



DEWESoft v6.5

Data acquisition, processing, analyzing and storage software

User manual

v1.1.0

General information

Software version

DEWESoft Tutorials corresponds with software version 6.5.

Printing notice

Specifications subject to change without notice.

Copyright notice

© 2007 DEWETRON elektronische Messgeraete Ges.m.b.H. All rights reserved. May not be duplicated or disseminated in any fashion without the express written permission of DEWETRON Ges.m.b.H.

Trademark notice

All trademarks are acknowledged to be the property of their owners, with all rights and privileges thereto. No infringement 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

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

Table of Contents

Part I	User Guide	2
Measurement setup		3
Analog in		5
Channel list		6
Input channel - Slot		7
Activating channel - On/Off		7
Channel color - C		8
Channel name - Name		9
Amplifier type and ranges, sensors		9
Physical values		11
Input zeroing		11
Channel setup form		12
General settings		14
Sensors setting		15
Amplifier settings		17
Scaling		19
Calibrate		22
Live data preview		24
Analog out		27
Operation modes		28
Control Options		32
Channel value		32
Channel setup		33
Waveform type		35
Arbitrary - additional settings		36
Noise - additional settings		37
Setting during Measurement		38
Counters		41
Event, gated and up/down counting		44
Encoder		46
Period and pulsewidth		48

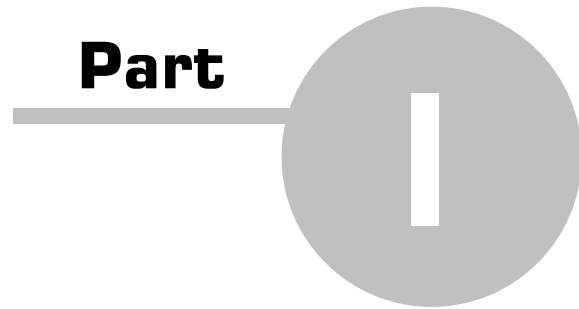
Two pulse edge separation	49
Frequency	50
Digital input	51
CAN bus	53
Channel setup	56
GPS	63
Video	65
Power module	70
General mathematics modules	86
Setup screen and basic operation	88
Formula editor	92
Operators and Mathematical Function	95
Example of logic functions	100
Example of input events	101
Example of signals	102
Example of measure functions	103
Example of stopwatch	104
Filters	105
IIR Filter setup	106
Standard filter	107
Integration and double integration	110
Derivation and double derivation	112
Custom filter	113
Custom filter import from Matlab	116
FIR Filter setup	117
FFT Filter setup	121
Statistic group	125
Basic statistics	126
Latch math	129
Reference curves	130
Reference curve	131
XY reference curve	134
FFT reference curve	135
Dynamic signal analysis section	137
Exact frequency	137
CA noise	139
Angle sensor	141

Dynamic signal analysis	143
Sound level	144
Torsional vibration	152
Human vibration	162
Order tracking	171
Combustion analysis	176
Trigger setup	187
Trigger Condition setup	191
Sensor database	199
Counter sensor editor	207
Alarm Monitoring	212
NET acquisition	218
 Measure - Data acquisition	 223
Displays design	224
Pre-defined displays	225
Display settings	227
Channels selector	229
Displays menu functions	234
Screen edit functions	235
Instrument appearance	237
Visual control settings	240
Group properties	240
Common graph settings	242
Digital meter	248
Horizontal / Vertical bar graph	250
Analog meter	251
Indicator lamp	253
Recorder	258
Vertical recorder	261
X-Y recorder	264
Scope	268
Setting - display tools	270
FFT Analysis	280
Octave Analysis	286
3D FFT	289

Orbit plot	292
Vector scope	295
Harmonics analysis	298
Video display	301
GPS	303
Overload indicator	307
Tabular display	309
CA p-v	312
CA scope	314
Additional controls	315
Recording Acquired Data	320
Basic Recording Setup	321
File Name Setup	326
Data Header Design	330
Freeze mode	338
Start and stop recording	339
NET acquisition data	344
Measurement	347
Analyse	352
Reviewing data files	353
Loading data files	353
Displaying data	357
Selecting data	360
Selecting triggered data	364
Replaying data	367
Working with events	368
Storing settings and events	370
Video post synchronization	371
Publishing data	374
Screen printout	374
Copy channel setup	377
Copy visual elements	378
Exporting data	379
to FlexPro and MS Excel	386
MergeDB - FlexPro	390

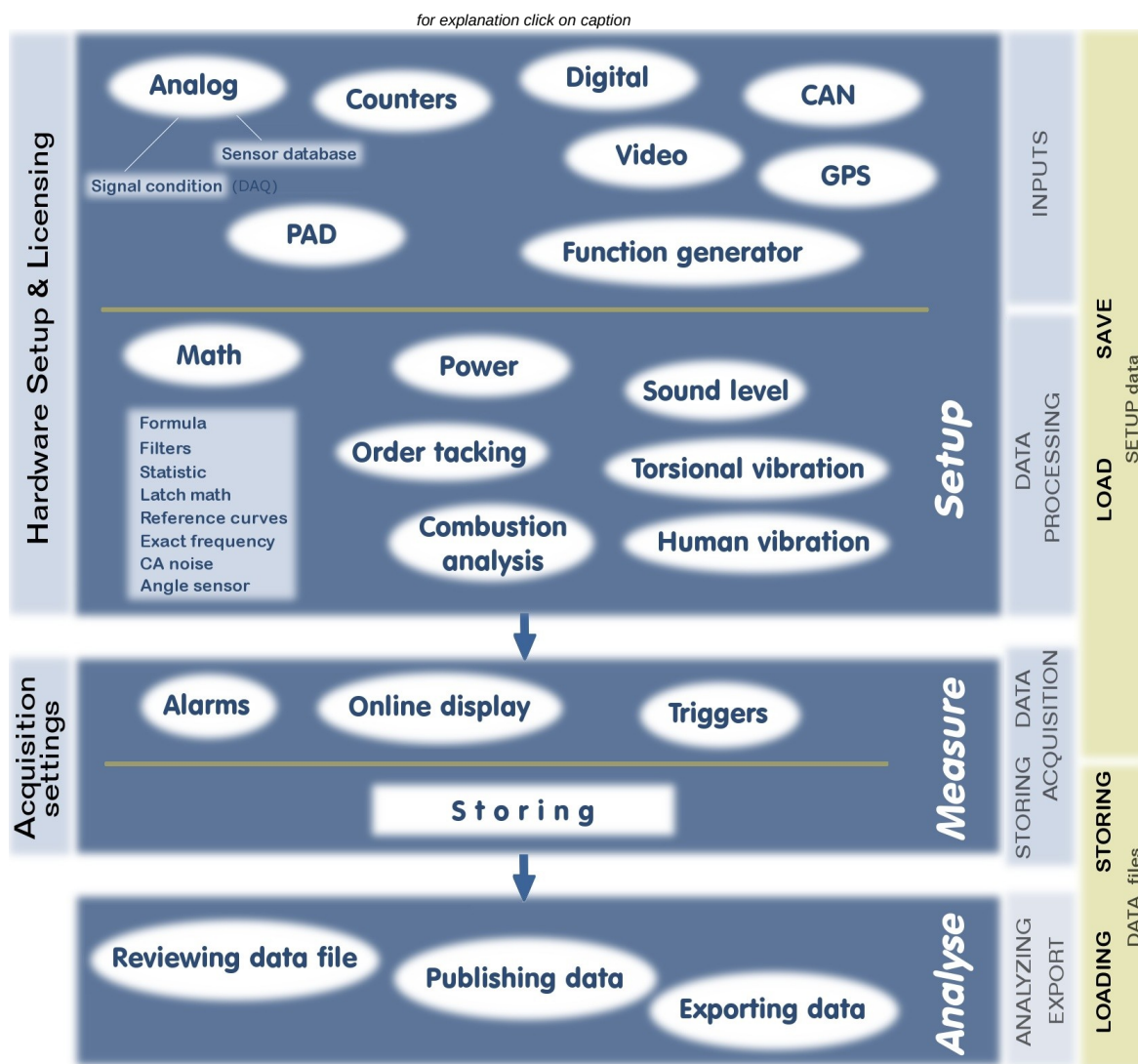
Export multiple files	392
Export instrument display to Video	393
Data management	396
Setup data	396
Structure of Text Setup file	401
Settings and Events	405
 Part II Reference Guide	 407
 Basics of A/D conversion	 408
Sampling rate	408
Amplitude resolution	410
Accuracy	413
 Theory of single channel frequency analysis	 415
What is frequency analysis	415
Theory of frequency analysis	416
Properties of the Fourier transform	417
Where we can get in trouble	420
How can we at least try to avoid it	422
Averaging - how to enhance the results	425
Representation of different signals in the FFT	426
Conclusion	428
 H1, H2 and HV algorithms for calculation	 429
 FlexPro Scripts	 431
 Index	 437

Part



User Guide

This help manual should give the user an overview on the **DEWESoft 6.5** and its **Application modules** functionality and its configuration possibilities in combination with various *measurement units*. It covers general operating system settings as well as the **DEWESoft** configurations for various *measurement topologies*.



For *background* information and *additional* information about supporting **DEWESoft 6.5** Application:

see → [Reference Guide](#)

1 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 the **Setup** screen will *automatically* appear or press direct **Setup** button on **DEWESoft** tool bar (in the **Analyze** mode this button have different function: display *Settings*, *Events* and *Data header* information → see → **Analyse mode**).



In upper part of the **DEWESoft Setup** screen we can:

define and set **Basic recording data settings**, which will be used for *data storage*, like:

- **DATA FILE OPTION**
- **STATIC/REDUCED RATE**
- **DYNAMIC ACQUISITION RATE**
- **STORING OPTIONS**

and with 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

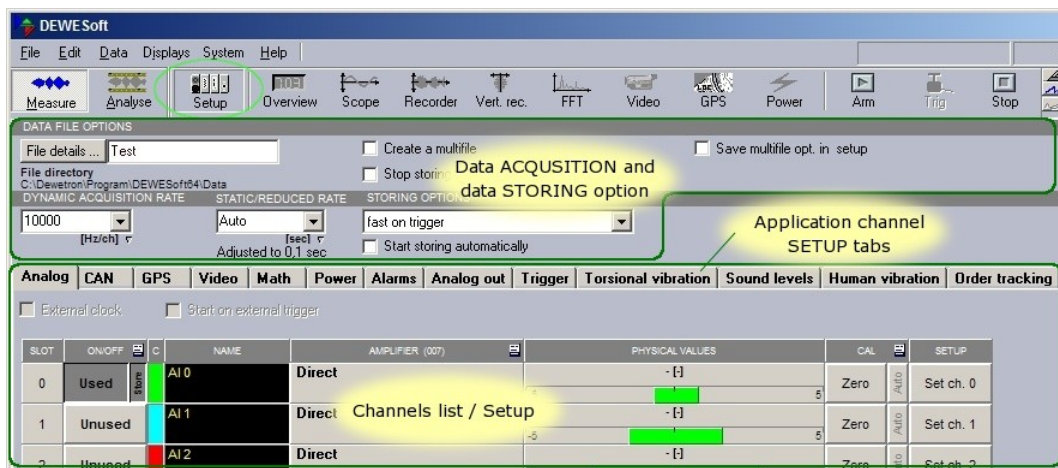
Supplementary functions

- **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
- **Trigger** setup setting any *channel(s)* to *start* and *stop* recording according to levels

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

set **Common Channel Settings** like: *slot, on/off, color, name.*



Displayed tabs in central lower part of **DEWESoft Setup screen** depend on selection in **System** → **Hardware setup** menu and consecutive on measurement mode (**PROF** or **DEMO**) → see → [Installation Guide](#) → **Hardware setup**.

1.1 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 installation new or changing input hardware), the **Analog** tab on **DEWESoft Setup screen** must be selected to **set all analog input channels** and this screen displays two main parts:

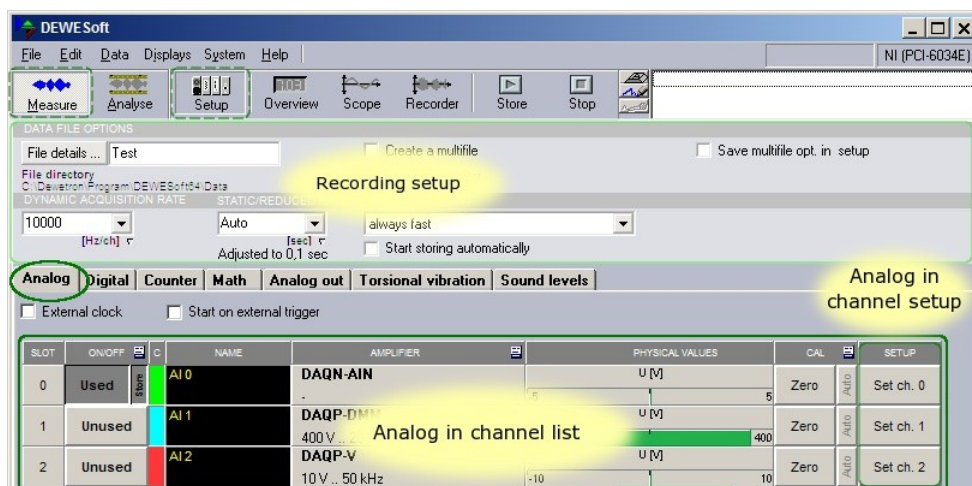
Recording setup with **Data file options** to set target of data storing and **Storing settings** to set up storing options and conditions
 → see → [User Guide](#) → **Recording Acquired Data**
 for information about **New**, **Load** and **Save** the **channel Setup data**
 see → [User Guide](#) → **Data management**

Analog tab displays **Analog in channel settings** in fields:

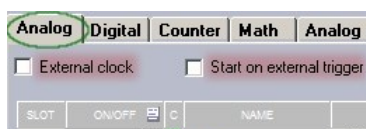
- **External Clock** - to use *external clock* as the **source of acquisition**
- **Start on external trigger** - to *start the acquisition* on **external event**

and columns of:

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



First we can check two fields above the **channel list**:



External clock

This option selects *external clock* as the *source* of acquisition.

Usually (if this option is not selected) the clock source is **internal** and *defined* by the *sample rate* selected in frequency span. If this option is checked, the *source* for acquisition comes from an *external clock source* such as *encoder* or *external oscillator*. There are two major settings of external clock:

- fixed external clock
- variable external clock

Fixed external clock option is used when we have external fixed clock source (like another device with fixed speed which is clocking the Dewetron system). In this case we have to enter the frequency of the clock that we still have a time as the x axis in the recorder.

If we have variable external clock (for example encoder), then we can enter samples per revolution to have the x axis units in *revs*. If we leave the entry field as 1, then the x units will be *samples*.

Start on external trigger

We can select this option to *start* the acquisition on *external event*. This option is useful when we want to start capturing data not on the start of the *measurement*, but on **external digital event**. It is most useful with *extremely high* sampling rates, but in a lot of cases the *software trigger* is better because we see *the data* even *before* the trigger occurs and we can also use *pretrigger*.

1.1.1 Channel list

SLOT	ON/OFF	C	NAME	AMPLIFIER	PHYSICAL VALUES	CAL	SETUP
0	Used	AI 0	DAQN-AIN		U [V]	Zero	Set ch. 0
1	Unused	AI 1	DAQP-DMM	SN 218100	U [V]	Zero	Set ch. 1
			400 V .. 20 kHz		-400 400		
2	Unused	AI 2	DAQP-V	SN 221800	U [V]	Zero	Set ch. 2
			10 V .. 50 kHz		-10 10		

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*

1.1.1.1 Input channel - Slot

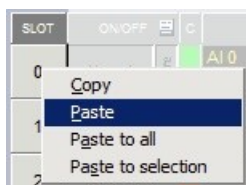
Slot column indicates the *number* of the *channel*. It always *starts* with **0** and counts up to the *maximum* number of *available module outputs*.

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

If you have a DEWE-3010 or DEWE-3000, the first 8 (**0** through **7**) slots are the ones on the mainframe itself, and the second group of 8 slots pertain to either the BNC inputs or RACK-8/3010 expansion rack, depending on the configuration of your system.

If you have a DEWE-2010, DEWE-4000, DEWE-RACK-16 or DEWE-BOOK-16, then each slot pertains to one of the 16 on the system 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. 0, 2 and 6) contain the *same* DEWE-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.

ATTENTION: This function is working only with amplifiers of the same type. Channel names will not be copied.

1.1.1.2 Activating channel - On/Off

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

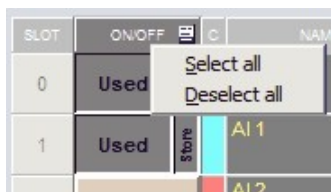
Select input channel for display

Input **0** 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.

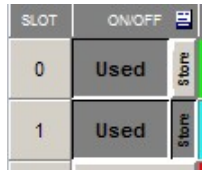
To *activate / deactivate* all channels at the same time, click on the  icon and the following selection window appears:



Now use either **Select all** to *activate* or **Deselect all** to *deactivate* all channels.

Select input channel for storage

Beside the **Used** buttons, a **Store** button is displayed:



The **Store** button is *only* visible when a channel is set to **Used**.

The *default* setting *after* activating the channel is **storing** - small  - **Store** button appear beside the **Used** button and *channel data* will be **stored**.

When you press the **Store** button, his appearance change to , 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*).

1.1.1.3 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.

1.1.1.4 Channel name - Name

Free text field for **nam**ing this channel.

NAME	
AI 0	MD
AI 1	MD
AI 2	MD
AI 3	MD
AI 4	MD
AI 5	MD

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**

1.1.1.5 Amplifier type and ranges, sensors

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

AMPLIFIER (014)	
MDAQ-ACC	SN:229400
2500 mV .. 300 kHz	-1
DAQP-CHARGE-B	SN:221654
100 pC .. 100 kHz	
DAQP-FREQ-A	SN:220728
200 kHz .. 200 kHz	

- for DAQP/MDAQ module it will show the *name* of the module and the *range* selected.
- for DAQN or DAQ series (non-programmable) module only the *name* is shown
- for PAD series module the *name* and *range* are shown.

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:

AMPLIFIER (003)		CHOICE OF VALUES	
MDAQ-ACC	2500 mV .. 300 k	DAQP/MDAQ modules	Set PowerOnDefault
MDAQ-ACC	2500 mV .. 300 k	Clear not found modules	Reset PowerOnDefault (DAQP only)
		Fill Rack	[9]

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).

Fill rack option will enable option to *start adding* modules to the system.

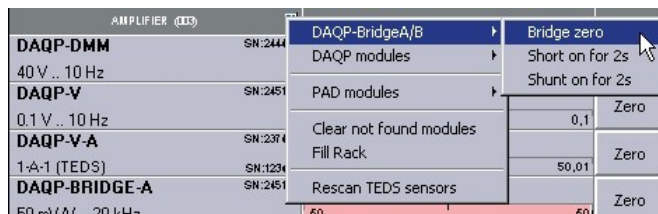
for more information about **add / replace modules** also see → [Installation Guide](#) → **Module Installation tips**

When there is at *least one special* module (like DAQP-BRIDGE), 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.

For BRIDGE A/B/S module we have:



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 ChargeB 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. If there is any PAD-CNT modules installed, there is an option to *reset* the *counters* on those modules.

Installing modules

You can **install** your DAQP and PAD modules (and any DAQP module that has a **small black** button near the top of the module) all at once, or if you **replace** just *one* module, you can *also replace* just one module in the **DEWESoft** without installing them *all again*.

for more information about **add / replace modules** also see → [Installation Guide](#) → **Module Installation tips**

Sensors and TEDS sensors

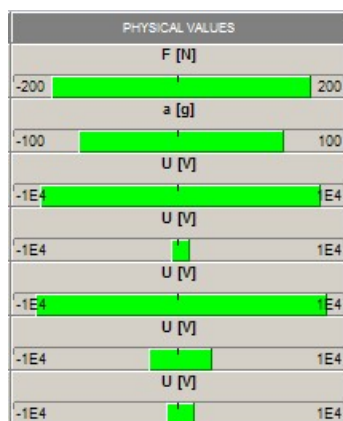
For *each* analog channel, **sensor** can be *predefined* in the **Sensor database** and **used** in the **Channel setup** for individual channel. To notify that the sensor is used, the sensor name and serial number is shown instead of the module range as shown in the picture below:

Direct
transformer (Voltage) SN:U_CH01

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. Amplifiers which support TEDS are MDAQ-ACC, MDAQ-ACC-A and DAQP-V-A. 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.

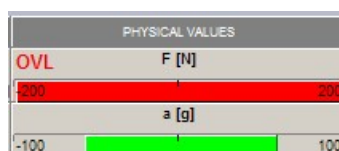
1.1.1.6 Physical values



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

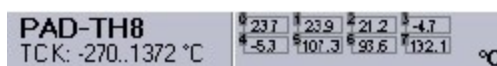
All this can be set in **Channel setup** column, which can be reached by *double click* on channel cell in this column.

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



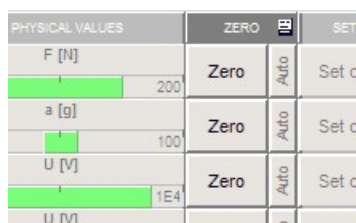
if this happens, *check your sensor and / or select another input range* (in **Channel setup**).

In the case of multi-channel PAD modules, like the PAD-V8-P, PAD-TH8P, et al, this field will be *divided* into *smaller* boxes, each containing a **dynamic representation** of your *inputs*:



1.1.1.7 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.



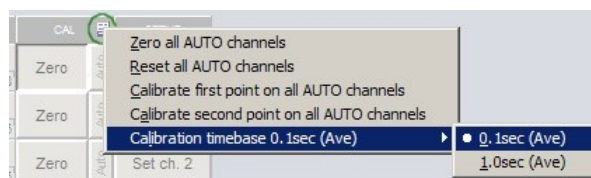
Press the *left* mouse button to *activate* **zeroing**, and the *right* mouse button to *deactivate* it (reset to *default* input range).



The **Auto** button is also *activated* by the *left* mouse click and *deactivated* by the *right* click.

You can set any channel to *auto* mode. If you click on the **Zero** button, the following selection window appears.





- **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**.
- 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.

