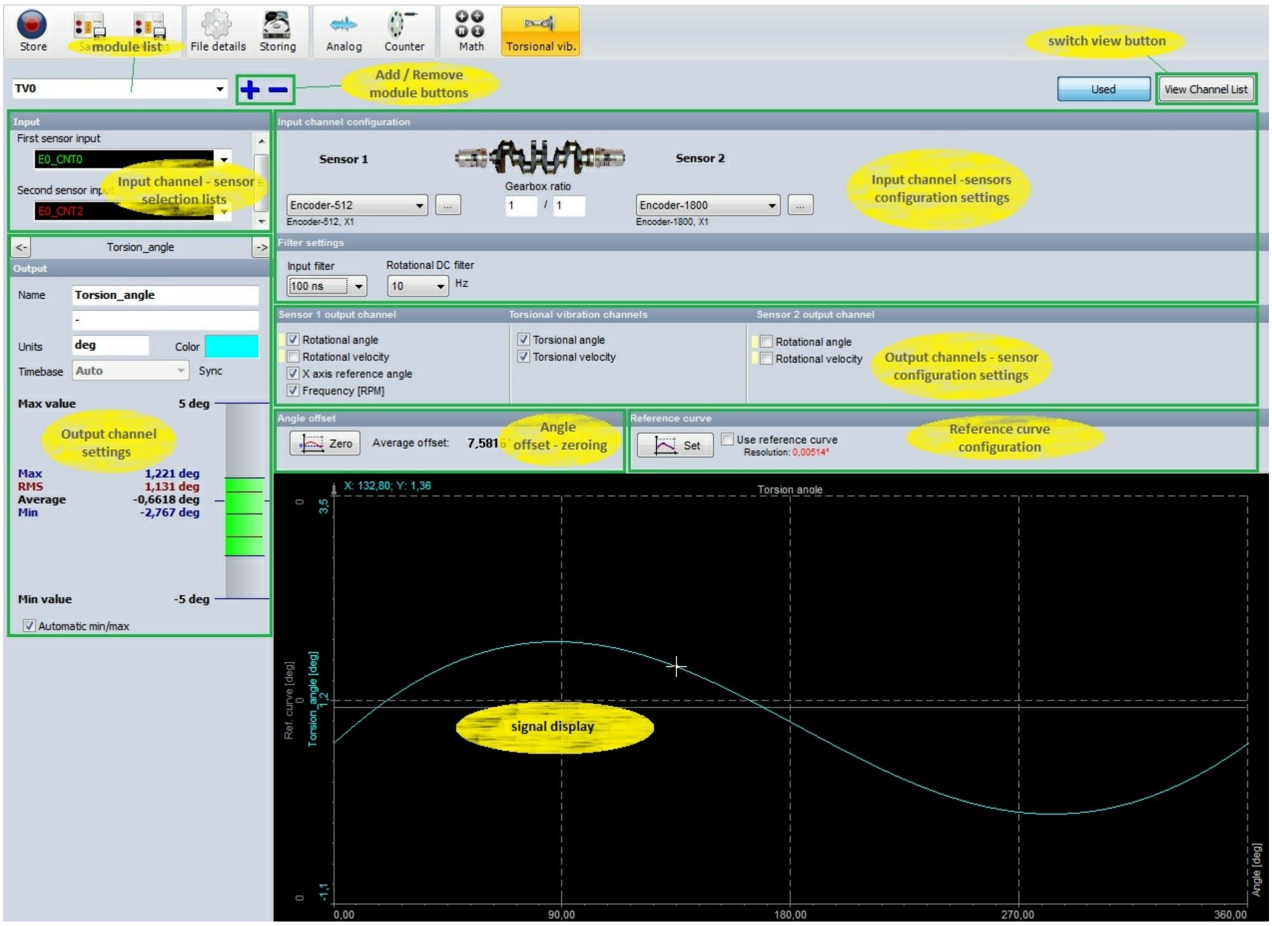
Beside *first* **input** *channel - sensor 1* also *second* **input** *channel - sensor 2* (the *second connected* sensor) from the Second sensor input drop down list must be *select*.



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
- **Angle offset / Zeroing**
- **Reference curve** configuration
- **Input channel configuration** - for *Sensor 1* and *Sensor 2*
- **Sensor output channels** definition for *Sensor 1* and *Sensor 2*
- **Torsional vibration channels** definition

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 by Torsional vibration <u>two</u> (*first* and *second*) sensors connected, on *Torsional vibration setup* screen beside for **Sensor 1** also second *Input channel configuration* for **Sensor 2** section appear.

On these sections we need to define *separate* for Sensor 1 and Sensor 2:

- **Pulses per revolution**
- **Sensor type**

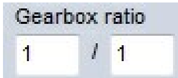
for information about ***Pulses per revolution*** and ***Sensor type*** settings *see* → User Guide → **Rotational vibration setup**

Some configuration settings are *common* for both sensors:

- **Input filter** for the counters
- **Rotational DC filter**

  for information about ***Rotational DC filter*** settings *see* → User Guide → **Rotational vibration setup**
- **Gearbox ratio**

  If we have gearbox in between sensors, we need to enter Gearbox ratio.



NOTE:	*Both sensors must be set to positive counting!*
	Both input filter must be set to same value!

Sensor output channels

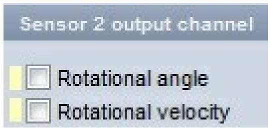
Sensor 1 output channel

The Sensor 1 output channel *is same as for* Rotation vibration *setup* → *see* → **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 with checking box beside caption.

For *each* selection in *Sensor 1* and *Sensor 2* section one *separate* **output channel** is created (*see* below).

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*

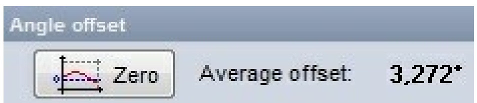
Select desired option with checking box beside caption.

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 detailed information about this ***Output channel*** *setting see* → User Guide → **Sound Level**

Torsional vibration module can have more **output** channels according selection on:

- *Sensor 1 output channel* section

  The output channel is same as for Rotation vibration *setup* → *see* → **Rotational vibration setup**

- *Torsional vibration channels* section

for on Torsional vibration channels section selected:

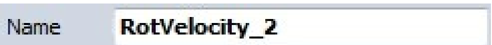
Torsional angle choice →  **output** channel is created

Torsional velocity choice →  **output** channel is created

- *Sensor 2 output channels* section

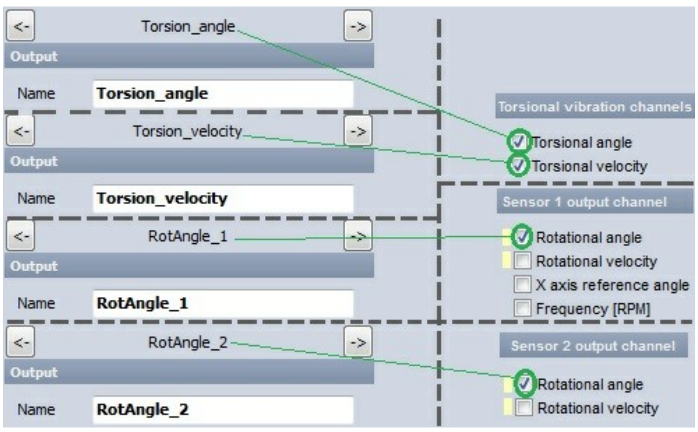
for on Sensor 2 output channels section selected:

Rotational angle choice →  **output** channel is created

Rotational velocity choice →  **output** channel is created

For *each* selection on these sections one *separate* **output channel** is created.

Example:



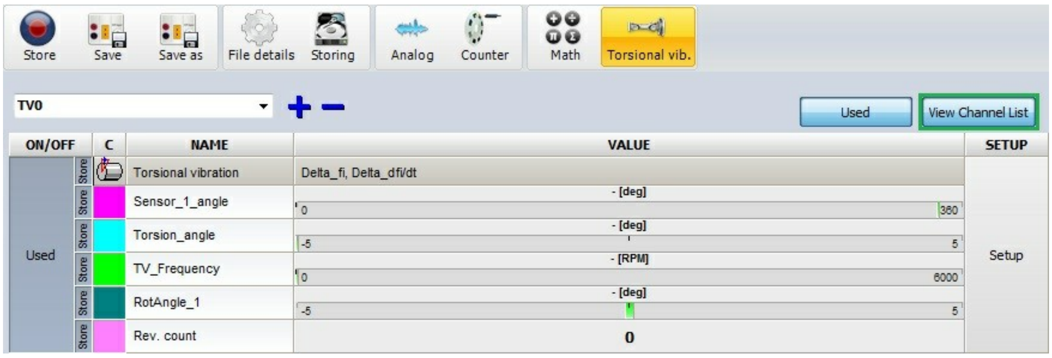
At least one **output** channels must be checked.

Use <- (*Previous*) and -> (*Next*) buttons to navigate between the **output** *channels* in OUTPUT *channel settings* part of screen.

Channel list view

After add *new* **Torsional vibration module** on upper right part of setup screen appear new button to switch between *module setup* and **channel list** view.

When this button is selected on *Torsional vibration module* setup, *Channel list* screen with all *defined output channels* appear:



for information about ***columns*** in ***Torsional vibration channel list*** *see* → User Guide → **Rotational vibration setup**

View Channel List button now change to  and we can call back *module setup* view with selecting this button or with selecting **Setup** button in channel list.

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.*

2.12.3 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.

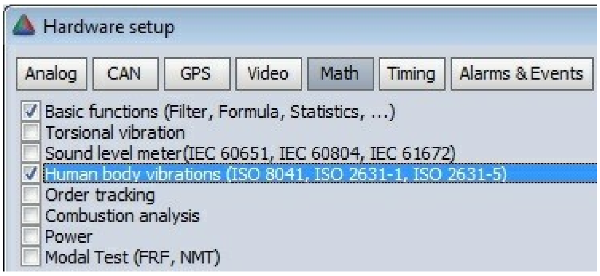
Hand arm measurements is measurement of hand arm where the sensors are installed on special adapters for holding them on the handle or *between fingers*.

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	Dewe 43, Sirius
Required software	SE or higher + HBV option, DSA or EE
Setup sample rate	At least 5 kHz

for hints about ***Human vibration Application*** see → DEWESoft Tutorials

After *required hardware - triaxial accelerometer installation* and *setup* (this procedure is to perform *only* by installation new or changing hardware) according to manufacturer instructions, DEWESoft **Human vibration** measurement is allowed by *selecting* the Human vibration checkbox in System → **Hardware setup** → **Math** tab.

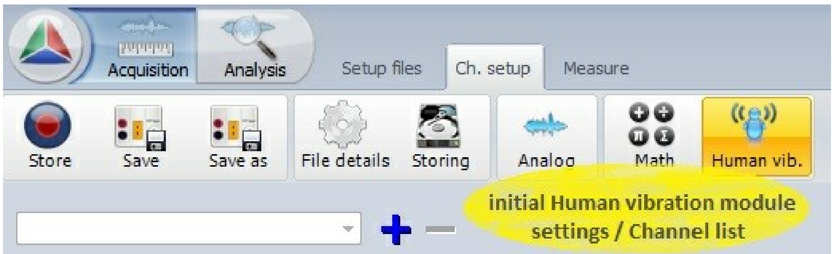


After selecting this option, a tab labeled 'Human vibration' appears in the DEWESoft *Setup* screen (*see* picture below) to *define* and *setup* *Human vibration module*(s).

To use human vibration module, please *select* and *set up* first at least *three* *vibration analog channels* in Analog tab

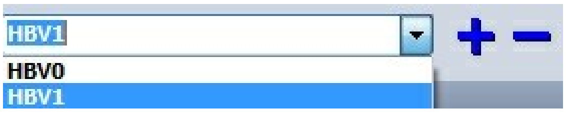
see → User Guide → **Analog in Channel Setup**

Add / New Human vibration module



When we *switch* first time to Human vibration tab, this tab area is empty (also *Human vibration module list*), except there only - **Add** button is active.

After *selecting* the Human vibration tab, press button to add *new* **Human vibration module**. *Several* modules can be used (added) within a session and we will need three *input channels* for *each* module. All new *Human vibration modules* are named automatically with Human vibration n, where n= *running number* and appear on *Human vibration module* drop down list, from which can be selected to set up.



Delete Human vibration module

Each module can be deleted by *selecting* him and then using button.

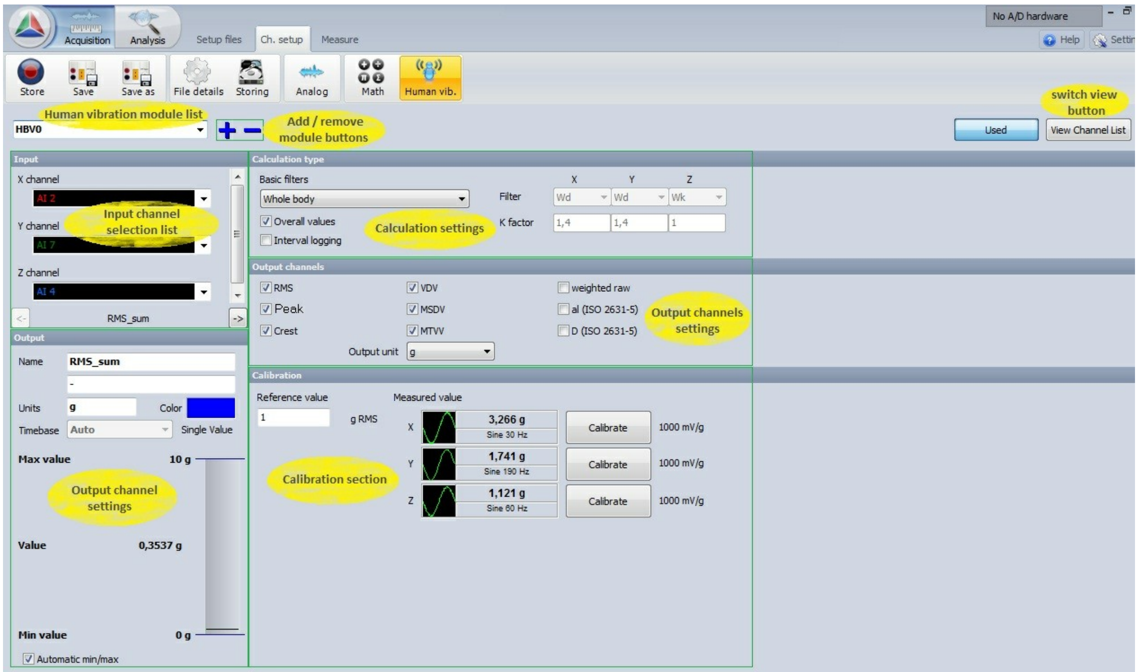
WARNING:	*Be careful - there is no UNDO function for this command. If you delete a Human vibration module, it cannot be recovered.*

Basic procedures of **Human vibration** application setup are:

- **Human vibration module setup** for applied hardware
- **Calibration**

Human vibration module setup

After add *new* **Human vibration module** screen like this will appear:



This screen has the following main sections:

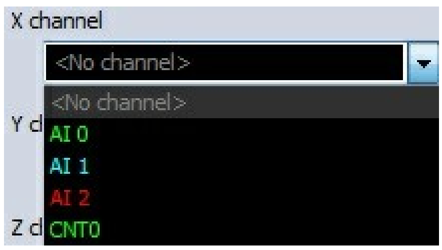
- **Input** channel selection
- **Output** channel settings
- **Calculation type**
- **Output channels** - calculated parameters definition
- triaxial accelerometer **Calibration**

With using **View Channel List** button we can show different view on Human vibration module setup → **Channel list** with all *defined* output channels appear.

Input channel

First of all is to **assign** the **input** channels to triaxial accelerometer sensor with selecting from drop down list 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.



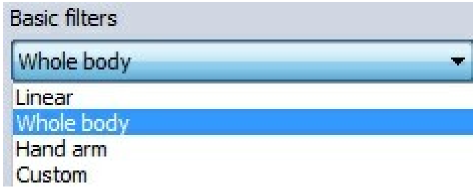
Calculation type

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 human response* to vibrations. Those filters are defined from numerous measurements of natural frequencies of certain parts of human body. Select desired mode from drop down list.



There are two **basic** modes of operation:

- Whole body mode

NOTE: individual Filter and K factor settings can't be chosen, these are *predefined*

- Hand arm mode

NOTE: individual Filter and K factor settings can't be chosen, these are *predefined*

We can also use:

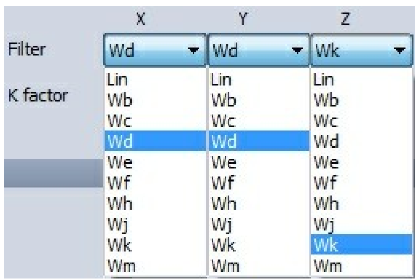
- Linear filter to *check* the measurement chain

NOTE: individual Filter and K factor settings can't be chosen, these are *predefined*

- Custom filters

  For *Custom* filter must be defined:

- **Filter** X, Y and Z value: this individual value can be selected from drop down lists on the *right* side to do *special* measurements. Filters are defined as following:



  Lin - unweighted linear

  Wb - vertical whole body, z axis (older ISO 2631-4)

  Wc - horizontal whole body, x axis

  Wd - horizontal whole body, x or y axis

  We - rotational whole body, all directions

  Wf - motion sickness, z axis

Wh - hand arm, all directions

Wj - vertical head vibration, x axis

Wk - vertical whole body, z axis

Wm - building vibration; all directions.

- With custom filter we need to define also 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 already 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 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.

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*.

- or both

Output channels definition

We can in the middle area of this screen *define* the **output channels**, additional 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.

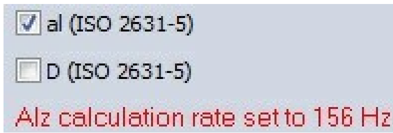
VDV	*fourth power* vibration dose value
MSDV	motion *sickness* dose value
MTVV	the *maximum transient* vibration value, calculated in one second interval
weighted raw	*full speed time* signal weighted with chosen filter We can use those channels for calculation of FFT or CPB spectrum.
aI (ISO 2631-5)	the *lumbar spine response* from excitation measured in all *three* directions
D (ISO 2631-5)	*acceleration dose*, measured from the lumbar spine response

NOTE: aI 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.

aI and D are enough to *evaluate* human vibration *exposure* according to ISO 2631-5.

When one of these calculated values is selected, notice about *AIz calculation rate* is displayed.



<u>**OUTPUT**</u> 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 detailed information about this ***Output channel*** *setting see* → User Guide → **Sound Level**

Human vibration module can have <u>more</u> **output** channels according selection in:

- *Calculation type* section

1. **Basic filters**

For whichever selected filters one Filter and K factor **output** channel *for each axis* individually is created with default *name* (this can be changed in Name field → *see* above):

Filter **output** channels

K factor **output** channels

NOTE: These **output** channels are created in any case, though in Calculated parameters and Output channels sections *none* whatever choice is selected.

2. **Calculated parameters** section

This 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:

Name	PEAK_X	PEAK_Y	PEAK_Z

for selected **Interval logging** in Calculated parameters section:

Name	PEAK_X_t	PEAK_Y_t	PEAK_Z_t

Crest output channels

for selected **Overall values** in Calculated parameters section:

Name	Crest_X	Crest_Y	Crest_Z

for selected **Interval logging** in Calculated parameters section:

Name	Crest_X_t	Crest_Y_t	Crest_Z_t

The RMS, MSDV, VDV and MTVV are calculated also for sum *of all three* axes, therefore for this value also one **output** channel is created.

RMS output channels

for selected **Overall values** in Calculated parameters section:

Name	RMS_sum	RMS_X	RMS_Y	RMS_Z

for selected **Interval logging** in Calculated parameters section:

Name	RMS_sum_t	RMS_X_t	RMS_Y_t	RMS_Z_t

VDV output channels

for selected **Overall values** in Calculated parameters section:

Name	VDV_sum	VDV_X	VDV_Y	VDV_Z

for selected **Interval logging** in Calculated parameters section:

Name	VDV_sum_t	VDV_X_t	VDV_Y_t	VDV_Z_t

MSDV output channels

for selected **Overall values** in Calculated parameters section:

Name	MSDV_sum	MSDV_X	MSDV_Y	MSDV_Z

for selected **Interval logging** in Calculated parameters section:

Name	MSDV_sum_t	MSDV_X_t	MSDV_Y_t	MSDV_Z_t

MTVV output channels

for selected **Overall values** in Calculated parameters section:

Name	MTVV_sum	MTVV_X	MTVV_Y	MTVV_Z

for selected **Interval logging** in Calculated parameters section:

Name	MTVV_sum_t	MTVV_X_t	MTVV_Y_t	MTVV_Z_t

NOTE: For creating **output** channels for above listed *Output channels* option either **Overall values** or **Interval logging** in Calculated parameters section (or both) must be *selected*.

weighted raw output channels

independently of selection in Calculated parameters section:

Name	X_w	Y_w	Z_w

aI (ISO 2631-5) output channels

independently of selection in Calculated parameters section:

Name	aIX	aIY	aIZ

D (ISO 2631-5) output channels

none of choices is selected in Calculated parameters section:

Name	aIX	aIY	aIZ

for selected **Overall values** in Calculated parameters section:

Name	DX	DY	DZ

for selected **Interval logging** in Calculated parameters section:

Name	DX_t	DY_t	DZ_t

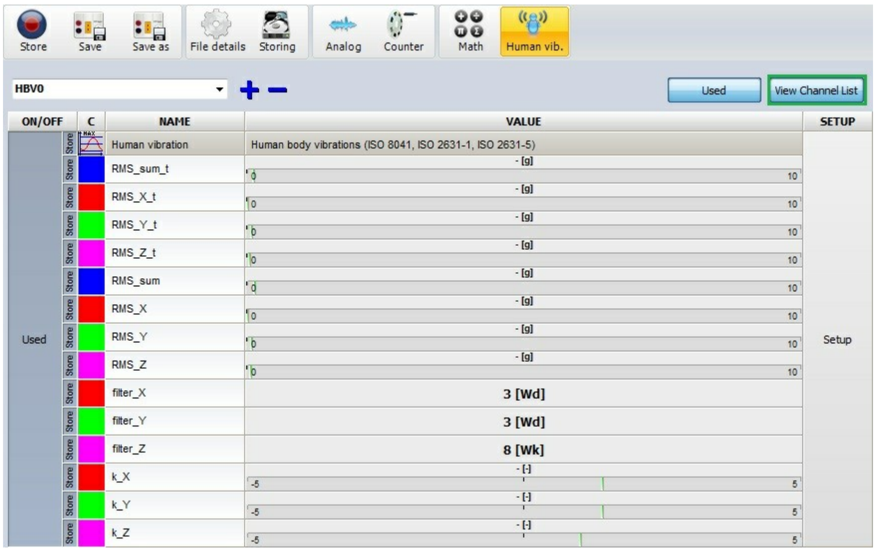
Use <- (Previous) and -> (Next) buttons to navigate between the **output** *channels* in OUTPUT *channel settings* part of screen.

Channel list view

After add *new* **Human vibration module** on upper right part of setup screen appear new button to switch between *module setup* and **channel list** view.

View Channel List

When this button is selected on *Human vibration module* setup, *Channel list* screen with all *defined output channels* appear:



First let's look at the columns that are shown in Human vibration *channel list*, and describe what each is for:

ON/OFF – *activate / deactivate* Human vibration channel - press the Unused button - it will change now to **Used** - to set the desired Human vibration channel ***active*** for *Human vibration 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 – Human vibration channel *name* with all defined *output channels*- just click into the **NAME** field of *output* channel and you can *enter* the new ***name*** as usual text

VALUE – current Human vibration channel *value*

SETUP – return to Human vibration module setup view (*see* above)

for information about ***On/Off***, ***C*** and ***Name*** column *see* → User Guide → **Channel setup**

View Channel List button now change to 

and we can call back *module setup* view with selecting this button or with selecting **Setup** button in 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 tab **Channel setup** for the **Human vibration** *Application module* used *channels*.

for information about ***Calibration analog input channel*** *see* → User Guide → **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/s^2 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 the 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 the Orion 1624 is used, the filter can be set to the *highest* values (because the card itself provides sharp anti aliasing filters) unless we intentionally want to have lower frequency range.

2. Calibrating the triaxial accelerometer with calibrator

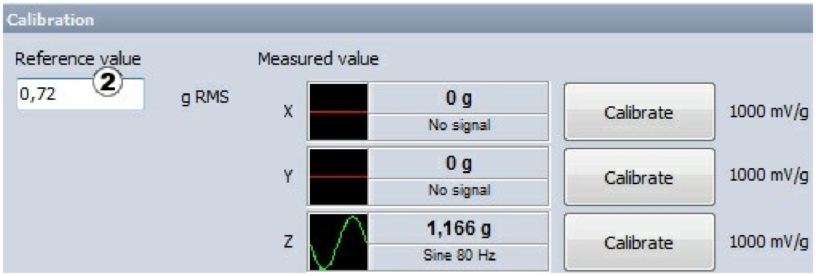
If we want to perform calibration with the calibrator, we can perform it here - in Human vibration module itself.

First enter the Reference value ② of vibration. This can be read from the **calibrator**.

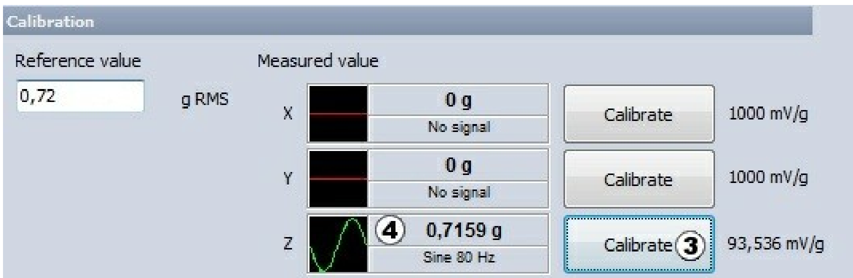
Example: Usually the calibration levels are 10 m/s^2 peak, which is 7,07 m/s^2 RMS or 0,72 g. We need to enter this value in Reference value field.

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 ③ 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 0,7197 g) ④.

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 Tuner Utility.*

2.12.4 Order tracking

DEWESoft **order tracking** method is used to *extract* the harmonic components related to rotational frequency of the machine. With order *extraction* we can see *specific* harmonic component which relates to certain machine ***fault***.

The *machine 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 and machine response function, which relates 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
- *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.

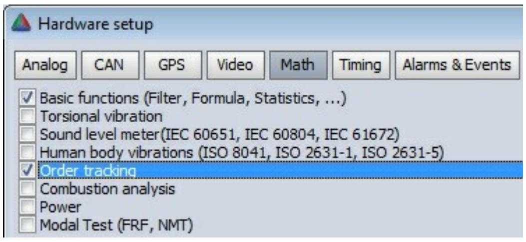
To measure *Order tracking*, we need at least one *accelerometer* for measuring the vibration and *encoder* for measuring RPM.

Required hardware	Dewe 43, Sirius
Required software	SE or higher + ORDTR option, DSA or EE
Setup sample rate	At least 10 kHz

for hints about ***Order tracking Application*** see → DEWESoft Tutorials

The Counter hardware can be *activating* on the **DEWESoft** System menu → Hardware setup... → **Analog** tab by set the required *counter channels* (CNT chnls column) in Device information list.

After activating the Counter hardware, DEWESoft **Order tracking** measurement is allowed by *selecting* the Order tracking checkbox in System → **Hardware setup** → **Math** tab.

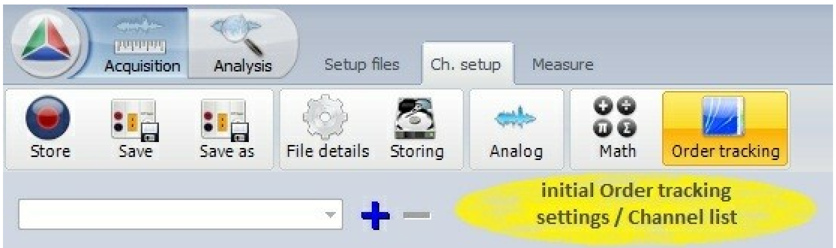


After selecting this option, a tab labeled 'Order tracking' appears in the DEWESoft *Setup* screen (*see* picture below) to *define* and *setup* *Order tracking module*(s).

To use Order tracking module, please *select* and *set up* first *vibration analog channels* in Analog tab

see → User Guide → **Analog in Channel Setup**

Add / New Order tracking module



When we *switch* first time to Order tracking tab, this tab area is empty (also *Order tracking module list*), except there only

 - **Add** button is active.

After *selecting* the Order tracking tab, press  button to add *new* **Order tracking module**. Only one module can be used within a session.

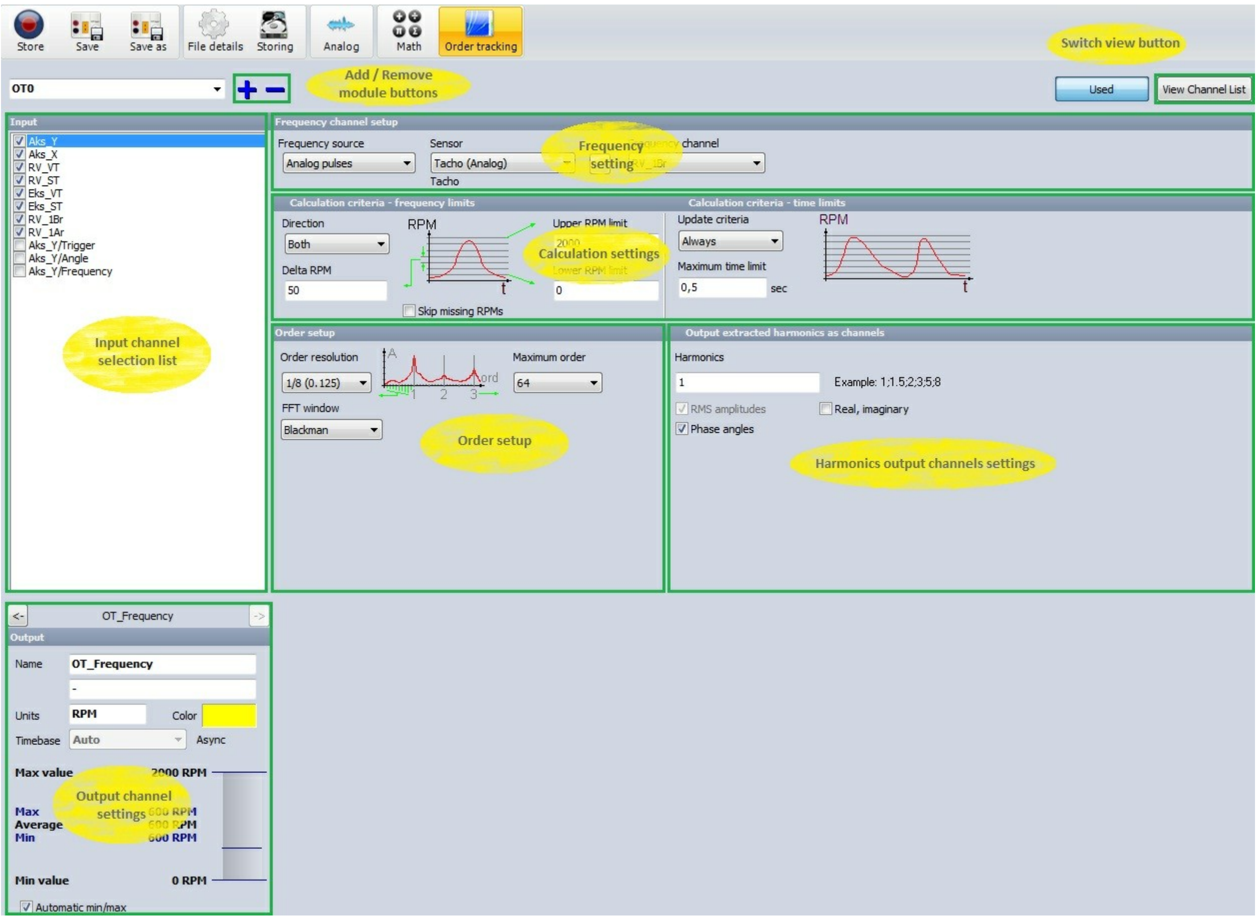
Delete Order tracking module

Module can be deleted by *selecting* him and then using  - minus button.

WARNING:	*Be careful - there is no UNDO function for this command. If you delete a Order tracking module, it cannot be recovered.*

Setup

When an order tracking is added, we get the following pictures with the **settings** for *order tracking*.



This screen has the following main sections:

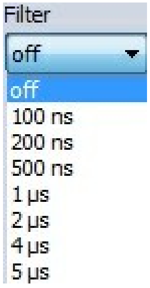
- **Input** *channel* selection
- **Output** *channel* settings
- **Frequency channel settings**
- **Order setup**
- **Calculation criteria** for *frequency* and *time limits*
- **Output extracted harmonics as channels**

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 **encoder** or **geartooth**. Then we select the *sensor* itself from Sensor drop down list and the *source* where this sensor is *connected* to.

  We can select any counter *sensor*.



If we choose the **encoder**, then we need to connect the encoder to the *counter input*.
Please note that only Orion counters can be used in this case.

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).
Invert options *inverts* the encoder direction if it counts in *negative* since order tracking supports only counting in positive direction.

- Second option is to select the Analog pulses as the *source*. In this case the sensor should be connected to *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 → after pressing this button we get *Angle sensor setup* window. → *see* → **Angle sensor**

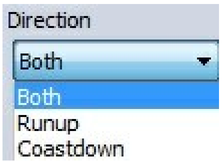
  When the ... button (on right of Sensor field) is selected, we get *Counter sensor editor* window where *angle sensor* should be defined. → *see* → **Counter sensor editor**

- Third option is the RPM channel. In this case the frequency source comes *directly* from *analog channel* which measures the *frequency* (like *DAQP-FREQ-A*).

- 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 Runup, Coastdown 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.

Calculation criteria - time limits

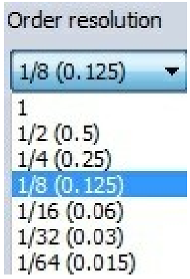
In this section we define the *time* Update criteria from drop down list for *taking new samples*. We can define to acquire the data Only at first time Runup / Coastdown or Always when *passing* through *some frequencies*.

We can also define the Maximum time limit. If the frequency doesn't *change* within specific time, we will *still get another* point for harmonics and 3D history.

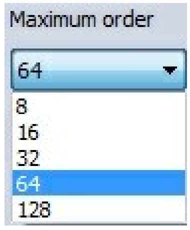
Order setup

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 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 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.

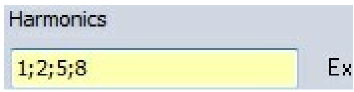
We set (select from drop down list) also the FFT window type for *calculation* of the history FFT.

for hints about recommended using ***Window type*** → *see* →
Reference Guide → **Theory of frequency analysis**

Output extracted harmonics as channels

Order extraction setup

We will see all the orders in the history FFT, but if we want to draw the Bode or Nyquist *runup 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 *reference point* (*tacho* or *encoder*), we can get also the *phase*, *real* and *imaginary* component of each harmonic.

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 detailed information about this ***Output channel*** *setting see* → User Guide → **Sound Level**

Order tracking can have more **output** channels according selection in *Output extracted harmonics as channels*.

2.12.5 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 hints about ***Combustion analysis*** see → DEWESoft Tutorials

To work with DEWESoft Combustion analyzer we need to do:

Activate hardware — or modify hardware settings:

- *analog* and *counter channels* on *Hardware setup* window with **Analog** setup
- **Counter setup**

Combustion module

- **Enable module**
- **Add module**
- **Delete module**

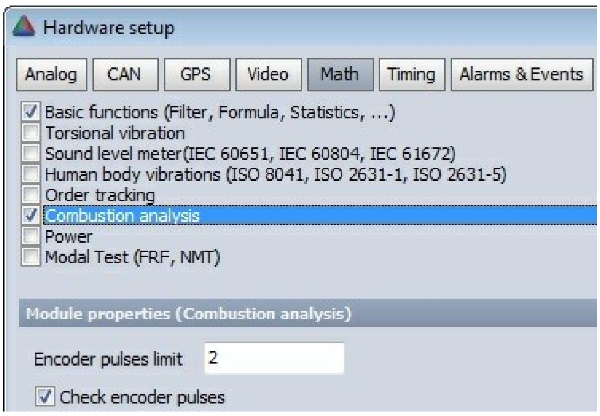
Combustion analysis setup — to select *CA type*: **Engine**, **Angle sensor**, **Calculations**, **Heat release**, **Knock detection** and then define all settings

Activate hardware

To **activate** your hardware or *modify* hardware settings, please select DEWESoft USB from the drop down list in the System menu → Hardware setup... → **Analog** tab.

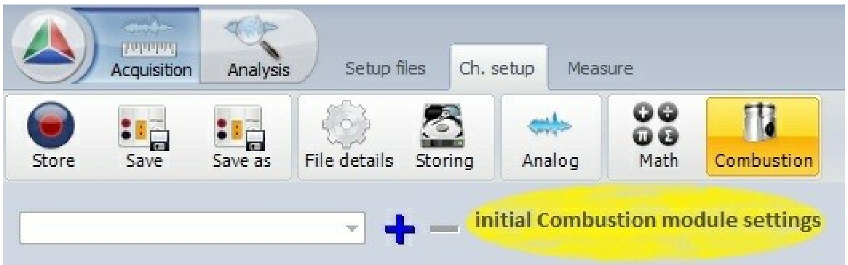
Enable Combustion analysis module

After *hardware setting* and channels *setup* DEWESoft **Combustion analysis module** measurement is allowed by *selecting* the Combustion analysis module checkbox in System menu → Hardware setup... → **Math** tab.



After selecting this options, a tab labeled Combustion analysis appears in the DEWESoft *Setup* screen (*see* picture in *Add Combustion analysis module* chapter below) to *define* and *setup* Combustion analysis module.

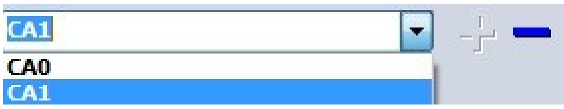
Add / New Combustion analysis module



When we *switch* first time to Combustion analysis tab, this tab area is empty (also *Combustion analysis module list*), except there only - **Add** button is active.

After *selecting* the Combustion analysis tab press button to add *new* **Combustion analysis module**. *Only one* module can be used (added) within a session.

New *Combustion analysis module* is named automatically with 'Combustion analysis'.



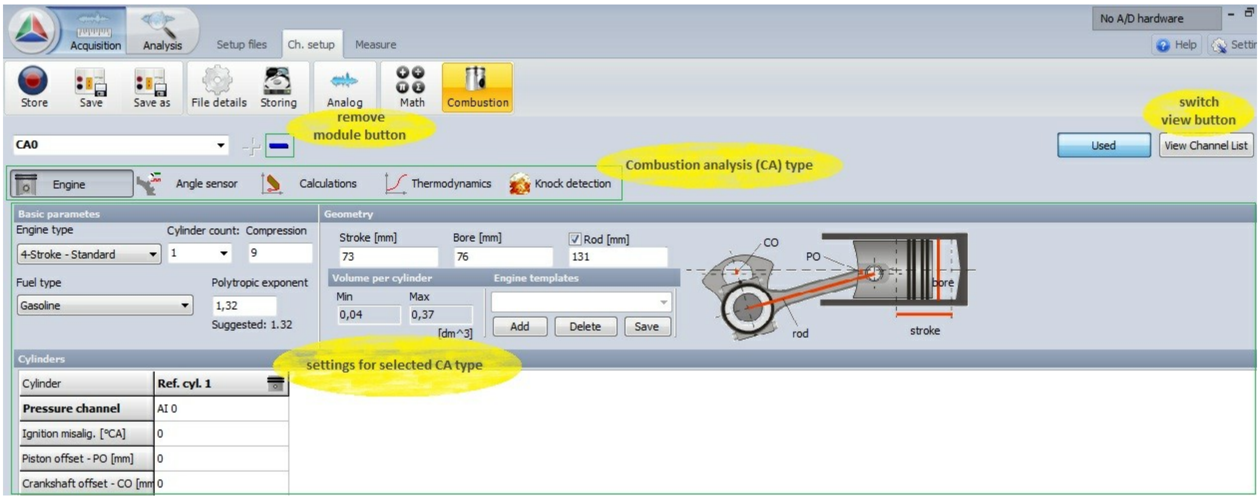
Delete Combustion analysis module

Combustion analysis module can be deleted by *selecting* him and then using - minus button.

WARNING:	*Be careful - there is no UNDO function for this command. If you delete a Combustion analysis module, it cannot be recovered.*

Combustion analysis setup

After add new *Combustion analysis module* initial (Engine) *setup* screen is displayed:



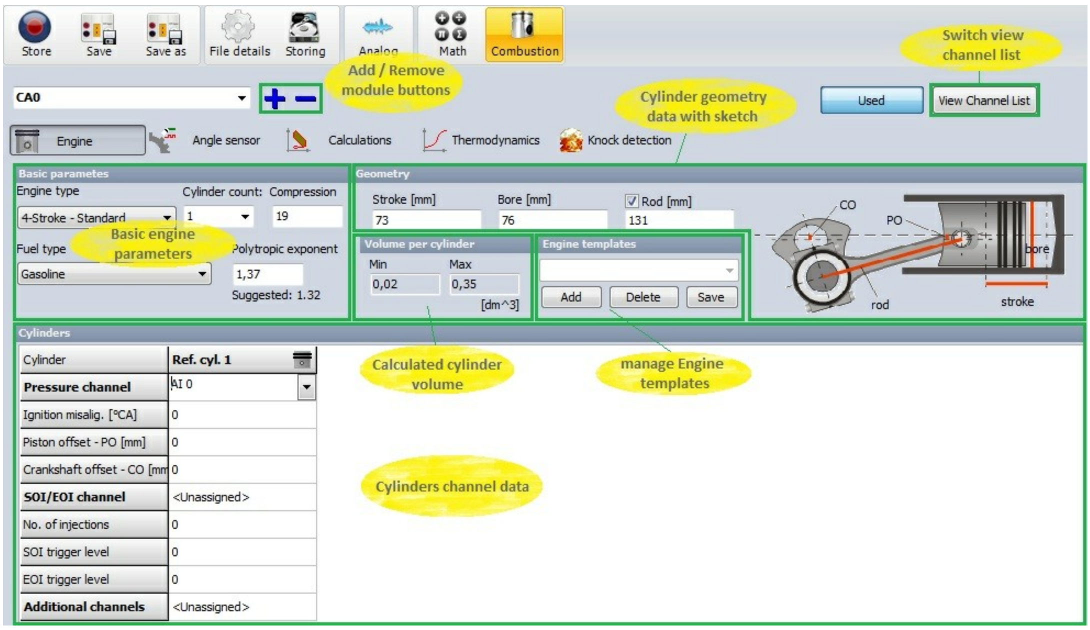
We can set **Combustion analysis** (**CA**) in any way we want, but usually the easiest way would be select CA type and then define all settings of particular type:

- **Engine**
- **Angle sensor**
- **Calculations**
- **Heat release**
- **Knock detection**

When the **View Channel List** button is selected, **Channel list** is displayed.

1. Engine setup

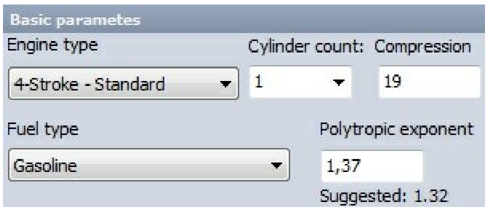
After selecting **Engine** type CA or after *new* CA module is added following `setup` screen is displayed:



Settings and entered *values* for **Engine** CA type are divided on following sections:

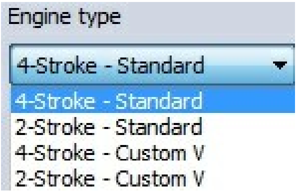
- **Basic parameters**
- **Geometry**
- **Calculated volume**
- **Engine templates**
- **Cylinders**

Basics parameters



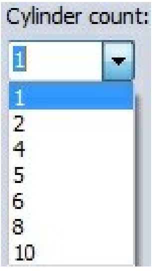
From drop down list can be selected:

Engine type

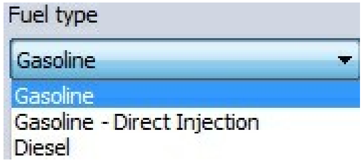


Engine type can be 4-stroke (for example car) or 2-stroke (motor bike).

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:

$$Rc = (Vd + Vc) / Vc$$

Vd = swept volume

Vc = 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 value should be enter in corresponding field.

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 keyed in

Delete ... deletes currently selected template

Save ... saves changes to currently selected template

Cylinders

Cylinders

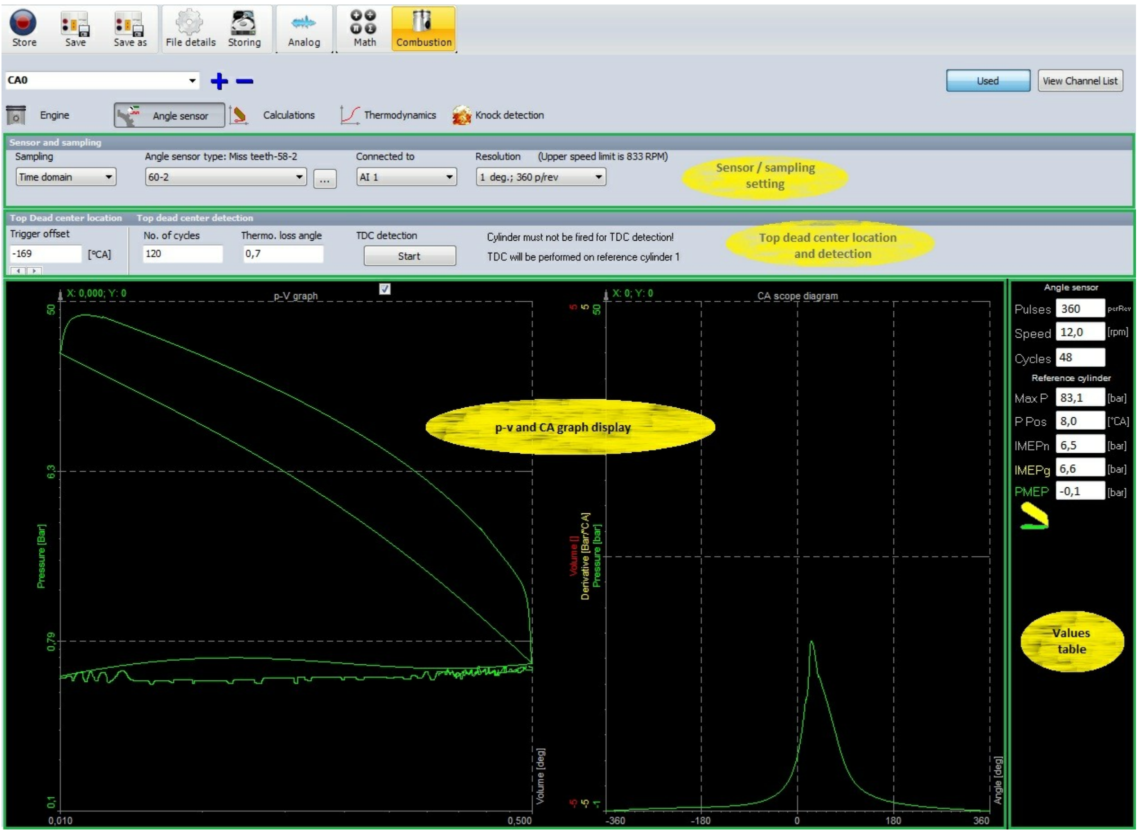
Cylinder	Ref. cyl. 1	Cyl. 2
Pressure channel	AI 0	<Unassigned>
Ignition misalig. [°CA]	0	0
Piston offset - PO [mm]	0	0
Crankshaft offset - CO [mm	0	0
SOI/EOI channel	<Unassigned>	<Unassigned>
No. of injections	0	0
SOI trigger level	0	0
EOI trigger level	0	0
Additional channels	<Unassigned>	<Unassigned>

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.
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).
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

Top dead center location

Top dead center detection

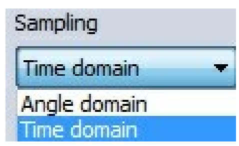
Display graphs

Values table

Sensor and sampling

From drop down list can be selected:

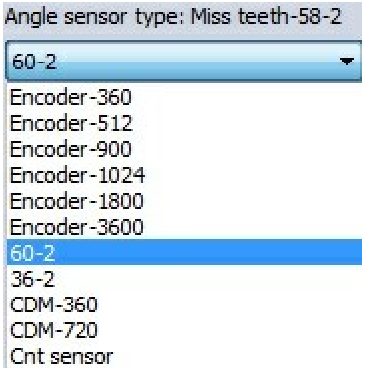
Sampling type



Sampling type is the *most important* selection for behavior of combustion analysis.

Internal clock will acquire the data in *time domain* and *recalculate to angle* domain. It will be slower, but we are able to do all time based calculations.

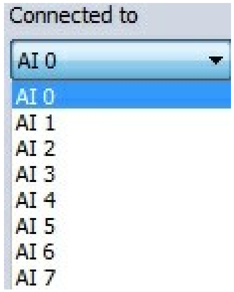
Angle sensor type



Angle sensor depends what sensor we connected.

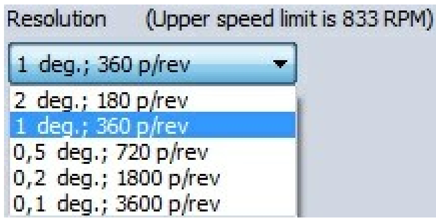
When button is selected, the **Counter sensor editor** window appears to define counter sensor.

Connected to



Here we define *physical connection* of the sensor. If use external clock with CA-CPU, it will show the CA-CPU *inputs*. If we use internal clock, it will offer the counter *inputs* for *encoder* and CDM sensors and analog *inputs* for *geartooth with double* or *missing teeth*.

Resolution



Resolution will define the *number of points* per *one engine revolution*. Higher number will give higher accuracy, but will also bring more calculation load.

When Setup button is selected, the **Angle sensor setup** window appears to define angle sensor.

for information about ***Angle sensor setup*** *see* → User Guide → **Angle sensor**

Top dead center location

Trigger offset

Trigger offset is the offset *from* the trigger *to top dead center* of the *reference* cylinder.

Value in this field we can decrease whit button and increase with button.

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.

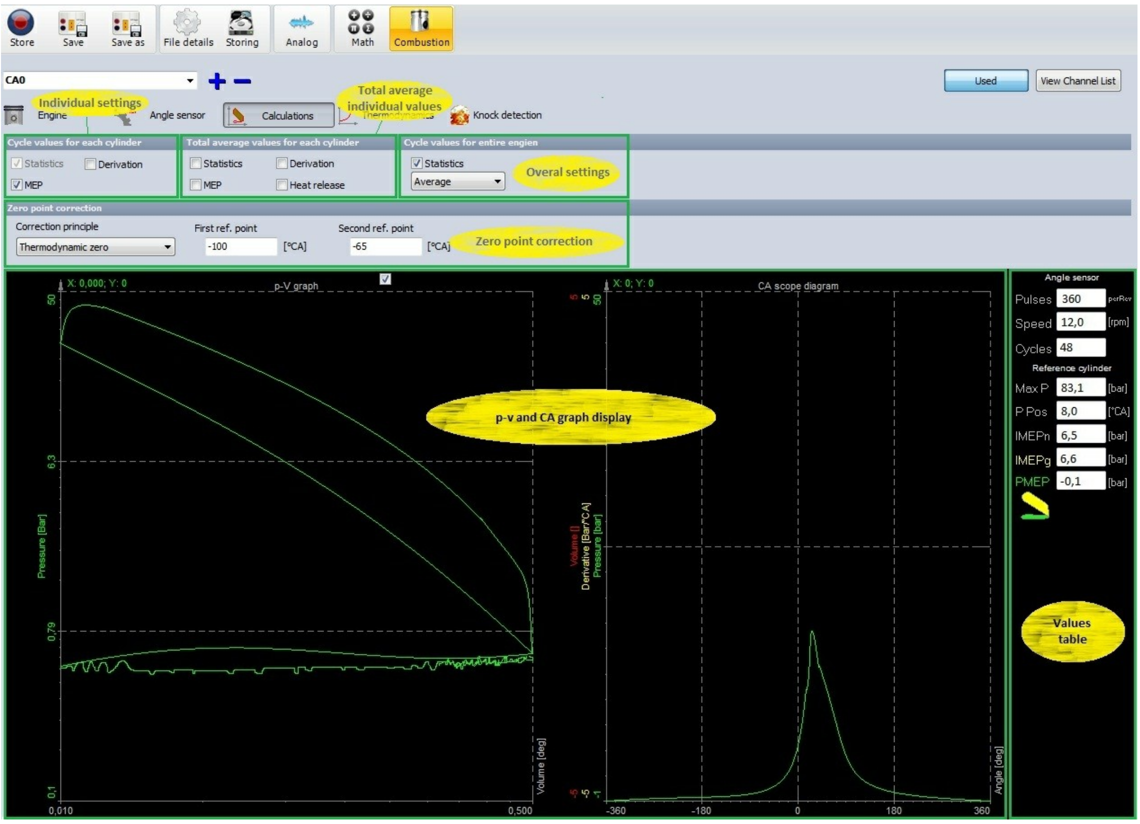
TDC detection

When Start button is selected, table with *values* appear instead on right side table:

Now the Start button change 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

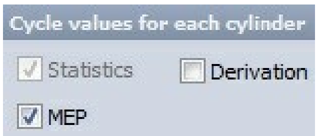
After selecting **Calculations** type CA following *setup* screen is displayed:



Settings and entered *values* for this **Calculations** CA type are divided on sections:

- **Individual calculations**
- **Overall calculations**
- **Zero point detection**
- **Display graphs**
- **Values table**

Individual calculations



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.

MEP option adds *net*, *gross* and *pumping* MEP values.

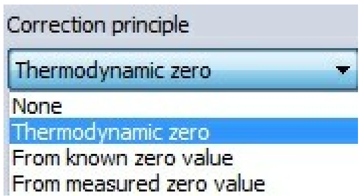
Overall calculations

With overall calculations the Sum for all cylinders are added. When we select Average, we will get an *average pressure* in *all* cylinders for *entire* stroke.

When Aligned is checked, we will get *pressures* for *each* cylinder *at the top dead center* to show the *envelope* of pressure *peaks* (for example at cold start).

Zero point detection

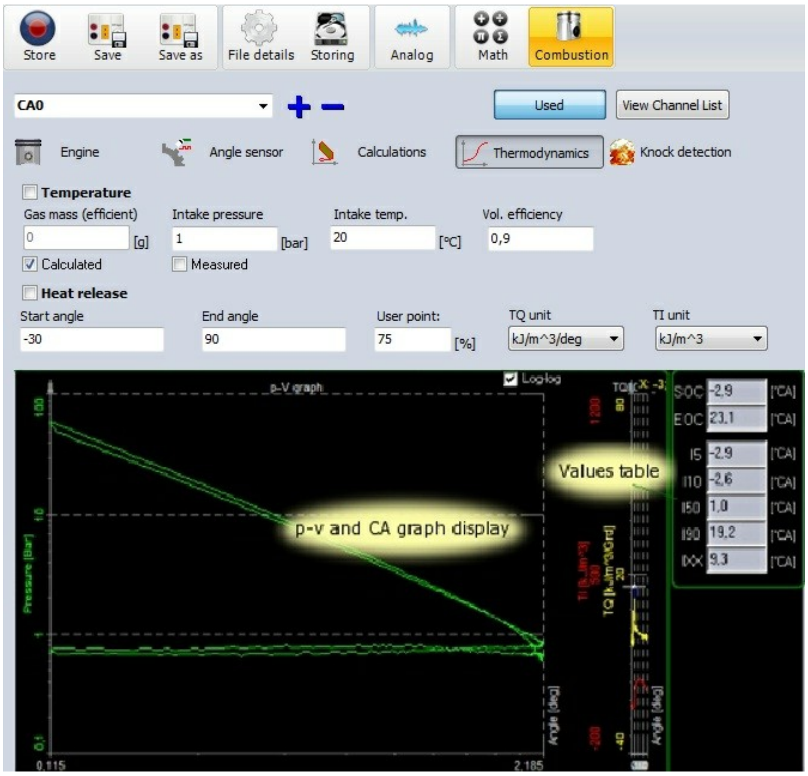
Correction principle



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, start and end angle for this calculation *needs to fit* the *real compression* (without ignition).

4. **Heat release** setup

After selecting **Heat release** type CA following *setup* screen is displayed:



Settings and entered *values* for this **Heat release** CA type are divided on sections:

Settings

Display graphs

Values table

Settings

Calculate heat release

This option switches *on* or *off* the calculation of the heat release.

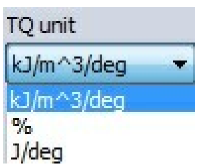
Starting and ending angle

Heat release is very intensive calculation and we can *limit the range* of calculation *only around top dead center*.

User point

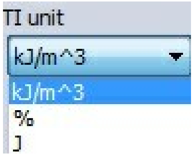
Heat release *creates several* output channels with *angle* values for *certain amplitude* values - 5, 10, 50 and 90% (called I5, I10, I50 and I90). Additionally we can define *one* user point called IXX where the XX is the *percentage* value of the heat release defined here.

TQUnit



We can either have the *physical* unit for the heat release or have it expressed in *percentage*.

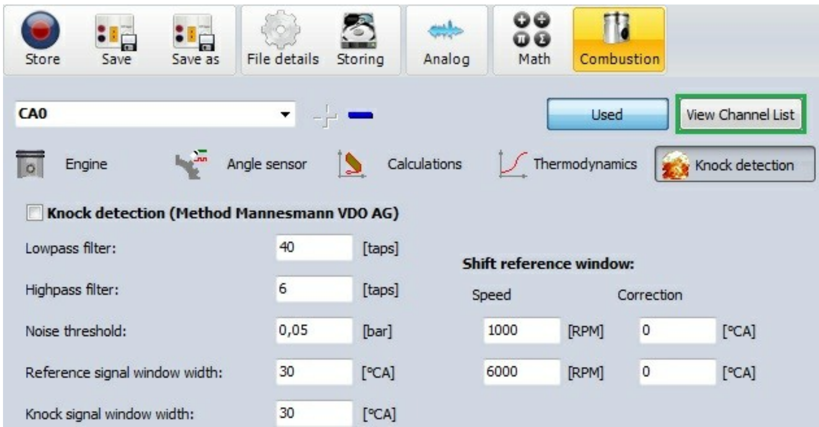
TIUnit



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:



Settings and entered *values* for this **Knock detection** CA type are:

Knock detection (Method Mannesmann VDO AG)

Knock detection is the principle for finding the *amount of knocking* due to late ignition.

Lowpass filter and Highpass filter

In these fields we define *Number of taps* (according to angle resolution) of 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.

Reference and knock signal window width

The *width of the window* on left and right side of the pressure signal.

Shift reference window

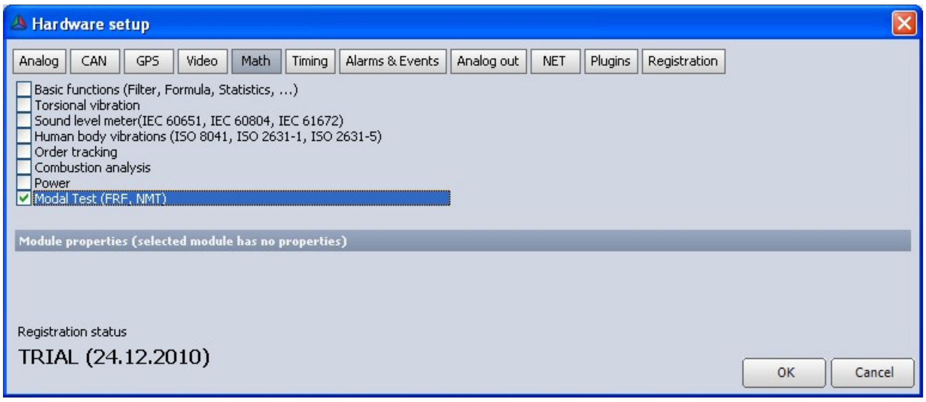
Is two point linear scaling offset definition how much the calculation window is *shifted* for different *speeds*.

2.12.6 Frequency response analysis

Frequency response analysis is a process to determine the transfer characteristic of system. It is most often used to find the natural frequencies of mechanical structure using hammer or shaker excitations.

Required hardware	Dewe43, Sirius
Required software	DEWESoft DSA or SE, PROF and FRF option

Activate hardware



Visualize transfer function calculation

How to *visualize* transfer function calculation with FRF Geometry visual control *see* **Geometry display**

MODAL TEST (intro)

Modal test is mathematical module that allows us to calculate **transfer function** (H) between *two signals*.

$$H = Output\,/\,Input$$

Input and **Output** signals are usually *mechanical* or *electrical* nature. Mechanical means *exciting structure* with hammer or shaker and measure response with *accelerometer*. Electrical can be for example input and output *voltage* of amplifier in order to get transfer function of *amplifier*.

Transfer function (H) is a *relation between two* signals in a complex form (real and imaginary vs frequency).

Two standard approaches for calculation of transfer function (H) are supported in DEWESoft:

a) **FRF calculation** (*triggered*, *free run*)

b) **NMT calculation** (*step sweep*, *manual*)

INPUTS

Excitation channels and response channels

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 acceleration sensor.

Excitation channels

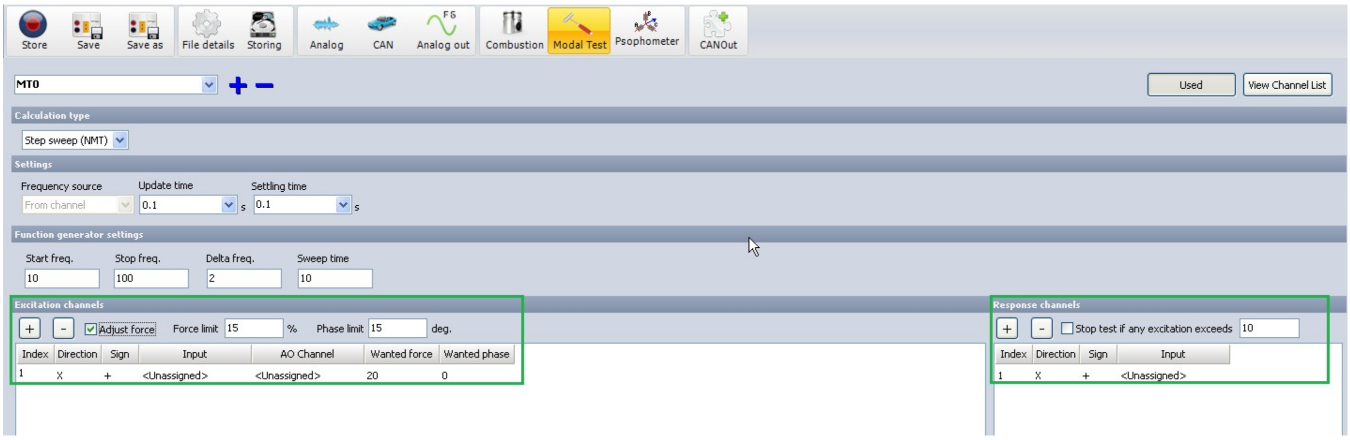
AO channel - AO channel which is used for driving shaker.

Wanted force - peak level at which the shakers will be held during measurement.

Force limit - error in percent from wanted force which is allowed during measurement.

Wanted phase - can be zero or 180 degrees. It depends on how our shakers are positioned and which mode we want to excite.

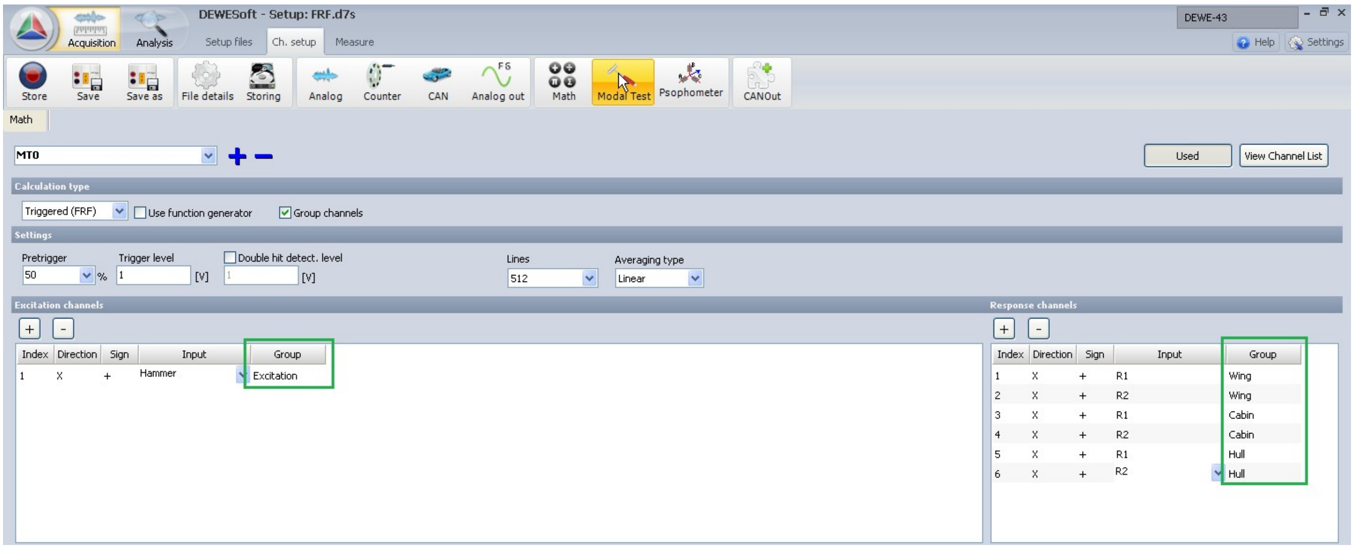
Phase limit - error in degrees from wanted phase which is allowed during measurement.



CALCULATION

FRF

Transfer function is calculated via standard **FRF** algorithm. It is possible to calculate *all excitation/response* pair at *once* or in *groups* (use group check box). When using group calculation you can have rowing hammer/accelerometer method. You can also move groups of sensors.

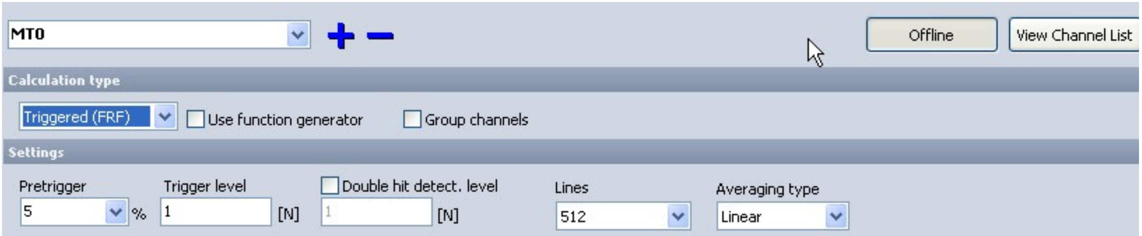


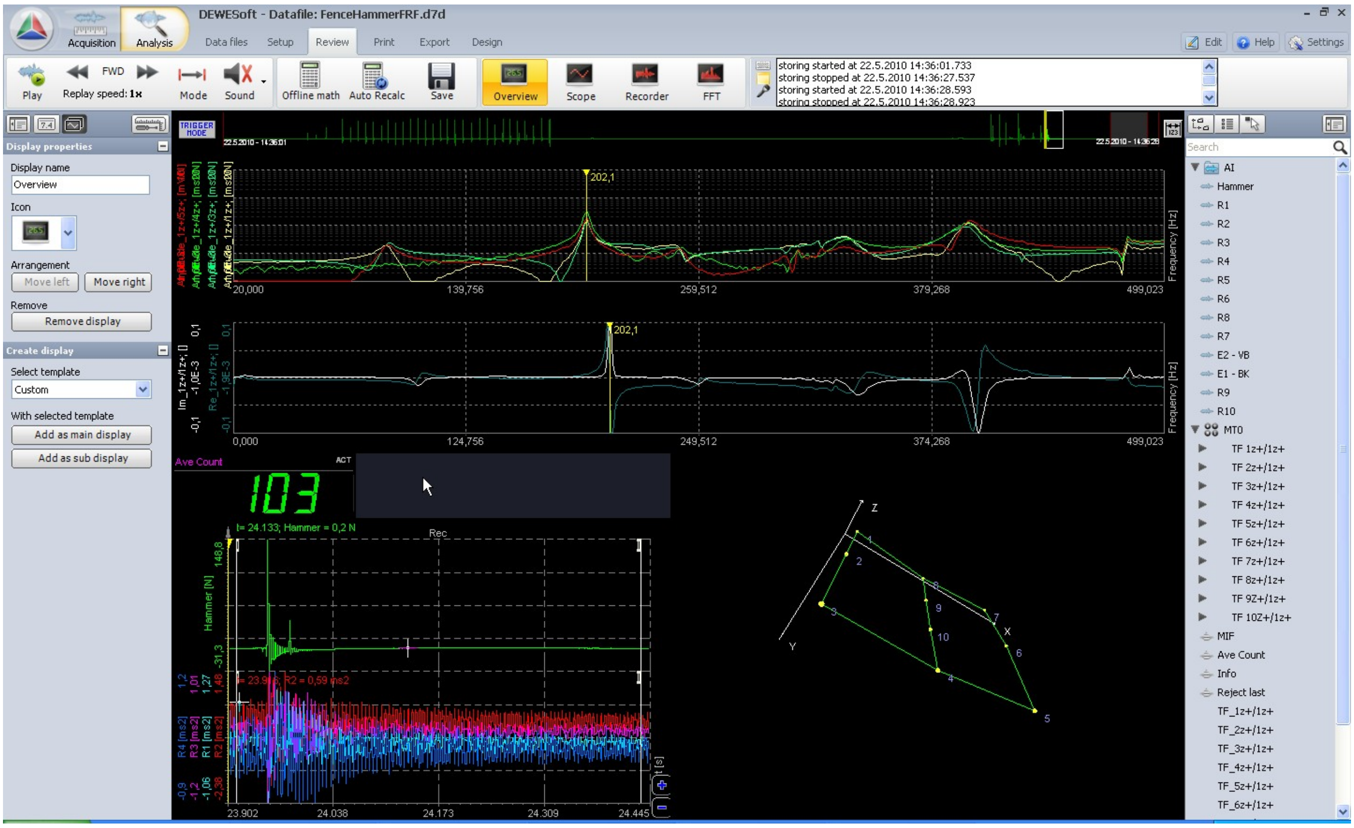
Data collection can be done in two ways:

a) Triggered - blocks of data are taken when trigger level is *exceeded*.

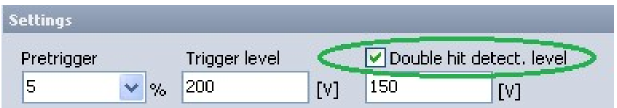
Pretrigger - how much data will be taken *before trigger* is reached (in percent of block size). If this is set to zero we will probably miss some data on the beginning.

Trigger level - trigger level in *units* of *input* channel.





Double hit detection level - 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 that on excitation spectrum. Then you can reject this measurement or continue.

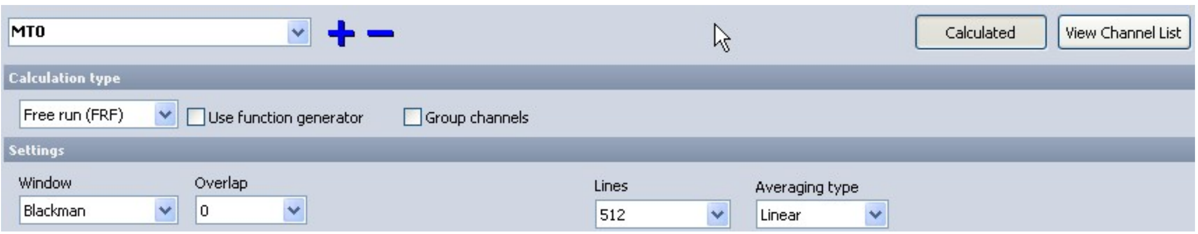


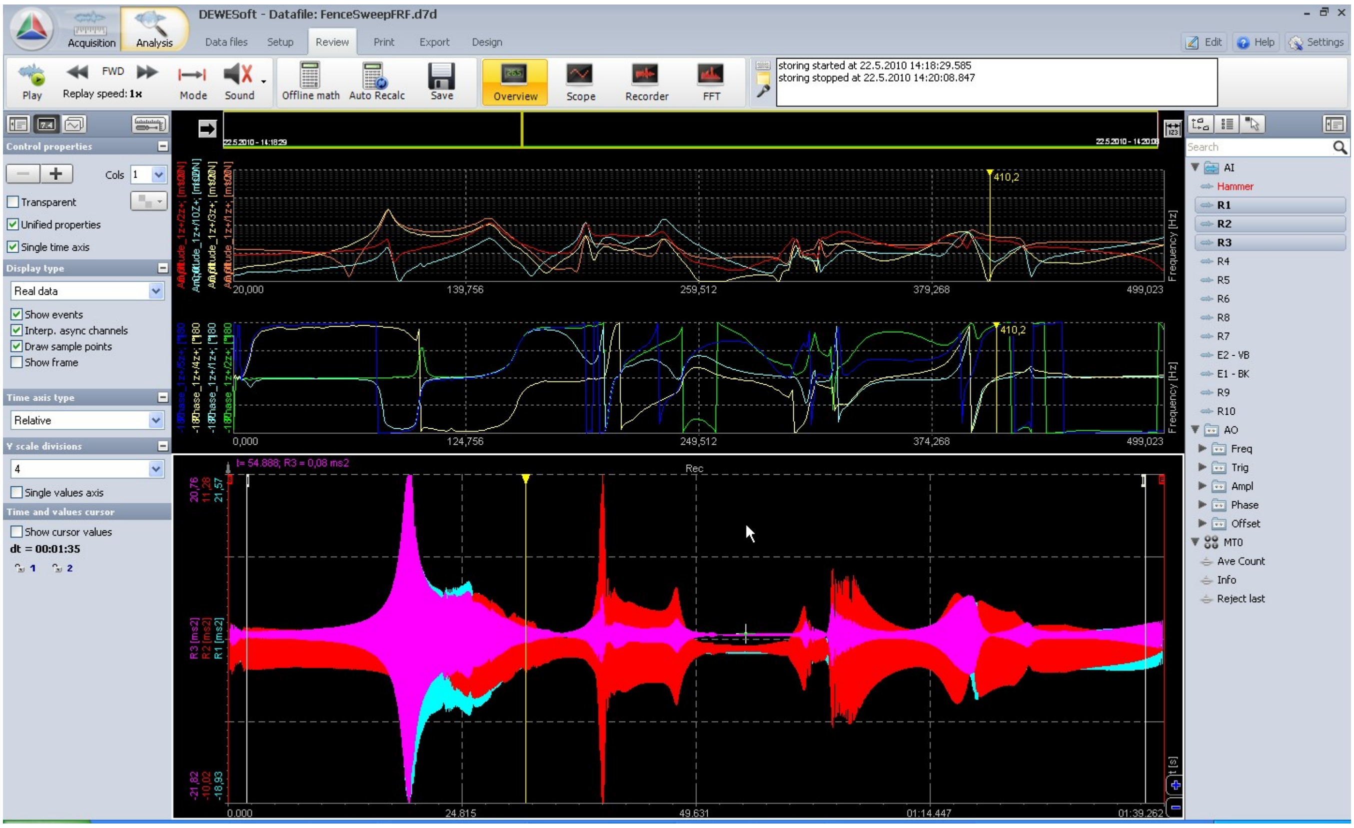
Reject Last - in triggered mode it is also possible to *reject last block* of data if we made a false measurement. This can be done via control button in the measurement. If we reject a block then the average count decreases by `1`.

b) Free run - blocks of data are taken *continuously* (one following another with overlap if specified).

Window - standard DEWESoft window functions for FFT leakage *reduction*.

Overlap - Block of data can also be *overlapped*.





Common settings

Lines - number of lines for FFT (block size equals two times lines).

Averaging type - Linear, peak hold, exponential (you can also define slope falloff in percent).

NMT

Normal mode technique for transfer function calculation where you can find frequency and phase of transfer function more exactly. That is because of two things:

- in manual mode you can really tune to the exact frequency of the resonance.
- in step sweep mode you go slowly up with the frequencies so that ringing from one frequency line before is interfering with the current frequency as little as possible.

a) Step sweep - in step sweep we excite the structure with sine wave calculate response, wait for some time and than continue to next frequency.

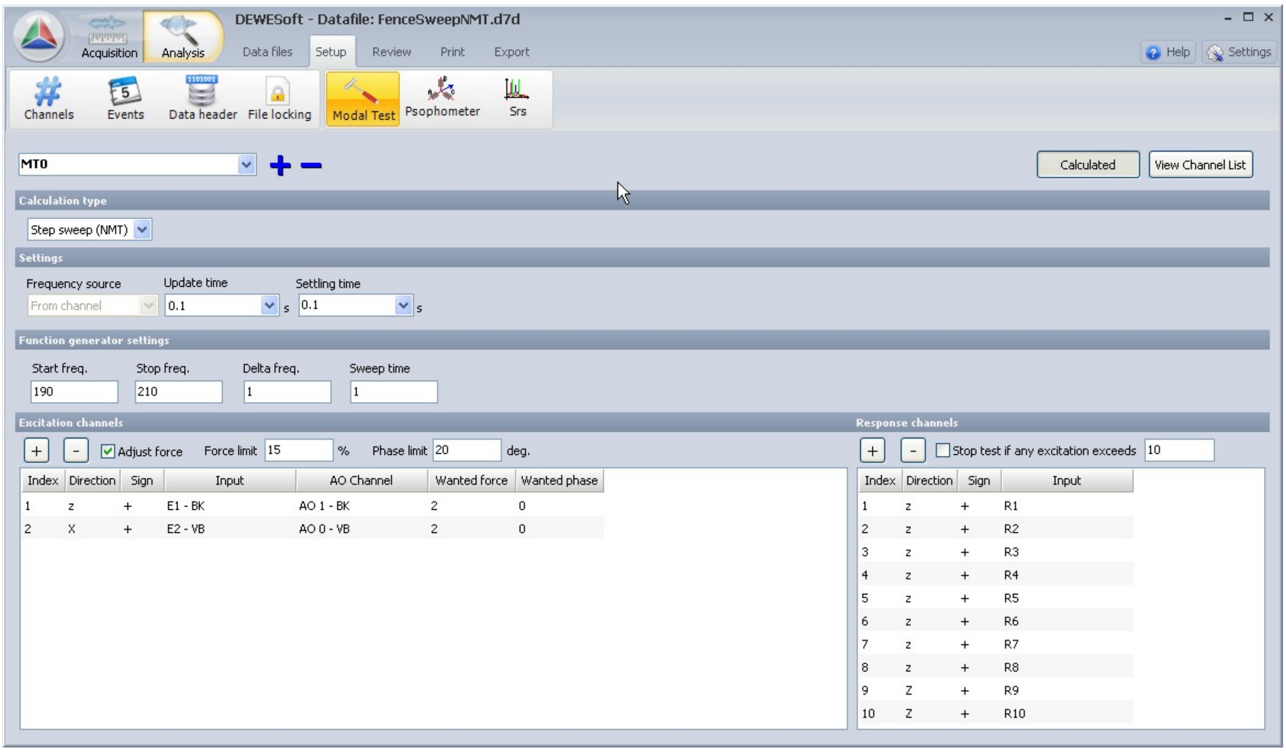
Settling time - NMT waits this amount of time before taking any new data (so the signal settles).

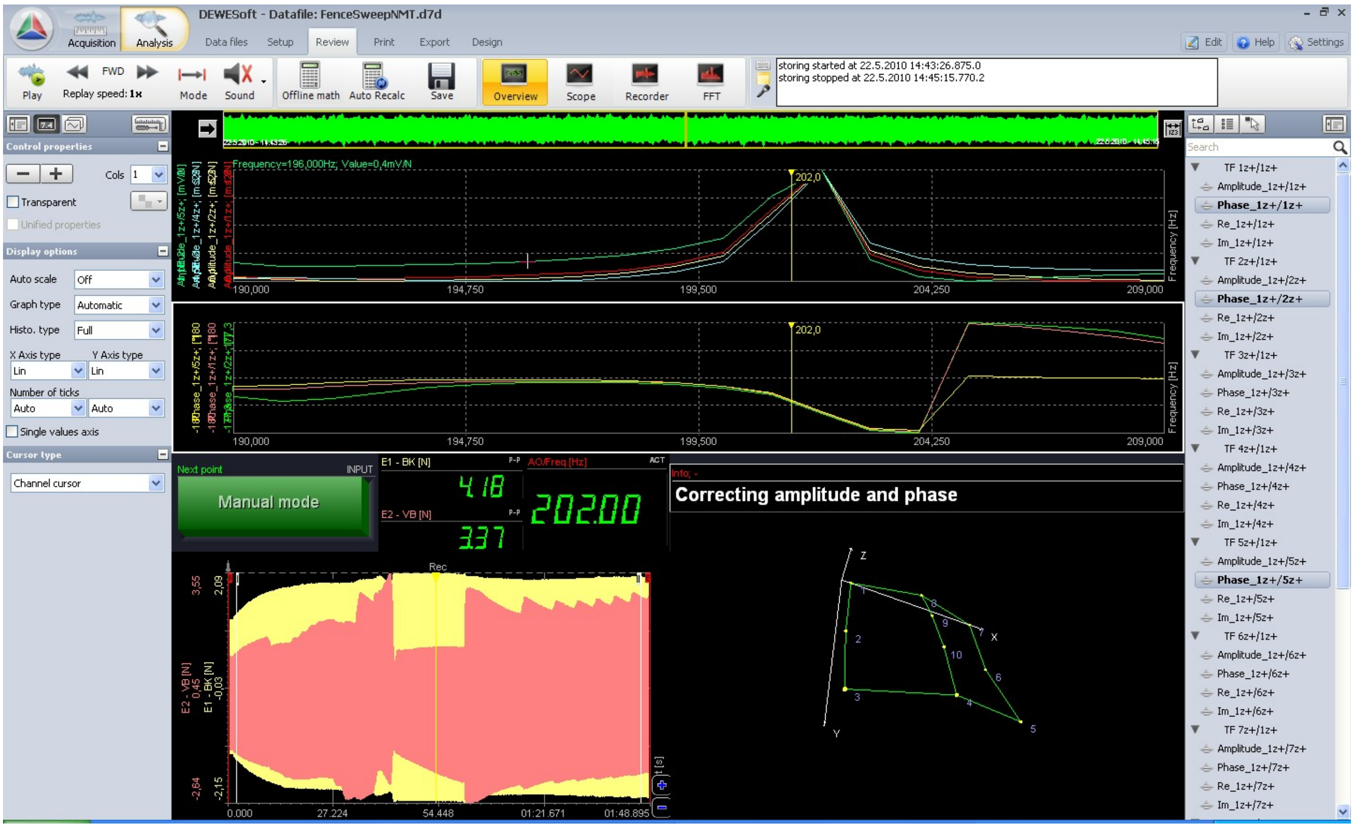
Start frequency - start frequency for sweep.

Stop frequency - stop frequency for sweep.

Delta frequency - frequency sweep step.

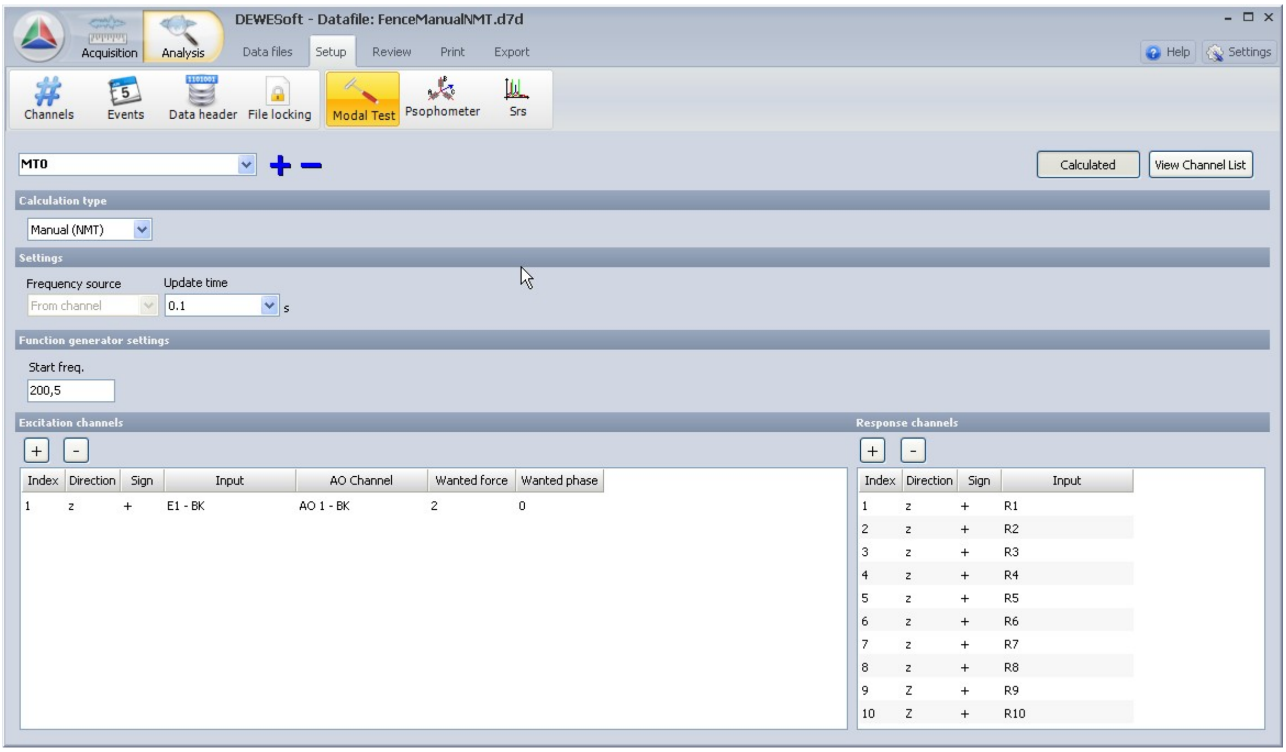
Sweep time - how fast you sweep from one frequency to the other.