1.1.2 Channel setup form

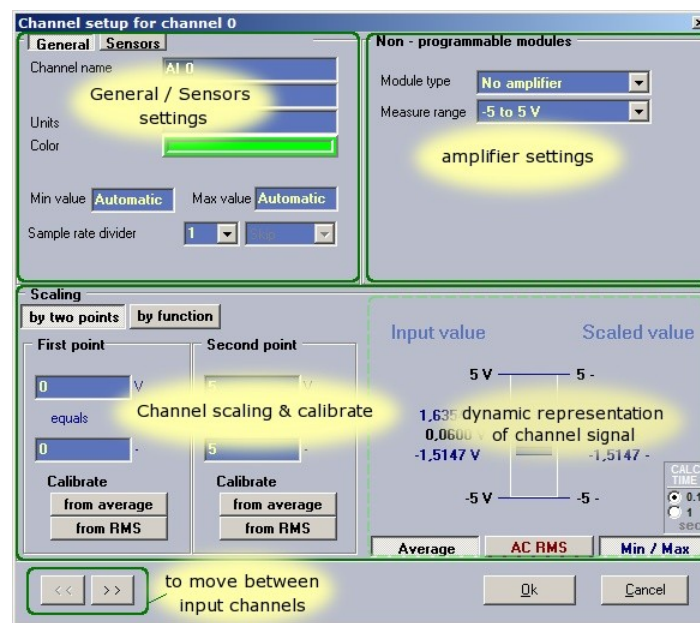
In **Channel setup** window we can do **settings** and **calibration** for selected **input channel**.

General settings	you can <i>enter</i> on the left side of General part the Channel name , the measurement value , Units of measurement, <i>define</i> 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	<i>define</i> and set sensors information and data in <i>sensor database</i>
Amplifier settings	<i>define</i> and set amplifier settings: <i>Non-programmable</i> , <i>series</i> and <i>special programmable modules</i>
Scaling	perform scaling and <i>manual</i> or <i>automated</i> 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 0 press **Set ch. 0** button.



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

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

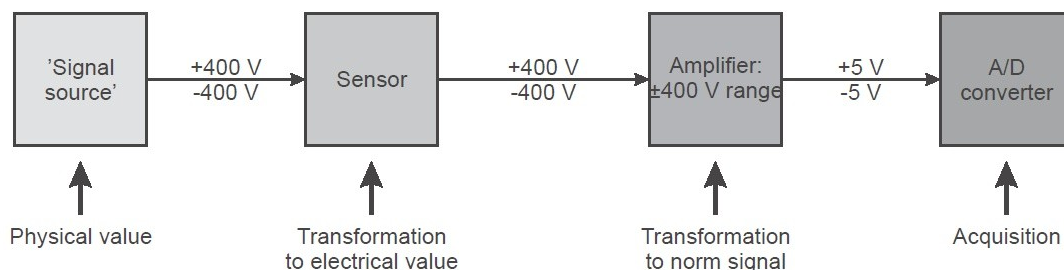


On bottom left part of screen are two buttons to *switch* between input channels without leaving this *Channel setup* window.

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*) will be measured with cables (= '*sensor*') connected to a DAQP-DMM module (= *amplifier*).

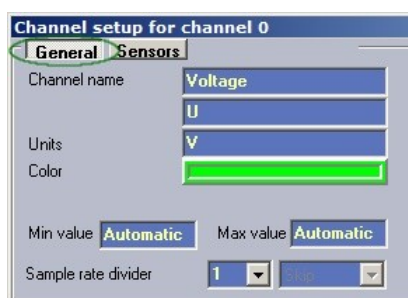


1.1.2.1 General settings

In this '**General**' part of *Channel setup for channel n* 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
- **Sample rate divider**

Click on the **General** button in the input *Channel setup for channel n* screen:



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.

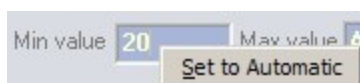
Min / Max value

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 divider

The **Sample rate divider** can be used to 'reduce' the sample rate for *each* channel. It offers *different types of reduction*:

- **Skip** selected number of values - this option will keep only each *n*th sample and discard all the rest,
- **Average** selected number of values - this option will calculate average of samples and keep the result,
- **4th, 6th or 8th order Filter** - this option will perform low pass IIR filtering at 40% of sample rate and keep only each *n*th sample of averaged data.

These types of reduction can be selected from drop down list when **Sample rate divider** number value is not 1:



As a standard, the number of values is set to 1 and Skip.

for details about *sampling rate* also see → [Reference Guide](#) → **Basics of A/D conversion**

1.1.2.2 Sensors setting

In this 'Sensor' part of **DEWESoft Channel setup for channel n** 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**.

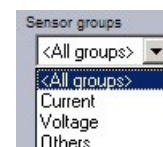
for details about **Sensors editor** see → [User Guide](#) → **Sensor database**

Open the **Sensors** tab and screen like this appear:

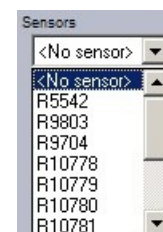


The upper left side shows the Sensor predefined values as well as additional values which can be defined in setup. The *predefined* values are **Scale**, **Offset**, **Description** and **Recalibration date**. If calibration is *overdue*, the date will be shown in red.

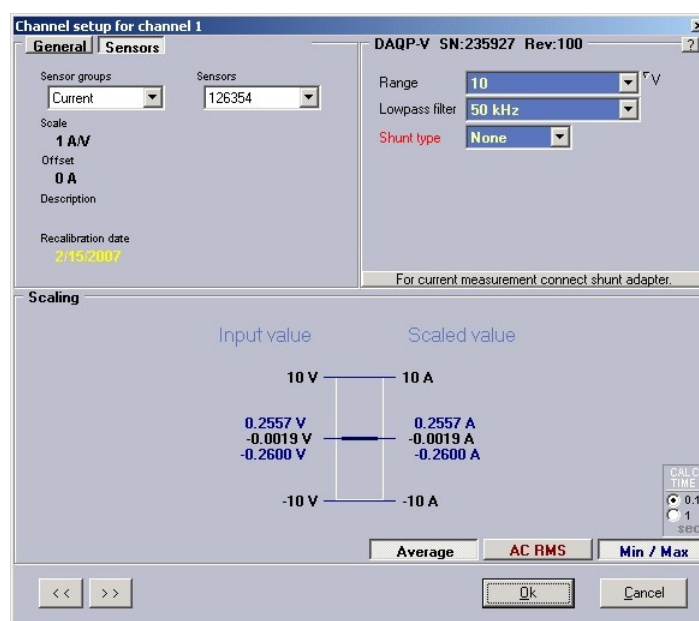
Choose from **Sensors group** selection list appropriate **sensor group** in sensor database.



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

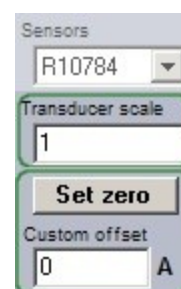


After sensor selection screen like this appear:



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**

1.1.2.3 Amplifier settings

In **DEWESoft** input *Channel setup for channel n* screen we can do for selected *input channel* on upper right part of this screen all **Amplifier settings** for:

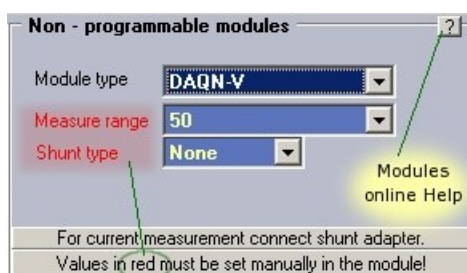
- **No amplifier** and **Non-programmable DEWE modules**
- **Programmable DEWE modules**

All boxes and fields at the top-right of screen depend from the *built-in module*.

Non-programmable DEWE modules

You could select any **non-programmable** DEWE **module** from **Module type** drop down list; older DAQN series are *supported*, making this software backwardly compatible with all modules. If you don't have built-in any module, you can select special **No amplifier** module.

After selecting module screen with settings for this module appear:



On this **Non-programmable module** section of *Channel setup for channel n* screen are displayed:

- **Module type**: selected module type or **No amplifier**
- settings for module type: for all module in this place is **Measure range**; for some modules there is also e.g. **Input type**, **Shunt type**, **Tmin**, **Tmax** ... (caption of all this fields is in red color)
- messages to set values in red in the module manually, connect shunt adapters...
- **Help** button to invoke modules online Help

Programmable DEWE modules

If a programmable module is installed in this slot, the **Module type**, **Serial number** and **Revision number** are displayed as *caption* of this section.

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

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

DAQP-BRIDGE-A SN:234575 Rev:203

Range: 2000 um/m

Lowpass filter: 100 Hz

Input type: Quarter (120 ohm)

Excitation: 5 V

Output offset: 0 %

Br.zero offset: -34.3 %

Buttons: Bridge zero, Amplifier zero, Short, Shunt

DEWETRON produce many series and special modules, e.g.:

- **Special DAQP series modules**

DAQP-BRIDGE-A, DAQP-BRIDGE-B, DAQP-CHARGE-A, DAQP-CHARGE-B
DAQP-FREQ-A, DAQP-V, DAQP-V-A, DAQP-V-B, DAQN-THERM

- **Special MDAQ series modules**

MDAQ-V, MDAQ-ACC, MDAQ-ACC-A, MDAQ-BRIDGE, MDAQ-STRAIN

- **PAD series modules**

for details about DEWE **Modules** and all *modules settings* see:

- [Manufacturer Modules Technical Reference manual](#)
- [DEWE Modules online Help](#)

for useful hints about DEWE **Modules settings** by different *Measurement application* see:

- [DEWESoft Tutorials](#)

1.1.2.4 Scaling

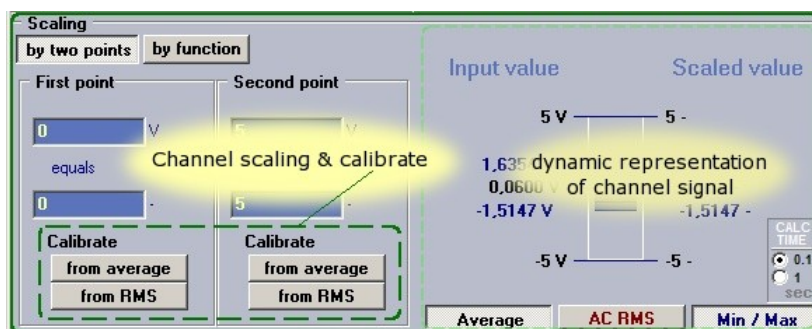
The **Scaling** procedure define **ratio** between the 'electrical' **input value** of selected **input channel** (from amplifier) and the scaled 'physical-engineering' **input value**, which is used in **DEWESoft** procedures.

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:

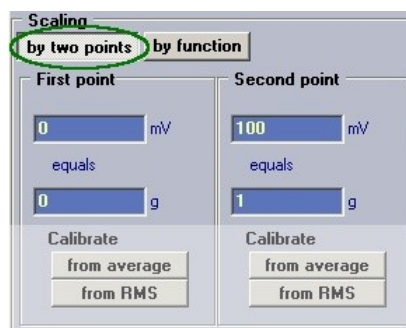
- **Scaling - calibrating on input:** either on a **2-point** or **functional basis**
- *manual / automated* **calibration**

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*.



Calibrating on input

Scaling - '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

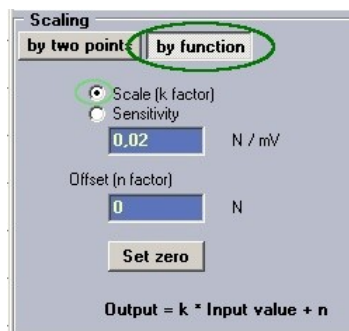
above to 'calibrate from average' of the first point, because the *scaling factor* will *change* as well. Please look at the following section - Scaling by function and perform a *sensor zero*.

for useful hints about **scaling factor** by different *Measurement application* see:

→ [DEWESoft Tutorials](#)

Scaling - '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:

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

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**:

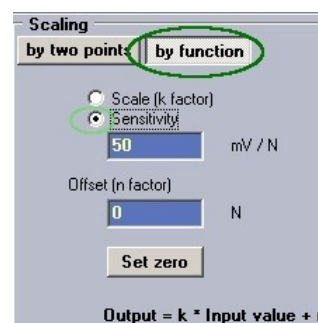
- **Scale (k factor)**; example above: $k = 0.02 \text{ N / mV}$

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

- **Sensitivity**; same example on right: $k = 50 \text{ mV / N}$

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.

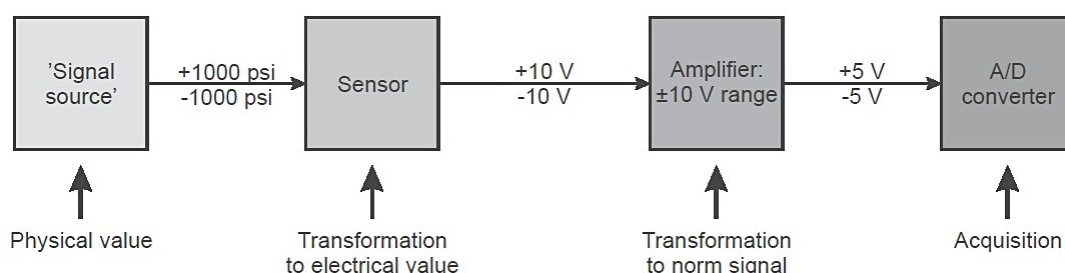
Since all PAD modules that measure *temperature* output their values in the scientific standard *Celsius* scale, this is a handy formula to know when the temperature is desired in *Fahrenheit*.

for useful hints about **scale (k factor)** and **Sensitivity** by different *Measurement applications* see:

→ [DEWESoft Tutorials](#)

Calibrating an input example

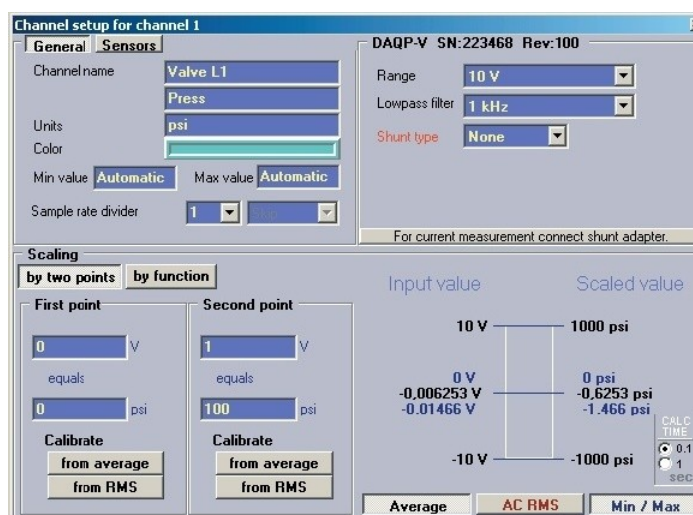
Pressure up to ± 1000 psi (= signal source) will be measured with a 100 psi / V sensor, connected to a DAQP-V module (= amplifier with 10V range).



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 to look like this:

Scaling dialog box, 'by function' tab. The 'Scale (k factor)' radio button is selected, and the sensitivity is set to 100 psi/V. The 'Offset (n factor)' is set to 0 psi. The 'Set zero' button is visible. The 'Input value' column shows 10 V, 0.004885 V, -0.006644 V, and -0.01954 V. The 'Scaled value' column shows 1000 psi, 0.4885 psi, -0.6644 psi, and -1.954 psi. The 'Output = k * Input value + n' formula is displayed at the bottom.

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

- **Scale (k factor)**; example above: $k = 100 \text{ psi / V}$
- **Sensitivity**; same example: $k = 0.01 \text{ V / psi}$

Scaling dialog box, 'by function' tab. The 'Sensitivity' radio button is selected, and the sensitivity is set to 0.01 V/psi. The 'Offset (n factor)' is set to 0 psi. The 'Set zero' button is visible. The 'Input value' column shows 10 V, 0 V, -0.006839 V, and -0.01954 V. The 'Scaled value' column shows 1000 psi, 0 psi, -0.6839 psi, and -1.954 psi. The 'Output = k * Input value + n' formula is displayed at the bottom.

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**.

1.1.2.5 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:

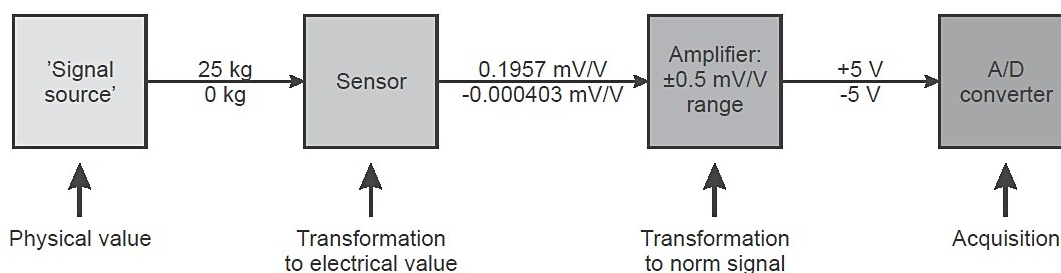
Now, we need to *apply* the known zero (0 kg) load. When it *stabilizes*, press the *left-most from average* button, and the software will *sample* the input for a short time, *average* the values that it receives, and then *calculate* the *required number* for the upper left box - *First point*.

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).

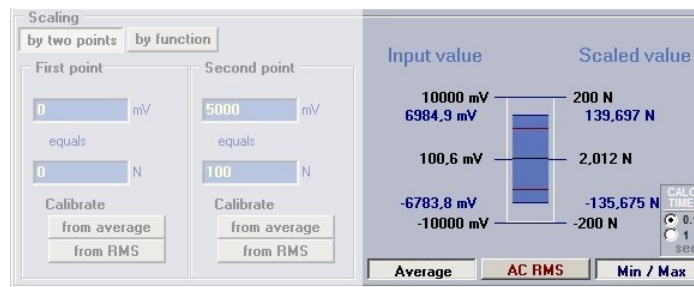


for useful hints about **Calibrate** by different *Measurement application* see:

→ [DEWESoft Tutorials](#)

1.1.2.6 Live data preview

The bottom-right 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 **Calculations time** for signal display.

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

Calculation time

On this section on graph you can change **CALC(ulations) TIME** and *signal display frequencies* of the *selected values*.

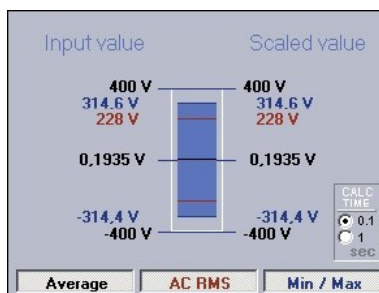


Predefined choices are:

- **0.1** second
- **1** second.

Dynamic representation of your signal

Now you can **verify** the current **input signal** at the bottom-right corner of the dialog - **dynamic representation** of your signal in **bar graph**:

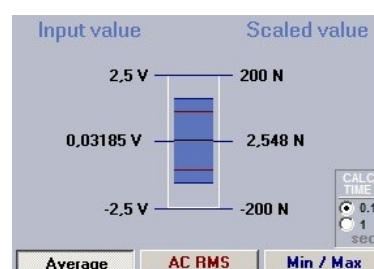


- The **bar graph** shows the *current input* signal. In addition, the **min / max**, **AC rms** and **average** values are displayed. If the input signal is higher than the selected input range, you will see a message **OVL** in this screen.
- The values at the *left* side of the bar graph shows the '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, also see → **dynamic graph**.
- Change the **CALC TIME** between **0.1** and **1 sec** to achieve the *best view* for your signal.

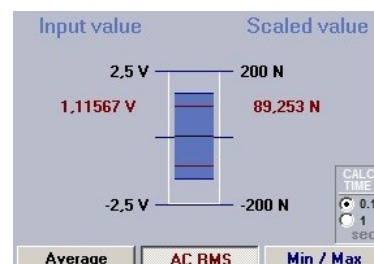
Dynamic graph

Depending on *pressed* button(s) in this area **Average** **AC RMS** **Min / Max**, different **Input** and **Scaled values** are represent on **dynamic graph**:

When **Average** button is pressed, **Average** value of your Input and Scaled **signal** is *represented* on dynamic graph.

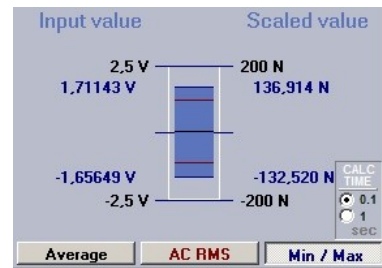


When **AC RMS** button is pressed, **AC RM** value of your Input and Scaled **signal** is *represented* on dynamic graph.

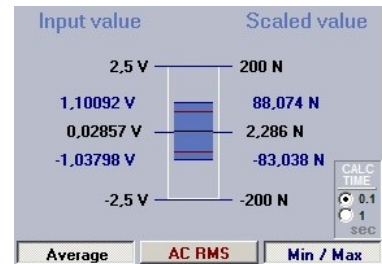


If you have an **AC** signal source, you should press the **AC RMS** button.

When **Min / Max** button is pressed, **Minimum / Maximum** value of your Input and Scaled **signal** is *represented* on dynamic graph.

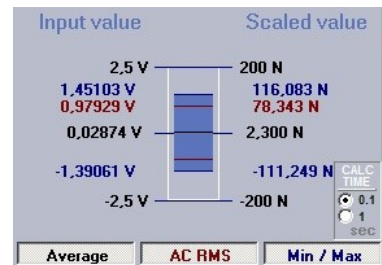


Usually the: **Average** and **Min / Max** buttons are pressed and this way **Average** and **Minimum / Maximum** values of your Input and Scaled **signal** are *represented* on dynamic graph.



If you have an **AC** signal source, you should also *press* the **AC RMS** button below the graph to *add* the **RMS values** to the dynamic display.

When **Average** and **Min / Max** and **AC RMS** buttons are *pressed*, **Average** and **Minimum / Maximum** and **AC RM** values of your Input and Scaled **signal** are *represented* on dynamic graph.



1.2 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

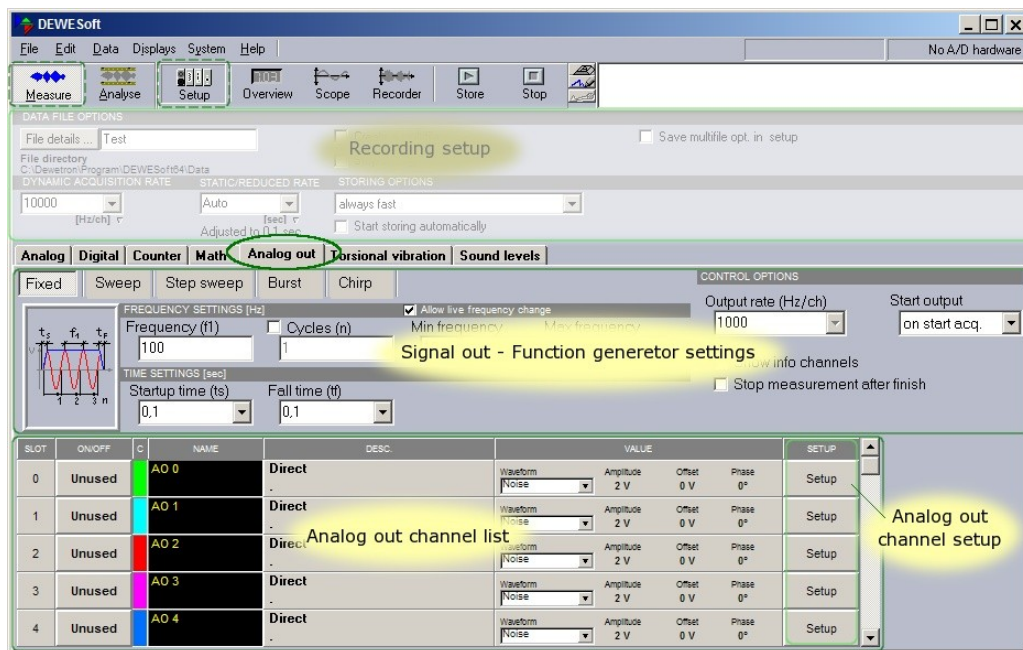
with all *definitions* and *settings*:

- **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:

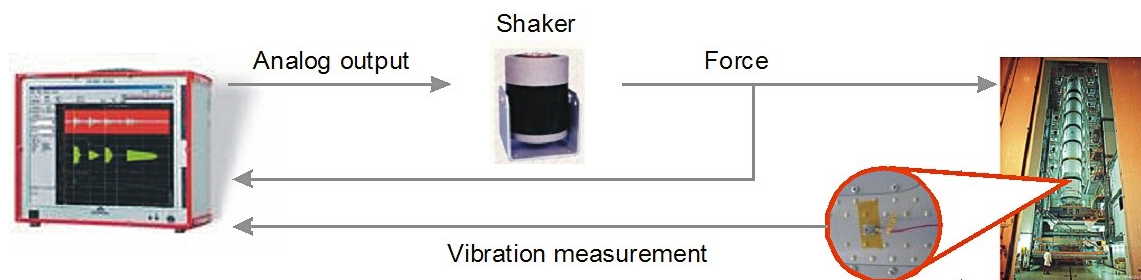
- **Slot** - indicates the *number* of the channel; it always starts with **0** and counts up to the *maximum* number of *available* output channels
- **On/Off** - *activate / deactivate* output channels; *click* on the button to activate or deactivate channels
- displays the state of the channel: **Unused** (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:

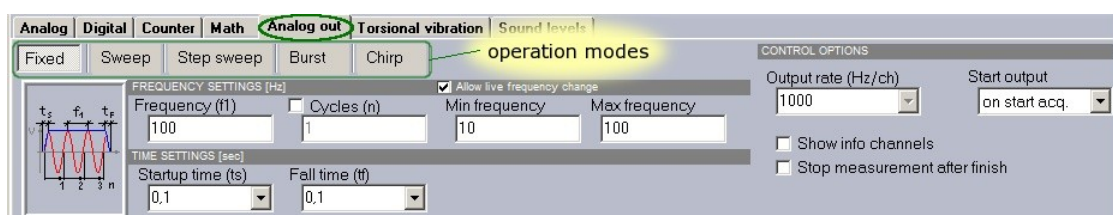
If the shaker is used for excitation of structures, single or multiple shakers are driven by either analog output or by external shaker system. The force is induced in the structure and the response of the structure is measured by the accelerometers or another vibration transducers.



1.2.1 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**



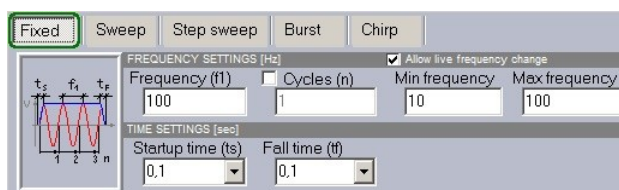
The function generator offers five *different operation modes*, which are set for all analog output channels:

- **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

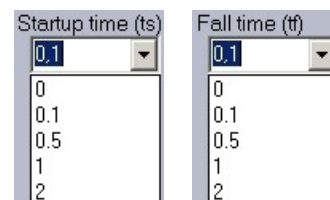
Fixed mode *outputs* the selected **signals** with a constant **Frequency (f1)**. Output frequency is the *same* for *all* the channels and is *defined* in the setup.



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**).

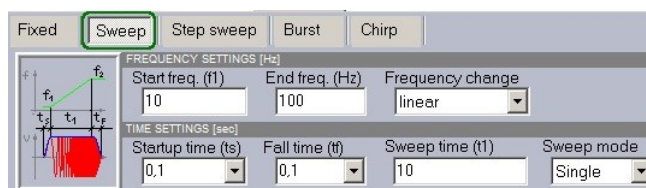
To avoid jumps in the signal a **Startup time (ts)** and a **Fall time (tf)** at the *beginning* and the *end* of the **signal output** can be set with selection from drop down list.



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.

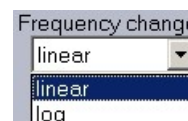
Sweep

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*.

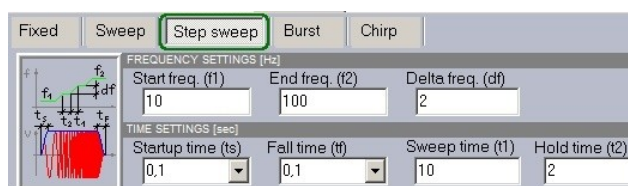


Applications: sweeps are useful for testing *electronic circuits* like filters as well as in *automated* modal analysis.

Step sweep

The step sweep mode is a special procedure for modal testing, where the *signal remains* at *certain fixed frequencies* to allow the structure to settle to that frequency. In addition to this more *exact* frequency resolution of *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*.

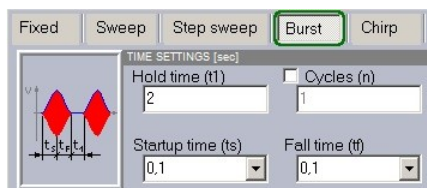
Sweep time (t1) defines the sweep **duration** to the *next* frequency step, **Hold time (t2)** sets the **duration** where the frequency keeps *fixed*.

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 (t_s)**, **Fall time (t_f)** and **Hold time (t_1)** 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.



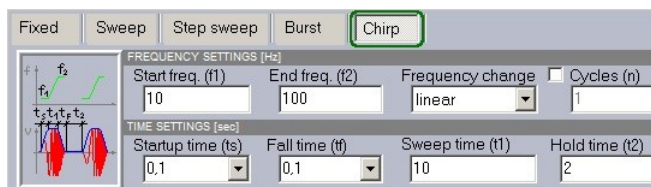
Within the *startup* and *fall* time you get **noise output**.

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. (f_1) and **End freq. (f_2)** *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 (t_s)** and **Fall time (t_f)** which *defines* the **ramp** at the *beginning* and the *end* of the chirp, **Sweep time (t_1)** - the chirp **duration** - and the **Hold time (t_2)** 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.

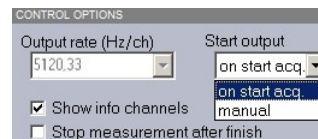
Applications: The **chirp** signal is used for *spectral analysis of nonlinear systems*.

1.2.2 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 than 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)** number for output in the **Operation mode** settings.

The **start** and **stop** of *generation* can be either **on start acq.**(quisition) 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*.

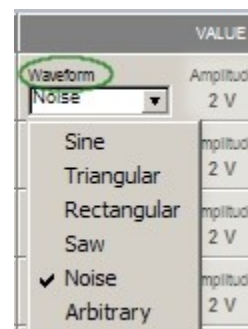
1.2.3 Channel value

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:



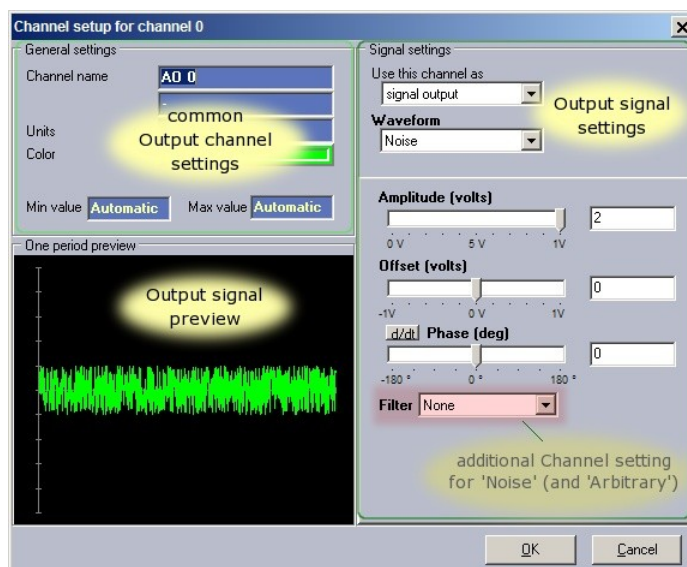
Amplitude, **offset** and **phase** can be set also in **Channel setup** with slider or enter the desired value directly.

1.2.4 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**



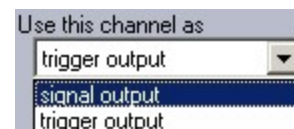
Signal settings

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

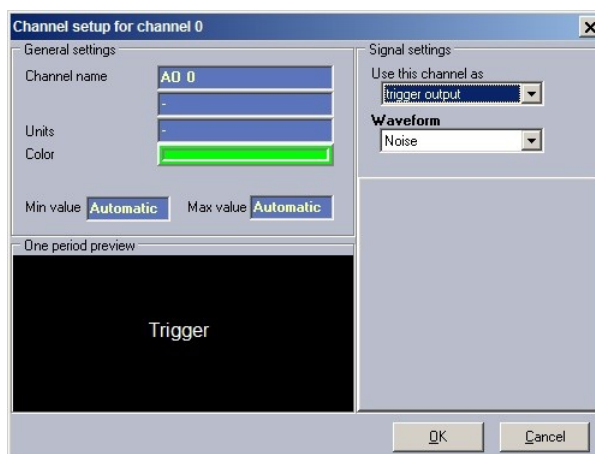
1. signal usage selection in Use this channel output as.

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



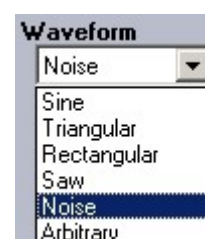
Often it is intended to *trigger* the *acquisition* on *measurement* side when *certain criterion* is *met*. When the sweep is active, the trigger value *goes up*. It is up at step sweep, at fixed frequency and during the bursts.



2. 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.

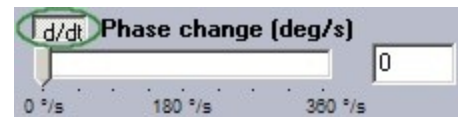


Amplitude, offset and phase angle

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**:

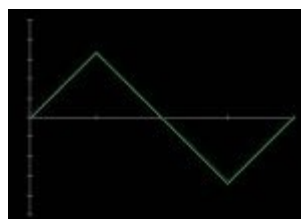
- for **Arbitrary** type of waveform there are sliders and numeric fields in **Signal** tab, additional is also **Points** tab for this **signal setting** → see → **Arbitrary - Additional settings**
- for **Noise** type of waveform bellow the sliders there is also an additional **Filter signal setting**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

1.2.4.1 Waveform type

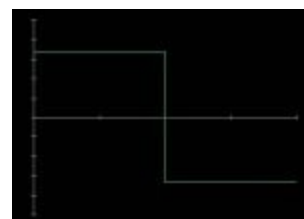
In **DEWESoft** software the **waveform types** are:



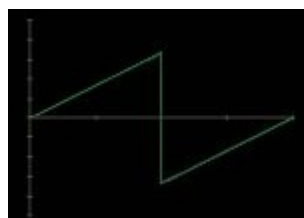
SINEWAVE



TRIANGLE



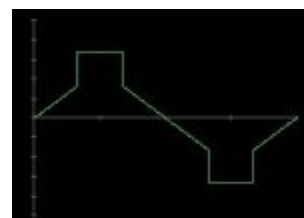
RECTANGLE



SAW



NOISE



ARBITRARY

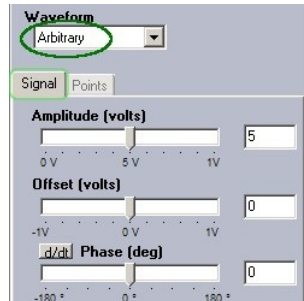
Waveform types can be set in:

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

1.2.4.2 Arbitrary - additional settings

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

Signal tab

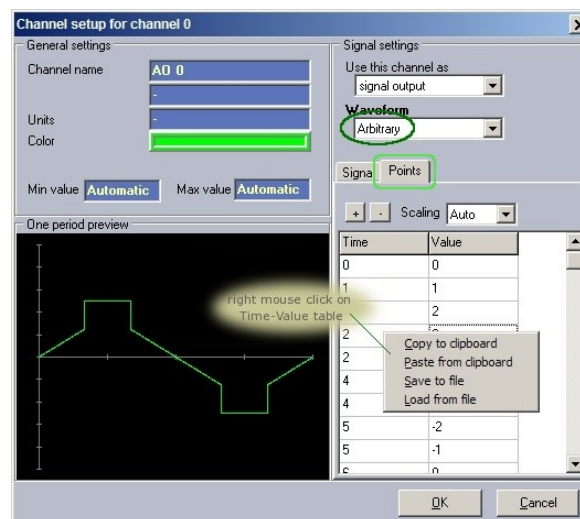


Points tab



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

The waveform itself is defined 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 directly in the table.
Press **+** or **-** buttons to add or remove rows in the table and enter values for **Time** and **Value** (*amplitude*).
- **Paste from clipboard**
For example 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!

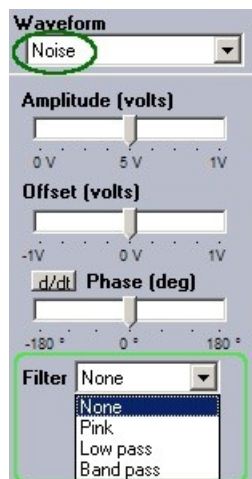
When this option is selected *additional* field appear to **enter** scaling factor:



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

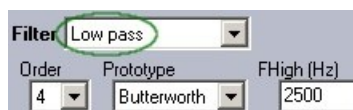
1.2.4.3 Noise - additional settings

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**

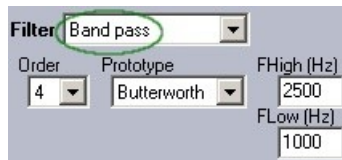
define a *low pass* filter:



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

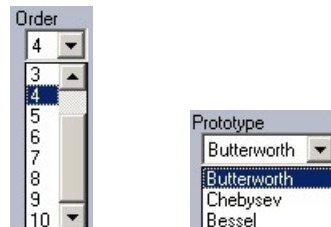
- **Band pass**

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:

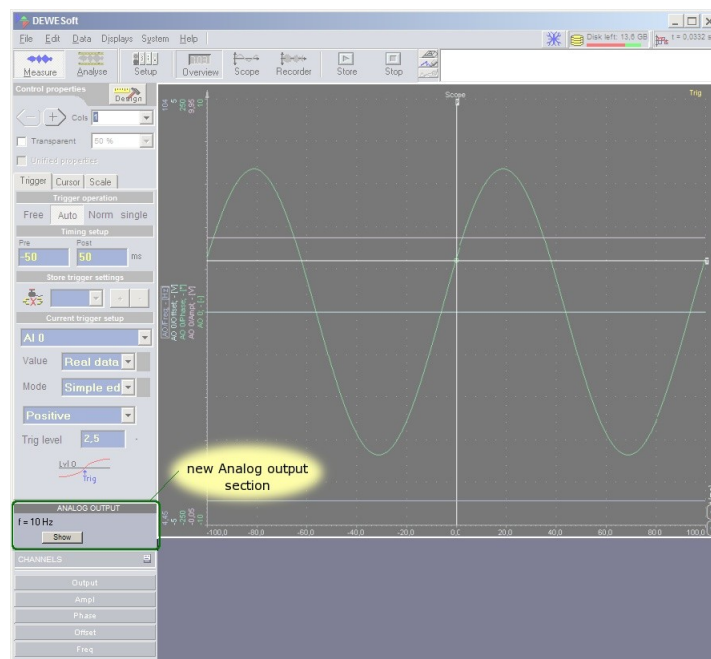


for information about **Order** and **Prototype** see → [User Guide](#) → **IIR Standard filter**

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.

1.2.5 Setting during Measurement

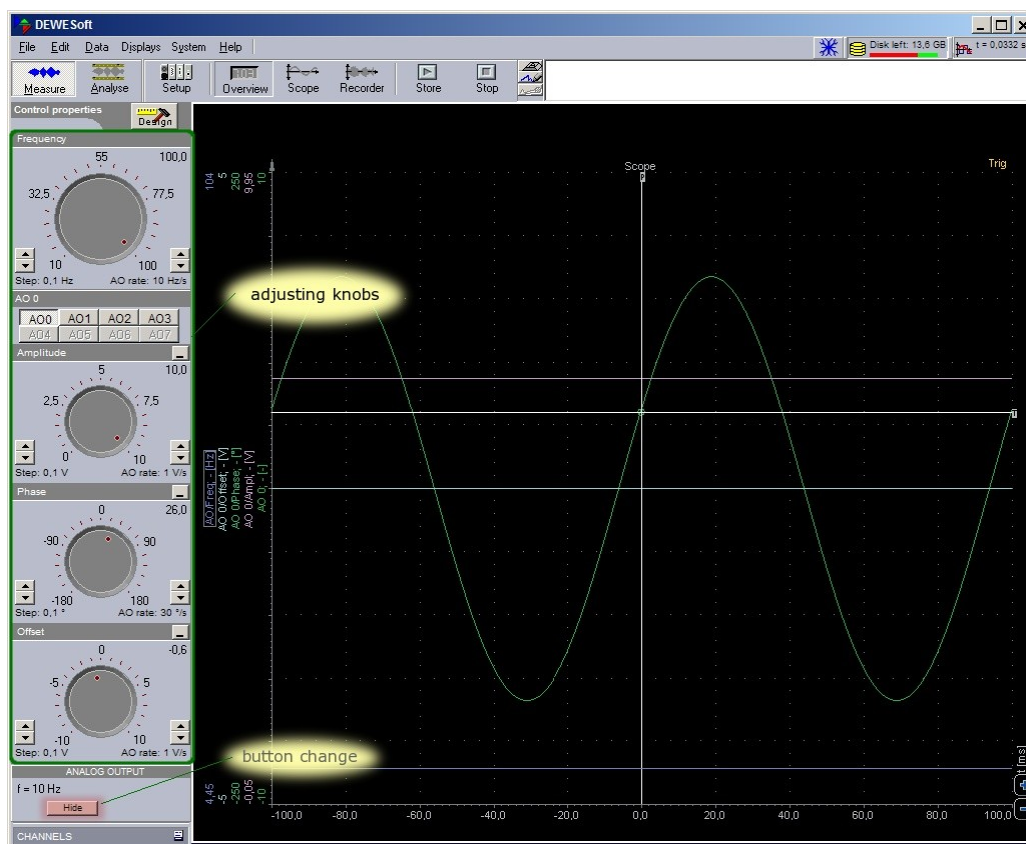
All analog *output channels* (including their parameters) are available in **DEWESoft** as *internal* channels also (if *hard sync* option is set in *hardware* setup). Therefore you can **display** them together with your *other signal* sources.



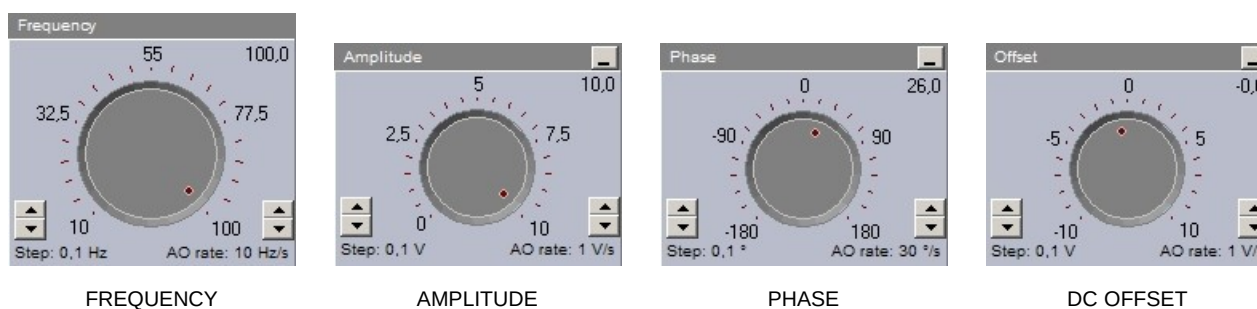
If analog outputs are activated in **DEWESoft**, a new section is visible directly above the channels selector. This section **Analog output** displays the *current frequency* of the output signals and contains one button. Press the **Show** button to

display the *output parameter controllers* (this button change to **Hide** and press this button to hide them - if required).

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



Use the **adjusting knobs** to change the parameters **Frequency**, **Amplitude**, **Phase** and **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*.

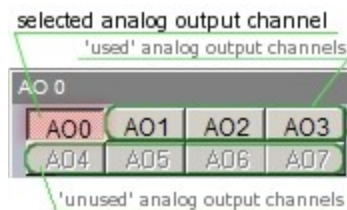
Example: *all controls knobs are minimized*



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:

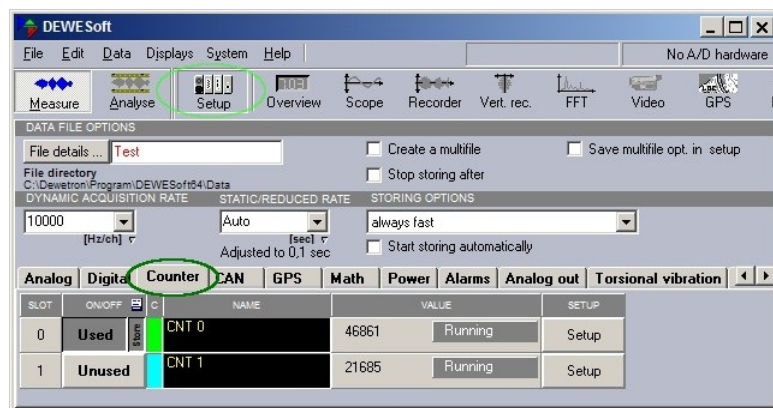


1.3 Counters

DEWESoft Counter module is used to perform **counting** and **frequency measurements**. The **counter** setup depends on the *used hardware* platform.