DEWESoft - Datafile: FenceSweepNMT.d7d
Acquisition
Analysis
Data files
Setup
Review
Print
Export
Help
Settings
Channels
Events
Data header
File locking
Modal Test
Psophometer
Srs
MT0
Calculated
View Channel List
Calculation type
Step sweep (NMT)
Settings
Frequency source
From channel
Update time
0.1
s
Settling time
0.1
s
Function generator settings
Start freq.
190
Stop freq.
210
Delta freq.
1
Sweep time
1
Excitation channels
Adjust force
Force limit
15
%
Phase limit
20
deg.
Index
Direction
Sign
Input
AO Channel
Wanted force
Wanted phase
1 z + E1 - BK AO 1 - BK 2 0
2 X + E2 - VB AO 0 - VB 2 0
Response channels
Stop test if any excitation exceeds
10
Index
Direction
Sign
Input
1 z + R1
2 z + R2
3 z + R3
4 z + R4
5 z + R5
6 z + R6
7 z + R7
8 z + R8
9 Z + R9
10 Z + R10


DEWESoft - Datafile: FenceSweepNMT.d7d
Acquisition
Analysis
Data files
Setup
Review
Print
Export
Design
Edit
Help
Settings
Play
FWD
Replay speed: 1x
Mode
Sound
Offline math
Auto Recalc
Save
Overview
Scope
Recorder
FFT
storing started at 22.5.2010 14:43:26.875.0
storing stopped at 22.5.2010 14:45:15.770.2
Control properties
Cols
Transparent
Unified properties
Display options
Auto scale
Off
Graph type
Automatic
Histo. type
Full
X Axis type
Y Axis type
Lin
Lin
Number of ticks
Auto
Auto
Single values axis
Cursor type
Channel cursor
Frequency=196,000Hz, Value=0,4mV/N
202,0
190,000
194,750
199,500
204,250
209,000
Frequency [Hz]
Next point
INPUT
Manual mode
E1 - BK [N]
E2 - VB [N]
4.18
3.37
AO/Freq [Hz]
202.00
Correcting amplitude and phase
Rec
0.000
27.224
54.448
01:21.671
01:48.895
Search
TF 1z+/1z+
Amplitude_1z+/1z+
Phase_1z+/1z+
Re_1z+/1z+
Im_1z+/1z+
TF 2z+/1z+
Amplitude_1z+/2z+
Phase_1z+/2z+
Re_1z+/2z+
Im_1z+/2z+
TF 3z+/1z+
Amplitude_1z+/3z+
Phase_1z+/3z+
Re_1z+/3z+
Im_1z+/3z+
TF 4z+/1z+
Amplitude_1z+/4z+
Phase_1z+/4z+
Re_1z+/4z+
Im_1z+/4z+
TF 5z+/1z+
Amplitude_1z+/5z+
Phase_1z+/5z+
Re_1z+/5z+
Im_1z+/5z+
TF 6z+/1z+
Amplitude_1z+/6z+
Phase_1z+/6z+
Re_1z+/6z+
Im_1z+/6z+
TF 7z+/1z+
Amplitude_1z+/7z+
Phase_1z+/7z+
Re_1z+/7z+
Im_1z+/7z+

b) Manual - in manual mode we can tune the frequency at which we want to calculate response really exactly (we drive AO manually)

Start frequency - frequency at which we start searching for resonance.



Common settings

Update time - size of block for NMT calculation in seconds.

Excitation channel settings

Adjust force and phase - if we want to use more than one shaker we need to mach phases and forces of that shakers that we can correctly calculate transfer function.

Response channels settings

Stop test if any channel exceeds - this is just a safety that we don't damage our structure. When the response acceleration from our system is above the limit analog out and measurement are stopped.

OUTPUTS

a) For each measured point:

Amplitude - signal amplitude.

Phase - signal phase.

Real - real part of a signal.

Imaginary - 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...).

b) For excitations:

Excitation spectrum - FFT spectrums of excitation signal.

c) Common channels:

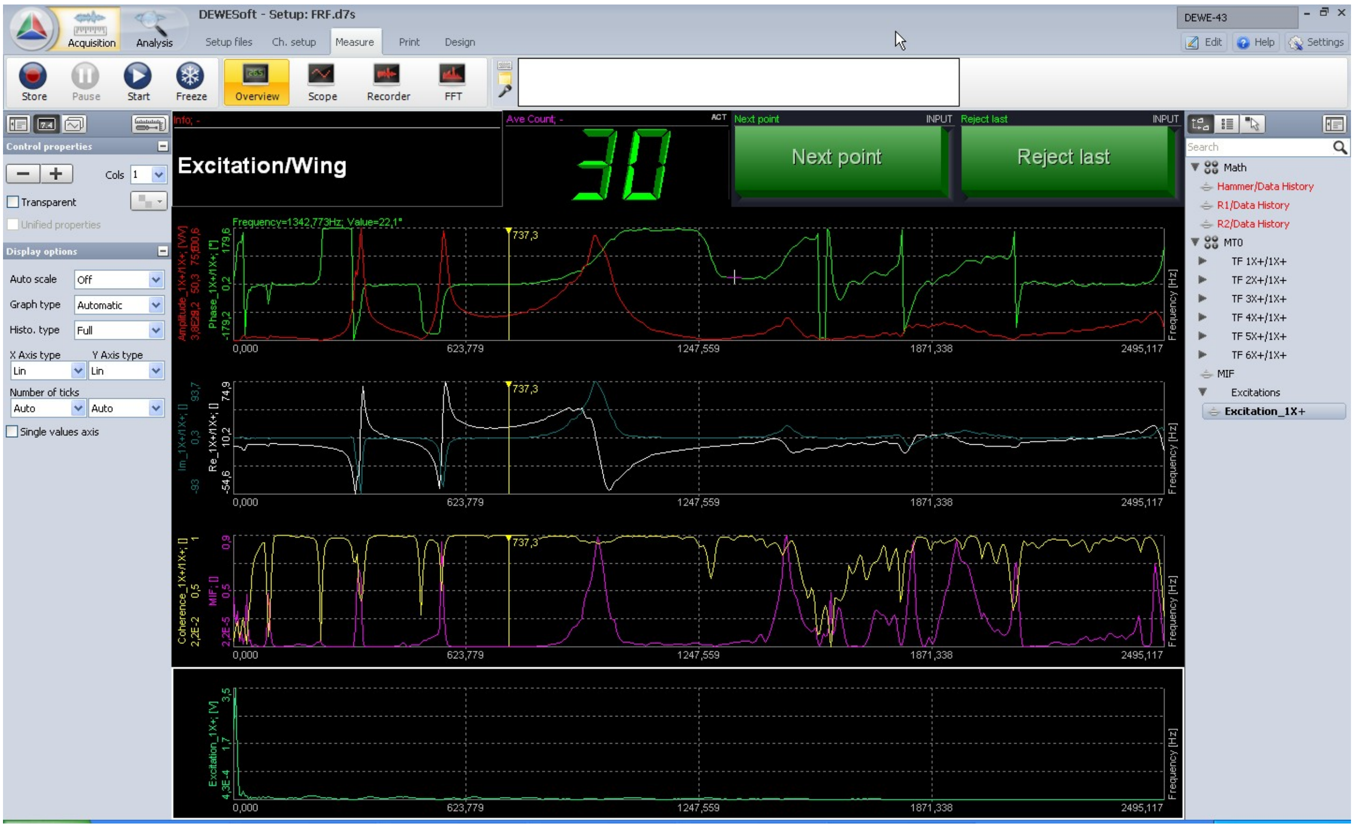
Info - information about which point is currently measured.

MIF - mode indicator function is a function ranging from zero to one and shows where resonances are.

Ave Count - 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 next point when grouping is used.



CIRCLE FIT (exact frequency damping estimation)

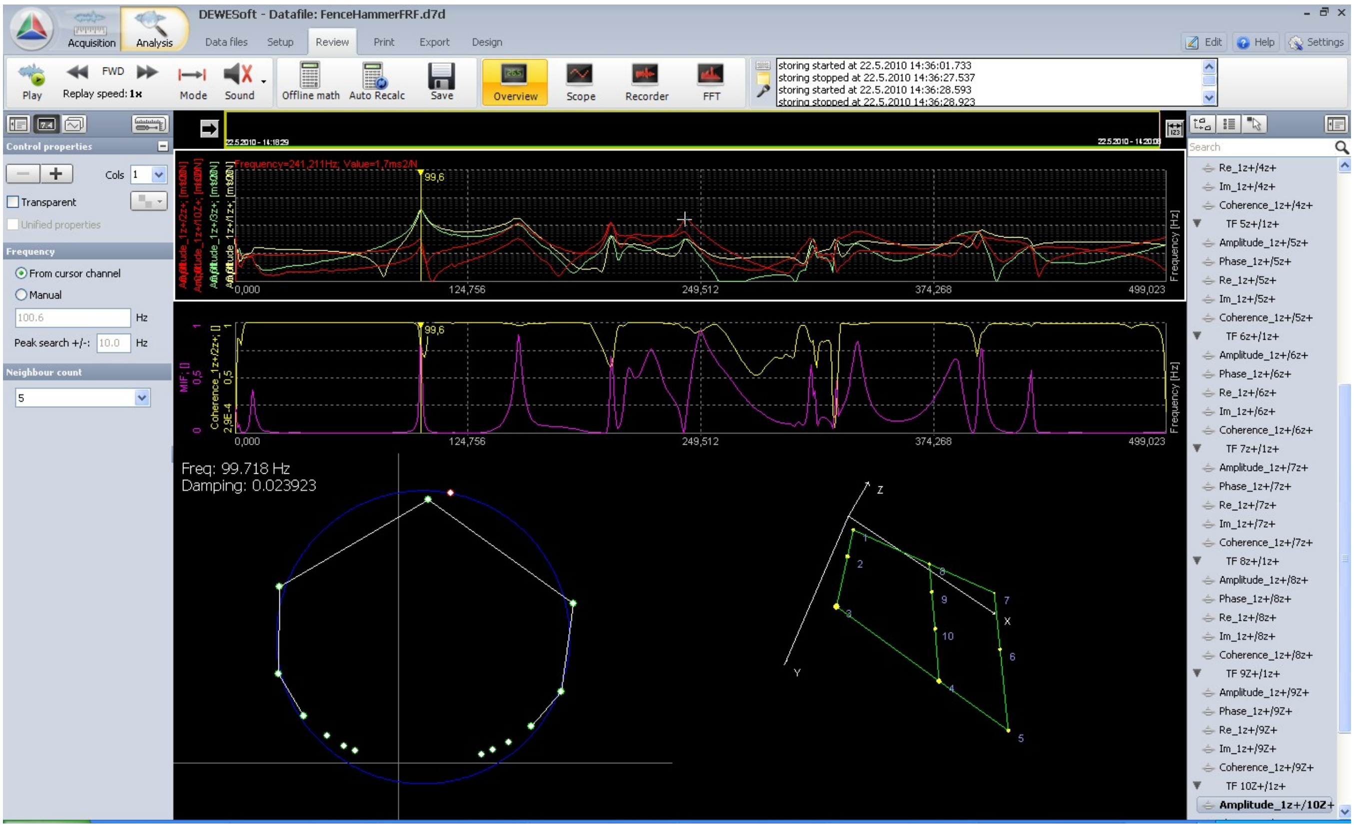
This is a visual control for basic modal analysis. You can use the yellow cursor in 2D graph and set it to frequency content of interest. Circle fit will fit will find the nearest peak (resonance) and will fit the circle into that data set. Than it will calculate damping factor and more exact frequency (between line resolution of FFT).

From cursor channel - yellow cursor on 2D graph is taken for frequency point determination.

Manual - you can manually enter frequency point for circle fit procedure.

Peak search (manual mode) - area in which we will search for peak for circle fit.

Neighbor count - number of neighbors taken into account when doing circle fit.

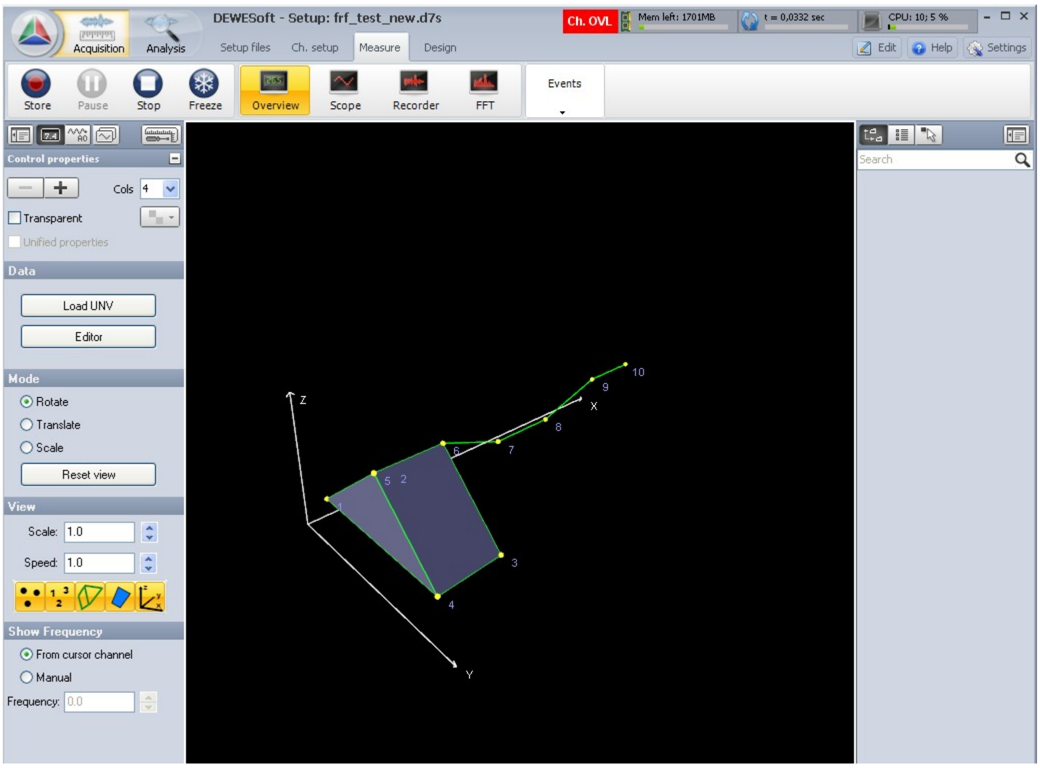


2.12.6.1 Geometry display

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*



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*.

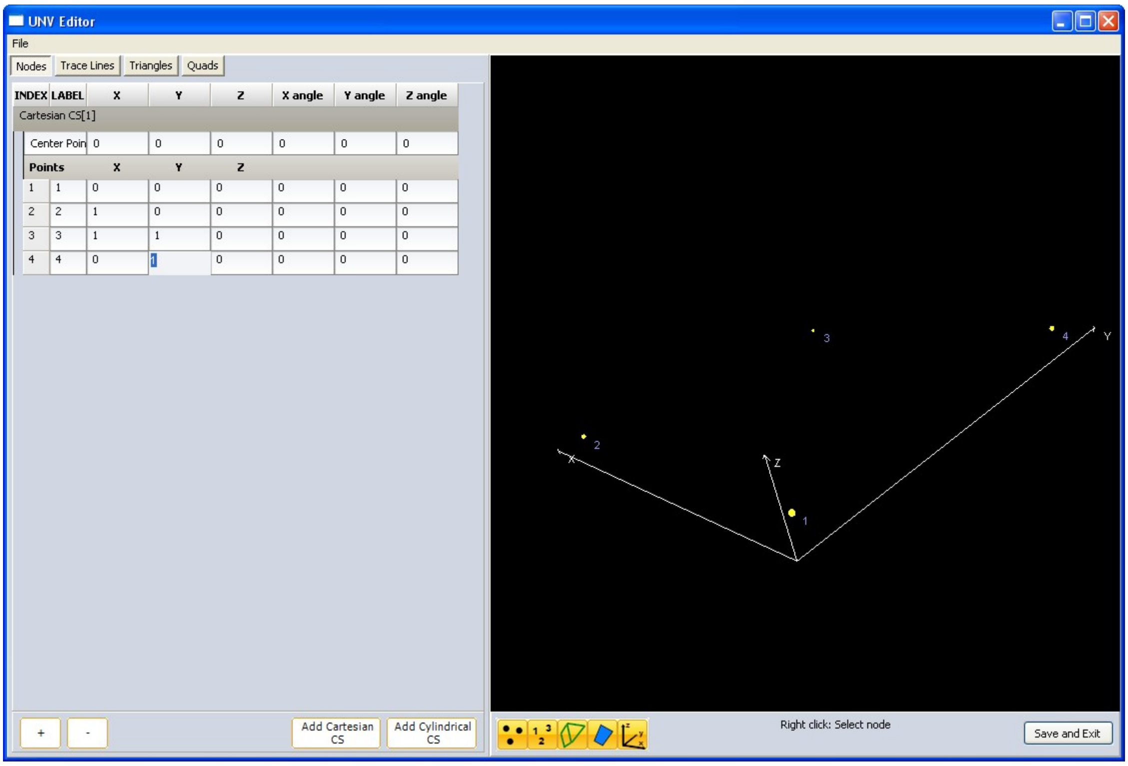
Creation of structure

If structure data is already loaded and we want to make a new one, we click on File → **Clear structure**.

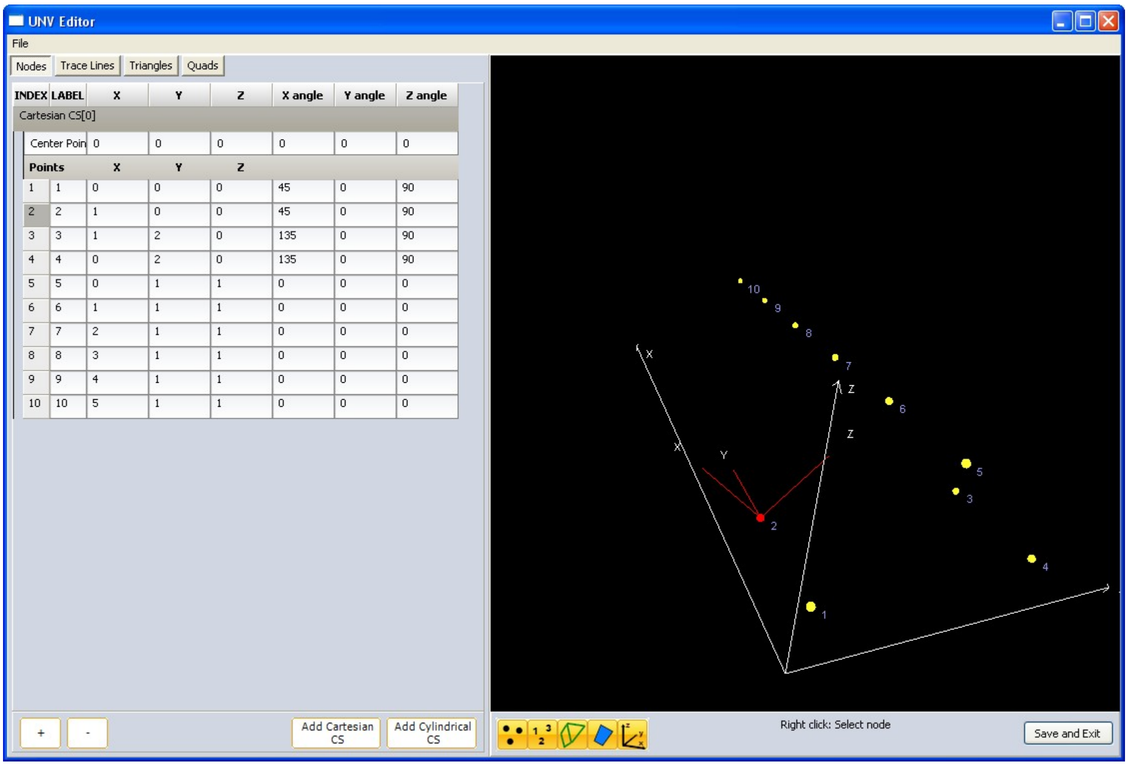
Nodes

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 + button.

In the picture below, you can see Cartesian coordinate system with 4 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.



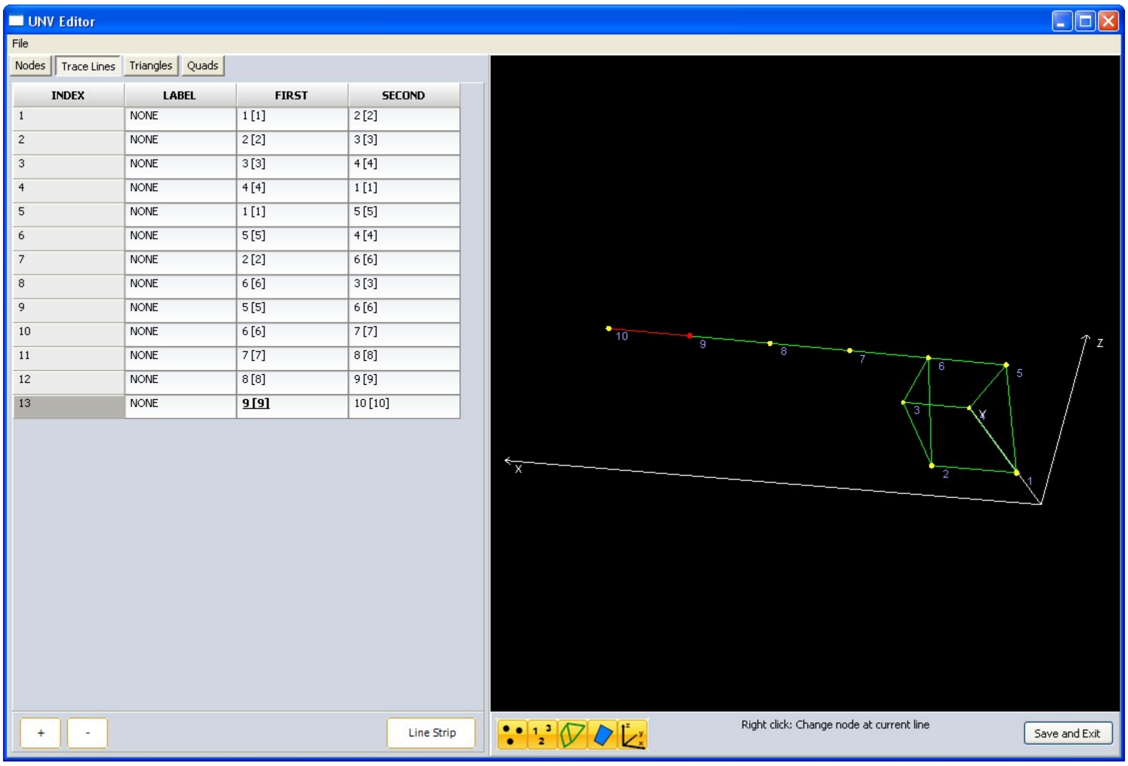
When node is selected, we can remove it by pressing button on nodes tab.

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 unselect 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 + 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.

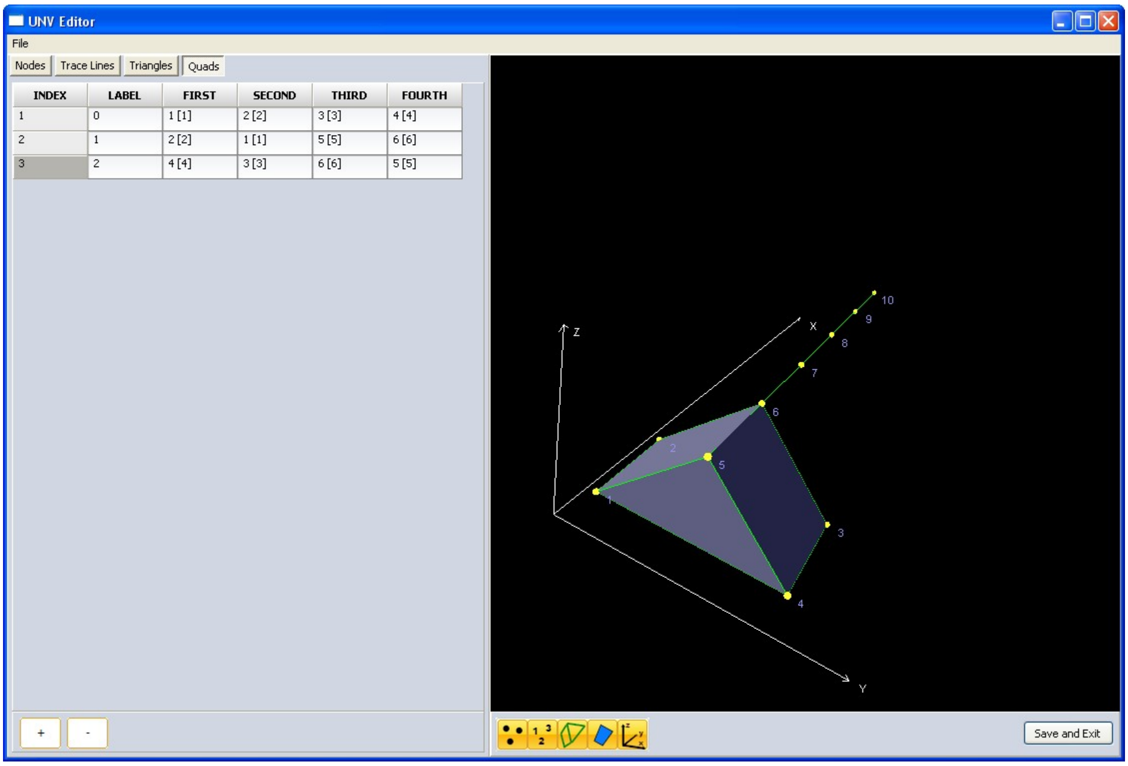
In the picture below we can see object with some trace lines added.



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 + 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.

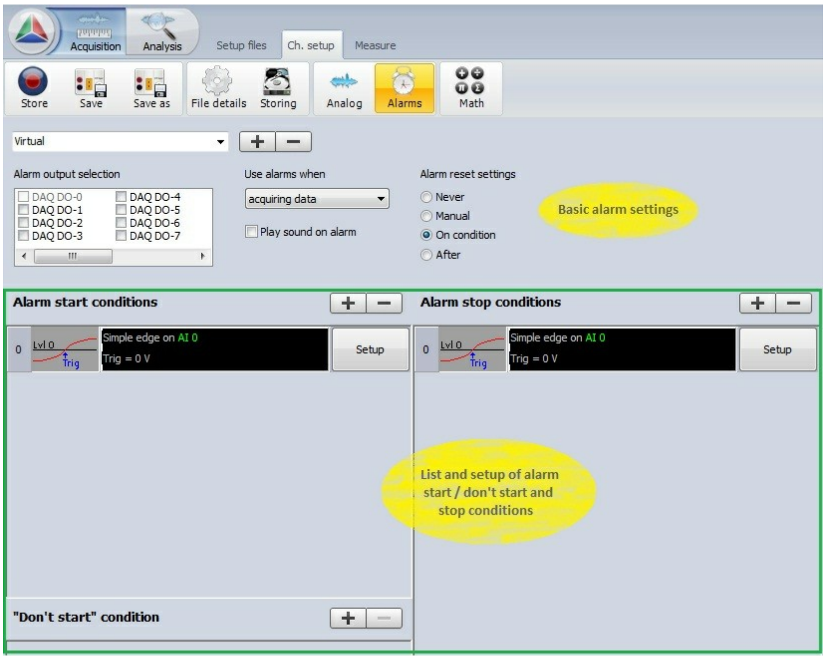
2.13 Alarm Monitoring

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 a **Alarms** tab on DEWESoft *Setup screen*, on 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)

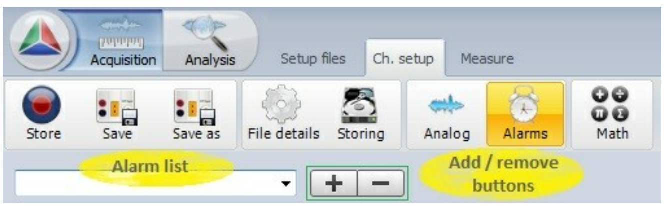
Alarm conditions — to **Add** (New) / **Change** / **Remove** and *define* ***Alarm start condition*** or ***"Don't start" condition*** and eventual ***Alarm stop conditions***



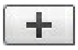
Basic settings

Add (New) / Delete alarm monitoring channel

When you select **Alarms** tab to define *new* **Alarm**, on lower part of screen an *empty* alarm list appear:

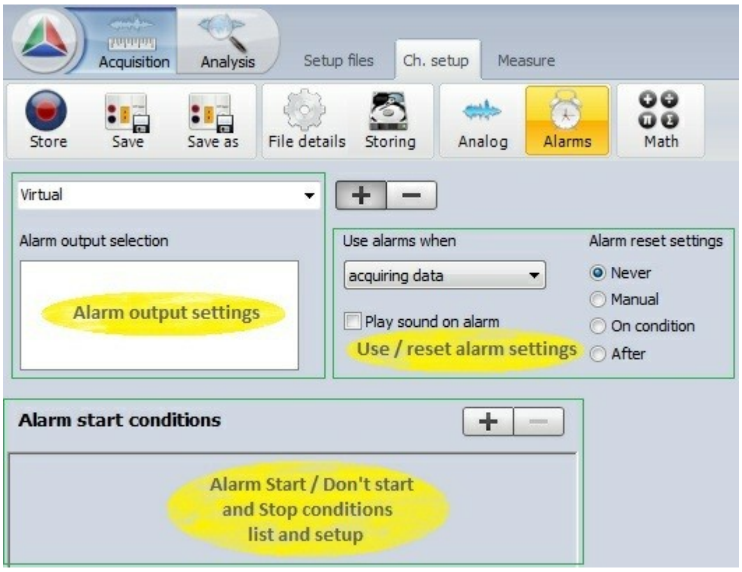


with *two icons*:

 to *add new* alarm condition

 to *remove 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)

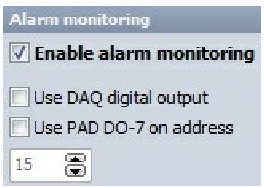
2. STEP

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

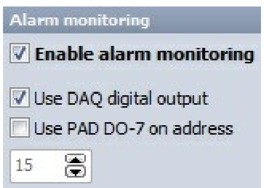
Depend on enabled choices in *Hardware setup* → *see* also → **Alarms Hardware setup**

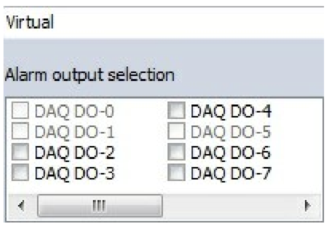
1. only the Enable alarm monitoring box is *checked*



Alarm output selection choice list is *empty* → *see* picture above

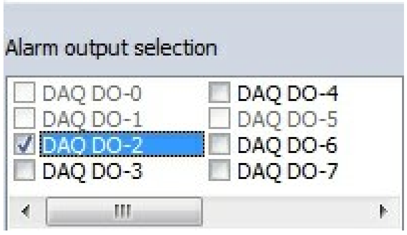
2. the Enable alarm monitoring and Use DAQ digital output box are *checked*





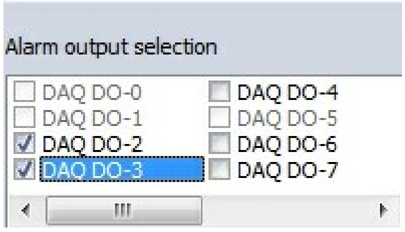
if no alarm is checked, the alarm will be virtual - only available for display

DAQ DIO2



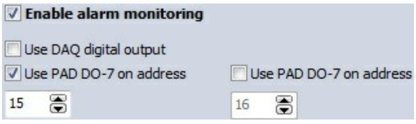
after checking one DAQ output on Alarm output selection the alarms will be output on selected channel

DAQ DIO2;DIO3

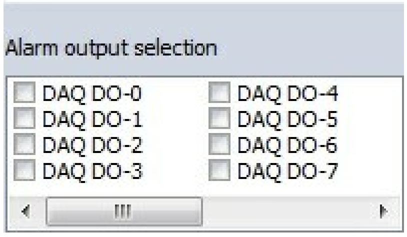


after checking two DAQ outputs on Alarm output selection the alarm will be output on two output channels

3. the Enable alarm monitoring and Use PAD DO-7 on address box are *checked*

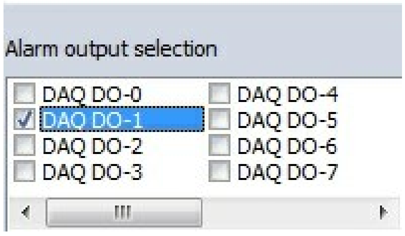


Virtual



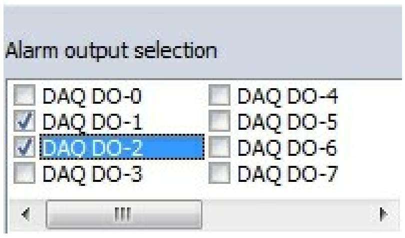
if no alarm is checked, the alarm will be virtual - only available for display

PAD DO/2



after checking one PAD output the alarms will be output on selected channel

PAD DO/2;DO/3



after checking two PAD outputs the alarm will be output on two output channels

The alarm can also use *two* PAD output modules (14 channels).

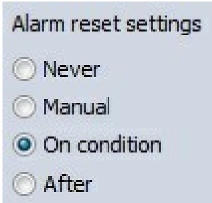
Use alarms when

Select from Use alarms when drop down list:

- alarm while **acquiring data**
- alarm during **storing**

Alarm reset settings

Select appropriate choice from displayed list:



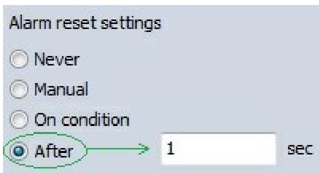
- Never when *Start condition* carry out, **Alarm** *remain* ***on*** to *end* of acquiring or storing data
- Manual when *Start condition* carry out, **Alarm** *go* ***on*** and on first row of display screen red **Alarm ON** button appear:

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 right lower part of screen also Alarm stop conditions section appear:

for information about ***setup alarm condition*** *see* → **Alarm conditions**

- After when *Start condition* carry out, **Alarm** *go* ***on*** and will be *reset after* amount of sec, which is entered in field beside this control:



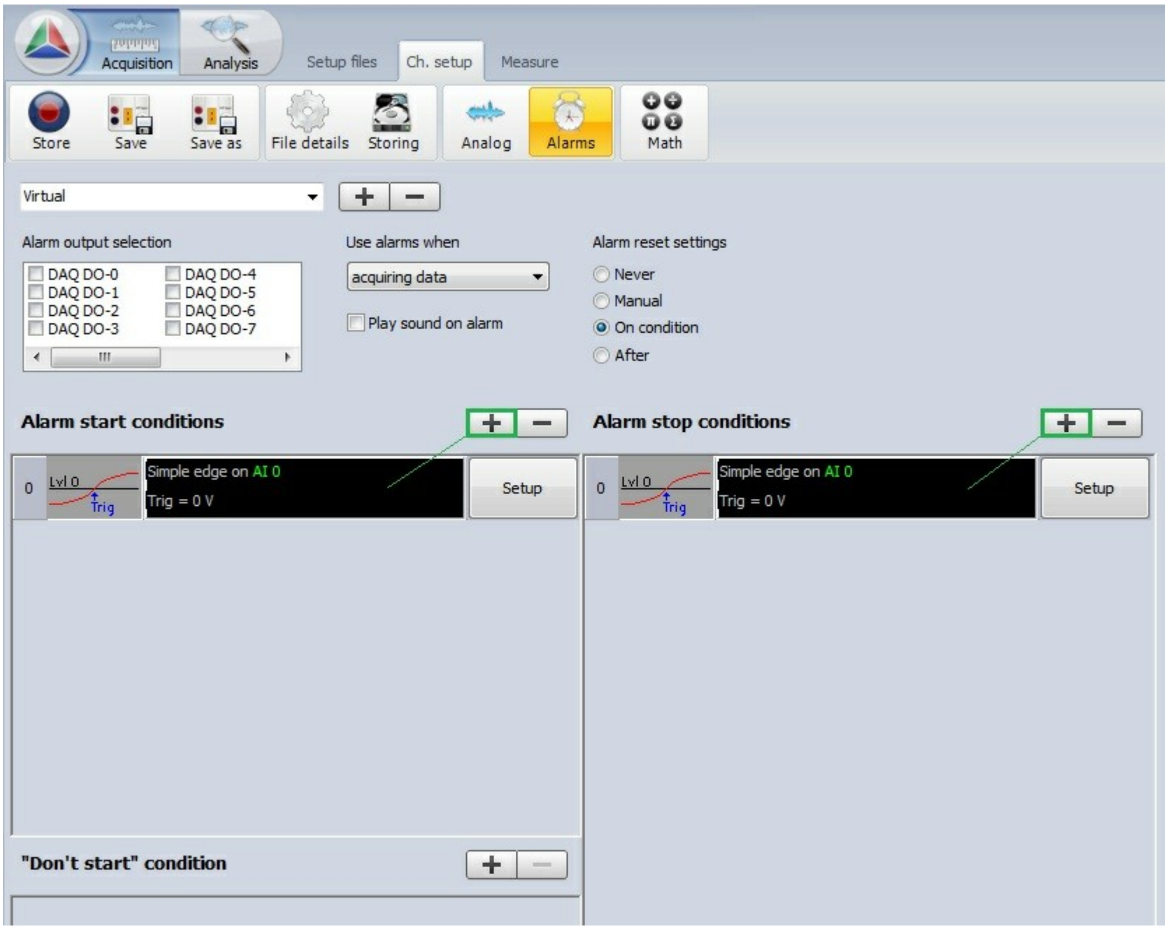
Set Alarm conditions

After you have done *add* and *basics alarm settings*, on lower part of screen can be *add / change* ***alarm conditions***:

- Alarm start condition or "Don't start" condition
- Alarm stop conditions (this part is displayed *only* for On condition *Alarm reset settings* option → *see* above)

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:

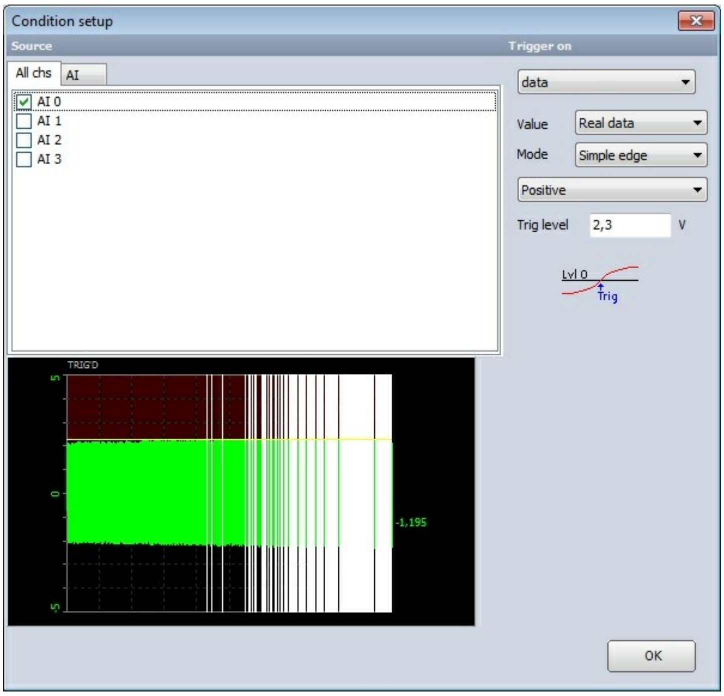


The **alarm monitoring settings** are similar to *trigger* conditions → *see* also → User Guide → **Setup - Triggers**

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. On 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* → User Guide → **Trigger Condition setup**

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

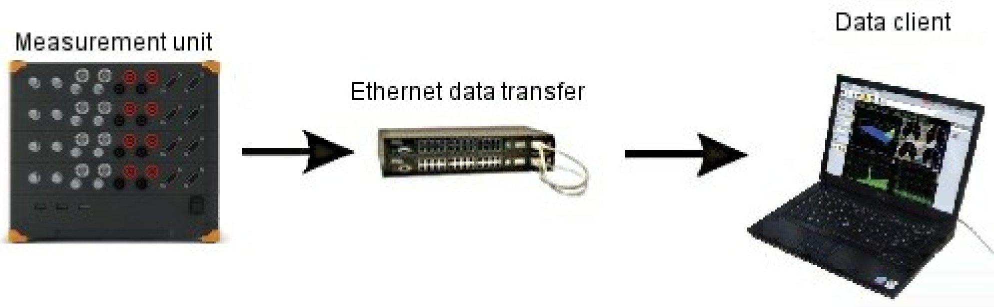
To *remove* an alarm condition, select the output channel and the condition to delete by clicking on it. After you have selected the condition in that way, press the − - **Remove** button.

2.14 NET acquisition

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.
It is important to note that which channels can be **viewed** on the CLIENT(s), the actual data are **stored** on the MU's.

Required hardware	Any AD card
Required software	SE or higher + NET option, EE
Setup sample rate	Depending on used math module



DEWESoft **NET data acquisition** is a software option, which must meet some **DEWESoft NET Requirements**.

This help manual should give the user an overview on the DEWESoft **NET** *functionality* with two DEWE-501 measurement units.

After DEWESoft **NET Hardware setup** (this procedure is to perform *only* by installation new or changing counter hardware) all *channel setups* must be *defined* and set up with:

Remotely controlling a Slave MU

- **Local NET setup**
- **Remote NET setup**:
  - **Channel setup**
  - **Transfer setup**

In this *Remotely controlling a Slave MU* section we are using only the **master client** computer. MU is set to **Slave MU** *mode*, and we have already *connected* to it using the steps from the **Measure-Connecting to the remote** units section.

NOTE: All steps are done on the CLIENT.

We are not touching the MU at all. It could be a few feet *away*, or 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 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** (green *indicates* the *selected unit*).

Note the buttons for **LOCAL** and **MINITAUR**. If you have *more* than one MU, 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.

On this screen you are doing exactly what you would be doing if LOCAL was a *stand-alone MU… set* the *dynamic* and *reduced sample rates*, choose a *filename*, set *storing option* and more. → *see* User Guide → **Recording Setup**

Math tab

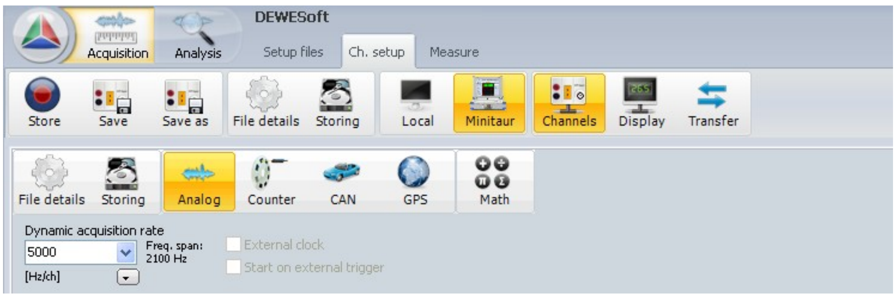
However, LOCAL computer still has a **Math** tab. It is interesting to note that you can *perform math functions* in *real time* on this **client** using any *channels* that are *transferred* from the MU's!

You can even combine *channels* from *more* than one MU here in *math channels* – as long as the MU's are synchronized!

Consult the User Guide → **Math Module** if you need thorough help in setting up a **Math** *channel*.

Remote NET Setup

Click on the **DEWE501B** (or **DEWE501A**) button so that we can proceed to *set up* this Slave MU:



The DEWE501B computer is our **Slave MU**, so it has also *channels* of its own.

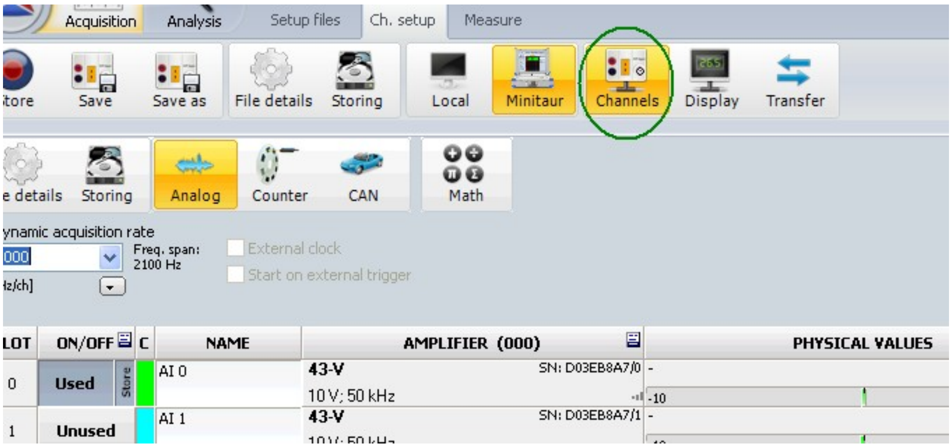
On central part of this screen you are doing exactly what you would be doing if Minitaur was a *stand-alone MU: set* the *dynamic* and *reduced sample rates*, choose a *filename*, set *storing option*... → *see* User Guide → **Recording Setup**

Note the *three tabs* under Minitaur:

- **CHANNEL SETUP**
- **DISPLAY SETUP** → *see* User Guide → **Measure - NET acquisition**
- **TRANSFER SETUP**

Channel setup (Remote)

In the **Remote** Channel setup screen each channel from the *selected remote* unit can be *configured* in the typical DEWESoft habit. The *Remote channel setup* screen is actually a copy of the *Channel setup* screen *from* the *remote* MU. This means that what is changed *locally* is simultaneously changed on the *remote* unit.



Remote channel setup screen for DEWE501B

On this screen you are doing exactly what you would be doing if Minitaur was a *stand-alone MU*... *activating channels* with the Used / Unused buttons, *scaling* them using the Set ch. # buttons, and so on.

for details → *see* User Guide → **Channel Setup**

In this example, Minitaur is a computer with 1 *analog input* channel. In addition, more **Math channels** can be *created*.

for details → *see* User Guide → **Math Channel Setup**

Transfer setup (Remote)

In the **Remote** Transfer setup screen we will set up the *transfer* from the MU → this mean *transfer which channels* will be *sent* across the **network** *during recording*, for **display** on the client'.

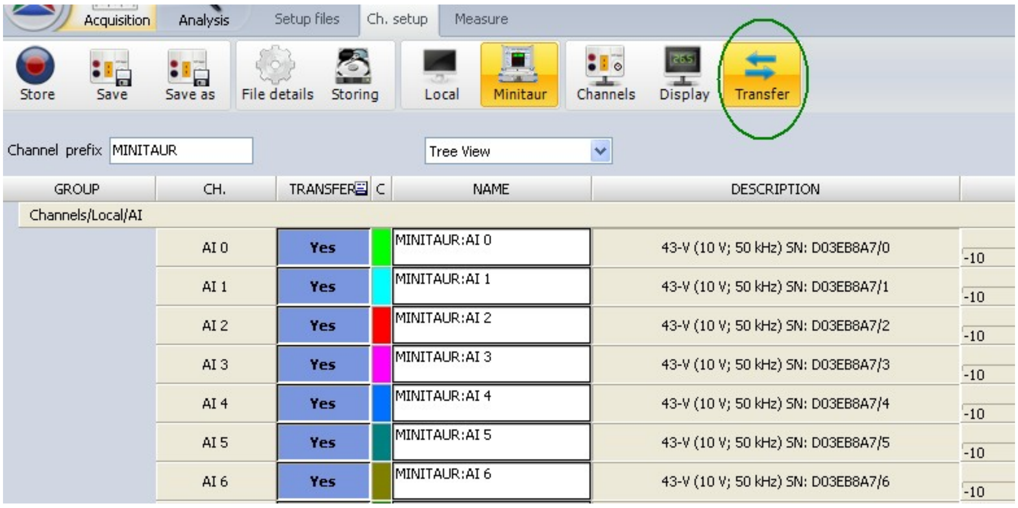
That is the entire scope of what transfer means. It has no effect on the actual Storage of data on the client (assuming that **local** storage is *enabled* – the default and highly *recommended* mode in System menu → Hardware setup... → **NET** tab).

Therefore, you can have **multiple** MU's, 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 MU's!

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 the **Remote** Transfer setup screen, click on the **Yes** / **No** buttons in the Transfer column to *select* the **channels** to be *transferred* to the Master unit during *measurement*.

The Remote transfer setup screen *only* shows *channels* which are activated in the *Channel setup*.

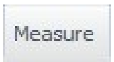


You can see that in our example Minitaur has *eleven* channels *set up* to be *transferred* in real time to the client. Of course they will also be *stored* on the **local MU**, because this has been selected by *default* on the *Hardware setup* screen (System menu → Hardware setup... → **NET** page).

NOTE: *The more channels are selected for transfer, the higher the network load! Choose* Store data on measurement units *to minimize network usage during measurement.*

The acquired data can be transferred from the local MU for viewing on the client computer *after* the measurement using **Transfer** button → *see* → User Guide → **Uploading Stored Data**.

3 Measure - Data acquisition



The **Measure** procedure is the most important part of DEWESoft ***Data Acquisition*** system. Acquired data from activated *channels* are *available* immediately and can be **viewed** in *all* online screen *instruments* either as ***digital values*** or ***graphs***. All live channel data can be **stored** on measurement unit *locally* or transferred via *Ethernet* to the *client*(s) on the *server* side.

Before measuring the data, please be sure that you set the hardware and measurement correctly in two DEWESoft procedures:

Hardware setup this procedure is to perform *only* by installation *new* hardware or when *changing* hardware

Acquisition setup set up and activate all *Analog Input* and *Output* channels; define and set up supplementary *Measuring Procedure* (like *General Mathematics Module*, Counter, Alarms, Triggers,...); *define* and *set* special *Measuring Applications* (like CAN, Video, GPS, Power, Torsional vibration, Sound level,... acquisition)

see → User Guide → **Measurement setup**

Measure procedure includes:

1. STEP

Design Display Screens define acquired data *appearance* on screen within DEWESoft; place *displays* (*instruments*) on the screen or use the *predefined screens*

for detailed information about ***predefined screens***
see → User Guide → **Predefined displays**

for detailed information about ***displays (instruments)***
see → User Guide → **Instrument setup**

2. STEP

Setup storing define the conditions data are *stored*

3. STEP

Storing data *start measure* and *store data*: Manually, Triggered, Remotely or Automatically

see → **Recording Acquired Data**

Special DEWESoft Measure procedures are:

- **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*
- **Freeze mode** - only displays are *stopped - frozen* with the last data

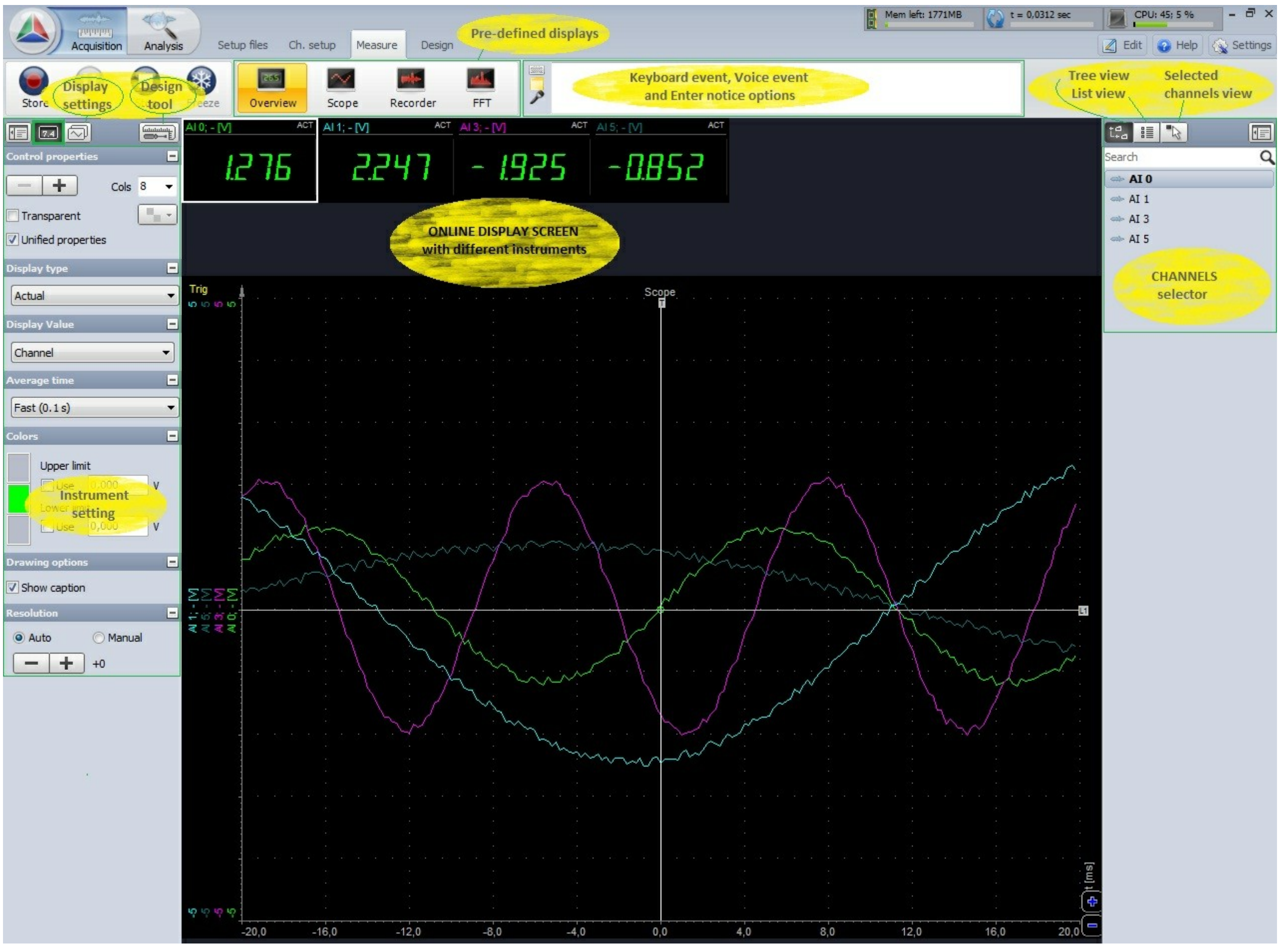
3.1 Displays design

Primary goal of DEWESoft **design** online **display screen** is to create *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 analyzer*,... and arrange them freely in front of any *graphic* in your system to have a *simple*, but efficient *overview* of your signals and measurement.

In DEWESoft we know basically four types of **visual controls**:

- controls which shows only one value (*digital meter*, *bar meter*, *analog meter*, *indicator lamp*)
- controls which typically shows all the data (*recorder*, *vertical recorder*, *xy recorder*, *GPS map*)
- controls which shows 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 **pre-defined** displays but these screens can be altered and **own specific displays** can be **created** with *different appearance*. The picture below shows typical display with standard elements for designing the display.



3.1.1 Pre-defined displays

DEWESoft provide some default screens with **pre-defined displays** of *different* instruments for *each* input, for example *scope*, *recorder*, *FFT analyzer,...*

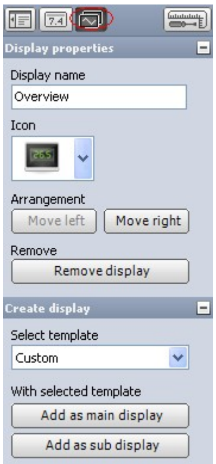
These instruments are *built as* a jump start that software acts like a *classic instruments* like strip chart recorders, classic oscilloscopes and so on. We have the following **pre-defined displays** - screens:

Name	Icon	Description
Overview		**Overview** screen include 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.
Scope		**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.
Recorder		**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***.
Vertical 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.
FFT analysis		**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.
Power analysis		**Power 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		**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		**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 acquired values in *digital meters* and a *recorder* view for time *domain* display of *acquired data*.

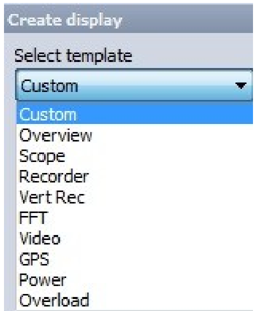
With new setup, there are only few displays which are added. These are Overview, Scope, Recorder and FFT.

Displays can be changed by selecting *icon* on DEWESoft toolbar.

A great new feature of version 7 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 - Display settings** for more information on this topic.

3.1.2 Display settings

All DEWESoft **pre-defined** and **added** displays can be **adapted** to meet the user's requirements and to have a *simple*, but 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 instruments** on the screen - Design mode
    - **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**
- **Displays 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 functions** — to *define* ***Instruments appearance*** on screen, *common* and *additional* ***control*** and ***function***

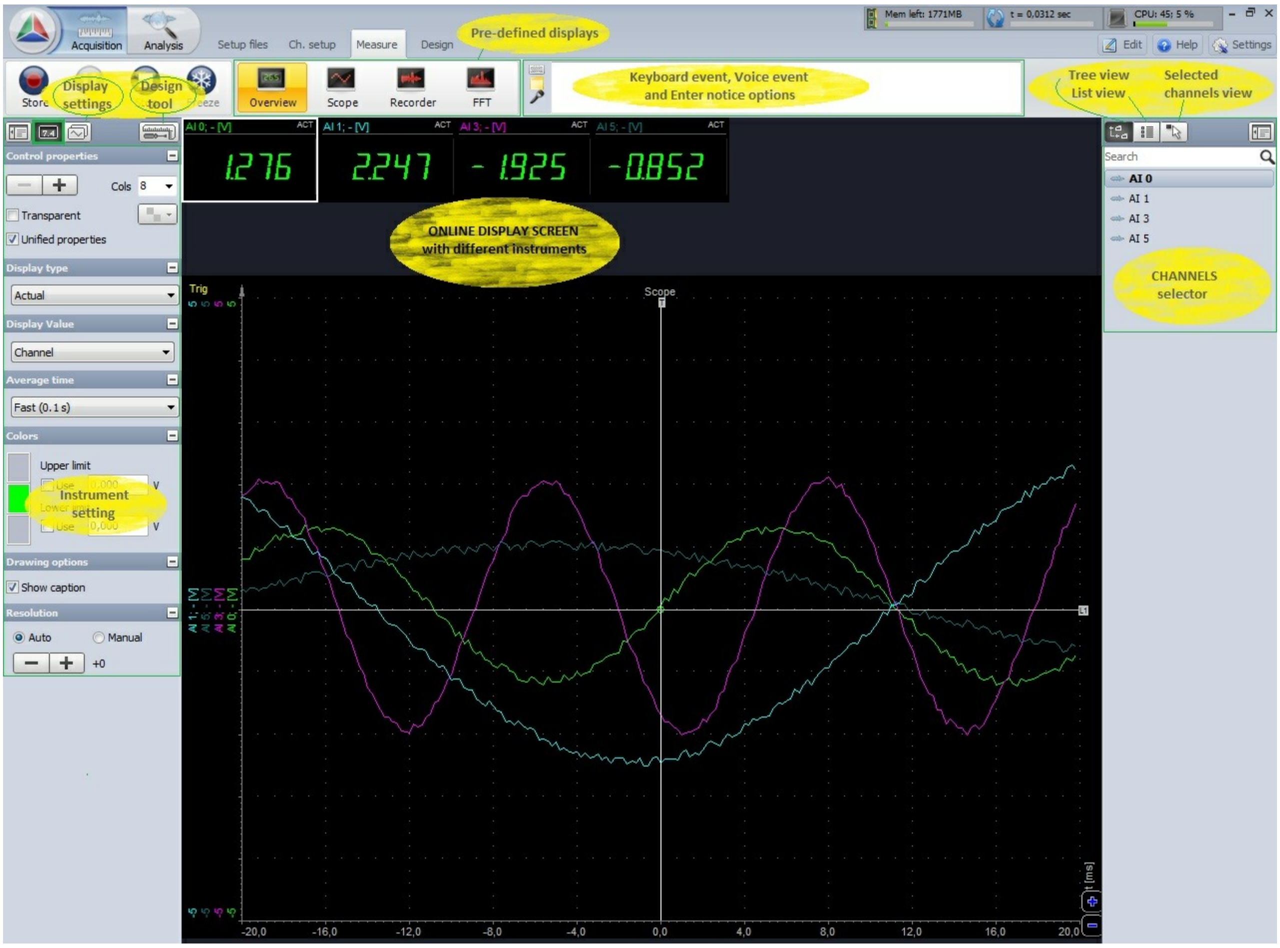
Design mode

When you press the 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* tool bar appear.

As soon as you enter *any* **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 tool bar *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* tool bar
- **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* tool bar 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* tool bar, 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 hints about *using* ***appropriate instrument*** *see also* → DEWESoft Tutorials

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 reight 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.

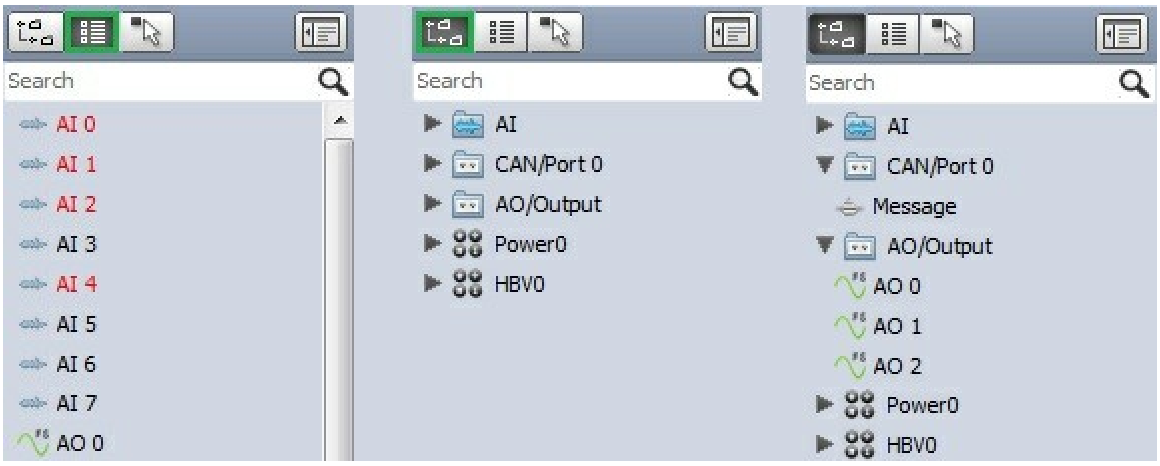
You can have the *same* channel assigned to multiple *instruments* if desired.

3.1.2.1 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:



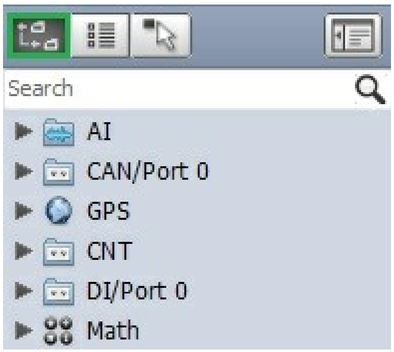
CHANNEL list view | GROUP view | GROUP view with subchannels list

CHANNEL (List) view / Group view

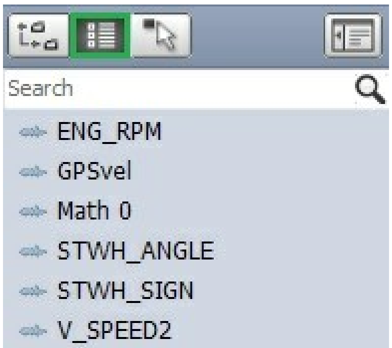
When you select the Group view (left picture below), channels will be grouped according to its *source*. we will have groups for analog input (AI), CAN, math and others. We can see the channels by expanding the group simply by clicking on it.

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*

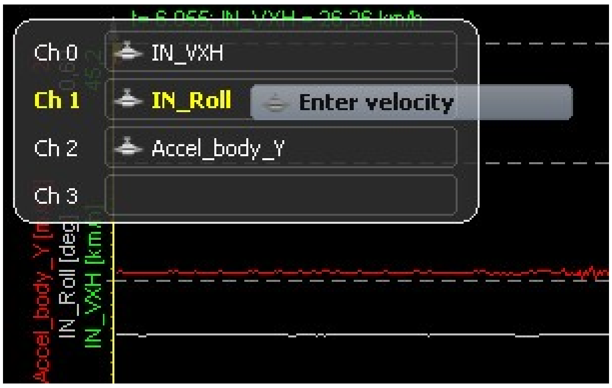
Selected channels *view*

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.

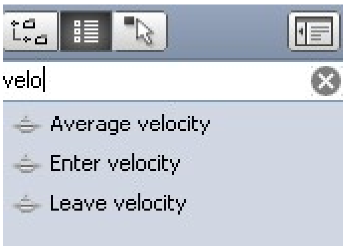
There are several other ways to assign channel to visual control. We can *drag and drop* the channel to the visual control. When we click and hold the left mouse button, channel will be selected and we can drag it over to any visual control. We can drop it to any place and that will add the channel to 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 Enter velocity channel.



In *Design* mode we can *drag and drop* channel to 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

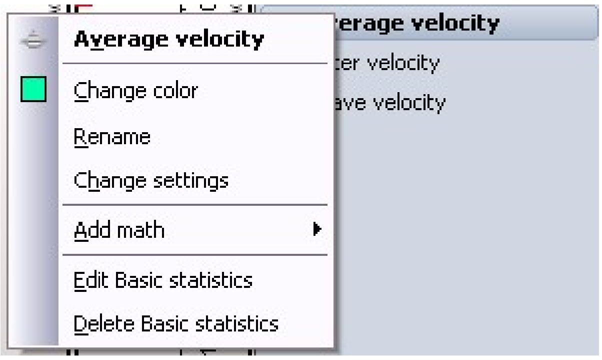
DEWESoft setups can easily have *hundreds* and even *thousands* of channels. To find channels we have added a feature to **search** for them by the *name*. If we enter the keyword in *Search* field, DEWESoft will look for channels with those keyword in the name (at any place). We can cancel the search 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.



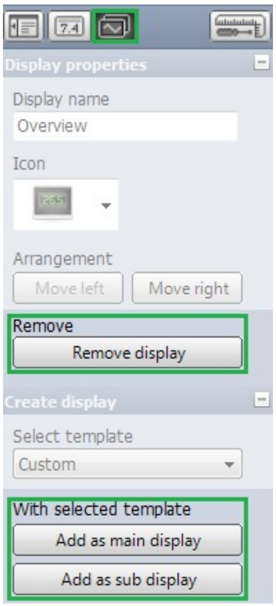
3.1.2.2 Displays menu functions

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).

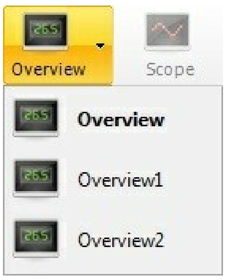


If we want to create *standard* display, then *choose* it from template list. If we want to create just an *empty* display, then Custom template must be chosen. We can add the display as main display, which will add a new screen right next to the selected display on the display menu bar. Then we can change the *name* and *icon* by changing Display properties.

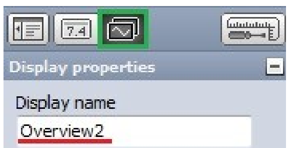
We can also *add* the display as a *sub* display, which will ad a new display as a *child* item of currently selected screen. The display icon will get a dropdown button to change the currently shown display.

Additional display screens

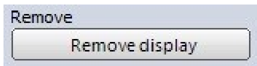
When you have added *additional* displays, the you can *select* them directly with the *down arrow* next to the appropriate *display icon*:



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 dropdown 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.

3.1.2.3 Screen edit functions

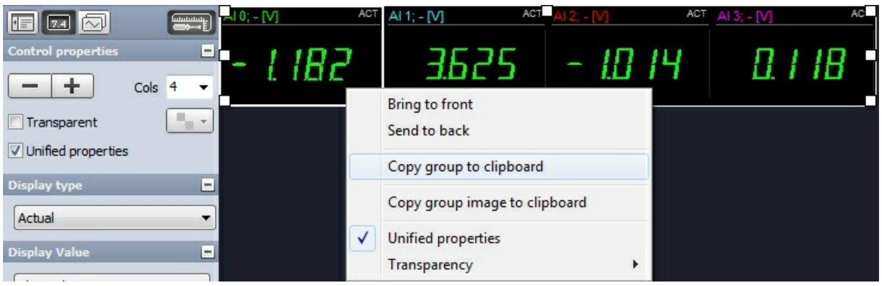
DEWESoft offers an additional **screen edit** function to create *intelligible display* all channels data on the screen.

- **Instruments 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 <u>same</u> ***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:

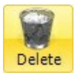


or select **Copy group to Clipboard** from the Edit menu.

Then *move* the cursor to the desired *position* (must be on screen *outside* group), *right-click* again and *select*  button which appear 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* tool bar 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!*

3.1.2.4 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**

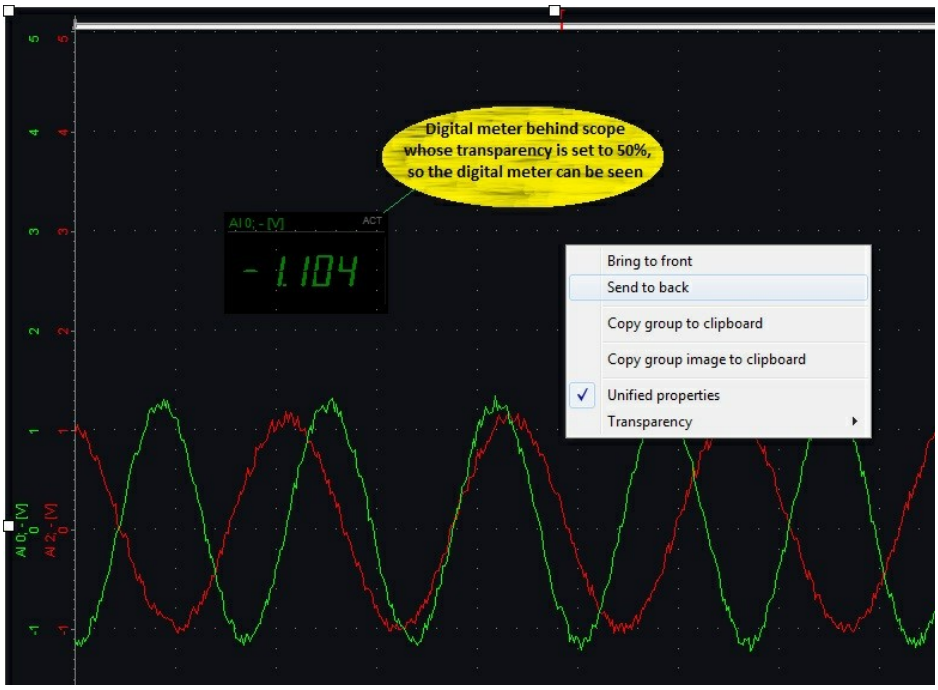
Position

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*

  Tip: use the SHIFT key simultaneously *with* the arrow keys to *increase* moving speed.

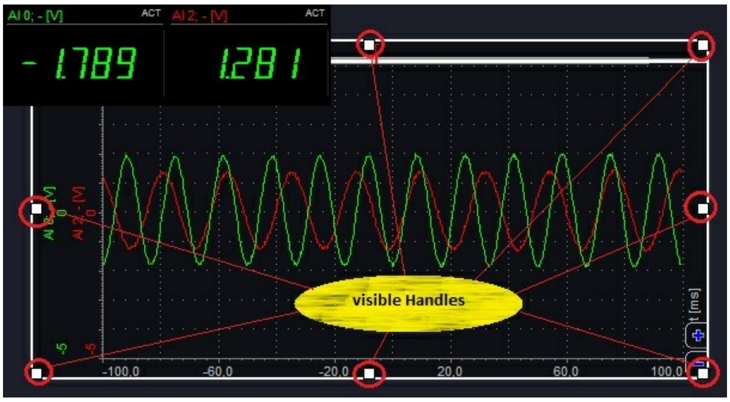
If the element comes in conflict with other elements, you can **bring** the element in *front* of the other one or *behind* it; therefore *right-click* on the element and then select Bring to front / Send to back from the appearing menu list.



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***" behaviour. 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

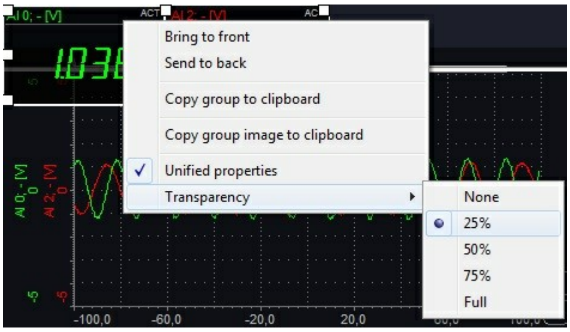
Now we have defined the instrument, its channel(s), size and its position. But what to do when instruments overlap one another?

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:

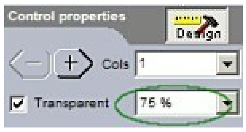
- use the **Transparent** checkbox in the Control properties bar and *select* the transparency *value* from the drop down list

  for detailed information about ***Control properties*** → *see* → **Common graph settings** and **Group properties**

- *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:



The following example should give you an impression of the function - the Digital meter transparency has been set to 75%:

3.1.3 Visual control settings

DEWESoft allows you to set up *different* **instruments** for *each* input, for example *digital meter*, *recorder*, *FFT analyzer*,... 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.

Standard instruments	*Additional instruments*	
Digital meter	**Recorder**	**Vector scope**
Horizontal 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**

3.1.3.1 Group properties

DEWESoft offers a Design **Control properties** to create *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)

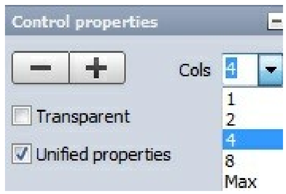
Create an instrument group

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 <u>same</u> *sort* of instruments.

Example digital meter:

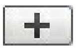
Columns number in group

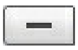
With the **Cols** drop down list, you can *define* how many ***columns*** of instruments should be allowed in group.



Add / Remove instrument to / from group

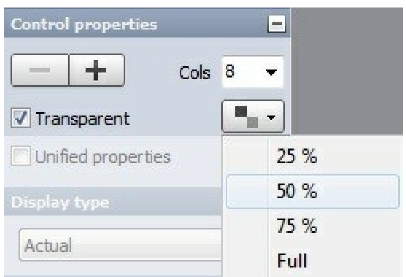
Sometimes you might require more graphs - instruments on one display according to your requirements. After defined *column number*, simply *press*:

 button to **add** *same* instrument or

 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 fully 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 <u>all</u> 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 <u>each</u> 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*...

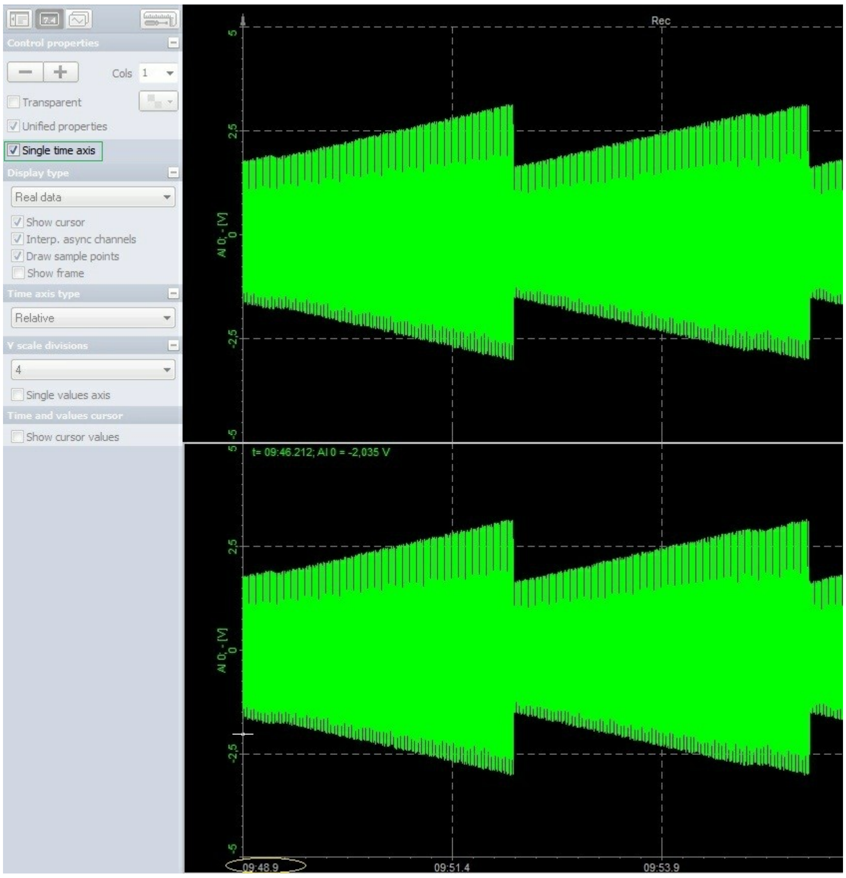
3.1.3.2 Common graph settings

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 checkbox to change between these two modes.

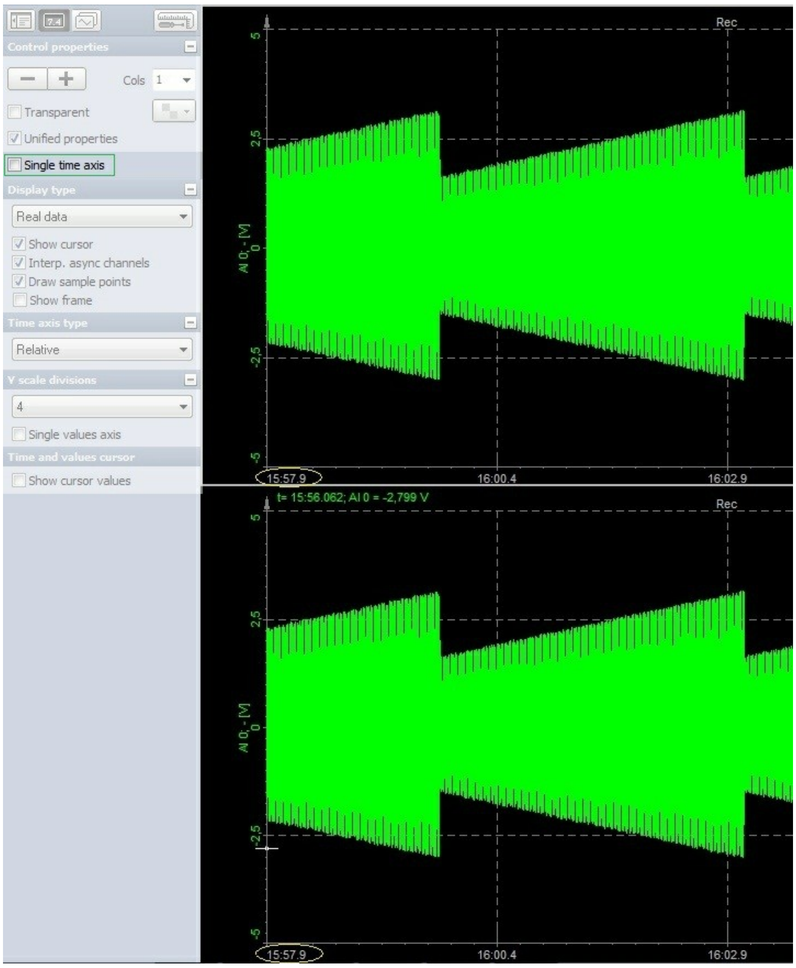
- ***single*** time axis → **Single time axis** box checked

The *advantage* of the ***single*** time axis is to have <u>more space</u> to *display* the *data*.



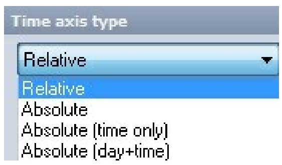
- ***multiple*** time axis → **Single time axis** box unchecked

The *advantage* of the ***multiple*** time axes is to make it easier to *read* time related *information*.



Appearance by *vertical instruments* is same, except Time axis is *vertical*.

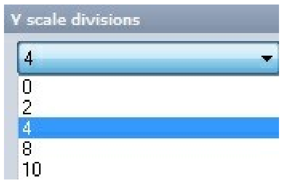
Time axis type



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.

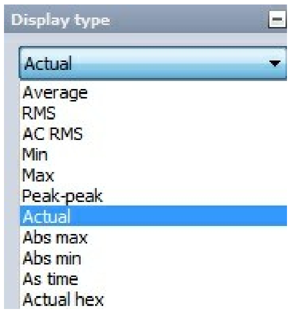
- **Relative** - *elapsed* ***time*** since *recording* or *monitoring* began
- **Absolute** - *current* ***time*** and ***date***; this format is *compatible* with the timing information according to UTC
- **Absolute (time only)** - *current* ***time*** only
- **Absolute (day+time)** - ***number*** of the *current* ***day*** in year and the ***time***

Grid resolution



To make the analysis more comfortable, DEWESoft contains the possibility to *change* the ***number*** of *grid lines* in the recorder display. Depending on range and signal, it may be interesting to change the number of grid lines in **Y scale divisions** from 4 (*standard*) to 10 - or *remove* them completely.

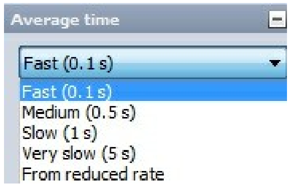
Display type



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



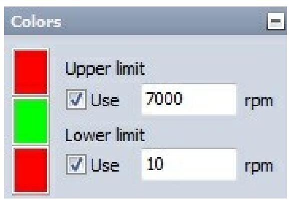
With the Average time drop-down list, you can *define* the update rate for the *selected instrument*. As a *standard*, the *values* for the display type will be *calculated* **Fast (0.1 s)** over a period of 0.1 seconds, which represents also the *internal minimum* calculation period.

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 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 → *see* → **Basic Recording Setup**

Upper & Lower limit

You can *set* ***high*** and/or ***low alarm limits*** for each *instrument*, which will cause the instrument number to *change* from green to red 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).

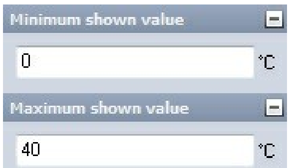
Note that the number changes to red as a standard when it is ***outside*** of both limit, and remains green when it is within the limit(s). You can set **Lower limit**, **Upper limit**, or *both* limits for each meter.

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.

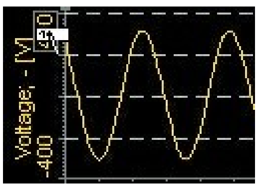
Appearance and functionality by *vertical instruments* is same, except Time axis is *vertical*.

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:

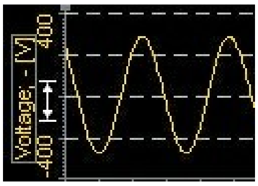
- ***Enter values***



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 grey outlined box around the scale value indicates the *selected* channel.

- ***Auto scale***



Move cursor to the axis scale. If you have reached the auto scale area, the cursor *changes* to the following *appearance*: .

A grey outlined box around the scale value indicates the *selected* channel.

Press *left* mouse button to **activate *auto scale*** for <u>this</u> 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

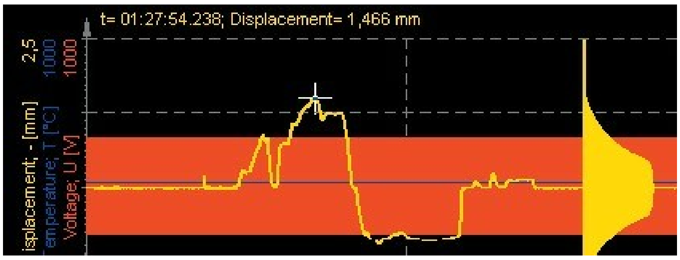
> WARNING: *This way displaying* X- *and* Y-axis *information can't be done in* Design *mode → available only in* Run *mode.*

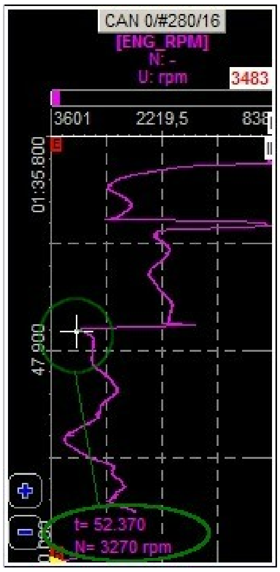
If you *move* the mouse cursor over the signal, a white cross appears, *showing* the current position of the **measurement cursor**. As soon as you *stop* moving the mouse or trackball, the cursor will *fix* its ***position*** within the signal and *move* with the ***time***. The cursor readout is displayed *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 is 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 right display gives an impression of the *cursor functionality* for **vertical** graph.