Typical applications are: event, gated event and up/down counting, encoder measurements, period and pulsewidth measurements, two pulse edge separation and frequency/supercounter.

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



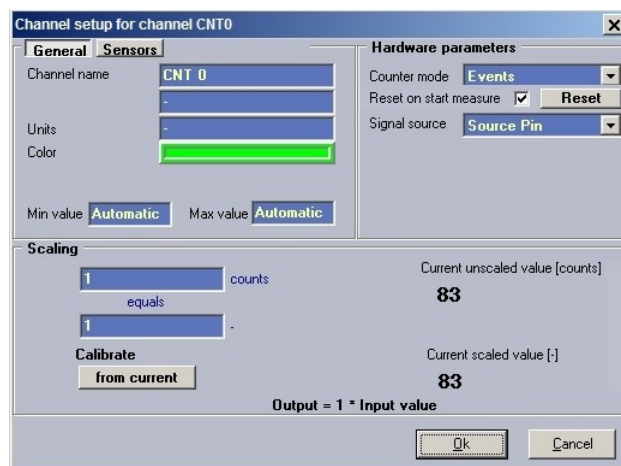
Counter channel list

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 Setting](#)

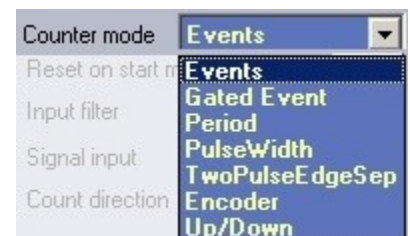
Hardware parameters

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](#)
- [frequency/supercounter](#)



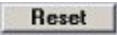
for information about **Hardware parameters setup** click on caption above

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.

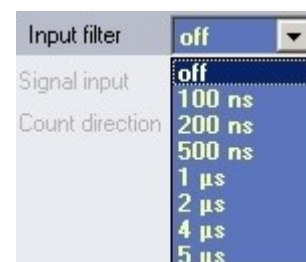
- **Reset** button

With press on  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 inverting**

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

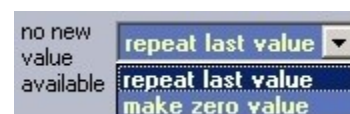
This has two effects: first is that the levels will *change*, so **normal** level will be **low** and consequentially the counter will count on *falling edges* of the *input signal*.



- **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.

1.3.1 Event, gated and up/down counting

Event counting is one of the simplest counter operations.

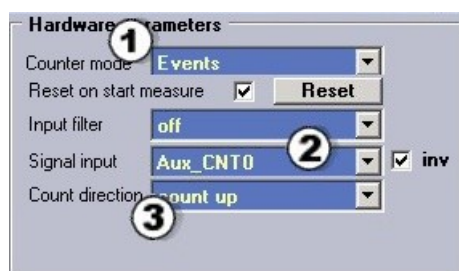
<i>Required hardware</i>	Orion, NI (cards with counters), NI-MX, DAP, Data translation
<i>Setup sample rate</i>	At least 1 kHz

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

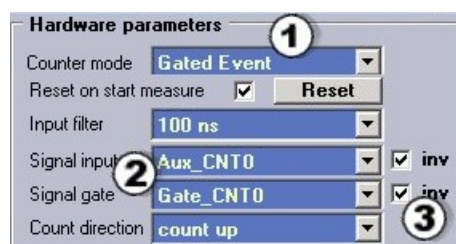


1. The **Counter mode** 'Events' ① are already selected by default.
2. Then we choose the **Signal input**. Usually the signal input is **Source Pin**, but if we have connected other signal, we need to select that signal ② as the input (e.g. **AUX_CNT0** above).
When the *normal state* is *high*, sometimes 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:

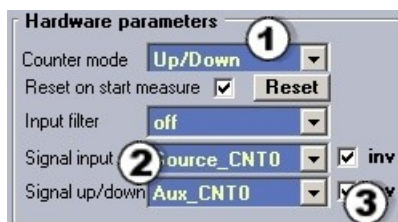
- **Signal gate** (e.g. **Gate_CNT0** above) ②. The counter will count the transitions from *low* to *high* only when a *signal gate* is *high*. Signal channel can be selected from drop down list.

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*' counting① 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 box③ (normally it is *high*), we choose to **invert** also the gate signal, that it will count **down** only when *gate signal* level is *high*.



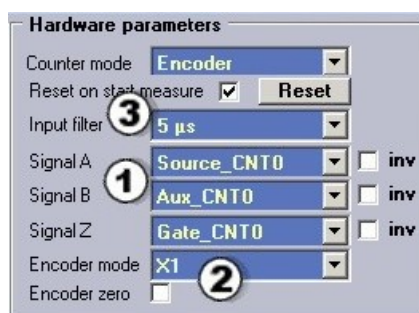
1.3.2 Encoder

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	fully available only on Orion cards, NI-MX supports some modes
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 → **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*.



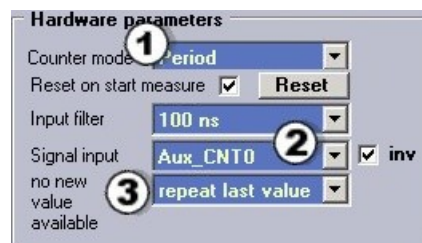
1.3.3 Period and pulsewidth

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	Orion cards
Setup sample rate	At least 1 kHz

for hints about **Period and pulsewidth setup** and **measurement** see → [DEWESoft Tutorials](#)

Period measurement

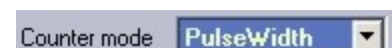


1. Select the **Counter mode** as 'Period' ①.
2. Choose the **Signal input** (e.g. `Aux_CNT0` in above example) and set the signal **Input filter** to *prevent glitches* ②.
see → [Counter hardware parameters](#)
3. In additional field **no new value available** tells the software what to do when *no 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 **Counter 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.

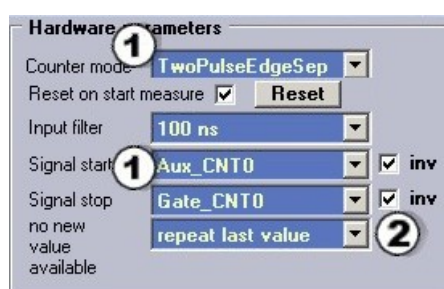
for hints about **Duty cycle setup** and **measurement** see → [DEWESoft Tutorials](#)

1.3.4 Two pulse edge separation

Required hardware	only on Orion cards
Setup sample rate	At least 1 kHz

Two pulse edge separation is a special mode of Orion *counters*. It **measures** the time between the *raising edges* of two *signals*. This is useful for measurement of the **time between two events** with a *very high precision*. The typical application is measure, for example, the *velocity* of an object passing by two sensors.

for hints about **Two pulse edge separation setup** and **measurement** see → [DEWESoft Tutorials](#)



1. Select the **Counter mode** as 'TwoPulseEdgeSep'.
2. Choose the **Signal start** and **Signal stop** ① *input signal* (e.g. `Aux_CNT0` and `Gate_CNT0` in above example). Both signals can be selected from drop down list:



When both of the signals are **inverted**, the *real trigger condition* will be *falling edge* of the *input signal*, because the values are *normally high* when event not occurs.

3. In field **no new value available** tell the software what to do when *no new value* is available ②.

see → [Counter hardware parameters](#)

4. Set the signal **Input filter** to *prevent glitches* → see → [Counter hardware parameters](#).

We can use this value for example for *speed* measurements, if we are *triggering* (for example) from two *light sensors*. If we know an *exact* distance between them, the speed would be $v = s / t$, where *s* is a constant and *t* is the *result* of the *edge separation measurement*. This can be doing in the **math** channel.

1.3.5 Frequency

Required hardware	only on Orion cards
Setup sample rate	At least 1 kHz

The problem with previous (traditional) counters is that the value of the counter is *latched only* at a sample rate interval. Therefore we have only discrete values on each sample.

But since the second counter of Orion 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*.

for details and hints about **Frequency / super-counter setup** and **measurement** see → [DEWESoft Tutorials](#)

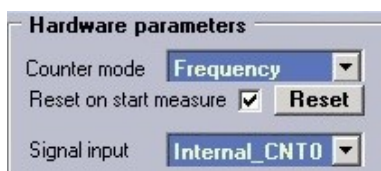
For **super-counter** and **frequency** measurement we need to use *two* counters. Another limitation is that *counter channel* needs to be an *even* counter (CNT0, CNT2 or CNT4) and the *frequency channel* needs to be the *next* counter - (CNT1, CNT3 or CNT5) ①. So the counter pairs are for example CNT0 and CNT1.

Analog	Digital	Counter	Video	Math	Order tracking
SLOT	ON/OFF	C	NAME	VALUE	SETUP
0	Used	SuperCounter	388.1	Event CNT	Setup
1	Used	Frequency	4.8	Frequency	Setup
2	Unused	CNT 2	388.1	Event CNT	Setup
3	Unused	CNT 3	4.9	Frequency	Setup
4	Used	EventCounter	388.1	Event CNT	Setup

The **counter 0** is set to simple **Events Counter mode**, with set **Input filter** and defined **Scaling**.

Frequency counter setup

The next step is to setup **counter 1**. We have for all *odd* counters (CNT1, CNT3 and CNT5) *additional mode* available - *frequency*. This mode will output on this counter *exact frequency* of the *signal*, which will have *no* phase shift to the input and will have an *update rate* same as the *input signal*.



1. Select the **Counter mode** as 'Frequency'.
2. Choose the **Signal input** as *internal count signal* (e.g. **Internal_CNT0**), which tells the system that the *special super-counter* mode will be used since the *resolution* of frequency measurement is limited by *internal clock*, not the sample rate.

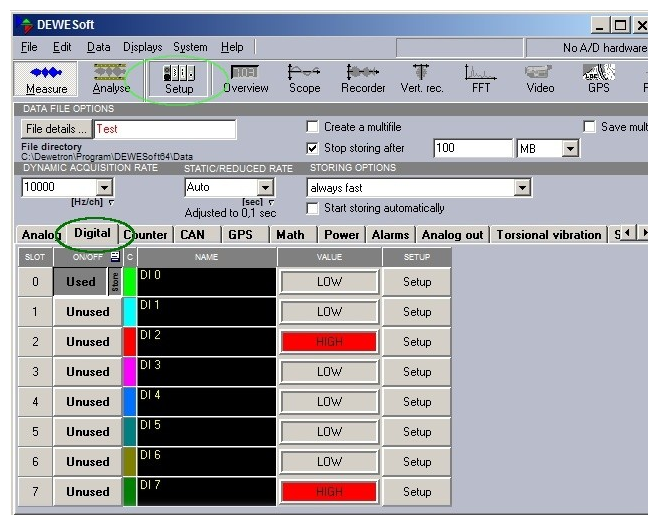
3. Define the **Scaling** factor of the *frequency output*.

When we set up the counter **1** as this, also the counter **0** will *change* its operation. It will not be a simple counter anymore, but it will *output* the values interpolated in between the *samples*.

1.4 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.

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**.

1.5 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	Orion, NI, Softing or Vector CAN
Setup sample rate	At least 1 kHz

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

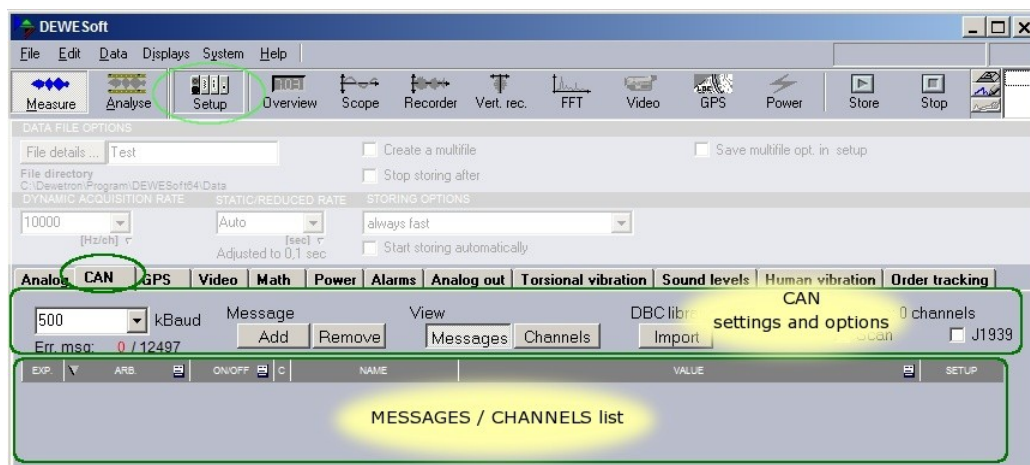
- setting common **CAN settings/options**
- create new:
 - **Import from existing .dbc file**
 - **Add manually** with **Can channel setup** procedure for **Message / Signal setup**
 - **Use scan mode to find automatically**
- change existing:
 - **Edit**
 - **Remove**
- **Select for storage**

for hints about **CAN bus Acquisition** see → [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**

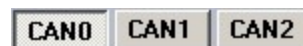
All this work can be done with *settings* and *options* on this part of **CAN Setup** screen.

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 → [Installation Guide](#) → **CAN Hardware setup**

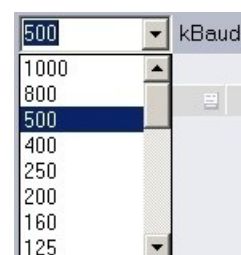
CAN speed- baud rate

The baud rate setting② 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.



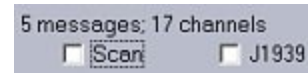
Import messages / channels from existing .dbc file

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](#)).

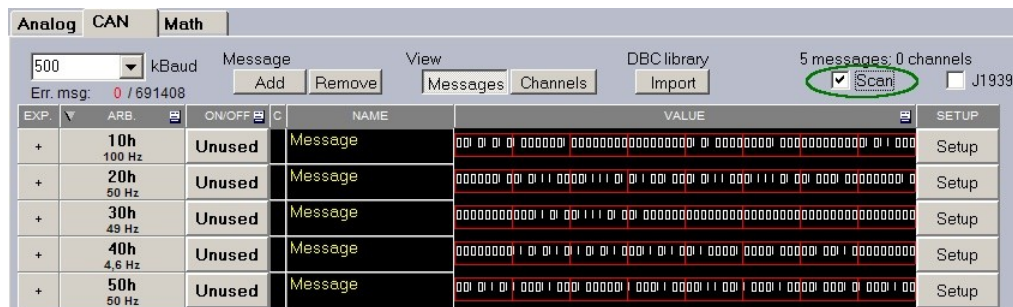
You can *edit* the messages and channels now also *manually*.

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.*

Therefore it is possible that it may take hours, days or even weeks before you get the whole message list - because sensors may send *only* error 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).

J1939 support

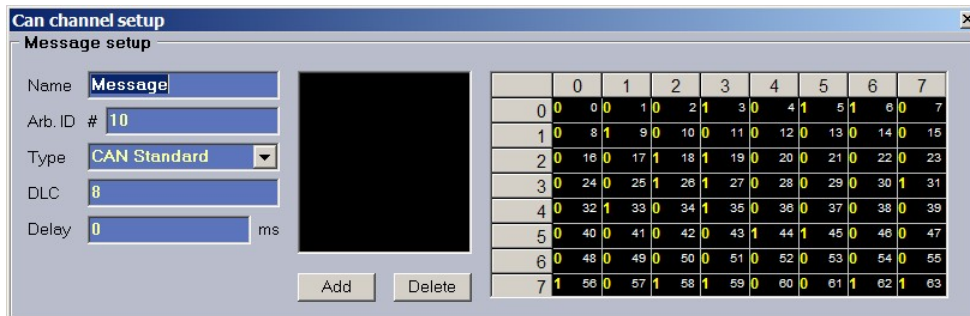
J1939 is used to enable special decoding of *arbitration ID* which includes the *sender*, *receiver* and the *message ID* itself. Arb ID is always extended in this case. This is most widely used on trucks. Please make sure that the bus type is really J1939 before enabling this option.

1.5.1 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** to **remove** the *selected* channel.

Can channel setup

Message setup

Name:

Arb. ID: #

Type:

DLC:

Delay: ms

Signal setup

Name:

Data format:

Data type:

Unit:

Color:

Start bit:

Length [bits]:

Factor:

Offset:

Maximum:

Minimum:

Current unscaled value: **136**

Current scaled value: **4**

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* within the message*
- Length [bits]** ④ *this value defines the **number** of bits used 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.*

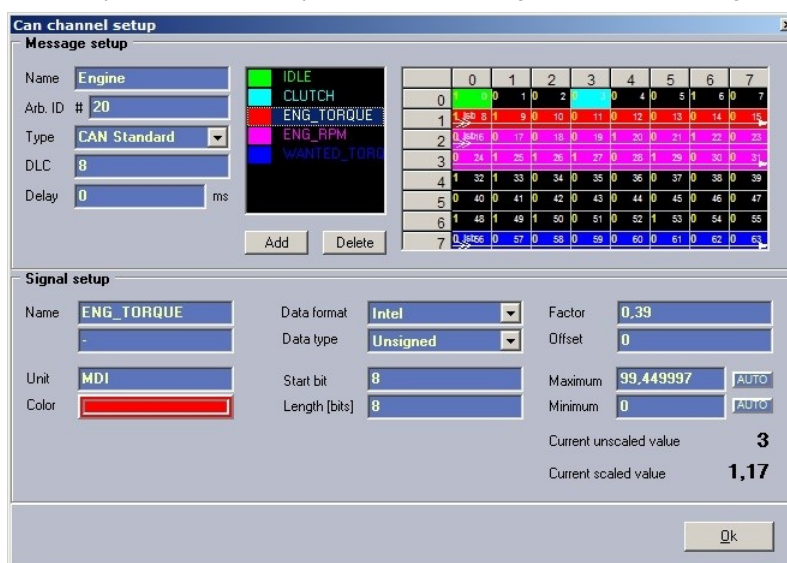
In addition to the signal settings, you have an *online display* of **Current unscaled value** and **Current scaled value** in lower right part of **Signal setup** area.

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:

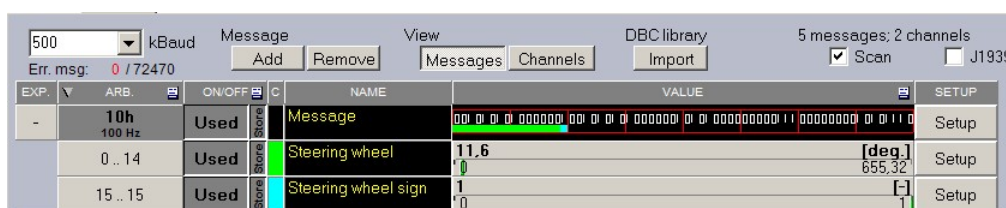


In this fields array the running number of bit is displayed on right side of field. **Start bit** is designated with *double* arrow, *end* bit with *simple* arrow and both arrows are connected with white line, which represent **Length** of signal.

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** - **Remove** button.

Message / channel list

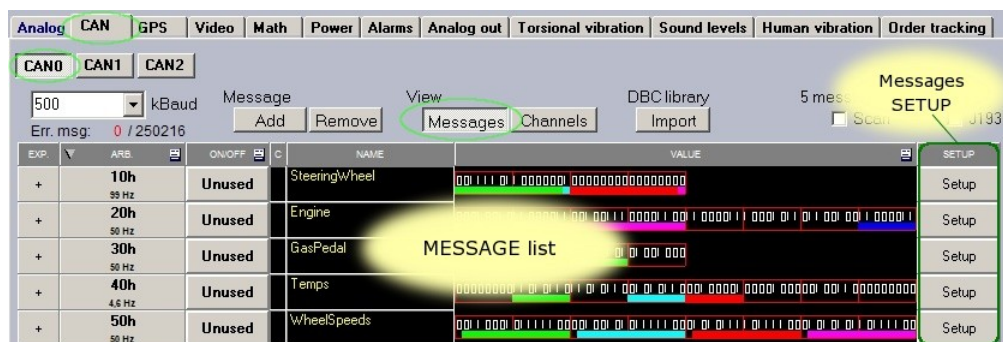
After you have defined *all signals (channels)* within *all messages* or you have import messages from *.dbc file, all this messages appear on **Message list** or **Channel list** (channel overview) on lower area of *CAN setup window*.

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

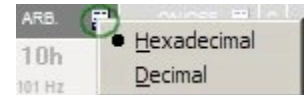
Use the **+** button in this column to **display all channels** of this *message* and channel list will have the following appearance:

EXP.	V	ARB	ON/OFF	C	NAME	VALUE	SETUP
-		10h 199 Hz	Unused		SteeringWheel	00 111 01 000000000000000000 superior message ←	Setup
		0 .. 14	Unused		STWH_ANGLE	19.42 [deg.]	Setup
		15 .. 15	Unused		STWH_SIGN	0	Setup
		16 .. 30	Unused		STWH_D_ANGLE	0 CHANNEL list of superior message	Setup
		31 .. 31	Unused		STWH_D_SIGN	0 ' SteeringWheel '	Setup
+		20h 49 Hz	Unused		Engine	000000 0000000000 01 01 01 01 00000000000000000000 0000 000000000000	Setup
+		30h 99 Hz	Unused		GasPedal	00000000000000000000 MESSAGE list	Setup
+		40h 5.5 Hz	Unused		Temps	000000 000000 000000 0000 Messages SETUP	Setup
+		50h 99 Hz	Unused		WheelSpeeds	0001 00 0111 0011 00000000 00 0001 00 0001 0001 0001 000000 111 00	Setup

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.



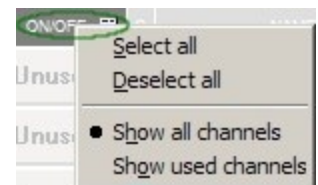
ON/OFF

- 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 _____

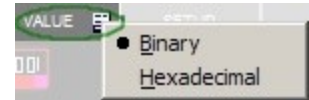
- CAN message (channels) name - just click into the **NAME** field and you can enter the new **name** as usual text.