ATTENTION: *The measurement cursor is not available when the time base is set to a short time. It's only working for long time measurements.*

3.1.3.3 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:

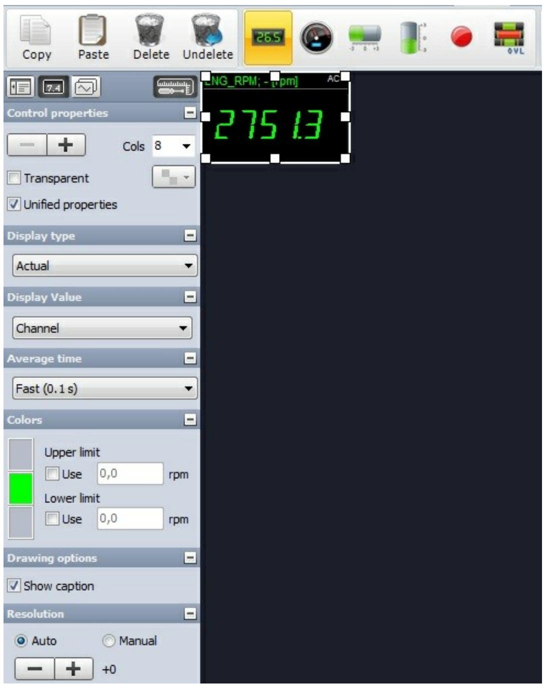
- **Control properties** — for detailed information about ***digital meter*** Control properties: *grouping, number of column, Add / Remove instruments, transparency,...*

  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*

Appearance on screen

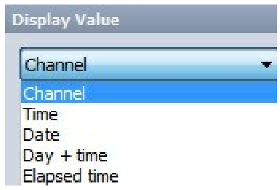
The digital meter has *only one* appearance: the ***channel name*** and ***unit*** is displayed at the top left, the ***display type*** at the top right side. The main space is used by the ***value*** letters itself.

for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

Setting

Display value

The digital meter can display *different* ***values***. Beside 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

Drawing options

Simply click on the Drawing options checkbox to **show** or **hide** the selected digital meter:

- ***caption***
- ***frame***

As a standard, the caption is *displayed*.

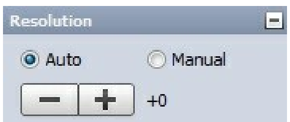
Example caption and frame ON:

Example caption and frame OFF:

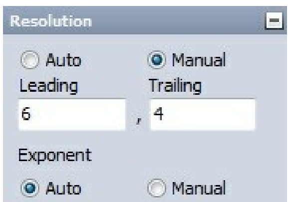
Resolution

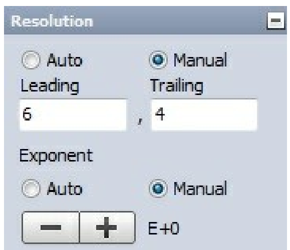
Sometimes it is *useful* to be able to add one or two *digital* of display resolution, or delete one or two, according to the type of data being displayed.

Select a meter by clicking once on it. If Auto is checked, click either **Inc**(rease) button or **Dec**(rease) button to **add** or **delete** ***digits*** to the *right* of the *decimal point* of the meter's display.

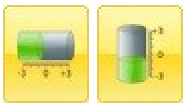


If **Manual** Resolution is checked, instead of Dec and Inc button Leading and Trailing field is displayed to enter number of digit. Below this also Exponent section appear. When **Manual** Exponent is checked, two buttons appear:  to define exponent E in step by +3 and  to define exponent E in step by -3.





3.1.3.4 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 column, Add / Remove instruments, transparency,...*

  see → **Control properties**

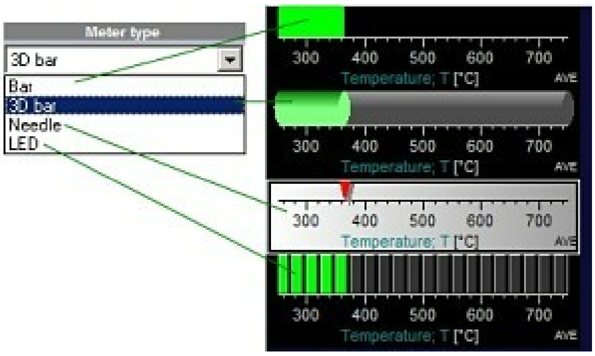
- **Horizontal / Vertical bar settings** — typical settings (same for ***Horizontal*** and ***Vertical bar***) are:
  - **Display type**
  - **Meter type**
  - **Average time**
  - **Upper / Lower limit**
  - **Minimum / Maximum shown value**

- **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 Meter Type drop down list.

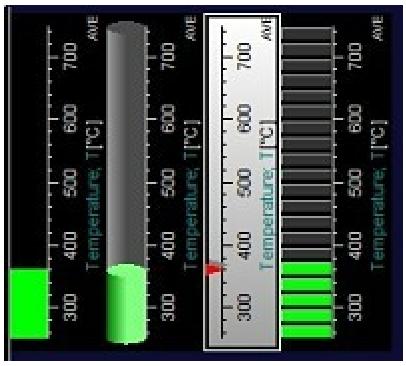
They all contain the ***channel name***, ***display type*** and ***unit*** beside the *measurement* ***values***.



All bar graph basic *appearances* can be in:

- ***horizontal*** - Horizontal bar (*see* above)
- ***vertical*** format - Vertical bar (*see* on right)

The orientation depends on the instrument *type* selected by the instrument *icon* in Design tool bar.
Changing the orientation is *not possible*; you have to select the *right* instrument when you click on the icon.



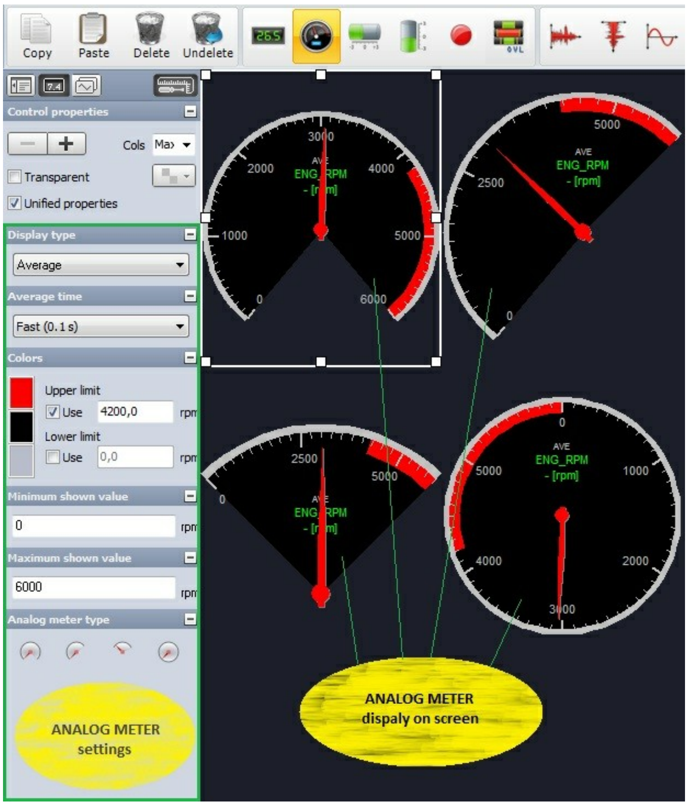
for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

3.1.3.5 Analog meter

The **Analog meter** is designed to give a *graphical indication* of the ***current value*** of a *signal* within the defined display range.

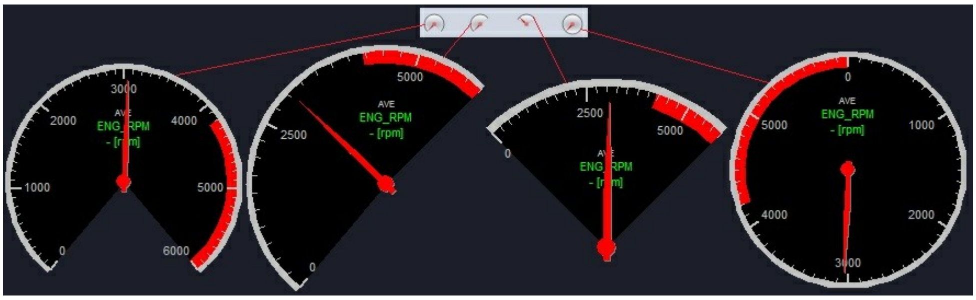
When you select an *Analog meter* in the design or run mode, following settings will appear on left and right part of the screen:

- **Control properties** — for detailed information about ***Analog meter*** Control properties: *grouping*, *number of column*, *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**



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:

1. *full circle* instrument (*see* above on the right most instrument)

2. *compass* instrument (appearance *see* right)

   To switch to this form **Compass** box must be checked:

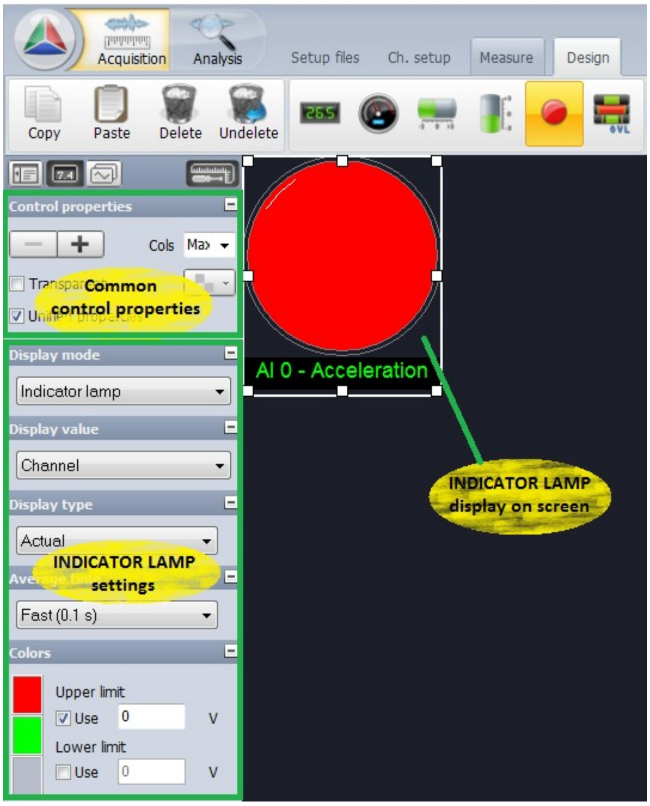
for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

3.1.3.6 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:

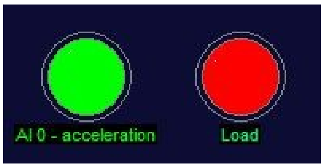
- **Control properties** — for detailed information about ***Indicator lamp*** Control properties: *grouping, number of column, Add / Remove, transparency,...*
  see → **Control properties**
- **Indicator lamp** Display mode **settings** — typical setting for ***Indicator lamp*** *Display* mode are:
  - **Display mode**
  - **Display value**
  - **Display type**
  - **Average time**
  - **Upper / Lower limit**
- **Discrete Display** Display mode **settings** — typical setting for ***Discrete Display*** *Display mode* are:
  - **Display mode**
  - **Discrete values**
  - **Options**
- **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 *Temperature* drops below 40°C, the lamp becomes green. As soon as the *Temperature* grows above 40°C, 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

The **Indicator lamp** instrument have two basic Display mode:

- **Indicator lamp**

  Shows the *state* of digital signals and to *supervise* the *value* of analog signal in a true / false way.

- **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.

Indicator lamp Display mode **setting**

Display value

Possible **Indicator lamp** settings depend from selection in this field.

For **Indicator lamp** you can select three basic **Display value**:

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.

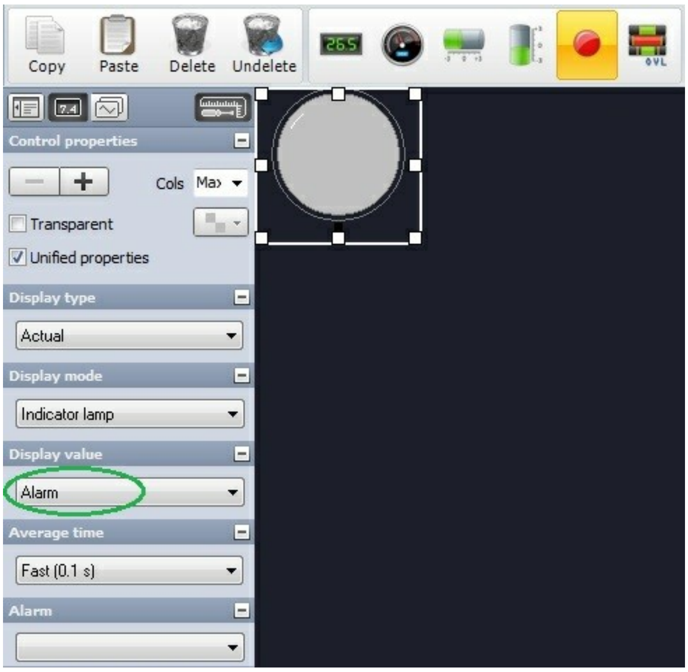
For Channel option you be able to set (*see picture above*):

- **Display type**
- **Average time**
- **Upper / Lower limit**

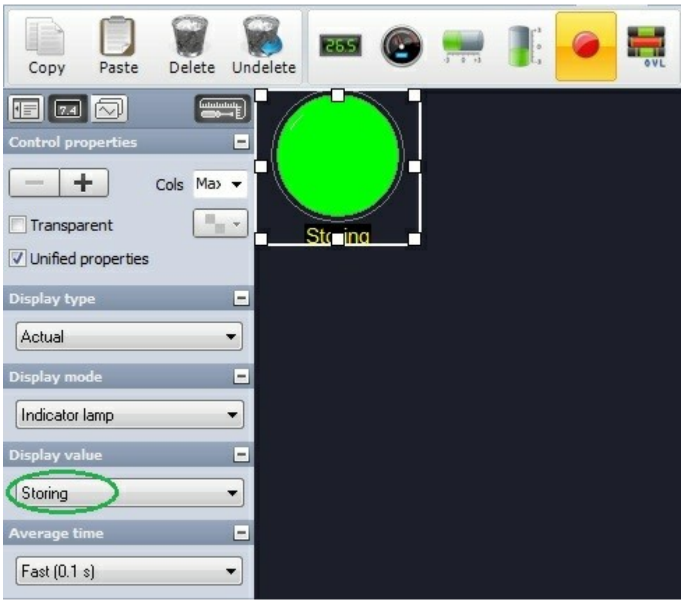
2. Alarm will show the *state* of alarm signals

For Alarm option you be able to set:

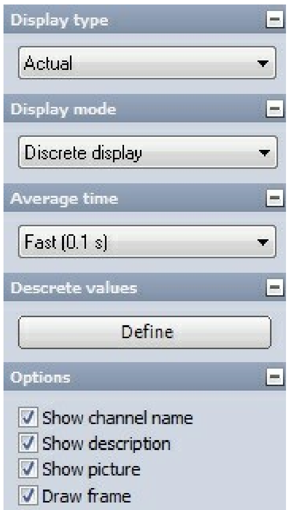
- **Alarm** from list; alarms are defined in Channel setup → Alarms tab which is visible if alarm option is *selected* in channels setup
- **Display type**
- **Average time**



3. Storing show the *state* of storing data. If data is *stored*, the lamp will go red, otherwise it will stay green



Discrete Display Display mode **setting**

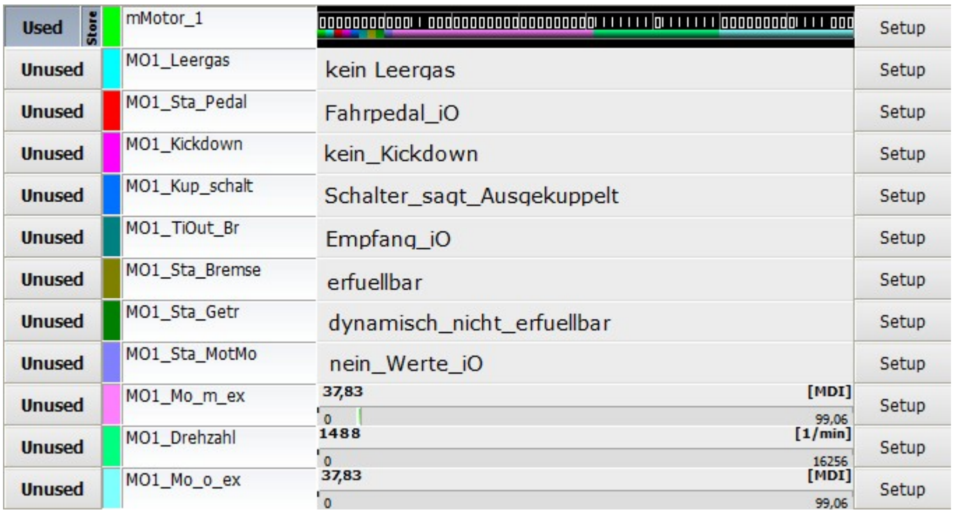


The **Discrete Display** instrument *show* ***defined discrete value***.

For **Discrete display** mode you be able to set (*see picture on right*):

- **Display type**
- **Average time**
- **Discrete values**

Let's take few CAN channels which have defined *discrete channels*. There are few channels which describes 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.



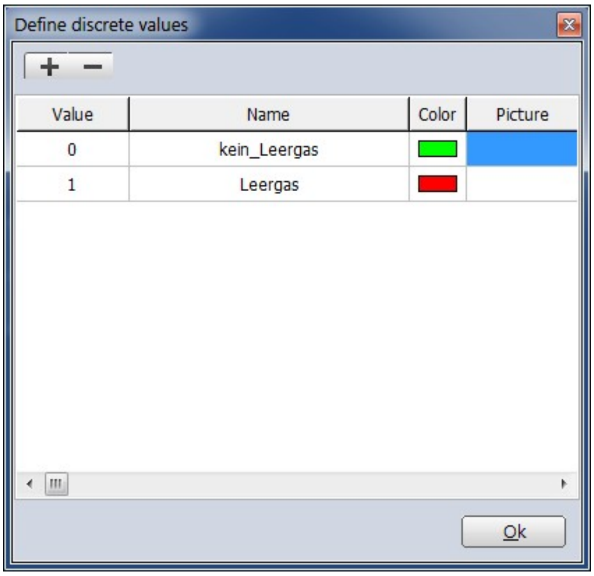
Now let's create a display with few controls to monitor the *position* of the *clutch* and *gas pedal*. So let's create a **discrete display** and put the MO1_Leergas in it. Since discrete values for this channel is *already defined*, we see already the status of the *channel*. The *bar* and *recorder* shows 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 [+] button you can *add* another value, and with [−] button delete value (table row).

Option

With checking appropriate box you can *change* **Discrete Display** ***appearance*** on online screen.

- All options switched *on*.

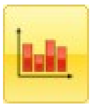
- Show channel name option shows the *channel* name caption. If it is switched off, it will *not* display the caption above.

- 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.

3.1.3.7 2D graph



The DEWESoft **2D graph** *shows* the drawing of any matrix channels.Some typical examples are FFT created from *math channels*, classification and others. It also replaces p-v scope from version 6.

When you select 2D 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*,...

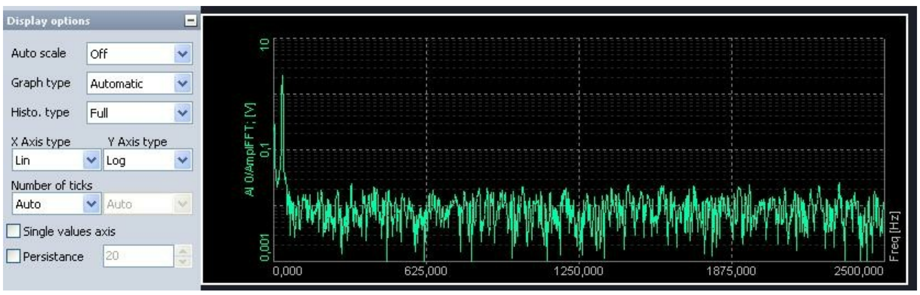
  see → **Control properties**

- **Drawing options** selects which parameters will be shown on the graph

  Available ***appearance*** setting in Run *mode* is:
  - **Axis scale/auto scale** (Common instruments tools)

- **Channels selector** for detailed information about assigning / reassigning channels →

  see → **Display settings**



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

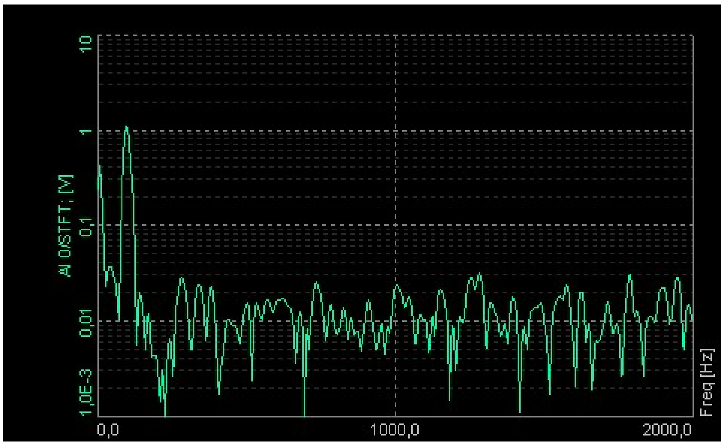
In short, 2D graph can show *any* array channel created by DEWESoft. In fact, with adding matrix channels in version 7 2D graph will replace more and more displays in the future.

Properties

There are several properties which can be set to 2D graph.

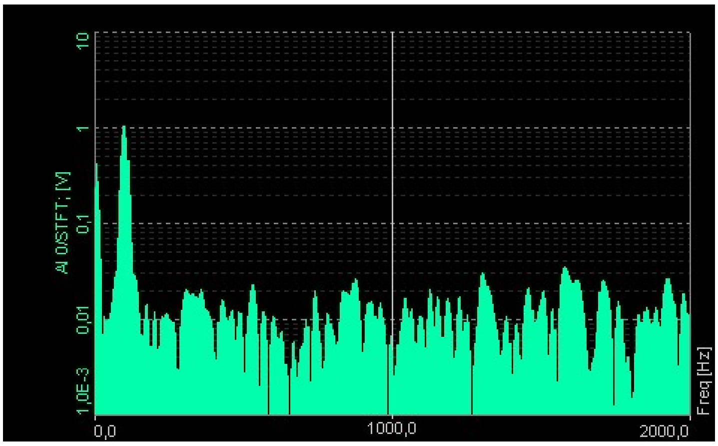
Autoscale - will automatically scale y axis

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*.

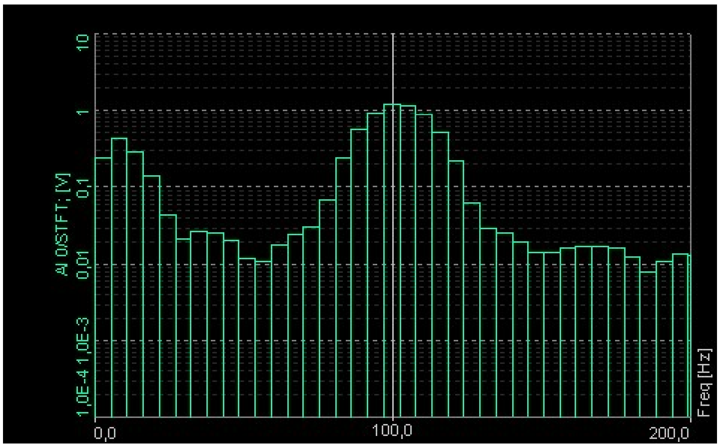


Line graph

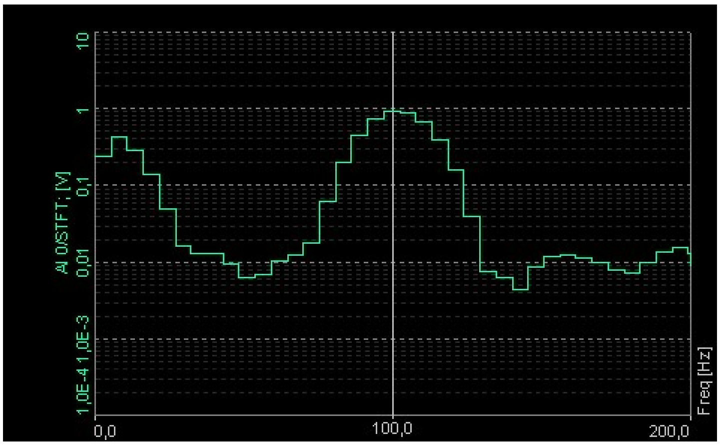
Histo. type - for histogram type, we can define to either fill the bars with Full option, or to draw empty bars with Empty option or to simply draw the Line at the top for a very classical instrument look.



Full histogram



Empty histogram



Line histogram

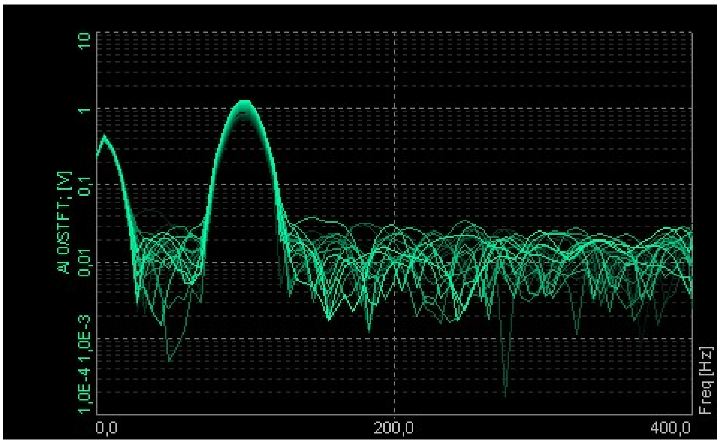
X Axis type - can be either *linear* or *logarithmic*

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

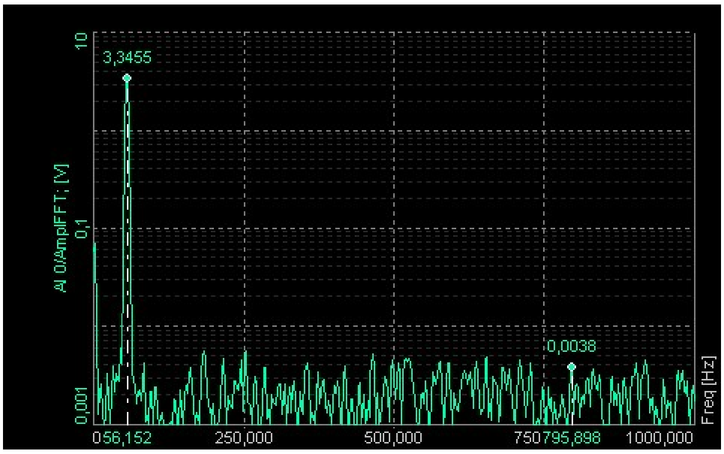
Persistence - will slowly *fade* the old data on the graph. We can define number of old arrays to be shown. The larger the number, more history will be seen.



Persistence graph

Cursor operations

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 value of certain point *above* the marked point. All points can be *removed* by pressing the right mouse button.

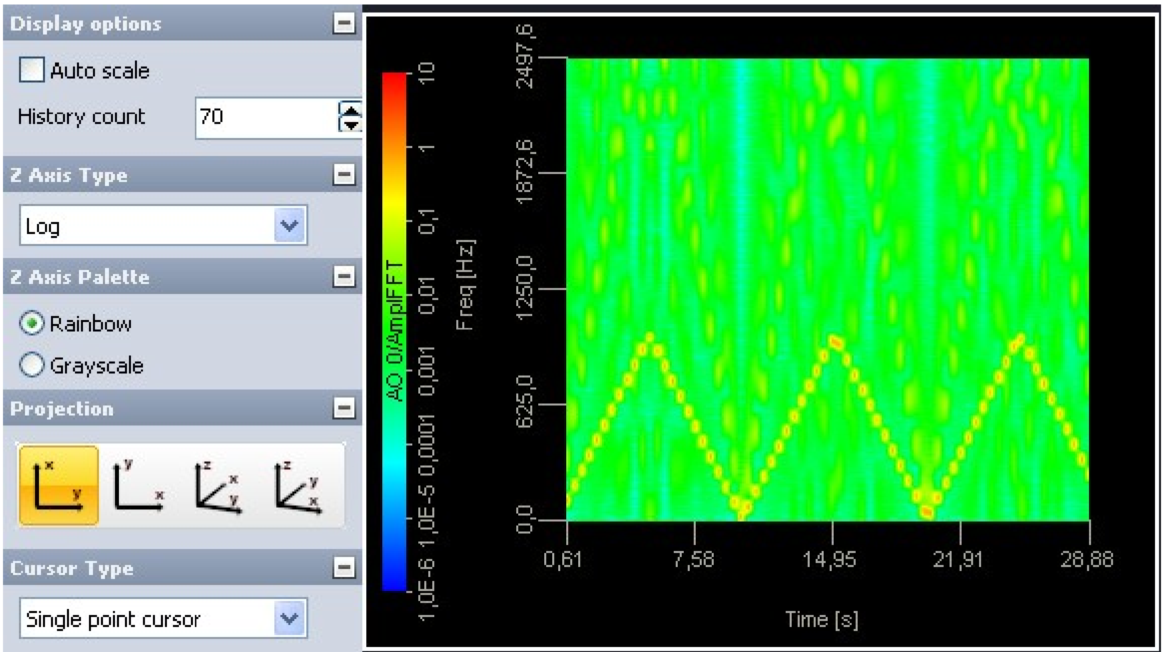


3.1.3.8 3D graph

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 thermovision data.

When you select **3D graph**, following settings will appear on left and right part of the screen:

- **Control properties** for detailed information about about Control properties: *grouping*, *number of column*, *Add / Remove*, *transparency*,...
  see → **Control properties**
- **Drawing options** selects which parameters will be shown on the graph.
  Available ***appearance*** setting in Run *mode* is:
  - **Axis scale/auto scale** (Common instruments tools)
- **Channels selector** for detailed information about about assigning / reassigning channels
  → *see* → **Display settings**



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 plugin).

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.

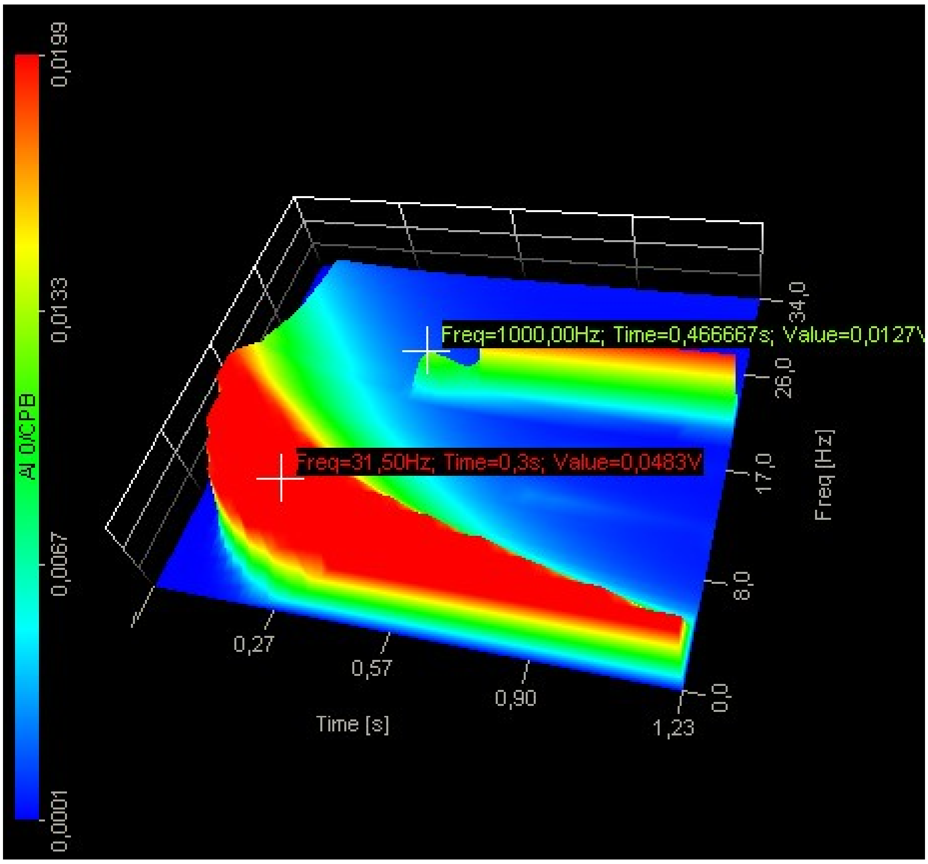
The Projection of the axis can be changed. First icon (x up, y right) is *planar* view and is mostly used when time based data is shown like FFT history, for example. The second one (x left, y up) is useful when displaying *matrix* channels like *rainflow count* or *thermovision picture*. Order tracking is in between, some users prefer first while another persons prefers second way. 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.

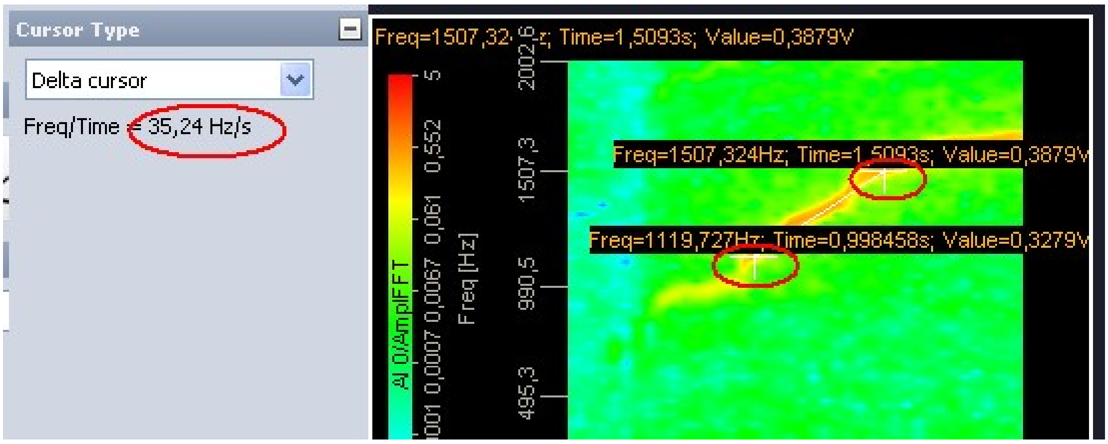
Cursors

Moving the mouse button around on the graph will *place* the cross hair cursor on the nearest point on the graph.

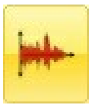
When clicking on the point for a short period of time (long click will rotate the display), the cross hair will be *held in place* and the *value* of all three axis will be *displayed* near the cursor. All the cursors can be *removed* with right mouse click.



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.



3.1.3.9 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**
  - **Y scale divisions**
  - **Display type**
  - **Single value axis**
  - **Time axis type**

  Available ***appearance*** setting for ***Recorder*** *instrument in* Run *mode* are:
  - **Time axis scaling**
  - **Y-axis scale/auto scale**
  - **X-/Y-axis information**

- **Channels selector** — for detailed information about about assigning / reassigning channels to/from ***Recorder***

  → *see* → **Display settings**

Appearance on screen

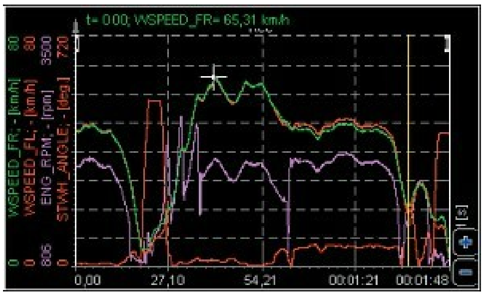
The **Recorder** element offers all *important* information:

channel name(s)

unit(s)

time information

zoom functions...

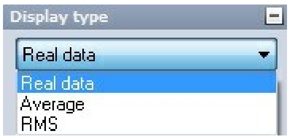


for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

Settings

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**.



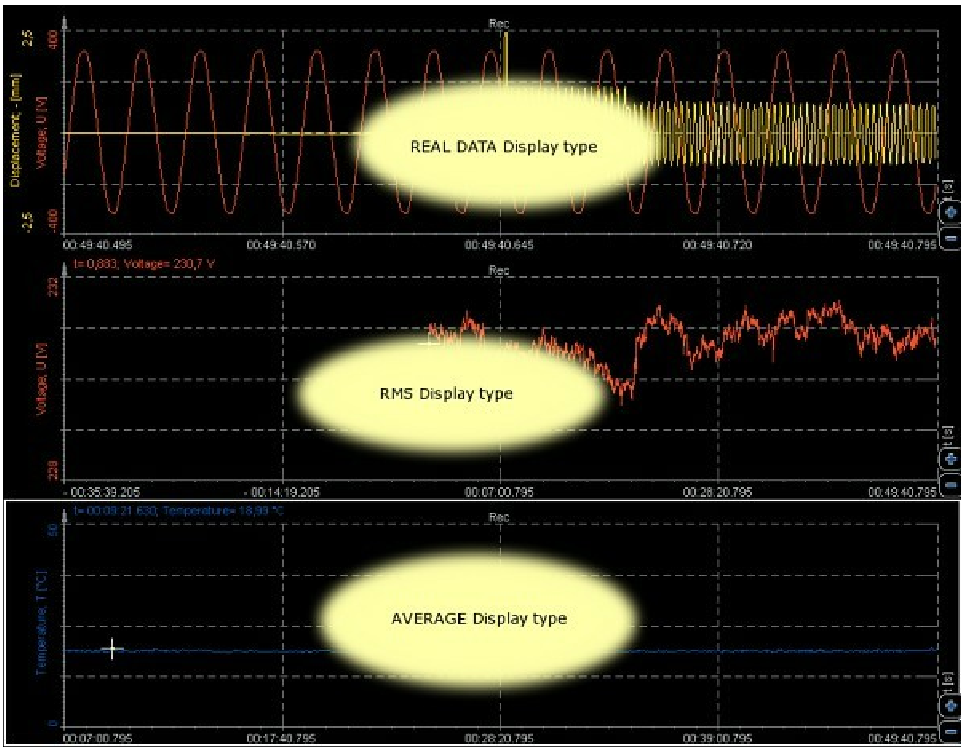
This is useful when monitoring AC signals, which are going to look like a *solid band* when a *long* duration is shown - not very useful to look at (unless you're just looking for overall amplitude envelopes, or obvious drop-outs). Changing the display type for this graph to **RMS** will show a more *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.

Just have a look to the following screen to see the *difference* between the three display types.



Use the **Show events** checkbox 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** checkbox to *interpolate* the *asynchronous channels*. If 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** checkbox to display *individual sample points* in analyse mode.

Single value axis

The **Single value axis** checkbox 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.

3.1.3.10 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:

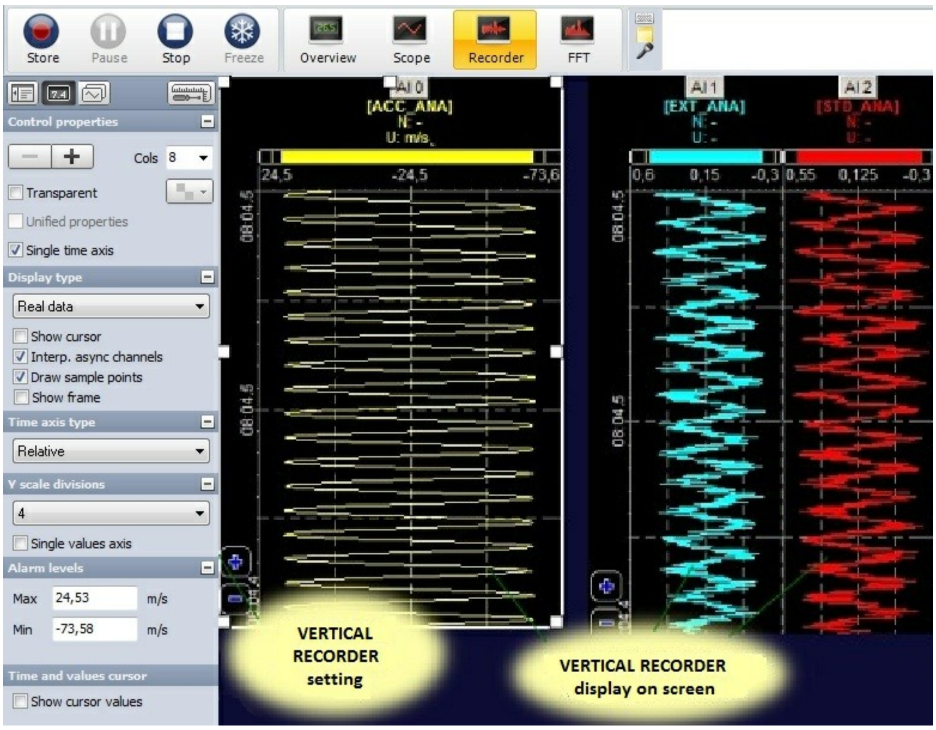
- **Control properties** — for detailed information about ***Vertical recorder*** Control properties: *grouping, number of column, Add / Remove, transparency,...*

  see → **Control properties**

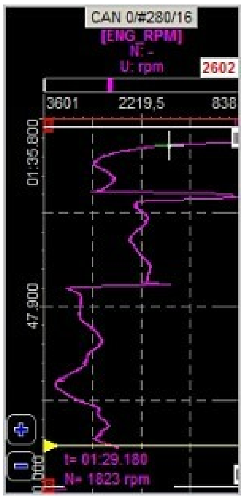
- **Vertical recorder settings** — typical ***Vertical recorder*** setting are:
  - **Single/Multiple Time axis**
  - **Display type**
  - **Time axis type**
  - **X scale divisions**
  - **Alarm levels**

  Available ***appearance*** setting for ***Vertical recorder*** *in* Run *mode* are:
  - **Time axis scaling**
  - **X-/Y-axis information**
  - **X-axis scale/auto scale**

- **Channels selector** — for detailed information about assigning / reassigning channels to/from ***Vertical recorder*** → *see* → **Display settings**



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*.

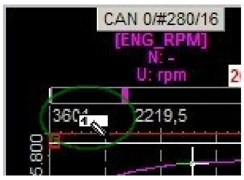
for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

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.*

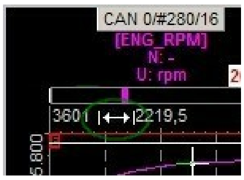
If you want to *change* the **X-axis** for *any* input, the recorder offers *two* possibilities:

- ***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. If you have reached 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).

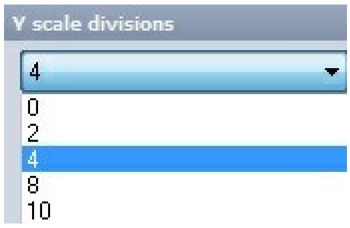
Settings

Display type

Display types of Vertical recorder are same as by Recorder, except Time axis type is *vertical*.

for detailed information about ***Display types*** *see* → **Recorders Display type**

X scale divisions



The **Vertical recorder** offers X scale divisions. You can select the number of X scale *divisions* from one to ten (=dashed white line which divide graph area) from 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 <u>above</u> the grid.

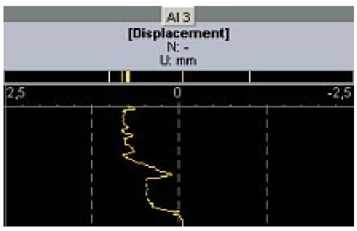
As a *standard*, the alarm values are set to the *minimum* and *maximum* ***range*** for the appropriate channel. If you want to use own limits, just *enter* the alarm level according to your *requirements*.

For example, if we want to supervise the signal within ±1 mm, we enter the following settings: → *see* above on right

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.

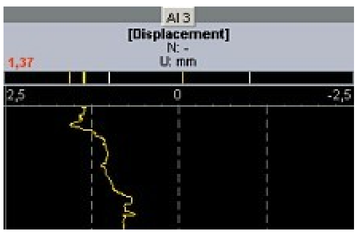
Example 1

The current signal is *within* the defined *limits* of ±1 mm, indicated by the two *white* markers.



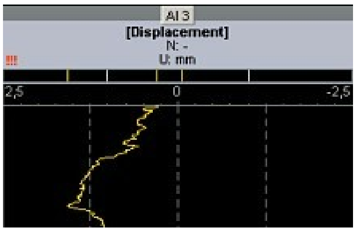
Example 2

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.

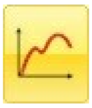


Example 3

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.



3.1.3.11 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**

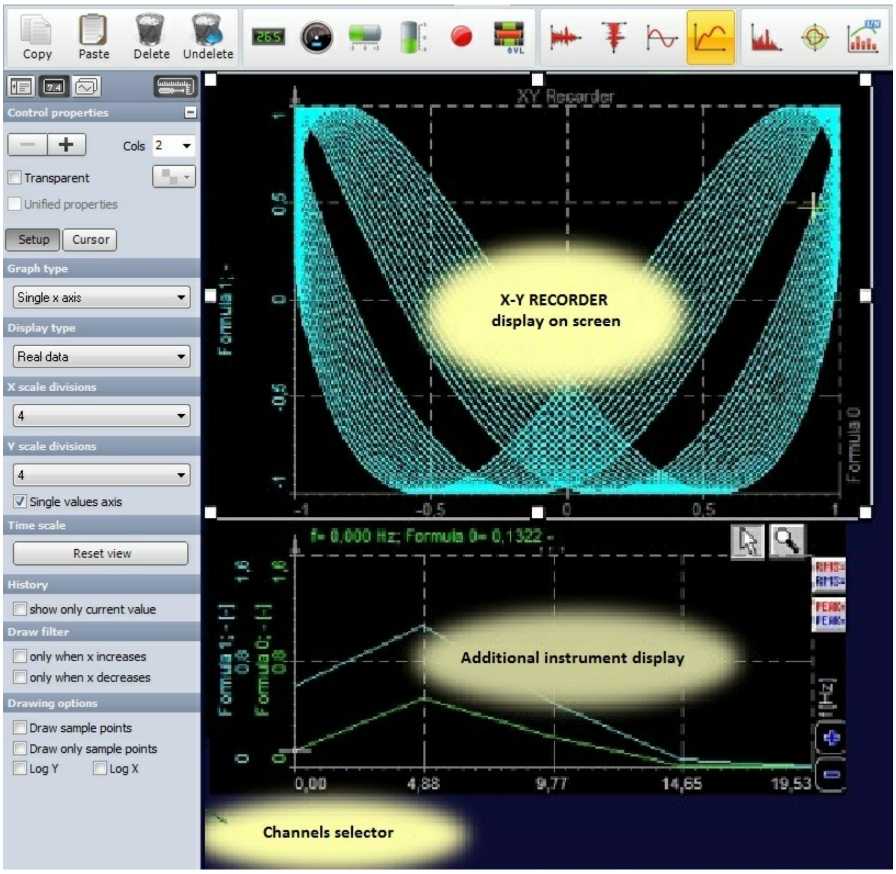
- **X-Y recorder settings** — typical ***X-Y recorder*** setting in Setup tab are:
  - **Graph type**
  - **Display type**
  - **X and Y scale divisions**
  - **Time scale**
  - **Drawing options**
  - **Draw filter**
  - **History**

  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*** →

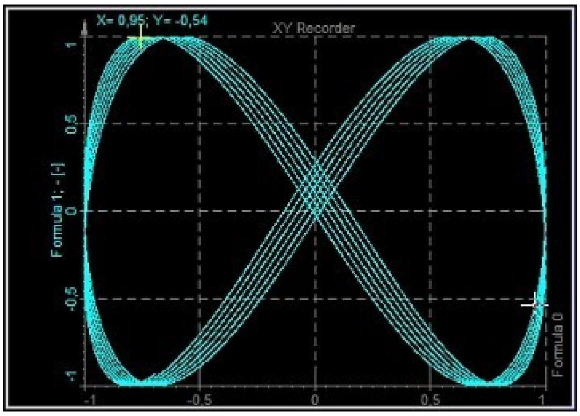
  see → **Display settings**



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* three ***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**

XY Recorder Setup

Graph type

The Graph type knows *three* modes:

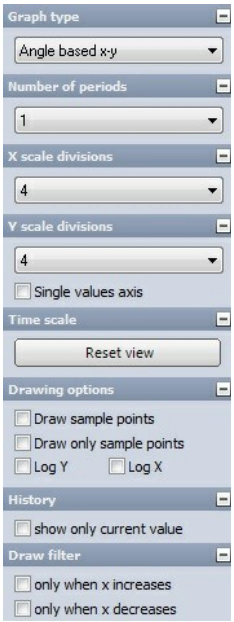
- **Single x-axis** — Allows <u>only</u> one channel **x-axis**. The ***first assigned*** channel is *used* for the x-axis, up to ***4 following*** channels the y-axis.

- **Pairs of x-y** — Allows 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.



For this Graph type following **Setup** option can be set (*see* picture on right):

- **Display type** (different as for other Graph type option)
- **X scale division** and **Y scale division**
- **Single value axis**
- **Drawing options**

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.

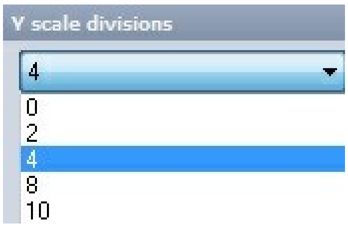
Select any graph by *clicking* on it, and then select this value from the Display type drop down list.

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.*

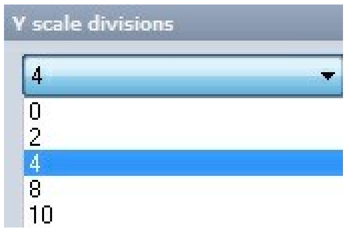
<u>Angle based x-y</u> graph

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 drop down list.

X and Y scale divisions



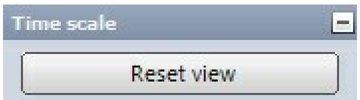
The **X-Y recorder** offers X scale divisions. You can select the number of X scale *divisions* from one to ten (=dashed white line which divide graph area), which can *differ* between ***X*** and ***Y axis***.



The **X-Y recorder** offers Y scale divisions. You can select the number of Y scale *divisions* from one to ten (=dashed white line which divide graph area), which can *differ* between ***X*** and ***Y axis***.

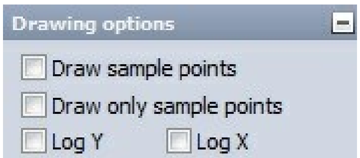
The **Single value axis** checkbox can be used to *set all active* channels of an *X-Y recorder* ***grid*** to *only one* **Y-axis**. If set, all channels will *use* the *same* ***scaling*** and as a visual result, there will be *only one* axis with ***values*** *left*.
This function is very helpful 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 <u>limited</u> **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.

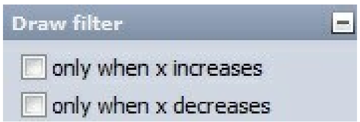
Drawing options



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* → Reference Guide → **Properties of the Fourier transform**

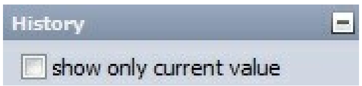
Draw filter



Especially for *run-up tests* you should use the **Only when x increases** checkbox 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.

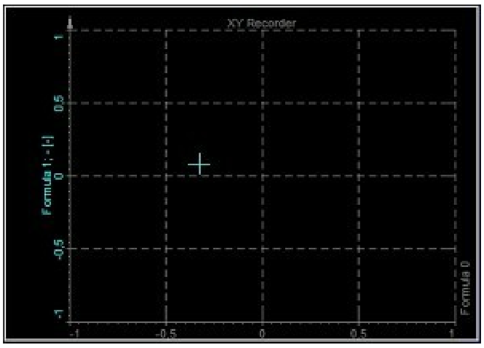
History



When the signal is changing very *strong*, it may be helpful to click the **Show only current value** checkbox 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.

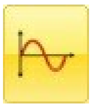
Example: appearance ***X-Y recorder*** with enabled show only current value



XY Recorder **Cursor**

By selected Cursor tab the **cursor readouts** for *each* Y channel in CURRENT POSITION section is displayed.

3.1.3.12 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*,...

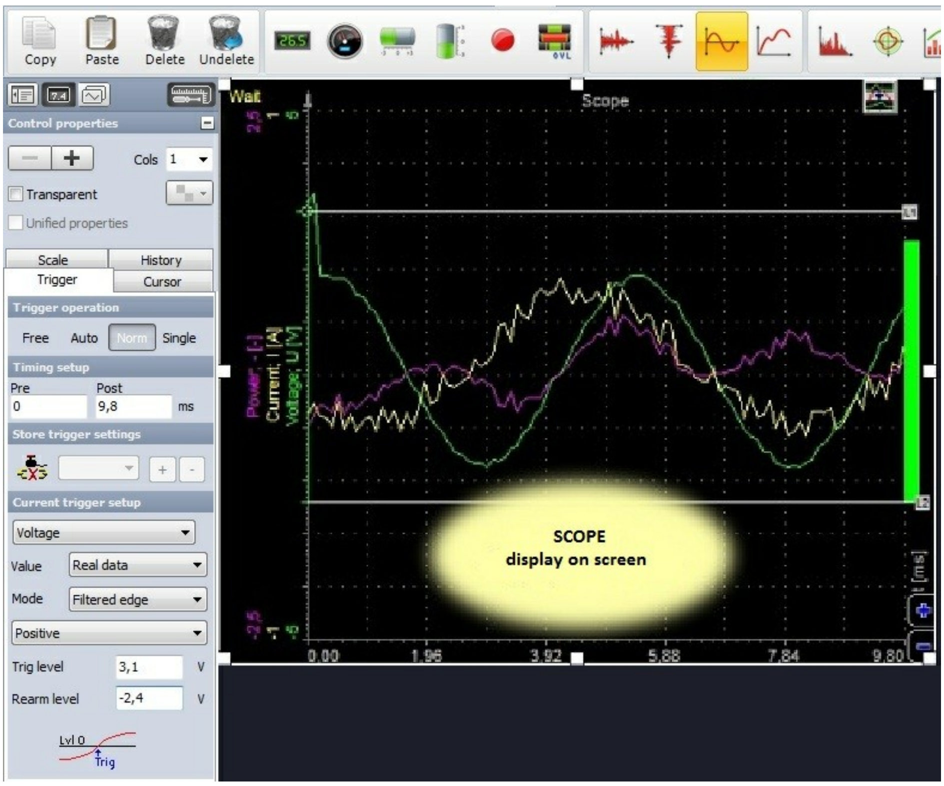
  see → **Control properties**

- **Scope settings** — typical setting for ***Scope*** *instrument* are in three main groups:
  - **Trigger**
  - **Cursor**
  - **Scale** (selected channel scaling and offset)

  and one dependent subgroup: **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:

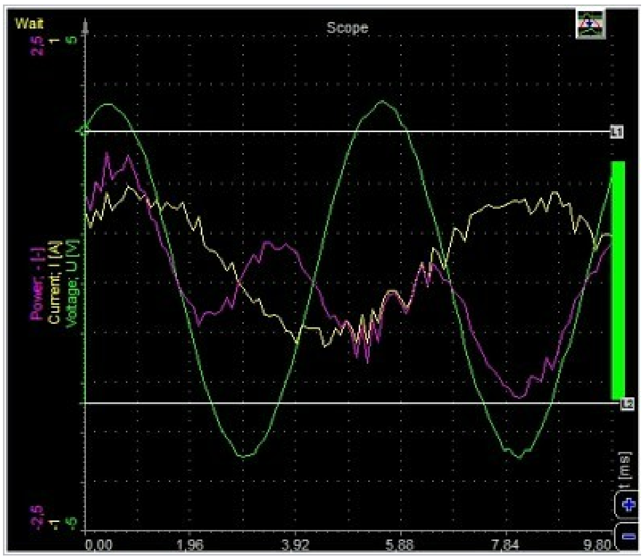
channel name(s)

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.

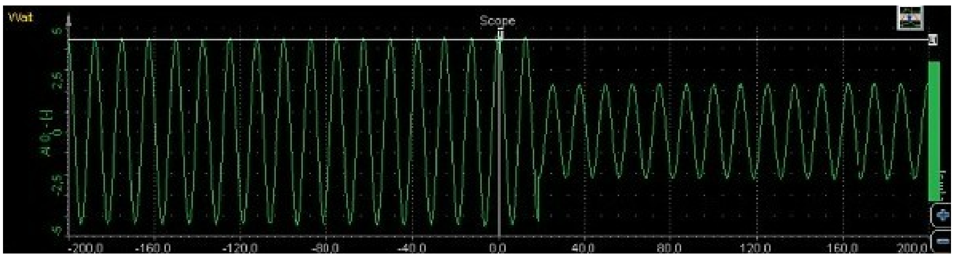


for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

Run mode Zoom (additional appearance setting)

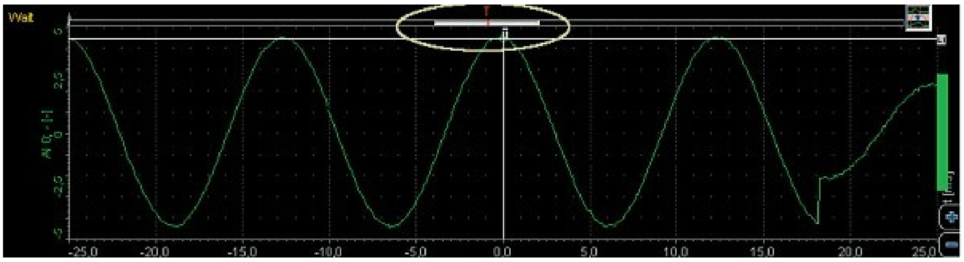
At the top right above each graph in **Norm** or **Single** Trigger mode you maybe have already noticed a small icon.

Pressing it *enables* / *disables* the ***zoom*** view *during acquisition*. Up to now, when you press the blue and buttons (*see* above) 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 button to ***zoom in*** (or 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* through the *whole* acquired data of the *current* ***trigger*** *shot*.

3.1.3.12.1 Setting - display tools

The **Scope** instrument typical *settings* include three main groups:

- **Trigger** → **Free run** - settings of triggered acquisition
  Auto
  Norm
  single
- **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

and one dependent subgroup **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

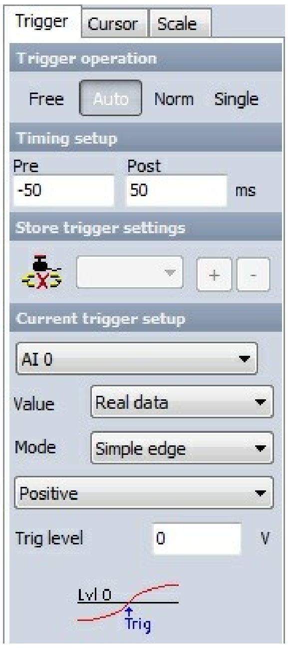
DEWESoft knows four types of Trigger operation → ***triggered acquisition***:

- **Free** run *All* values are *displayed*, no trigger active.

  There are not 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.

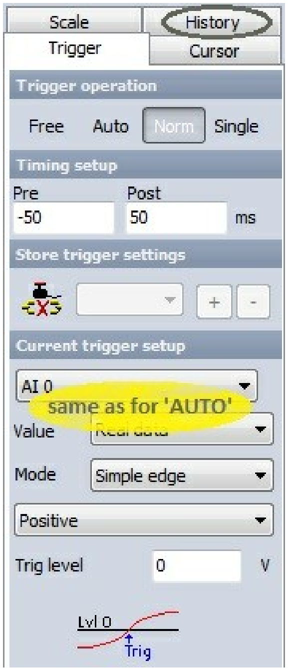


For this type of Trigger operation can be set:

- **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** **Pulse-Width** **Filtered edge** **Window and pulse-width** **Window** **Slope**
- **Store trigger settings**

- **Norm**

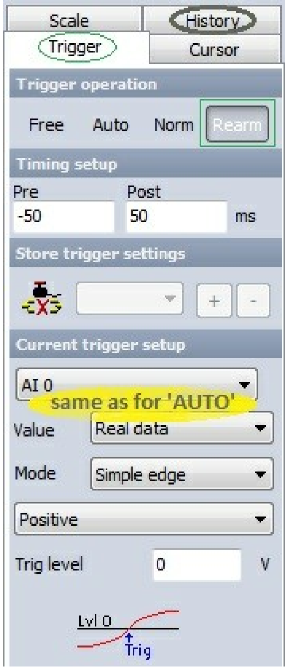
The **normal trigger** *displays* only values if the *trigger condition* is true.



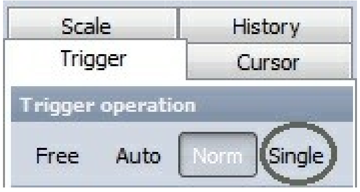
For this type of Trigger operation can be set the *same* setting as for **Auto** trigger → *see* above.

When the Norm (or Single) trigger is selected, another tab appears → the History.

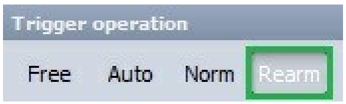
- **Single** This function can be used to *acquire* **single events**.



After *selecting* **single** button:



and event appear this button changes to **Rearm** (*see* also right):

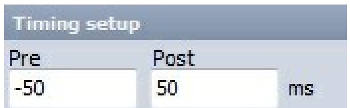


Press it to *get another* single shot event.

For this type of Trigger operation can be set the *same* setting as for **Auto** trigger → *see* above.

When the Single (or Norm) trigger is selected, another tab appears → the History.

Timing setup



The Timing setup can be used to *define* the displayed **Pre** and **Post** trigger ***time*** in milliseconds.

Hint 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* mouse button pressed and *move* the line to the desired position.

The *time window* can also be *changed* using the + and − buttons at the *right bottom* of each graph.

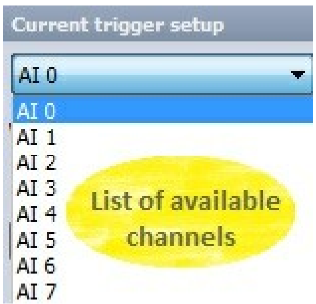
Current trigger setup

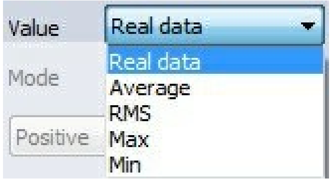
The **trigger** conditions for **Auto**, **Norm** and **Single *data*** trigger are the *same* and *work* in the same way than described in Using trigger to start and stop recording.

for detailed information about ***using triggers*** *see* → **Using Triggers to start and stop recording**

1. *select* the desired **channel**

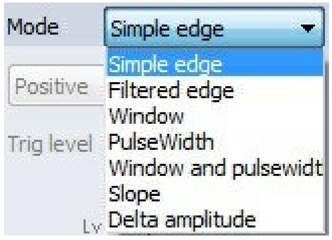
First of all you have to select the *desired* channel out of the drop down list. It displays all *available* **channels**.





2. *define* the Value

Select the **Real data**, **Average** or **RMS** from the drop down list.



3. *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.

4. *setup* ***other***

These settings (e.g. **Slope**, **Trigger level**, **Rearm level**, **Pulse time**,...) depend on *selected* trigger ***type*** in Mode field.

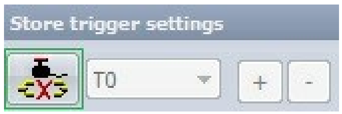
for detailed information about ***trigger mode*** and ***set up triggers*** *see* → **Trigger Setup**

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.

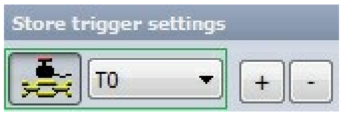
Store trigger settings

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.

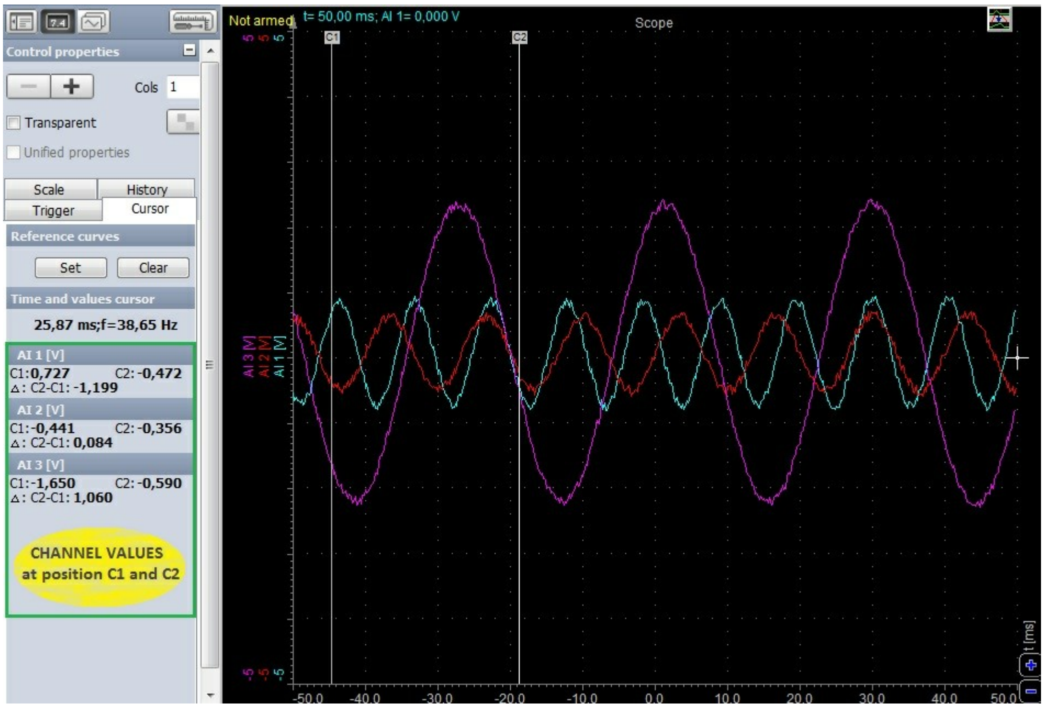
The  buttons can be used to *delete* selected *additional* conditions.

WARNING: *As long as the Link storage trigger button is not pressed, the data is only displayed - not stored!*

Cursor

Cursor measurements

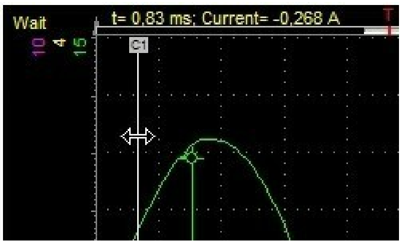
DEWESoft offers also **Cursor measurements** function to *show* the ***cursor readouts*** for *each* channel within the selected scope. Select the Cursor tab to activate the cursor measurement:

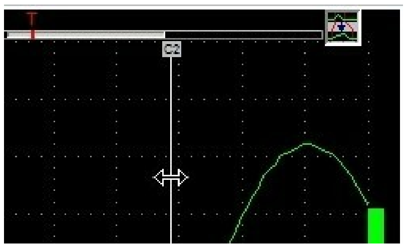


The *scope* offers two *measurement* **cursors** for the *active* graph, which can be positioned in two ways:

- ***Move*** the mouse cursor to the *first* position of interest. ***Click*** the left mouse button - cursor C1 will appear. Move 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.

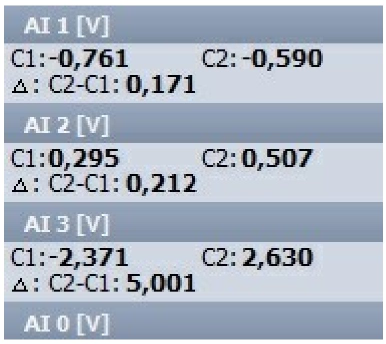
- ***Drag*** the *first* cursor *from left* side of the graph to the position of interest and the *second* cursor *from the right* side of the graph. In both case cursor appearance on *measurement cursors .jpg*must change to ⟺:



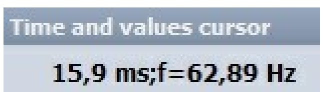


The settings menu at the *left* side of the screen now displays:

- 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



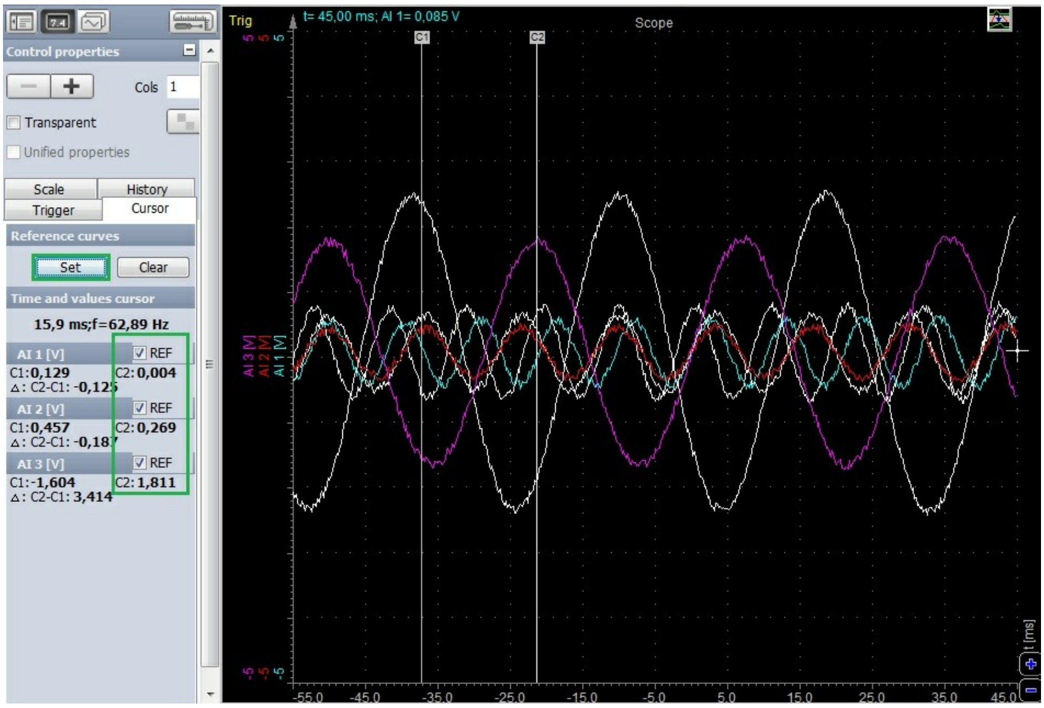
WARNING:	*The freeze function is not working in the scope!*

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:



and in the cursor readouts for *each channel* REF checkbox appear:

To *hide* the reference curve *uncheck* the **REF** checkbox for the *desired* channel(s). This will *keep* the reference curve within memory. To display it *again* just check the **REF** checkbox *again*.

Press the **Set** button at any time to *update* the reference curve to the ***current*** triggered ***data***.

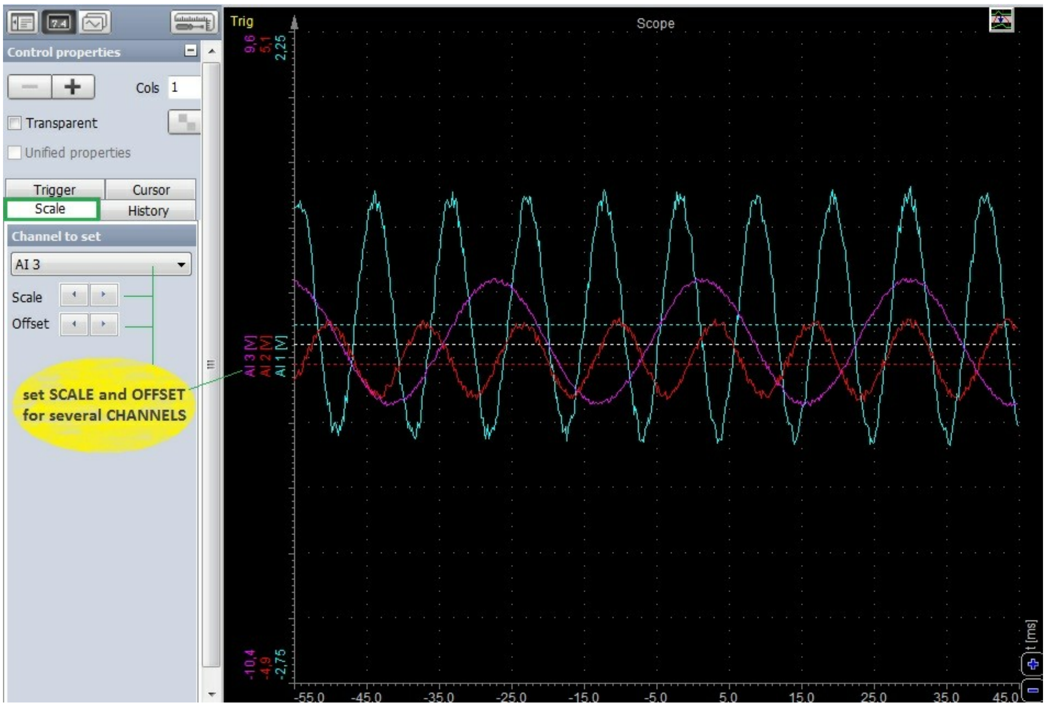
To *remove* the reference curve permanently press the **Clear** button. This will *remove* the reference curve data from *memory*. To define a *new* reference curve press the **Set** button *again*.

Scale

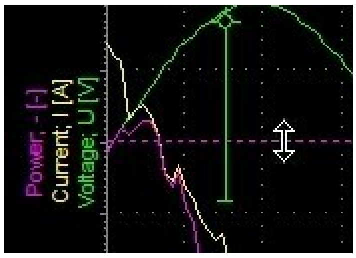
The **Scale** function can be used to *change* displayed ***offset*** and ***scaling*** of signals.

First select from Channel to set drop down list several channel to set his Scale and / or Offset:

Press the Up / Down **Scale** buttons to '***zoom***' the amplitude *in* or *out* and Up / Down **Offset** buttons to ***move*** the signal within the grid:



NOTE: *These settings have no influence on the acquisition resolution or accuracy. It's just a display feature!*



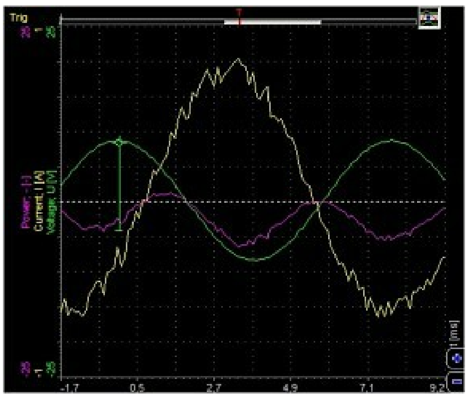
A dashed *horizontal* line indicates the current **offset *setting*** of *each* channel. It has the *same* color than the channel itself. You can *use* this line also to *change* the channel **offset**. Simply move the mouse cursor over the line and the mouse cursor appearance will change to

 .

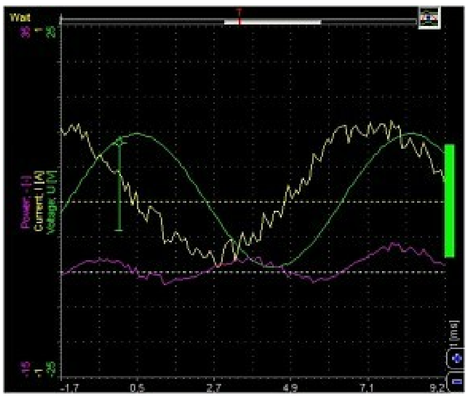
Now click (and keep mouse button pressed) and move the dashed line to the *desired offset* position.

Examples for *scaling* and *offset* functions:

Scope Scale view after scale down Power channel:



and after set negative offset for Power channel:

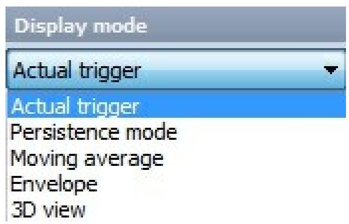


History

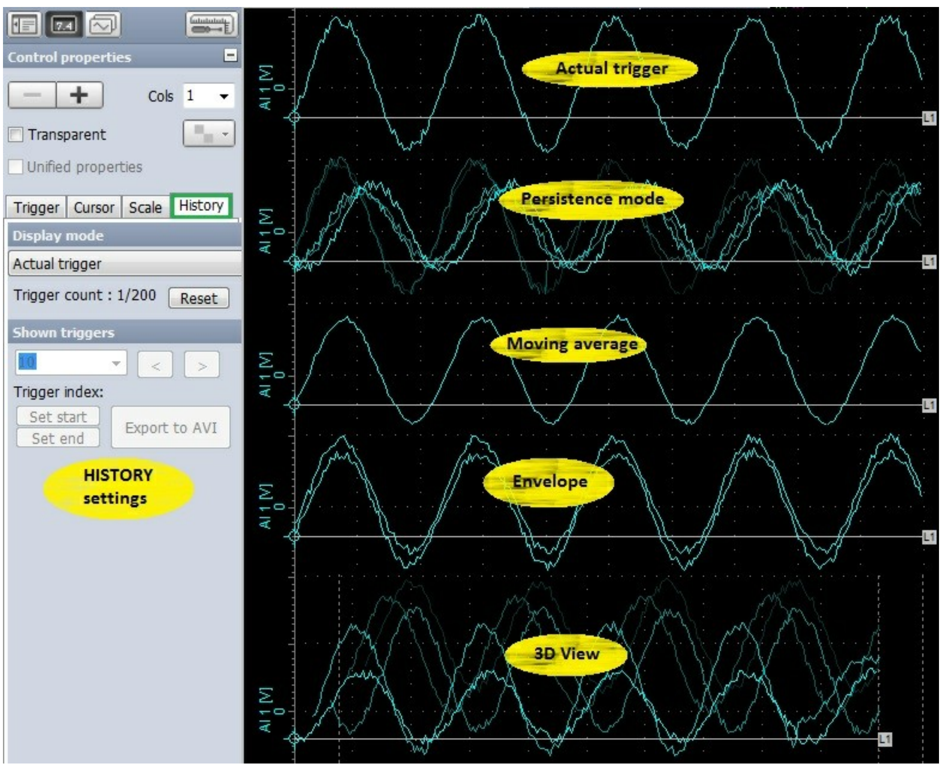
When the Norm or Single trigger is selected, another tab appears: the **History**. This feature allows you to *display* the following ***information*** - **history type**:

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).

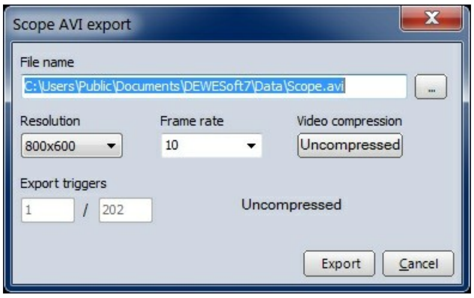
If you want to *clear* the *history memory* simply press the **Reset** button.

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** button 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.



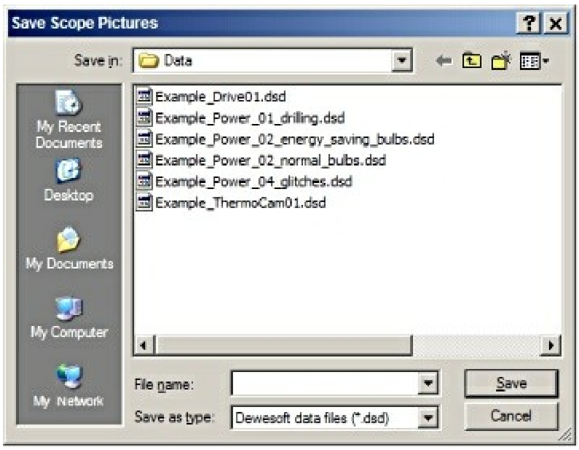
When **Export to AVI** is selected *Scope AVI export* window appear to set: File name, Resolution, Frame rate, Video compression

for information about this settings *see* → **Export screen to AVI**

and Export triggers field, in which we can define to save *only* interesting triggered events (to *reduce* the amount of data).

Select **Export** button to *export* '**avi**' data or **Cancel** button to *cancel* exporting.

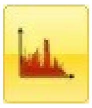
Whit **Save to file** button we call *Save Scope Pictures* window to *save* ***acquired data*** from history memory in to DEWESoft *.dsd (*DEWESoft data*) *file*:



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.

3.1.3.13 FFT Analysis



The DEWESoft **FFT** (`Fast Fourier transformation`) instrument *shows* the ***frequency*** components of *acquired* signals in *amplitude* and *frequency*.

for background information about ***FFT analysis*** → *see* → Reference Guide → **Theory of frequency analysis**

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*,...

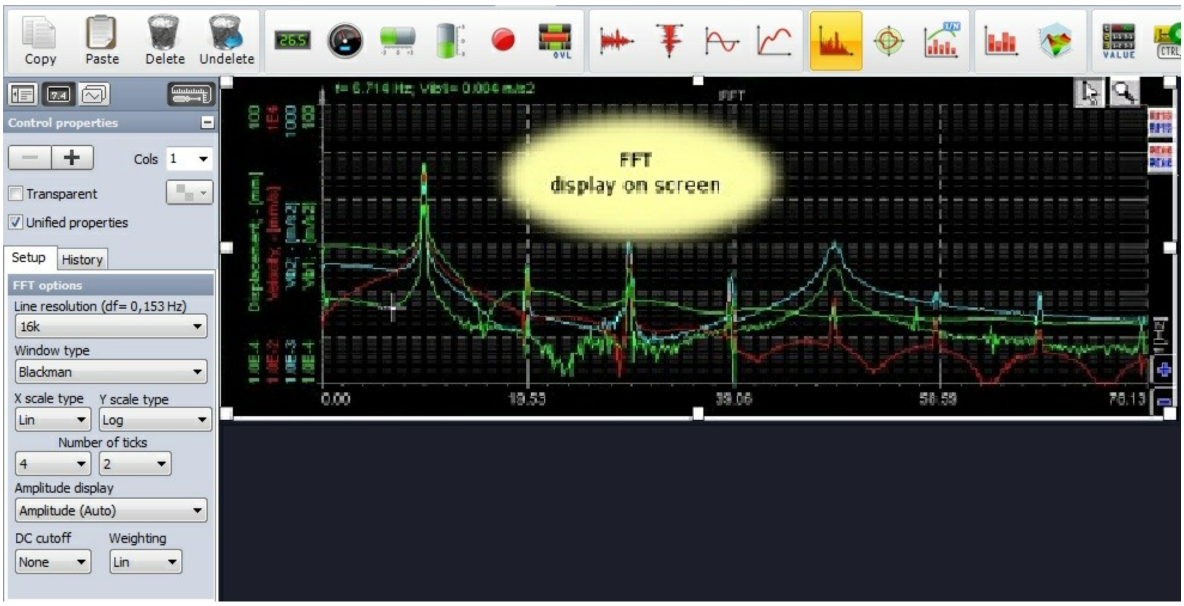
  see → **Control properties**

- **FFT settings** — typical setting for ***FFT*** *instrument* are combined in two tabs:
  - **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

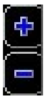
zoom functions...

for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

Changing the Frequency axis scale

Run mode Frequency axis scaling

WARNING: *This way changing* Frequency 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 **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: *This way changing* Frequency axis *can't be done in* Design *mode* → *available only in* Run *mode.*

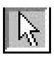
Enter values

You may *enter* the values by yourself - simply *click* on the number at either *extreme* or *minimum* (*see* left) and then *type in a new number*. This is possible when the cursor has following *appearance*:  .

When click on this cursor, *enter* field for *new* value appear (*see* left). After enter new ***min*** and/or ***max value*** press **OK** button and *new* Frequency axis scale appear on 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.

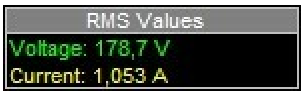
To *remove* the readout, ***click again*** on this button - cursor marker.

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.

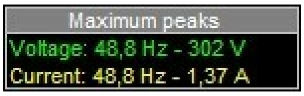
Online calculation

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*** →

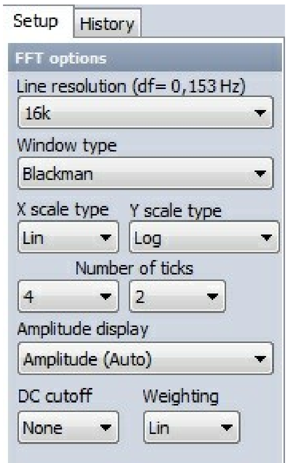


for the ***peak values*** →

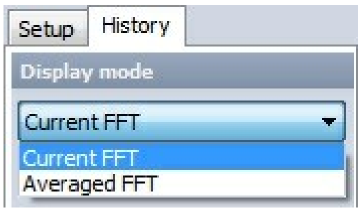


FFT Settings

Serious **FFT analysis** requires several settings to get a useful result. All these settings DEWESoft combine in two tabs:



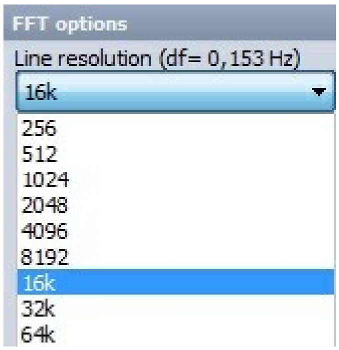
1. **Setup** In FFT Option can be set:
   - **Line resolution**
   - **Window type**
   - **X scale and Y scale type**
   - **Number of ticks for X and Y axis**
   - **Amplitude display**
   - **DC cutoff**
   - **Weighting**



2. **History** This setting define 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 *lower* frequency resolution. If we want to see *exact* frequency, we set *higher* line resolution. 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 analyses frequency by the FFT lines (5000 Hz / 1 024 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.

for detailed information about ***Line resolution*** → *see* → Reference Guide → **Properties of the Fourier transform**

Window type

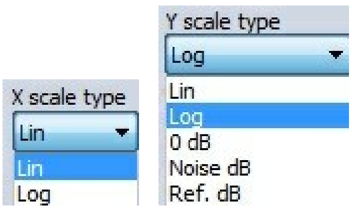
DEWESoft supports the most common **Window types** for FFT analysis. Select the *window* from drop down list according to your application.

for hints about recommended using ***Window type***
→ *see* → Reference Guide → **Theory of frequency analysis**

X and Y scale type

DEWESoft allows:

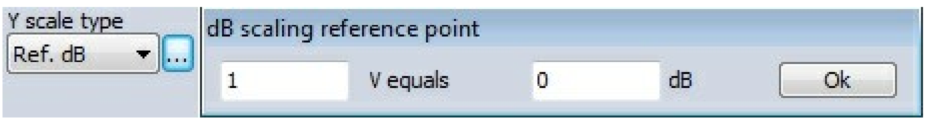
- two different ***X*** (frequencies) ***axis*** types (**Lin**ear and **Log**arithmic)
- four ***Y*** (amplitudes) ***axis*** scaling types (**Lin**ear, **Log**arithmic, **0 dB** and **Noise dB** and **Ref. dB** scaled)



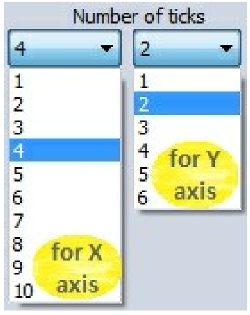
Select the *axis type* from drop down list according to your application.

for detailed information and using hints about ***X and Y scale type***
→ *see* → Reference Guide → **Properties of the Fourier transform**

When in Y scale type **Ref. dB** is selected, 'Db scaling reference point' window appear to *enter* this reference point and confirming that with **OK**.



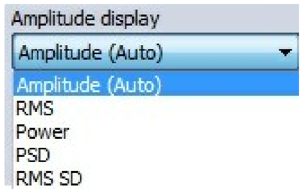
Number of ticks



You can select the **Number of ticks** from one to six **ticks** (dashed white line which divide graph area) for your FFT display, which can *differ* between ***X*** and ***Y axis***.

The first field in row is to select the ***X axis*** ticks, the second row for the ***Y axis*** ticks.

Amplitude display



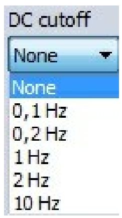
The Amplitude display section defines display in Y-amplitude axis.

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
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 *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** analyses use a **Lin**ear 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

1. Current FFT

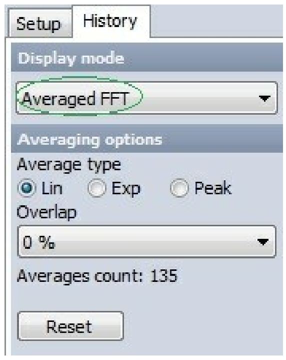
*Display **current FFT*** with settings in Setup tab (*see* above). Just select **Current FFT** from Display mode drop down list. Only this one setting is necessary for this *Display mode* type.

2. 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* right).

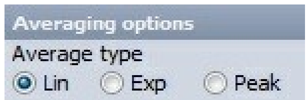
All necessary setting can be done on Averaging options section.

for detailed information about ***Averaging***
→ *see* → Reference Guide → **Theory of frequency analysis**



Average type

From Averaging options select **Average type**: **Lin**ear, **Exp**onential 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.

for detailed information and hints about ***Overlap*** → *see* → Reference Guide → **Theory of frequency analysis**

3.1.3.14 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.

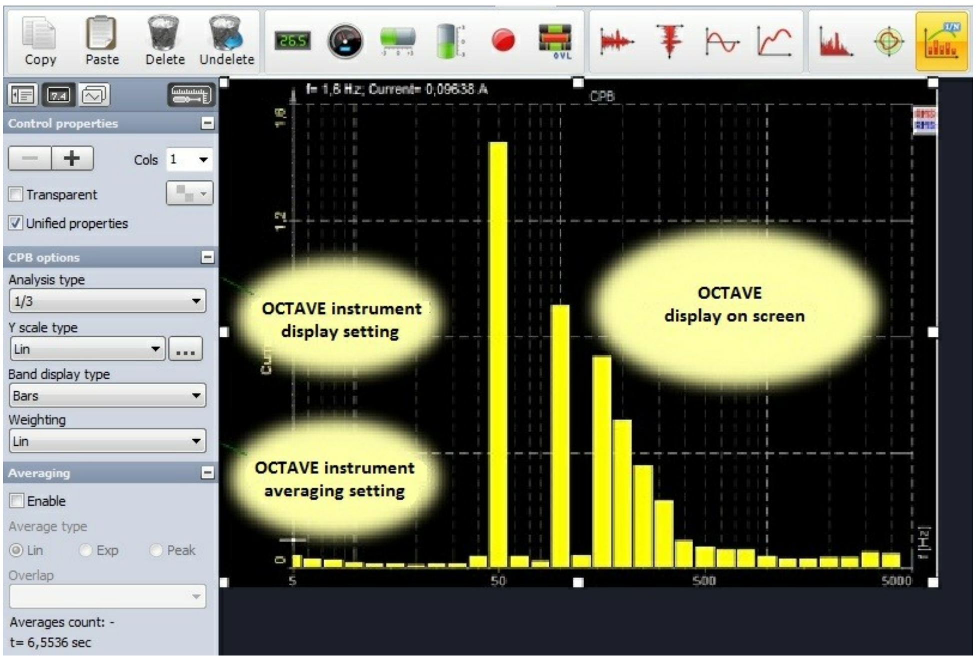
for background information about ***FFT analysis*** → *see* → Reference Guide → **Theory of frequency analysis**

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 column, Add / Remove, transparency,...*
  see → **Control properties**
- **Octave settings** — typical setting for ***Octave*** *instrument* are combined in two tabs:
  - **Display settings** — with *Analysis type, Y scale type, Band display type* and *Weighting*
  - **Averaging settings** — *Averaging type, Overlap* and *Averages number*

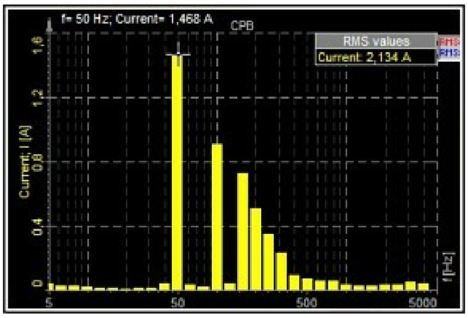
  Available ***appearance*** setting for ***Octave*** *instrument* in Run mode are:
  - **Y-axis scale/auto scale** (Common instruments tools)
  - **Online calculation**
- **Channels selector** — for detailed information about assigning / reassigning channels to/from ***Octave***
  → *see* → **Display settings**



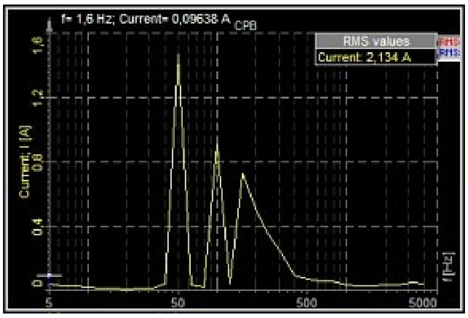
Appearance on screen

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**

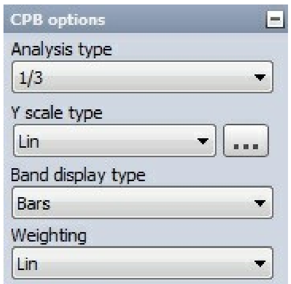
Online calculation

To keep an overview over the **Octave**, DEWESoft can calculate the ***RMS*** values *displayed signals*. To *activate* the calculation, just *click* on:

for 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:



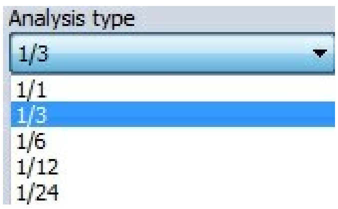
- **Analysis type**
- **Y scale type**

  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 $2^{\wedge}(AnalysisType)$ 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*.

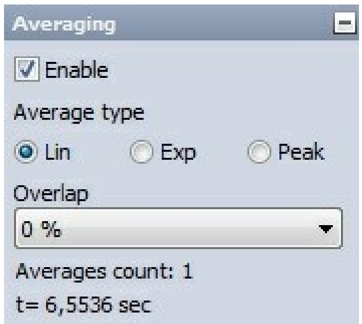
Band display type

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)

Octave Averaging settings



Use averaging mode to get a more ***stable*** **Octave** display.

To *activate* the averaging just *click* the **Enable** checkbox on Averaging section and all control become available.

for detailed information about ***Averaging settings*** *see* → **FFT Averaging**

3.1.3.15 Orbit plot

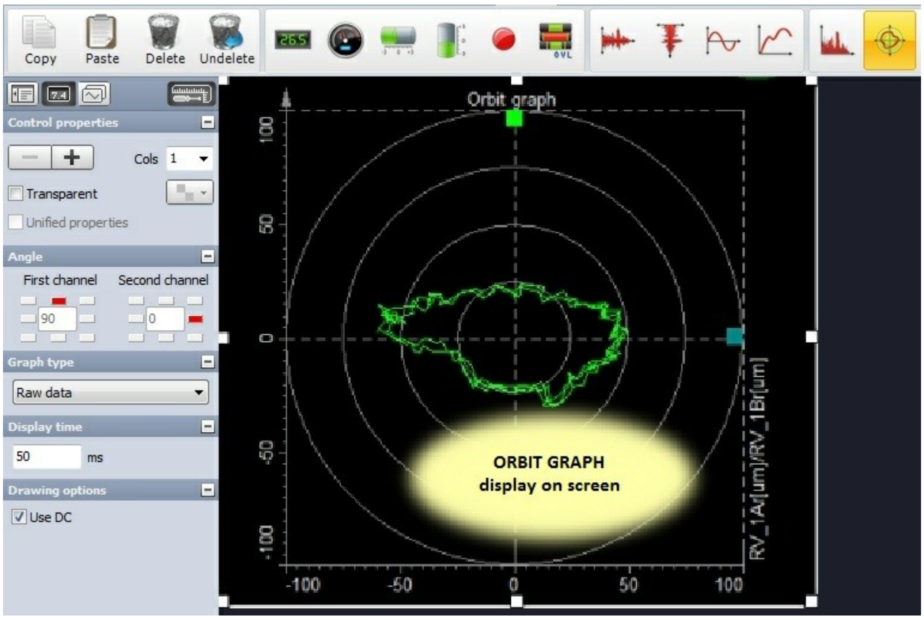
The DEWESoft **orbit plot** *shows* the *x-y scope* with a chance to rotate **x** and **y** axis. It is mainly used for displaying the *axis movement* in DSA analysis.

When you select *orbit plot* in the design or run mode, following settings will appear on left part of the screen:

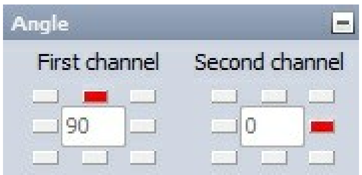
- **Control properties** — for detailed information about Control properties: *grouping*, *number of column*, *Add / Remove*, *transparency*,...
  see → **Control properties**
- **Mounting angle** — describes the *orientation* of the sensor
- **Graph type** — There are two basic graph types - **Normal**, 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** (Common instruments tools)
- **Channels selector** — for detailed information about assigning / reassigning channels
  see → **Display settings**



Mounting angle

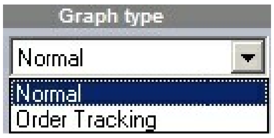


Angle is *defined* for *first* and *second* channel and depends on the mounting of the sensors.

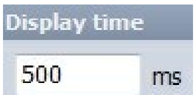
For example if the first channel angle is mounted from the *top*, we select 90 degrees. If it is from the *right side*, we select *180 degrees*.

Please note that the sensors should not be mounted in a *straight line* - there should be an angle offset between them.

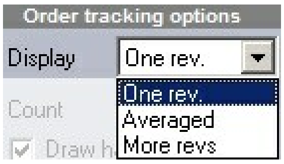
Graph type

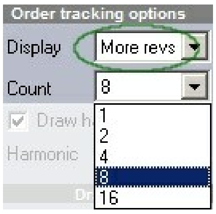


There are two possible graph types - **Raw data** and **Order tracking**.



- **Normal** 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.

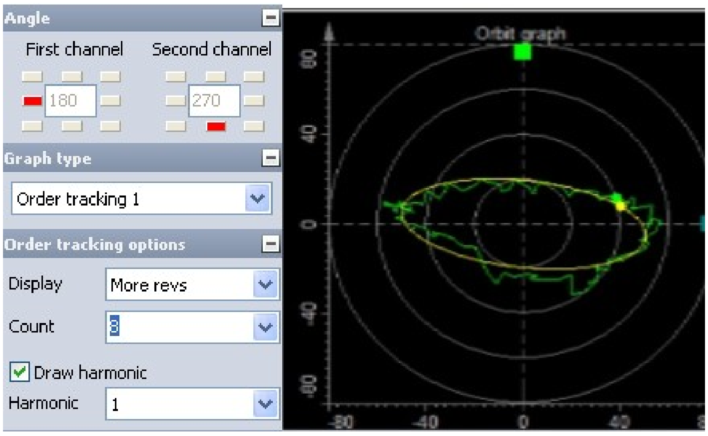


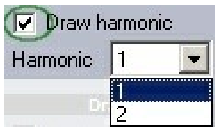


For last two modes we need to define Count - the *number of cycles* to average or display.

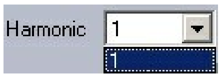
When for Display One rev. is selected, Count field is not available (value 1 is defined).

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* or *second* harmonic on the display.



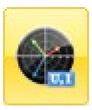
These harmonics must be defined in list of *Output extracted harmonics as channels* section of Order tracking module *setup* screen, otherwise *only first* harmonics is available on list:

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.

3.1.3.16 Vector scope

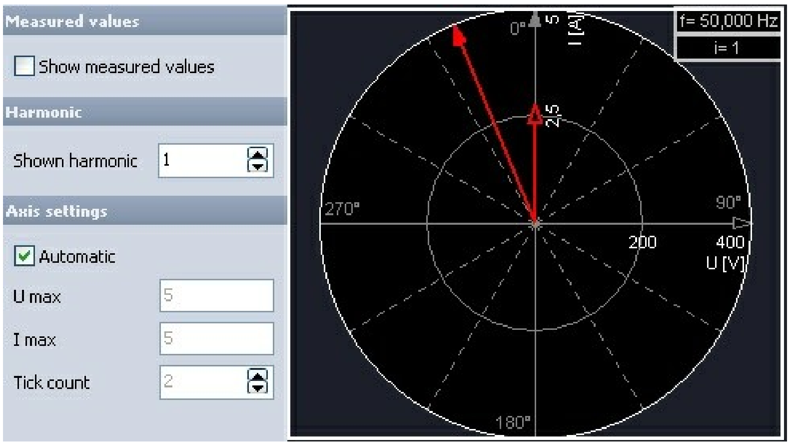


The DEWESoft **Vector scope** instrument is used for *displaying* the ***amplitudes*** and ***phase angle*** between the voltage and current power module ***channels*** and additional to the vector also the most important *measurement values* for *each phase*: Ui, Ii, Phi, cos Phi, Pi, Qi and Si.

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** checkbox:

to show important *measurement values* (*see* example on right and above).

Harmonic selection

Harmonic shown

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

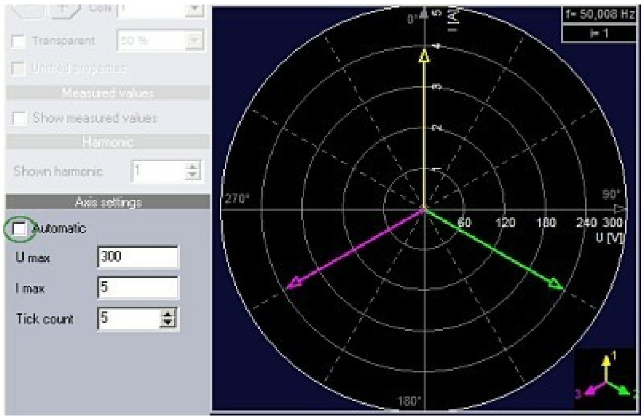
With **Automatic** enabled, the vector scope always *scales* to the *maximum* of all ***displayed channels***.

Manual set

When Automatic is disabled, you can *enter* value for:

- **Umax** and
- **Imax**

Use the Up / Down arrow to *increase /decrease* ***number of ticks*** -**Tick count**.



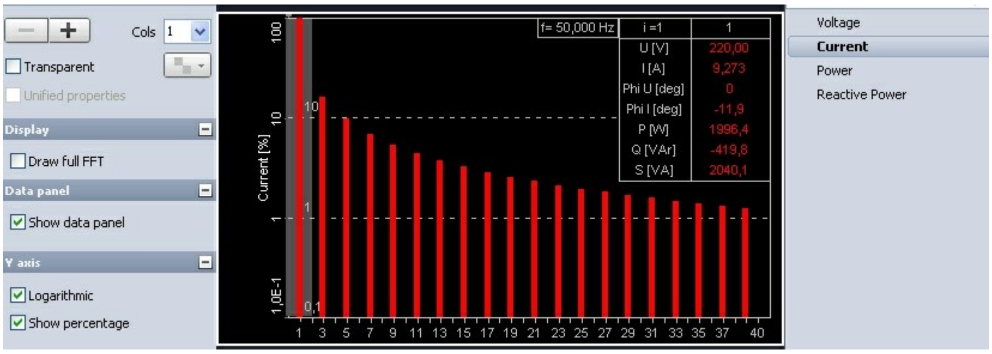
3.1.3.17 Harmonics analysis

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:

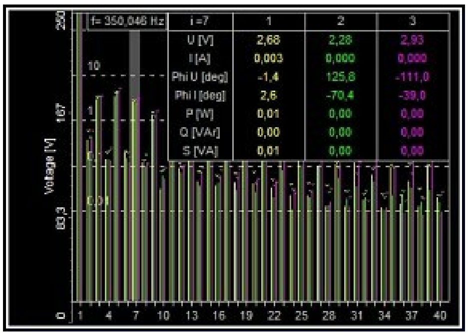
- **Control properties** — for detailed information about ***Harmonics display*** Control properties: *grouping*, *number of column*, *Add / Remove*, *transparency*,...
  see → **Control properties**
- **Harmonics settings** — available ***appearance*** setting for ***Harmonics display*** instrument:
  - **Display value**
  - **Show data panel**
  - **Y Axis display**
  - **Auto scale Y Axis** (Common instruments tools - see Auto scale)
- **Channels selector** — for detailed information about assigning / reassigning channels to/from ***Harmonics display*** → *see* → **Display settings**



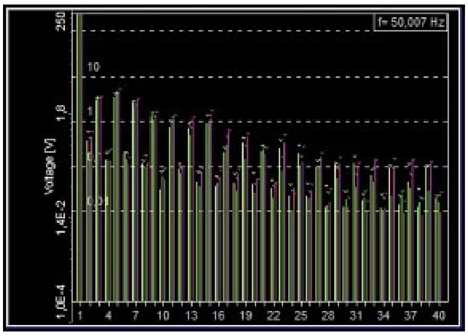
Appearance on screen

The **Harmonics** displays *shows*: ***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**

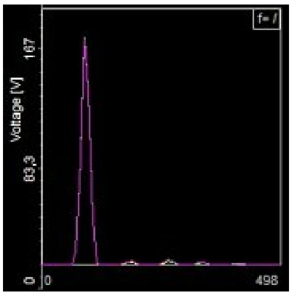
Display value

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** checkbox in Display value section instead harmonics bars full FFT spectrum can be displayed (*see* right).

Data panel

Show 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 this values check the **Show data panel** checkbox (example picture *see* above) in Data panel section.

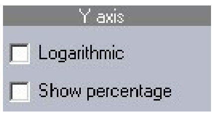
To *select* a **Harmonics**, which values will be displayed in data panel, simply move the mouse cursor over the bars, a grey *harmonics* cursor (rectangle) will follow and indicates your *selection*.

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.

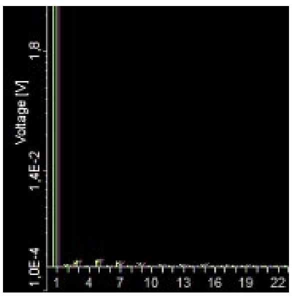
NOTE:	*Harmonics cursor works only by checked Show data panel checkbox.*

Y axis display



In this part of **Harmonics** display settings you can choose with *check / uncheck* appropriate box different Y axis ***scaling***:

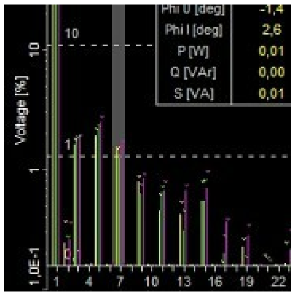
Logarithmic



Logarithmic checkbox in Y axis section is:

- *Selected* → ***logarithmic*** Y axis scaling (example *see* right)
- *Unselected* → ***linear*** Y axis scaling (example picture *see* above)

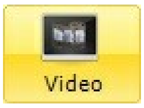
Show percentage



Show percentage checkbox in Y axis section is:

- *Selected* → the Y axis is scaled in % (example *see* right)
- *Unselected* → the Y axis is scaled in it's ***units*** (V, A, W or VAr)
  (example picture *see* above)

3.1.3.18 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.

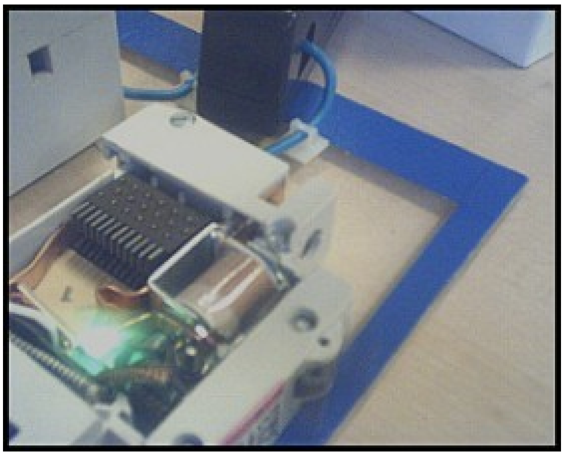
NOTE: *Video display possibility on Design tool bar is available only in case of physical connection and set up of camera in DEWESoft Setup → Video tab.*

When you select *Video display* in the design or run mode, following settings will appear on left part of the screen:

- **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*

Appearance on screen

The **Video** display present content sensitive *acquired* ***video information***.



for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

CAMERA (Video) setting

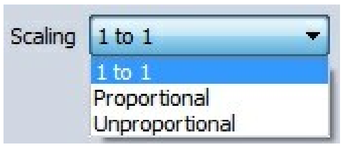
Camera information

In 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**



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

Cameras selector

It will display the *available* cameras to select the right one.

3.1.3.19 GPS

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*.

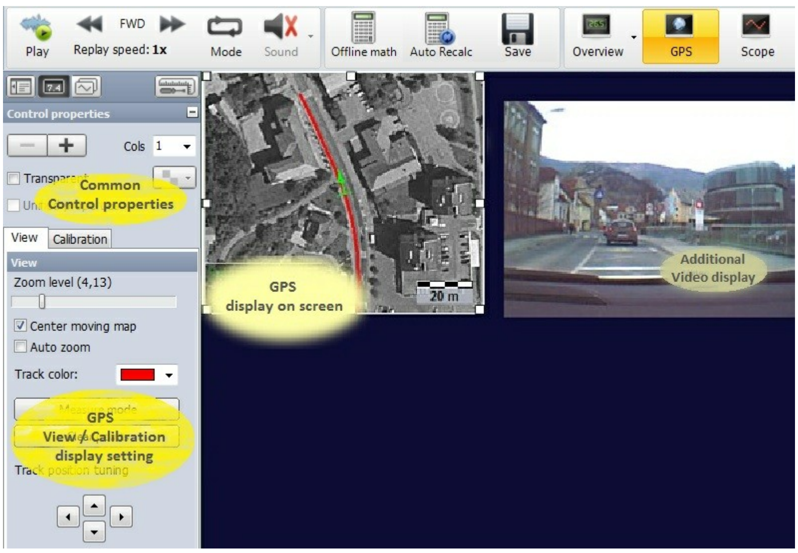
When you select GPS display in the design or run mode, following settings will appear on left part of the screen:

- **Control properties** — for detailed information about ***GPS display*** Control properties: *grouping, number of column, Add / Remove, transparency,...*

  see → **Control properties**

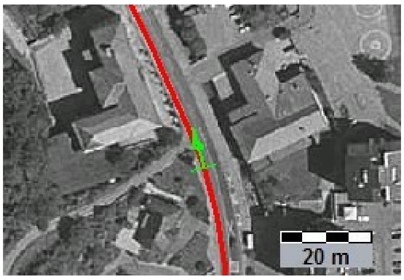
- **GPS settings** — typical setting for ***GPS*** *instrument* are divided in two modes:
  - **View**
  - **Calibrate map**

- **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

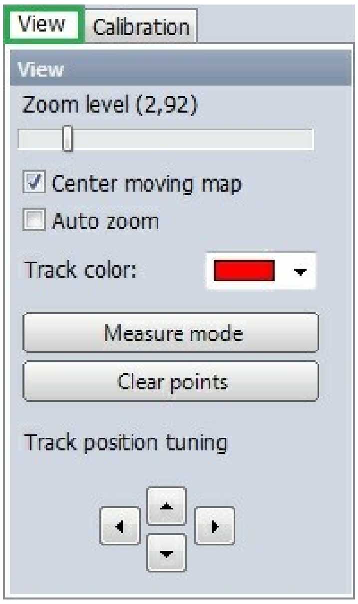
for detailed information about *instruments* ***positioning***, ***size*** and ***transparency*** *see* → **Screen edit functions**

Settings

The **GPS** instrument can be switched in *two* ***modes*** using the tabs:

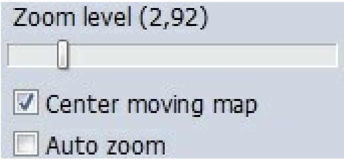
- **View**
- **Calibrate map**

View setting



- **Zoom**

  DEWESoft offers two ways to scale the track: *Auto* zoom or zoom *manually*.



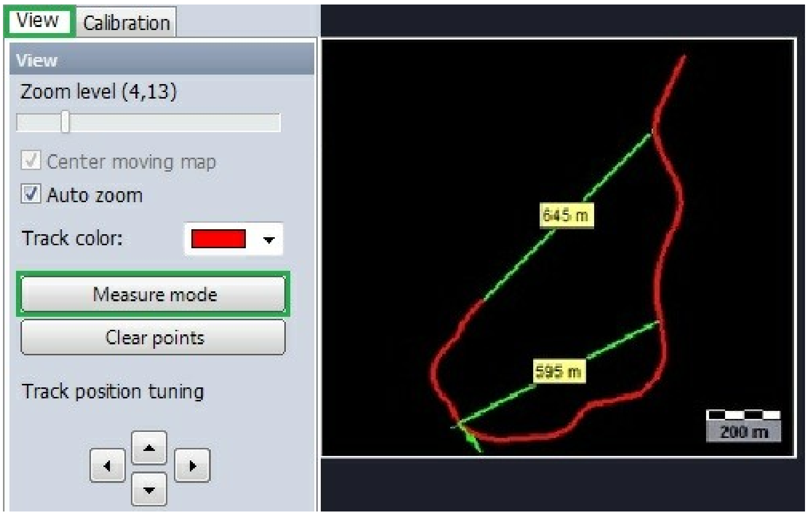
  Use the **Zoom level** slider to zoom *manually*; above the slider after caption, you can see the ***zoom factor***.

- 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 then 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 *de selected*, 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`.

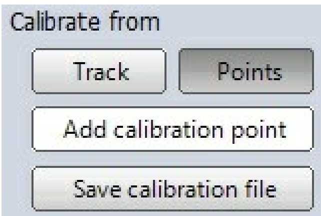
Now you can define the ***map position***. When Calibrate from → **track** is selected, *click* on the map image, keep the mouse button pressed and *move* the map image as required. To ***resize*** the map image, use the Design zoom slider.

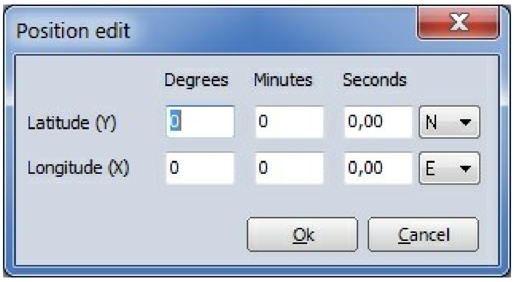
If we *don't know* the coordinates of the map, we can take a *short tour* in the area *storing* the measurement. Then we go to GPS screen, go to Calibration and Load map. The map is shown in behind and the traveled path gets four handles around it. Now we can *drag* and *adjust* those handles to match 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 analyze the recorded data.

3.1.3.20 Overload indicator

Together with other different data of measurement in various instruments DEWESoft provide **Overload indicator** display to *show* list *information* about ***overloaded*** *signals*.

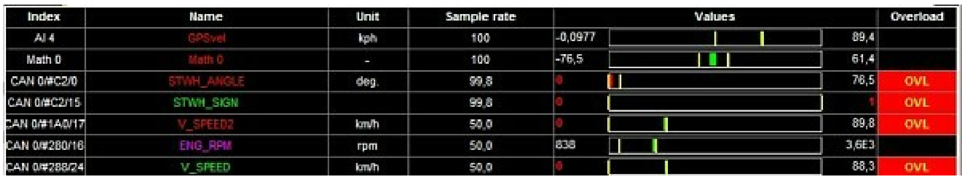
When you select *Overload indicator display* in the design or run mode, following settings will appear on left part of the screen:

- **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 selection**
- **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		Overload
AI 4	GPSvel	kph	100	-0,0977	89,4	
Math 0	Math 0	-	100	-76,5	61,4	
CAN 0/#C2/0	STWH_ANGLE	deg.	99,8	0	76,5	OVL
CAN 0/#C2/15	STWH_SIGN		99,8	0	1	OVL
CAN 0/#1A0/17	V_SPEED2	km/h	50,0	0	89,8	OVL
CAN 0/#280/16	ENG_RPM	rpm	50,0	838	3,6E3	
CAN 0/#288/24	V_SPEED	km/h	50,0	0	88,3	OVL

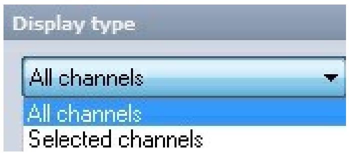
In Values column also *channel* *min* and *max* values is displayed, bar graphically shows the *current levels* of the signal with their *limits*.

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



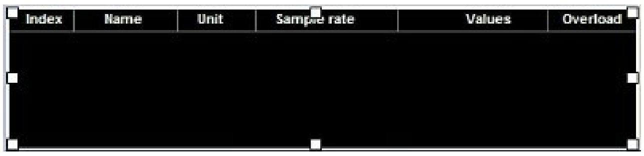
The **Overload indicator** display can *show* *channel* information for:

- **All channels**
- **Selected channels**

Select the **Display type** from drop down list according to your requirements.

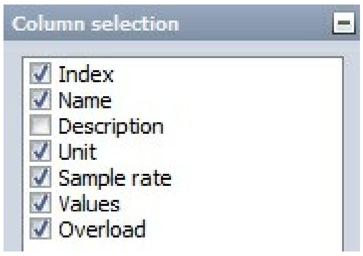
When we select **All channels** option then display the **Overload indicator** information for *all* *channels*.

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.

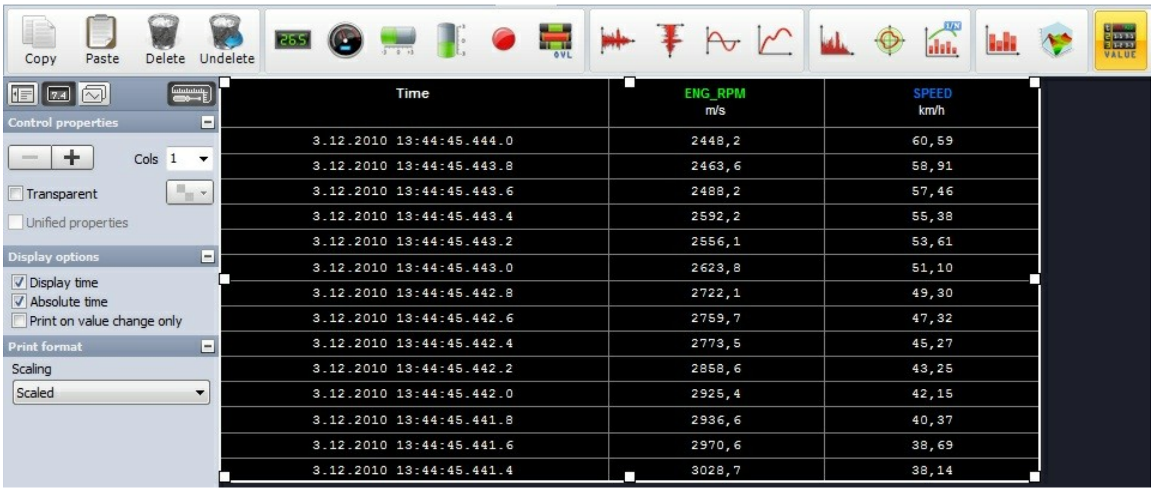
Simply *check* box in Column selection in front of desired column name to show this column on **Overload indicator** (*see* right).

3.1.3.21 Tabular display

DEWESoft provide 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 values 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 *check* box in the Display option to:

1. **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 *selected* channel are *displayed*.

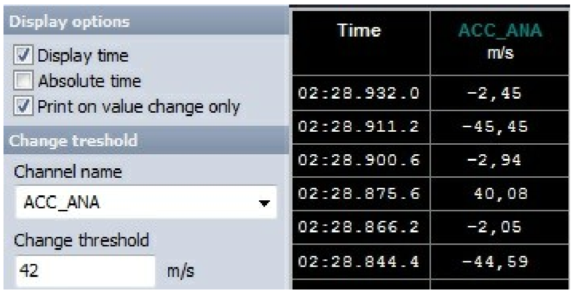
2. **Absolute time**

   When we select this field check box, the ***date*** is *displayed* additional in *absolute* ***time format.***

   All ***values*** of *selected* channels are *displayed*.

3. **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.

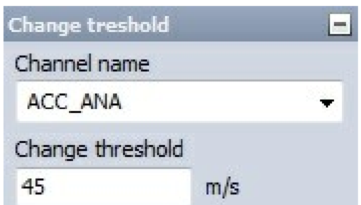
Change threshold

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

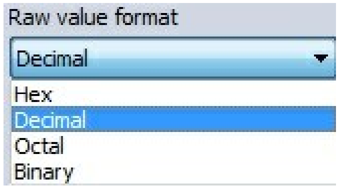
The **Tabular values** *display* offer settings for displayed *channels* ***values***:

- **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).

When Raw type is selected, then new field appear:

- **Raw value format**

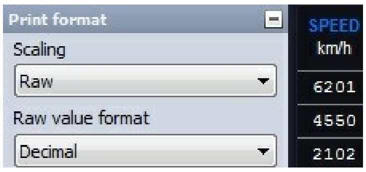


Select the **Hex**, **Decimal**, **Octal** or **Binary** format type from drop down list according to your requirements. This is valid only when raw values will be displayed.

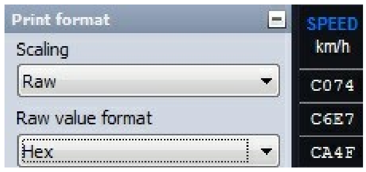
Examples different Tabular values formats:

Scaled Raw Scaling

Decimal Raw Scaling



Hex Raw Scaling



3.1.3.22 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

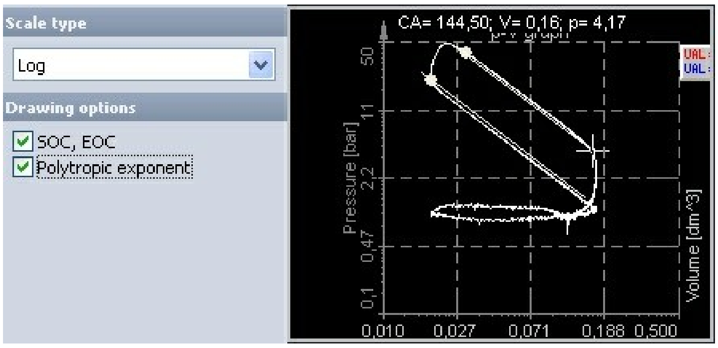
- **Drawing options** — There are two basic graph types - **Normal**, where the inputs are any analog channels, or from **Order tracking**, where the inputs are only the channels which are used in Order tracking math module.

  Available ***appearance*** setting for ***CA p-v graph*** in Run mode are:

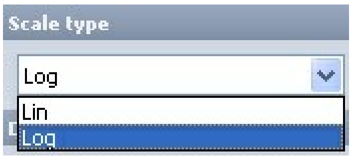
  - **Axis scale/auto scale** (Common instruments tools)
  - **Online calculation**

- **Channels selector** — for detailed information about assigning / reassigning channels

  → *see* → **Display settings**

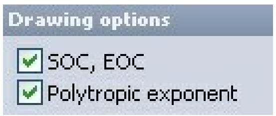


Scale type



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



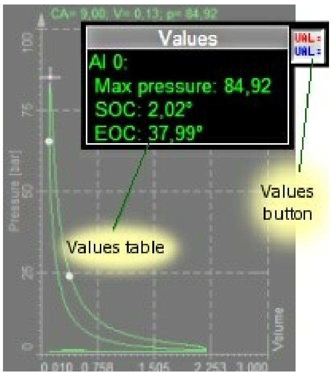
We have also an option to draw ***start*** of combustion and ***end*** of combustion points (SOC, EOC). This shows the points where start of combustion and end of combustion happens (*calculated* from the heat release).

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.

Online calculation

To keep an overview over the **p-v**, DEWESoft can calculate the Max pressure, SOC (***start*** of combustion points) and EOC (***end*** of combustion points) **values**.

To *activate* the calculation, just *click* on VAL: VAL: button and Values table appear:



3.1.3.23 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 have two basic mode 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 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

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*

For turn knob, horizontal and vertical slider we can also define minimum and maximum limit. In analyse mode **Control element** has **no function**.

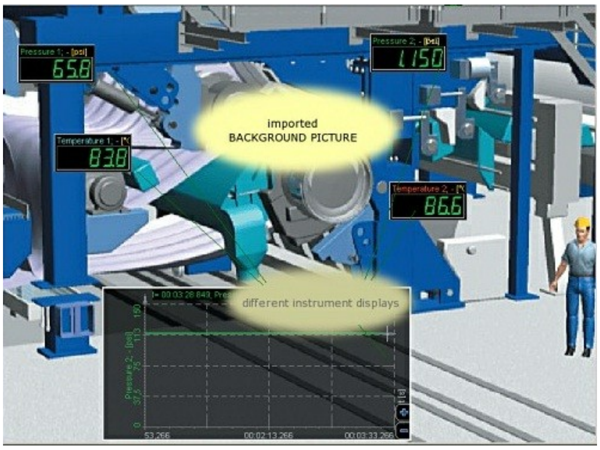
3.1.3.24 Additional controls

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,...

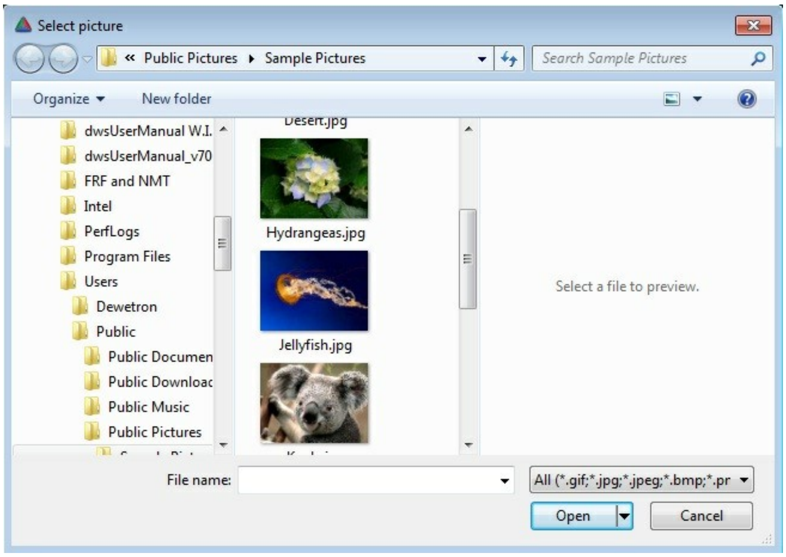
Static image

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 allow *to load* any **jpg**, **jpeg**, **bmp**, **ico**, **emf** and **wmf** image. Pressing **Open** will load the image.

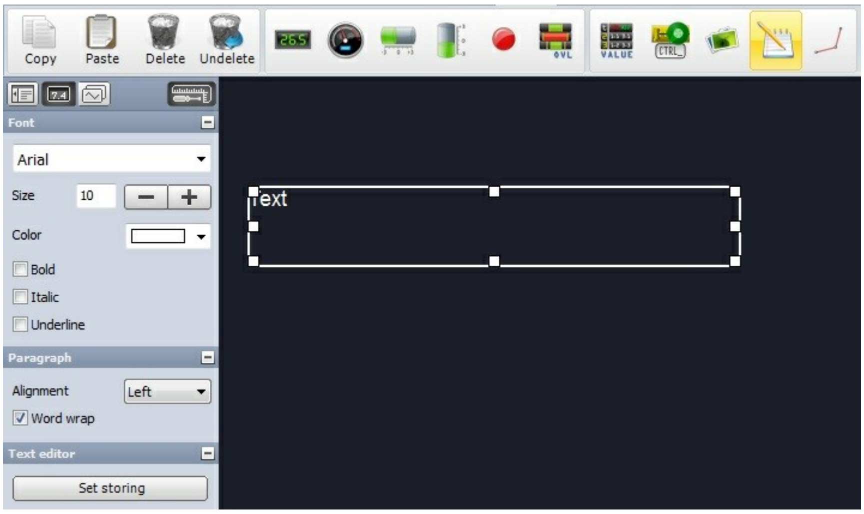


We have an option to Scale the image proportionally to the size of the placeholder with **Full** option, we can scale it to *proportionally* or keep the original size.

Text element

The **Text element** can be used to ***write*** any *text* at any *position* of the screens like *caption*, *comments* and *reminder*.

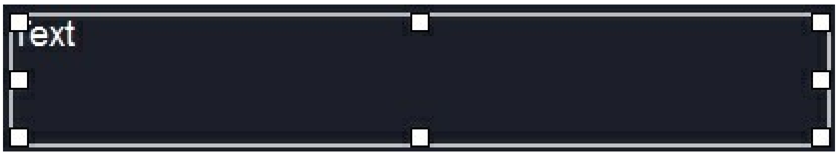
When you select the Text notice in the design or run mode, on left part of the *overview* screen a text *controls* will appear:



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**

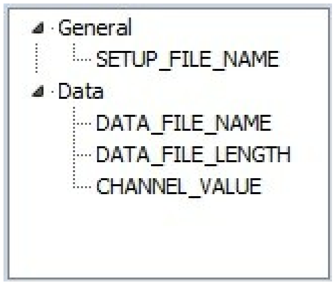
Font and paragraph properties

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

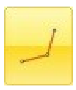
Press the **Edit** button on Text editor section or simply ***double-click*** on the text field to *enter* or *change* the text.



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
<GLOBAL_HEADER SECTION=Comment>

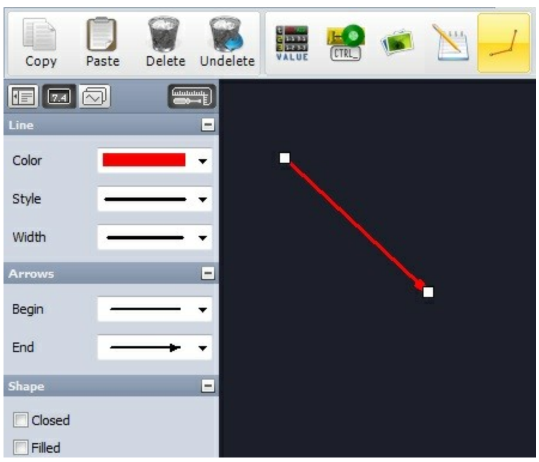
When you have finished, simply *click* anywhere *outside* of the text element or *press* once again **Edit** button to *confirm* the *changes*.

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 that 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 betw een tw o 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.

  On 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.

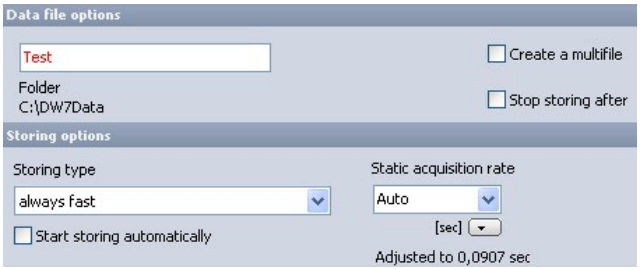
- To *move whole* **line** or **shape**, simply *click* once on it to *select* it and then *move* (with cursor *on* line / shape and with *pressed left mouse* button) *whole* line or shape to the desired *new* location.

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.

3.2 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 multi file 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

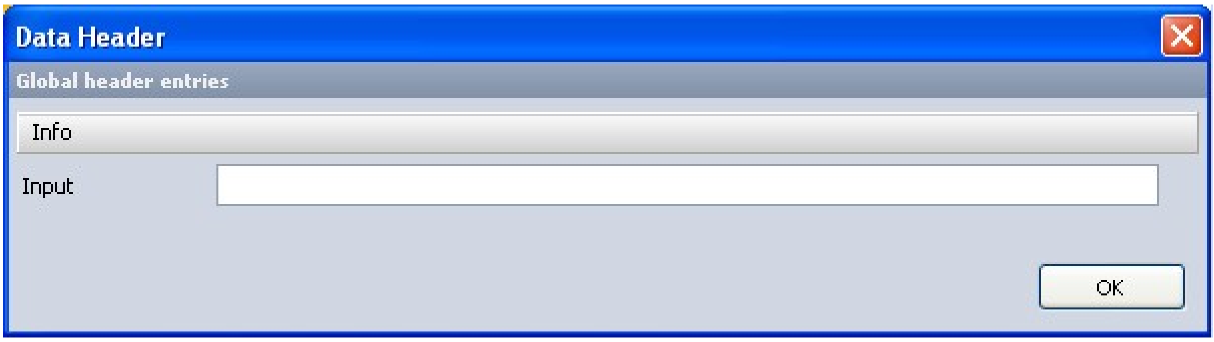
– **Start / Stop Recording** how data recording can be *activated*:
- **Manually**
- **Triggered**
- **Remotely**
- **Automatically**

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*.

3.2.1 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 *start* and at the *end* of measurement. When we do so, a popup 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 how to design **Data header** see →* User Guide → **Data Header Design**

3.2.2 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 Analyze 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*.

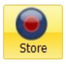
3.2.3 Start and stop recording

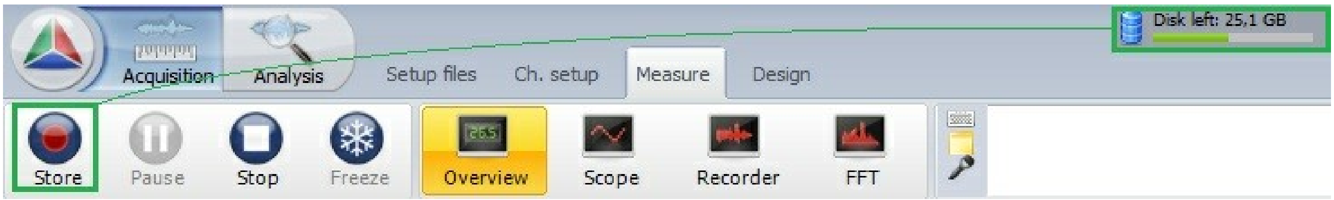
With DEWESoft you can **activate** / **deactivate** recording in four ways:

- **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*
- **Remotely** by using the *digital I/O* lines found in most *National Instruments A/D boards* also to **Start acquisition** only
- **Automatically** ***start*** storing

Manually

<u>**Start**</u> data recording

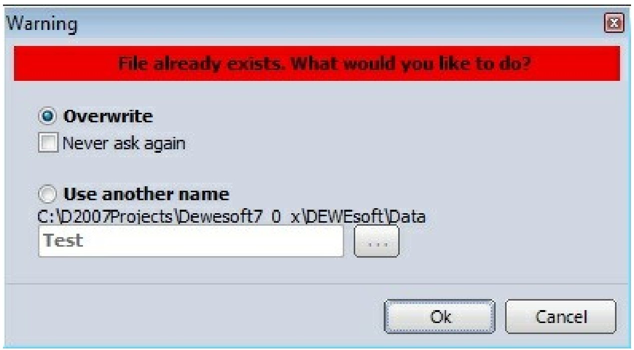
It is as simple as pressing  - the **Store** button on the main toolbar or press F5 on the keyboard.



On the right top part of this screen you will see the currently ***available space*** on your *hard disk drive*.

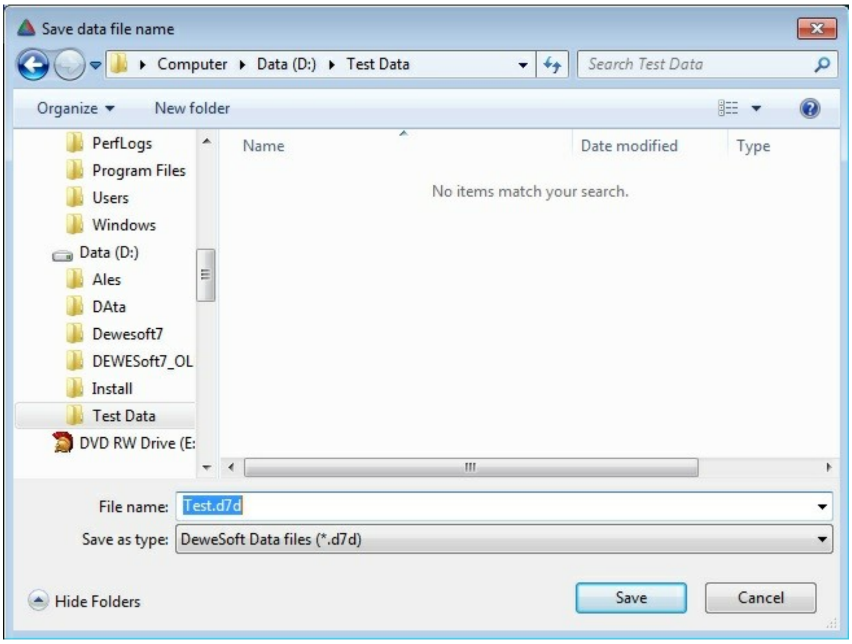
As you can see in the example screen, the hard disk is nearly full - better free some hard disk memory before you start the acquisition or select another file location.

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 → *see* also → **File Name Setup** and **Data Header Design**):



On this window you can:

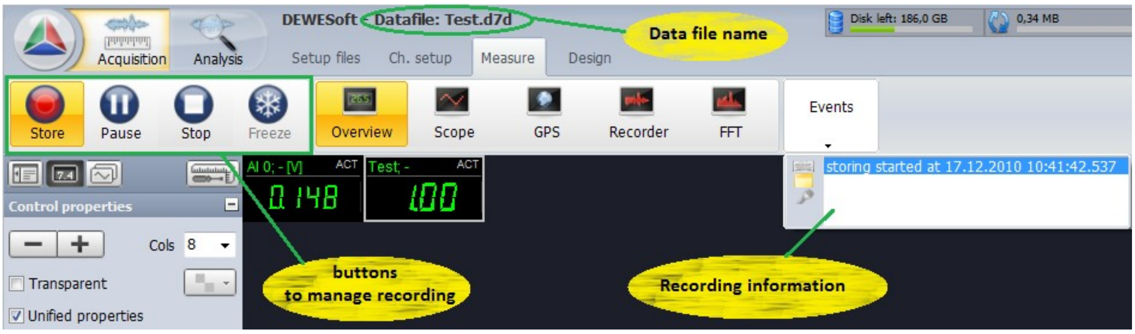
- **overwrite** the *existing file* with selecting **Overwrite** option; there is an additional option - **Never ask again** to always override the file *without asking*; so this form will not be shown anymore until DEWESoft is *restarted*
- **change** the *name* here with selecting **Use another name** 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**.

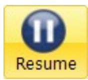
The top of the screen will show you the *current filename*, how *large* the file is at *all times*, and *other* important parameters. The **Store** button itself *changes* color and is *labeled* with **Pause** to indicate that *recording* is **activated**:

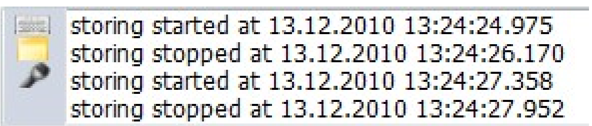


If data was already shown on the screen it is cleared and a red vertical line indicates where recording *began*. In addition, the scrolling EVENT box at the top-right of the screen tells us that "storing started at [date] [time]".

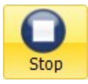
The **Store** button on the main toolbar now changes to **Pause** button.

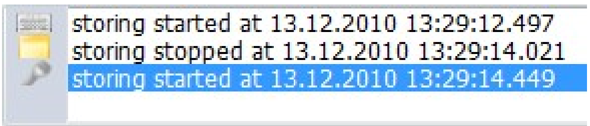
Pause recording

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 <u>same</u> *data file*.



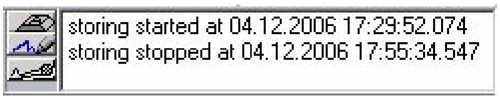
Stop recording

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*** - just 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 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 grey line.

Events can be ***added***, ***edited*** or ***deleted*** also in Analyse mode → *see* → User Guide → **Working with events**

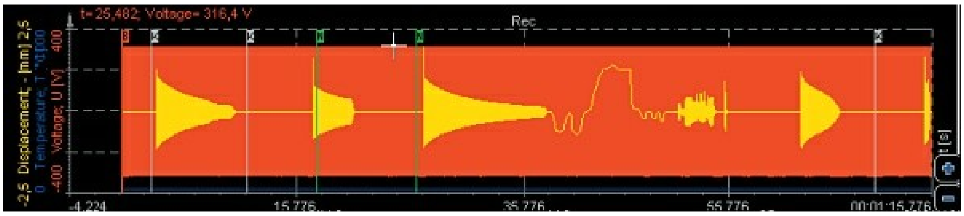
Internal markers

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)

- the *end* of recording marked with the icon and a thin vertical red line (information about this is also automatically noted as *event* - *see* above)

*Example of several **marked events** in on stored signal*

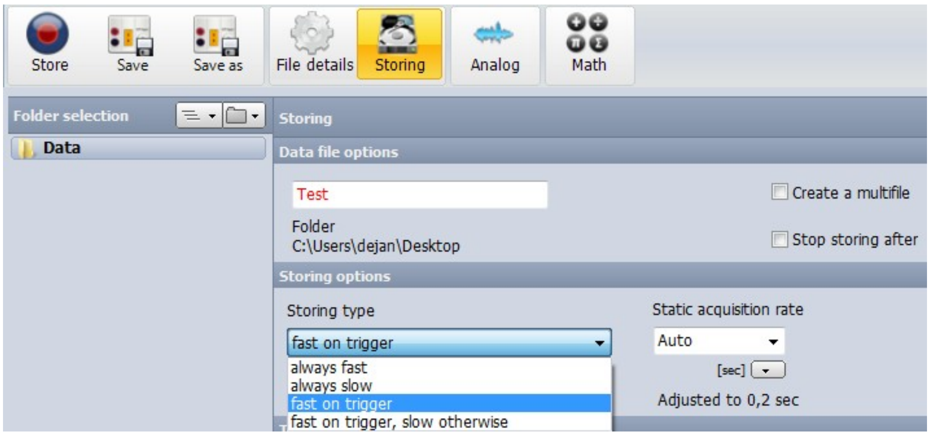


Triggered

You can **trigger** from your signals to **start** and **stop** recording.

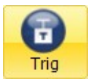
You can *select* two different *types of trigger storing* from the STORING OPTIONS drop down list on the DEWESoft *Setup* screen:

- **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**

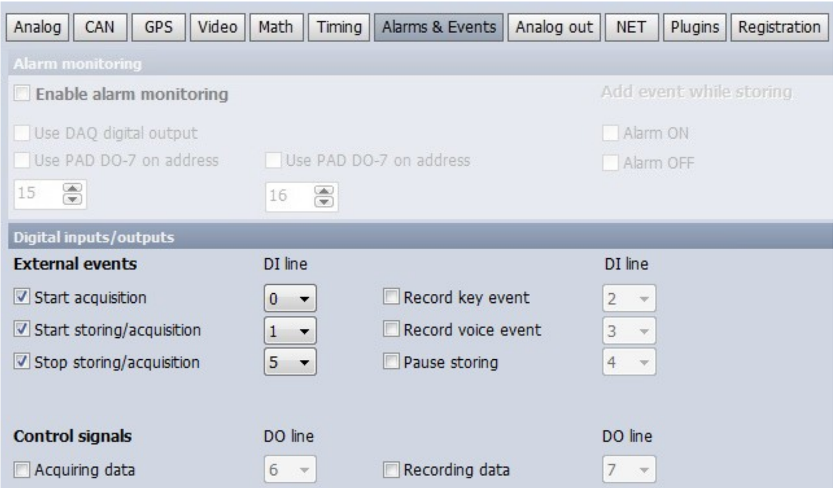
After we have set our *trigger criteria* (manually or also achieve automatically) we can store data. When we press **Store** button, this button *change* the name to  **Arm** button and we see additional  **Trig** button, which tells us that we are *using* triggered storing. We can also press this button 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.

Remotely

You can use the **digital I/O lines** available on some of the supported cards to ***start*** and ***stop*** *recording*.

This option should be *set up* under the **System** menu, on the *Alarms&Events* dialog box:



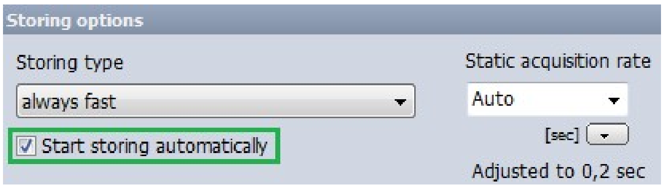
In the example above, we are using the **DIO** lines 1 to **start** and 5 to **stop** *recording*. You could use *other* lines to "read" the current status, about whether DEWESoft is in the acquisition mode (i.e., able to start recording, as opposed to being the Analyse mode), as well as whether it is *presently* recording data or not. Click on any checkbox to *activate* or *deactivate* the named function, and use the small selectors to *assign* the **DIO** lines freely to each.

Start acquisition

On this window can be checked also **Start acquisition** field to ***remotely start*** *acquisition* (without storing data) using **DIO** line (in example above 0).

Automatically

If the **Start storage automatically** checkbox 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 autoload function of DEWESoft for highest automation.

for information about ***autostart*** of the measurement *see* → System Settings → **General setting**

3.3 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*:

- Hardware setup → *see* System settings → **NET module**
- DEWESoft NET Setup → *see* User Guide → **Setup - NET data acquisition**

Measurement — Manage measurement units with **NET menu option** including *Measure transfer speed*; ***Creating a display***, *measure-acquire* data and ***store*** this data on net

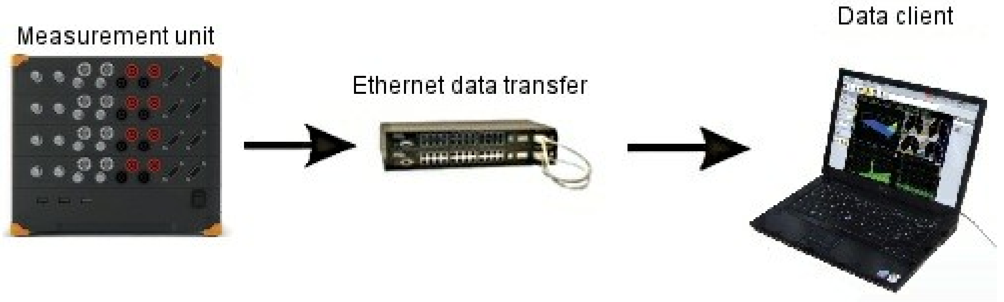
Analyse — *Analyze* acquired and stored data on net, *export* measured data

General information

It is important to note that even their *channels* can be **viewed** on the CLIENT(s), the actual data are **stored** on the MU's. This is critical *to guard* against data loss which might be caused by the network going down or transmission being interrupted. Even if this happens, the data are safely stored on the MU(s). When the network connection is reestablish, it is possible to *reconnect automatically*.

Even if the network going down or transmission was being interrupted, the data are safely stored on the MU's. The *idea* of the DEWESoft **NET** technology is to have a distributed system when:

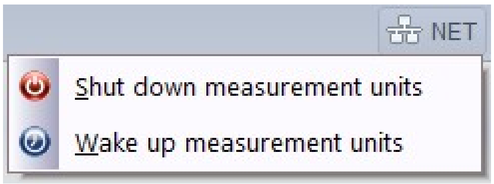
- the *required computing power* is too high for a single measurement unit (e.g. many channels sampled with a high sample rate)
- there is too much *distance* between the units for analog data transfer
- the measurement unit is *not accessible* (e.g.: dangerous measurements, test rig measurements,...)
- data from measurement units shall be *displayed on several* client computers
- measurements have to be *remotely controlled* or *supervised*.



Within DEWESoft **NET** the **Master-Unit** (*Master* client or *Master* measurement unit) *totally* **controls** the **Slave-Units** (*Slave* measurement unit) – when the *Master-Unit* switches to the Setup screen, also the *Slave-Unit* switches to the Setup screen.

NET menu option

Click the **NET** menu option to see the list of options:

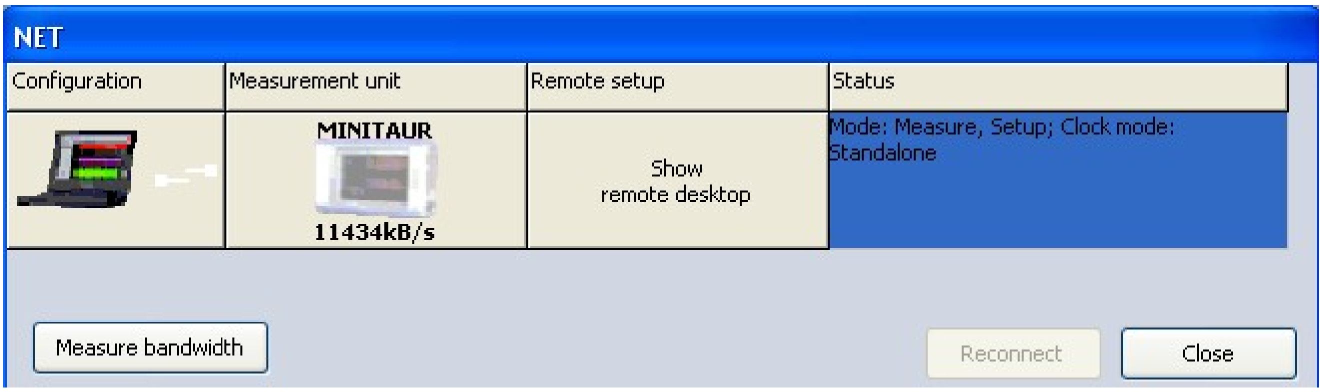


Notice from the menu that you have several other useful capabilities:

- Connect / Disconnect from MU's - *connect* to all MU's / *releases* the *connection*
- Close DEWESoft on MU's - *closes* the DEWESoft application on all MU's
- Measure transfer speed - *measuring* the **bandwidth** (*transfer speed*) between the MU(s) and this client
- Reboot MU's - *reboots* the MU computers (useful if they have crashed or hung up)
- Shut down MU's - really shuts them down (requires ACPI power system on the MU's)
- Wakeup MU's - requires "**Wake-up on LAN**" option enabled on the MU's

Measure transfer speed

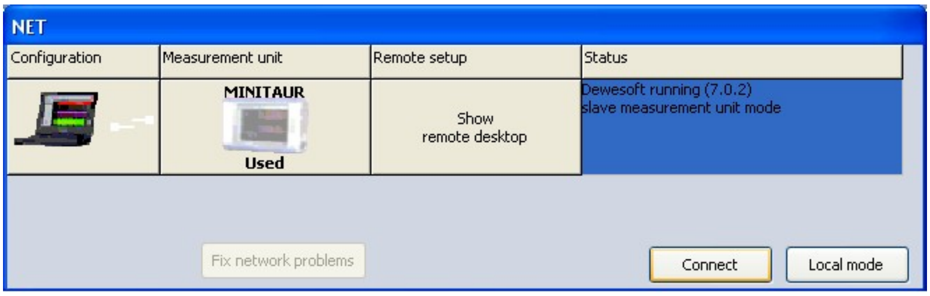
Click the Measure transfer speed item and DEWESoft will *perform* a **test**, *measuring* the **bandwidth** (transfer speed) between the MU(s) and this client:



This will take a few seconds, and then will reveal the maximum possible transfer speed. In this case it has been found to be *only* 92 kB/s. In this example, the network is a relatively slow one.

Connecting to the remote units

When DEWESoft **NET** has been *configured*, the Master unit can *connect* to the Slave units by *selecting* **NET** → Connect to measurement units from the *menu* (*see* picture above) or clicking the **NET** button and then click on **Connect** button:



Remotely Controlling a Slave MU

We can use only the **master client** computer. The MU is set to Slave MU *mode*, and we have already *connected* to it using the steps from the preceding section.

NOTE: *All steps are done on the CLIENT!*

We are not touching the MU at all. It could be a few feet *away*, or 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**!

3.3.1 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**

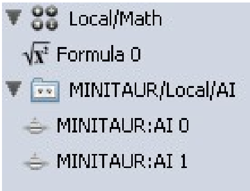
Creating a display on the client

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.

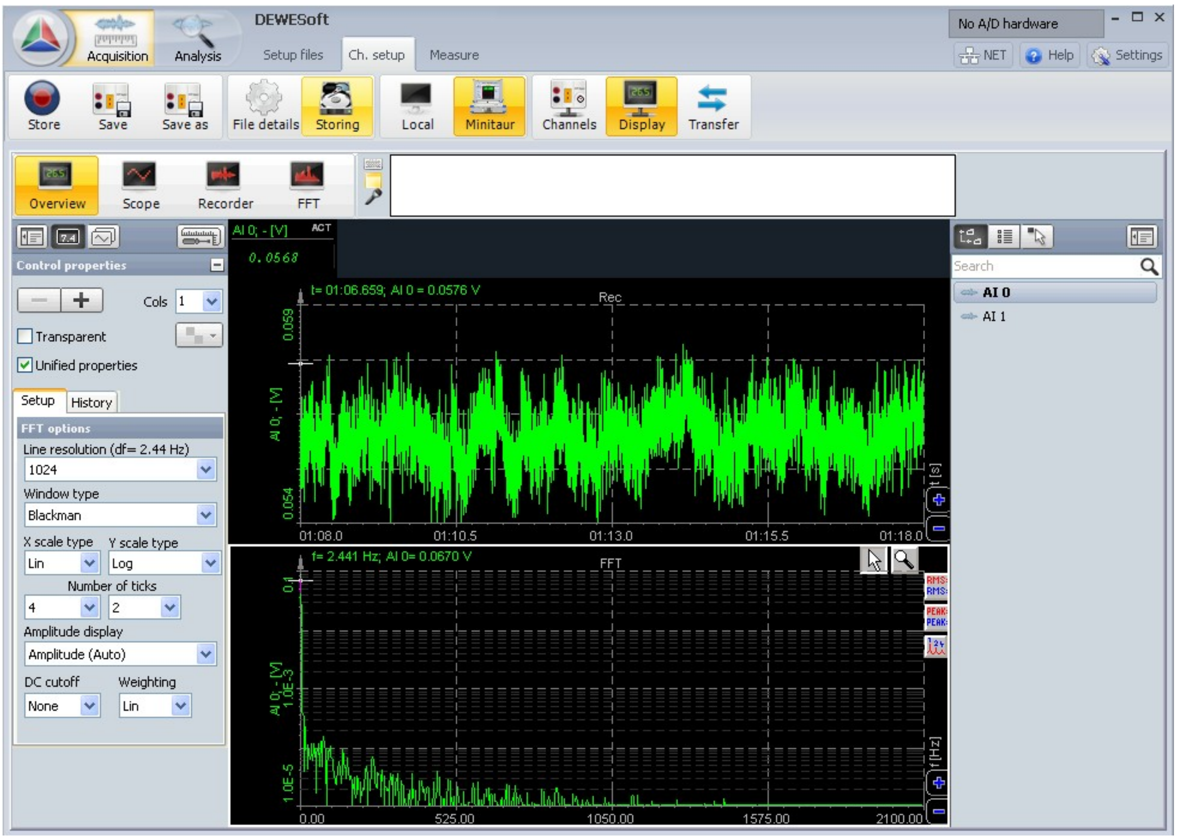
The *channels* from each MU will be shown this way *automatically*. This is the only thing that is different from setting up a screen in the standalone mode of DEWESoft, for years now.

Consult 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 <u>not</u> 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 <u>avoid</u> 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 <u>local</u> 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 <u>remote</u> storage is* 8 MByte/sec.

- **Local** & **Remote**

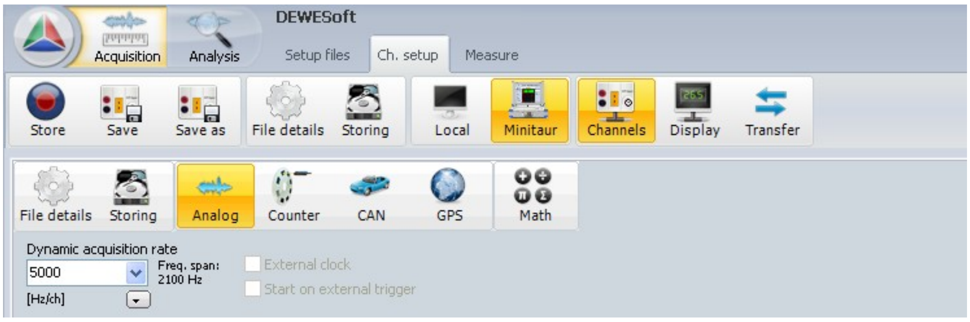
It is often *not possible* to **transfer all** possible channels in real time to the client for storage *even* with a gigabit Ethernet interface.

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

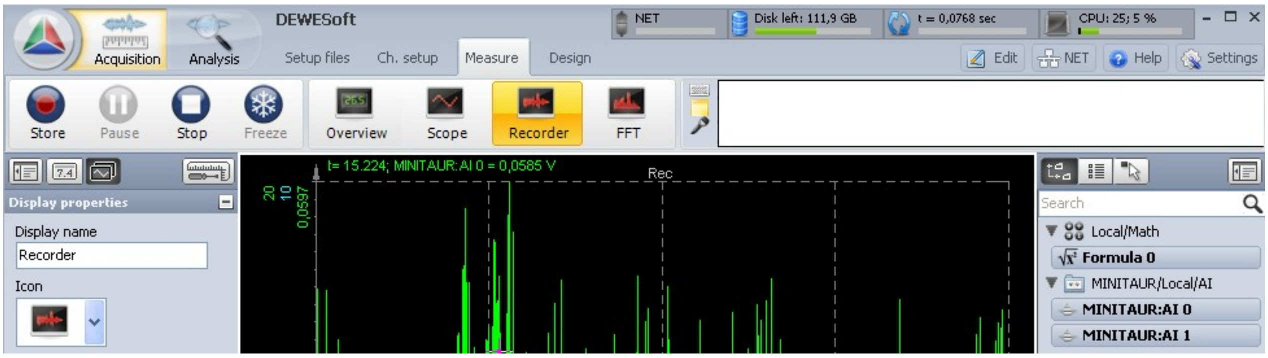
- When the *connection* was *successful*, the NET button turns green and the DEWESoft *Setup* screen *appears*. Each remote unit's *system* setup can be accessed via the *button bar* (green indicates the *selected unit*), therefore *click* on the name of remote unit's button (in example below MINIATUR) so that we can proceed to *set up* <u>this</u> Slave MU:



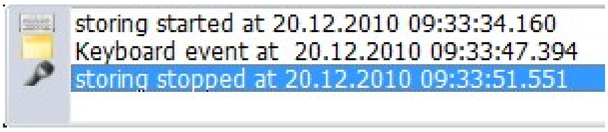
Now click the **Setup** button and on central part of this screen you are doing <u>exactly</u> what you would be doing by *stand-alone MU*: *set* the *dynamic* and *reduced sample rates*, choose a *filename*, set *storing option*,...

Storing data on the MU(s)

With the client and MU properly <u>configured</u>, 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.

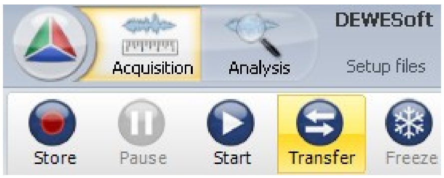


for details *see* → User Guide → **Recording Data**

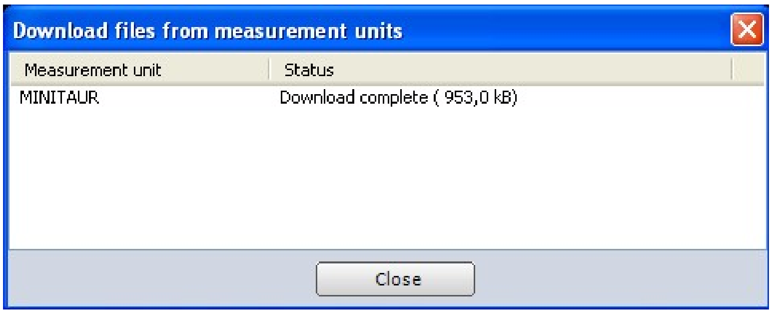
Uploading stored data to the client

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**.

4 Analyse

The **Analyse** procedure is important part of DEWESoft for ***analyzing acquired data***.

Once *data* has been *acquired*, there are a number of things that you can do with it: *review data, display data in selected instrument display with signal overview, replay data, start / stop replay and sound output, arranging instruments, (re-) assigning channels, watch events, signal analysis, reload triggered file, printout of instruments, store settings, copy Channel setup and display elements to Clipboard, exporting data,...*

for ***practical hints*** concerning establishment of efficient conditions ***analysis*** → *see* also → DEWESoft Tutorials

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 data file; display *data file information*: Settings, Events or Data header
- **Displaying data** and **Replaying data** to ***display*** data in selected instrument display with 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**

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**

To ***enter*** the DEWESoft Analyse mode press the  - **Analyse** button on main tool bar.

4.1 Reviewing data files

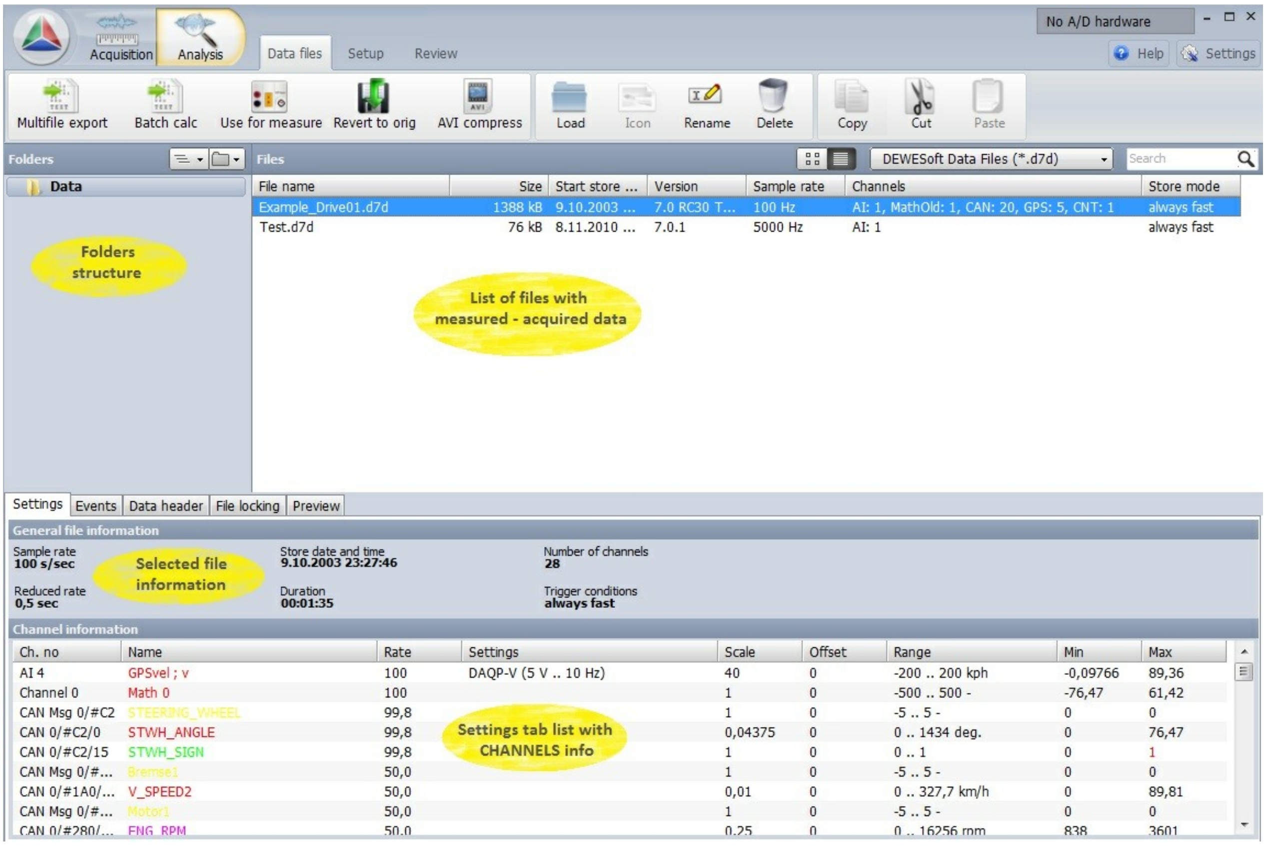
Reviewing data files is very helpful to analyse *measured data* and include following procedures:

Loading data file	to ***load*** *acquired measured* data stored in data file; ***display*** data file information: *Settings, Events* or *Data header*
Displaying data	***display*** data in selected instrument display with *signal overview*, ***selecting*** a *channel for overview window, Time selector*

Include Video file	*Video* data can be ***included*** and ***synchronized*** in DEWESoft, *zoomed*, *scrolled*, *scaled*, *played*, ***resynchronized***, ***removed*** from *data* file
Replaying data	*start / stop*, *Sound output* selection, *Replay mode*, *Replay direction* and *Replay speed* in different Display Screens
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**
Working with events	to ***add***, ***display*** and ***save*** *Keyboard*, *Notice* and *Voice events*

4.1.1 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 *sub folders* 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 *sub folders*, we can choose them by double clicking on the sub folder.

The first *level data folder* can be *changed* on either clicking on 'up' button, which brings us one level higher in the folder structure or by clicking 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.

for information about ***folders management*** *see* also → User Guide → **Folder tree view navigation**

File selection list

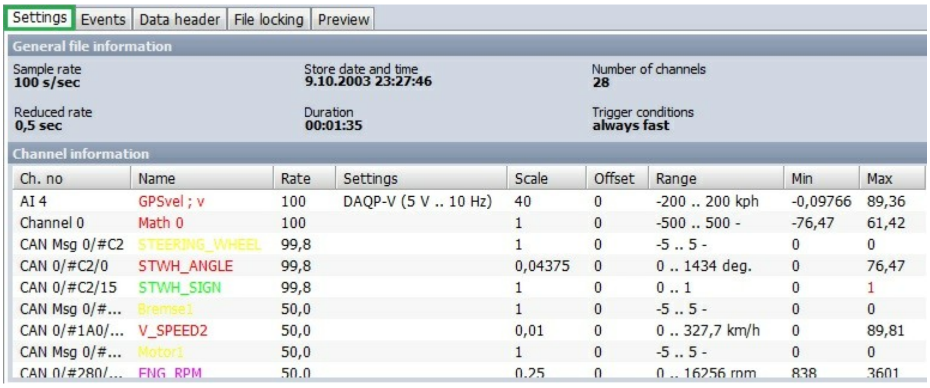
In the upper right section, you can select the ***file*** which should be loaded for analysis with double click on it. It provides you also with plenty of *information* about *all* files available in the selected *folder:*

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 ...	Version	Sample rate	Channels	Store mode
Example_Drive01.d7d	1388 kB	9.10.2003 ...	7.0 RC30 T...	100 Hz	AI: 1, MathOld: 1, CAN: 20...	always fast
Test.d7d	76 kB	8.11.2010 ...	7.0.1	5000 Hz	AI: 1	always fast

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*.

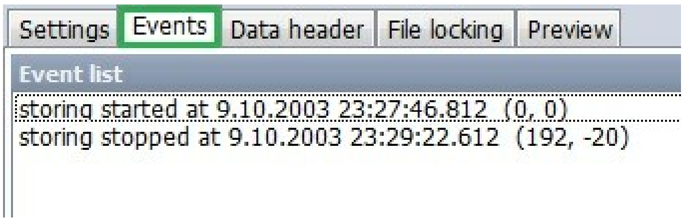


This part *shows* now **information** about the selected *file* on General file information section (Sample rate, Store date and time, Number of channels and Trigger condition) and more *detailed* **information** on Channel info section:

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

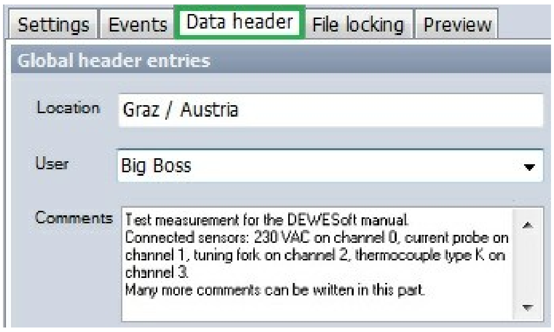
When you change to **Events**, *all* happened ***events*** will be *displayed*. Events are start and stop of measurements, keyboard events, notice events and also voice events.



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 tool bar is selected, Settings, Events and Data header information are displayed on whole screen and *Analyse* mode is interrupted → *see* above → **File settings**.

To continue with Analyse mode the 

 - **Analysis** button on main tool bar must be selected *again*.

4.1.2 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 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:

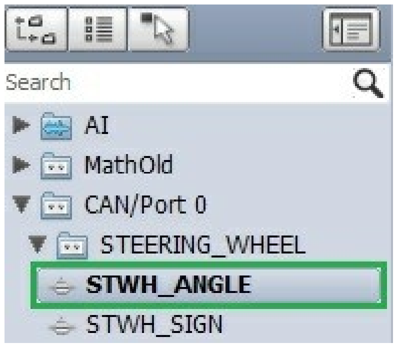
Stored data offers at the top of the screen a small bar - **Signal overview window / Time overview bar** with:

- *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.



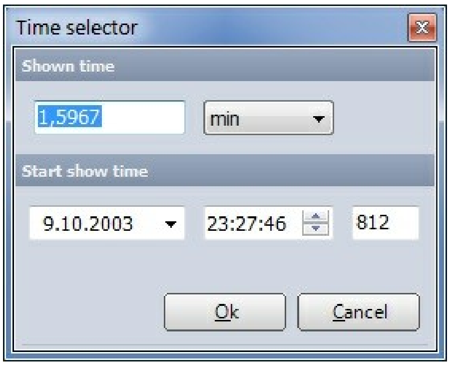
for information about ***Channels selector*** *see* → **CHANNELS selector**

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

When you click on the - **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 h, min or sec *time* unit can be selected
- exact Start show time position within the file; in left field date is displayed, which can be selected from drop down list (calendar is displayed); in right field start time to show measurement is displayed in form h:min:sec - we can *direct enter* values or select group h, min or sec and after that set values with up and down button.