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

NAME	VALUE
SteeringWheel	54 B1 AC B2
Engine	41 22 CC D 22 0 20 23
GasPedal	0 5A 0 0

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](#)

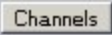
for information about **C** and **Name** column also see → [User Guide](#) → [Common channel setup](#)

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  **View** button **list** of defined **CAN channels** appear:



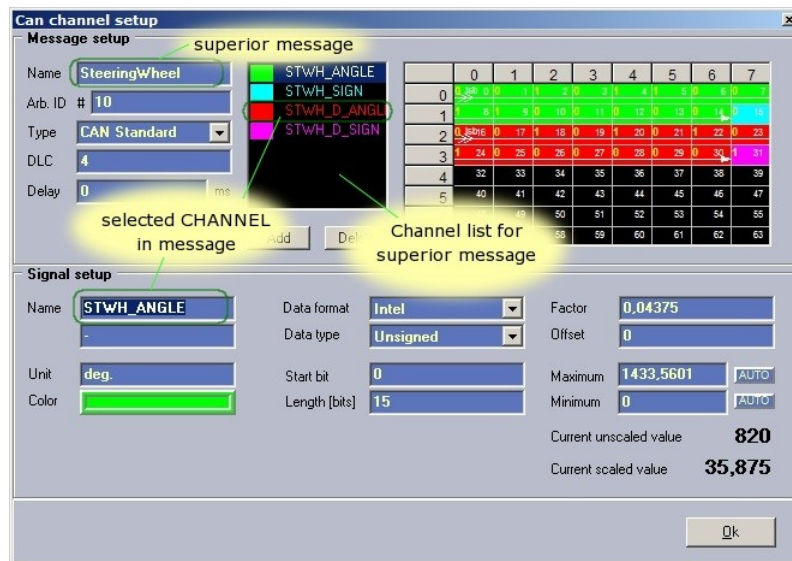
Columns that are shown in your **CAN channels list** are similar as in **Messages view**, except:

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 below)

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**

1.6 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	Dewetron 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](#)

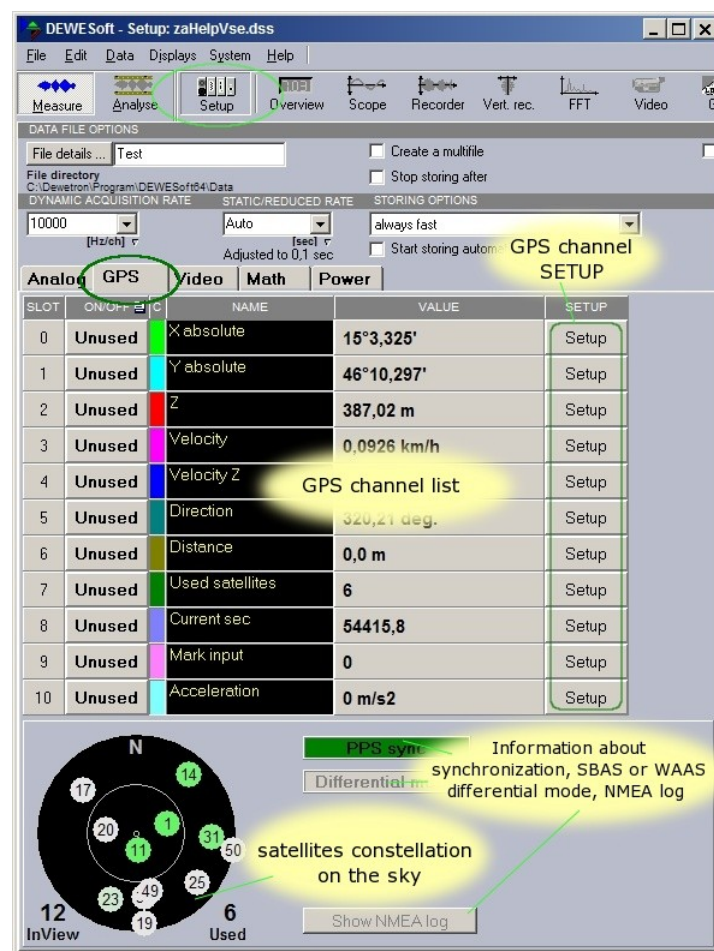
When you select **GPS** tab on **DEWESoft Setup** screen, on lower part of screen **GPS settings** will appear:

GPS channel list

with information about GPS channel Slot number, On / Off state, Name, Value column and primary **Setup** 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 channel list

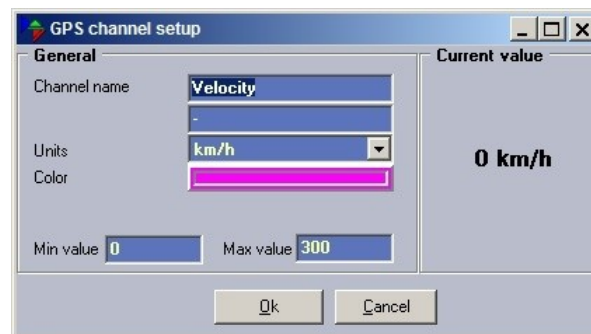
First let's look at the columns that are shown in *GPS channel list*, and describe what each is for:

- 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.

1.7 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 and Basler/Dewecam
Setup sample rate	At least 1 kHz

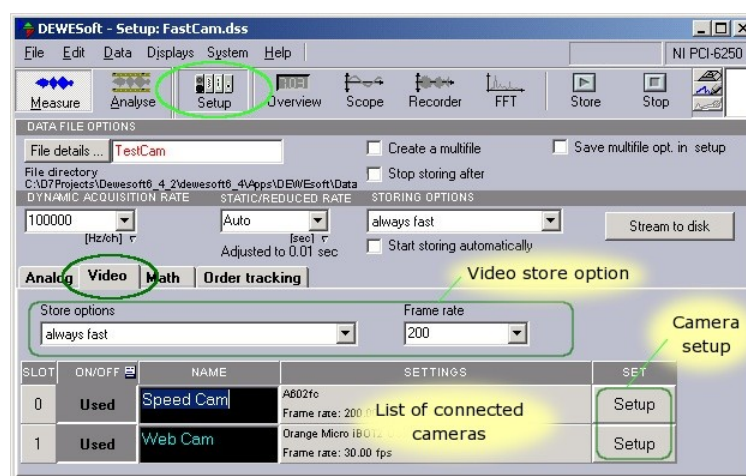
for hints about **Video Acquisition** see → [DEWESoft Tutorials](#)

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 below)

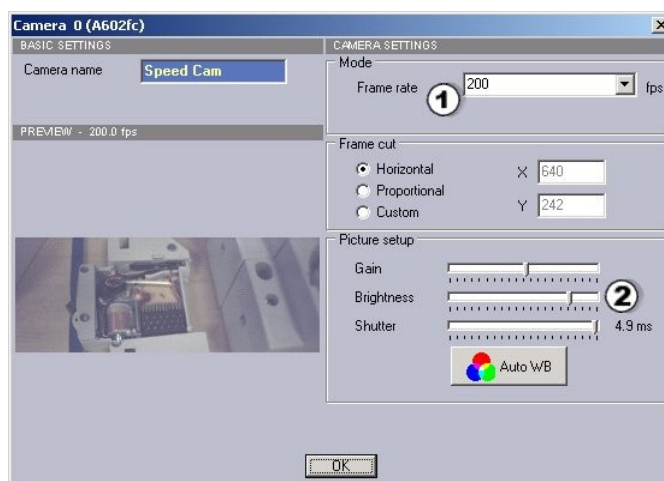
for information about **Slot**, **On/Off** and **Name** column see → [User Guide](#) → **Channel setup**

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...

1. Basler (high speed) camera setup



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

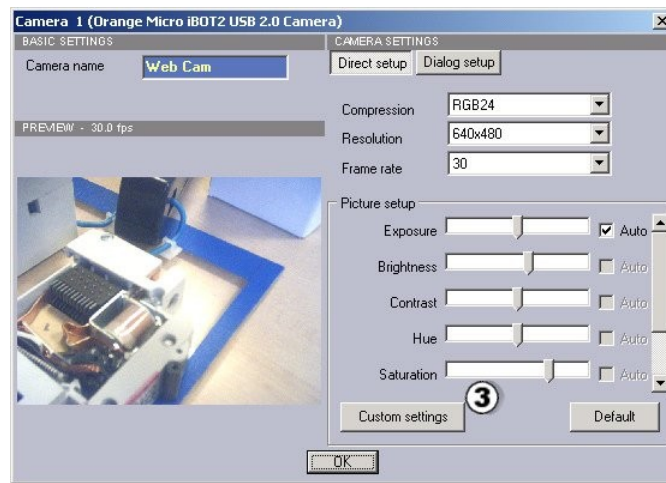
Frame cut when we select higher *frame rate*, the *resolution* can't be 640x480 (VGA), but it is **reduced** (e.g. by 200 fps is reduced to 640x242); with selecting **Horizontal**, **Proportional** or **Custom**, we can choose how the camera cuts pictures. The only way to *increase* the frame rate is to *reduce* the *height* of the picture (reducing the width will not increase maximum frame rate).

Picture setup **Shutter** speed② 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.

2. directX (web cams and handy cams) *camera setup*



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. Dewetron is all the time testing new cameras on the market, so it is worth asking them 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.

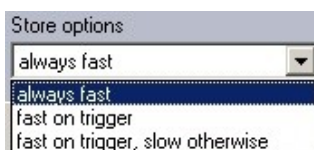
After you have done your settings press **OK** to leave the camera setup.

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*:

SLOT	ON/OFF 	NAME	
0	Used	Speed Cam	A602fc Frame rate: 200.0
1	Used	Web Cam	Orange Micro iBC Frame rate: 30.0

Store options

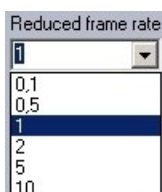
DEWESoft offers three different ways - Storing strategies to **store** your **video data**. Select option from drop-down list:



- **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.

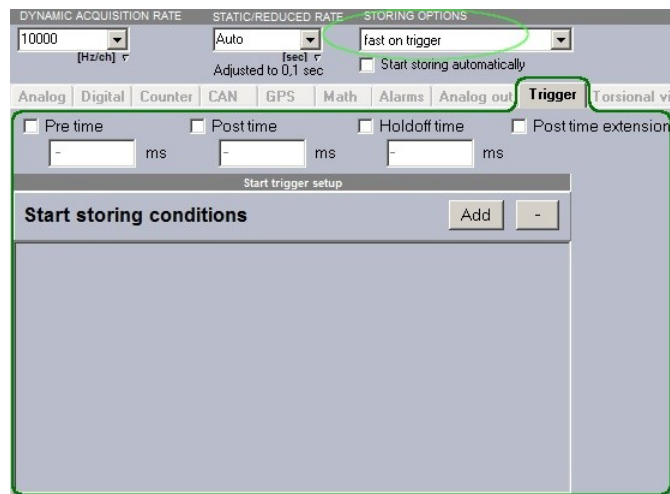
- **fast on trigger** the video data will only be **stored** with the *full image acquisition rate* when **trigger** condition is *true*
- **fast on trigger, slow otherwise** to be able to acquire data with two speeds: **stores** data with a *reduced image acquisition rate* until the trigger condition *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**

1.8 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:

- With just **two input channels** connected to your instrument (voltage and current *inputs*) and with the **DEWESoft power module** functions not only a lots of *parameters* can be **calculated** but also *many channels* can be produced.
- **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
 - Symmetrical Components (positive, negative and zero sequence components)
 - Cos Phi, Power factor
 - Period values (1/2 cycle, cycle,...)
 - P, Q, cos Phi for each *harmonic*
- 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.

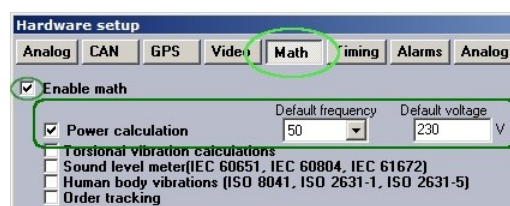
To measure **Power**, we need at least two *input channels* connected to your instrument for measuring voltage and current.

Required hardware	Any AD card, DAQP-DMM or DAQP-HV, DAQP-V or MDAQ-V or PQL print
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](#)

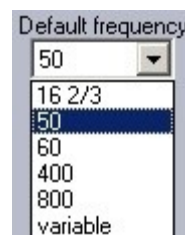
for information **Power module** see → [DEWETRON DEWESoft Power Software User Manual](#)

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.

SLOT	ON/OFF	C	NAME	AMPLIFIER (G14)	PHYSICAL VALUES	CAL	SETUP
0	Used	U	DAQP-DMM	400 V ... 20 kHz	U [V]	Zero	Set ch. 0
1	Used	I	DAQP-V	10 V ... 50 kHz	I [A]	Zero	Set ch. 1

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. Settings on **Module** section depend on used DEWETRON amplifier (see above → '*Required hardware*').

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. In most cases it is more than enough to measure the **voltage** directly with Dewetron amplifier.

Example: Since the *input voltage* of DAQP-DMM is ± 1000 V, the direct low voltage grid can be measured *directly* without any need of additional transducers.

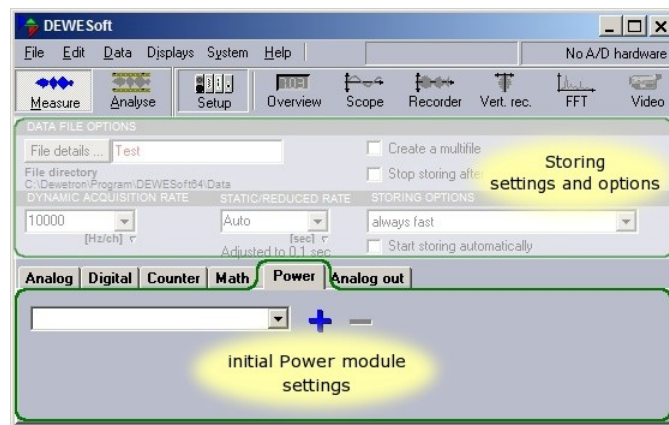
It is very important that the *measurement Range* is chosen *according* to the expected signal.

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.

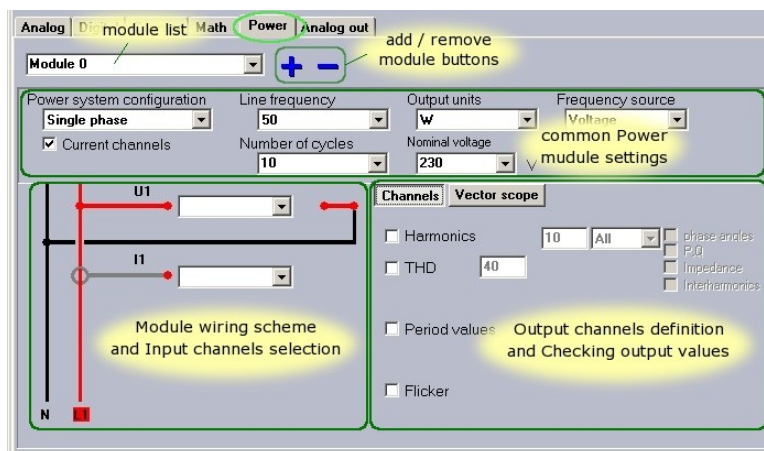
WARNING *Think well before using this command, while we can't return deleted module not at all!*

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:

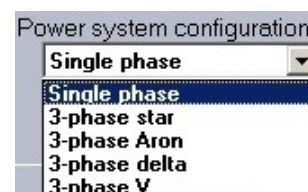


This screen has the following main sections:

- **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

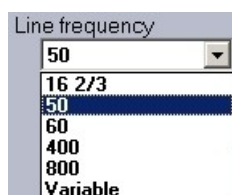
On this section can be defined settings:

- **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**



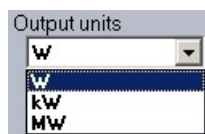
Line frequency depends on the grid frequency. We can have $16 \frac{2}{3}$ (in trains), 50 Hz (Europe), 60 Hz (USA); 400 and 800 Hz (vehicle power systems) or Variable (for frequency converters).

- **Number of cycles**



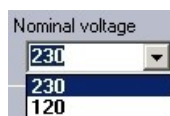
Choose from drop down list. Standard values are 10 for 50 Hz grids and 12 for 60 Hz grids. This basically defines the *measurement interval*.

- **Output units**



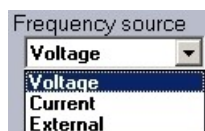
Choose *power unit* for your system from drop down list.

- **Nominal voltage**



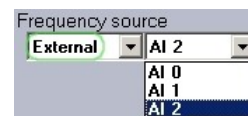
Choose *nominal voltage* of your system from drop down list.

- **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

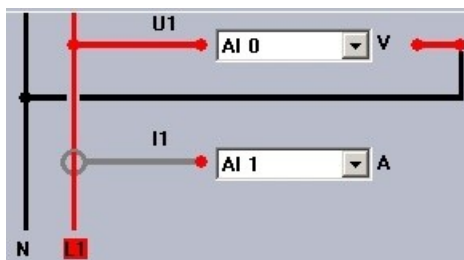
On this left lower part of the **DEWESoft Power Setup** screen *principle module wiring scheme* with *input channels* selection fields are displayed.

Module wiring scheme

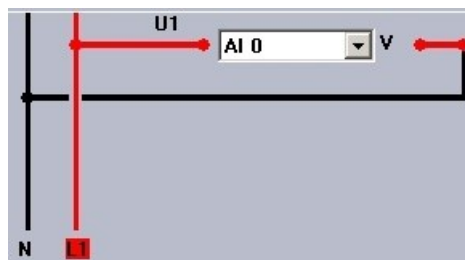
This scheme depends from selection in **Power system configuration** and **Current channels** field:

- **Single phase**

Current channels choice = *enabled* (default)

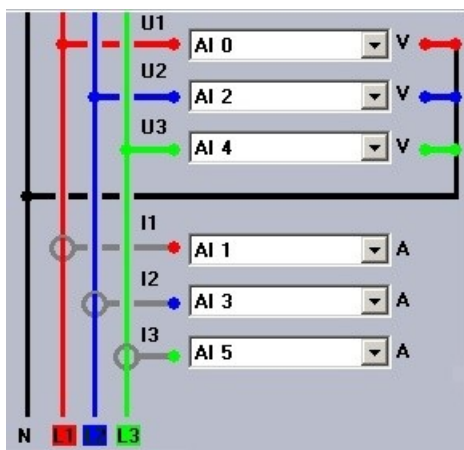


Current channels choice = *disabled*

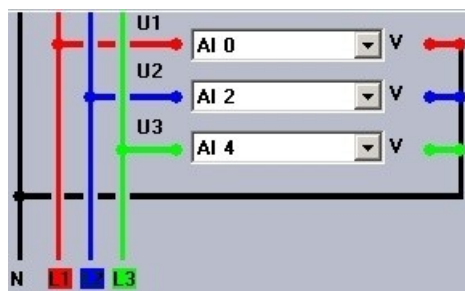


- **3 - Phase star**

Current channels choice = *enabled* (default)

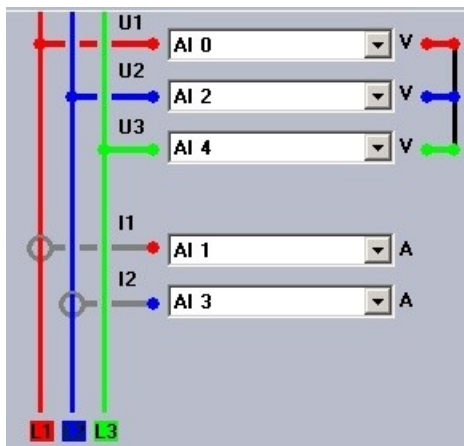


Current channels choice = *disabled*

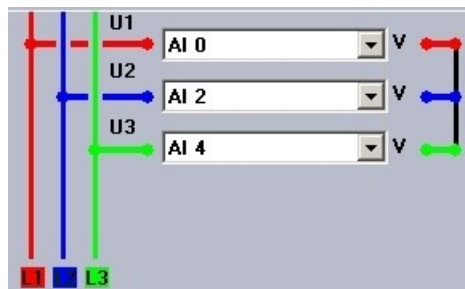


- **3 - phase Aron**

Current channels choice = *enabled* (default)

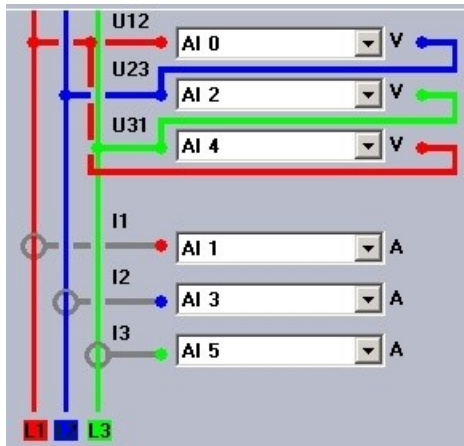


Current channels choice = *disabled*

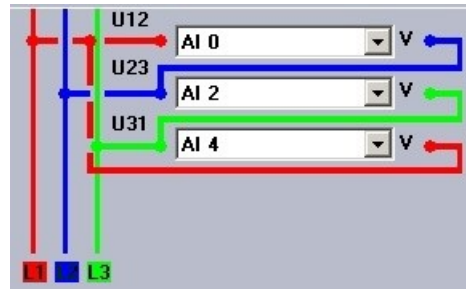


• 3 - phase delta

Current channels choice = *enabled* (default)

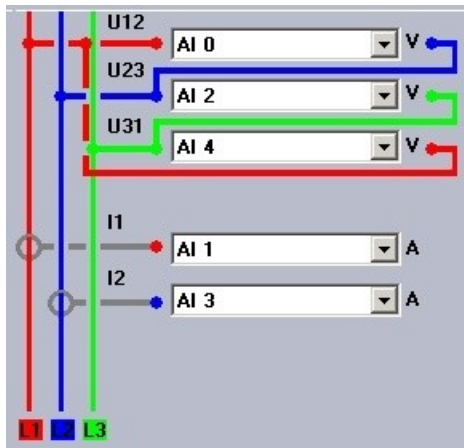


Current channels choice = *disabled*

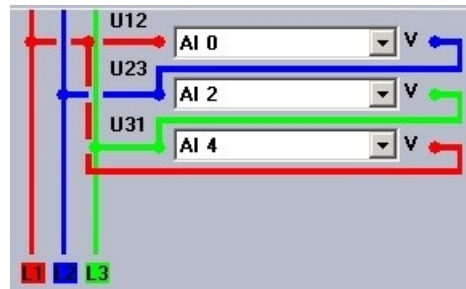


• 3 - phase V

Current channels choice = *enabled* (default)

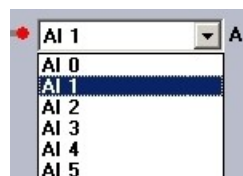
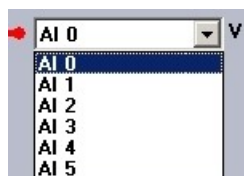


Current channels choice = *disabled*



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:



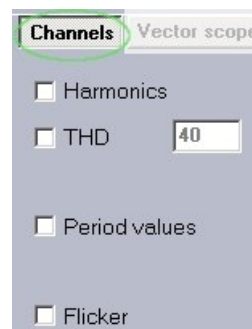
Output channels definition

When **Channels** 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

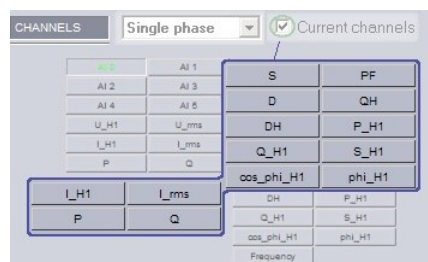
DEWESoft automatically create following *output channels*:

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:

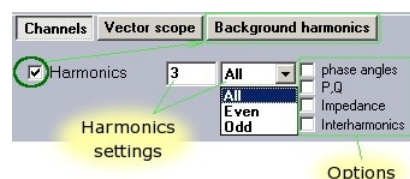
- *voltage* and *current channels* (**U** and **I**)
- *power channels* (**P**, **Q**, **S**, **phi**, **cos phi**)

2. **Harmonics** 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*.