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 - *Show time*)
- *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 change 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 to and then you can drag this cursor within box (*see* picture on right):

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 whit press this switch we *enlarge* selection to initial - *whole* range of stored data and Trigger mode button return to .

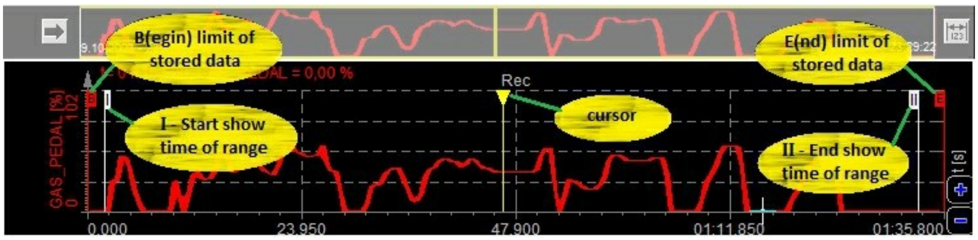
4.1.3 Selecting data

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***

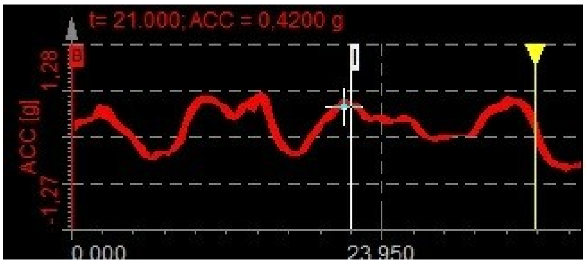
Selection range of stored data

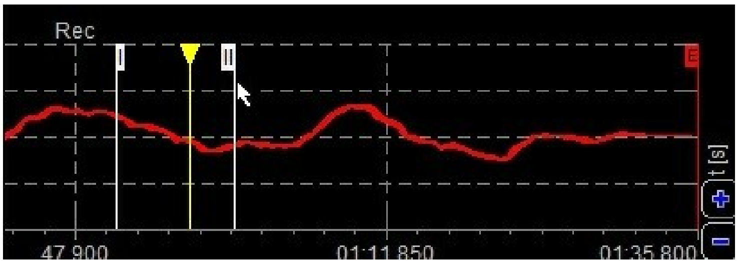
1. Measurement cursor



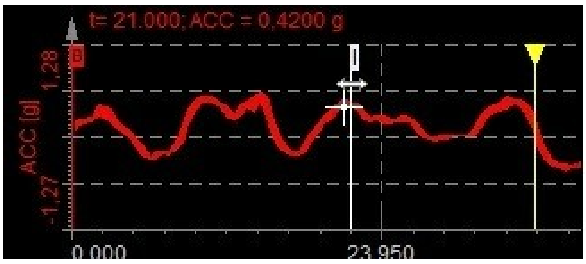
The *recorder* and *vertical recorder* offer two **measurement cursors** for the active graph on top labeled with I for *Start* show time and II for *End* show time. You can *drag* these cursors to ***select*** a ***certain region*** in two ways:

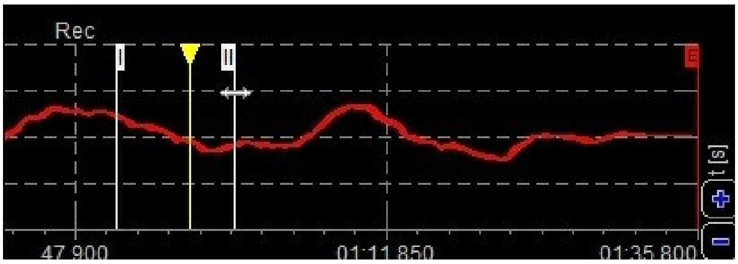
- *move* mouse cursor to the *first* position of interest; *click* (measurement cursor I appear on this position) and *hold* left mouse button and *move* (on mouse cursor appear measurement cursor II and move whit him) to the *second* position of interest





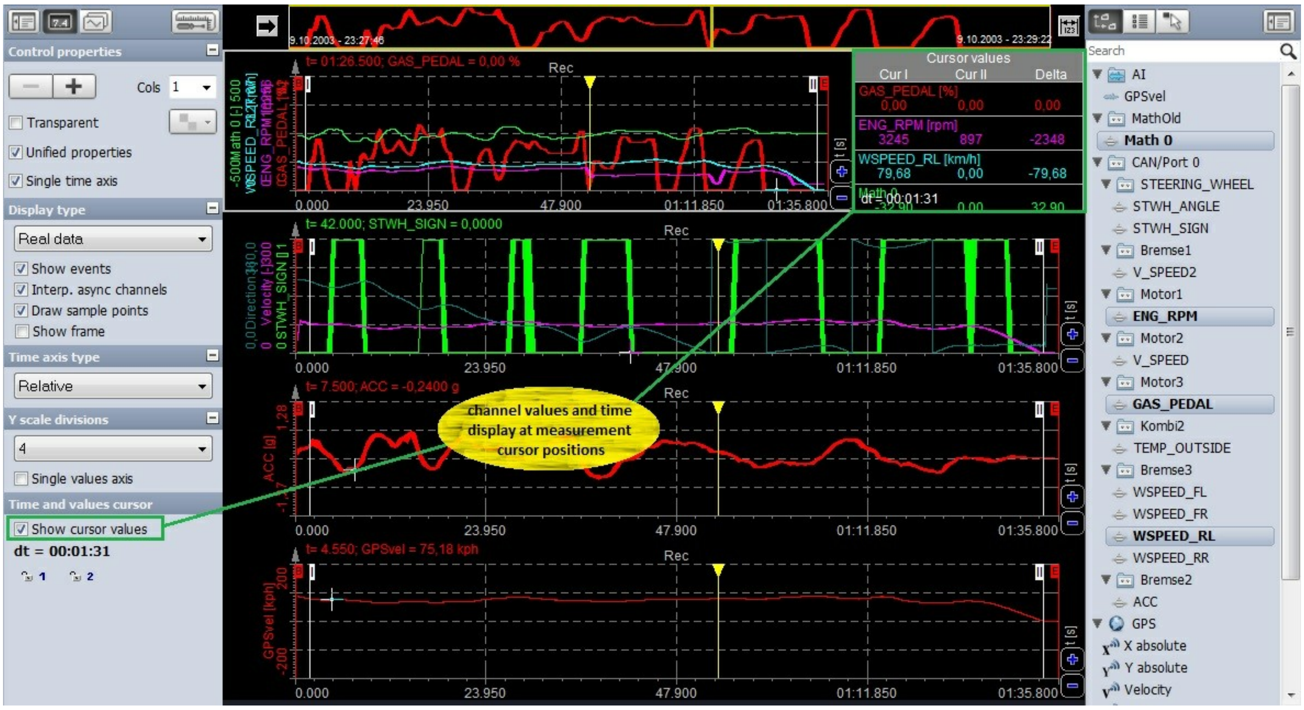
- *drag* the *first* cursor I from *leftmost* side of the graph to the *position* of interest and the *second* cursor II from the *rightmost* side of the graph (when mouse point on vertical line of measurement cursor mouse cursor change to two-sided arrow which indicate feasibility to move measurement cursor)





It's no matter in *which* recorder you use the cursor; it will *automatically* move in <u>all</u> 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:

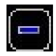


As we can see above, in Analyse mode on left part of screen below *recorder* settings (same also by *vertical recorder*) appear new section to display readout ***values of signals*** and ***time*** at *cursor* position - Time and values cursor with icon to **lock** cursor (*see* also **Lock Cursor 1 to selected position**).

2. 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 the 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  button on right side of graph to zoom *into* the selected area. To *undo* the zoom use  button (below +) 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.

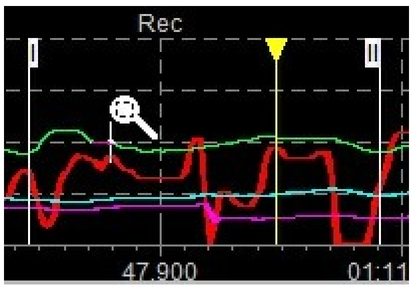
  You can do that action *several* times to *zoom into* the selected area step by step (by each step is decrease of range about 1/25 prior range) and is meaningless in what place cursor there is.
  By first *zoom out* graph return to *previous* zoom (range) and after that step by step. If we use before multiple zoom (with +*button* or *measurement cursor*), in first step displayed range return to about 3/4 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.

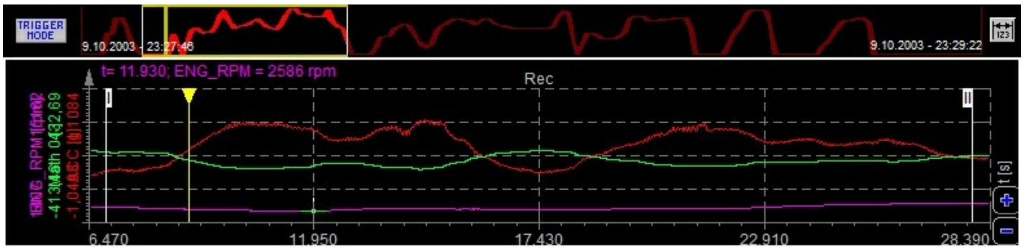
You can do that action *several* times to see the information of real interest.

3. Window zoom



Additionally to the described method you can also zoom *in both axes* using the **window zoom** (also known as *boxed zoom*, because you open a 'box'). To do that *place* the mouse cursor at the *first* point (normally the upper left corner of the area of interest), press the SHIFT key and keep it pressed - the mouse cursor will change by little move it's appearance to:

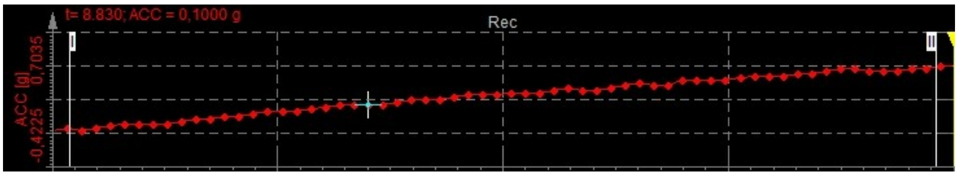
Now move the mouse to the *second* point of your zoom window like here on the right. *Release* the mouse button (and afterwards also the SHIFT key) to zoom now into the selected window:



You can do this procedure as *many* times as required, also in combination with the 'standard' zoom in time axis. There is difference to "standard" zoom that here is *zoomed in* ***both*** *axes* (but *only* signals in zoom window, signals of other graphs - recorders are zoomed in only at *time* axis because there is *only* one time base).

To **undo** the last zoom simply *right click* once - as in the standard zoom.

When you zoom in very *deep*, you will see the really acquired data *points* marked with points, the time between is a linear line.



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 I at its top changes to L1 (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 L1 disappears and cursor 1 is again available for *new* locks. We can also lock **cursor 2**.

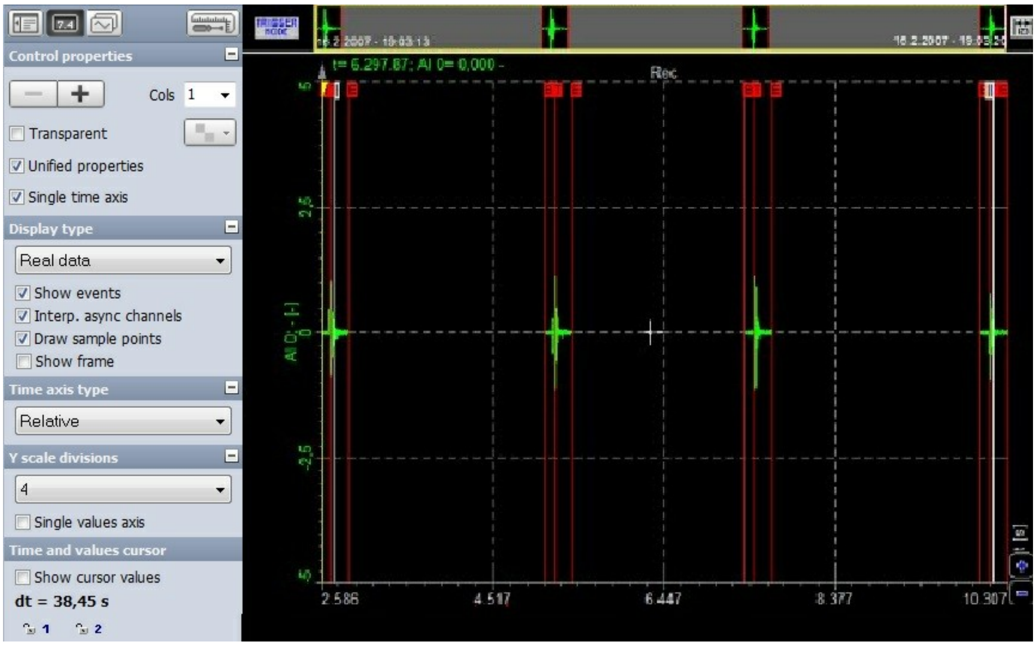
4.1.4 Selecting triggered data

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 time the data is blank.



Note that there is a new TRIGGER MODE - '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.

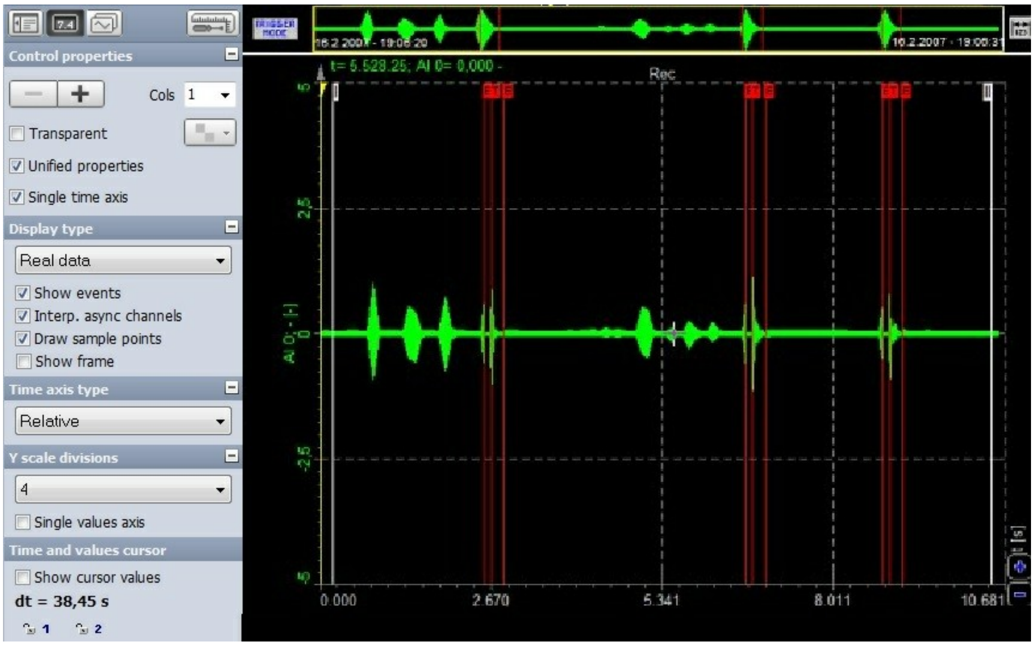
To display each event *separately*, just press the TRIGGER MODE button and after that *use* the arrows to **move** *forward* and *backward between* the events - select event. On *first* trigger only arrow is available - to navigate *forward* to second trigger and on last trigger only arrow is available - 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**.

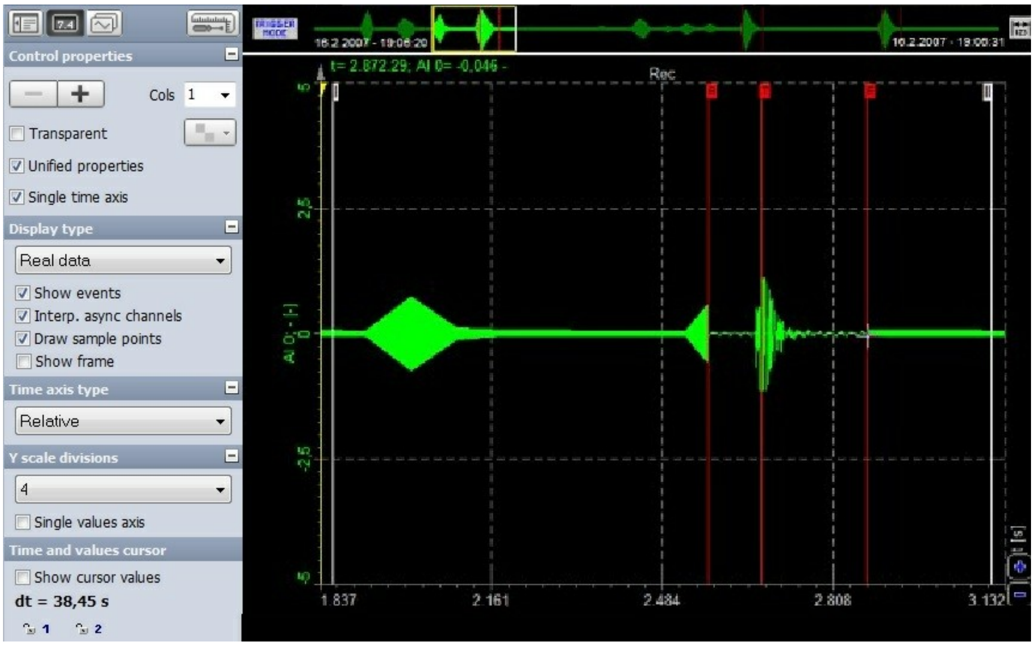
'fast on trigger, slow otherwise' triggered data file

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.



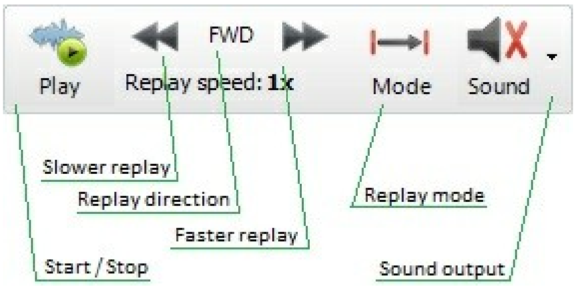
All other Analyse action and procedure on *triggered* signals are *same* as described → **Reviewing data files**.

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.



4.1.5 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  - 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 started the replay, 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).

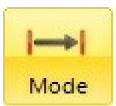
Sound output

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.

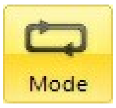
Replay mode

You can select between three different replay modes (simply click on the third button from left to change it):



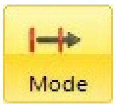
Normal

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 direction

Change between ***forward*** and ***rewind***. When  button is displayed, with click on it, this button change to  and data replays ***backwards***.

The replay *direction* 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.

4.1.6 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)

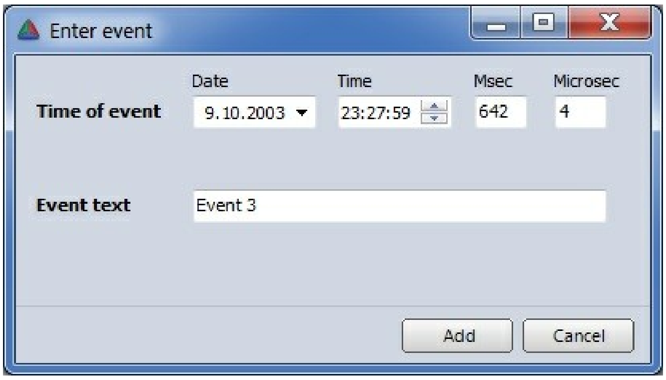
- if you click on any ***notice*** event, it will show in the EVENT section, and just by *hovering* the mouse over the gray *vertical* line, the *text* that you typed in *at* the time of recording
- ***Keyboard*** events show the *exact time* that they occurred

for information about ***marking recording data with events*** *see* → User Guide → **Manually Start / Stop Recording**

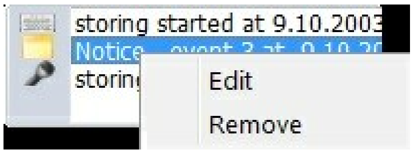
Manage events

As mentioned before, all **events** are *displayed* in the recorder.

To **add** an event just *clicks* on - **notice** icon or press the shortcut key, for example < n > 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.

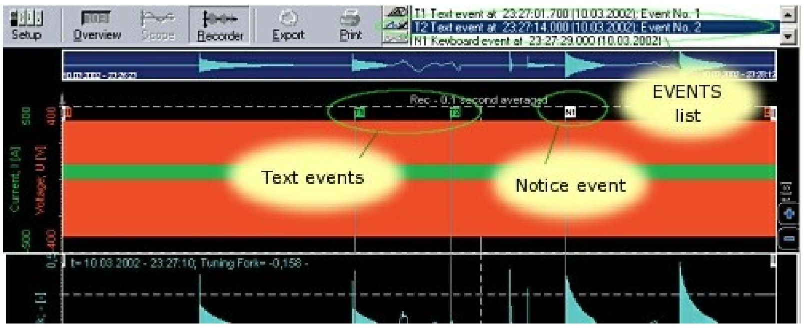


To **change** previous entered notice *right click* on this notice in the list, from displayed menu choose Edit and Enter event window will pop up. In this window now **Add** change to **Update** button, with which we accept changes to carry out in Event text field.



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*.



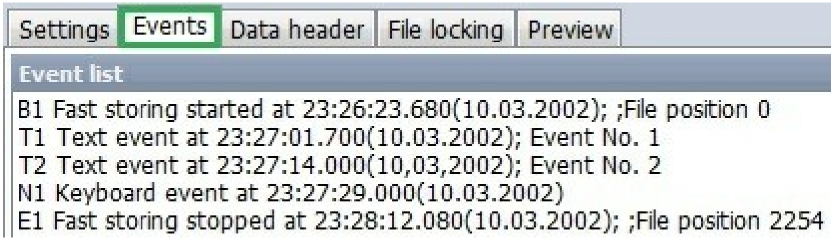
After all your new events are defined, select **Store settings and events** from the Data main menu (*see* also → User Guide → **Settings and Events**). All events will now be stored into the already *open data* file.

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***:



4.1.7 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.

for information about ***Load*** / ***Save*** *events* and changes in the *display appearance see* → **Store settings and events**

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.*

4.1.8 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. STEP

Include Video file to ***copy*** *video file* and ***rename*** it according **File name conventions**

2. STEP

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**

WARNING: AVI *format is required for synchronization. Other formats are not supported!*

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.

Name	Date modified	Type	Size
Example_Drive01.cam0.avi	4.12.2009 14:56	Video Clip	9.877 KB
Example_Drive01.d7d	7.12.2009 12:51	DEWESoft7 Data File	1.388 KB

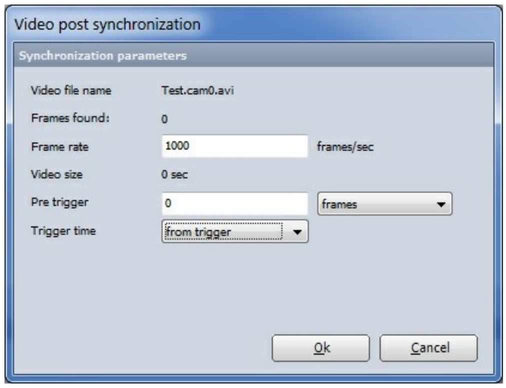
The picture above shows the example how this looks like. Original DEWESoft file is `test.dat` and therefore a video clip should be named test.cam0.avi.

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*.

The window displays the ***video file** name* and the *number* of stored ***frames*** (Frames found). *Change* the **Frame rate** to the *real capture* rate of the *camera* (sometimes the **.avi** files hold correct values, but most of the times not).



As an additional help, the Video size field shows you the *video duration* (length in seconds) at the currently *selected* frame rate.

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 *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 for 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:*

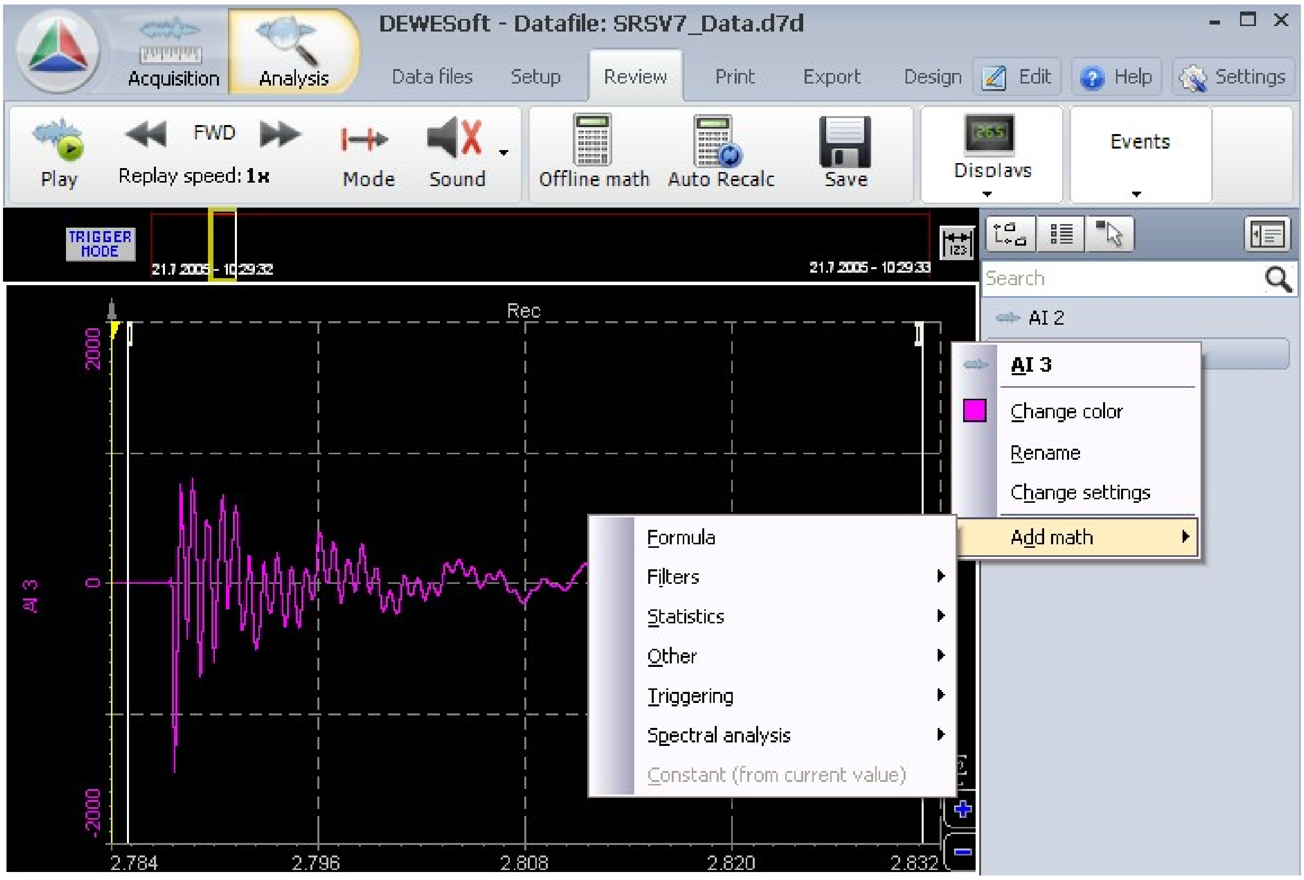
Rename the video back to the data file name and open it in DEWESoft - the video is *back* and still *synchronized*.

4.2 Postprocessing

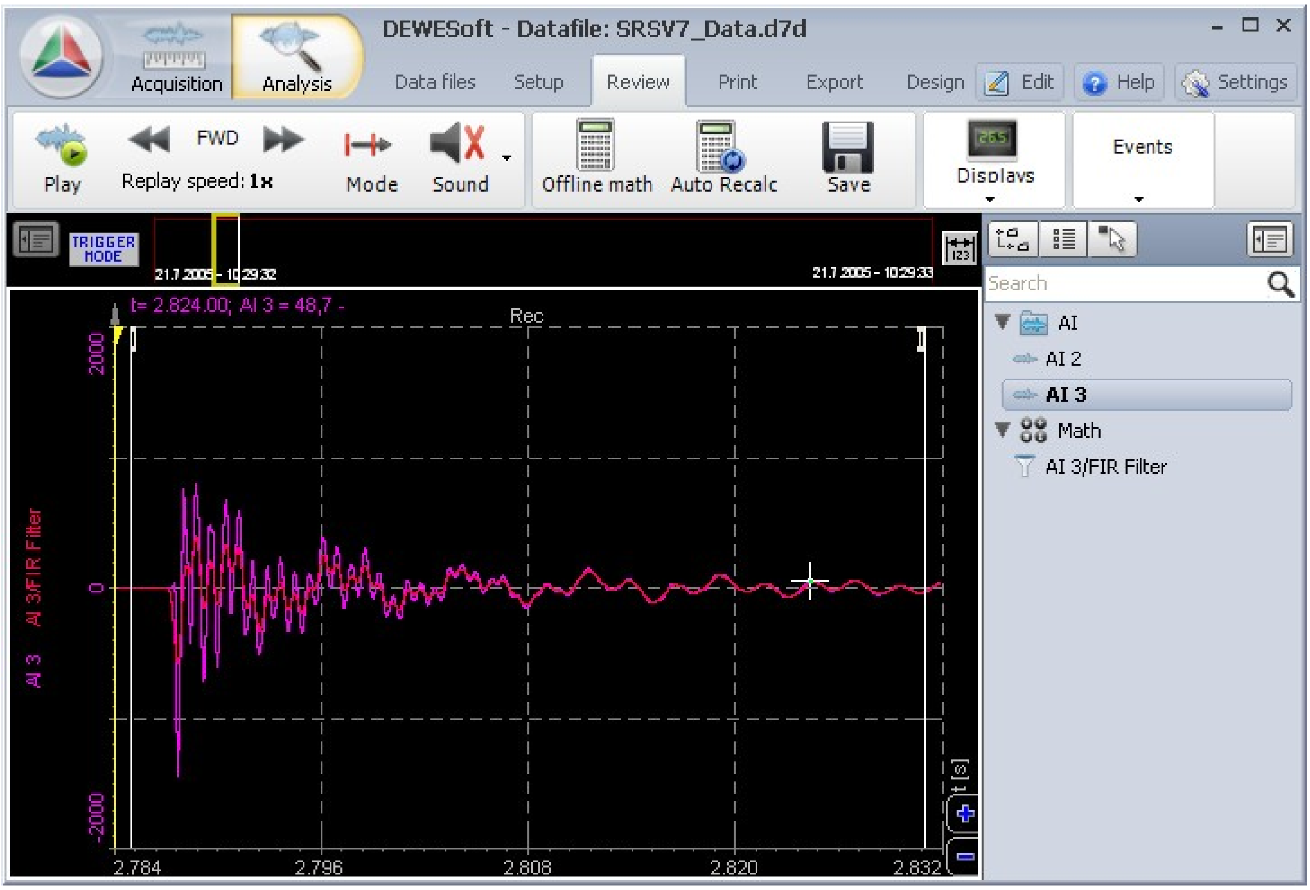
Post processing is a great feature which allows to:

- add *new math channels* in analyse mode
- *modify* existing math channels
- *change scaling* of online or offline channels.

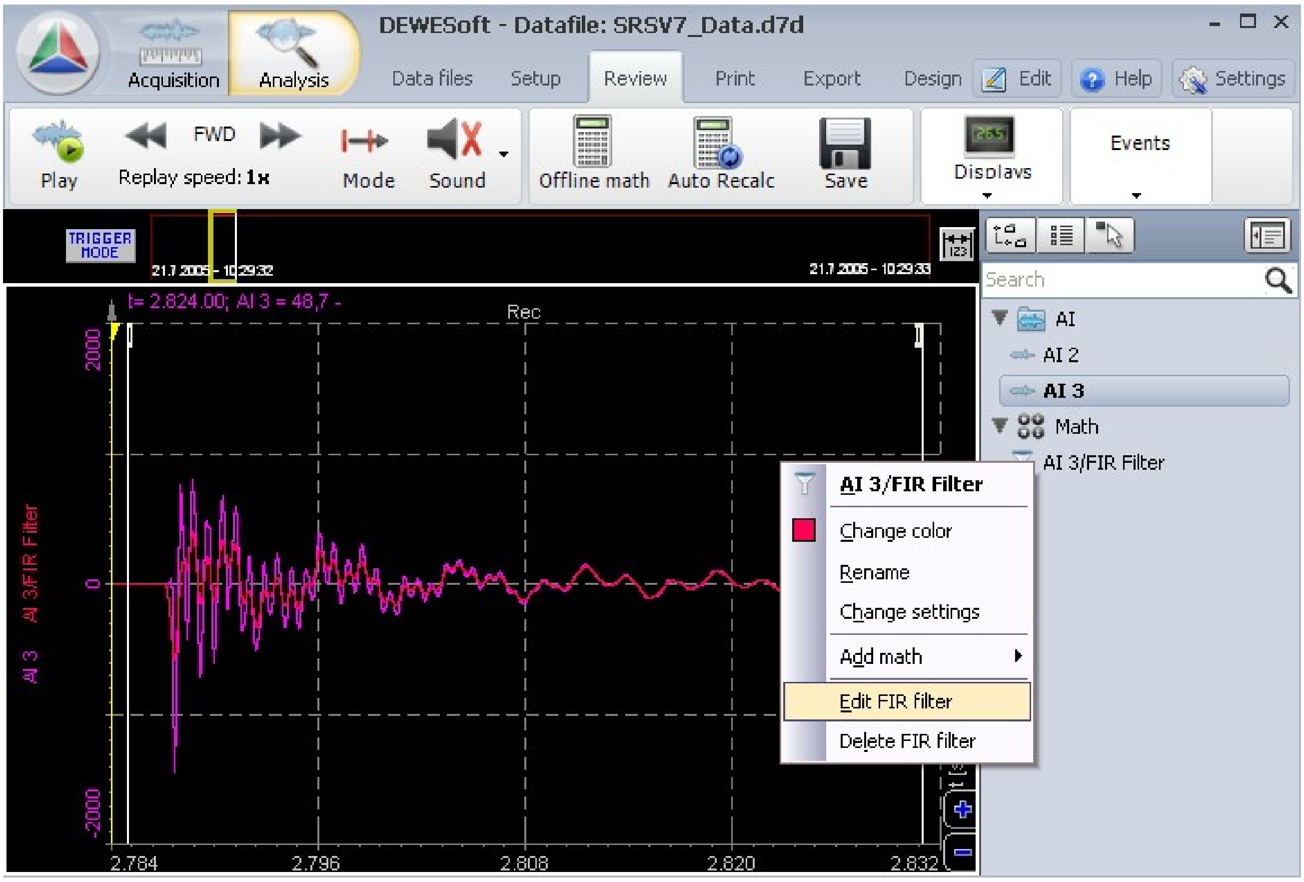
So you could store only the *raw* data and then in analyse perform all needed **mathematics**. There are several ways to perform math analysis on already acquired data. Let's start with a simple one. To add a simple math channel like filter we *select* the channel from *channel selector*, right click and select Add math and then choose appropriate math.



In this example I have added the FIR filter. When filter is set, data is automatically recalculated and put into current display.



Next we 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**).



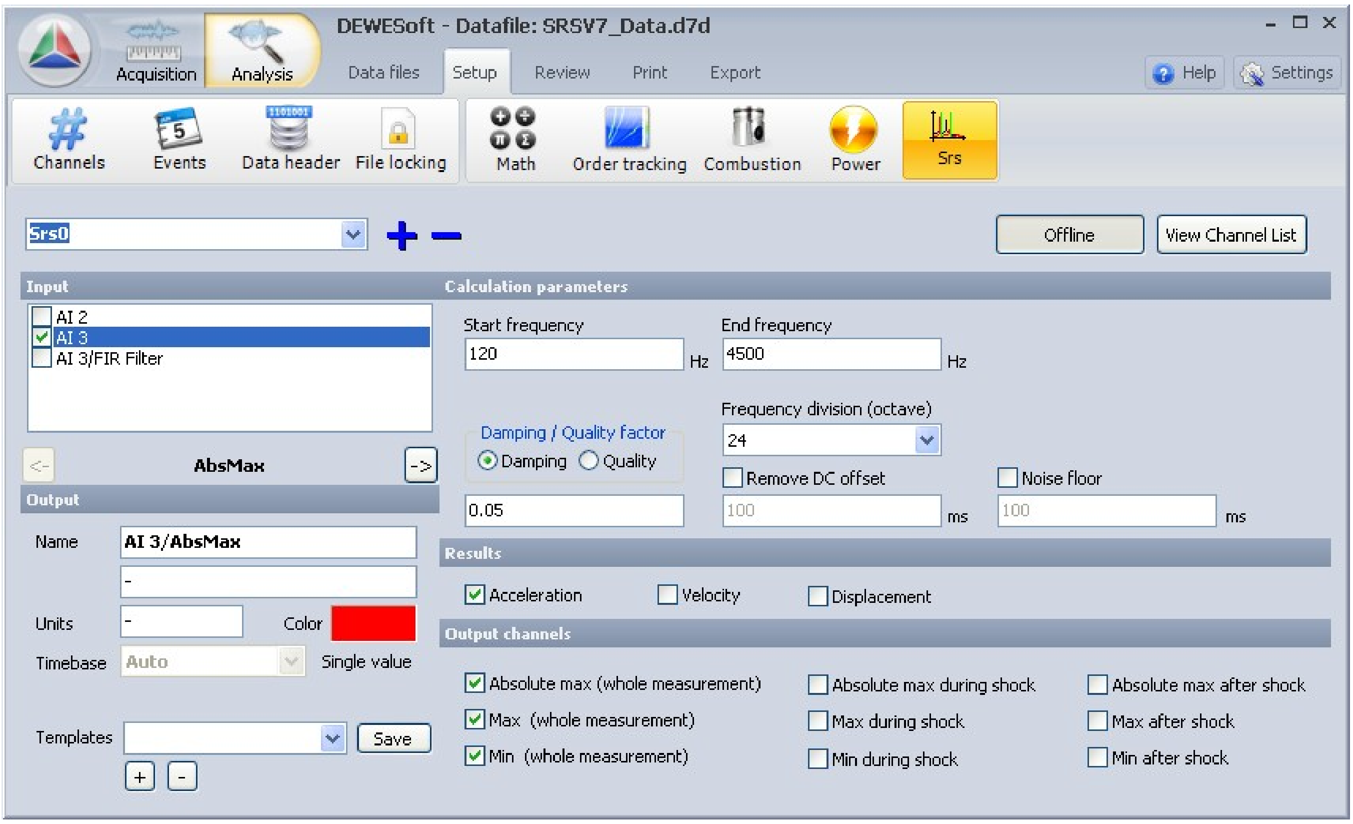
A window of chosen math will pop up, allowing us to change calculation parameters. Upon closing the channels will be *automatically recalculated*.

IMPORTANT: *data file will be recalculated only on chosen time interval. To reduce the time needed to recalculate huge data files, we can zoom in and define all needed calculations on small area and then zoom out, press Recalculate and get the cup of coffee when DEWESoft recalculates the whole file.*

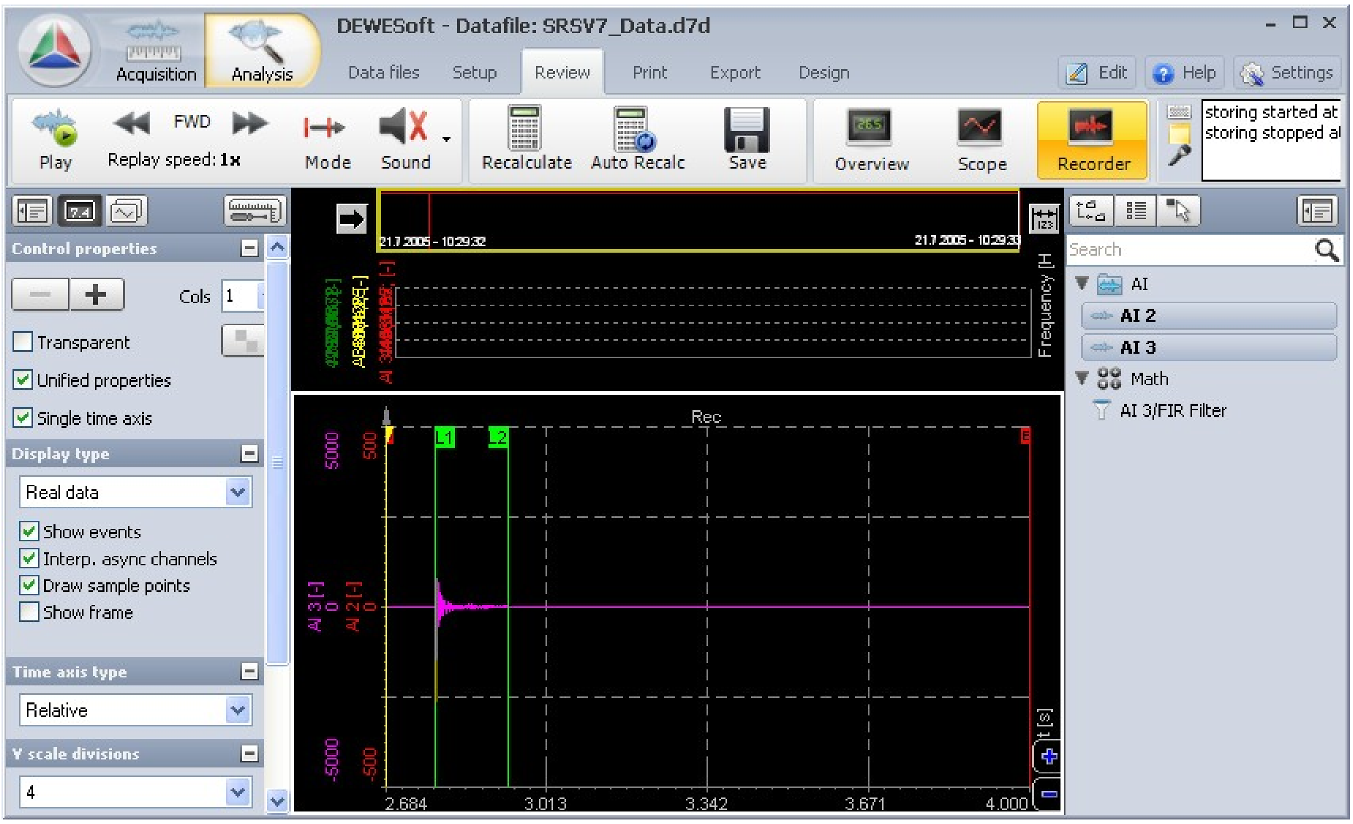
More complicated math can be defined in the math section by pressing **Offline math** button or **Setup** button from top menu.

Now we prepare math channels like in *Measure* → *Setup* mode and add advanced analysis like Power, Order tracking, Order analysis and all others.

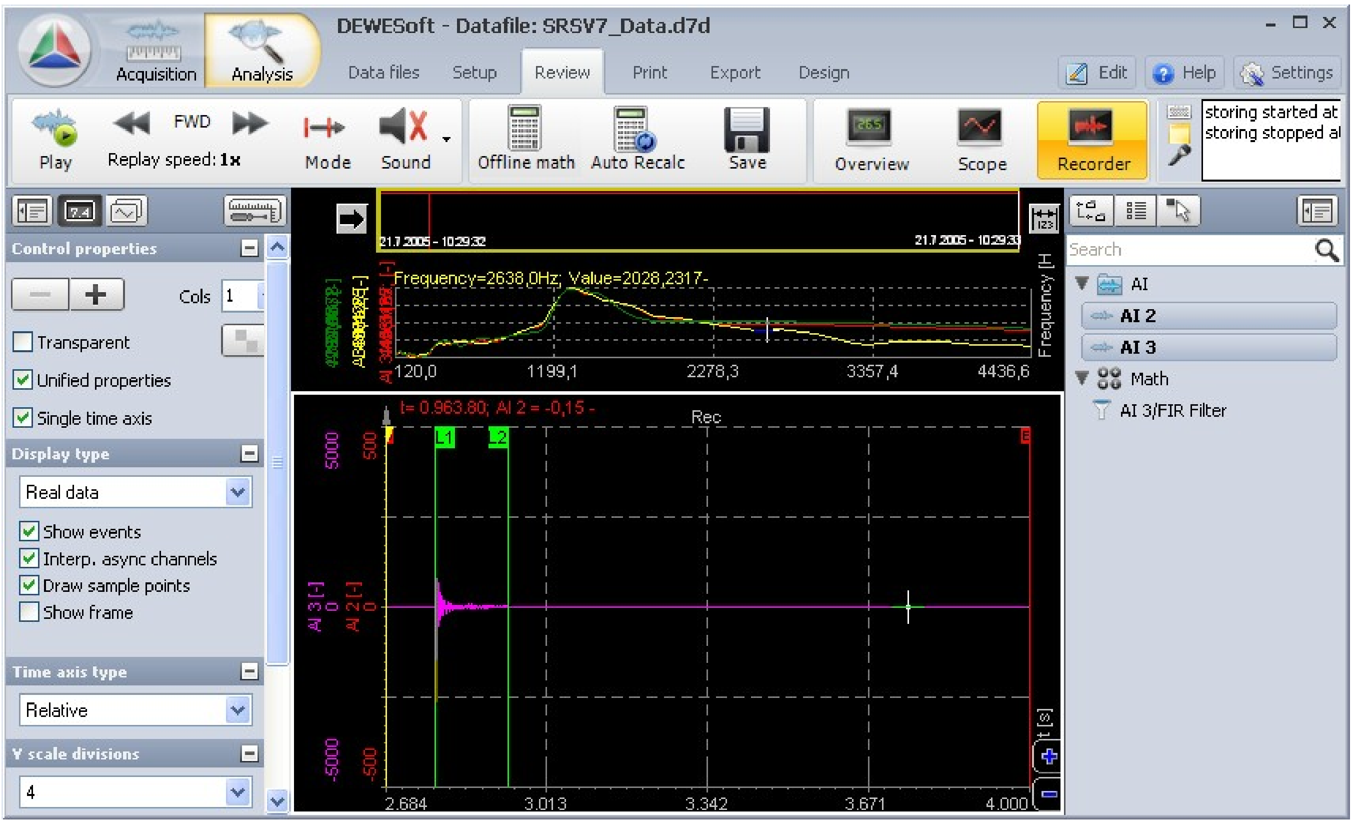
In our example let's continue to add **SRS** (shock response spectrum) from the Setup menu.



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 channel selector topic. We can *change* the color, scale and offset of any channel.



NOTE: *Lengthy calculations can be interrupted with Escape key on the keyboard.*

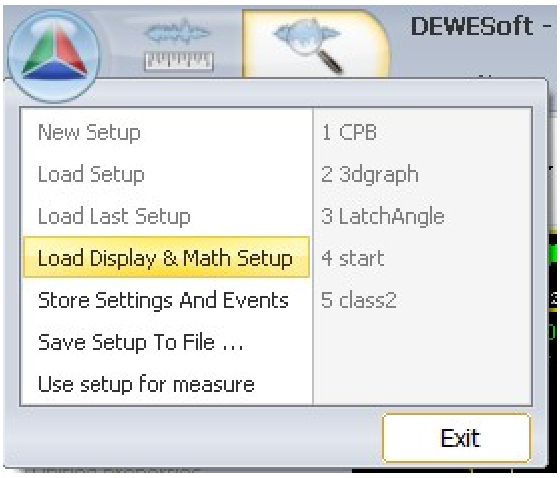
We can also choose that DEWESoft *automatically recalculates* the data when zooming in or out. This is for example useful when searching for absolute maximum for a specific time slice.

We 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 *entire* time of the file, *also* the recalculated results will be stored to the data file.

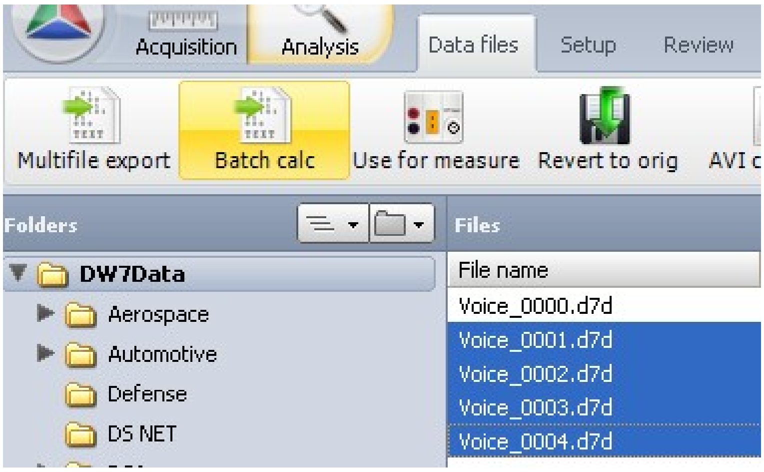
4.2.1 Batch calculation

When we have worked on **math analysis** for a *single* data file, this calculation can be applied to *several* data files. One chance is to press the round DEWESoft button and choose the **Load Display&Math** and *choose* the appropriate setup or data file holding the right calculation. Pressing **Recalculate** and **Save** will finish the process.



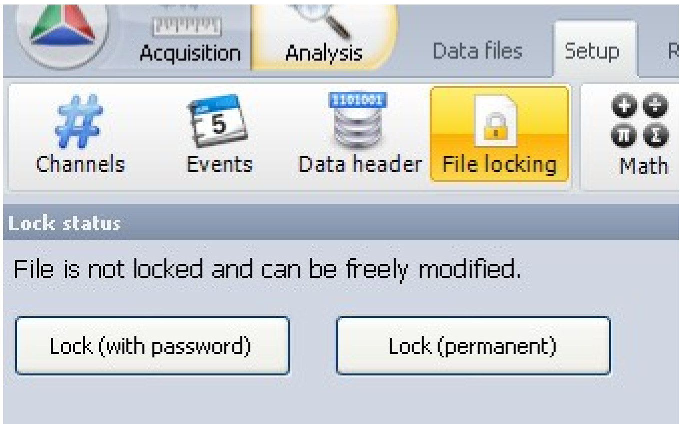
Another option is to *choose multiple* files in the Analysis → Data files and then choosing **Batch calc**. Select an *original* setup or data file *with defined* offline math. Make sure that the channels in *selected* files are the same as in *original* file.

A progress bar will be shown and all files will be recalculated.



4.2.2 File locking

We have seen 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 password which must be entered next time the file is opened or we can also lock the file permanently.



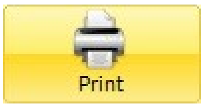
The file locking can be already done *while measuring* with Project settings → Security.

4.3 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**
  - **Multi page printout**
- **Copy channel setup** — copy to ***clipboard***
- **Copy Visual elements** — copy to ***clipboard***, than paste this data in *other applications*

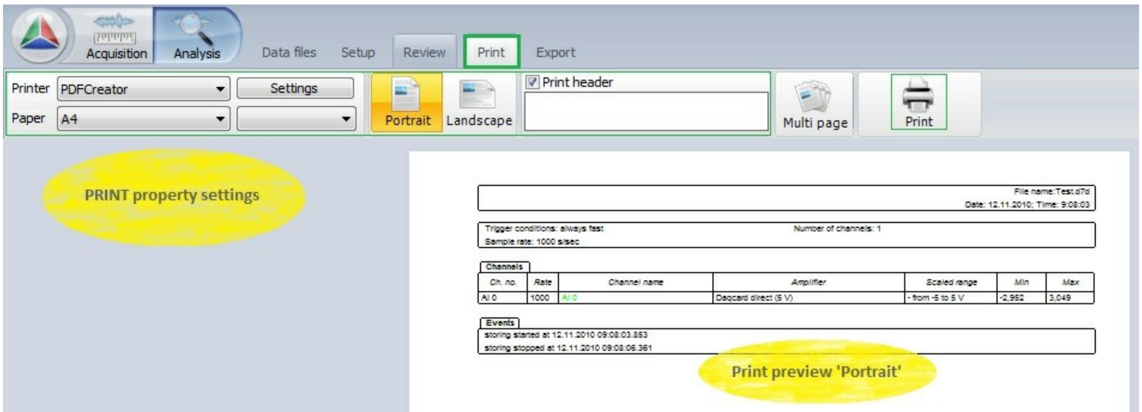
4.3.1 Screen printout



Print button.

In the *Analyse* mode, the 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

The following screen will appear:

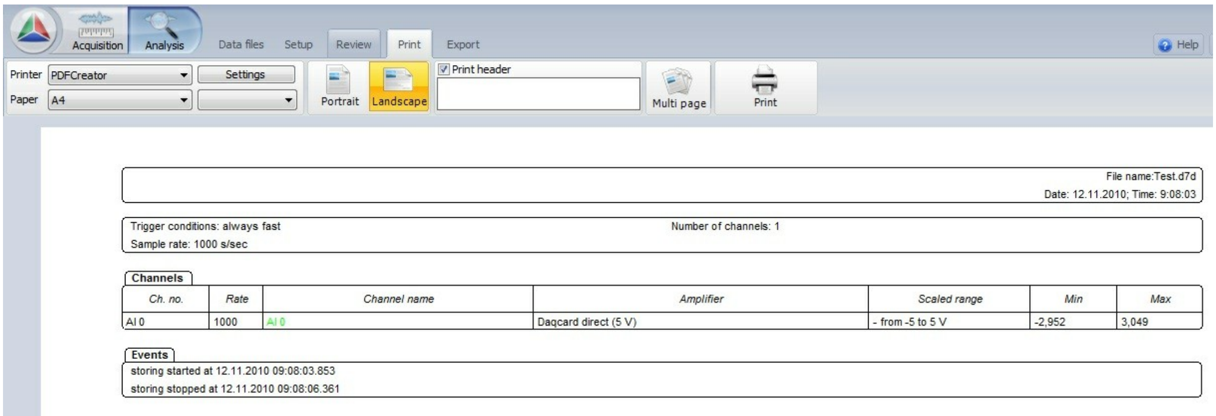


To **leave** the *printout* window simply *click* on any <u>other</u> instrument to change directly, for example the recorder.

On PRINT MENU can be set different ***Print properties***:

Page orientation

Select between **Landscape** and **Portrait** (*see* above) format:



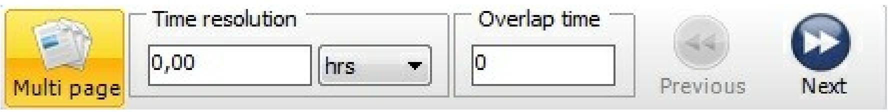
Notes on measurement and page header

In Notes on measurement field enter a ***comment****:*

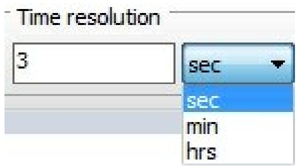
Beside this comment on page header appear: *license information, file name, page / pages number, date* and *time of measurement, show time.*

Multi page printout

Multi page printout can be selected by *checking* box by this field:



After that *select* time unit: sec, min or hrs from drop down list and *enter* **Time resolution**:



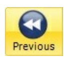
Bellow *Time resolution* unit field the page / from pages number of multi page printout is displayed.

> *Example:* when for measurement with duration 1min 35sec the *Time resolution*=40sec is entered, multi page printout with 3 pages in all is defined → first page for measurement time 0 ÷ 40sec, 2nd for 40 ÷ 80sec and 3rd for 80 ÷ 120sec.

In field bellow *Time resolution* value can be eventually set also **Overlap time**. The overlap time is the time which will be displayed twice in the printout - on the end of first page and in 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 → first page for measurement time 0 ÷ 40sec, 2nd for 25 ÷ 65sec, 3rd for 50 ÷ 90sec and 4th for 75 ÷ 115sec.

We can *display* ***preview*** of every page in multi page printout → use 

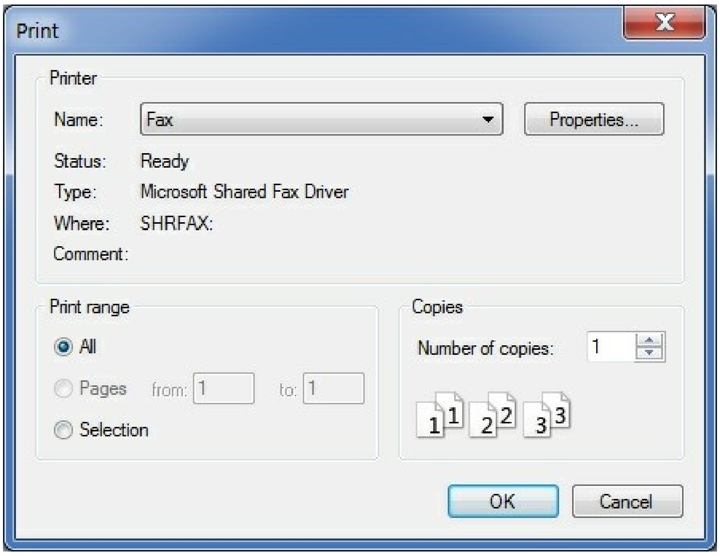
button for navigation *up* and 

button for navigation *down* between pages.

Printing

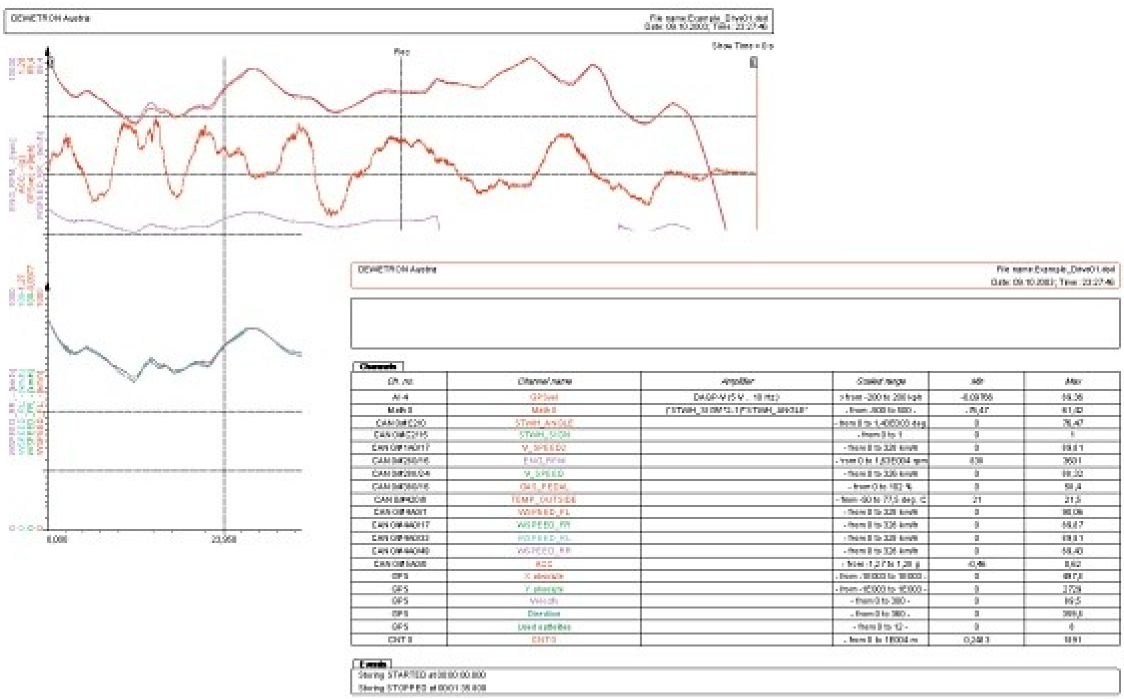
After all desired properties are set press the **Print** command button below print menu settings. This will open the *standard Windows® Print window*, where you may define your printer *properties*, *print range* and *copies* as in other programs and *start* the printing job.

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 use also the printer spooler.



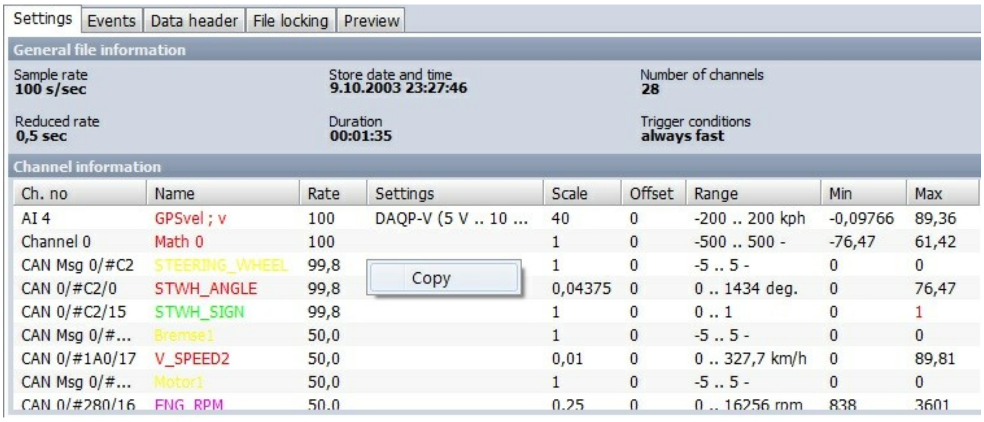
After starting the printout, you can start *immediately continue working* with DEWESoft, even if the printer is still running.

Example: typical ***Setup*** and ***Recorder*** printouts in landscape format:



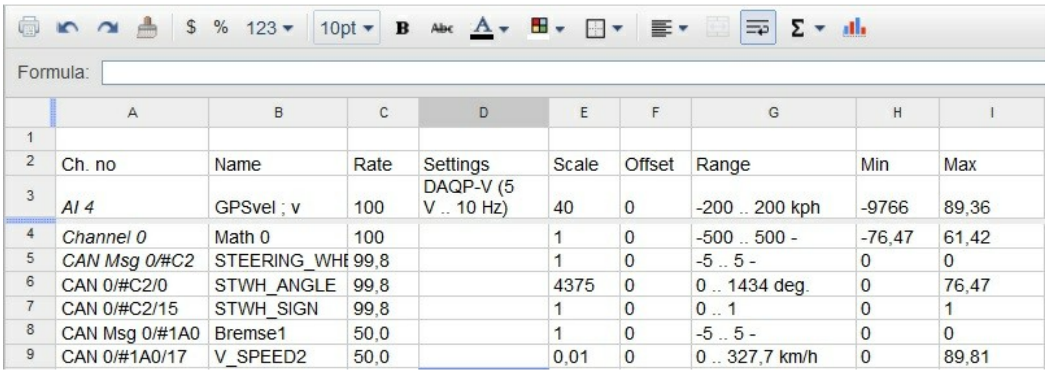
4.3.2 Copy channel setup

If you work with plenty of channels, it would be nice to have them also *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* anywhere within the channel list and select **Copy** from the appearing menu.



Ch. no	Name	Rate	Settings	Scale	Offset	Range	Min	Max
AI 4	GPSvel ; v	100	DAQP-V (5 V .. 10 ...	40	0	-200 .. 200 kph	-0,09766	89,36
Channel 0	Math 0	100		1	0	-500 .. 500 -	-76,47	61,42
CAN Msg 0/#C2	STEERING_WHEEL	99,8		1	0	-5 .. 5 -	0	0
CAN 0/#C2/0	STWH_ANGLE	99,8		0,04375	0	0 .. 1434 deg.	0	76,47
CAN 0/#C2/15	STWH_SIGN	99,8		1	0	0 .. 1	0	1
CAN Msg 0/#...	Bremse1	50,0		1	0	-5 .. 5 -	0	0
CAN 0/#1A0/17	V_SPEED2	50,0		0,01	0	0 .. 327,7 km/h	0	89,81
CAN Msg 0/#...	Motor1	50,0		1	0	-5 .. 5 -	0	0
CAN 0/#280/16	ENG_RPM	50,0		0,25	0	0 .. 16256 rpm	838	3601

Now open and change to the other application and **paste** it.

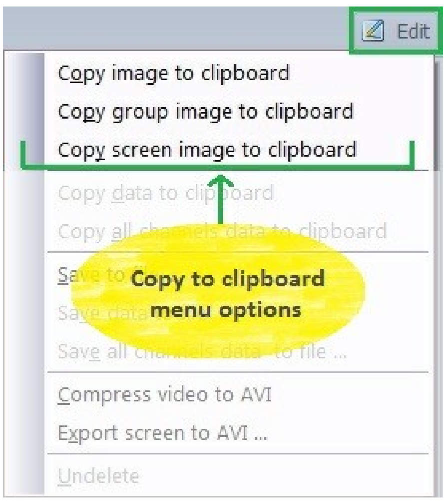


	A	B	C	D	E	F	G	H	I
1									
2	Ch. no	Name	Rate	Settings	Scale	Offset	Range	Min	Max
3	*AI 4*	GPSvel ; v	100	DAQP-V (5 V .. 10 Hz)	40	0	-200 .. 200 kph	-9766	89,36
4	*Channel 0*	Math 0	100		1	0	-500 .. 500 -	-76,47	61,42
5	*CAN Msg 0/#C2*	STEERING_WHE	99,8		1	0	-5 .. 5 -	0	0
6	CAN 0/#C2/0	STWH_ANGLE	99,8		4375	0	0 .. 1434 deg.	0	76,47
7	CAN 0/#C2/15	STWH_SIGN	99,8		1	0	0 .. 1	0	1
8	CAN Msg 0/#1A0	Bremse1	50,0		1	0	-5 .. 5 -	0	0
9	CAN 0/#1A0/17	V_SPEED2	50,0		0,01	0	0 .. 327,7 km/h	0	89,81

NOTE: *This feature is also available in the settings during file selection.*

4.3.3 Copy visual elements

For documentation in *external* applications, you can **copy** the *display elements* also into the clipboard.



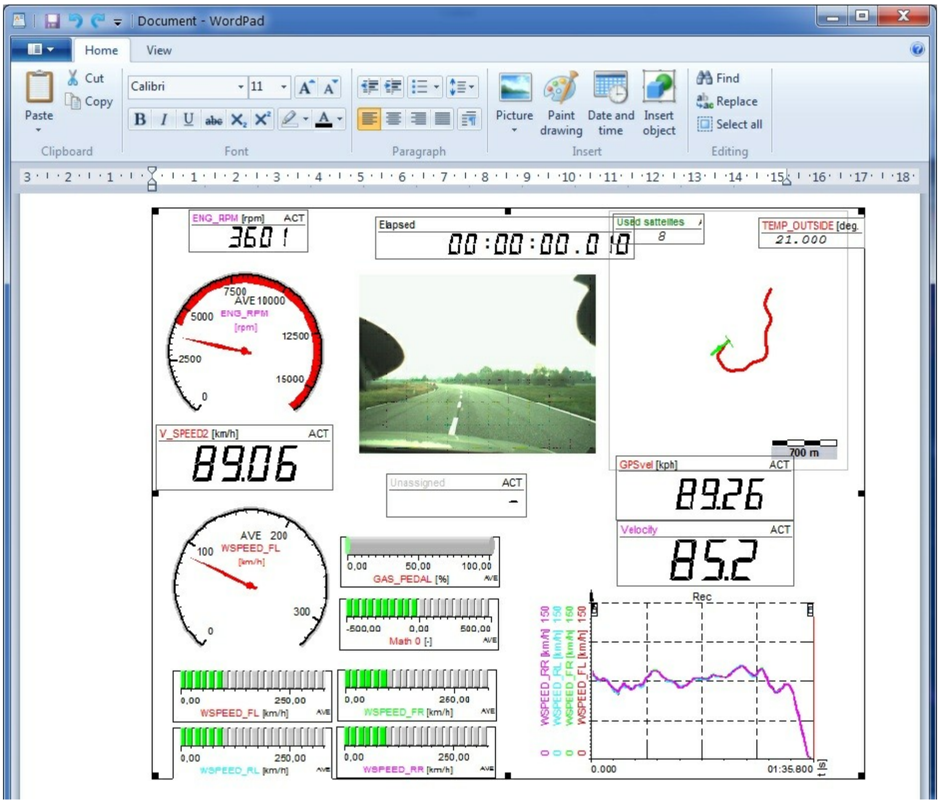
Select the desired instrument and *select* from the **Edit** menu:

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 external application



4.4 Exporting data

The important purpose of the DEWESoft Analyse mode is also to **export** *acquired data* for ***off-line analysis*** using *other* software.

DEWESoft is intended to be an **acquisition package.** We 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 following procedures:

1. STEP

Set export data properties — to define ***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 — select another software application as target for data export and ***off-line analysis***:

- **Microsoft Excel**
- **File Export**
- **Clipboard**
- **FlexPro**
- **DEWESoft**

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

We have also two special procedures to:

- **Export multiple files**
- **Export instrument display to Video**

Time range of data

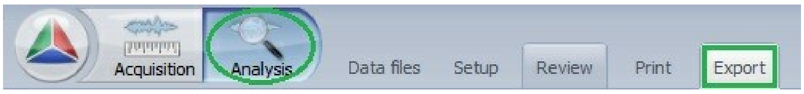
To export only **part** of acquired data *select time range* on the recorder or vertical recorder with *zooming* in the portion of the data.

for detailed information about ***Time range selection*** *see* → User Guide → **Selecting data**

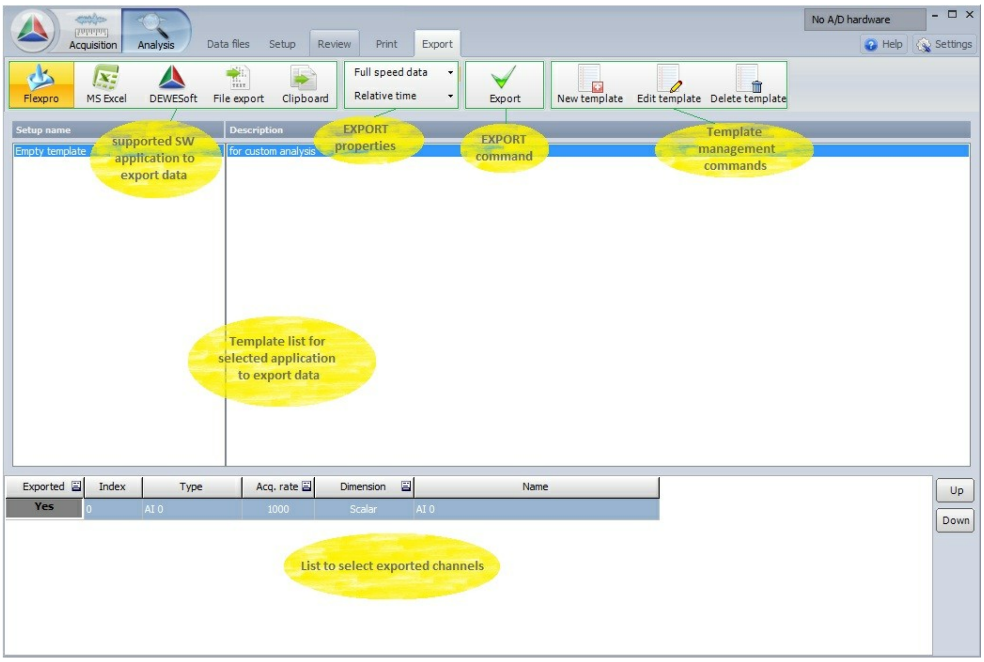
NOTE:	*Be aware that only selected portion of data will be exported. To export full range of data, zoom out first to entire length of the data.*

To **export** data, press the 

- **Export** button on main DEWESoft tool bar, 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:

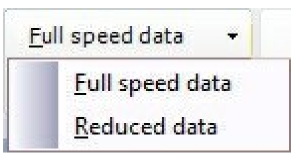


Set export data properties

Data export

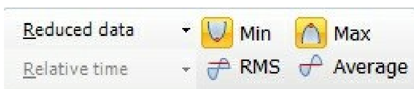
Select ***type*** of data from Data export drop down list:

- **Real data**

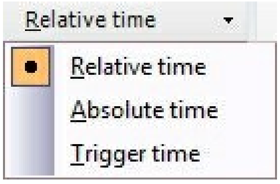


- **Reduced**

  For this type of data can be selected by checking appropriate box: **Min**, **Max**, **RMS** and/or **Average** type.



Time axis



Select *type* of time scale from Time axis drop down list:

Relative, **Absolute** or **Trigger** type.

Relative axis will start with time 0 at the *beginning* of the *measurement*.

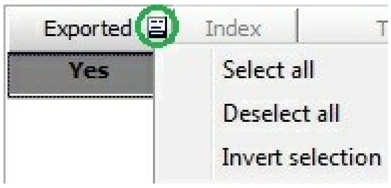
Absolute axis will export *absolute* date and time of *measurement* as the time axis.

Trigger time axis will start with time 0 at the *trigger point* (*pretrigger* will show *negative* time values).

Exported channels

The next step is to *select* the *channels* to **export** from displayed *Channels* list (at the bottom of the window).

Selection can be made in two ways:



- Click on the icon in the Exported column to select from displayed menu:
  - **Select all** channels
  - **Deselect all** channels or
  - **Invert selection**

- Click **Yes** / **No** button in the Exported column to toggle between this choices to ***export*** / ***don't export*** *particular* channel.

As a standard, *all* channels will be exported.

Channel can be **moved** with click on Up button *up* and on Down 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 main Data menu.

Export option - formats

DEWESoft support 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 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 offer *easy* export to this format, and advanced *automation* when data is exported to this program, including the ability to run custom *macro* 'scripts' inside.

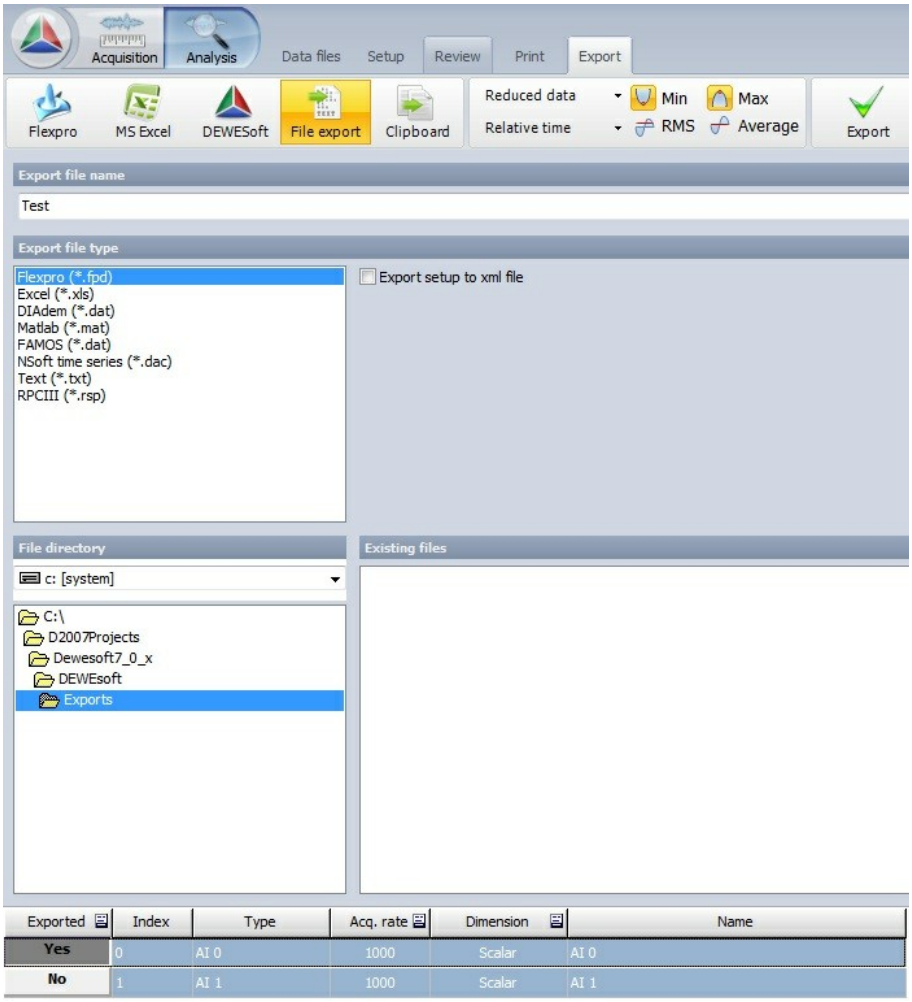
The other file formats except *FlexPro* and *MS Excel* just prompt you for a ***filename***, which you supply using a standard Windows file dialog box. Click **OK** and the software will do the rest. Be sure to supply the proper filename ***extension***.

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 3-D 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

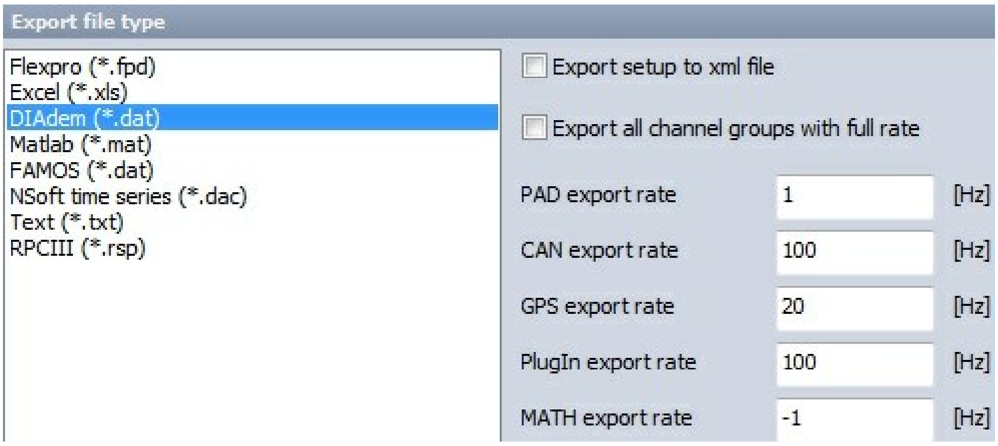
If you select the **File Export**, following display appears:



On central part following *Export to different file formats* sections is displayed:

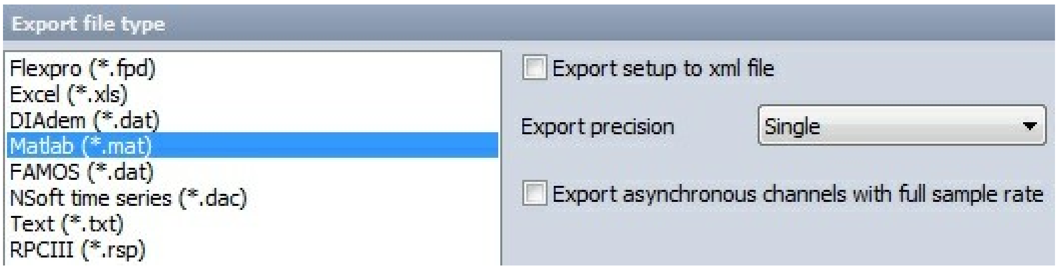
Export file name	*name* of exported *file*; enter name according to OS name convention
File directory	*name* of *directory* where exported file to be saved; select with double click from drop down list
Existing files	*list* of *existing* files same type in selected directory
Export file type	list of available *type exported file* with optional file settings for:

- `DIAdem *.dat` file type



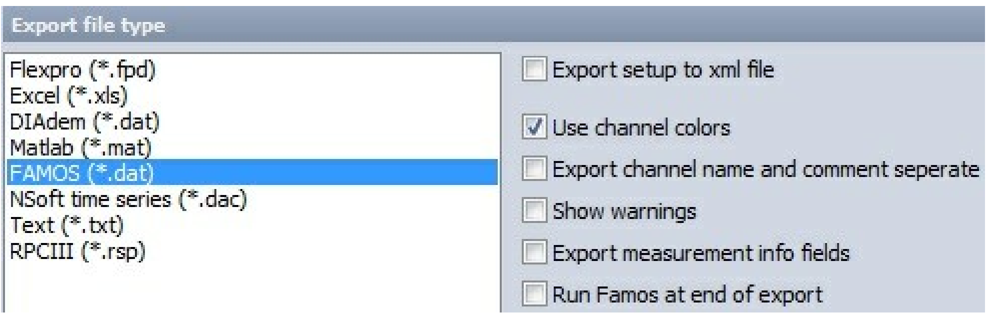
Diadem has an option to export certain groups of channels at specific sample rate, so the resulting data file is shorter. We can achieve this by de selecting **Export all channel groups with full rate** and defining the *rate* for each channel group (PAD, CAN, GPS, Plugin and MATH). If -1 is entered, the data will be exported at ***full rate***.

- `Matlab *.mat` file type



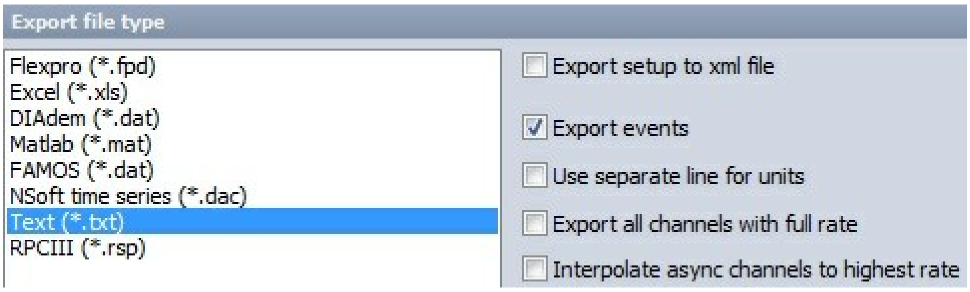
Export precision can be selected from drop down list between **Single** and **Double**. Usually single precision is enough, but sometimes Matlab functions expect data in double precision. Please be aware that the amount of exported data will be *doubled* if this option is chosen and Matlab files <u>can't exceed</u> 2 GB. Matlab can also export *each* channel with *separate time* channel or exports *all* data with *full speed* if Export asynchronous channels with full sample rate check box is checked.

- `FAMOS *.dat` file type



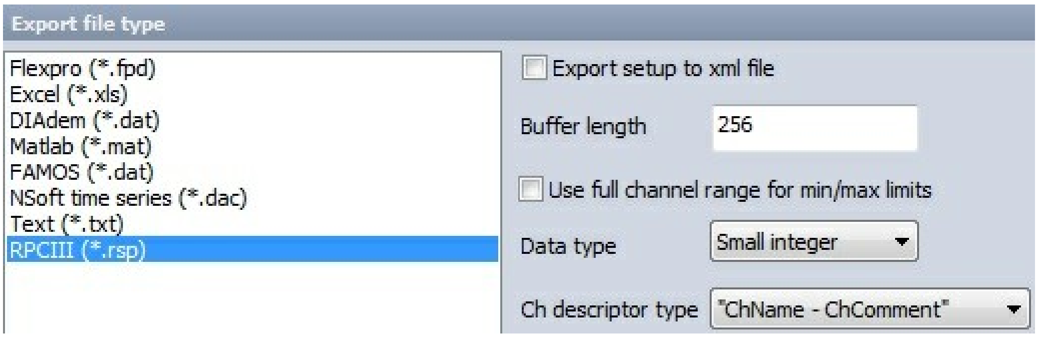
Use channel colors can be checked to use DEWESoft *channel colors* otherwise *default Famos* color is chosen.

- *Text* `*.txt` file type



With checked Export events also ***events*** will be *exported* in text file.

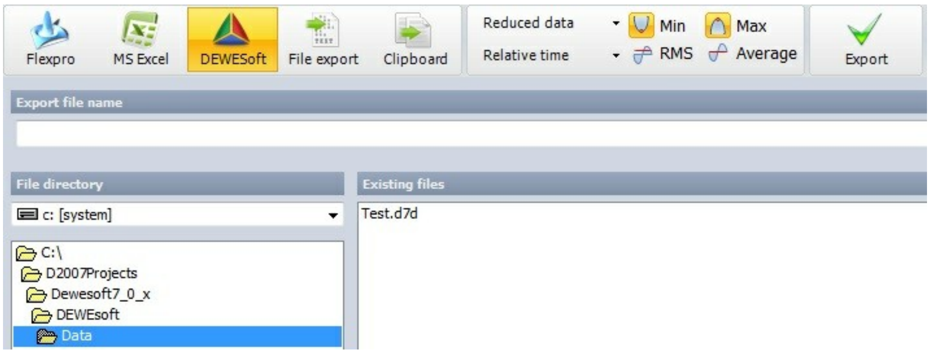
- *RPCIII* `*.rsp` file type



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:

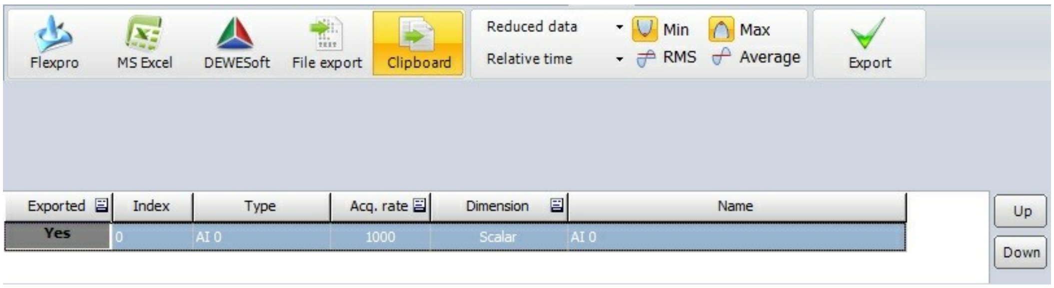
Export file name	*name* of exported *file*; enter *new* file name according to OS name convention
File directory	*name* of *directory* where exported file to be saved; select with double click from drop down list
Existing files	*list* of *existing* files same type in selected directory

When **Export to DEWESoft** is selected, *Channels list* disappear, therefore ***all*** *channels* will be exported.