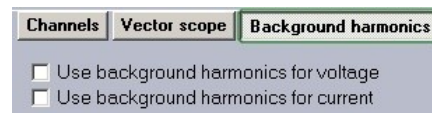
When the **Harmonics** option is checked we can:

- 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)



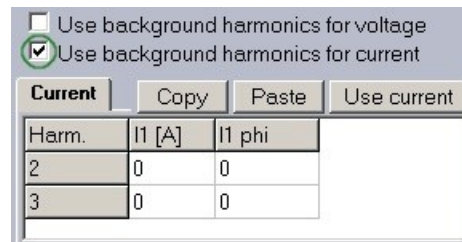
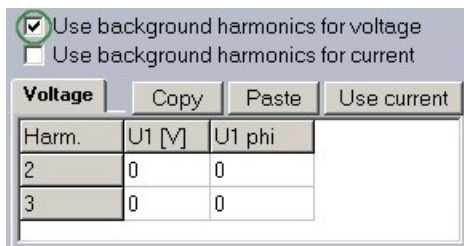
Background harmonics

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

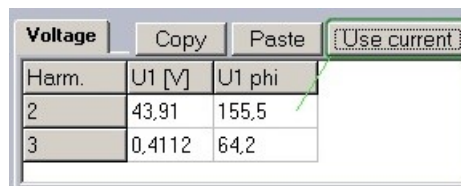


When one from this options (or both) is checked, table with *voltage* (U1 in V unit and U1 phi - phase angle) / *current values* (I1 in A unit and I1 phi - phase angle) for defined *number - 1* background harmonics appear.

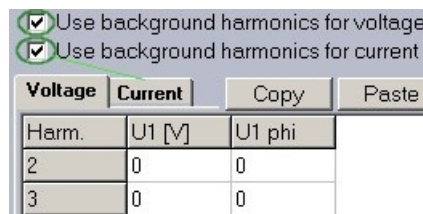
In example above we have selected 3 harmonics, therefore voltage / current value for *second* and *3rd* harmonics can be defined.

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)



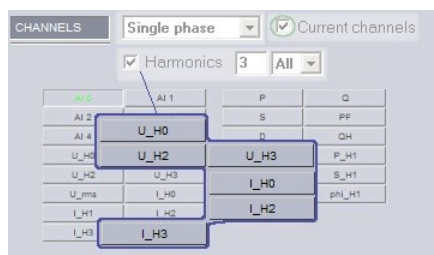
When both options for *background harmonics* are checked, table with *current values* appear in *separate tab* (see right).



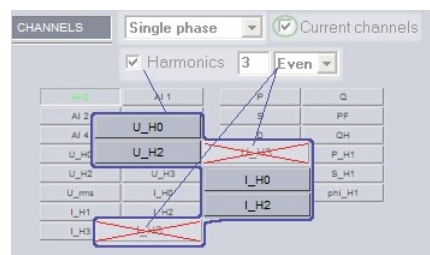
DEWESoft create following *harmonics output channels* for **Single phase** Power module

(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.



To basic channels for **Even** Harmonics (see **above**) *emphasized* channels are added → *current* and *power harmonics* channels.



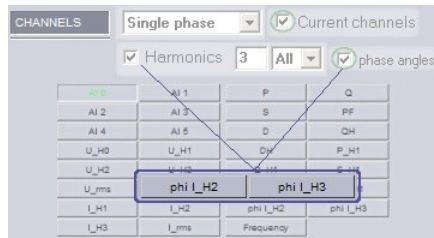
When **Current channels** choice = *disabled*, *current* channels are not created (e.g. above only U_... channels).

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

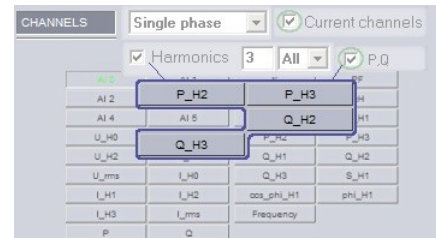
To basic channels for **All** Harmonics (see above left) *emphasized phase angles* **phi** channels for each *harmonic* are added:



For **Even** Harmonics *odd* channels are not created (except ..._H1 → see note above).

b) **P,Q** option is checked

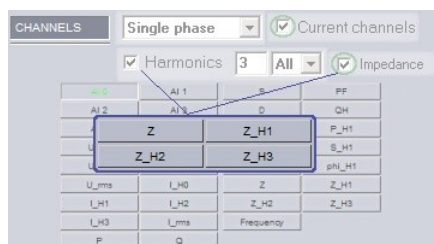
To basic channels for **All** Harmonics (see above left) *emphasized P - active power* and *Q - reactive power* channels for each *harmonic* are added:



For **Even** Harmonics *odd* channels are not created (except ..._H1 → see note above).

c) **Impedance** option is checked

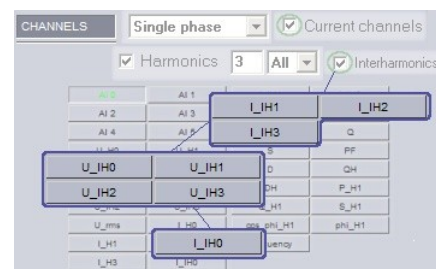
To basic channels for **All** Harmonics (see above left) *emphasized Impedance* **Z** channels are added



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.

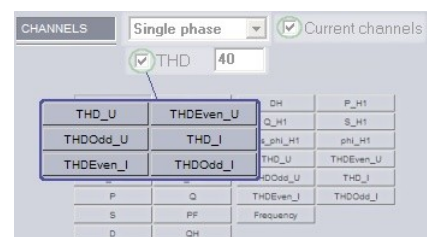
3. **THD** option

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*).



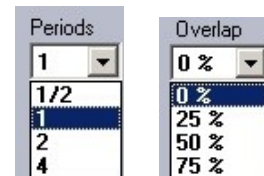
4. Period values option

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

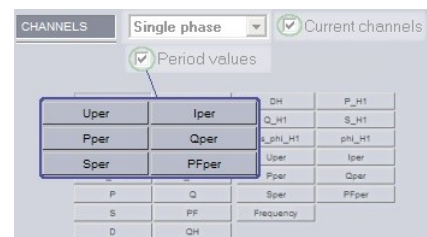
This is very helpful for *triggering*.

for information about **Overlap** see → [User Guide](#) → **FFT instrument settings**



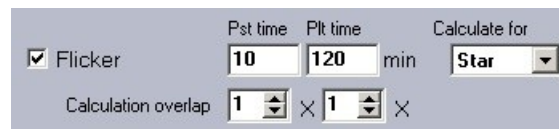
To create *period values output channels* this option must be checked.

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*.



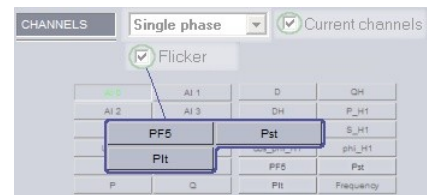
For this option we can set:

- 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.

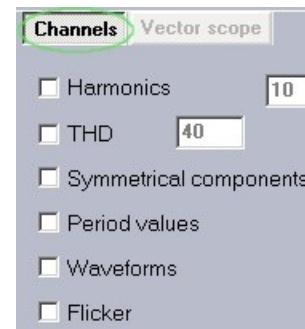
To basic channels (see [above](#)) *emphasized flicker (powers)* P... channels are added.



• **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**



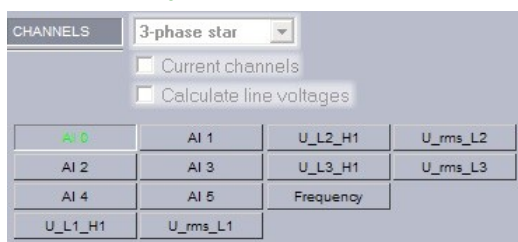
NOTE: Following description and examples to be valid for all types 3 phase systems:
3-phase star, 3-phase Aron, 3-phase delta and 3-phase V.

1. **basic** output channels → no option are checked

DEWESoft automatically create following *output channels*:

Current channels choice = *disabled*

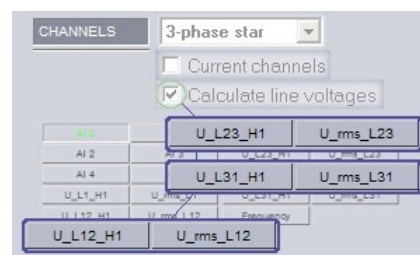
Calculate line voltages choice = *disabled*



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 = *disabled*

Calculate line voltages choice = *enabled*

CHANNELS					
3 phase star					
☒ Current channels ☐ Calculate line voltages					
AI 0	AI 1	I_L2_H1	I_rms_L2	QH_L3	DH_L3
AI 2	AI 3	P_L2	Q_L2	P_L3_H1	Q_L3_H1
AI 4	AI 5	S_L2	PF_L2	S_L3_H1	phi U_L3_H1
U_L1_H1	U_rms_L1	D_L2	QH_L2	cos_phi_L3_H1	phi_L3_H1
I_L1_H1	I_rms_L1	DH_L2	P_L2_H1	P	Q
P_L1	Q_L1	Q_L2_H1	S_L2_H1	S	PF
S_L1	PF_L1	phi U_L2_H1	cos_phi_L2_H1	D	QH
D_L1	QH_L1	phi_L2_H1	U_L3_H1	DH	P_H1
DH_L1	P_L1_H1	U_rms_L3	I_L3_H1	Q_H1	S_H1
Q_L1_H1	S_L1_H1	I_rms_L3	P_L3	Frequency	
cos_phi_L1_H1	phi_L1_H1	Q_L3	S_L3		
U_L2_H1	U_rms_L2	PF_L3	D_L3		

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.

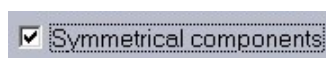
3. **THD** output channels → **THD** option is checked

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 - **L3** phase 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.

CHANNELS					
3-phase star					
☒ Current channels ☒ Symmetrical components					
☐ Calculate line voltages					
AI 0	AI 1	S_L2	PF_L2	P	Q
AI 2				S	PF
AI 4				DH	QH
U_L1_H1	U			Q_H1	S_H1
I_L1_H1	I	u2	u0	u2	u0
P_L1		u2_1	u0_1	u2_1	u0_1
S_L1		U_0	U_1	U_2	U_1
Q_L1		U_2	i2	U_2	i2
DH_L1	P	i0	i2_1	i0	i2_1
Q_L1_H1	S	i0_1	I_0	i0_1	I_0
cos_phi_L1_H1	PF	I_1	I_2	I_1	I_2
U_L2_H1	U				
I_L2_H1	I				
P_L2	P				
Q_L2	Q				
cos_phi_L2_H1	PF				
phi_U_L3_H1					
phi_L3_H1					
Frequency					

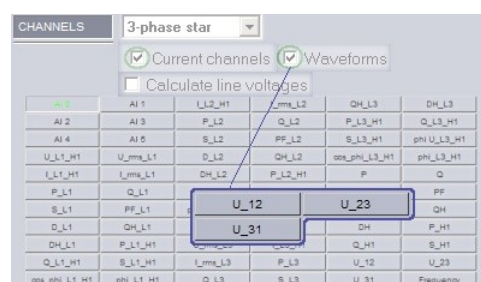
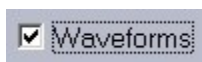
5. Period values output channels → **Period values** option is checked

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 this option:



7. Flicker output channels → **Flicker** option is checked

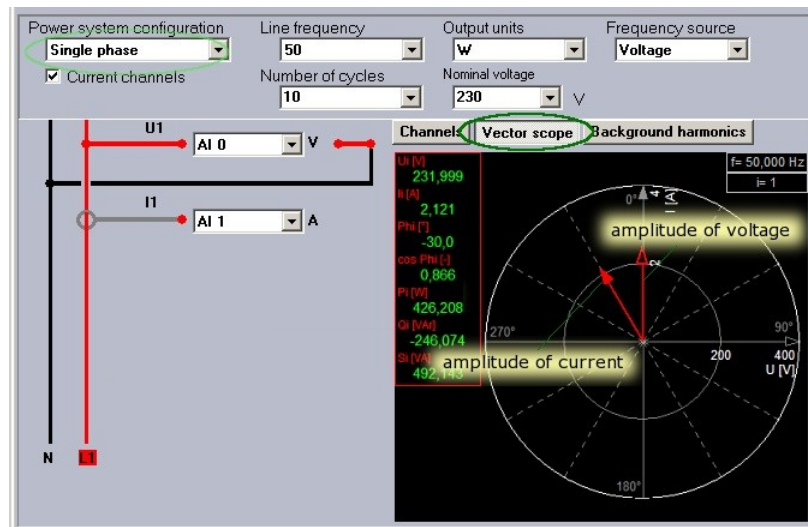
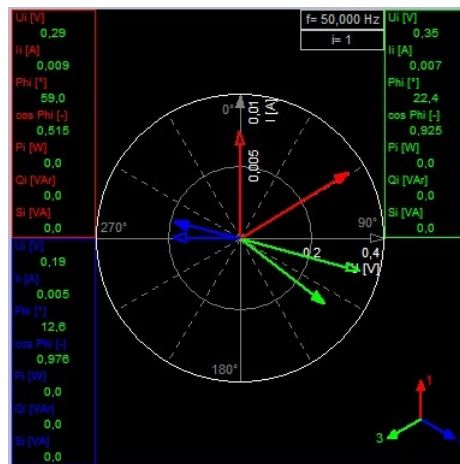
For information about this option *settings* and *output channels* → see above → **Single phase Flicker option**

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** - **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

→ [DEWETRON DEWESoft Power Software Reference Manual](#)

→ [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.

The current *converters* have the *voltage output*, so we measure this with Dewetron DAQP-V, DAQP-V-A or MDAQ-V module.

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*.

For background information about **Power and power quality measurements** → see book →

[Power quality the new challenge](#) by W. Schoeffler, published by Dewetron

1.9 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 [Installation Guide](#) → **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**



– **Filters** →

- **IIR filter**
- **FIR filter**
- **FFT Filter**



– **Dynamic signal analysis section** →

- **CA Noise**
- **Exact frequency**
- **Angle sensor**

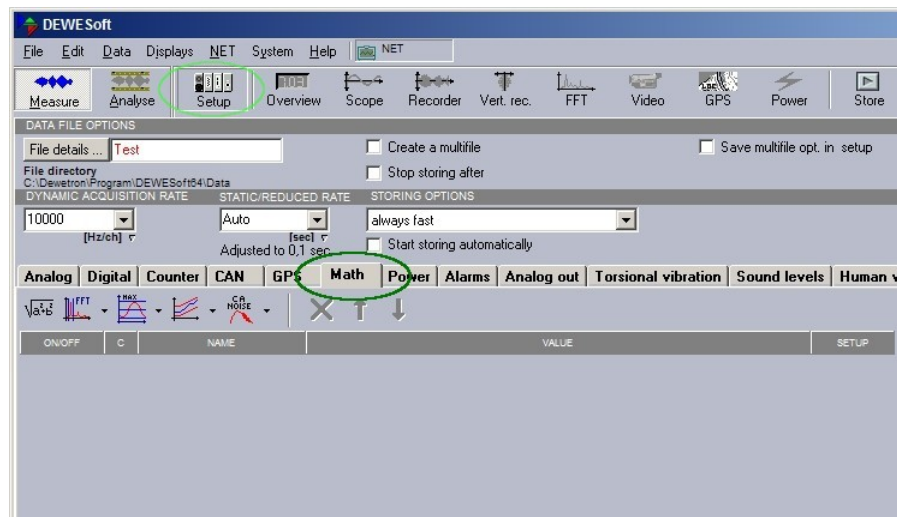


– **Statistics** →

- **Basic statistics**
- **Latch math**

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.

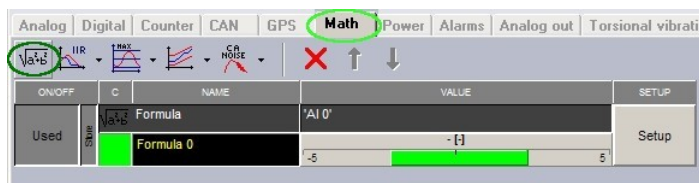
Moving the modules up and down *doesn't change* the *calculation order*. **DEWESoft** automatically *detects* the **references** and **orders** the mathematic channels. So in general we can first select a *filter* on *formula* channel and after that *define* this formula channel. **DEWESoft** will then 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.

1.9.1 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.

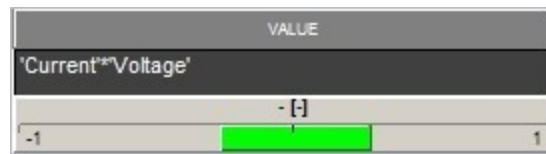


In second channel row of **Name** column the **name** of the output channel created with the formula is displayed.

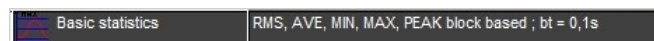


VALUE

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:



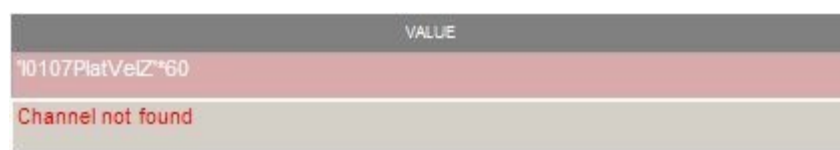
In second channel row of **Value** column the **live preview** of calculated value is displayed.

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.

Basic statistics	RMS, AVE, MIN, MAX, PEAK block based ; bt = 0,1s		
Voltage/RMS	0	- [-]	2
Voltage/AVE	-5	- [-]	5
Voltage/MIN	-5	- [-]	5
Voltage/MAX	-5	- [-]	5
Voltage/PEAK	0	- [-]	2

If there is an **error** in the math module, the **error caption** will appear:



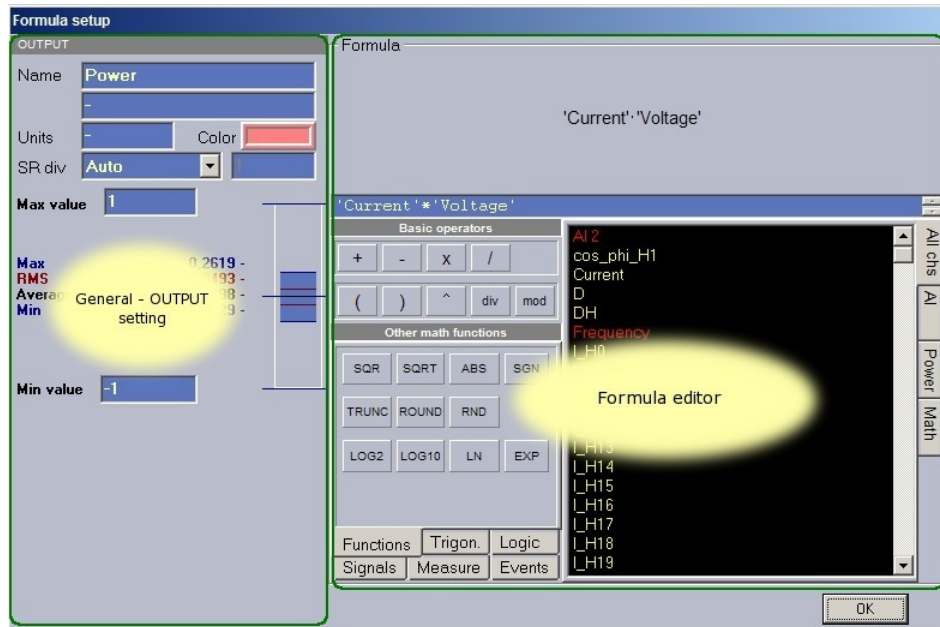
There are several possible **errors**:

- 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 **de selected** 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**

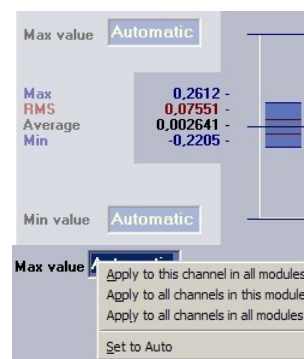
Name The *first* line of channel name holds the *name* of the *output channel*. This name will appear in the channel setup list and in the channel selector for the display.

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 $V \cdot A = W$. These units must be **defined** by the *user*.

Min val The user scale *Minimum* and *Maximum* values can be set here.

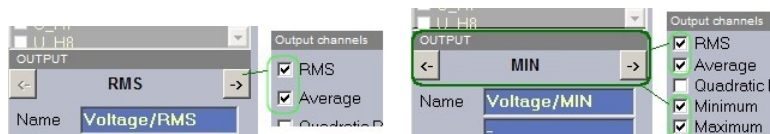
Max val Minimum and maximum scale is used in the *display* as the default and full display range. The maximum and minimum can be set *automatically* or *manually* by entering the value in the fields. Then the max value is not shown in the bar graph since it is displayed in edit box.



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 <i>specific</i> channel in <i>all</i> modules (like for all RMS values in statistic module). |
| Apply to all channels in this module | Applies the settings for <i>all</i> 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 <i>current</i> setting for <i>all</i> output channels. |
| Set to Auto | Sets back the value to <i>Automatic</i> . |

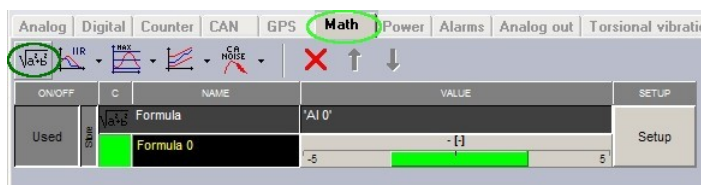
Modules like **filters** (with *more input* channels) or **statistics** can have *more* output channels from the mathematic module. In this case the upper left section shows chosen *input channels*, lower left part shows the *general output channel settings*, where the only difference is that we have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



1.9.2 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*.

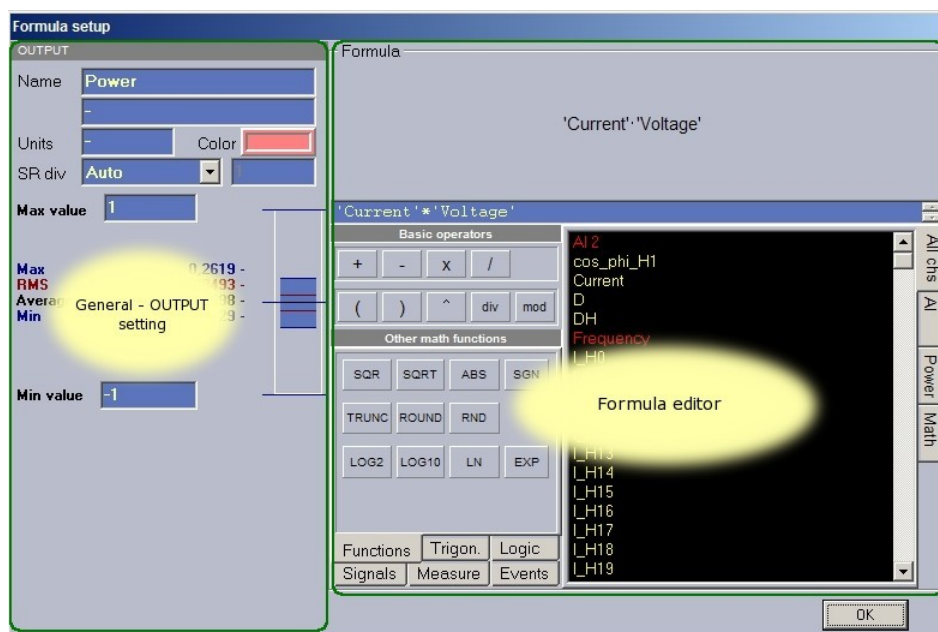
When you choose on the **DEWESoft Math setup** screen button with  - formula icon, a *formula Math channel* is displayed:



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:

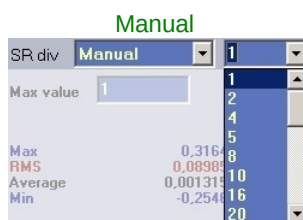
- If the input is synchronous channel with *Sr div* of 4, the resulting channel will also have a sample rate divider of four.
- If the input is asynchronous channel, the output will also be asynchronous channel.
- If there are more input channels, output will be matched to the input channel with the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

Manual

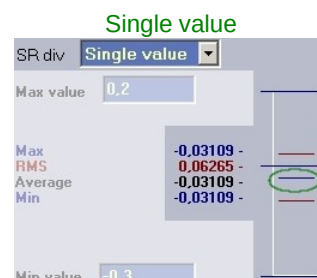
This function will override the logic of automatic *Sr dDiv* assignment and will set the sample rate divider *manually*. This is useful to make *interpolation* of *input asynchronous channel*, because each asynchronous channel is *interpolated* before it is calculated. This gives a smoother curve and 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*.



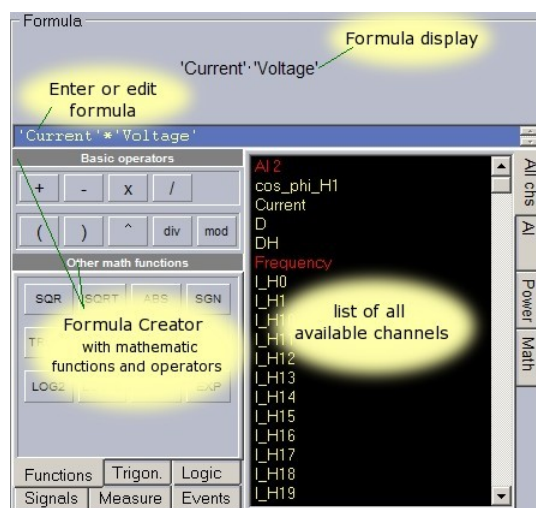
From list we can select **Manual** values from 1 to 500.



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**

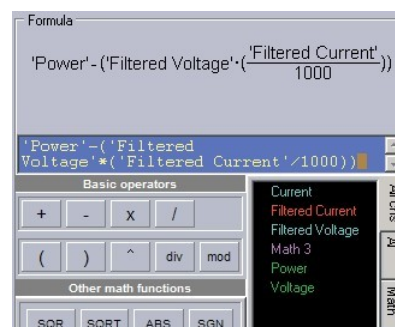


To **enter** a channel in the formula just:

- *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 right).

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

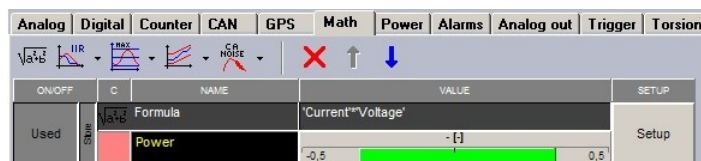
DEWESoft allows *different* types of mathematics, which are grouped in six tabs. Please learn more about these functions in → **Operators and mathematical function**.

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 page to have a complete list of available **operators and mathematical functions**.

1.9.2.1 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**

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:

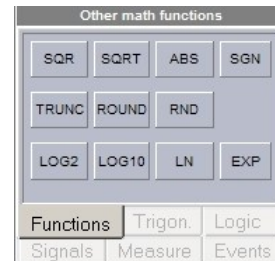
General mathematics		
Function	Syntax	Description
+	<i>expression1 + expression2</i>	<i>expression1</i> plus <i>expression2</i>
-	<i>expression1 - expression2</i>	<i>expression1</i> minus <i>expression2</i>
×	<i>expression1 * expression2</i>	multiplies <i>expression1</i> by <i>expression2</i>
/	<i>expression1 / expression2</i>	divides <i>expression1</i> by <i>expression2</i>
()	<i>(expression)</i>	brackets <i>expression</i>
^	<i>expression1 ^ expression2</i>	raises <i>expression1</i> to the power of <i>expression2</i>

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

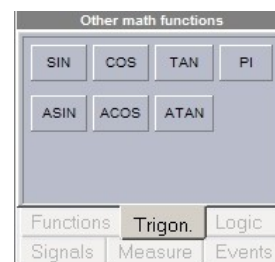
The **Functions** tab contains *additional* mathematic functions.



Functions		
Function	Syntax	Description
SQR	$\text{SQR}(\text{expression})$	square
SQRT	$\text{SQRT}(\text{expression})$	square root
ABS	$\text{ABS}(\text{expression})$	absolute value of <i>expression</i>
SGN	$\text{SGN}(\text{expression})$	sign <i>expression</i> > 0: output = 1 <i>expression</i> = 0: output = 0 <i>expression</i> < 0: output = -1
TRUNC	$\text{TRUNC}(\text{expression})$	truncate to integer <i>expression</i>
ROUND	$\text{ROUND}(\text{expression})$	round <i>expression</i> to nearest whole number
RND	RND	random number between 0 and 1
LOG2	$\text{LOG2}(\text{expression})$	logarithm (base 2)
LOG10	$\text{LOG10}(\text{expression})$	logarithm (base 10)
LN	$\text{LN}(\text{expression})$	natural logarithm
EXP	$\text{EXP}(\text{expression})$	exponent ($e^{\text{expression}}$)

Trigonometry

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

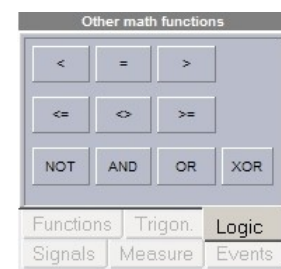


Trigonometric functions		
Function	Syntax	Description
SIN	$\text{SIN}(\text{expression})$	sine of <i>expression</i>
COS	$\text{COS}(\text{expression})$	cosine of <i>expression</i>
TAN	$\text{TAN}(\text{expression})$	tangent of <i>expression</i>
PI	PI	constant pi
ASIN	$\text{ASIN}(\text{expression})$	arcsine of <i>expression</i>
ACOS	$\text{ACOS}(\text{expression})$	arccosine of <i>expression</i>
ATAN	$\text{ATAN}(\text{expression})$ or $\text{ATAN}(x,y)$	arctangent of <i>expression</i> or arctangent of <i>x/y</i>

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](#)

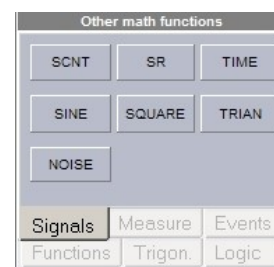


Logic functions		
Function	Syntax	Description
<	$\text{expression1} < \text{expression2}$	if <i>expression1</i> is less than <i>expression2</i> then output is 1, else 0
=	$\text{expression1} = \text{expression2}$	if <i>expression1</i> is equal to <i>expression2</i> then output is 1, else 0
>	$\text{expression1} > \text{expression2}$	if <i>expression1</i> is greater than <i>expression2</i> then output is 1, else 0
<=	$\text{expression1} <= \text{expression2}$	if <i>expression1</i> is less than or equal to <i>expression2</i> then output is 1, else 0
<>	$\text{expression1} <> \text{expression2}$	if <i>expression1</i> is less or greater than <i>expression2</i> then output is 1, else 0
>=	$\text{expression1} >= \text{expression2}$	if <i>expression1</i> is greater than or equal to <i>expression2</i> then output is 1, else 0
NOT	NOT <i>expression</i>	negation; <i>expression</i> = 0: 1; <i>expression</i> = 1: 0;
AND	$\text{expression1 AND expression2}$	logic and 1 AND 1 = 1 1 AND 0 = 0 0 AND 1 = 0 0 AND 0 = 0
OR	$\text{expression1 OR expression2}$	logic or 1 OR 1 = 1 1 OR 0 = 1 0 OR 1 = 1 0 OR 0 = 0
XOR	$\text{expression1 XOR expression2}$	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 <i>sine</i> wave with frequency f [Hz] optionally with phase shift ps [radian]
SQUARE	SQUARE(f [,ps])	Generates a <i>square</i> wave with frequency f [Hz] optionally with phase shift ps [radian]
TRIAN	TRIAN(f [,ps])	Generates a <i>triangle</i> wave with frequency f [Hz] optionally with phase shift ps [radian]
NOISE	NOISE	Generates <i>noise</i> (random number between ± 1)

NOTICE: *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](#)

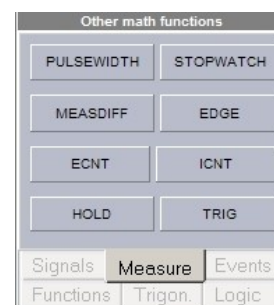
Measure

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](#)

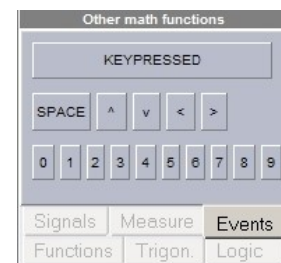


Measurement functions		
Function	Syntax	Description
PULSEWIDTH	PULSEWIDTH(<i>cond</i> [, <i>rearm</i>])	Measures time [s] between two <i>cond</i> edges (<i>cond</i> jumps from 0 to 1), rearm edge optional
STOPWATCH	STOPWATCH(<i>cond1</i> , <i>cond2</i>)	Measures time [s] between <i>cond1</i> and <i>cond2</i> edges (<i>cond</i> jumps from 0 to 1)
MEASDIFF	MEASDIFF(<i>value</i> , <i>cond1</i> , <i>cond2</i>)	Measures <i>value</i> difference between <i>cond1</i> and <i>cond2</i> edges (<i>cond</i> jumps from 0 to 1)
EDGE	EDGE(<i>cond</i> [, <i>rearm</i>])	Returns 1 when the <i>cond</i> changes from 0 to 1 and have again optional rearm conditions
ECNT	ECNT(<i>cond</i>)	Counts number of edges for <i>cond</i> (<i>cond</i> jumps from 0 to 1)
ICNT	ICNT(<i>cond</i>)	Counts all the samples where the <i>cond</i> have a logical value 1
HOLD	HOLD(<i>value</i> , <i>cond</i> [, <i>rearm</i>])	shows and holds value, when <i>cond</i> goes from 0 to 1 with optional rearm condition
TRIG	TRIG	has a value of 1 when a store condition appears

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](#)



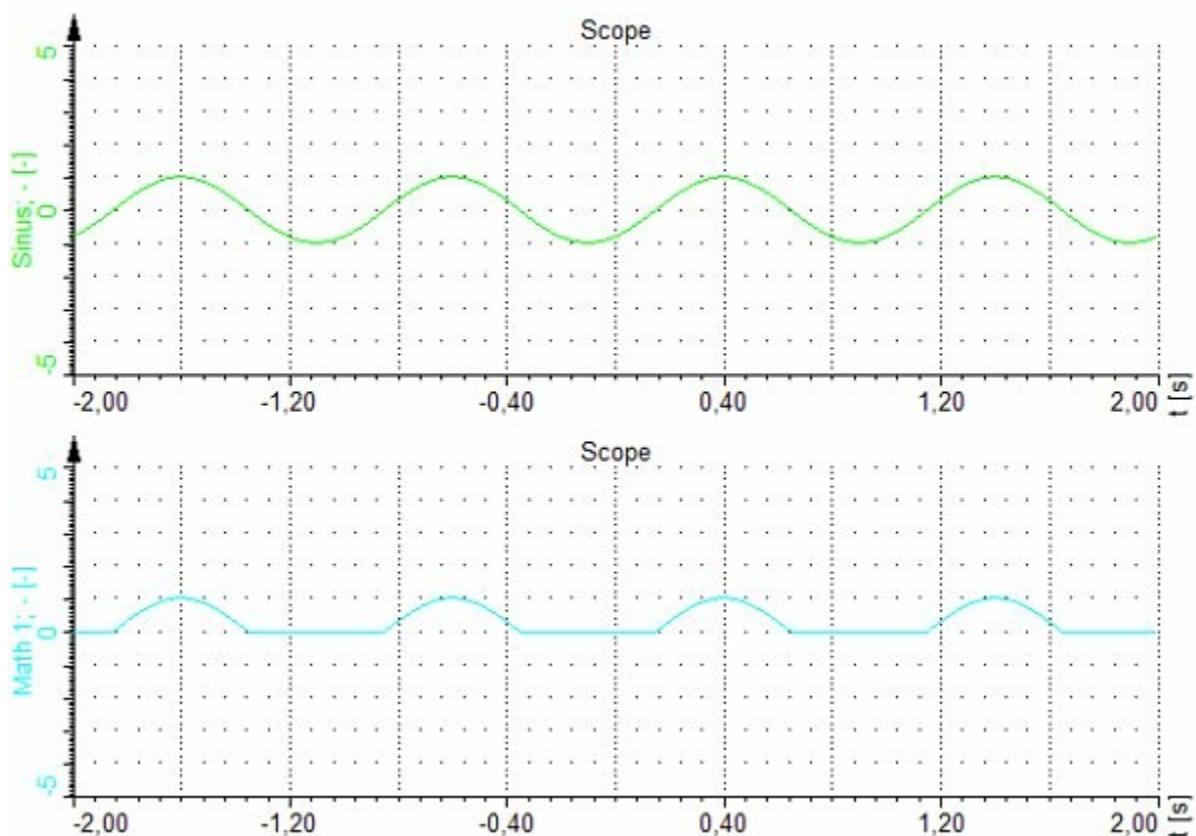
Event functions		
Function	Syntax	Description
KEYPRESSED	KEYPRESSED(<i>ASCII code</i>)	Generates pulse (transition 0 to 1 and back) if key with matching <i>ASCII code</i> is pressed. Example: KEYPRESSED(32) outputs pulse if Space key is pressed.

1.9.2.2 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.

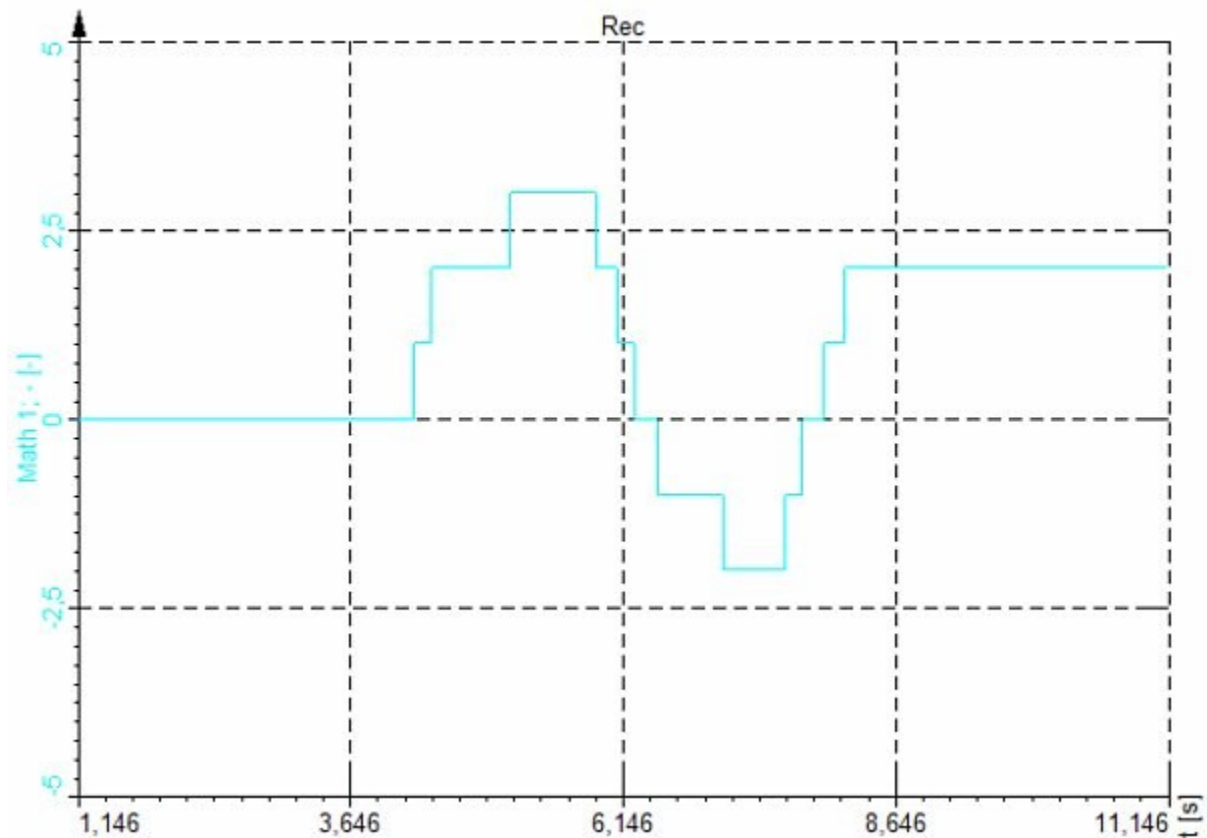


1.9.2.3 Example of input events

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.



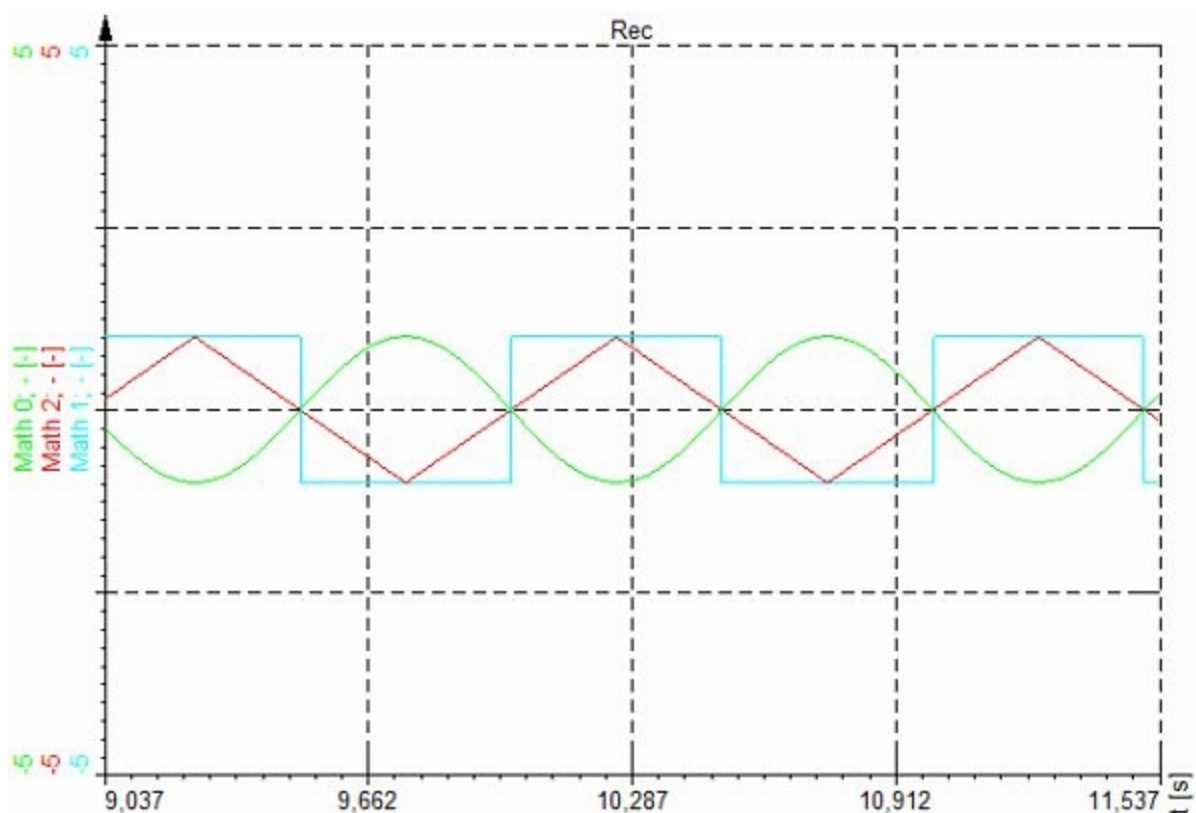
1.9.2.4 Example of signals

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.