After selecting *time range* on the *recorder* or *vertical recorder* display (*see above* → **Time range of data**), entering Export file name and selecting **Export 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).

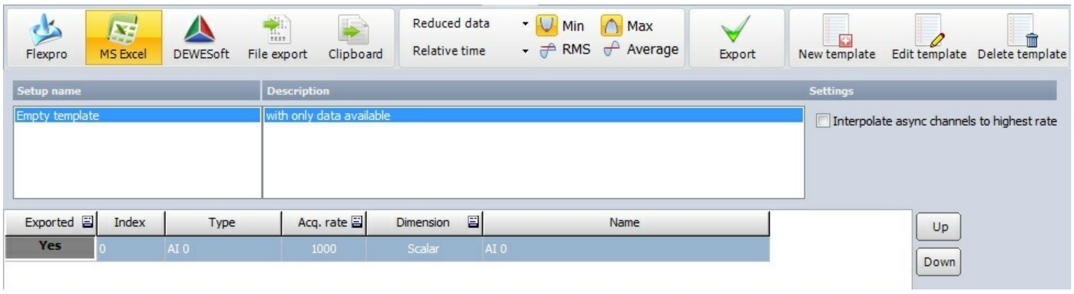
4.4.1 to FlexPro and MS Excel

To enhance **export data** to *Microsoft Excel* and *FlexPro*, following procedures can be used:

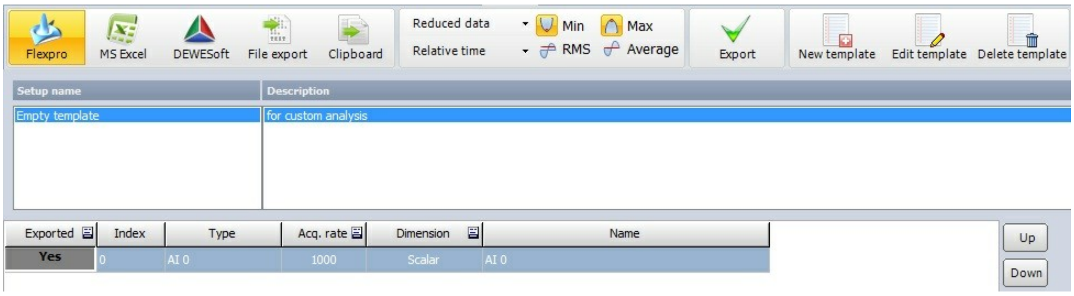
1. STEP **Select application**	click on either **FlexPro (ActiveX)** or **MS Excel (ActiveX)** on Export option
2. STEP **Template Management**	**Template Management controls** are only available for *Microsoft Excel* and *FlexPro export option* to manage ***template - scripts***, which are listed on central Postprocessing 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 Postprocessing 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 *preformatted 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.*

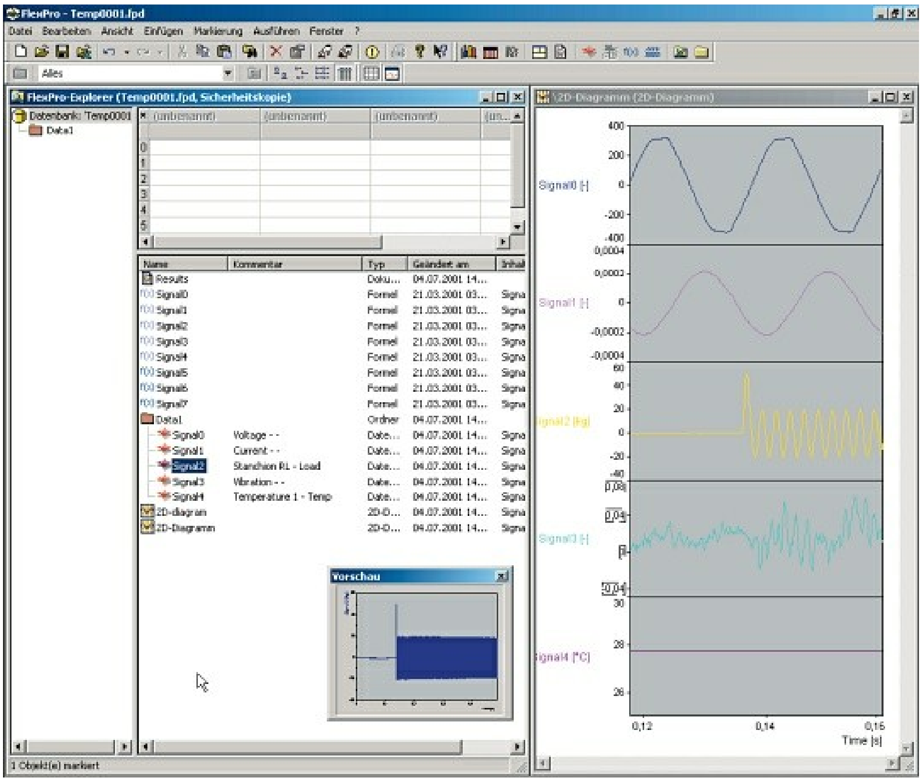
The *concept* is a nice one: once the script is created, just *load* any ***data file*** and *run* the **script**. What happens next is a big time-saver. DEWESoft runs this script:

- *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 dataset and
- *runs* the script.

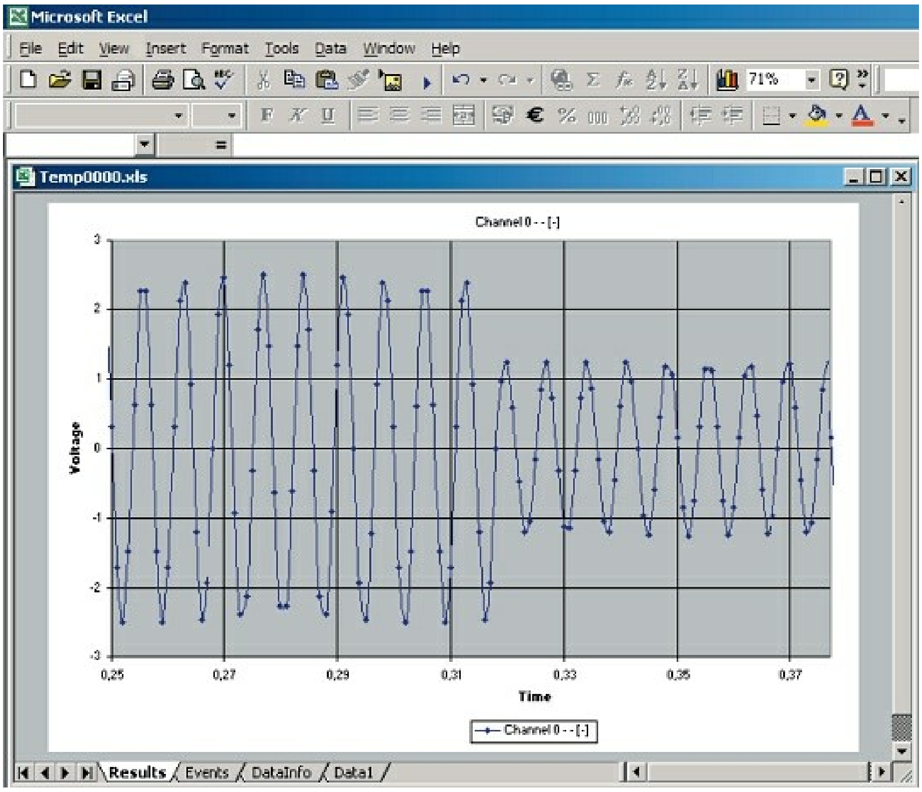
Let's say that you have a script that takes four incoming signals, runs 3rd octave analysis on them, creates a polished looking report, complete with your company's title bar and logo, automatically puts in the data and other variables from the data file, and formats it for your color printer. After 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.

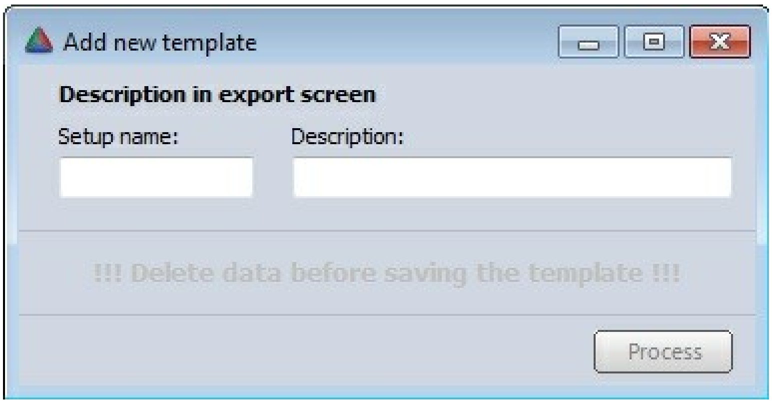
Template management controls

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*

for information about ***creating FlexPro script*** also *see* → Reference Guide → **Scripts**

4.4.1.1 MergeDB - FlexPro

The MergeDB script allows the user 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!*

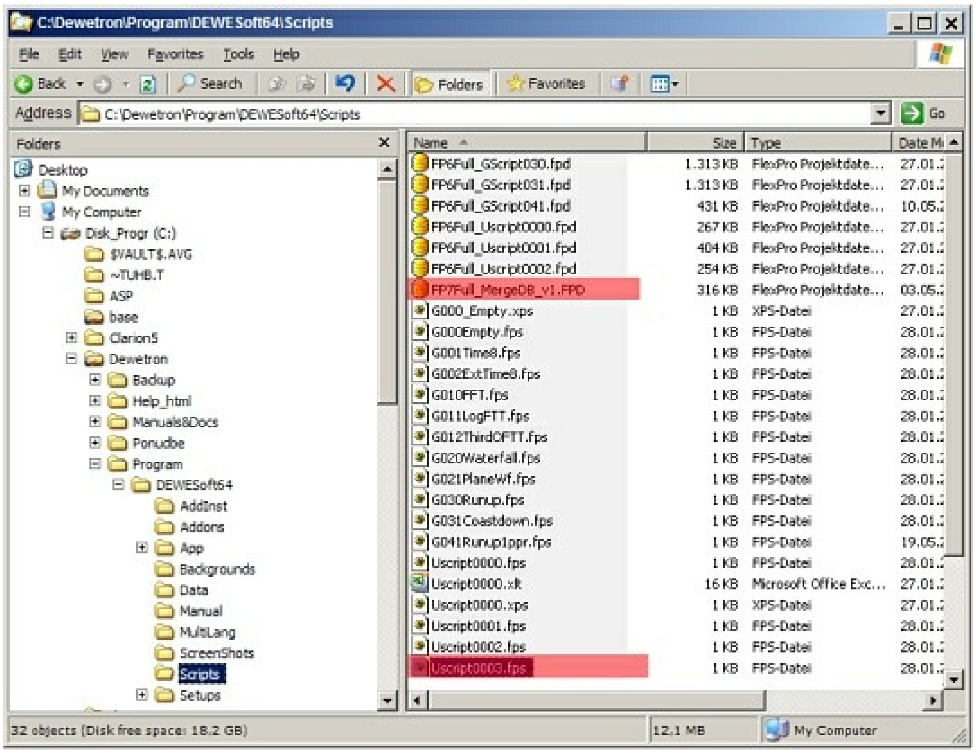
Installation

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:



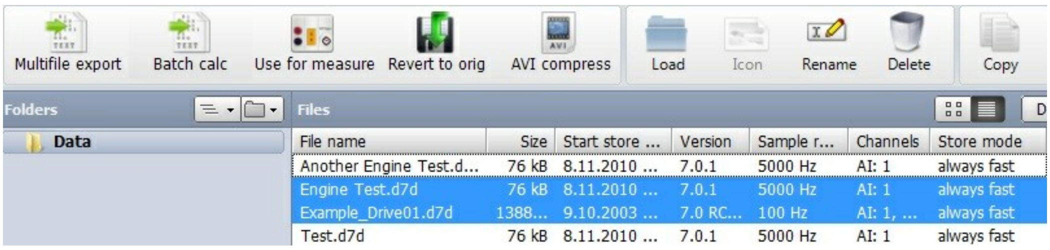
Usage of MergeDB

After selecting *MergeDB-Script* on the *DEWESoft export* screen Merge Database window 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*.

4.4.2 Export multiple files

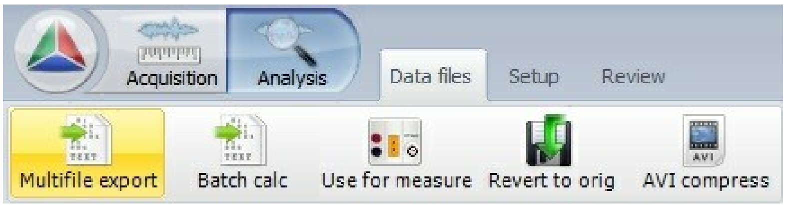
If you want to **export** *several files* at *once*, there is a nice function available. Press the **Analyze** button to enter the *file explorer*. *Select all* ***files*** you want to *export*.



How can we *mark* several files? There are two possibilities:

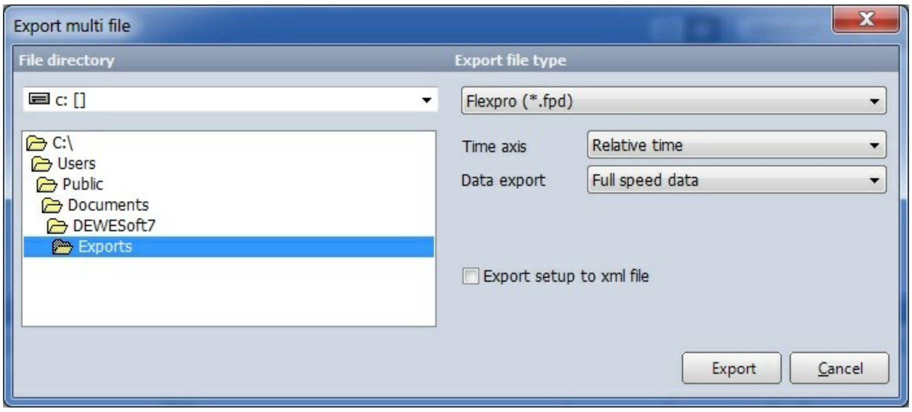
– to *mark several* files which are listed near to each other, keep the SHIFT button pressed *during* selection
– to *mark several* files picked out of the list, keep the Ctrl button pressed *during* selection

After all files to export are *marked*, select **Multifile export** from the Data menu.

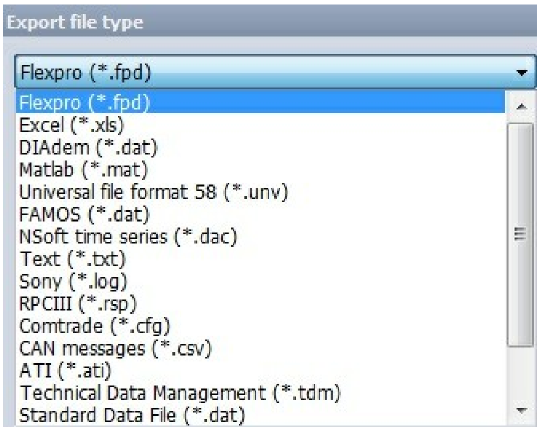


Now the *Export multi file* window appears. This window allows you to select:

- ***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 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:

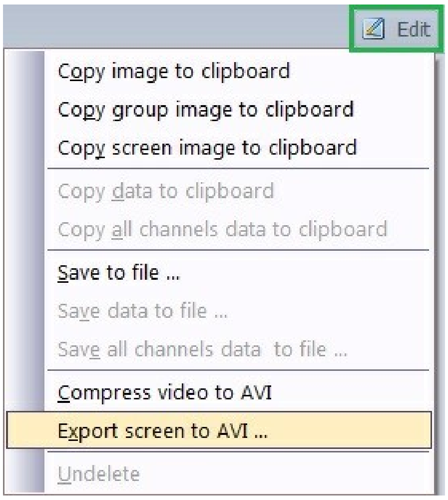
Press the **Export** button to start the *export* process or **Cancel** to leave *without* exporting the files.

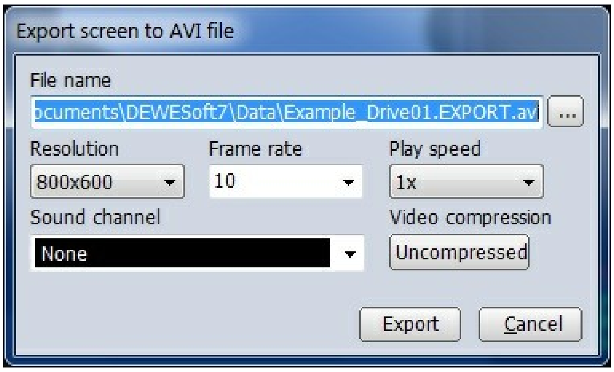
NOTE: *The export keeps the original file names for the exported files.*

4.4.3 Export instrument display to Video

A new way of presenting data is the ***multimedia documentation***. 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:

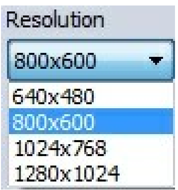




Please change the ***export settings*** on this window according to your requirements. The following settings can be changed:

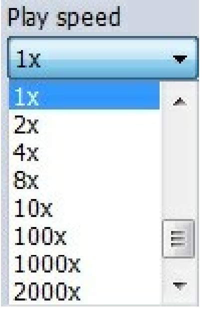
- **File name** and storage *path*; after pressing ... - *browse* button *Save As* standard window appear to select folder and file name → *see* also → **Data management**

- **Resolution**: from 640 x 480 to 1280 x 1024 selectable from drop down list - *high* resolutions will create *larger* files!



- **Frame rate**: from 1 to 30 - standard video player can not handle faster data; select from drop down list

- **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 drop down list



- **Sound channel**: select from drop down list one of the available *analog* channels to be your ***audio*** channel

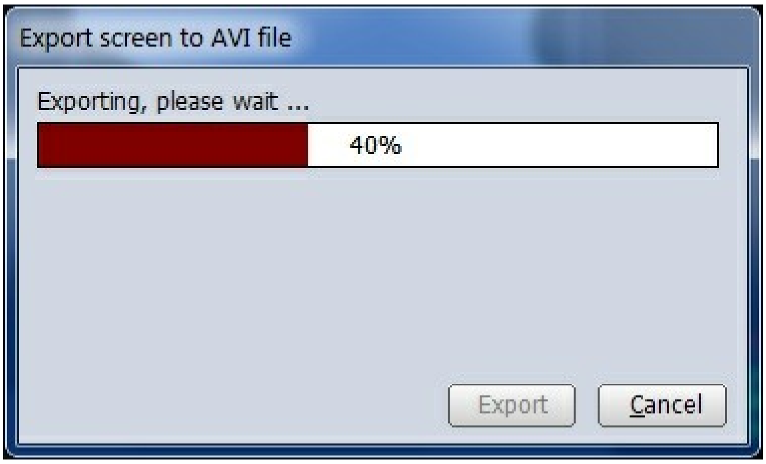
- **Video compression** → DivX codec or similar is recommended

Press Uncompressed button and Choose compressor window appear for selecting *standard compression* codec from the compressor drop-down list and for configuring them..jpg After confirmation of selected codec **Uncompressed** button in the Video compression field change to name of *selected* codec (e.g. XVID).

for detailed information about ***codec selecting and configuring*** *see* → **Video setup**

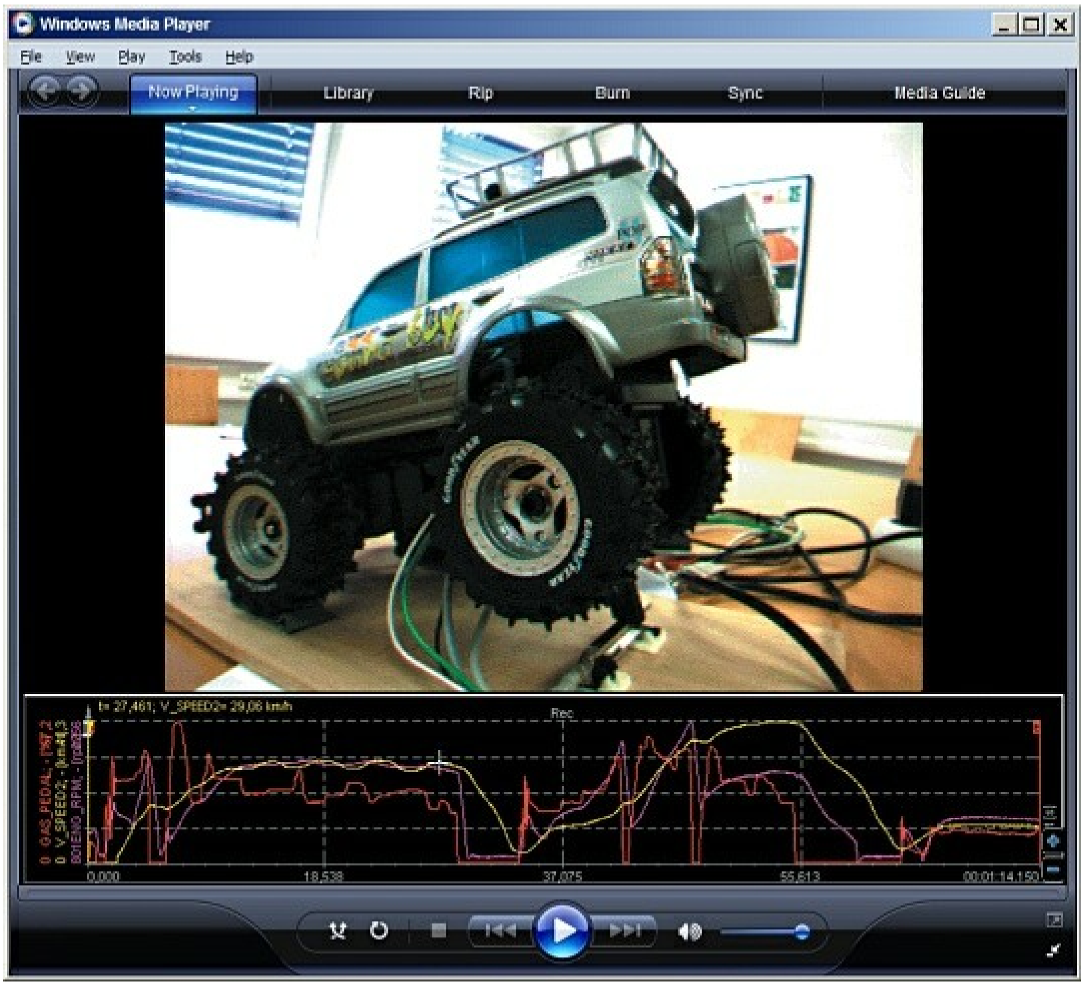
When you have done all changes press the **Export** button to start the **export** or **Cancel** to leave *without* exporting the video.

The *export progress* will be displayed in an *Export screen to AVI file* window.



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*:



5 Data management

In DEWESoft there are several typical files which are created for its purpose:

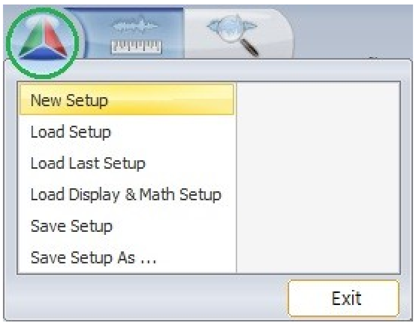
type of file	type of data	action
Setup data *.d7s (*DEWESoft setup* data *file*) or *.xml (*DEWESoft xml setup* file) - for detailed information about this file *see* → **XMLFileStructure** document	**Channel setup** setting of all *analog channels*, math*modules*, instruments appearance, storing options, multifile setting...	**New** **Load** and **Load last** **Load display setup** **Save** and **Save As**
Acquired data, settings and events *.d7d (*DEWESoft* data *file*)	**Measured data** measured data and uploaded data on NET **Complete Setup data** with all settings and events	**Record**, **Load** **Export** **Store** and **Load** settings and events
Project files *.d7p (*DEWESoft* project *file*)	**Project and Hardware setup**	**Hardware setup**
Sequence files *.d7t (*DEWESoft* sequence *file*)	**DEWESoft sequences** with option to include setup files as well	**Sequencer**
Video files *.dvi *file* and *.avi *file*	**Video files** online video data from *cameras* or *exported* instrument screen like *Overview*, *Scope*, *Recorder*,...	**Video setup** **Export display**
XML files sensors.xml, cntsensors.xml, modules.xml, CAengines.xml	**Sensors database**, **counter sensor database**, **amplifier transfer curves** and typical **combustion engines**	**Sensor database** **Counter sensor** **Engine templates**

5.1 Setup data

In DEWESoft *.d7s file (DEWESoft ***setup data*** file) or *.xml (DEWESoft ***xml setup*** file) *Channel setup* data and DEWESoft *Application modules setup data* are **stored**.

WARNING: *DEWESoft setup data file does not include any measured data!*

New Setup data



We can start with a **new Setup** by selecting:

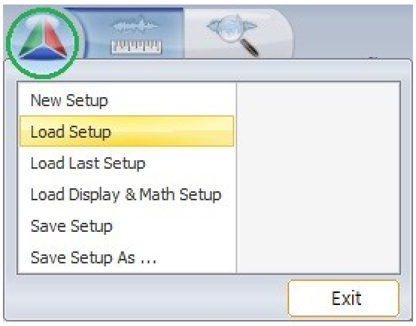
File menu → **New Setup**

and Channel setup screen is *displayed*.

You can define all *new* **Setup** data (**Channel setup** and **Application modules setup data**).

Don't forget to *save* Setup data after making a setup.

Load Setup data



We can directly load an *existing Setup* by selecting:

File menu → **Load Setup**

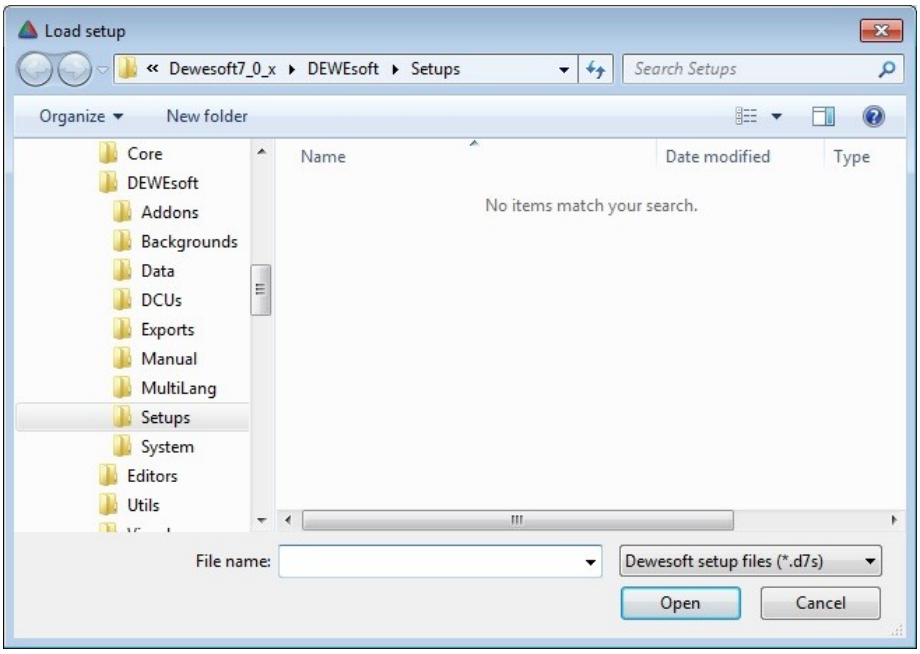
and *choose* from the file list,

or by click on:

File menu → *name of recent project* on lower part of this menu

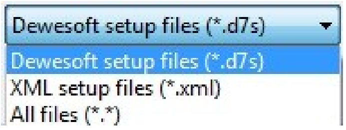
Setup loads *automatically when selecting most recent data.*

When we select the first option, the standard *Load setup window* is displayed to load *existing* **Setup** from `*.d7s` (*DEWESoft setup data*) *file* or `*.xml` (*DEWESoft **xml** setup*) *file*:



NOTE: *In order to see available XML setups within the file dialog, it is necessary to switch the file type to XML setup files (*.xml).*

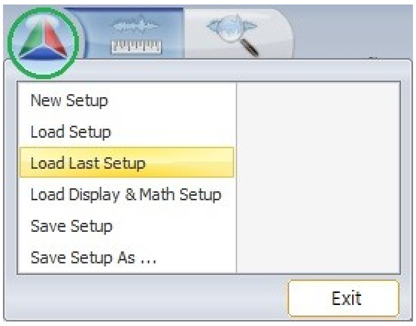
The change can be made in the drop down box where the file type is set. There are two choices - to load DEWESoft (.**d7s**) or to load **XML** (.**xml**) setup file.



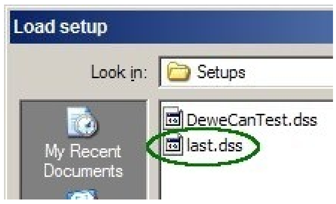
Load Last Setup data

If you have closed DEWESoft without storing the *setup* data, you can recall it after the *next startup*. To do that select:

File menu → **Load Last Setup**

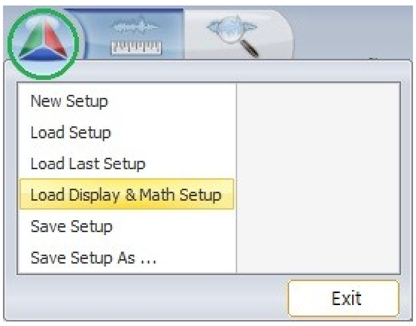


and look for a file called **last.d7s**:



WARNING: *Be aware that this file will be overwritten immediately when you change anything or load another setup.*

Load display Setup data



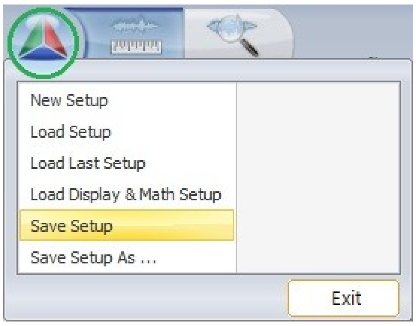
Load display setup options *keeps* the *channel settings*, but overwrites the instrument appearance with the appearance *found in the setup* file:

File menu → **Load Display Setup**

and the standard *Load setup window* (see above) is displayed to load *existing Display* **Setup** from *.**d7s** (*DEWESoft* ***setup data****) file*

WARNING: *Be aware that this file will be overwritten immediately when you change anything or load another setup.*

Save Setup data



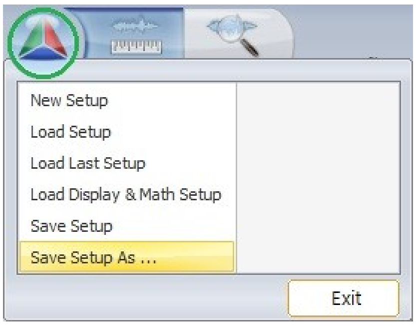
After setting all *channels* and *Application* modules data is reasonable to save this *new* or *changed* **Setup** by selecting:

File menu → **Save Setup**

By *new* project Standard *Save setup window* is displayed - *see 'Save Setup as...'*. *Changed* Setup is automatically stored under *existing* name.

WARNING: *existing project is saved under same project name and overwrite previous data.*

Save Setup as ...



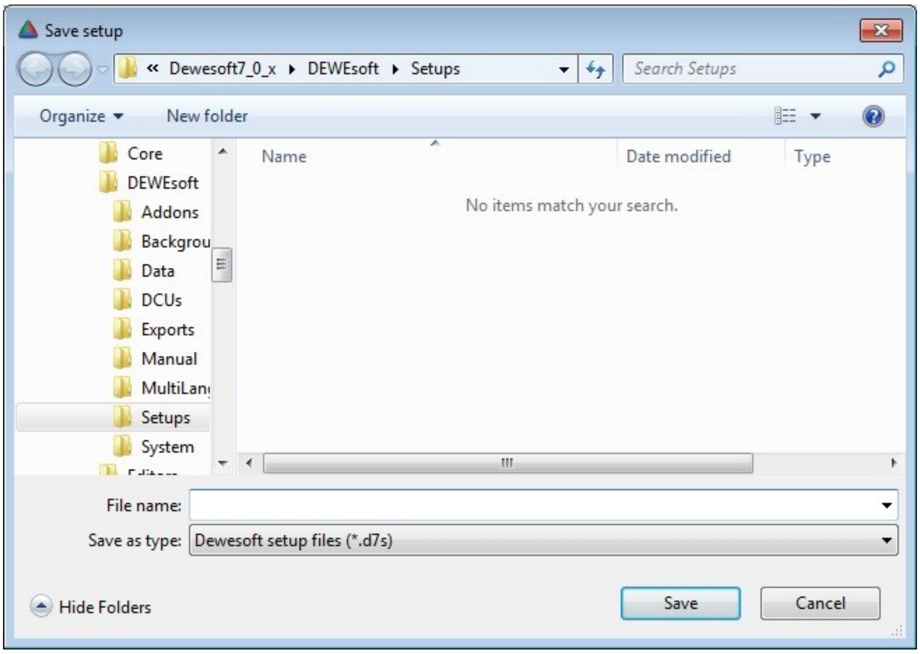
We can save *existing* (modified) **Setup** data under the ***new name*** by selecting:

File menu → **Save project as ...**

if you want to give the current setup a new name.

NOTE: *It is not possible to save a* text *setup. It has to be generated by separate custom software or manually by means of a spreadsheet editor or using a simple text editor. Be aware that in simple text editors, items belonging to one column may appear at different tab-stop positions.*

Standard **Save setup** *window* is displayed for save *new* or *modified* **Setup** to *.d7s (*DEWESoft* ***setup data***) *file*:



Is very reasonable to give easy to remember ***name*** of Setup, as it can serve as a nice starting point for any number of setups.

Setups are *automatically stored* in the \Setups subdirectory, and given an extension of *.d7s - *DEWESoft* *Setup*.

NOTE:	If you want to make this setup to your ***default startup*** ***setup***, define the file in the: System menu → **General settings** → **Starting setup** menu also *see* → System Settings → **General settings - Starting setup**

With checking **Save multifile opt. in setup** field on DATA FILE OPTIONS section of *Setup* window all multifile settings are *stored* in **setup** file.

see also → User Guide → **Basic Recording Setup**

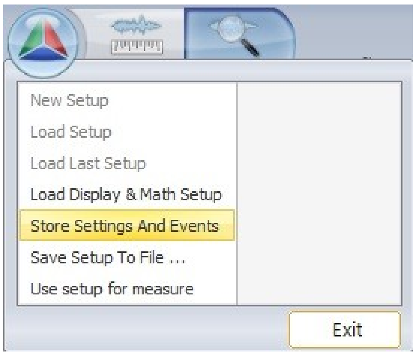
5.2 Settings and Events

Display appearance settings and **Events** (entered information about *measurement*) are stored in DEWESoft *.d7d (*DEWESoft* ***data****) file*.

Display appearance settings can be also stored in *.d7s (*DEWESoft setup* data) *file* (in Measure mode).

for detailed information about ***New / Load*** / ***Save*** the *.d7s (*DEWESoft setup data)* file *see* → **Setup Data**

Store Display appearance settings and Events



All defined events can be *stored* into the already *open* *.d7d (*DEWESoft data*) *file* by selecting:

Data menu → **Store settings and events**

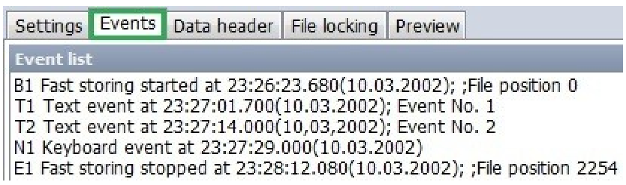
NOTE: *This function stores also the current display settings into the file!*

If you have changed anything in your ***display appearance***, it will now be *stored also*.

Load Display appearance settings and Events

To *load Display appearance settings* and *Events* click *again* the **Analyse** button.

When you select the **Events** on Analyse *initial* screen, you see the entries for all *old* and *new* ***events***:

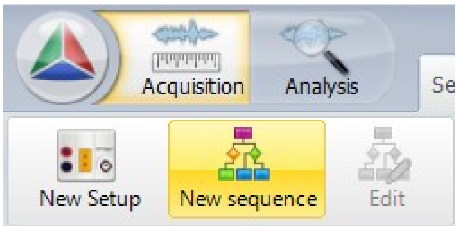


6 Sequencer

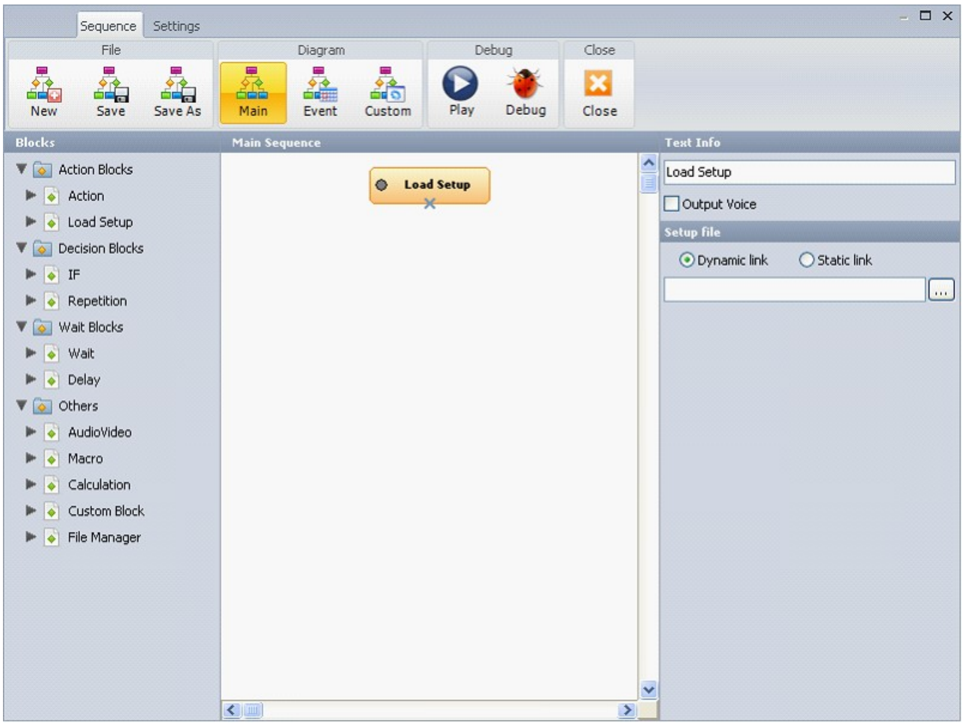
Sequencer in five minutes

Before going to details about **Sequence building blocks**, let's first try to make a first simple sequence to show how easy it is. A typical application is to *load* the setup, *perform* some measurement, review the data and then ask the user *to control* the **sequence**.

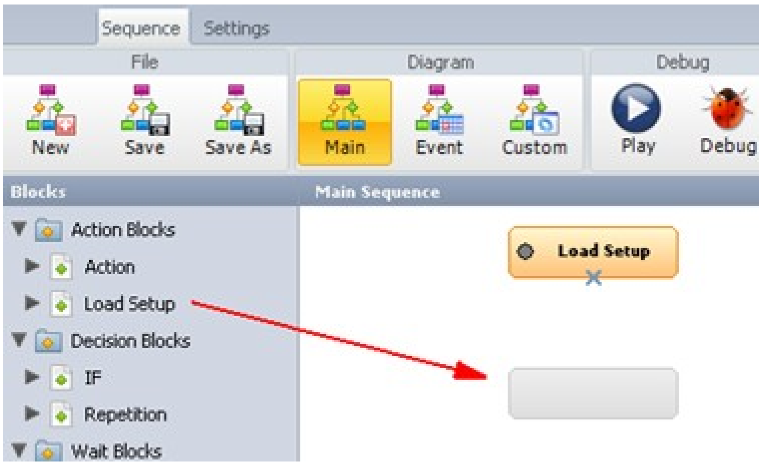
New sequence can be started from *Acquisition* – *Setup* files tool bar by pressing **New sequence** button.



The main screen of the *Sequence editor* has a main tool bar on the top, list of available blocks on the left, Main Sequence sheet in the middle and property tab on the right side.



The sequence can be started by clicking on the block name and while not releasing the mouse button, *dragging* it into the sequencer sheet area.

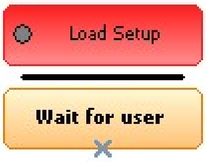


Adding modules

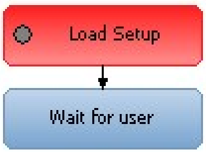
Let's *add* few modules. First I would like to **Load setup**. Then I would like the user to confirm that the measurement can be started. This can be done with **Wait** block. Then I would like to perform a DEWESoft action, so I choose the **Action** block. This block contains many actions which can be performed on DEWESoft.

Connecting modules

Now let's *connect* those together. This can be done either by *dragging* the block *over* the previous one in the sequence. Let's grab Wait for user block and pull it over Load setup block. There will be black line shown in the bottom of the Load setup block indicating that those two will be connected.

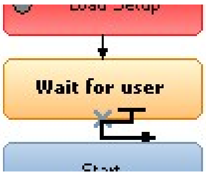


Release the mouse button to connect the two blocks. The correct connection will be shown as an arrow pointing from the first to the second block.

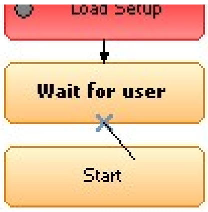


As an alternative you can also *select* the block *from* which you would like to connect. The connection arrows will be shown.

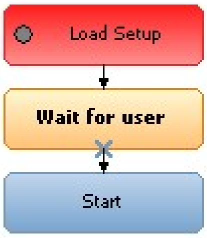
When we move with the mouse over the connection point, the mouse cursor will change as indicated on the picture below.



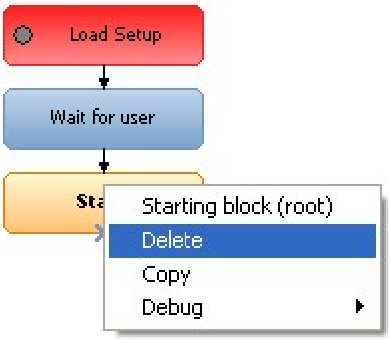
When we now *press* and *hold* the mouse button, the connection line will start to be drawn. If we *move* over the module, the module will turn to orange color.



If we release the mouse at this moment, the *connection arrow* will be drawn.



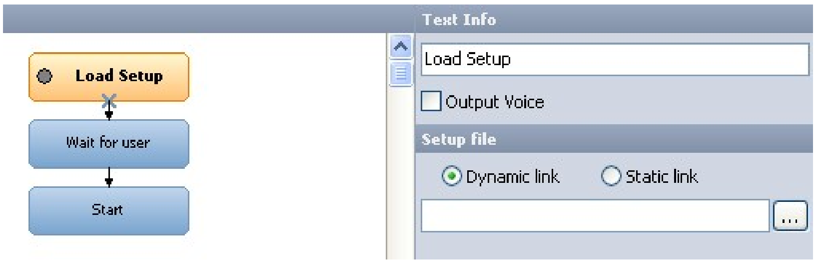
If we made a mistake either by adding the module or adding a connection line, *any* element can be **deleted** by right clicking on the element and choosing **Delete** button.



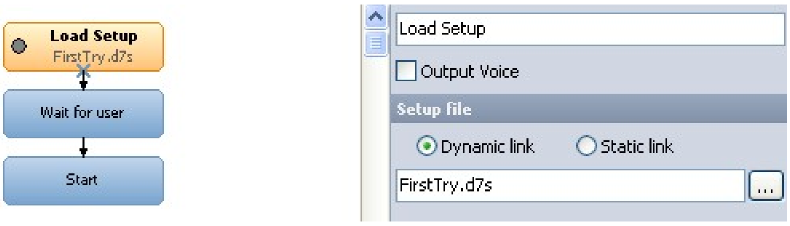
Setting properties

Ok, now that we have connected the modules together, let's *set* the **properties** of the building blocks. First we need to choose the block we want to **setup** and then we need to define its properties on the right side of the sequencer screen. In the upper section Text Info we define the name of the block which will appear while running the sequence in DEWESoft.

Second is the **Output Voice** option. This option will speak out the Text Info through Windows Text-To-Speech option. In Windows OS you can define the human voices which allow disabled people to operate the system. In our case there are several applications where the user can't really look at the screen while performing the test and for that purpose this option can help a lot.

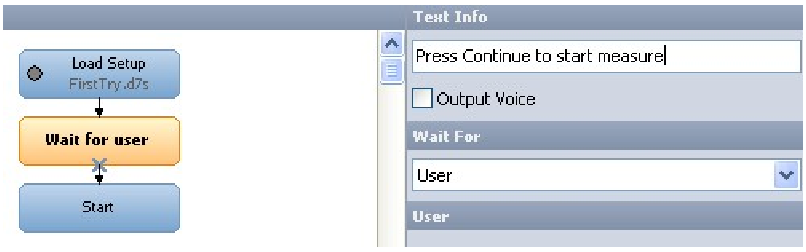


Second section depends on the block. For **Load Setup** block we can define the setup file which will be loaded. In my case I have prepared up front a simple setup called `FirstTry`. By clicking on the three dots push button we can choose the setup to load.

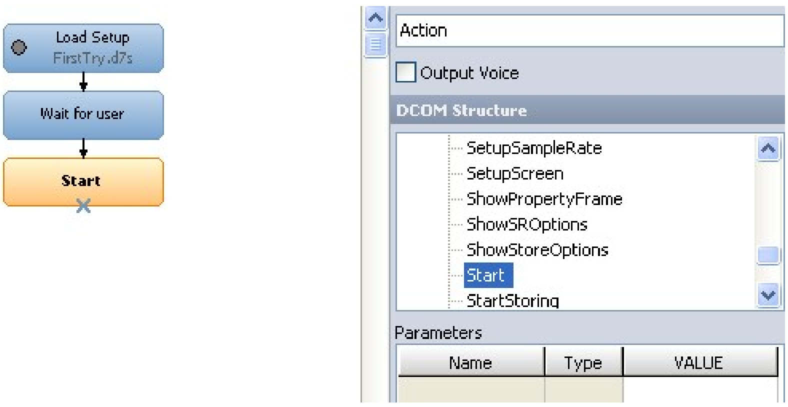


The **name** will be *noted* on the block.

Then we select the **Wait for user** block. We can wait for user interaction, trigger or certain value of the selected channel. At this point, the only thing needed is to inform the user what to do. That can be entered in the Text info field.



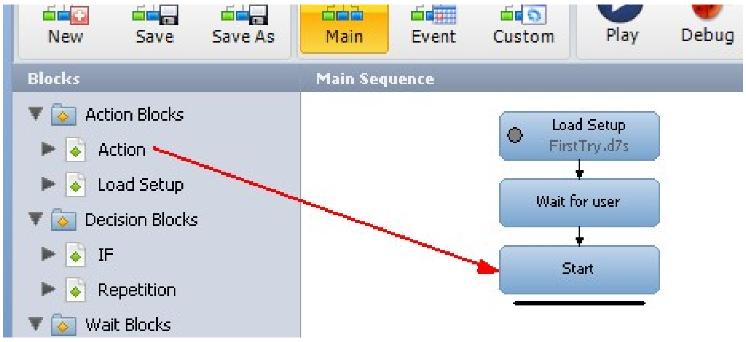
The action button needs a selection of DEWESoft event. There are many things which can be done with this block, but let's start with simple one, with **Start** action.



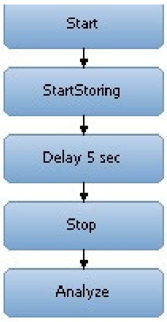
This action will *start* the **measurement**.

Finishing the first example

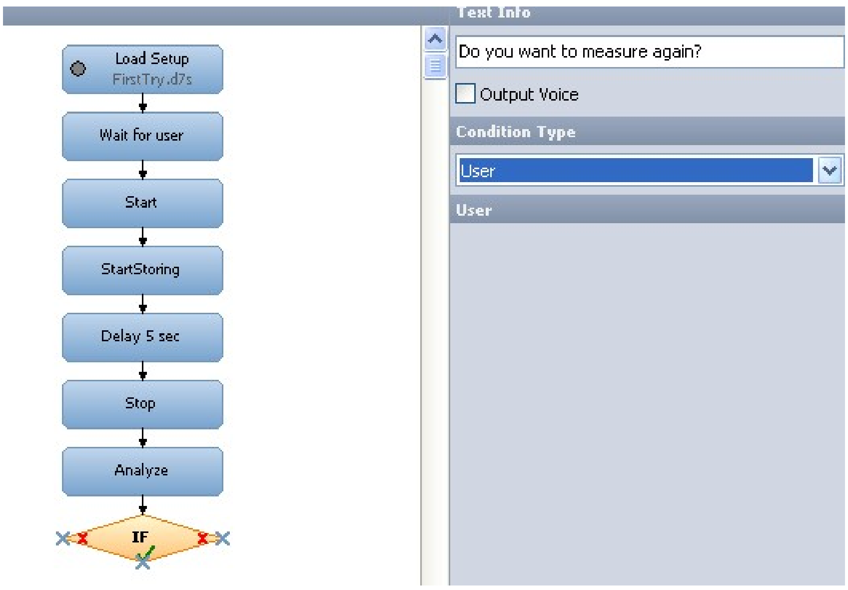
Now let's *add* some more blocks. Let's add another action. This time let's not drag it into the empty space, but over the Start block. The black line will appear on the bottom of the Start block and when releasing the mouse button, a *new* action module will be *added* and automatically connected to the Start block.



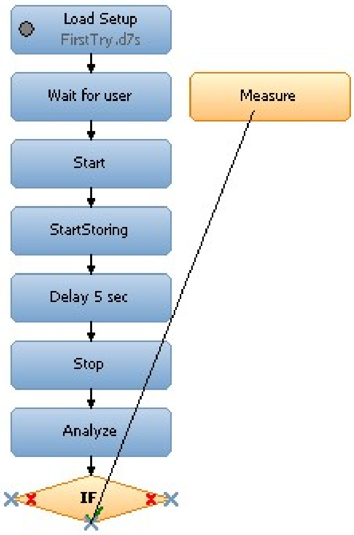
Let's choose Start storing action. Next we can continue by adding a Delay block from the left side. This block will wait with the execution of the sequence for the specific time period. Let's add few more Actions: Stop and Analyze by adding Action block and selecting appropriate action from the list.

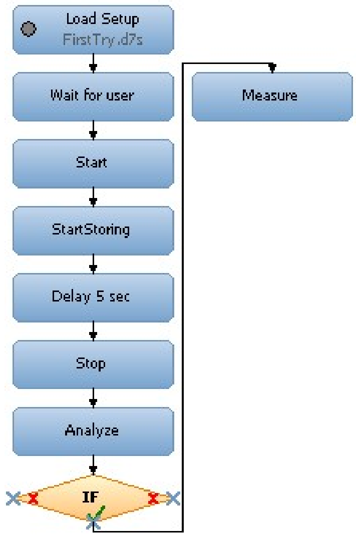


Now let's do something more interesting - let's *make* a **decision**. The decision can be made either by the user or from the formula. In this example I would like to ask the user *if another measurement should be performed*. So I *enter* this text in the Text info field.

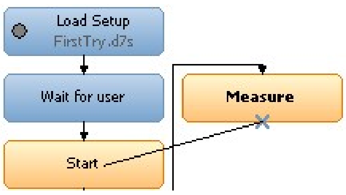


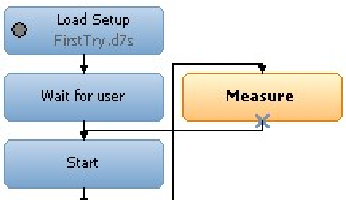
Now let's *connect the* blocks. If the answer is No, then the sequence can be *stopped* and therefore no connection line is needed. If the answer is Yes, I have to draw a line to some point above to *repeat* the execution. But we should remember that we are in Analyze mode. Therefore I add another Action called Measure to *go back* to measure mode. Then I can connect the If by clicking on the green tick and *drag it* to the Measure action. The correct connection will be shown when releasing the mouse button.





Then I need to *come back* to the *previous* flow. The right place would be the Start block. Therefore I need to connect the Measure to the Start block.

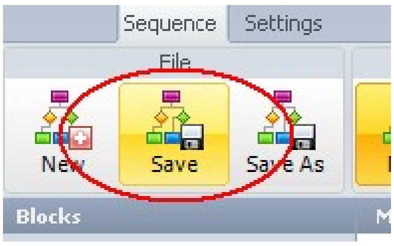




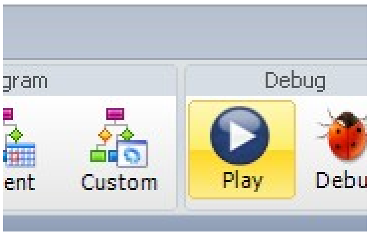
Don't forget, any mistake can be corrected by *right* clicking on the connection line and choosing Delete or by *left* clicking on the connection line and pressing Delete button on the keyboard.

Running the sequence

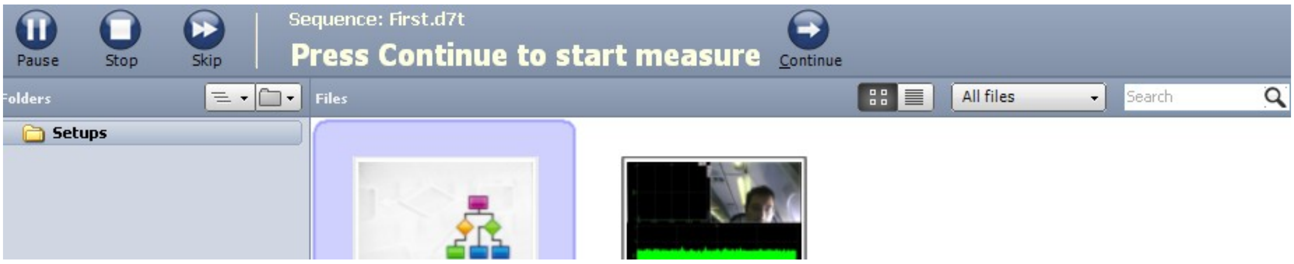
Ok, enough of "programming", let's see what we have done. Let's **save** the sequence by pressing **Save**.



Now let's try to run it by pressing the **Play** button. In real life the sequence can be run *just by loading* it from the startup screen, but for testing purposes Play and Debug buttons are close at hand.



The sequence will load the setup and show the top bar. There are several actions which the user can perform: **Pause** the sequence, Stop the sequence or Skip one block of the sequence. These buttons are available always. Based on the chosen block the *additional* buttons will be shown. At the wait block we could see the text info we have entered and the button **Continue**. This allows the user to enter some parameters, prepare the measurements and so on.



In this case Continue button will **start** the measurement. The delay will store the data for five seconds and the user will *need to wait* for that time period:

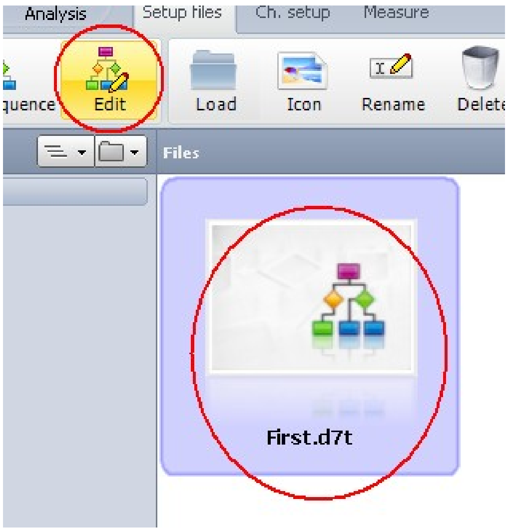
After five seconds the **stored** file will be *reloaded* and the question will come to the user if *another* measurement should be taken. If we say Yes, another data file will be *stored*. If we say No, the execution will **stop** and the software will go to *startup screen* or to the *sequence editor*.

In this simple example we have seen that the sequence is easy to operate if we know the basics of operation. With sophisticated measurement equipment I can also see that I need to shave (and perhaps get some more sleep).

Refining the example

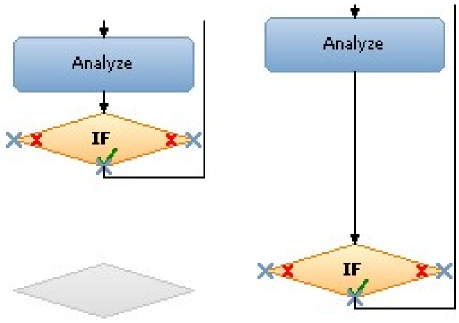
Now let's refine the example a little bit. Let's try to *eliminate* the user's ability to interfere with the flow execution and on the other hand let's *give* him a chance to **enter** *important information* about the measurement *conditions*.

If we have left the sequence editor, the **editor** can be *reopened* from the *startup screen* by *choosing* the right sequence and pressing **Edit** sequence button.

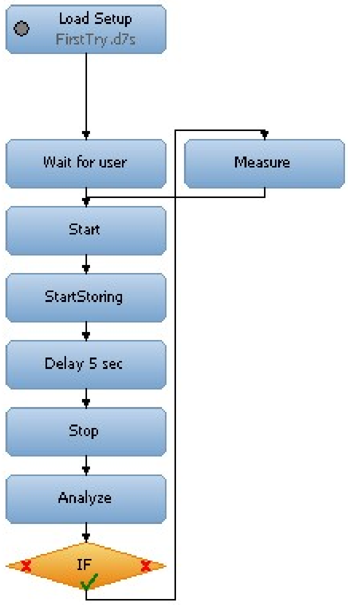


Now let's *rearrange* blocks a bit *to add* few more actions. We can drag and drop the block in the sequence window to

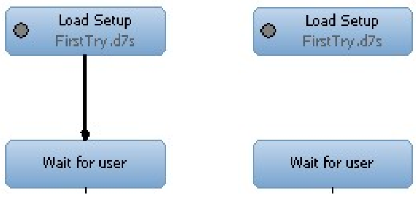
move them. *Move* with the mouse button over the block, but <u>take care</u> that the cursor *doesn't change* to drag the *connection lines*. Then press and hold the left mouse button to *move* the block. In the example below I have moved the IF block to *gain more space*.



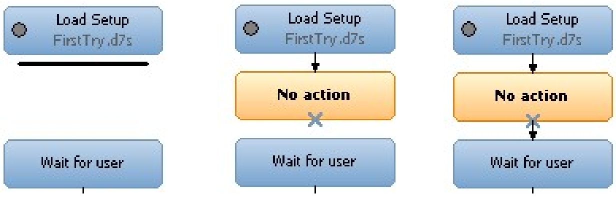
In the same way we should move other blocks to gain space for one more block.



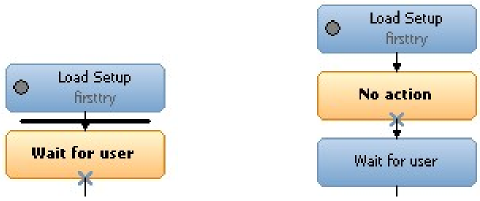
Click on the connection line between Load setup and Wait for user and press Delete button on the keyboard. Connection line is gone.



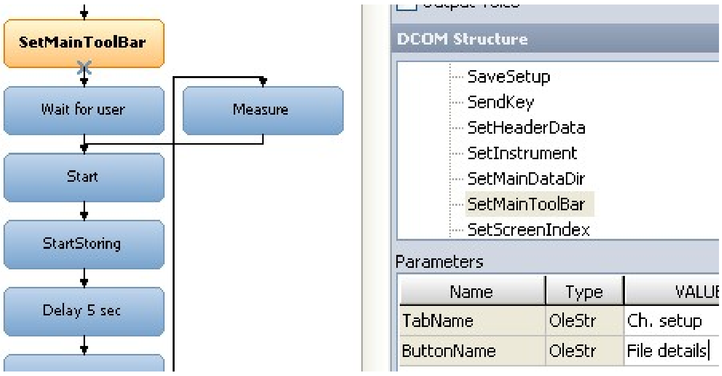
Now let's *add* another Action. Add it over the Load Setup to connect to that and then *manually connect* the Action with Wait for user block.



I wanted to show you the longer way that you get skilled with moving around the modules. If a *single* module needs to be **added**, there is a much simpler way to add a block. Simply *place* the *new* block *on the upper one* under which the new module should be **inserted** and all other modules will be pushed away. New module will be *connected* with the upper and lower one.



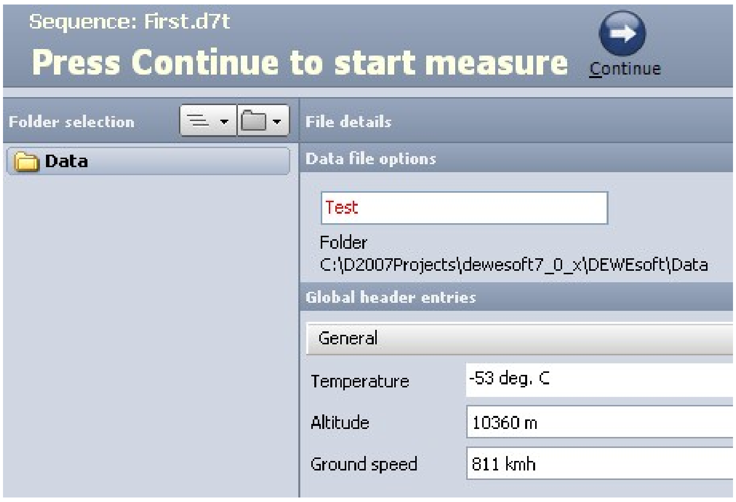
Now let's choose *SetMainToolBar* action. This is a great action which allows us *to navigate* through DEWESoft *menus* as we would press the buttons manually.



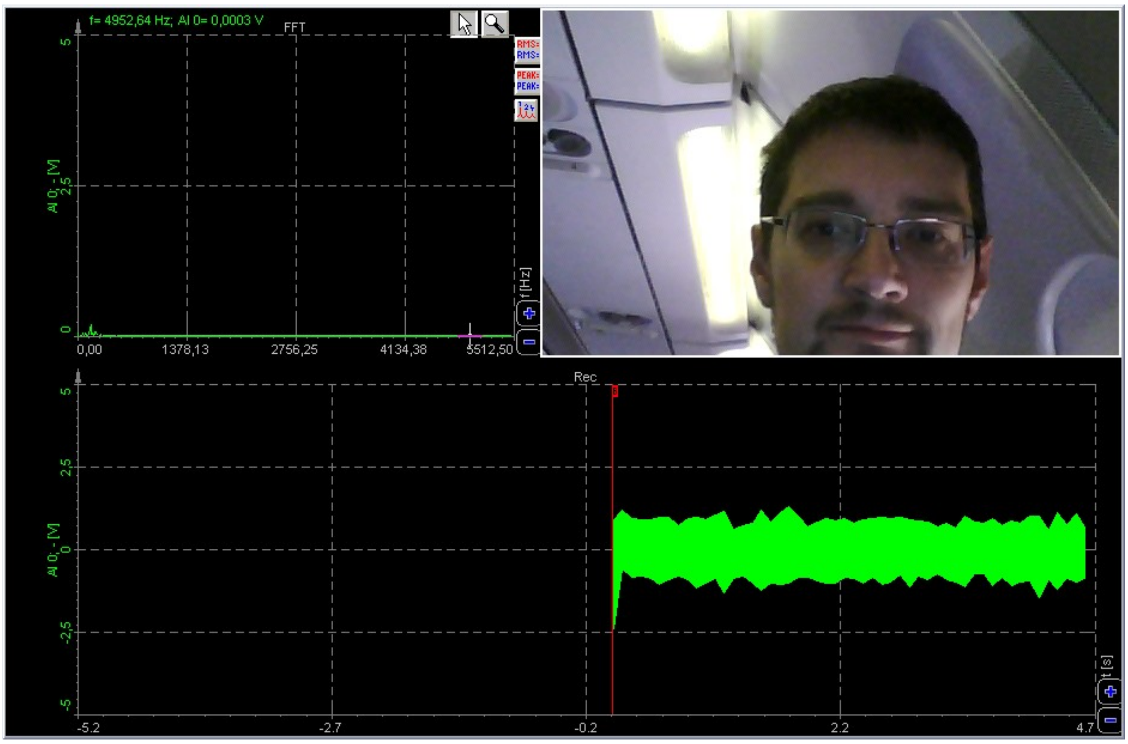
In the property panels we need to enter the Tab and the Button name. The *TabName* is the name of the *main tabs* of DEWESoft. In our case I want the software to go to Ch. Setup – File details screen that the user can *enter file name* and the *file header*.

Additionally I want to *prevent* the user *to change* the screens and to stop, block or accelerate the execution of the sequence. This can be done in Settings tab of the sequencer. Let's *remove access* to the sequence control tool bar (the buttons for stop/skip/pause) and the display property panels.

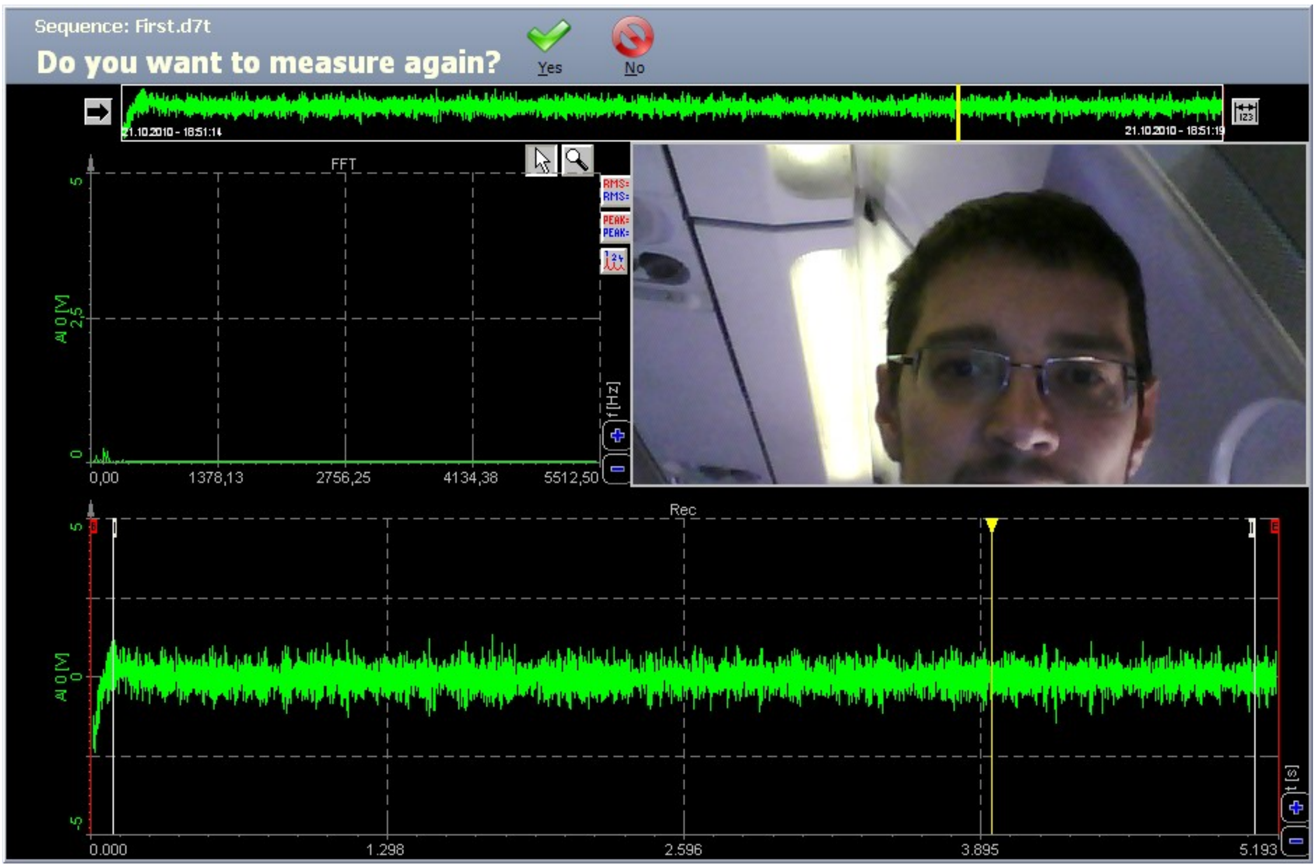
Let's *run* the sequence again. This time the sequence will *stop* in File details screen allowing the user *to enter* the file header.



Please note that we *don't have* a chance now *to control* the execution of the sequence. The only available button is the **Continue** button. In measurement screen all the properties are hidden.



In the analyze mode the only available buttons are the selection to *make another* measurement (**Yes** button) or *exit* (**No** button) the sequence.



With few simple settings we have made very *simple user interface* for the end user and *hide* all in depth abilities while and therefore *prevent* mistakes during measurement process.

6.1 Sequence building blocks

In this section we will *look in detail* to each block which can be used. Since there are many possibilities in sequencer, there are also many options to choose from.

Each option has a Text info which will show at the top. We have seen the usage of the text info already in the short example, but it has also one interesting option. It can *parse* variables or channels and show them in the upper bar. As we can enter a variable in the formula we can also enter it in the text info.

During measurement it will show up as a real value and will update with display refresh rate.

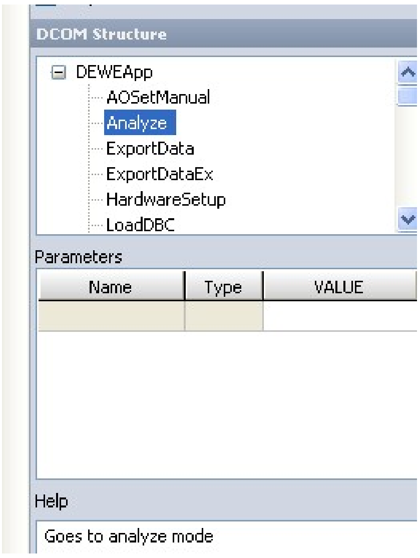
There are several blocks to help you build your sequence:

Action block	**Audio/video block**
IF block	**Macro block**
Repetition block	**Calculation block**
Wait block	**Custom block**
Delay block	**File manager**

6.1.1 Action block

Actually here are all functions available also from DEWESoft programming DCOM interface, so the sequence is actually an easy visual way to **program** the execution work flow like it could be done in any external programming language. Commands are the same as in DEWESoft DCOM... We can also look at the sequencer as the easier to use *replacement* for the classic programming languages.

Action can be chosen from the list shown on the right side under DCOM structure caption.



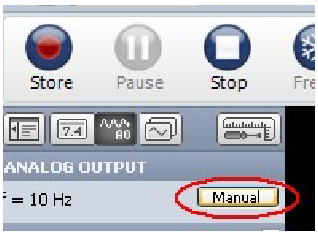
Some of the properties have the *parameters* which can be edited in Parameters list. Help will provide short form information what the action does.

There are many possible actions shown here in alphabetical order.

AOSetManual action

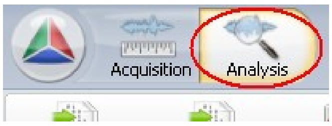
If the function generator is set to Manual output, the user can *choose* whenever the output will *start*.

The action is the same as if the user would press the **Manual** button to start the function generator (shown below).



Analyze action

This action puts DEWESoft in Analysis mode like pressing **Analysis** button on the screen. Please take a look also to the **Measure** action.



ExportData action

This action **exports** the data. It is replaced by ExportDataEx which has more parameters, but this function can still be used if these additional parameters are not needed.

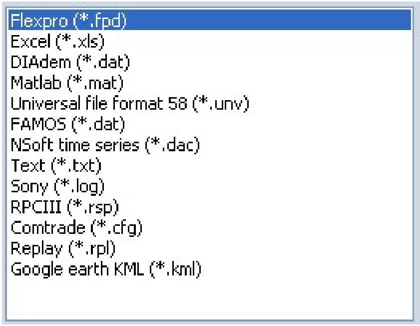
ExportDataEx action

ExportDataEx will export the opened file with extended set of parameters. Please note that the file must be open in Analysis mode to call this action.

There are several parameters which must be set.

Export type

Export type defines the **type** of the export to be used. The list of exports is enumerated in the same way as the drop down list in the export is shown.



So we need to enter:

0 ... *Flexpro* (*. **fpd**)
1 ... *Excel* (*. **xls**)
2 ... *DIAdem* (*. **dat**)
3 ... *Matlab* (*. **mat**)
4 ... *Universal File Format 58* (*. **unv**)
5 ... *FAMOS* (*. **dat**)
6 ... *NSoft time series* (*. **dac**)
7 ... *Text* (*. **txt**)
8 ... *Sony* (*. **log**)
9 ... *RPCIII* (*. **rsp**)
10... *Comtrade* (*. **cfg**)

Custom exports must be *enumerated* with negative values starting with -2. The custom exports depends on the installed exported (.exp) files. In our case from the screenshot above it would be:

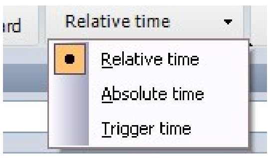
-2 ... Replay
-3 ... Google earth KML

Time Axis

Time axis defines the how the time will be *exported*:

0 ... relative
1 ... absolute
2 .. .from trigger

The property corresponds with this selection:

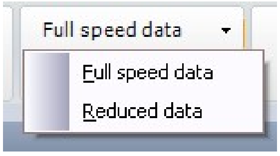


ExportData Type

Defines which data is to be exported:

0 ... full speed
1 ... reduced (with speed defined in the reduced rate)

The property corresponds with this selection:



ExportOptions

Defines which data will be exported *at reduced rate*. The value is a set from:

1 … min
2 … max
4 … average
8 … rms

So if we want to export min and max, we need to enter: 1+2=3. If we want to export min and RMS, we need to enter 1+8=9.

The property is important only if reduced data is exported and corresponds to this selection:

FileName

Defines the file name to be exported.

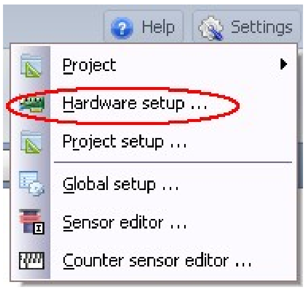
If the file *extension* is not defined, the *default file* extension will be taken.

If the file *path* is not defined (just the file name is entered), then the *default export folder* will be taken.

If not even a *file name* is defined (field is blank), then the *loaded DEWESoft file* name will be taken.

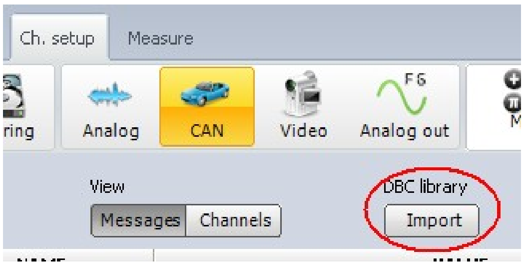
HardwareSection action

This action will open the Hardware screen of DEWESoft.



LoadDBC action

This action will load the DBC file for CAN ports. You have to define the port number and File name of DBC library to load. It is the same action as this one found in CAN setup:



LoadFile action

This action will load DEWESoft data file. If the file name is wrong or if it is not defined, it will show the *dialog to choose* the file *manually*. If the path is not defined, it will try to load the data from the *main data folder* of the currently selected project.

LoadProject action

This action will open the project file. This brings the sequence above setup as well as above project files. It allows to load in one single sequence different projects and different setups.

LoadSetup action

This action loads the setup. A file name must be defined. The action is the same as the Load Setup main block.

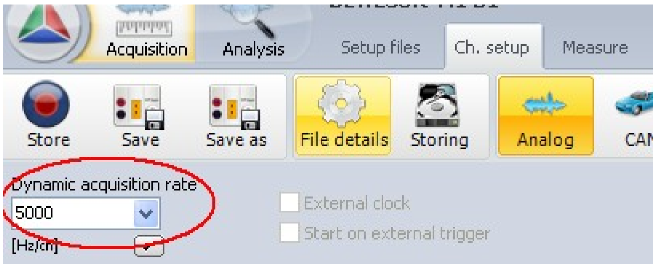
Measure action

This action will *put* DEWESoft in Measure (Acquisition) mode. It is the opposite action from Analysis and has the same effect as clicking on the **Acquisition** button.

MeasureSampleRateEx action

This action will change the measure sample rate. Property MeasureSamplerate must be set to the new wanted sample rate.

IMPORTANT: DEWESoft must be running in *Acquistion – Ch. setup* screen that this command is taken.



NewSetup action

This action will open a new setup. It is equal to the DEWESoft button – **New setup** menu item.

PauseStoring action

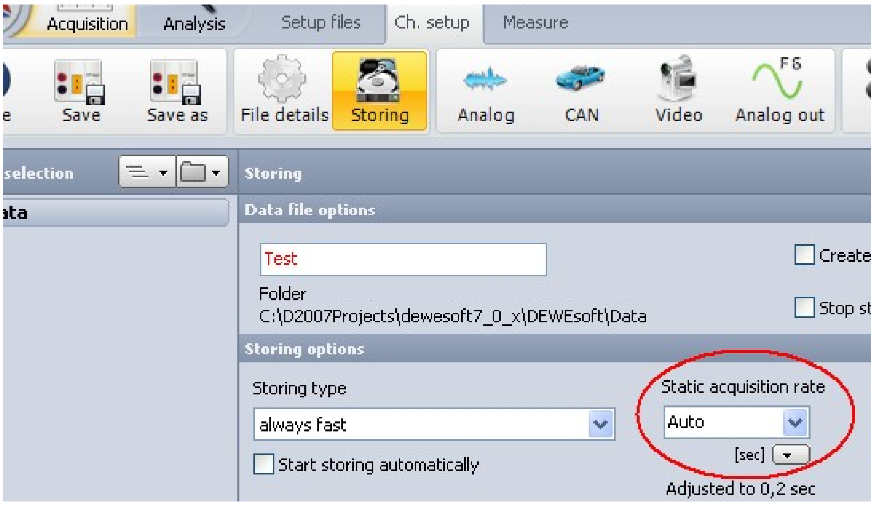
This action will pause storing. It is equivalent to the **Pause** button and works in conjunction with **Resume**. DEWESoft must be in Measure mode and should be storing the data that this command is valid.

PrintScreen action

This action will make a screen dump to the printer. It is the same as the Analysis Print command.

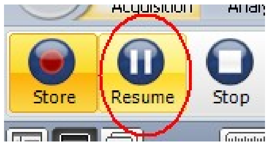
ReducedRate action

This action will set the reduced rate of the data. Please note that you need to be in *Measure* – *Setup* screen that this command is valid. It is the same as the Static acquisition rate entry field.



ResumeStoring action

This action will *resume storing* if paused. It is equivalent to the **Resume** button and works in conjunction with **Pause**. DEWESoft must be in Measure mode, should be *storing* the data and should be *paused* that this command is valid.



SaveSetup action

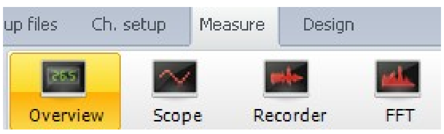
This action will *save* the setup under the defined file name. If the file name is empty, it will store the *currently loaded* setup.

SendKey action

This action will *send* the key stroke to DEWESoft. The parameter *Key* is the key code according to the character map.

SetInstrument action

This action will *change* the shown display. The Id parameter defines the main index of the screen. This command is valid only in the measure mode. In this case 0 will set *Overview*, 1 will set the *Scope*, 2 will set the *Recorder* and 3 will set the *FFT* regardless of the display name.



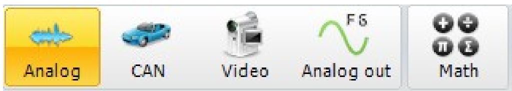
SetMainDataDir action

This action sets the main folder of the data.

SetMainToolbar action

This action is very *important*. It sets DEWESoft to the certain mode. The *TabName* property defines the name of the *main tool bar*. Here a Name of the main tool bar must be entered (like *Ch. Setup*, *Measure*, *Print* and so on). Please note that some of the tabs are not always available and depends if DEWESoft is in Measure or Analyze mode.

The next selection – Button name defines which *main menu button* will be selected. Here it is again important to take care which buttons are available in which menu.



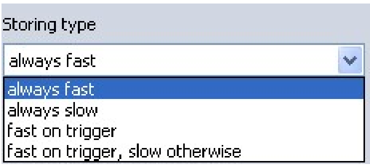
SetScreenIndex action

This action *changes* the sub displays. If the main display has several sub-screens, this action can change between them.

SetStoreMode action

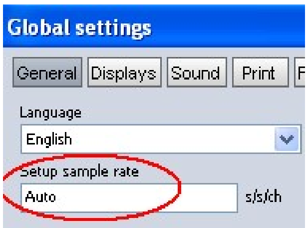
This action can *set* the store mode. Possible values are:

0 … always fast
1 … always slow
2 … fast on trigger
3 … fast on trigger, slow otherwise



SetupSampleRate action

This action *changes* the setup sample rate. It is similar to the *Global settings* – Setup sample rate entry. Please note that DEWESoft must NOT be in the *acquisition setup mode* when this parameter is set.

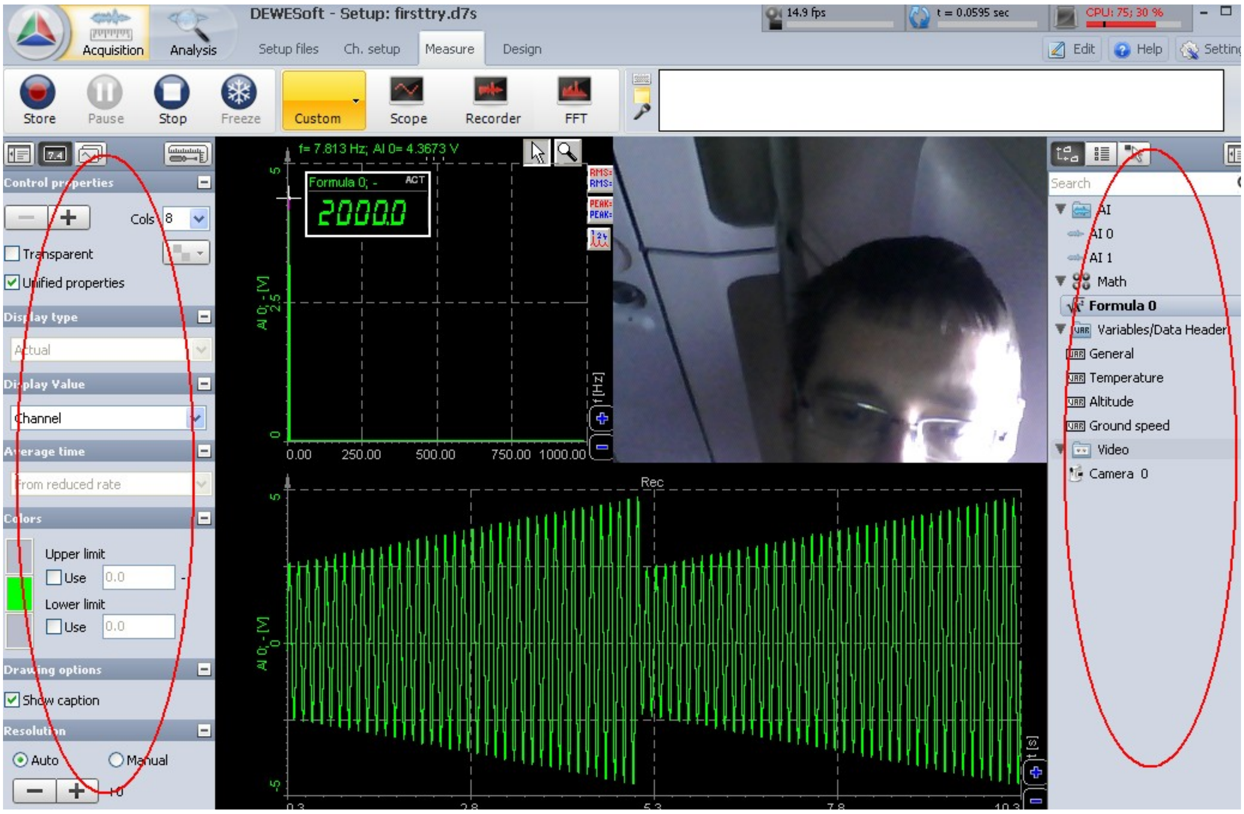


SetupScreen action

This action *changes* DEWESoft to the setup screen mode.

ShowPropertyFrame action

This action *shows* or *hides* the property frame and channel list while *measuring*.

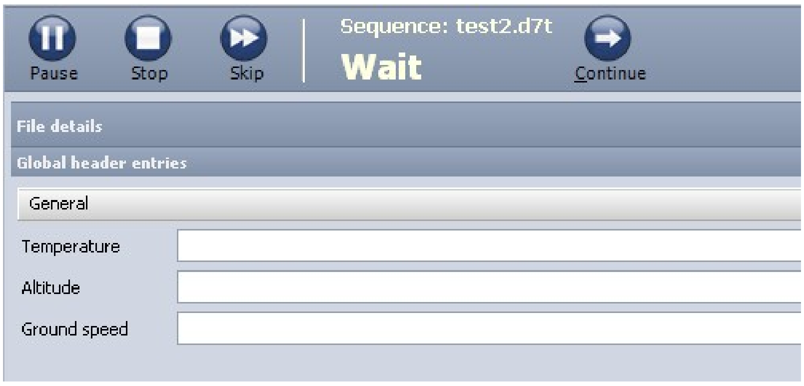


ShowSROptions action

This action will *show* or *hide* the sample rate panel in Ch. Setup – Storing and Ch. Setup – Analog tabs.

ShowStoreOptions action

This action will *show* or *hide* the storing options in Ch. Setup – Storing and Ch. Setup – File details. It is important if we want the user to be able to change the *header* entries, but *not* the file name.



Start action

This action will *start* the measurement.

StartStoring action

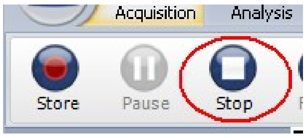
This action will *start* storing the data. DEWESoft must be in the Measure mode that this command is working. It is the same as the **Store** button.

StayOnTop action

If the property *StayOnTop* is true, then DEWESoft will be always on top even if another application has the focus.

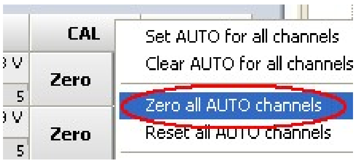
Stop action

This action will *stop* the measurement.



ZeroAllAutoChannels action

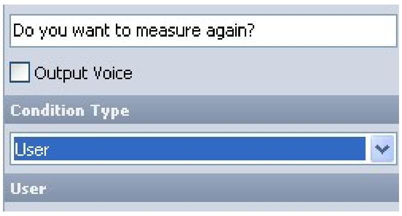
This action will *perform* channels zero for all Auto channels. In the Ch. Setup it is the same action as **Zero** all AUTO Channels.



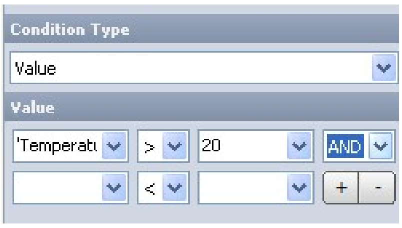
In the measure mode it is the *same* action as the Zero, but it is important that DEWESoft is *not storing the data* when this action is performed.

6.1.2 IF block

This block is used for **making a decision**. Decision is based on user choice where the question is asked in the Text info and the user answers with Yes or No.

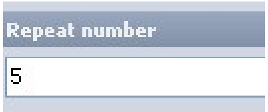


Another option is to make a decision based on the value. We can *add* any number of conditions by pressing plus and minus. In the first field we can add any channel and then choose either >, <, = and != (not equal). In the second field we can also add a *variable* OR we can enter a *fixed value*.



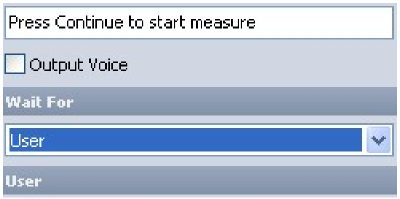
6.1.3 Repetition block

This block is used to *repeat* a certain work flow fixed *number of times*. The only property is the number of repetitions. We need to connect to Yes the blocks which needs *to be repeated* and to No the blocks which are *following* the Repeat block.



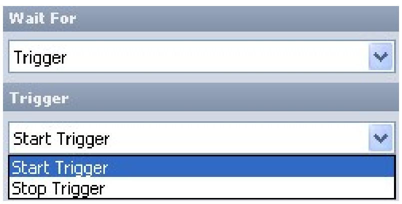
6.1.4 Wait block

This block is used to **wait** for a certain event. It can wait for *user interaction*. It will output a Text info and show **Continue** button. The execution of the work flow will continue when the user will press it.



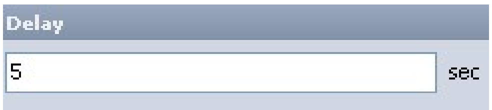
Another option is to **wait** for the value. We can *add* any number of conditions by pressing plus and minus. In the first field we can add any channel and then choose either >, <, = and *!=* (not equal). In the second field we can also add a *variable* OR we can enter a *fixed value.*

Third option is to **wait** for the trigger. We can either wait for the *start* or *stop trigger*. DEWESoft must be in *store mode with triggering* that this option is valid.



6.1.5 Delay block

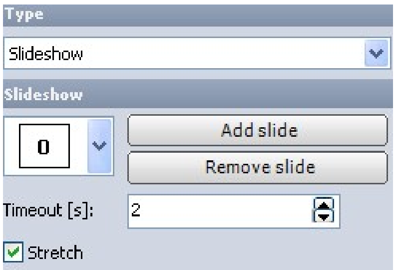
This action is used to *wait* for a certain amount of time defined in the Delay field. Time is defined in seconds.



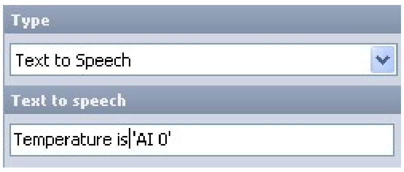
6.1.6 Audio/Video block

This block is used **play** audio, video files, slideshows or **outputs** text-to-speech. A short video or audio instruction might be valuable help for unskilled user. We can choose from **.avi**, **.mkv**, **.mov**, **.mp4**, **.mpg**, **.wmv**, **.mp3** or **.wav** files.

Another option is to **create** a slideshow to show. We can enter *any number* of pictures and define *delay* between them.



Third option is text-to-speech. This option gives us ability that the text is *output to the speaker* which is written in the entry field. Also here a value of the channel can be entered, so if we have AI 0 at 40, computer will say "Temperature is 40".



6.1.7 Macro block

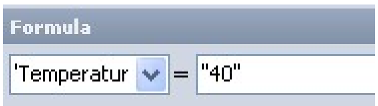
This block is used to **record** and **replay** Macro. With pressing the Play sequence and Record Macro DEWESoft will *execute* the sequence up to the point where Macro needs to be recorded and then we can *operate* the mouse, keyboard or even *record* the voice during our actions.

When *playing it back*, DEWESoft will *repeat* the same actions.

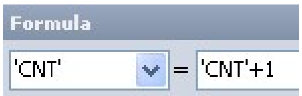
6.1.8 Calculation block

This block is used to **set variables** or **control channels**. We can set the data header values, internal variables used in math or control channels which operates *analog* or *digital output*.

First entry is the channel which we will set. In the example below we will define the data header *variable*. Since this variable is set to Text, we must enter the *value* with **double apostrophe**.



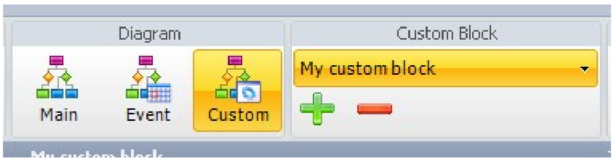
If the variable is integer or float, we can enter the *value* or enter any DEWESoft *channel* with full function of math formulas. We can for example define the counter. In the *Project setup* – Internal variable I defined the "***CNT***" variable which can be used like shown below. On each transition of this block the value will be increased by 1.



This block can also **set** the control channels. In the example below the frequency of the function generator is set to 50.

6.1.9 Custom block

This block is used as the **black box** which can hold *more modules* to reduce the complexity of the main sequence. Please note that on the *upper* bar you have a choice to see the **Main sequence**, **Event** *actions* or **Custom blocks**. If we go to the custom block, we can *add a new* one by pressing + button. After defining the blocks, we can enter this *single* or *multiple* times in the main sequence by *adding* a custom block.



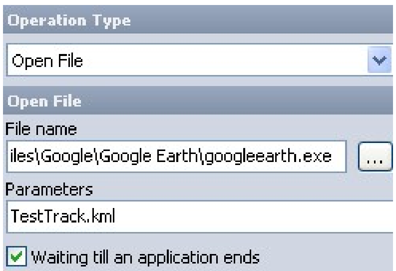
The only thing to define in the custom block is the name of the block.

6.1.10 File manager

This block is used to **perform actions** on the files. We can for example run external program with some parameters, delete, copy or rename the files.

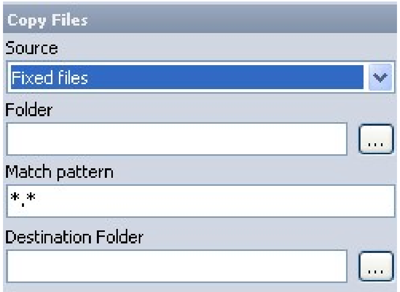
Open

Opens or **executes** the file defined in the file name. This is similar to Start – Run ... option in Windows. We can also define parameters for executables.



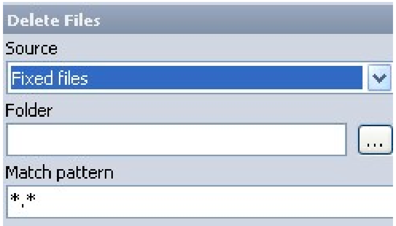
Copy

Second option is to Copy Files. We can either copy a *series* of files or choose to copy *last stored* file to a defined Destination folder. This is used for example to create backups on network drives.



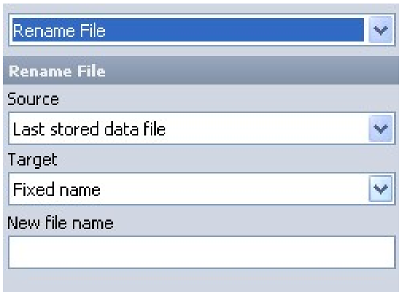
Delete

Another option is to Delete Files (defined or last stored file). We can for example ask the user if the measurement is ok and if it was not, delete the last stored file.



Rename File

At last, we can also Rename File (last stored) to a different name which can be either fixed or defined by the user.



Wow, that was quite a list and it required a trip over the Atlantic to describe all of them. And these are only single commands like a single instrument in the orchestra. To really understand the power of sequencer, we should look at some typical examples how to use those building blocks to create applications.

Part II

System settings

For effective DEWESoft use all needed ***hardware units*** and *their* ***drivers*** must be proper **installed** on respective computer; after that DEWESoft must be **set up** to *acquiring* and *processing* ***measured data*** with this hardware.

DEWESoft X1 can be used with recommended *system* hardware and software.

for information about ***recommended system hardware*** and ***software*** *see* → **Application requirements**

1. STEP

Hardware installation — all needed ***hardware units*** and *their* ***drivers*** must be proper *installed* on computer; for detailed information about ***hardware*** and ***drivers installation***

see → **Hardware driver installation**

2. STEP

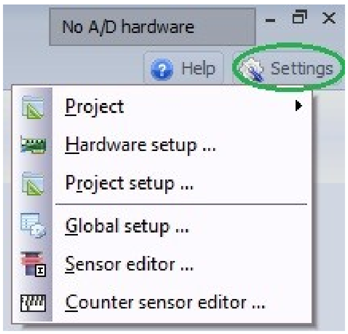
DEWESoft **System setup**

- enter licence key - **Licensing**
- appropriate DEWESoft Settings menu for:
  - **Project** settings
  - **Hardware setup** procedure: we can *set* and *activate* all applied measurements hardware for work with DEWESoft and his ***application modules***
  - setting for *Project folders, Starting setup, Security, Internal variables, Data header* and *Memory* in **Project setup** menu item
  - *general, displays*, *sound*, *print*, *folders and amplifier* options in **Global setup**
  - **Sensor editor**
  - **Counter sensor editor**

Before use DEWESoft software:

- for working in **REAL** mode, you have to install appropriate **hardware** and **Hardware driver**(s). After installing software you have to install **licence key** - **Licensing**
- first of all choose is appropriate **hardware** and **general setting**. In DEWESoft **Settings** menu you can choose:

- **Hardware setup**
- **Project**
- **Project setup**
  - **Project folders**
  - **Starting setup**
  - **Security**
  - **Internal variables**
  - **Data header**
  - **Memory**
- **Global setup**
  - **General**
  - **Displays**
  - **Sound**
  - **Print**
  - **Folders**
  - **Amplifier**
- **Sensor editor**
- **Counter sensor editor**



High-performance application requirements

For all high-performance applications like high-speed *video* or high-speed *data acquisition*, we recommend the following hardware and software:

- WINDOWS XP / Windows 7 [1)]
- Intel Core2Duo processor or higher
- 2GB RAM or higher
- Serial ATA or SCSI hard disks and/or RAID systems
- Approx. 200 MB free hard disk space for DEWESoft
  Additional 20 to 200 MB for drivers and post processing software packages
- A/D board for using **real** mode (requires licence key)

NOTE: *We strongly recommend to running DEWESoft-NET with Windows Administration rights - otherwise file access or driver problems might occur!*
With Windows Vista it is necessary to switch off User Account Controls (UAC) that DEWESoft works property.
[1)] DEWESoft is not running under WINDOWS 95/98/ME/NT4/2000 operating systems!

for information about ***NET module hardware*** and ***software requirements*** *see* → **NET module**

1 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 project, 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. New project will not be created from scratch, but it will take all the settings of *currently selected* project.

Rename project

Rename project menu item will open the dialog box to **rename** 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.

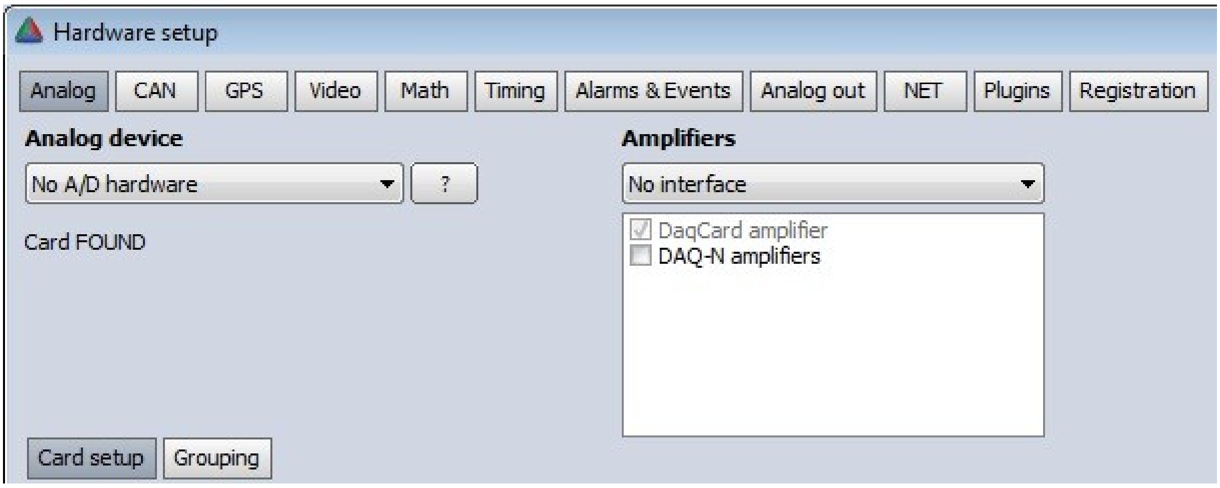
Import project from file

This menu item allows to **import** the project from *setup file* (`d7s`), *data file* (`d7d`) 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.

2 Hardware setup

When you first run DEWESoft, it does not know which *hardware* you intend to use it with, so we must determine this in appropriate tabs on *Hardware setup* screen.

The *Hardware setup* screen is same in all DEWESoft software. Please select System menu → **Hardware setup**:



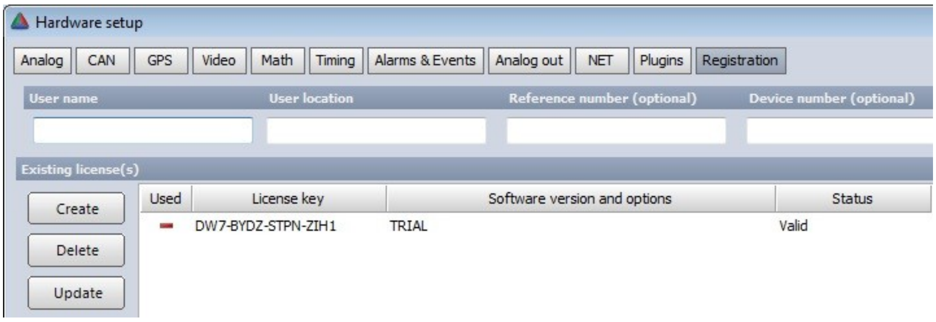
With **Hardware setup** procedure we can *set* and *activate* all applied and proper installed **measurements hardware** for work with DEWESoft:

Analog input	depending on hardware you can *run* DEWESoft software on a computer: • with an A/D card in **PROF (REAL)** mode • that does not have an **A/D** card in **DEMO** mode on this screen is also available *channels* **Grouping**
Analog output	to activate and set **analog out** *channels* for use with *Function Generator*
CAN input	to activate **CAN** board
GPS input	to activate and set **GPS** data acquisition
Video input	to activate and set **video** data acquisition
Math	to setting up **calculations** with signals, **filtering** them or **set digital states** according to your acquired data from *input channels*; to activate other DEWESoft **Application Module** like *Power Calculation*, *Torsional vibration calculation*, *Sound level meter*, *Human body vibration*, *Order tracking* and *FRF*
Timing	supports the **time synchronization**
Alarms	activate **alarm monitoring** to set *digital states* according to acquired data and for *digital output* to A/D board or PAD-DO7 modules
NET	to activate and set **network environment** where one or more measurement units to be under the **control** of

	other computers
Plugins	to create of user defined **interfaces**
Licensing	for easy **license** handling of software and options

2.1 Licensing

A **Licensing** (**Registration** button) offers the possibility for easy **license handling** of *software* and *options*. You have to enter your User name and User location first, and then click into the license key(s) field and enter your license key(s).



If the license key is correct, the red cross in Status will be replaced by a green *check mark* and the Registration status displays the version: LT, SE (Standard Edition), PROF (Professional Edition), DSA (Dynamic Signal Analyzer)....

The systems usually comes with valid license tied to the measurement hardware, but if you have purchased the software alone, you will get the license key along with the ordered software which needs to be registered.

The license key have such form: DW7-LJ2U-NVH4-9HY1.

Entering new license

Choosing **Create** button will allow us to enter the new license key. If the computer is connected to the Internet, then the registration process is very simple. Enter the key in the *Enter new license* edit box.

IMPORTANT:	*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.*

Register online will connect to the Internet and register the license. Once the license is registered, doing that again will simply return the same license file and the rules above will apply.

IMPORTANT: *In case of evaluation license, you can press* Register online *even if the computer is not connected to Internet.* DEWESoft *will recognize evaluation code and will register it automatically for the time period defined in the license.*

If the computer is *not* connected to Internet, you can perform **Register offline** registration by following these steps:

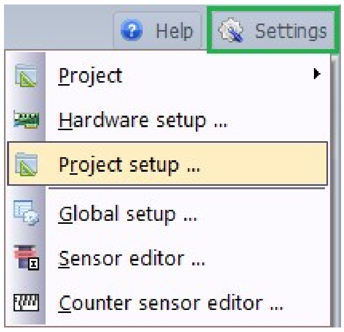
1. Fill out desired User name, User location, Reference number (optional) and Device number (optional) information
2. Enter your DEWESoft 7 License key and press **Register offline** button
3. Copy the created license (`.lic`) file from measurement computer from DEWESoft *system folder*, browse to http://www.dewesoft.com/registration and press **Register** button.
4. Website will return you back registered license file. *Save* and *copy* this new file over the "`.lic`" file created in step 2.
5. Restart DEWESoft 7.

3 Project setup

The **project** is a set of settings which includes the Hardware setup and **Project setup**. By *changing* the project the settings within *Project setup* will therefore change.

To enter **Project setup**, please select:

Settings menu → **Project setup ...**

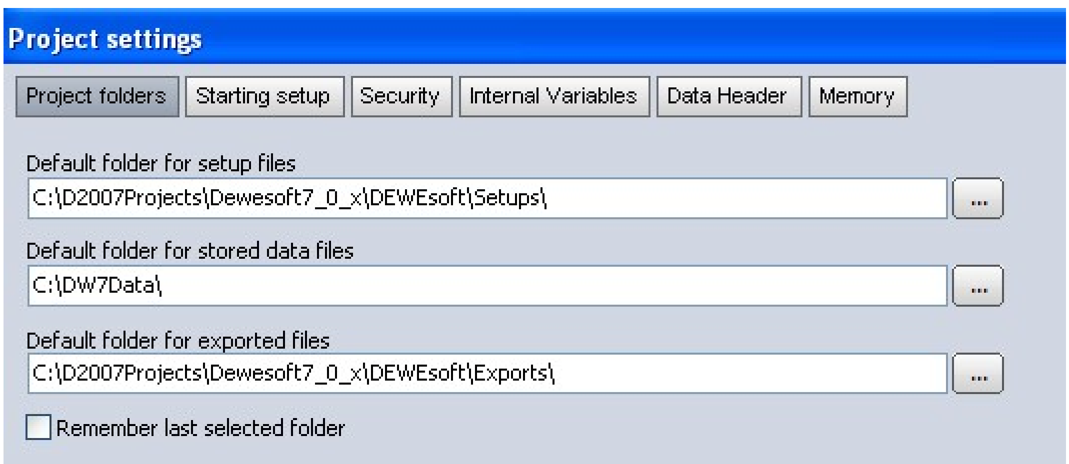


The *Project settings* window has several tabs:

- **Project folders** ... defines the standard place to *save* setups, data files and exported files
- **Starting setup** ... defines how DEWESoft *starts*
- **Security** ... to define *codes* for stopping measurement, altering the setup...
- **Internal variables** ... defines *internal variables* which can be used in sequences, math and displays
- **Data header** ... defines *additional information* which operator can enter at start or end of measurement
- **Memory** ... defines *memory sizes* for data and video

3.1 Project folders

Project folder defines a *starting point* within folder structure.



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**.

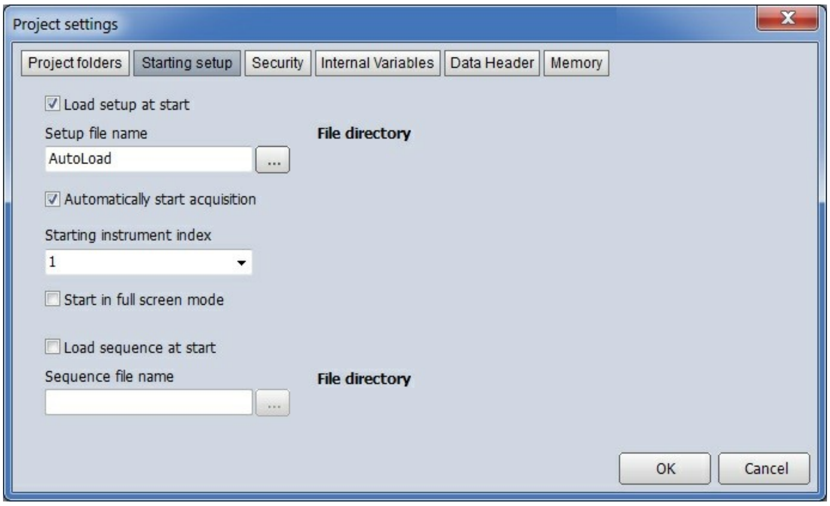
Default folder for stored data files

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 export files

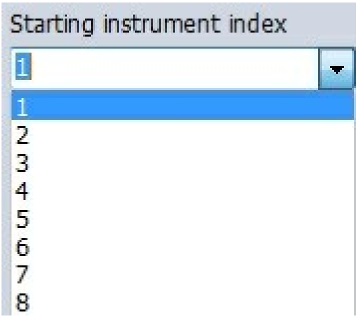
Defines the *default* place where the files are *exported*. If we select **Remember last selected folder**, then the folder which was last selected when exporting data will be set as default and remembered for next export.

3.2 Starting setup



- Allows to *load* your defined **setup** automatically at *startup* (otherwise the system defaults are used). When we select  button, standard *Windows Save data file name* window appear to define *different file* (`Setup file for DEWESoft` type) name as predefined name - `AutoLoad`.

- Allows to select **autostart** for the measurement and to *select* the *instrument* which should be *shown* from drop down list:

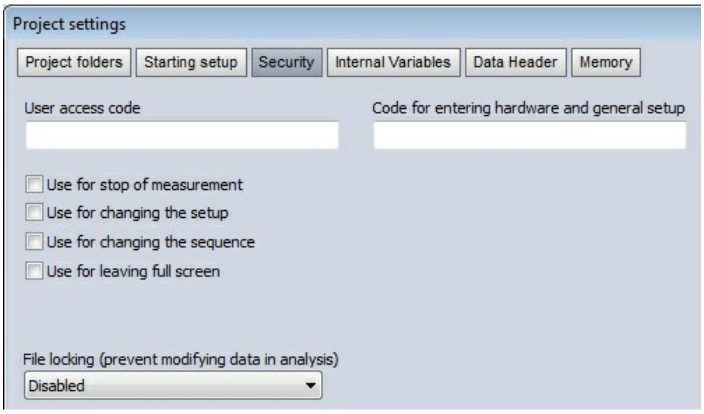


- Defines to *start* DEWESoft in *full screen* mode (combined with **security** this allows to run DEWESoft unattended)

- **Load sequence at start** will load the sequence defined in the file name and run when DEWESoft starts. Please don't use this function together with loading the setup on start.

3.3 Security

On this *General settings* window you can enter code for *preventing* unwanted **access** to DEWESoft procedures:



- **User access code**; this code is used (with checking box beside caption) for:
  - Use for stop of measuring;
  - Use for changing the setup
  - Use for leaving full screen

- **Code for entering hardware and general setup**
  code to separately *protect* access to hardware and general setup procedures.

The security tab allows the system administrator to **lock** certain parts of the software. Therefore the users might only be able to load the setup and run the measurement. We can also set DEWESoft to **run unattended**. In conjunction with Auto load setup/Auto start/Start in full screen we can issue that DEWESoft starts with certain setup in full screen, then we can *define* the code for **leaving** full screen and **stopping** the measurement. In this the system will run DEWESoft *without the risk* of someone stopping the software.

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

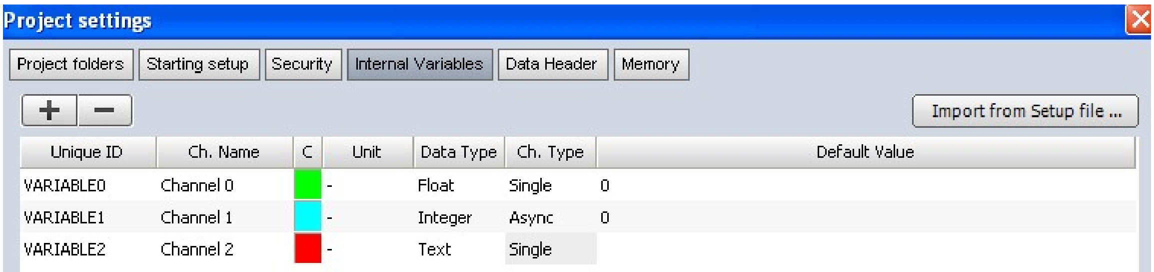
- Disabled ... file locking will be disabled and the files can be post processed

- With password ... file locking will be enabled and password will be defined to unlock the file in analyse

- Permanent ... file will be permanently locked and we can be sure it will not be modified in analyse.

3.4 Internal variables

Internal **variables** defines the channels, which can be used in *sequencer*, *math section* and *displays*.



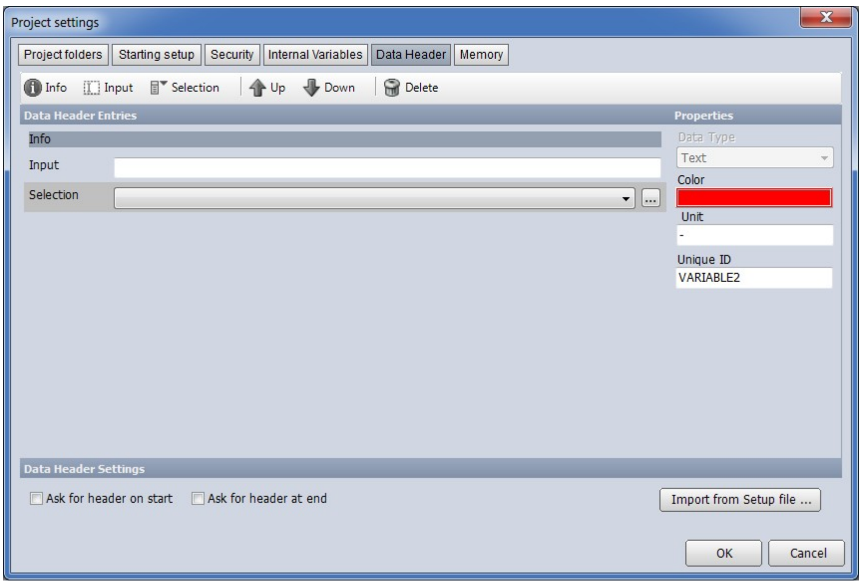
New variables can be added with a button. The Unique ID must be defined, which is used as reference for all others places where internal variables are used. Channel name can be freely defined, we can define the color and the unit.

Data type can be either Float, Integer or Text. Channel Type can be Single as for *Single value* (storing only one value per entire measurement) or Async (adding a value each time the channel changes from sequencer or math).

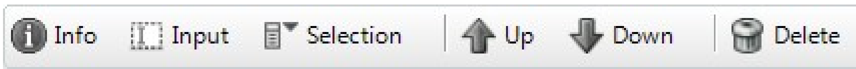
Default value describes what value the channel has when starting up.

3.5 Data header

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



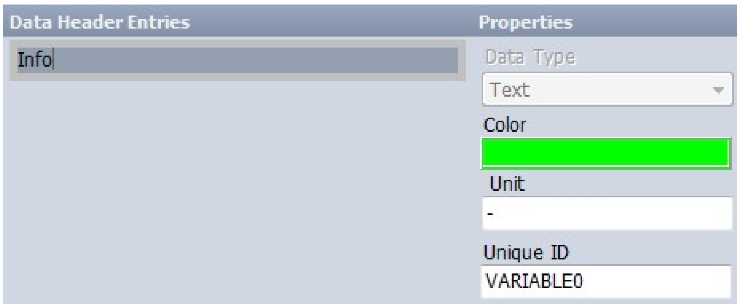
Icon bar



The icon bar at the top of the window allows you to change the right section of the window. We will now check icon by icon the functions. They can be used to ***create***, ***delete*** or ***move*** *fields*. The currently *selected* field will have a red *box* around.

Info field 

The Info button **adds** a new line in the right section of the window.



Just *click* in 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*...).

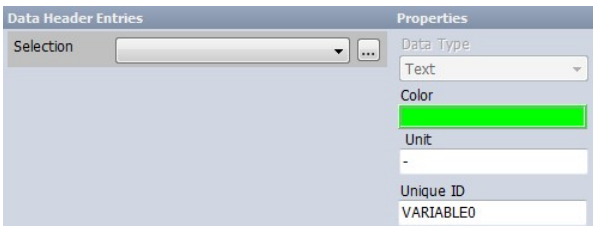
Let us *create* a **new field** with press the **Input** icon:

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* **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 *user names* or also a list of *departments* within your company.

Let us *create* a *new list* with press the **Selection** icon:

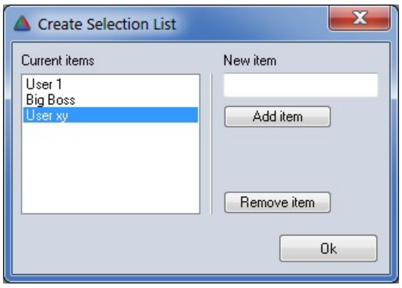


and *label* it (for example 'User' - click on 'Selection' and *overwrite* it):

Now you have to *define* the ***content*** of your *list*. To do that, press the ... button on the right side and **Create Selection List** window appear.

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:

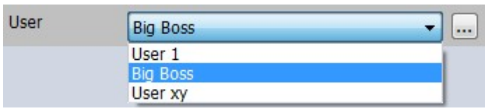


Enter as *many* names as *required*. You can also edit values direct in the list by select an item.

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:



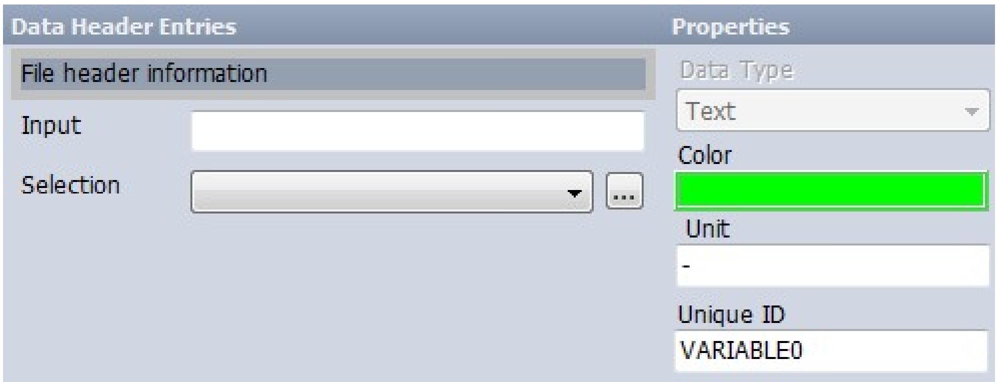
Position up/down

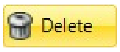
With the both buttons ***up*** and ***down***, you can now *change* the **position** of the:

- File name 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 appropriate button.

For example, we want to move above created **info** field to the *top* of the window, press the Move up button *twice*:



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.

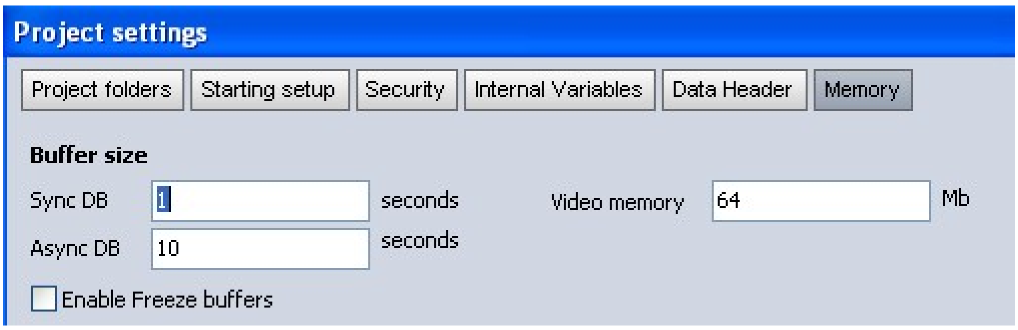
Ask for header on start option will pop up a window with all entries before start of storing. Ask for header on end will open the same window when measurement is stopped.

3.6 Memory

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

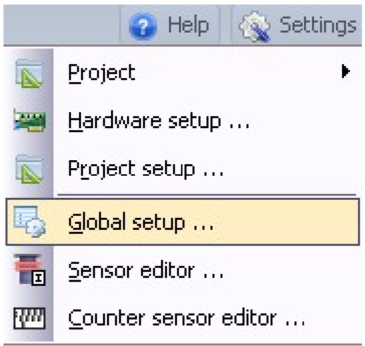
- Sync DB ... this is the memory size in seconds for all synchronous channels (analog, counters, ...); value should be larger than maximum refresh time - 2 seconds is default
- Async DB ... this is the memory size for all asynchronous channels (CAN, GPS and many others...)
- Video memory ... size of of the video buffer; with lots of cameras the default 64 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).



4 Global setup

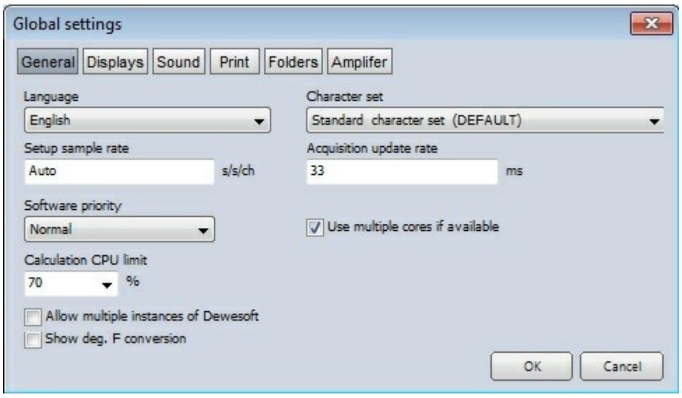
To enter **Global setup**, please select: Settings menu → **Global setup ...**



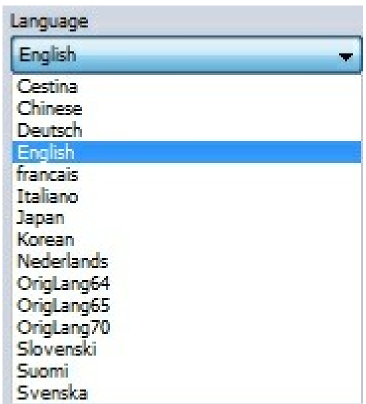
Global setup does not depend on the project and has six different tabs:

- **General** ... to set up general properties
- **Displays** ... to set up general display settings
- **Sound** ... sets up voice events
- **Print** ... sets up printing
- **Folders** ... displays system folder
- **Amplifier** ... sets up amplifier properties

4.1 General



– Language file selection (Chinese, Deutsch, English, Francais, Japan, Korean, Nederlans, Slovenski, Svenska, Cestina) can be selected from drop down list:



– Setup sample rate ... defines the rate that setup screen is running. In some cases AD card doesn't allow high rate because of multiplexing and we have to reduce the default value. In other cases some math procedures requires high sample rate to allow calibration (typical example is sound measurement). Default value is 5-10 kHz, depending on AD card.

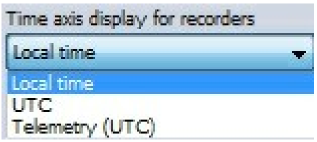
– Allow multiple instance of DEWESoft can be selected to allow that DEWESoft is started multiple times on the same computer for special applications (running different devices, analyzing old data while measurement process is running...).

– Special units setup ... to automatic *units conversion* from °C to °F check **Show deg. F conversion** field in channel setup of therm modules.

– Software 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. **Realtime** is useful only in special applications. Even though thinking that realtime priority would be the best, but in reality it might stall acquisition low level drivers which must have enough CPU time. Setting DEWESoft to realtime would mean that tasks like displaying data on the screen would have too high priority.

– Calculation CPU limit defines the overall CPU load above which DEWESoft will start to reduce the refresh interval on screen to *reduce* the CPU load; default value is 70%, but it can easily be set to 90% if we want to get maximum system performance.

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



This selection will affect the absolute axis of recorder, time display in multi meter 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 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.

– Acquisition update rate defines the minimum refresh time of internal. In other words this is the maximum **update rate** of the display during *acquisition*. By default it is set to 33 msec, therefore the display will update with 30 Hz refresh rate. DEWESoft will block the display if the acquisition will take too long and drop the update rate. We can set this value to the *lower* number if we want the display to *update faster* (like 10 msec - 100 Hz) but a very strong PC is needed for this or we can *reduce* the display update (for example to 200 ms = 5 Hz) to reduce CPU load for computers which runs unattended or if we have extensive calculations (on the limit of the performance). This defines the minimum, but the display will also not refresh if the CPU load is higher than defined as maximum.

– Use multiple cores is very important function to *split* the data acquisition and math processing between different CPU cores.

4.2 Displays

There are several items to set for displays:

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

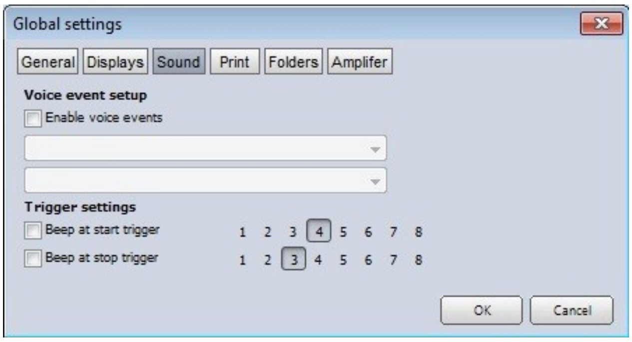
- Show instrument toolbar in full screen ... displays instruments button in full screen mode; full screen (available with Ctrl-F) usually shows only instruments, but in this case also a toolbar with standard displays are shown as in picture below:

- Allows you to select Background color of instruments from drop down list for *instrument displays* (white, light gray or dark blue):

Dark blue is useful for displaying data in bright sunshine while light background is most useful for displaying data on the beamer.

- Time axis display for recorders ... can be defined to show Local time (based on UTC bias), UTC time (universal time coordinated - same in all time zones) and telemetry UTC (utc and the number of days within a year).

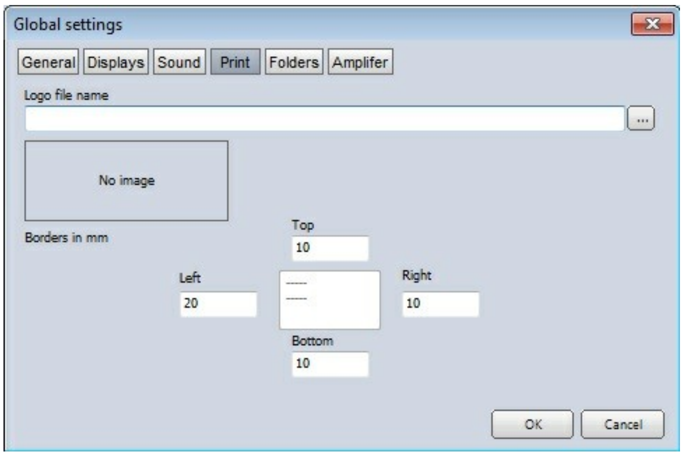
4.3 Sound



- Allows you to tell DEWESoft which DirectX capable *sound card* in this computer that you want to use for making **voice annotations** when storing data.
- Allows **definition** of the *sound frequency* for start and stop trigger in 8 steps.

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

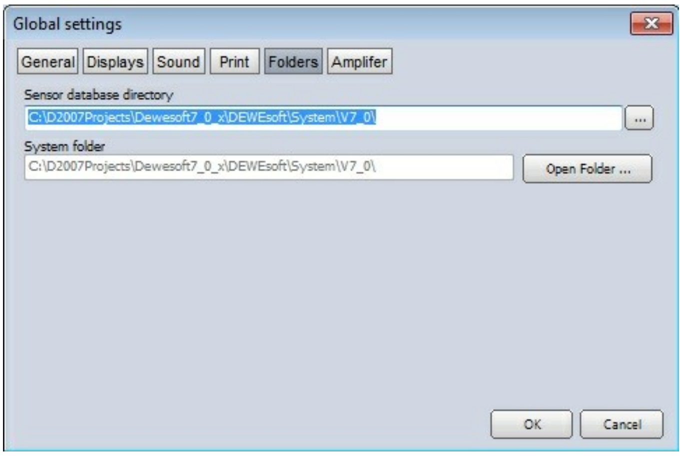
4.4 Print



Use your *own company* **logo** on *printouts*: after selecting ... button standard *Windows Open* window appear to simply select a bitmap file (*.bmp type).

In this section we can define also printer border in millimeters.

4.5 Folders

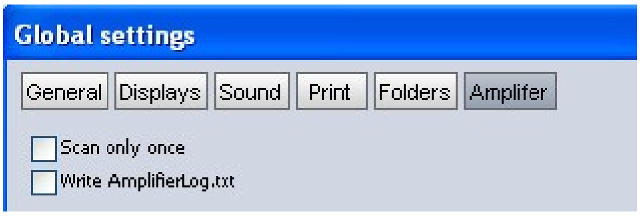


Define the *directory path* for the **sensor database** file; after selecting ... button standard Windows Open window appear to simply select *directory path*. As a standard, it is set to DEWESoft system folder.

Please find more information about the sensor database in User Guide → **Sensor database** .

In this section we can also **Open** the *system folder* (to copy licenses, add plugins ...).

4.6 Amplifier



Amplifier section has two options:

- Scan only once ... will perform only one scan of amplifiers and then stop scanning (disabled by default).
- Write AmplifierLog.txt ... will write all the log file of all requests and responses to amplifiers.

5 Sensor editor

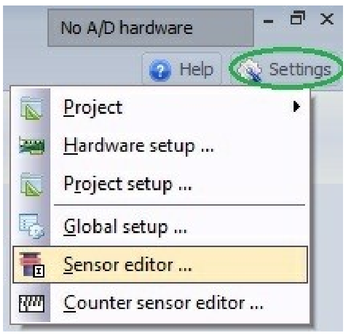
DEWESoft offers **sensor database** which **holds** the *list* and the *properties* of *all* sensors which will be used. In addition to the *linear scaling*, which can be done also in the input *Channel setup*, the sensor database offers also ***scaling*** by table or polynomial, even transfer curves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

Pre-defined *demo* **sensors** are for sure not what you would need for your application. The **sensor database** can be *created* and *adapted* according to your *requirements* (used *measurement hardware*). The **Sensors** (database) **editor** offers several *functions* for ***create***, ***edit*** and ***manage*** **sensors** and **sensor group**:

- **Add sensor**
  Creates a **new** *sensor* in the selected *group*
- **Remove sensor**
  Delete a defined sensor
- **Add group**
  Define a **new** *group* for sensors
- **Remove group**
  Delete a sensor **group** and **all sensors** *within* this group
- **Rename group**
  Rename a sensor *group*
- **Save file**
  Store the sensor *database* to **file**
- **Exit**
  Close the sensor database *editor*

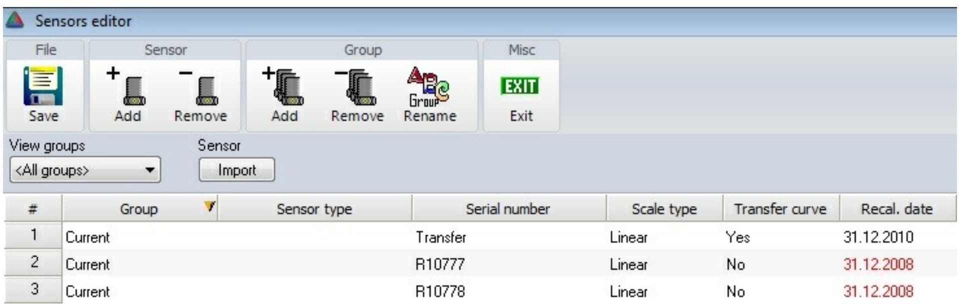
Each **sensor** is **defined** with ***information*** in *database*:

- **Sensor type**, **Serial number**, **Recal. date**
- **General sensor information**
- **Scaling type**:
  - **Linear**
  - **Polynomial**
  - **Table**
- **Transfer curve** (also for **Sensor correction**)



Sensor definition - Edit sensor

To enter the **edit** mode, please select in Data menu **Sensor editor ...** and the *Sensors editor* with the list of all possible sensors window will open:

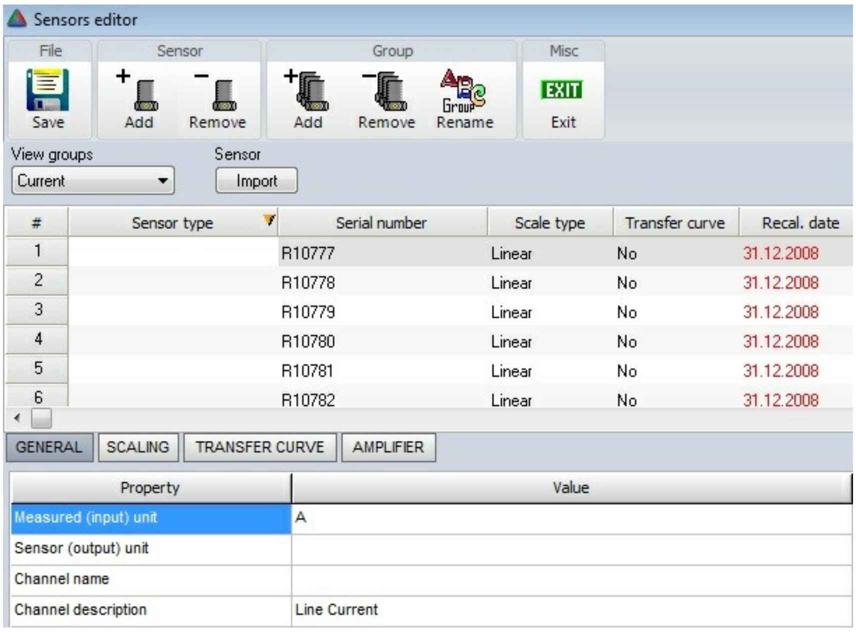


Any already *defined* (and pre-defined) sensors and/or connected TEDS sensors will be listed *automatically* as our example above shows. The Sensor editor offers several functions as you can also see as icons on icons bar (*see* above).

In *table* of sensors following *columns* are available:

Group	The Group column is only available when the *View groups* selection is set to <All groups>. Use it to **move** a sensor into *another group*.
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 (even in different groups), because this information is used for sensor *identification*.
Scale type	The sensors editor differs between *linear*, *polynomial* and *table scaling*.
Transfer curve	Can be used for sensors with *phase* characteristics like current clamps...
Recal. date	Enter the **date** when the sensor has to be *recalibrated*. When the date expires, the *Recal. date* will change to red color.

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



At the bottom of the sensor database editor are *additional information* of the selected sensor:

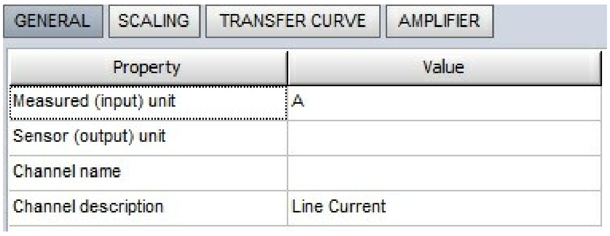
General	- the general section contains ***additional** information* about the *sensor*
Scaling	- contains the ***scaling** information* of the selected sensor
Transfer curve	- only available if Transfer curve is *enabled*

GENERAL sensor information

The General tab at the bottom of the screen will *automatically* open when you select the Group, Sensor type, Serial number or Recal. date column for the *desired sensor* (*see* screen above). It contains following information:

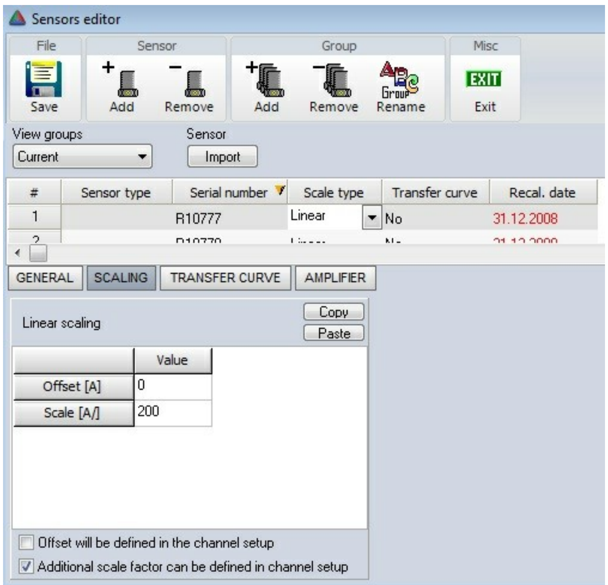
Physical (input) unit	the *physical* **unit** of the sensor, e.g. V, A, °C, mm, psi, %,...
Electrical (output) unit	the *electrical* output **unit** of the sensor, most times V or A
Channel name	use this field to *pre-define* the **channel name** for the *setup*
Channel description	**additional** *information* about the sensor, helps to select the right sensor in the input setup
Wanted range min	*Minimum* electrical **input range**
Wanted range max	*Maximum* electrical **input range**
Wanted Filter [Hz]	desired *lowpass* input **filter range**

Example of general sensor setting:



Sensor SCALING

DEWESoft *supports* different **scaling** *types* within the sensor database. When we select in *Sensor editor* window **Scaling type** *field*, the **scaling type** *selection list* appear in this field and SCALING tab is selected *automatically*. After selecting desired scaling types from this selection list in *Scaling* tab automatically appears information about scaling.



DEWESoft *supports* three different **scaling** types within the sensor database.

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 the transfer curve will deliver more accurate results.*

- **Linear** used for ***linear*** *sensors*, calculated by:

$$y=k*x+d \qquad (\text{physical value} = \text{scale} * \text{measured value} + \text{offset})$$

... *enter* this value in Scaling tab - **Linear scaling**

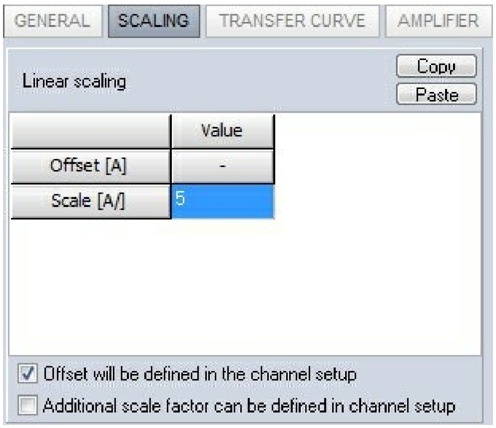
Example → *see* above

Option:

Copy, **Paste** usually *Windows* operation to **copy** / **paste** *value to* / *from* another sensor

Offset will be defined in the channel setup

after check this option Offset [] value we set in channel setup and in Scaling tab - Linear scaling is yet inaccessible:



Additional Scale[] factor can be defined in channel setup

to enter scale factor also in channel setup (e.g. to be able to *reverse* sensor polarity)

- **Polynomial** used for ***nonlinear*** *sensors*, calculated by

$$y = (a0 + a1*x + a2*x2 + ... + an*xn)$$

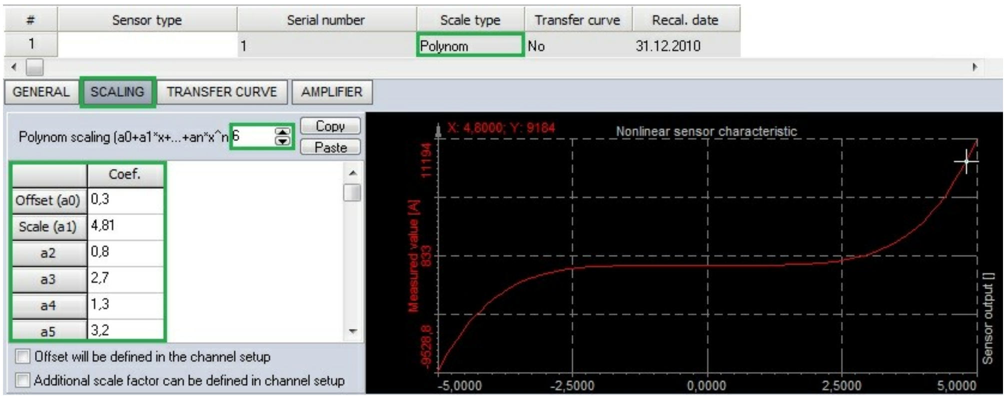
Option:

- Polynom scaling (a0+a1*x+...an*x^n)

in this field enter n and ***coefficients*** in Coef. column (coefficient a0 = offset)

- other fields: same as for *Linear* (*see* above)

- example:

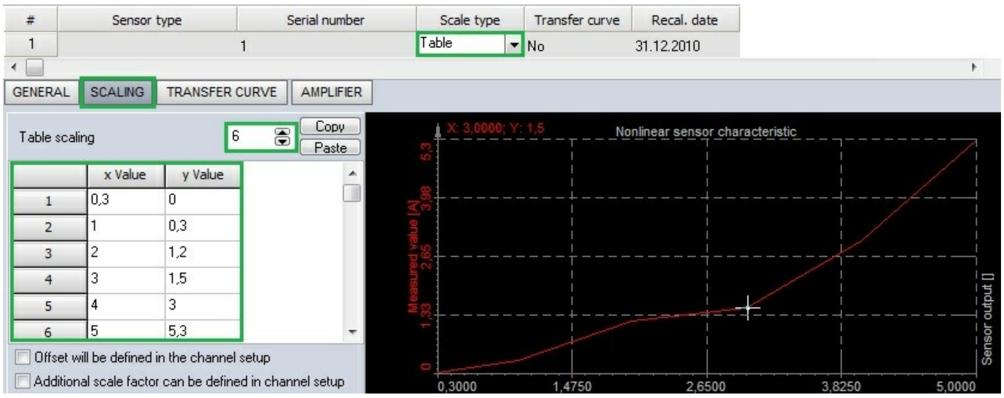


- **Table** also used for ***nonlinear** sensors*, but normally *easier* to enter because most *calibration information* contains *several* **calibration points**

Option:

- Table scaling in this field enter *number of points* (rows of table) and in table below xValue and yValue
- other fields: same as for *Linear* (*see* above)

- example:



TRANSFER CURVE

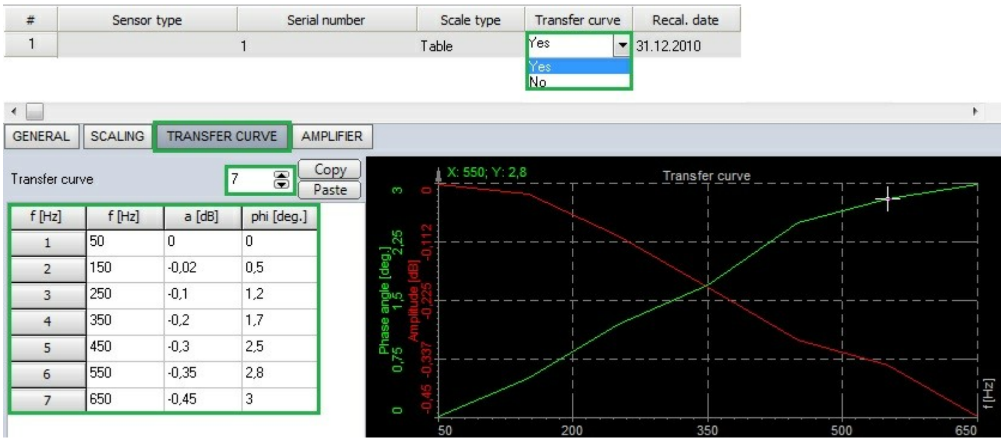
The **transfer curve calibration** can be used when the *frequency behaviour* of the *sensor* is known (acquire transfer curves for most common sensors *already measured*; to *copy it from the calibration sheet* of the sensor, if the cal sheet includes the transfer curve; the third option is to *measure it* with **DeweFRF**, but this requires some equipment). Some companies offer calibration reports for sensors also in 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 choose Yes to tell that the transfer curve *will be defined* and now we need to enter the points of the curve.

Option:

- Transfer curve in this field enter *number of points* (rows of table) and in in table below f[Hz] - signal frequency, a[dB] - amplitude deviation in dB and the fi[deg] - phase angle in degrees

- **Copy**, **Paste** usually *Windows* operation to **copy** / **paste** *value to* / *from* another sensor

- example:



NOTE:	*Be aware that the transfer curve is only helpful in frequency domain applications, like FFT, harmonics or octave analysis!*

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

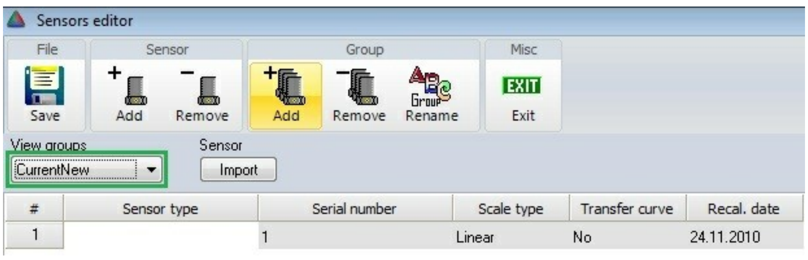
Manage sensors and sensor group

Add group

Before you start editing sensors you should think a few seconds about **group**ing your **sensors**. Imagine that you have dozen or hundreds of sensors, it will become hard work to find them again. Think of *easy* names, related to the sensor *type* (for example Temperature, Vibration, Current...), the *manufacturer name* or the *application*.

We create now a *new* **group** called "CurrentNew". Simply click on the **Add group** icon and *enter* the **group** *name*:

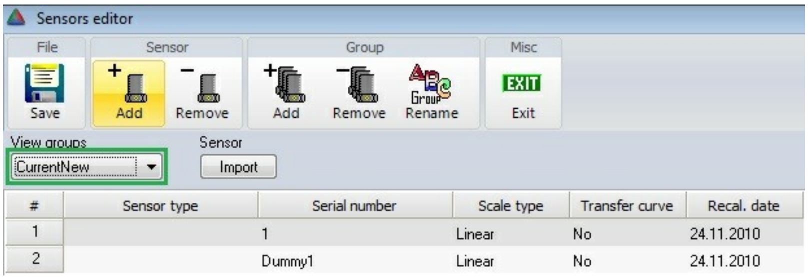
When you press the **OK** button, the *new* **group** will be **created**, the View groups selection will change to the *new group name* and *new* **sensors** will be *created automatically*:



NOTE: *Sensor groups which contain no sensor will be automatically deleted*

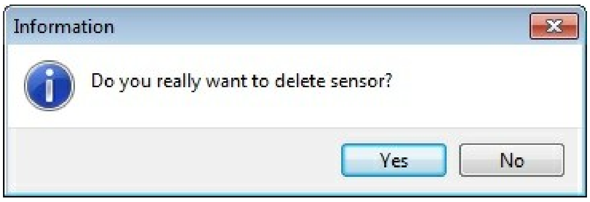
Add sensor

If you already have an existing *sensor group* and just want to ***add*** *new* **sensors**, *select* the *desired group* from the View groups selection and press the **Add sensor** button and *new* row - sensor is *added* to table:



Remove sensor

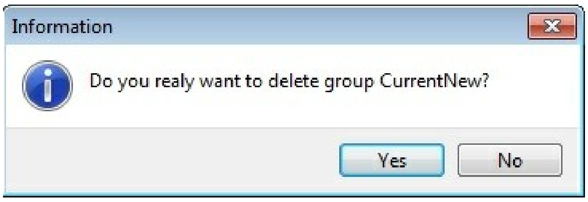
To ***remove*** a **sensor** just *click* on it - the whole *line* in the table will receive a grey *background* - and press the **Remove sensor** button.
A warning will appear and ask if you really want to delete the sensor.



ATTENTION: *It is not possible to restore deleted sensors*

Remove group

To ***remove*** a **group** of sensors *select* the desired *group* from the View groups *selection* and press the **Remove group** button. A warning will appear and ask if you really want to delete the *selected* group.



ATTENTION: *All sensors within the selected group will be deleted! It is not possible to restore deleted groups and it's sensors*

Rename group

To ***rename*** a **sensor group** select the desired *group* from the View groups *selection* and press the **Rename group** button. You will be asked to enter the new group name:

Save file (sensor database)

To ***save*** *all changes* in the **sensors database** just click on the **Save file** icon. The data will be stored immediately in a XML styled file called **sensors.xml**.

ATTENTION: *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.

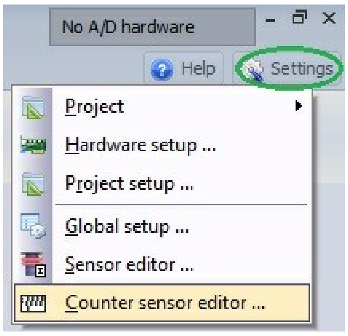
Yes → ***save*** changes or **No** → *leave without* changes

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

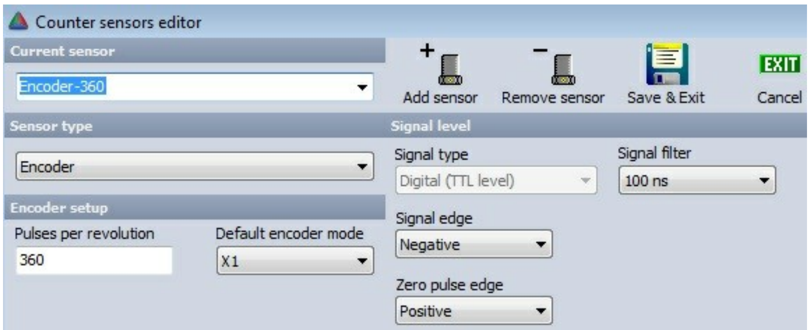
6 Counter sensor editor

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 sensors editor* can be accesses via Data main menu → **Counter sensor editor ...** item.



When this menu item is selected, the *Counter sensor editor* window appears to define *counter sensor*.



There are *several predefined* sensors already added when first starting DEWESoft, but we can always **add** new ones, **modify** existing or **delete** the sensors. On upper right side of *Counter sensor editor* window common command icons appear to:

Add sensor button will ***add new*** sensor. The sensor will be named '**New sensor**', but we can *rename* it in the current sensor drop down.

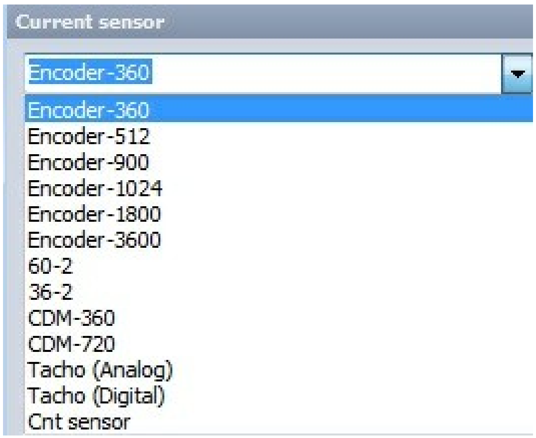
Remove sensor button will ***remove*** *currently selected* sensor.

Save & 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.

for more information about above ***Commands*** *see* → User Guide → **Sensor database**

On *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 on following sections:

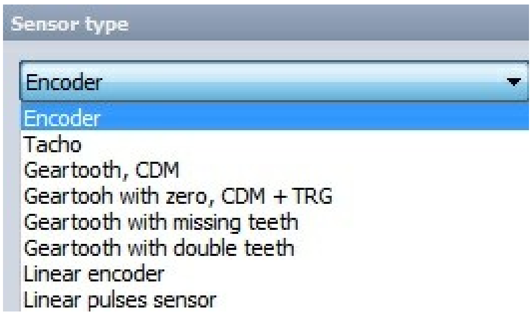
Sensor type

Signal level

Encoder setup or **Geartooth setup** (depend on selected Sensor type)

Sensor type

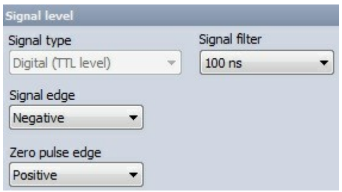
There are several basic *sensor types* available which can be selected from Sensor type drop down list :



Encoder	classic angle encoder with A, B and Z signals, can be *only digital* and used with Orion *counter*
Tacho	*one pulse* per revolution, can be either *analog* level or *digital*
Geartooth	sensor with *defined number of pulses* per revolution, but *without any zero* pulse
Geartooth with zero	sensor with *defined number of pulses* per revolution *with zero* pulse, sometimes also called CDM sensor
Geartooth with missing teeth	a classic in-vehicle sensor with *any number of pulses* where some teeth are *missing* for *zero* pulse recognition. 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 for two teeth
Geartooth with double teeth	in-vehicle sensor with *any number of pulses* per revolution with *some double* teeth. A typical example is 36+1

Signal level

There are several *signal level* settings.

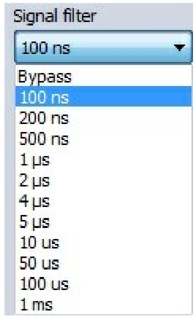


From drop down list can be selected:

Signal type

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

Signal filter

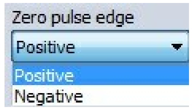


is the *holdoff time* after the *trigger* in which the triggering logic is *disabled.*
It is used to *prevent double* triggers.

Signal edge

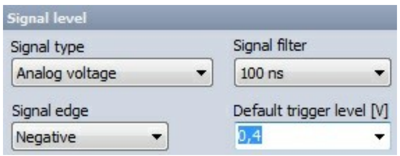
can be either *positive* or *negative.*

Zero pulse edge

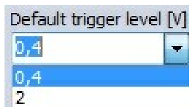


For *encoders* and *geartooth* with *zero* we also define the **Zero pulse edge** which can and can be either *positive* or *negative.*

For *analog sensors* we also define the Default trigger level [V].

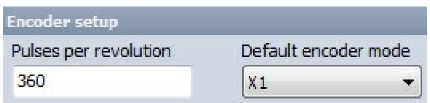


Default trigger level



Please note that these are *default* values and can be *adjusted* when the sensor is used in *each specific* application.

Encoder setup

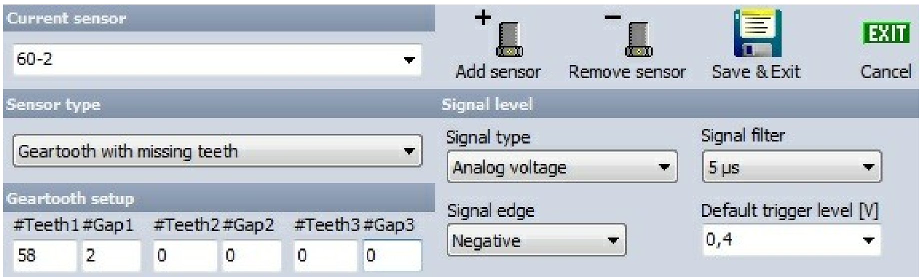


For *encoder sensor* we define:

- Number of Pulses per revolution. Standard values are multiple of 2 (256, 512, 1024). These sensors are mainly used for *external clocking* that we can have the frequency spectrum as a direct multiple of number of revolutions to easily *see* the *harmonic components*.
  Another standard values are related to degrees (360, 720, 1800, 3600) where the reason is obvious - nice *angle resolution*.

- We also define the Default encoder modes. Encoder modes are explained already in counter chapter.

Geartooth sensor setup



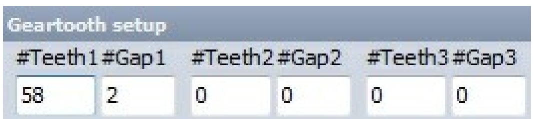
for more information about ***Sensor type*** and ***Sensor level*** *settings → see above*

Geartooth and geartooth with zero sensor settings

For these sensors we only need to define the *number of pulses per revolution*, or in different words, *number of teeth*.

Geartooth with missing teeth sensor settings

For these sensors we define the *number of teeth* - #Tooth and the *gap lengths* - #Gap.
For the number of teeth we need to enter the number of teeth which *exists*. The example in 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 *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 on right shows the settings for 36+1, so geartooth with 36 *teeth* and *one double* tooth.

Part

III

Installation Guide

This section describes the installation procedures of DEWESoft itself. For effective DEWESoft use all needed ***hardware devices*** and *their* ***drivers*** must be properly *installed* on respective computer, after that DEWESoft must be *set up* to *acquiring* and *processing* ***measured data*** with this hardware.

DEWESoft X1 can be used with recommended *system* hardware and software.

for information about ***recommended system hardware*** and ***software*** *see* → **Application requirements**

for common information about DEWESoft *see* → **General Information**

Before using DEWESoft X1 for working in **real** mode, you have to *install* the **A/D board** *drivers*. Be aware that the proper drivers are installed. Detailed information about the installation procedure is available in the manufactures Drivers documentation.

In next chapters following procedures are described:

DEWESoft installation

Hardware driver installation

Where to find the drivers

High-performance application requirements

For all high-performance applications like high-speed *video* or high-speed *data acquisition*, we recommend the following hardware and software:

- WINDOWS 2000 / XP / Vista [1)]
- Intel Pentium 4 2.4 GHz processor or higher
- 512 MB RAM or higher
- Serial ATA or SCSI hard disks and/or RAID systems
- Approx. 40 MB free hard disk space for DEWESoft
  Additional 20 to 200 MB for drivers and post processing software packages
- A/D board for using **real** mode (requires license key)

NOTE: *We strongly recommend to running DEWESoft-NET with Windows Administration rights - otherwise file access or driver problems might occur!*

[1)] DEWESoft-NET is not running under WINDOWS 95/98/ME/NT4 operating systems!

Windows Vista note

If running on Microsoft Windows Vista, it is not only necessary to run with administrator rights, but it is also important to turn **off** User account control (UAC). This is the special option switched on by default which prevents the software to run administrative tasks. For most of the programs this is fine, but DEWESoft uses lots of registry calls (for example performance checking for defining refresh rates) and the UAC prevents the normal operation of the software.

This option can be found in Control Panel → User Accounts → Turn User Account Control On or Off.

1 General Information

Trademark notice

All trademarks are acknowledged to be the property of their owners, with all rights and privileges thereto. No infringement is intended.

DEWESoft and DeweFRF is a registered trademark of DEWESoft d.o.o.
Microsoft, Excel and Word are registered trademarks of Microsoft Corporation.
National Instruments is a trademark of National Instruments.
Microstar, DAP and iDSC are trademarks of Microstar Labs Inc.
Data Translation is a trademark of Data Translation, Inc.

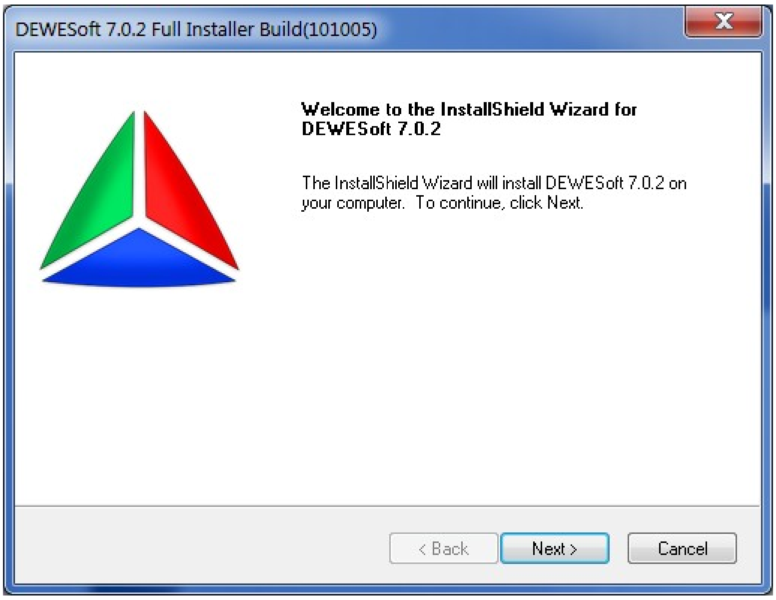
Disclaimer

DEWESoft make no claim about the efficiency or accuracy of the information contained herein. Use of this manual is entirely at the user’s own risk. Under no circumstances will DEWESoft assume any liability caused by the use, proper or improper, of this manual or the information, textual, graphical or otherwise, contained within it.

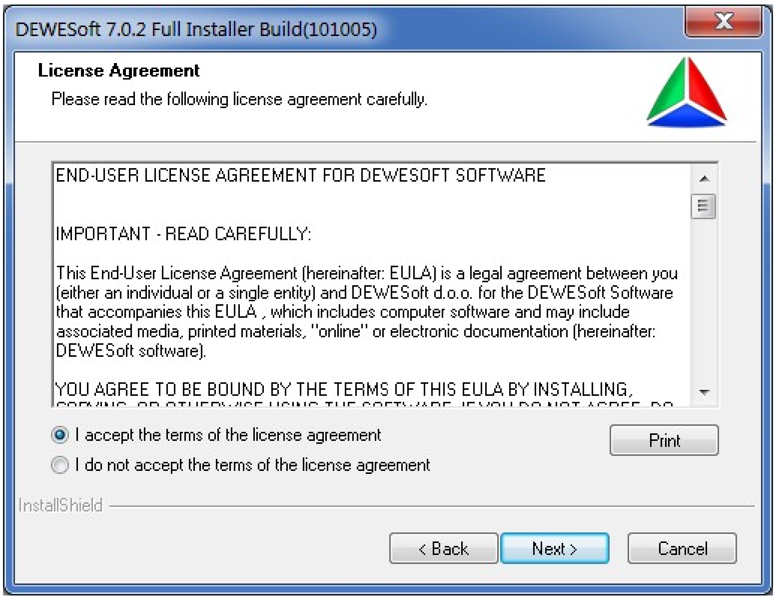
2 DEWESoft installation

In order to install DEWESoft for the first time, please Run `Dewesoft Full Installer`.

When the installer starts, it shows the message for installation. The installation procedures might ask to *remove old* before installing new software. If these versions are not installed, simply select **Next**.

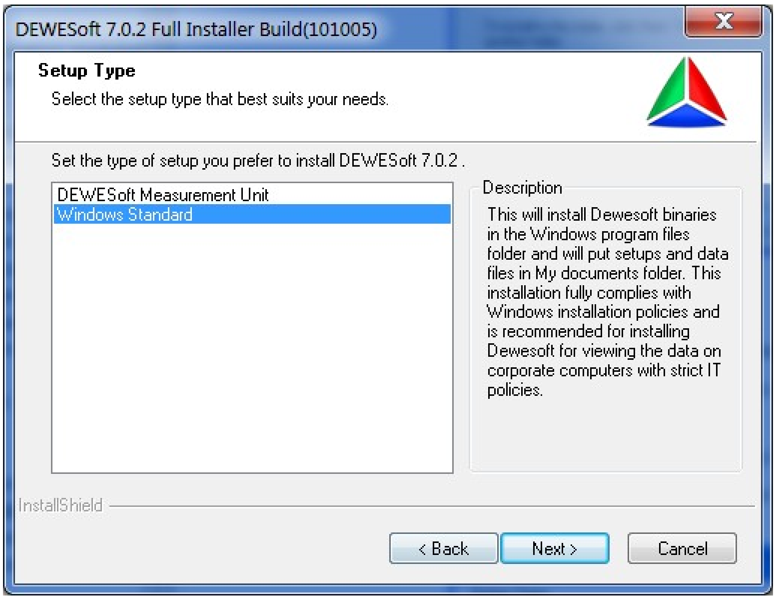


Following screen will show the end user license agreement. In order to continue with installation, license agreement must be accepted with I agree to the terms of the license agreement check box. Please continue with pressing the **Next** button.

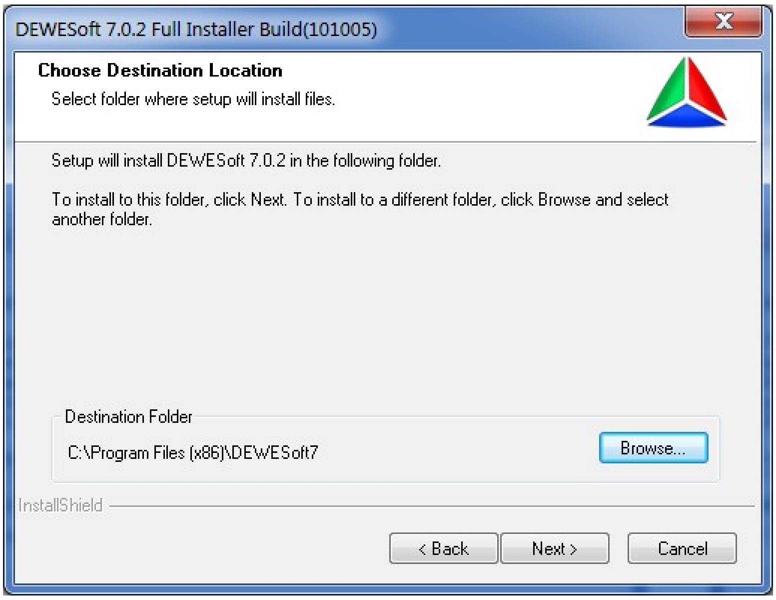


Next screen shows some *basic installation options*:

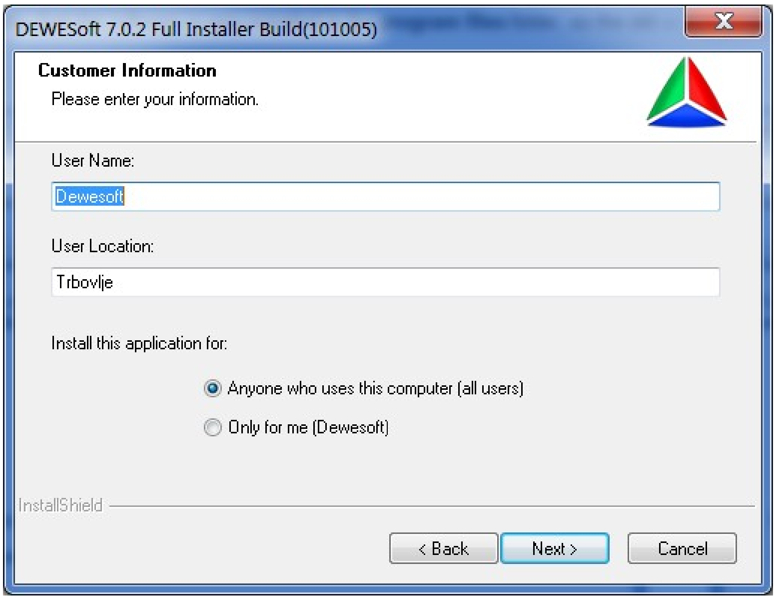
- DEWESoft Measurement Unit: this mode is highly recommended if the software is installed on unit intended to take measurements. All the binaries and other files will be installed to one folder, which should not be located on the system drive (like `D:`). In case of OS failure, it can be restored without a need of reinstalling DEWESoft. Additionally, this mode is recommended for fast writing of data. OS drive gets fragmented with time and therefore additional drive is much better for sustained fast read/write access.
- Windows Standard: this mode complies with the Windows standard way of installing the software. Binary files are copied to **`Program files`** folder and the setups, scripts and data folder is in the **`Document and settings`** section for each user. This installation is intended to be used with computers regulated from company IT departments (office computers used for data review and analysis).



The next screen asks for *installation folder* for the software. In case of Windows Standard installation, the software is installed in the **`C:\Program files`** folder. In case of DEWESoft Measurement Unit, the recommended folder (if exists) is **`d:\dewesoft7`**. The folder can be changed, but if it is acceptable, simply click **Next**.

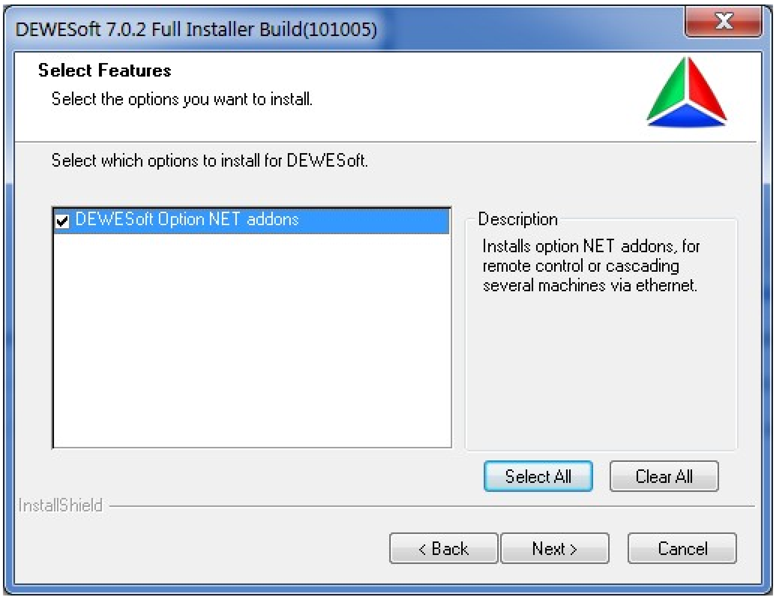


Please enter the User name and User Location, which will be used for registering the software (if needed). Please also choose if the application will be installed for *every user* or just *current user* (which *should have enough rights* for installing the software).

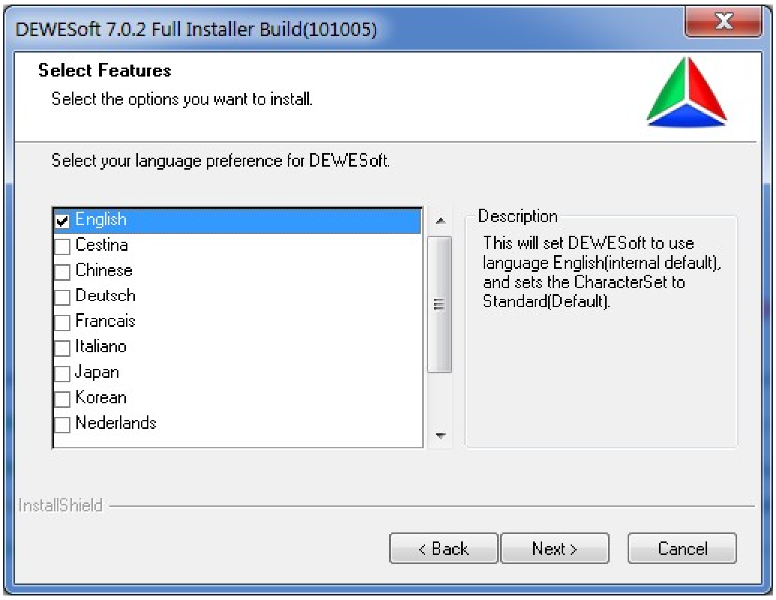


Next screen shows the options to be installed with DEWESoft. There is only one option which can be installed additionally:

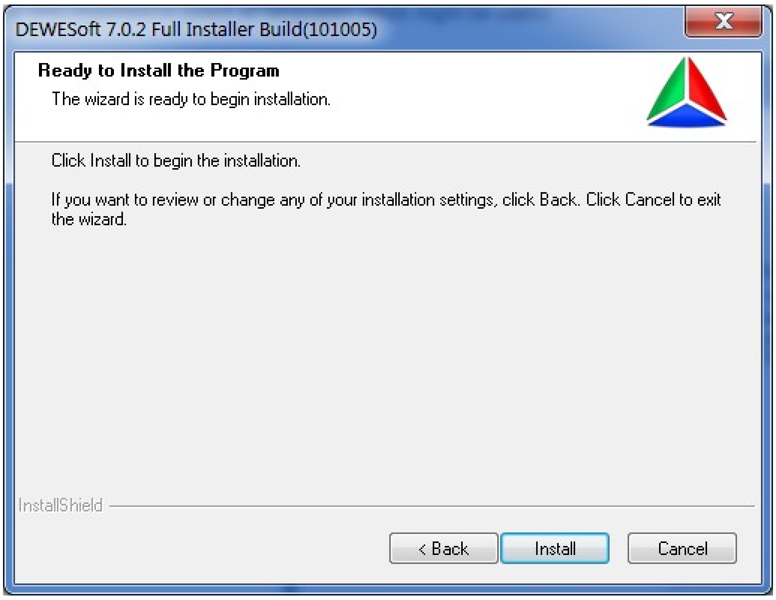
- DEWESoft option NET addons: will install option **NET** for *networked data acquisition* and additional sub packages like VNC server for remote connection



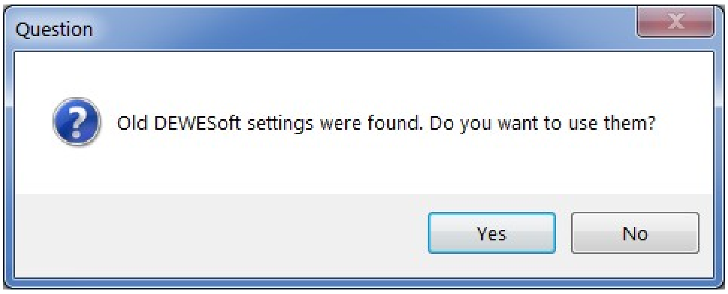
Next screen will show the **language** options - you can select as many languages as you like, they will appear in *System → General setup* and can be changed afterwards. At this point it is only important to select *all* languages which *might be useful*.



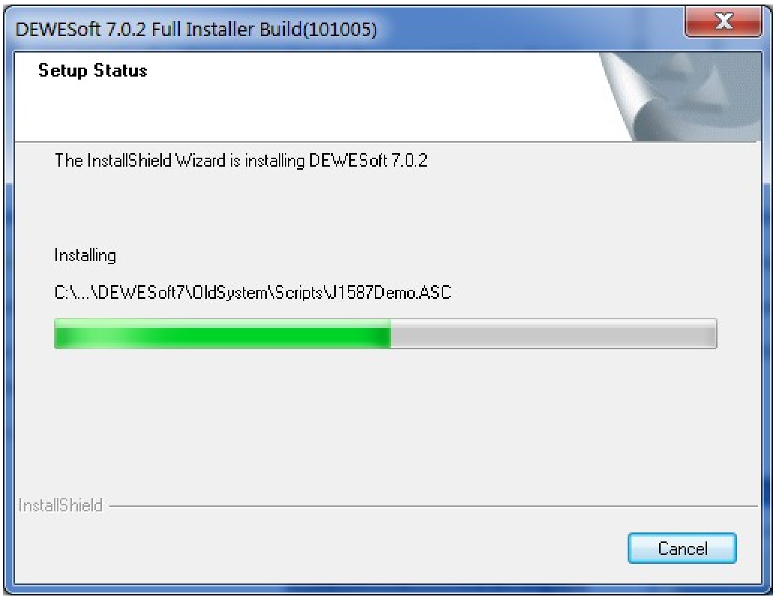
Next step will ask again for a *confirmation to install* the software. After pressing **Install** button DEWESoft will be installed.



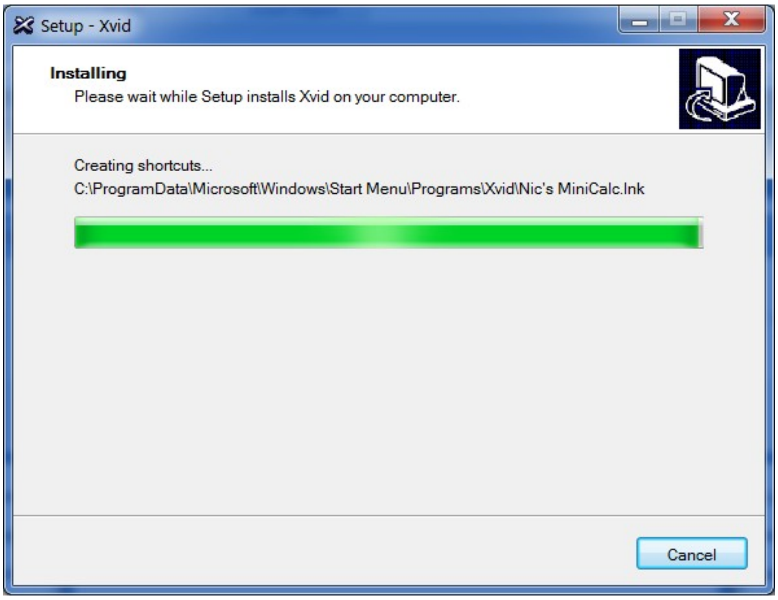
In case that *old* installation is found, Installer will ask if old **settings** should be kept. If pressing **No**, the default settings will *replace the* existing ones.



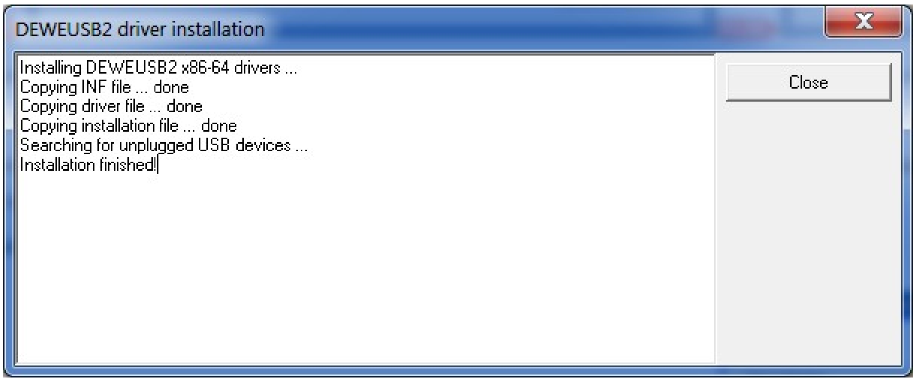
Next step will show the progress of the installation.



A setup will also install *additional components* needed for operating DEWESoft (VNC, XVid...).

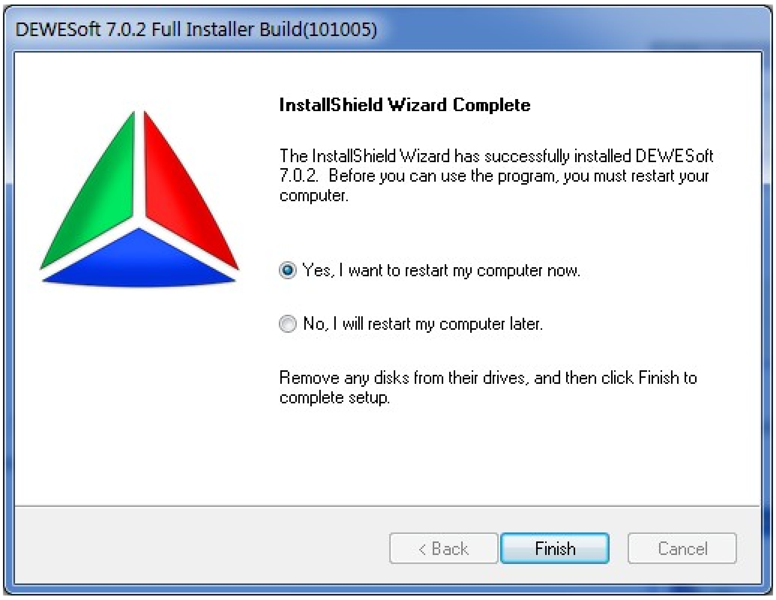


It will also *automatically install* the **drivers** needed for DEWESoft USB devices.



To *complete* the drivers installation, you need to *replug all* the DEWESoft devices which might be connected during installation of the driver.

At the end, DEWESoft will ask to *restart* the computer to *finalize* the **installation**. Computer can be restarted also at later time.



This step completes the installation of the software.

After restarting and opening DEWESoft, *Hardware settings* will appear (if we didn't choose the **demo** mode). Please consult **Hardware setting** part of the manual for further steps how to **enable** and **register** the hardware.

Part

IV

Reference Guide

Reference Guide is designed to give some DEWESoft *background* information and *additional* information about *supporting software* Application.

DEWESoft ***background information***	• **Basics of A/D conversion**
	• **Theory of frequency analysis**
	• **H1, H2 and Hv calculation algorithms**
supporting ***software Application***	• **FlexPro scripts**

1 Basics of A/D conversion

This article helps to understand the theory and answers some of the most asked questions regarding A/D conversion.

In the measurement world of *analog* recorders and *analog* oscilloscopes, we had the chance to see at *any time any amplitude* value. The *digital* has limitations; the measured signal from an A/D converter is *always* discrete in both *time* and *amplitude*. This is caused by the used conversion technology, the system speed and also the limited storage capabilities.

But the digital world is becoming more and more common, so we have to know some backgrounds about the functionality of **A/D converters**. Understand the technology helps to achieve best *measurement results* and *saving time* and *money* at the same time.

There are three important *elements* in the *digital* measurement world:

- Time resolution (= **Sampling rate**)
- **Amplitude resolution** (= number of bits of the converter)
- **Accuracy**

1.1 Sampling rate

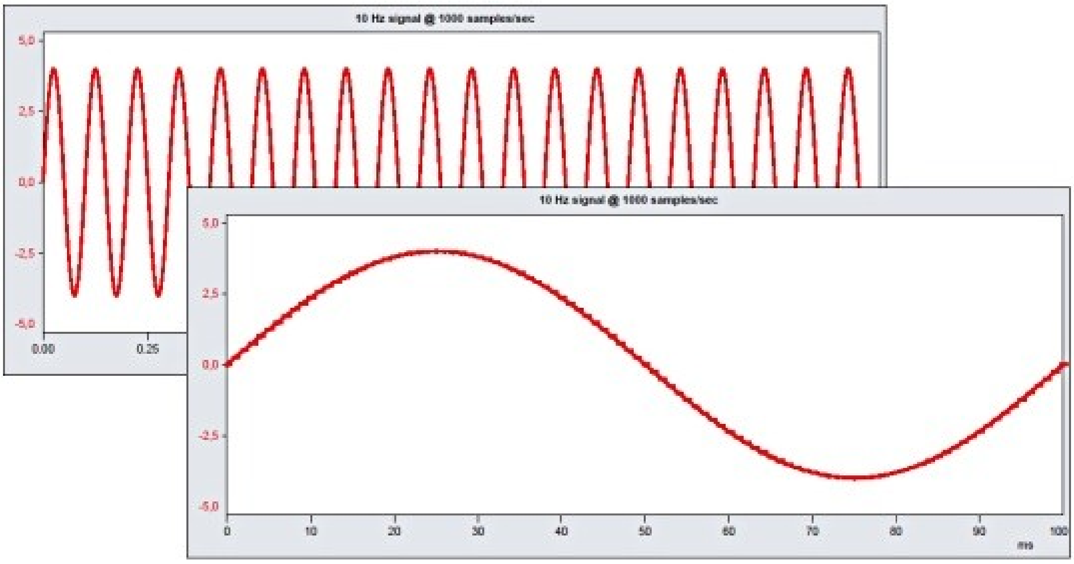
Selecting the *correct* **sampling rate** is very important to obtain good measurement results. When the sampling rate is too *high*, you will see a perfect signal, but you will also receive plenty of *unnecessary* data - waste of hard disk space.

A sampling rate *below* the *signal frequency* will save hard disk space - but delivers an unusable result: The displayed signal looks interesting, but is completely wrong (= aliasing effect).

The following example should show you the relation between signal frequency and sampling rate. The example shows always a 10 Hz sine wave with ±4 V amplitude. We will decrease the sampling rate from too high until too low.

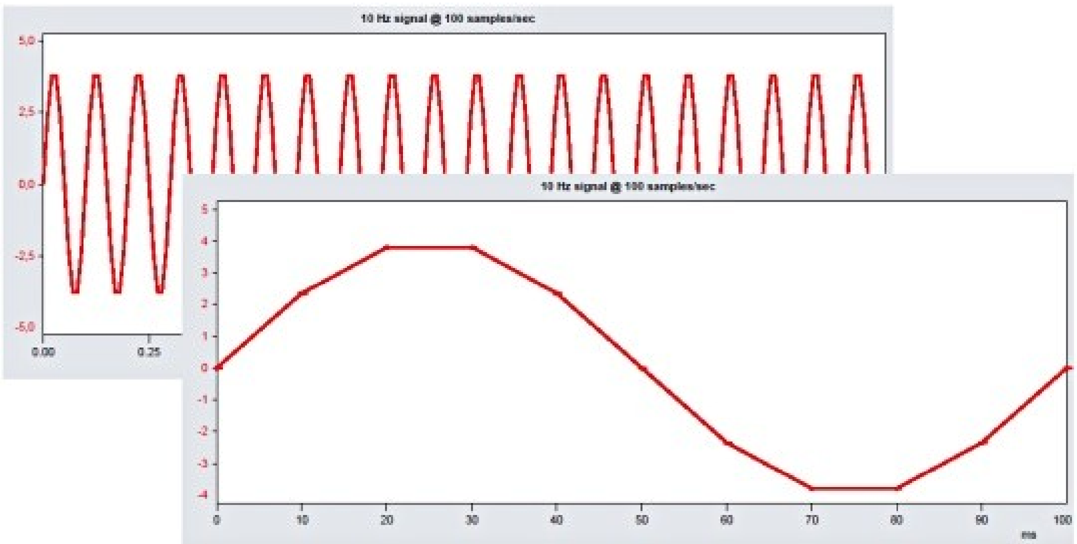
- *10 Hz sine wave, 1000 samples/sec => 100 sampling points per period*

  shows a wonderful *precise* sine wave signal - but requires *plenty* of hard disk space



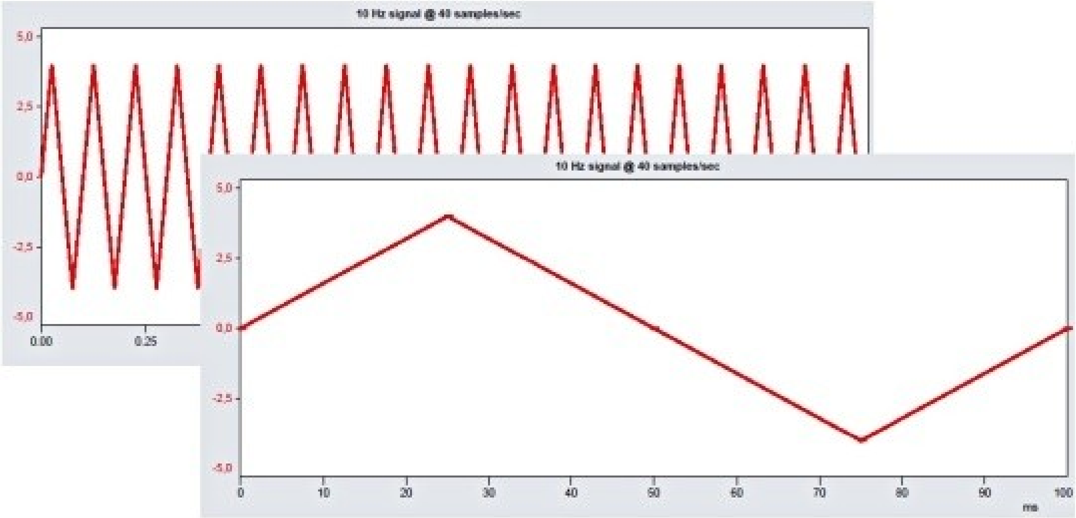
- *10 Hz sine wave, 100 samples/sec => 10 sampling points per period*

  delivers a quite *useful* sine wave with *moderate* disk space requirements



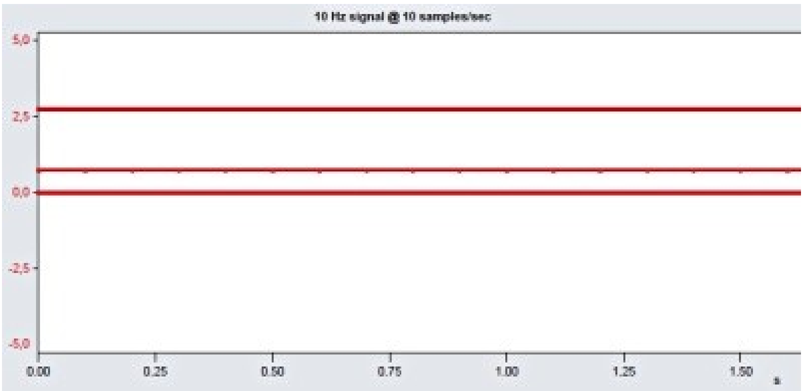
- *10 Hz sine wave, 40 samples/sec => 4 sampling points per period*

  delivers **no** sine wave signal any more. The amplitude information will get *completely* **lost**, because you may sample at any time, but not at the maximum. Even the time and frequency analysis can be a *problem*, depending on the sampling time



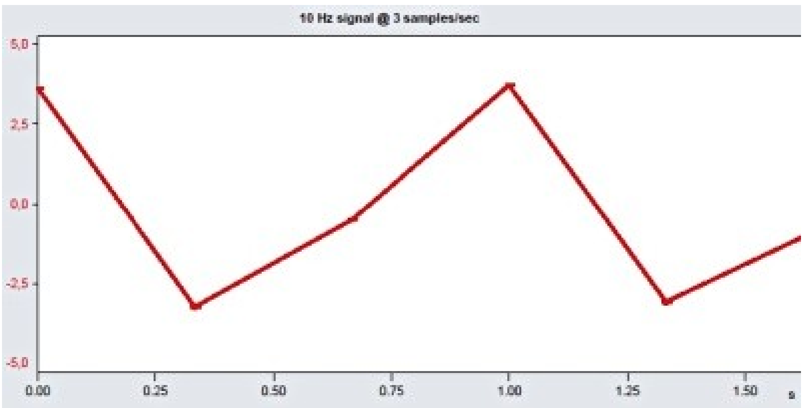
- *10 Hz sine wave, 10 samples/sec => 1 sampling point per period*

  shows a *flat line*, independent from the sampling time



- *10 Hz sine wave, 3 samples/sec => 0.33 sampling points per period*

  delivers an interesting result: there **seems** to be a signal, but the frequency is much below the 10 Hz → this is a typical aliasing effect



All the examples are *idealized* and won't appear that extreme in the reality. But they can give you an idea how signal frequency and sampling rate corresponds. For the reality, we would *recommend* a **sampling rate** which is about 5 to 20 times higher than the *highest expected signal frequency* - this varies from application to application.

1.2 Amplitude resolution

Up to now, we just worried about the time resolution of our signal. But the **amplitude resolution** is also important. It is given by the *number of bits* of the A/D converter and gives the *smallest difference* between *two* values of an *input signal*. The most common values are 8, 12, 16 or even 24 bits. The following table shows the relation between bit resolution and input range resolution:

Number of bits	Discrete steps	Minimum resolution for ±1 V input range		±5 V input range		±10 V input range	
8	256	0,007812500 V	7,8 mV	0,039062500 V	39,1 mV	0,078125000 V	78,1 mV
12	4096	0,000488281 V	0,488 mV	0,002441406 V	2,441 mV	0,004882813 V	4,883 mV
14	16384	0,000122070 V	0,122 mV	0,000610352 V	0,610 mV	0,001220703 V	1,221 mV
16	65536	0,000030518 V	0,031 mV	0,000152588 V	0,153 mV	0,000305176 V	0,305 mV
24	16777216	0,000000119 V	0,119 µV	0,000000596 V	0,596 µV	0,000001192 V	1,192 µV

You can easily calculate the resolution by yourself. First calculate the number of discrete steps:

$$DiscreteSteps = 2^{NumberOfBits}$$

Then divide your input range by the number of discrete steps:

$$InputResolution = \frac{InputRange}{DiscreteSteps}$$

or calculate directly:

$$InputReslution = \frac{InputRange}{2^{NumberOfBits}}$$

The more bits, the higher are the resolution. A high amplitude resolution is very important for enhanced mathematics, like FFT analysis. The higher the resolution, the better is the dynamic range - and the higher the costs!

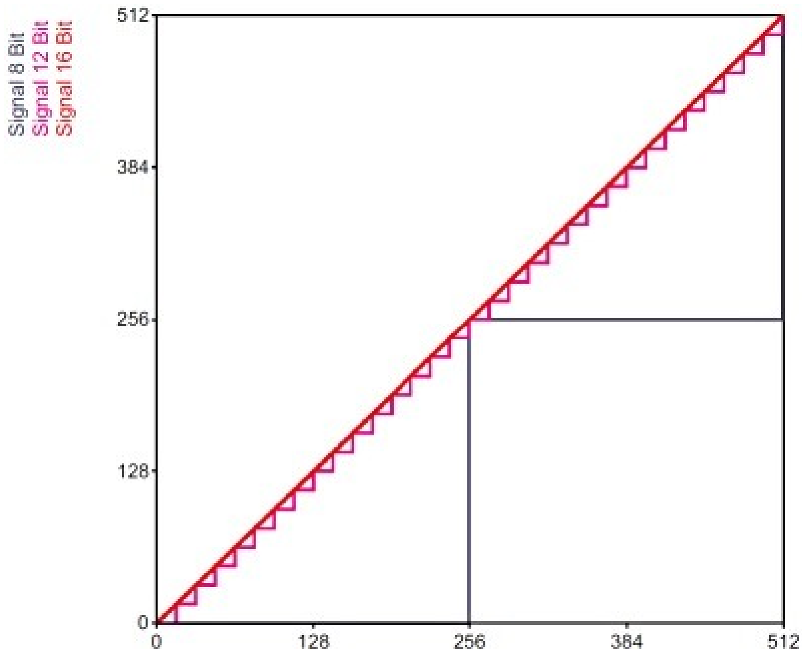
But high resolution has also other disadvantages: To get a stable signal, the A/D converter requires more time. The higher the resolution, the more time is required for the conversion:

Number of bits	Discrete steps	Minimum sampling time per channel using standard A/D boards		with special A/D boards	
8	256	-	-	2 ns	500 MS/s
12	4096	0.8 us	1.25 MS/s	10 ns	100 MS/s
14	16384	1,25 us	800 kS/s	-	-
16	65536	3 us	333 kS/s	1 us	1 MS/s
24	16777216	0.17 s	6 Hz	-	-

A/D boards you can do continuous measurements, streaming your data to hard disk, like you have done that before with analog recorders.

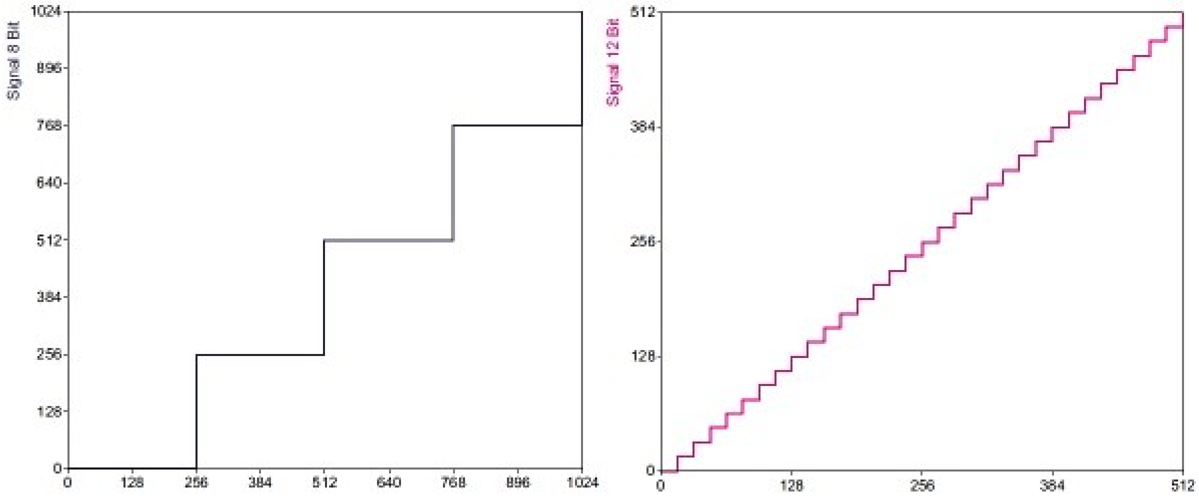
The special A/D boards are normally transient recorder boards with their own fast acquisition memory on-board. They are working ‘block oriented’, which means you *can’t do* a *continuous* measurement. The technology is the same which is used in modern digital storage oscilloscopes.

When we compare now the same input signal sampled with 8, 12 and 16 bits, the signal will look like that:

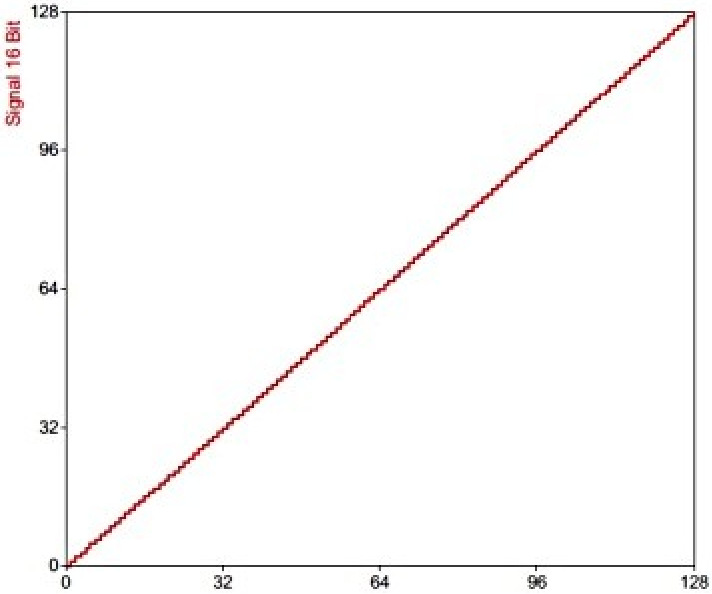


With three A/D boards (8, 12 and 16 bit resolution) set to 1 kS/s, we have acquired 512 ms of a signal, which is continuously growing with 1000 per second (ideal signal without units). While the 16 bit board has an own discrete amplitude value for each sampling point, the 12 bit board always shows discrete amplitude steps of 16. And the 8 bit shows the steps each 256.

The following three graphs show the amplitude resolution more detailed:



Signal with 12-bit resolution (right): 16 times higher resolution than 8 bit (left)



Signal with 16-bit resolution: 16 times higher resolution than 12 bit, 256 times higher resolution than 8 bit

1.3 Accuracy

Most users mix up **amplitude resolution** and **accuracy**, but there is a big difference:

- the **amplitude resolution** is given by the number of bits of the A/D converter and is the *smallest difference* between two values the system *can display*.
- the **accuracy** declares the *difference* in percent between the *nominal* and the *real value* of the *measurement system*

Each part of a measurement system has its *own* accuracy: the A/D board, the *amplifiers*, the *sensors*. Even the *cables* may have an influence to the system accuracy.

All these accuracies are not really interesting for the instrument user. It's just *important* to know the *system* accuracy - the *sum* of all parts. So it is very easy to calculate the accuracy: connect a well known and high precise signal to the measurement system and have a look on the result. The difference is the **error**, which can be *removed* by doing a system calibration directly in DEWESoft.

There are two different ways to mention the accuracy.

- Accuracy, related to the input *range*
- Accuracy, related to the input *signal*

Input range related accuracy

The most common way is to mention the accuracy related to the *input range*. The formula is simple:

$$Accuracy[\%] = \frac{InputSignal - ExpectedInputSignal}{FullScaleInputRange} \times 100$$

Three examples for a 10 V input range:

$$Accuracy = \frac{9.505V - 9.5V}{10V} \times 100 = \frac{0.005V}{10V} \times 100 = 0.05\%$$
$$Accuracy = \frac{4.505V - 4.5V}{10V} \times 100 = \frac{0.005V}{10V} \times 100 = 0.05\%$$
$$Accuracy = \frac{0.505V - 0.5V}{10V} \times 100 = \frac{0.005V}{10V} \times 100 = 0.05\%$$

In all three cases, the error is the **same**, because the absolute fault of 0.005 V in relation to the 10 V input range is always the same.

Input signal related accuracy

Sometimes the accuracy is related to the *input signal*. The formula is similar to the range related formula:

$$Accuracy[\%] = \frac{InputSignal - ExpectedInputSignal}{ExpectedInputRange} \times 100$$

The same three examples, again with 10 V input range, but related to the input signal:

$$Accuracy = \frac{9.505V - 9.5V}{9.5V} \times 100 = \frac{0.005V}{9.5V} \times 100 = 0.053\%$$
$$Accuracy = \frac{4.505V - 4.5V}{4.5V} \times 100 = \frac{0.005V}{4.5V} \times 100 = 0.11\%$$
$$Accuracy = \frac{0.505V - 0.5V}{0.5V} \times 100 = \frac{0.005V}{0.5V} \times 100 = 1\%$$

As the input *range* is <u>not used</u> in the formula, the input range itself has *no influence* to the accuracy. The absolute fault is still 0.005 V, but the lower input signal, the worse the accuracy in percent!

Resolution and accuracy

The *amplitude resolution* has to be higher than the *accuracy of the system*. As a standard, it should be 5 to 10 times *higher* to compensate the bit *noise* of the A/D converter and other faults - this is very important to get reproduce able results. The following table shows the required accuracy for the different A/D converter resolutions:

As the system accuracy is *normally* in ranges between 0.05 % and 3 % (depending on the system and sensors), the A/D resolution of each bit is much below the system accuracy. *Only* with *8-bit A/D* converters you can come into troubles.

Number of bits	Total number of discrete steps (= 100 % input range)	1 discrete step (related to 100 % input range)
8	256	0,39063%
12	4096	0,02441%
14	16384	0,00610%
16	65536	0,00153%
24	16777216	0,00001%

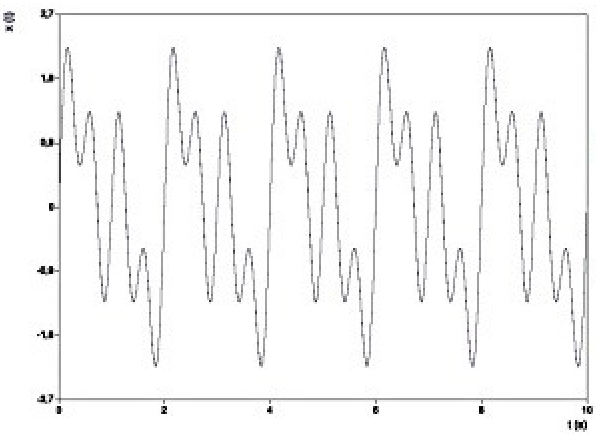
2 Theory of single channel frequency analysis

Frequency analysis is widely used tool because of its power to **reveal** the *real nature* of *dynamic signals*. Working for many years in practice doing *measurements* and lately also developing *FFT analyzers*, it became very obvious: knowing the theory of frequency analysis helps to reveal the problems we can anticipate and the ways to get around them. This article helps to understand the theory and answers some of the most asked questions regarding **FFT**.

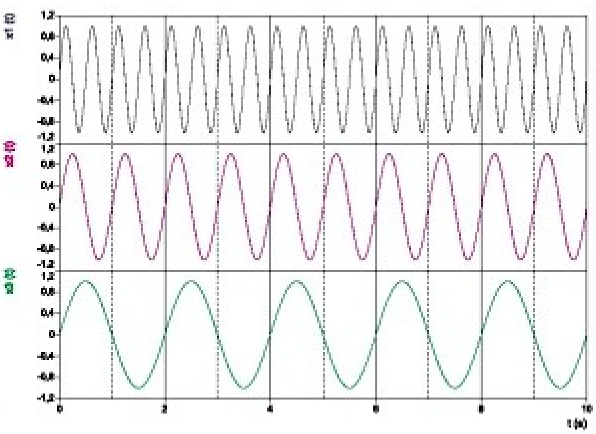
2.1 What is frequency analysis

Frequency analysis is just *another way* of looking at the same data. Instead of observing the data in the *time* domain, with some not very difficult, yet inventive mathematics frequency analysis *decomposes* time data in the ***series*** of ***sinus waves***.

Picture 1 shows the signal, which consist of three sine waves with the frequencies of 0.5, 1 and 2 Hz, and then the decomposed signal (*Picture* 2).

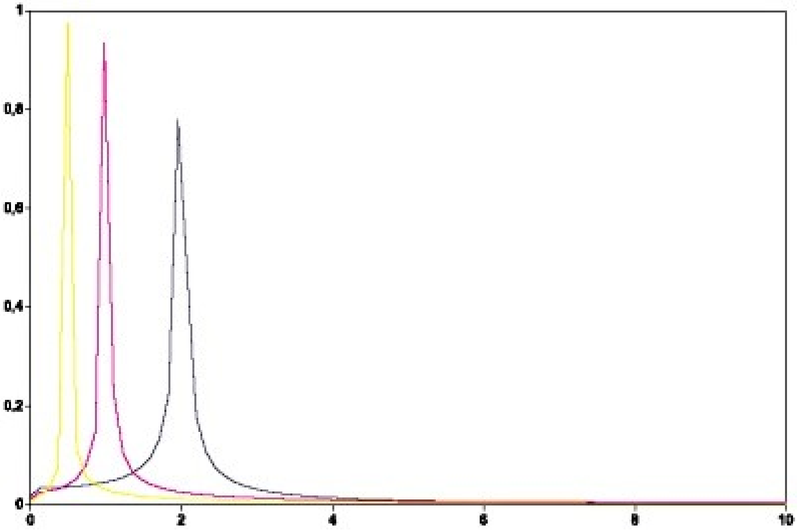


Picture 1: ORIGINAL SIGNAL



Picture 2: DECOMPOSED SIGNAL

Just to make those sine waves better visible, let's show them in a nicer way (*Picture* 3). On x-axis there are frequencies and on the y-axis there are amplitudes of the *sine waves*.



Picture 3: DECOMPOSED FREQUENCY DOMAIN DATA

And this is really what the frequency analysis is all about: showing the signal as the ***sum of sinus signals***. And the understanding how that works helps us to overcome problems that it brings with it.

2.2 Theory of frequency analysis

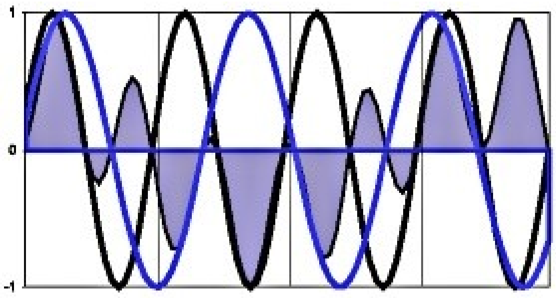
Frequency analysis is **converting** *time domain* data to the ***set*** of ***sinus waves***. So the theory says. But I like more if we say that frequency analysis *checks* the presence of *certain fixed* frequencies. Let's look at the equation of discrete Fourier's transform to figure out why:

$$X(k\omega_0) = \sum_{0}^{N-1} x[n] \cdot (\cos(2 \cdot \pi \cdot k \cdot n / N) + j \cdot \sin(2 \cdot \pi \cdot k \cdot n / N)) \quad (1)$$

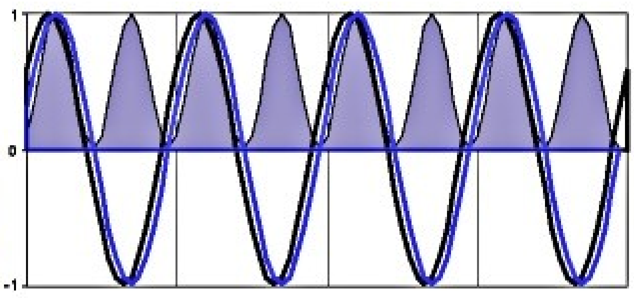
Let's think about how does it work. To check the presence of certain sine wave in a data sample, the equation does the following:

1) Multiplies the signal *with sine wave* of *that* frequency which we *want to extract*. *Pictures* 4 and 5 show the signal (black line), which consists only of sine wave with 50 Hz. We try to extract the 36 Hz on *Picture* 4 and 50 Hz on *Picture* 5 (they are shown as blue lines). Light blue filled wave shows multiplied values.
2) Multiplied values are *summed together* and this is the main trick. If there is a component in signal like in *Picture* 5 the multiplication of positive signal parts and extraction sine waves gives *positive* result. Also the multiplication of negative signal parts and negative extraction sine waves gives the *positive* results (observe *Picture* 5). In this case, the sum of the multiplied sine waves will be *nonzero* and will show the amplitude of the 50 Hz part of the signal. In the case of 36 Hz, there are both positive and negative sides of multiplication values and the sum will be (almost, as we will see further on) *zero*.

3) And that's it. That *sum gives* the ***estimate of presence of frequencies*** in the signal sample. We check sine and cosine to get also *phase shift* (in the worst case, if phase shift would be 90 deg, sum of sine functions would always give zero).



Picture 4: UNSUCCESSFUL EXTRACTION



Picture 5: SUCCESSFUL EXTRACTION

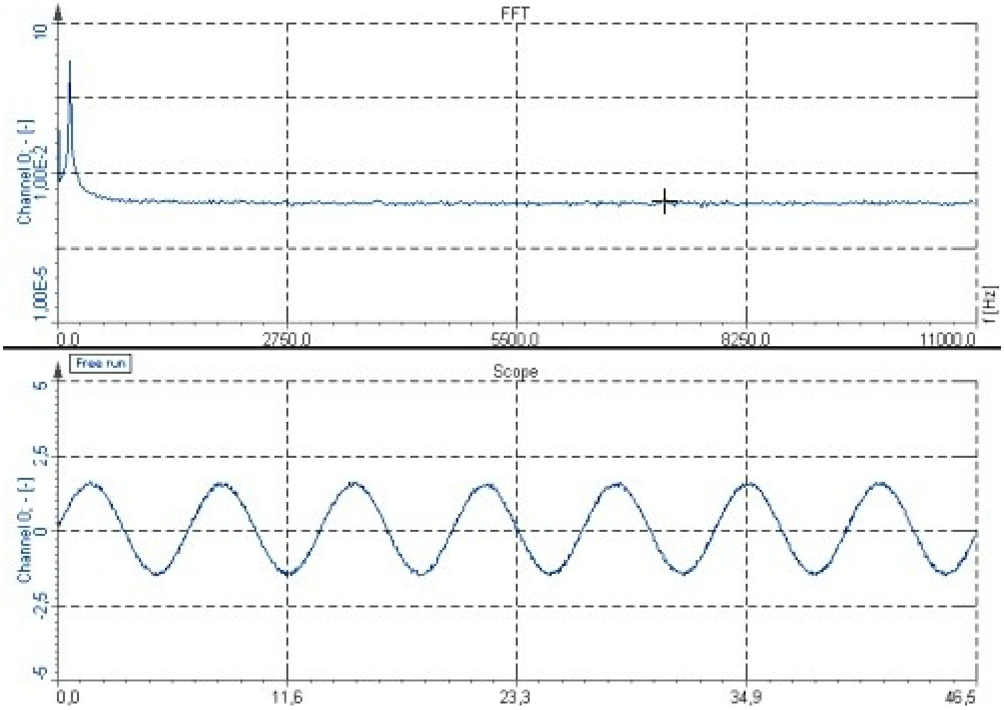
The principle shown above can extract basically any frequency from the sine wave, but it has one disadvantage. It is awfully *slow*. The next important step in the usage of DFT was the FFT algorithm. Because there is more than enough theory, let's just say that this analysis reduces amount of calculations by rearranging the data. The disadvantage is only that the data *samples must* be of *length*, which *is power of two* (like 256, 512, 1024 and so on). Apart from that, the result is practically the same as for the DFT.

2.3 Properties of the Fourier transform

Let's take a look on the typical **FFT** screen on *Picture* 6. The maximum frequency of the FFT is half of the ***signal sampling frequency*** (in this case the sample rate was 22000 samples/sec), but in the upper region the results are never reliable, so the sampling result should be set to:

(2) $$\boldsymbol{SampleRate = MaximumSignalFrequence \cdot 2 \cdot 1.25}$$

1.25 is absolute *minimum factor* for getting right values also in the upper region of the FFT. This is the equation other way around of famous Nyquist criteria, which says that maximal signal frequency adequately presented in digitized wave is the *half* of sampling rate.



Picture 6: TYPICAL **FFT** SCREEN

Result of FFT is a *set of amplitudes* of *certain frequencies*. **Number of lines** in the set is user selectable, but they *only* change the resolution of the FFT. **Line resolution** is a change in frequency between two frequency lines, which are extracted from the signal and is calculated with equation:

(3) $$LineResolution = \frac{SampleRate / 2}{NumberOfLines}$$

So the question is: why not always use maximum number of available frequency lines, which gives more exact results? The answer is simple: because with larger frequency lines it takes *more time* to calculate FFT.

(4) $$TimeToCalculate = \frac{NumberOfLines \cdot 2}{SampleRate}$$

Just for fun we can also combine the equations above and we get:

(5) $$LineResolution = \frac{1}{TimeToCalculate}$$

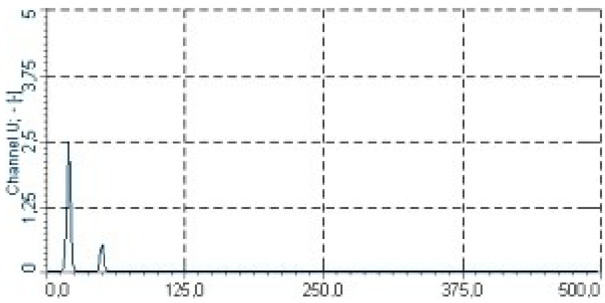
Let‘s look to the equations above and make a list for 22 kHz sample rate.

Number of lines	Line resolution	Calculation time
512	21,5 Hz	0,046 s
1024	10,75 Hz	0,093 s
4096	2,685 Hz	0,372 s
16384	0,67 Hz	1,49 s

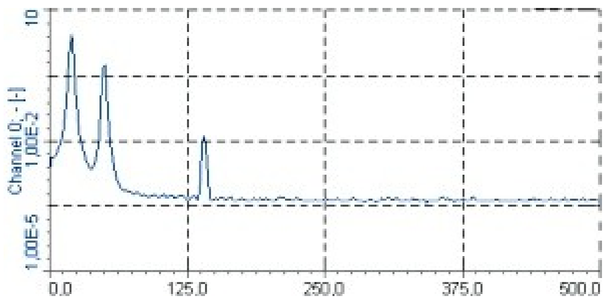
Table 1: EXAMPLE OF FFT SETTING

So the number of lines combined with sample rate also *defines the speed* of the FFT when non-stationary signals are applied. With *more* lines, FFT will appear *slower* and changes in signal will not be shown that *rapidly*.

Different ***amplitude scales*** of FFT can reveal more about the signal if used correctly. **Linear** amplitude scale gives the best view of *maximum peaks* in the signal, **logarithmic** can show more *invisible peaks* and signal *noise* but gives worse comparison of high and low peaks. Scale in dB gives the best estimation of signal *noise* if 0 dB is *maximum measurable* value and is also used in noise measurements, where the dB scaling is actually the result since the human ear has logarithmic sensitivity to noise.

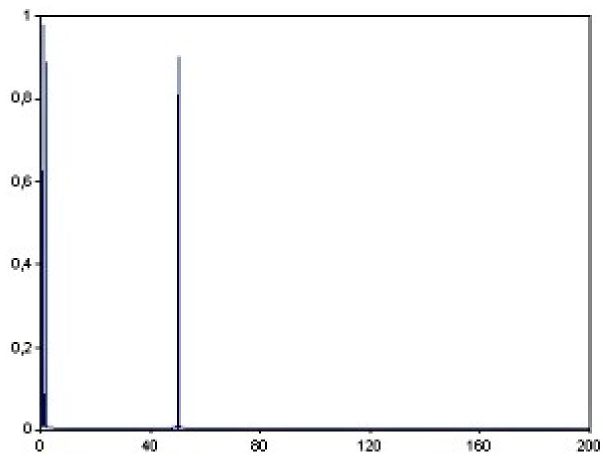


Picture 7: **LINEAR** *AMPLITUDE* SCALE

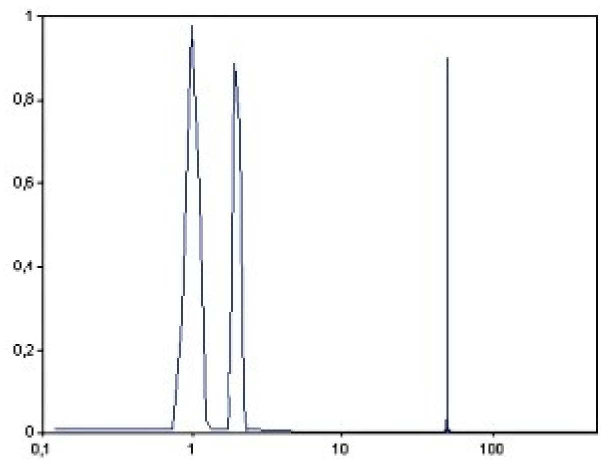


Picture 8: **LOGARITHMIC** *AMPLITUDE* SCALE

The Pictures 7 and 8 give good overview about the *differences* in linear and logarithmic scale. In linear scale (*picture* 7), noise and the peak at 140 Hz is not even visible. But on the other side peak at 140 Hz is 100 times lower than the other two and that difference is not very well visible in logarithmic scale (*picture* 8). In the real world, often there are lots of sine waves or spikes in the signal and linear amplitude display shows most relevant only.



Picture 9: **LINEAR** *FREQUENCY* SCALE



Picture 10: **LOGARITHMIC** *FREQUENCY* SCALE

X ***scale*** can be either *linear* or *logarithmic*. **Linear scaling** is the *correct representation* of the mathematic transformation and usually gives best information for analysis. Sometimes like in the example shown in *picture* 9 it is nice to see x-axis in **logarithmic** values since most interesting frequencies are in a *lower* region. We have to know that just to set the x scale to logarithmic does *not enhance* the results in the *lower* region, so the resolution will be better in the upper region, since there are more frequency lines available there.

If we use another technique, called **CPB** (for *constant percentage bandwidth*) or **octave analysis**, this will give us *same* resolution in *all regions* when x-axis is **logarithmic**. This is achieved by the fact that *upper* region lines *covers wider frequency* ranges than the lower one.

The *resolution of the bands* is defined by ***1/n*** description, where ***n*** is the number of bands in *one* octave. Most widely used is 1/3 octave analysis, which is the standard for noise measurements. 1/12 and even better 1/24 octave analysis already gives *good resolution* also for signal analysis.

2.4 Where we can get in trouble

Because there should be at least one equation in each chapter, let's use the following one:

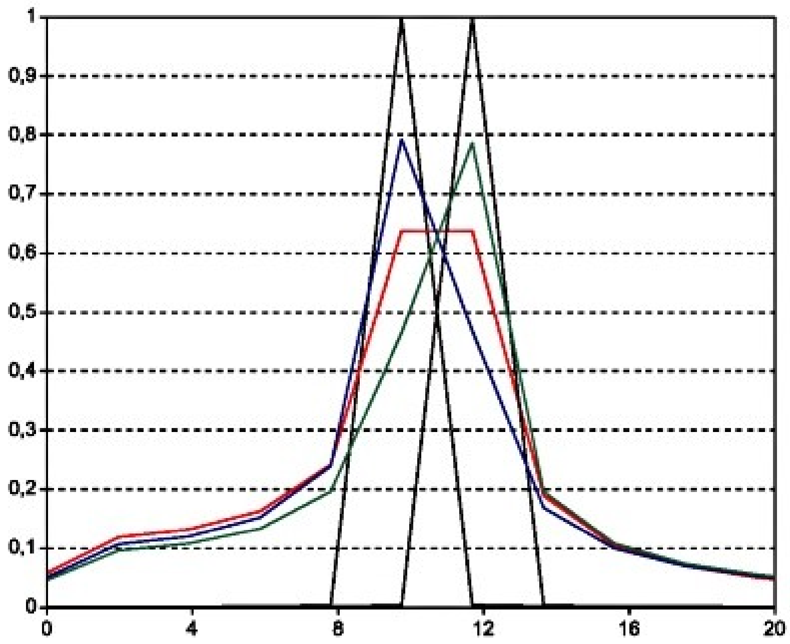
$$X(j\omega) = \sum_{n\to-\infty}^{n\to\infty} X[n] \cdot e^{-j\omega nT} \tag{6}$$

This is the *theoretical discrete Fourier transformation* and it has absolutely no error. The only thing, which could be bothering, is the sum goes from minus infinity to plus infinity. Because in the fast world we live in we don‘t have the time to wait that long, we run in problems.

1. Amplitude error (picket - fence effect)

The first problem is shown already in theory chapter. The sum can produce 'not null' results even if the signal does not correspond to the frequencies extracted from the signal. The pure frequencies are because of that somehow 'leaked' over neighbor frequencies. For the same reason if the frequency does not exactly fall on frequency line, amplitudes seem to be lower. This is called 'picket fence' effect.

Let‘s look at the *Picture* 11 if a frequency is exactly on the spectrum line and some examples in between. On the picture below 10 and 12 Hz are the exact frequency lines. On the example there are 10 and 12 sine waves marked as black, which are transformed correctly, and there are frequencies in between which has lower amplitudes. Maximum amplitude error can go up to 35% of the correct value.



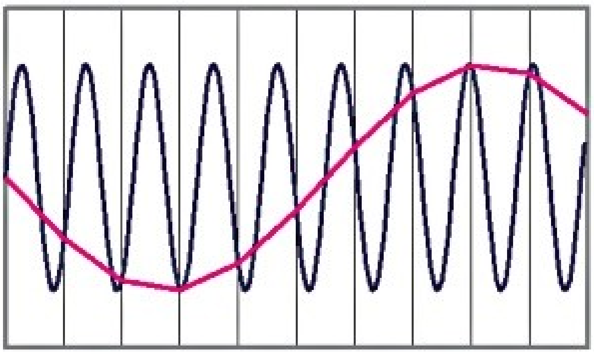
Picture 11: AMPLITUDE ERROR AND LEAKAGE OF THE FFT (NO WINDOW)

2. Leakage

Also we observe that especially if the sine wave is not on the frequency line, we get high amplitude values on both sides of the main band. The amplitudes are really high (with no window, it is about 10% of the original values for about 10 neighbor lines). If there is another sine wave in the signal in this region, which is lower than this 10%, it will be completely hidden by the leakage effect.

3. Aliasing

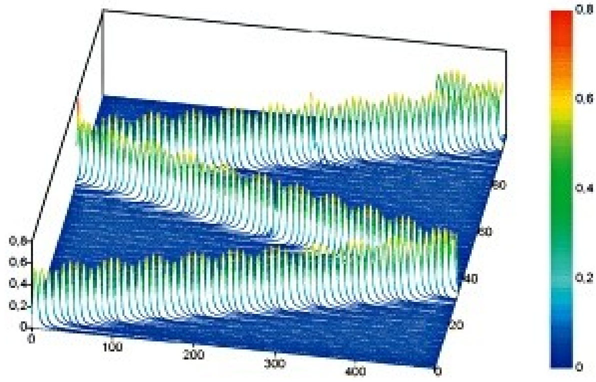
The other problem comes from the fact of the signal conditioning. If *simple* A/D converters are used, the sampling frequency must be at least twice higher than the maximum frequency of the signal. This effect is called aliasing effect. *Picture* 12 shows the reason for it. Vertical lines represent samples taken with A/D converter and the blue line is the original signal. But if we look at the red line, which is the signal form A/D converter, the signal is totally wrong because too fewer samples per period were taken to correctly represent the signal.



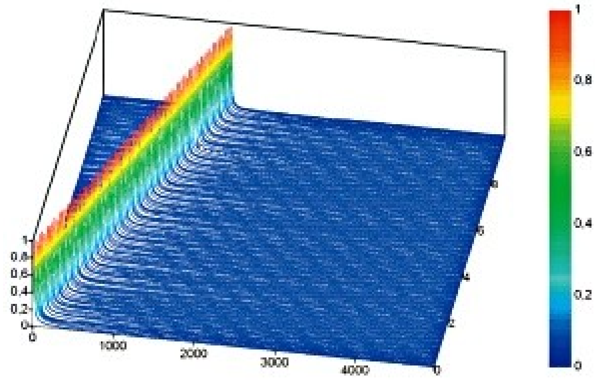
Picture 12: ALIASING

Of course the problem above is not FFT problem, but it is very important to know to correctly identify the cause for the error. And sometimes there are some lines in FFT, which can be only explained in terms of aliases. In FFT, if we change the frequency to the ranges above the maximum frequency limit, that line will not disappear but will bounce back and will show the fake frequency.

To really see that effect, it is the best if function generator and simple A/D card is used and the online FFT analyzer perfectly shows the problem. Since this is not possible in the written article, we will use waterfall diagrams to see the problem. Both *Picture* 13 and 14 show the sine sweep from zero to 1.5 kHz. On *Picture* 13, the signal is sampled with 1 kHz and on *Picture* 14 the signal is sampled with 10 kHz. On picture 14 we see right result, but on the left we can clearly see that the signal above 500 Hz bounces back zero and then over 1 kHz is starts to rise again, but the truth is all that frequencies are not correct.



Picture 13: ALIASING IN THE FFT



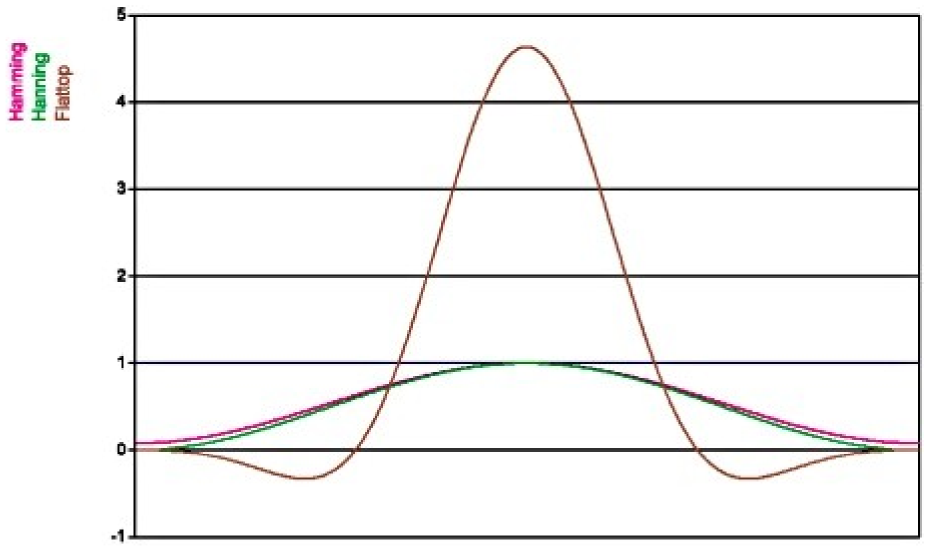
Picture 14: CORRECTLY SAMPLED SIGNAL

2.5 How can we at least try to avoid it

For the problem of ***aliasing***, there is not much to do in the FFT domain. Actually, there is absolutely nothing we can do when the samples are already taken. So the first way would be to choose the A/D board who has *anti-aliasing filters* in front, the second way would be to use *external filters* or we can simply set the sampling rate to more than *twice the maximum frequency* present in the signal (according to equation 2).

For ***amplitude errors*** a bunch of people tried to minimize that problem. Those were Hamming, Hanning, Blackmann, Harris and others. They have created assortment of *functions*, which tries to correct the errors. **Window functions** are *multiplied* with *original* time signal and because they are usually 0 at beginning and the end, sine waves could be also in-between lines or phase-shifted and they will be *less leaked* over *neighbor frequencies*.

Picture 15 shows some of these functions in the *time* domain.



Picture 15: WINDOWS IN THE TIME DOMAIN

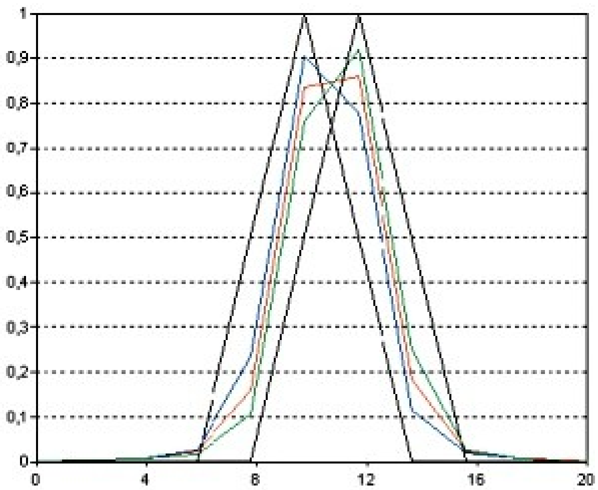
And here is the most common question to FFT: what are the differences between windows and when to use certain windows?

The rule of thumb is when we want a ***pure transformation*** with *no window's side effects* (for advanced calculations), we should use Rectangular window (which is by the way *equal to no window*).

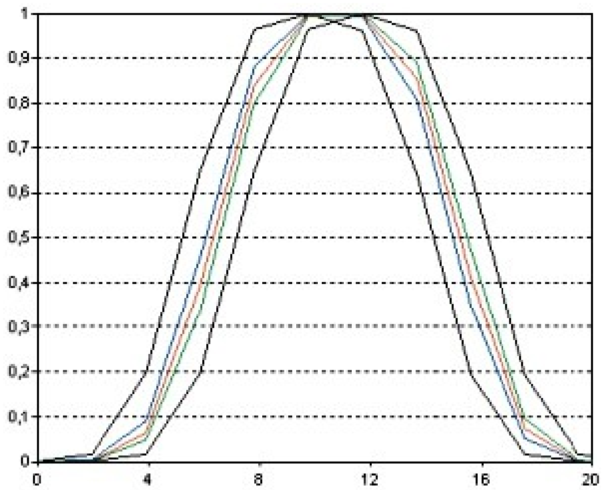
For ***general*** purpose, Hanning or Hamming are commonly used because they provide a good compromise between fall-off and amplitude error (maximum of 15%). This comes from the fact that old frequency analyzers didn't have that many possibilities in terms of frequency lines and these two windows have narrow sideband. But still if 12 bit AD cards are used; Hanning or Hamming is a good choice.

When more ***dynamic range*** is necessary (we want to see *very small* signal *among large* ones), Blackmann or Kaiser window is better choice, because sidebands are 10 times *lower* than with the Hanning window. However the sideband width is *wider*. Here it comes to the point - if *more lines* in FFT are chosen, we can use these windows and *still larger sideband's* have *no real* disadvantage.

If ***correct amplitudes*** are searched, we should use the flat-top window. The amplitudes would be wrong by *only a fraction* (as low as 1%). Of course there is a penalty - *neighbor frequencies* are also *very high* (sideband width is *high*). This window is most suitable for calibration. But here it is the same: with modern equipment with lots of lines this is no longer that much of a problem.

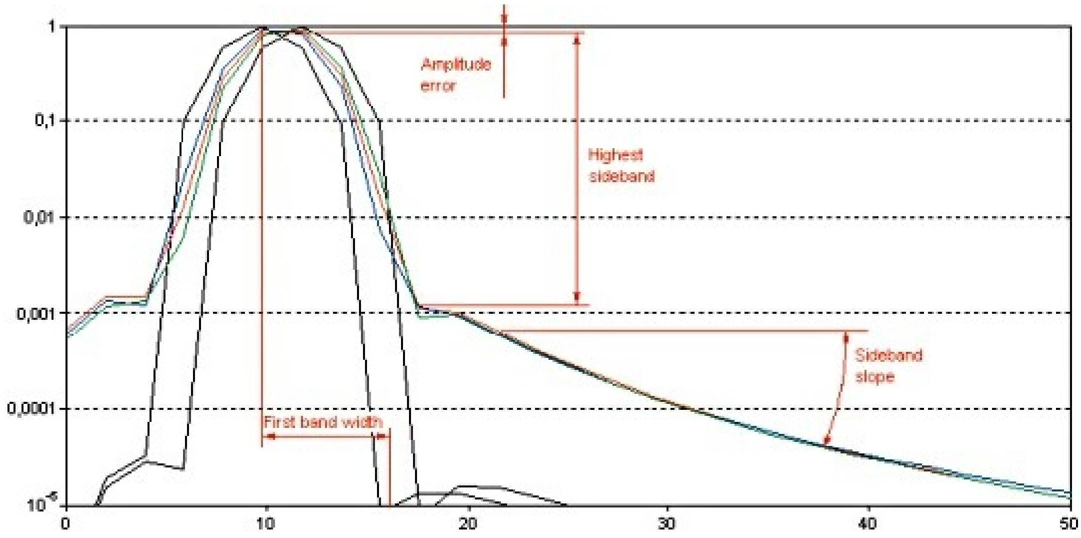


Picture 16: HANNING WINDOW



Picture 17: FLAT TOP WINDOW

Window characteristics (maximum *amplitude error*, *sideband width*, *highest sideband attenuation*, *sideband slope attenuation*) are best described on *Picture* 18. We have already discussed about maximum amplitude error: it is an error of amplitude if sine waves does not fall on frequency line. Windows try to eliminate this problem and because of that they widen the first band. The sine waves are not longer one line in FFT, but spread along several lines. Ability to recognize small sine waves among larger ones is determined by highest sideband attenuation and sideband slope attenuation. These two values determine the leakage of the FFT and that's nicely seen on *Picture* 18. If for example there would be a frequency of 30 Hz with amplitude of 0.0001, we would never see it because the 10.5 Hz frequency leakage is bigger than requested frequency. But for example if rectangular window is used, we would never see even signal with amplitude of 0.01.



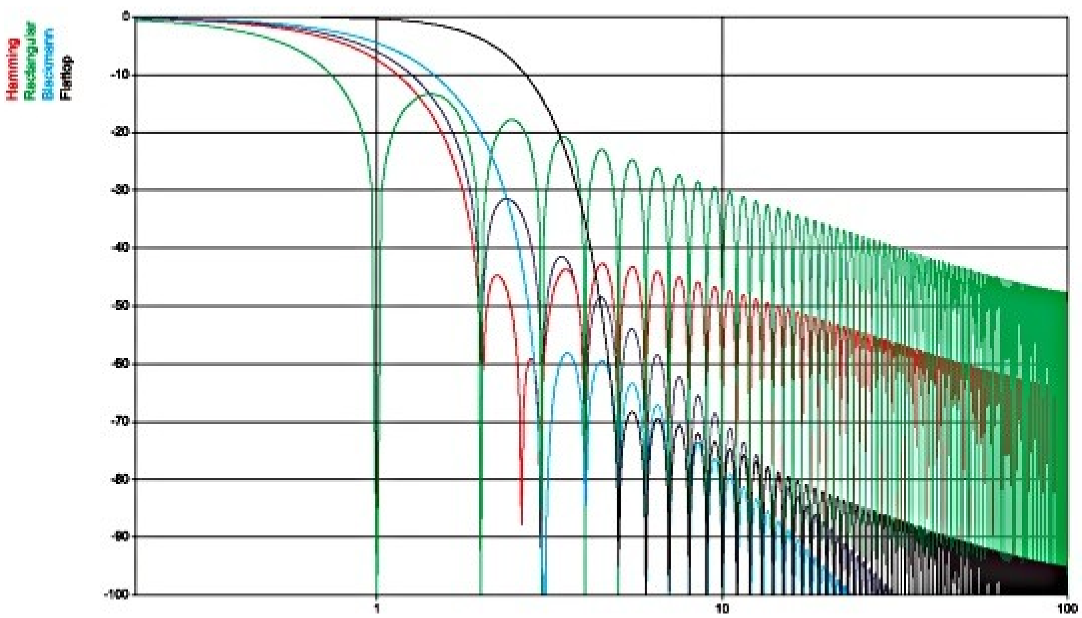
Picture 18: WINDOW PROPERTIES DESCRIPTION

For different kind of windows, *Table* 2 shows the values of all window properties. This is a numerical representation of above-mentioned rules.

Window type	Maximum amplitude error	Width of first band	Highest sideband	Sideband slope
Rectangular	36%	1 line	22%	-20 db/decade
Hanning	15%	2 lines	2,50%	-60 db/decade
Hamming	18%	2 lines	0,70%	-20 db/decade
Blackmann	12%	3 lines	0,12%	-40 db/decade
Flat top	0,20%	5 lines	0,04%	-20 db/decade

Table 2: WINDOW PROPERTIES

Picture 19 shows zoomed FFT of the pure sine wave, which fits to the exact frequency line. Abscissa axis shows the lines value. In normal FFT, only values of the 0, 1, 2, etc are calculated, so only those values are shown in the FFT. We can see width of the first sideband, highest sideband and sideband attenuation very clearly.



Picture 19: WINDOW PROPERTIES

If a signal sine wave frequency falls between two lines, we see only the values of 0.5, 1.5, 2.5..., which always produces higher sidebands. This is best seen if we take the function generator, set the frequency to exact frequency line, set the amplitude scaling to logarithmic and the FFT will look fantastic. No leakage, exact amplitude. Now switch the frequency from function generator to the one between two lines in the FFT and the result will be just terrible: large amplitude errors, huge leakage.

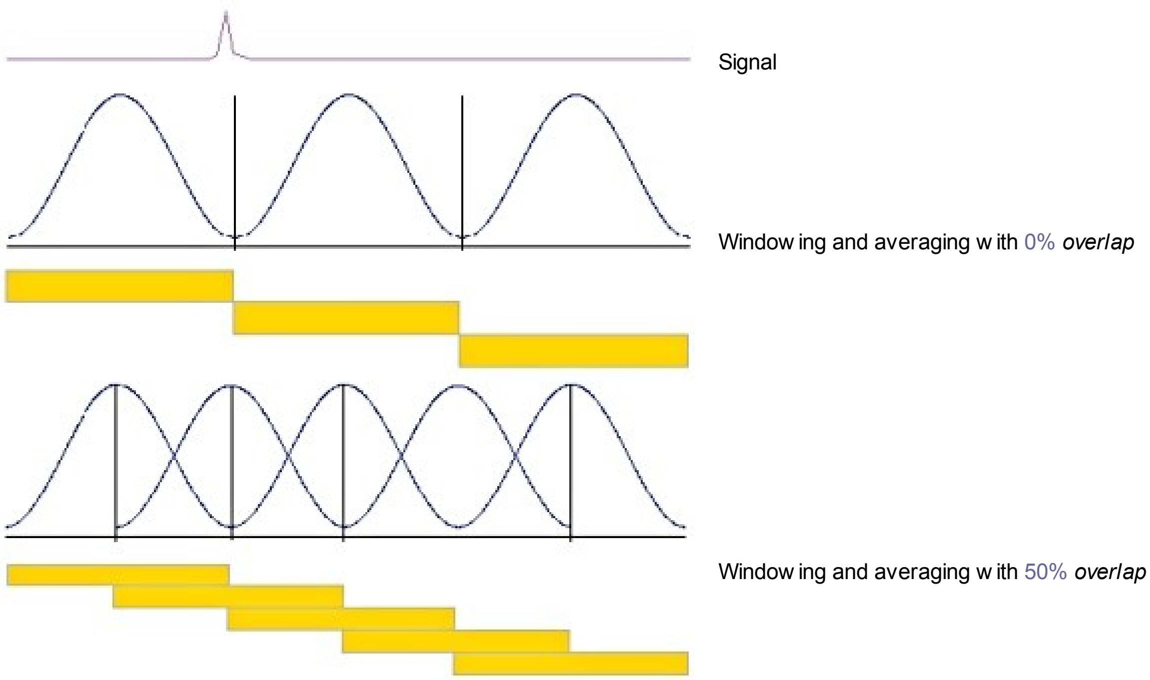
There is one more trick with windows: if we are sure that *all the frequencies* will *fall on its frequency lines*, rectangular window will give us the best result. For example to measure the *harmonics* of the *power line* (50 Hz in Europe or higher), choose 6400 or 9600 sample/sec sampling rate, that the line resolution will give exact 50, 100, 150 Hz... FFT lines, choose rectangular window and observe the *perfect result* in the y log scale.

2.6 Averaging - how to enhance the results

To enhance the result, we can use **averaging** of the signal in the *frequency domain*. There are many ways to average the signal, but the most important are: linear, exponential and peak hold average.

Averaging means that we *calculate many* FFTs during the time and *averaging the frequency lines*. We can do linear averaging (*each* FFT counts the *same* in the results), exponential (FFTs becomes *less and less important* with time) and the peak hold (*only maximum results* are stored and shown).

There is one more thing about the averaging: loss of information. When averaging is used with window functions, we could *loose some data* due to the window multiplication effects.



Picture 20: OVERLAPPING

In *Picture* 20 there is one example where the signal only consists of one pulse. If we average the result, use window function and we are unlucky, signal will fall in region where the window sets the values to zero and in resulting FFT we will never see this pulse.

That's why there is a procedure called **overlapping** which overcomes this problem. It no longer calculates averages one after another, but takes *some part* of the time signal, which is already calculated and *uses it again* for calculation. There could be any number for overlap, but usually there is 25 %, 50%, 66.7% and 75% overlapping.
50% overlapping means that the calculation will take *half of the old* data. Now all data will be *for sure shown* in the resulting FFT.
With 66.7% and higher overlapping, *every* sample in the time domain *will count exactly the same* in the frequency domain, so if it's possible, we should use this value for overlapping to get mathematical correct results.

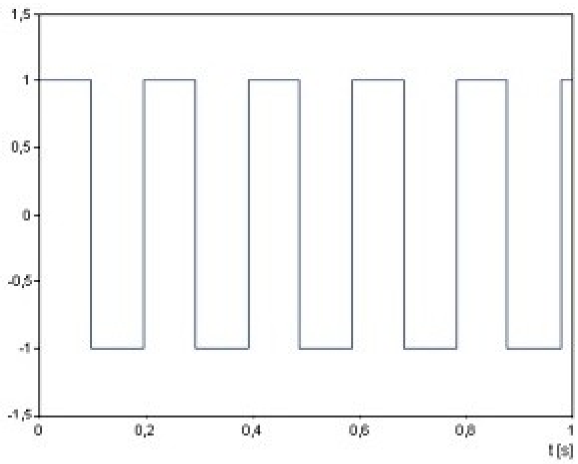
By the way, do you know what 'real-time' frequency analyzer means? It means that it is able to calculate and show data with 66.7% (or 50 % in some literature) *overlapping* and therefore *has no data loss*.

2.7 Representation of different signals in the FFT

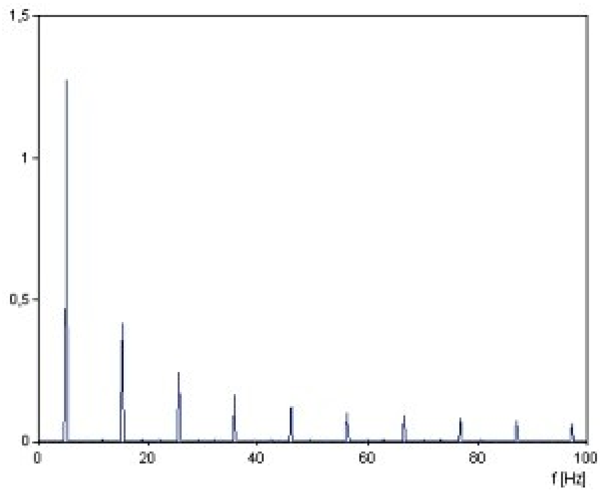
For now, we have only seen how the sine waves looks like in the FFT, but let's observe how *other types* of the signals look like:

1. Triangle, rectangular

All signals that are *periodic* with time but are *not pure sine* waves, produces base harmonic component as well as *additional* higher harmonics. More the signal is not like sinus, *higher* the harmonics are.



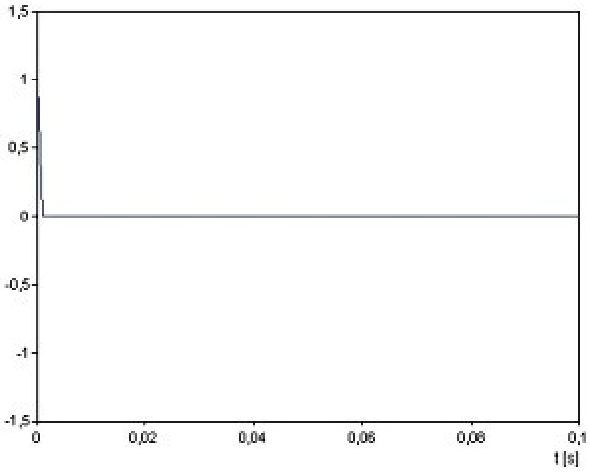
Picture 21: TIME DOMAIN SQUARE WAVE



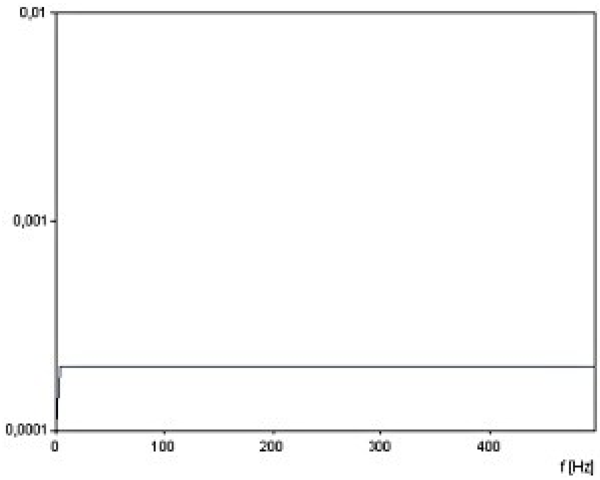
Picture 22: FREQUENCY DOMAIN SQUARE WAVE

2. Impulse

Impulse is quite interesting thing. It cannot be described with sum of sine waves. Or in different words: it is shown *equally on all* of the frequency lines. That's the reason why we use it as the basic excitation principle to get frequency responses of the system. The other ones are swept sine and noise, but this is already a part of another story - dual channel frequency analysis.



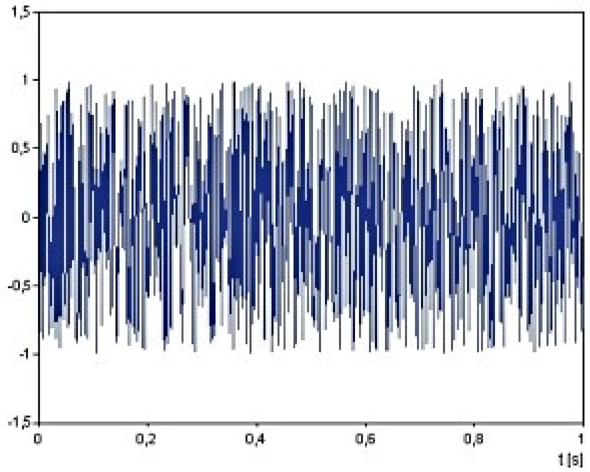
Picture 23: TIME DOMAIN IMPULSE



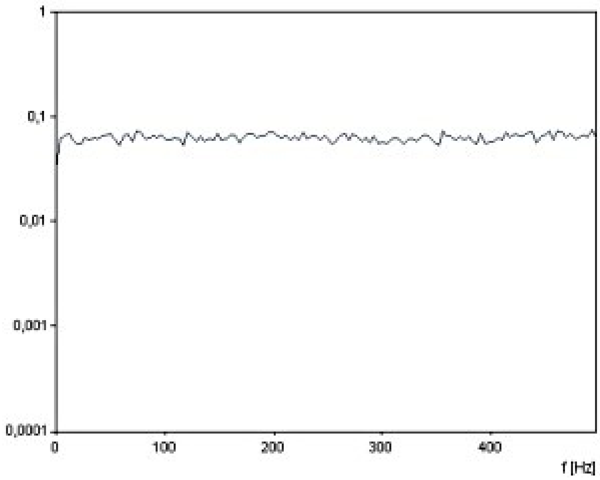
Picture 24: FREQUENCY DOMAIN IMPULSE

3. White noise

The theory says that white noise *consists of all frequencies*. That's why the infinite frequency spectrum of the white noise is the straight line. However the *shorter the samples* are, *more different amplitudes* for *certain frequencies* we get in the noise level. It is the best to use averaging to get fixed noise line. *Picture* 24 shows already averaged FFT of the white noise.



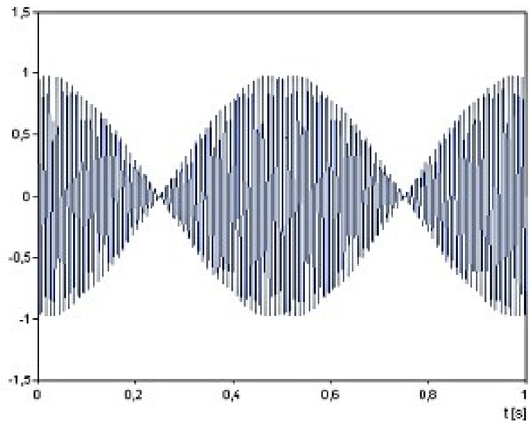
Picture 25: TIME DOMAIN NOISE



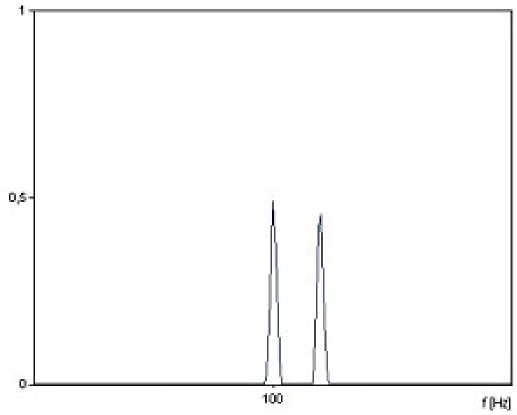
Picture 26: FREQUENCY DOMAIN NOISE

4. Beating (two closely spaced signals)

Beating in time domain is somehow hidden and looks like *one frequency with changing amplitudes*. Only FFT reveals *two frequency lines*, if we choose high enough line resolution. The *difference* between two frequencies is the *modulation frequency* shown in the time domain.



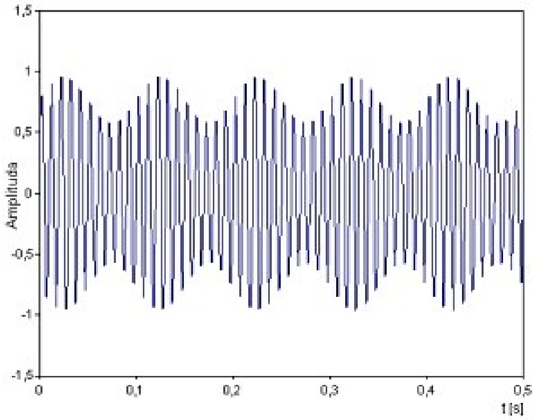
Picture 27: BEATING IN TIME DOMAIN



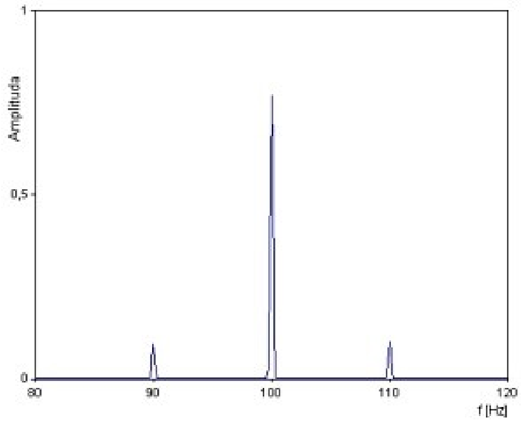
Picture 28: BEATING IN FREQUENCY DOMAIN

5. Amplitude modulated signal

Amplitude modulated (AM) signal is shown as *two sideband frequencies*. *Difference* between the *base* frequency and the *sideband* frequency is the modulated frequency (10 Hz in this case) also seen clearly in the *time domain*. The rule here is the same like with beating - to reveal the modulation; we should choose high enough line resolution. In fact, the time signal, which is the base for the FFT calculation, should show some modulation peaks. When windowing is used (we know that the base band could be even 4 lines wide) and the main band, which is always the highest, covers the modulation with low line resolution, time signal should show at least 16 or 32 modulation peaks that the modulation is shown in the FFT.



Picture 29: TIME DOMAIN AM SIGNAL



Picture 30: FREQUENCY DOMAIN AM SIGNAL

2.8 Conclusion

There's no doubt about it - FFT is a powerful analysis. But it is just like a car. If we know its limits, we can get so much more performance from it. That's why it is important to know how it is calculated and the tricks that might come handy.
The next step in one channel FFT analysis is the real signal analysis, which comes from the knowledge of the measurement. If it's *machine vibration*, we have to know the structure and the properties of mechanical elements and structures. If it is an *electronic circuit*, we have to know the properties of electronic components to figure out the responses.

With this step the FFT analysis steps out of being a mathematical principle and becomes just a handy tool, like it really should be.

3 H1, H2 and HV algorithms for calculation

H1 algorithm

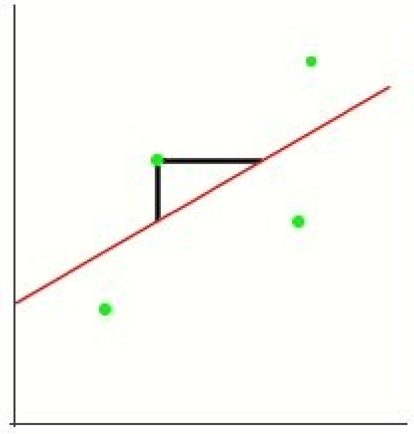
H1 algorithm expects **noise** on output.

H1 algorithm tends to *minimize the noise* on *output*. In the presence of noise it underestimates amplitude at resonances and anti-resonances.

H2 algorithm

H2 algorithm expects **noise** on input.

H2 algorithm tends to *minimize the noise* on *input*. In the presence of noise it underestimates amplitude at resonances and anti-resonances.

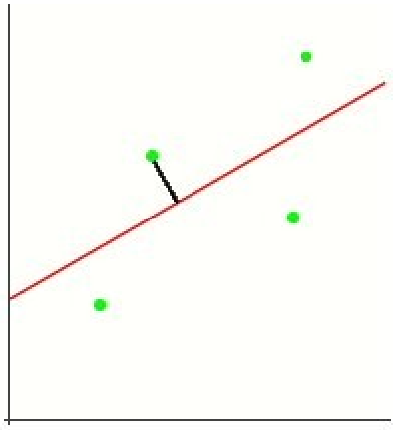


Vertical offset - leas squares (minimizes distance in x or y direction).

Hv algorithm

Hv algorithm expects **noise** on input and output.

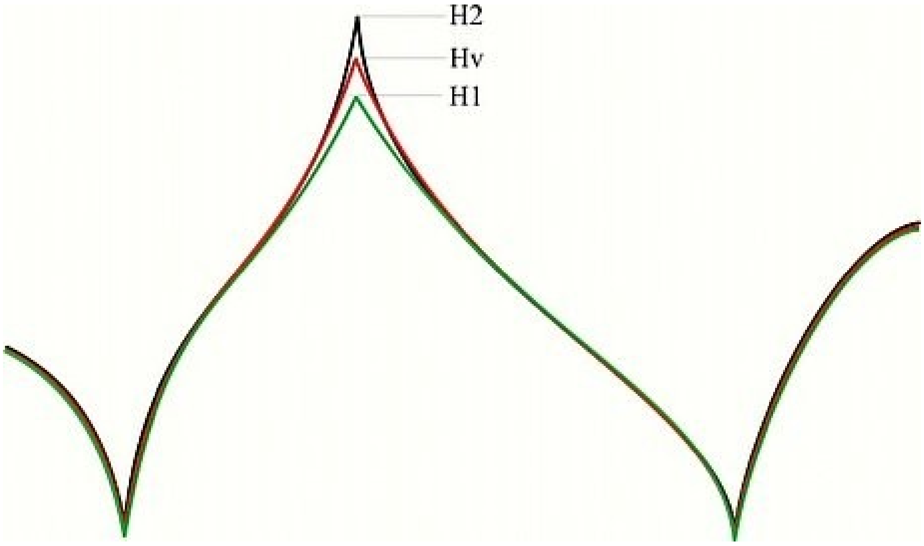
Hv algorithm tends to *minimize the noise* on *input and output*. It is always bounded by H1 and H2. The only cut down is that because of it's complexity it uses a lot of computing power.



Perpendicular offset - total leas squares (minimize distance in x *and* y direction).

Comparison of given method

If there is *no* noise present *all three* algorithms give the *same* results. That is why everything that is possible should be done in order to eliminate noise on input and output.



Index