<code>sine(1 , Pi)</code>
<code>square(1)</code>
<code>trian(1)</code>

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.

1.9.2.5 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 then measure the *time* between each transition. If we want to measure the *frequency*, we can calculate *inverse* of this function as seen below.

$$\frac{1}{\text{pulsewidth}('F1' > 0)}$$

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



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

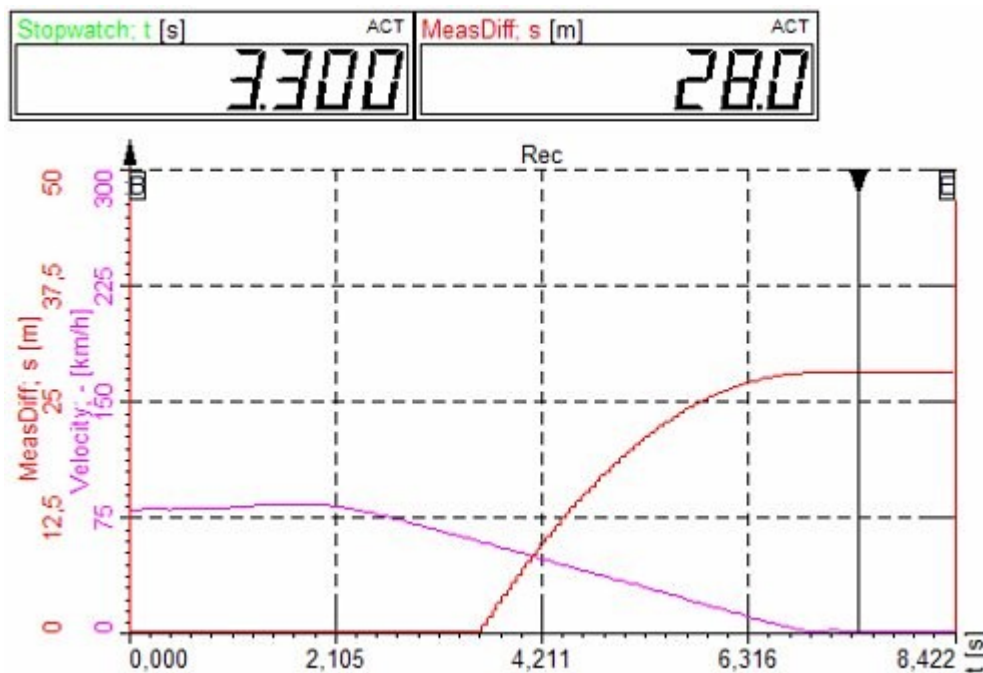
1.9.2.6 Example of stopwatch

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)
```

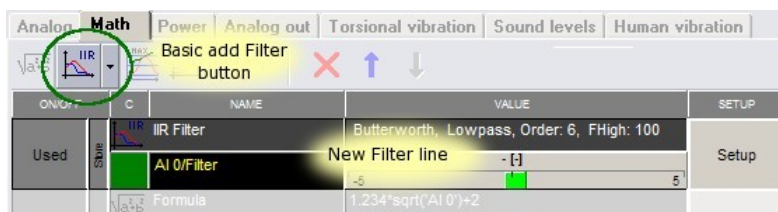
The graph below shows the example of measured data - *input* channels of *velocity* and *distance*. The *meter* shows the time and distance of braking.



Version 6.5 is by far better in calculating the distance and time of travel since it *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.

1.9.3 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:

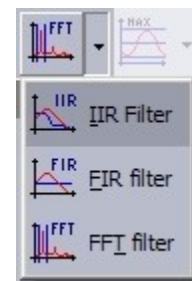
- **IIR Filter setup**
- **FIR Filter setup**
- **FFT Filter setup**

Add new Filter

Available **Filters** on **Math setup** screen are: **IIR Filter**, **FIR Filter** and **FFT Filter**. 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.



1.9.3.1 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:



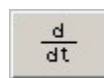
Standard



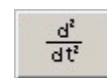
Integration



**Double
integration**



Derivation



**Double
derivation**



Custom

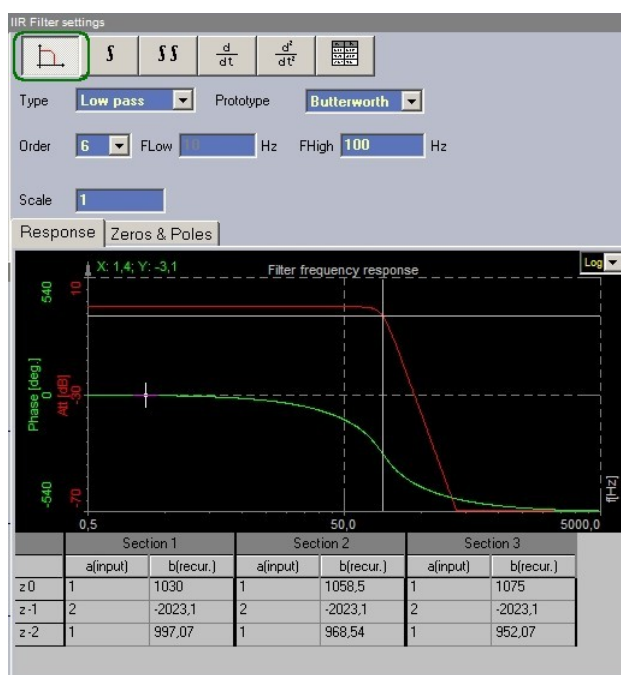
1.9.3.1.1 Standard filter

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**

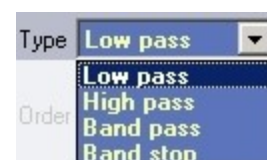
You can see *effect* of these settings directly on **Response curve / Zeros & Poles** display for **Filter Type: Low pass**, **High pass**, **Band pass**, **Band stop** and different **Prototype**.



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 $(-20 \text{ db/decade}) * \text{order}$. 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 steep 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 $(-20 \text{ db/decade}) * \text{order}$, 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 higher the order is, more *calculation power* will be needed to calculate the filter. We need **6** multiplications for *each two orders* of the filter.

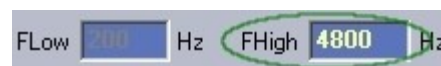


Cut-off frequency

You can enter Cut-off frequency:

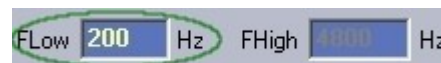
- **FHigh** - High frequency

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



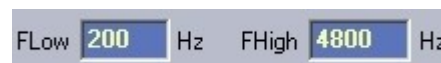
- **FLow** - Low frequency

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*.

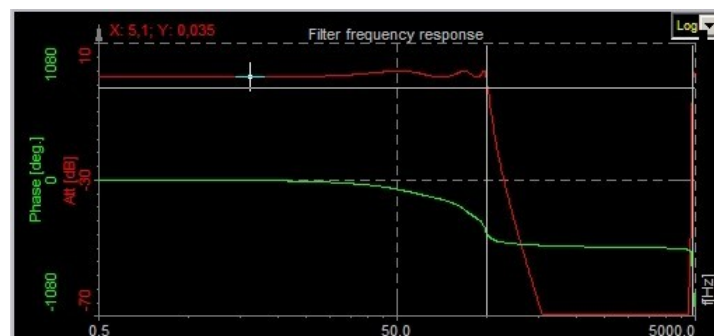
Scale



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 $a \text{ [dB]} = 20 * \log_{10}(A)$, so the attenuation ratio is calculated with $A = 10^{(a/20)}$.

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.

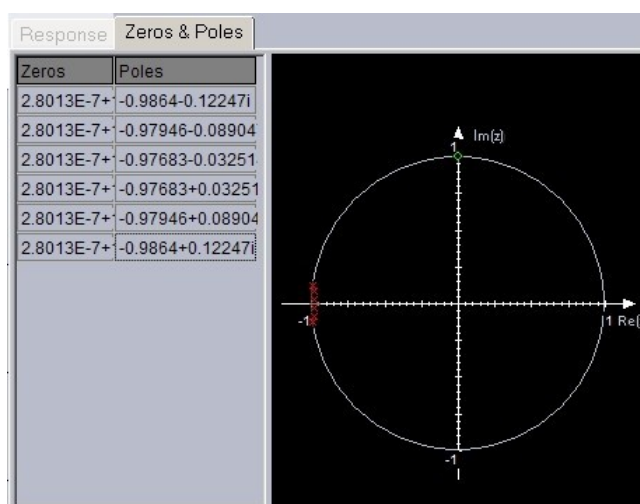
	Section 1		Section 2		Section 3	
	a(input)	b(recur.)	a(input)	b(recur.)	a(input)	b(recur.)
z 0	1	8,1553	1	11,045	1	25,338
z -1	-5,6027E-7	16,089	-5,6027E-7	21,637	-5,6027E-7	49,501
z -2	1	8,0573	1	10,684	1	24,204

	Section 4		Section 5		Section 6	
	a(input)	b(recur.)	a(input)	b(recur.)	a(input)	b(recur.)
z 0	1	8,1553	1	11,045	1	25,338
z -1	-5,6027E-7	-16,089	-5,6027E-7	-21,637	-5,6027E-7	-49,501
z -2	1	8,0573	1	10,684	1	24,204

On **Response** curve preview you can choose between **Logarithmic** and **Linear** display, you can also *edit* coordinates value and *auto scale* Y axis.



Zeroes & poles diagram shows the *position* of **filter** zeroes and **filter** poles and can suggest the *stability* of the filter.



1.9.3.1.2 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.

- 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.

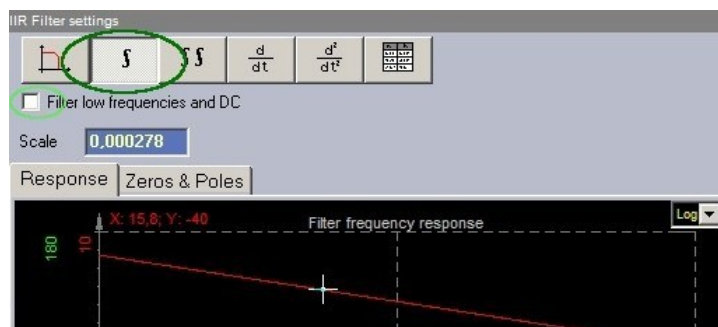
So if we have **km/h** as *input*, we have at *output*:

$$\text{km/h} * \text{sec} = 1000 \text{ m} / 3600 \text{ sec} * \text{sec} = 0.278 \text{ m}$$

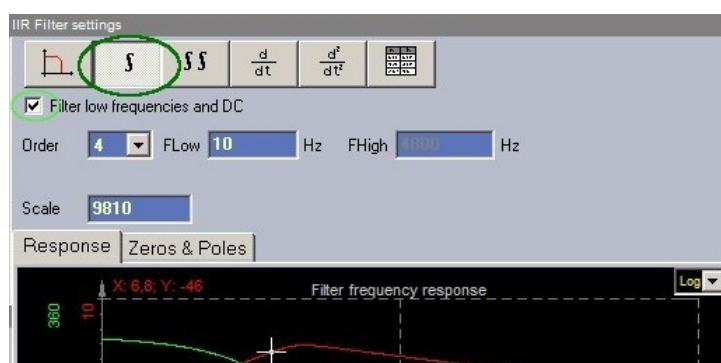
Therefore if we want to have the output in **meters**, we have to enter **0.278** as the *scale factor*. If we want to have output in **km**, we have to enter **0.000278**.

- Let's practice on another example - if we have input channel as power in **kW**, we have at output:

$$\text{kW} * \text{sec} = \text{kW} * 1/3600 * \text{h} = 0.000278 \text{ kWh} - \text{this is our scaling factor}$$



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:

$$1 g * sec = 9,81 \text{ m/sec/sec} * sec = 9,81 \text{ m/s} = 9810 \text{ mm /s}$$

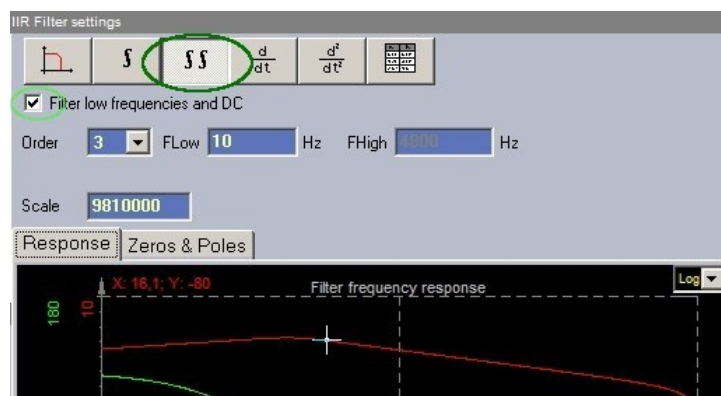
So we need to enter **9810** in the field scaling.

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:

$$1 g * sec * sec = 9,81 \text{ m/sec/sec} * sec * sec = 9,81 \text{ m} = 9,81E6 \mu m$$



1.9.3.1.3 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.

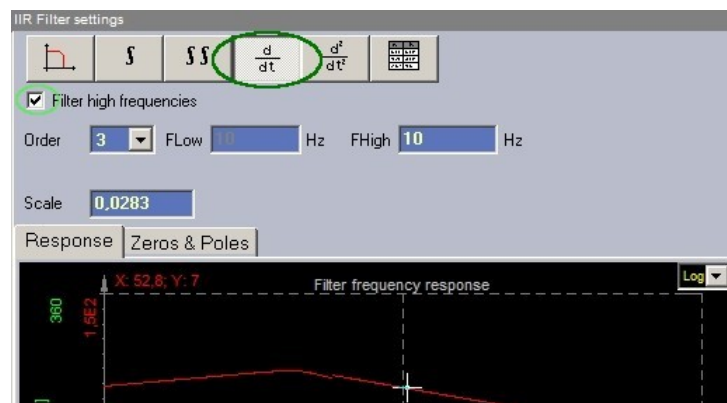
Let's assume the example of calculating **acceleration** out of **speed** of the vehicle. With car it is a fact that we can't have the acceleration higher than for example 10 Hz coming from the real vehicle acceleration - higher values are basically vibrations.

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:

$$1 \text{ km/hour} / \text{sec} = 1000 \text{ m} / 3600 \text{ sec} / \text{sec} = 0.278 \text{ m/s}^2 = 0.278 / 9,81 \text{ g} = 0.0283 \text{ g}$$

We have to enter this value in the scale field to get correct results.



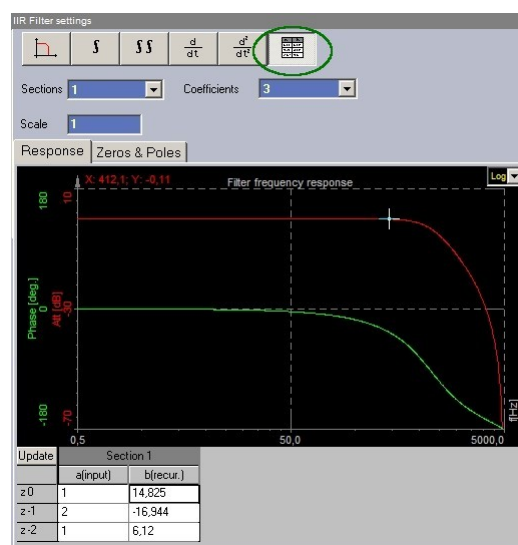
1.9.3.1.4 Custom filter

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

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 thing 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:

$$H(s) = (g_0 + g_1 \cdot s + g_2 \cdot s^2) / (h_0 + h_1 \cdot s + h_2 \cdot s^2) \quad (1)$$

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

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

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:

$$g \cdot (1 + z^{-1})^2 + 2 \cdot g_1 \cdot f_s \cdot (1 - z^{-1}) \cdot (1 + z^{-1}) + 4 \cdot g_2 \cdot f_s^2 \cdot (1 - z^{-1})^2 = \\ (g_0 + 2 \cdot g_1 \cdot f_s + 4 \cdot g_2 \cdot f_s^2) + (2 \cdot g_0 - 8 \cdot g_2 \cdot f_s^2) \cdot z^{-1} + (g_0 - 2 \cdot g_1 \cdot f_s + 4 \cdot g_2 \cdot f_s^2) \cdot z^{-2} \quad (3)$$

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**:

$$H(s) = 1 / (1 + \text{sqrt}(2) \cdot s / \omega_c + (s / \omega_c)^2) \quad (4)$$

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

$$\omega_c = 2 \cdot f_s \cdot \tan(\pi \cdot f_c / f_s) \quad (5)$$

If we write out the factors for this filter:

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

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:

$$f_{cp} = 2 * 1000 * \tan(\pi * 100 / 1000) = 649,8 \text{ rd/s}$$

And now we need to calculate the *coefficients* for direct and recursive part of the filter with substituting factors (6) in equation (3):

$$a(z-2) = 1-0+0 = 1$$

$$b(z-2) = 1 - 2 * \sqrt{2} * 1000/649,8 + 4 * 1000^2/649,8^2 = 6,12$$

$$a(z-1) = 2*1-0 = 2$$

$$b(z-1) = 2 * 1 - 8 * 1000^2/649,8^2 = -16,944$$

$$a(z0) = 1+0+0 = 1$$

$$b(z0) = 1 + 2 * \sqrt{2} * 1000/649,8 + 4 * 1000^2/649,8^2 = 14,825$$

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.

	Section 1	
	a(input)	b(recur.)
z0	1	14,825
z-1	2	-16,944
z-2	1	6,12

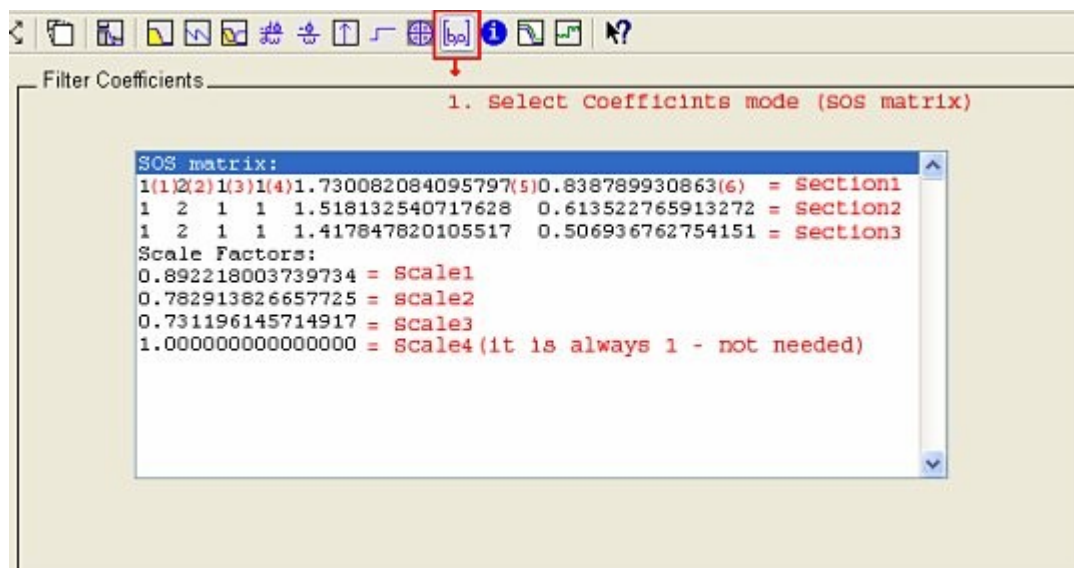
Remember, this is *valid only* for a sampling rate of 1000 Hz. For others, we need to *recalculate* the f_{cp} and coefficients.

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**.

1.9.3.1.5 Custom filter import from Matlab

It is possible to import custom filters 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*:

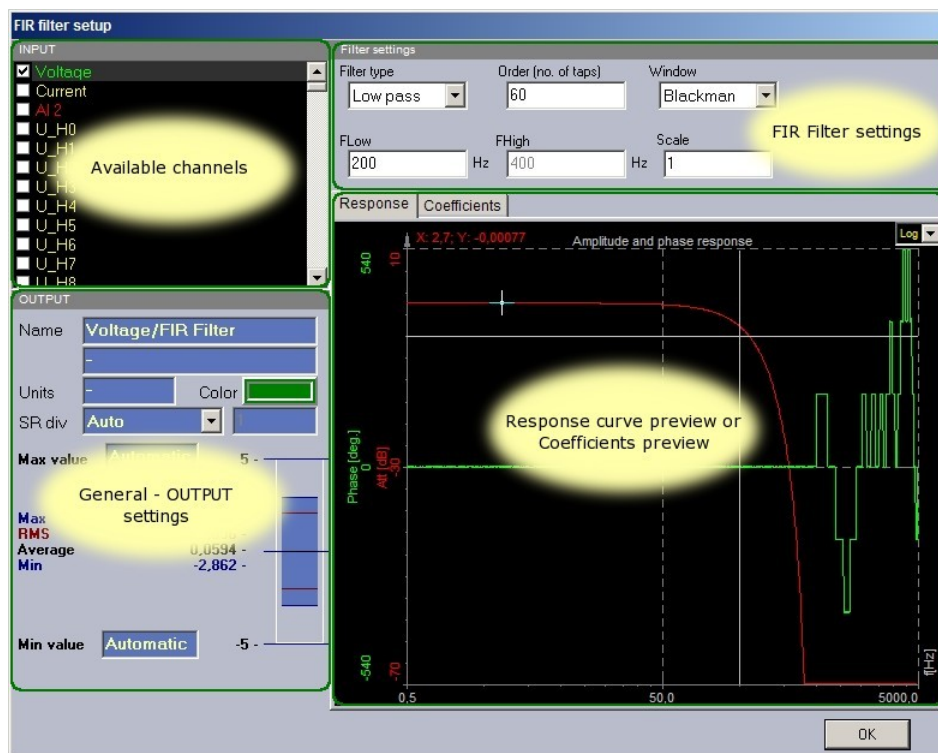
	<i>a(input)</i>	<i>b(recur.)</i>
z0	Scale <i>i</i> * Section <i>i</i> (1)	Section <i>i</i> (4)
z-1	Scale <i>i</i> * Section <i>i</i> (2)	Section <i>i</i> (5)
z-2	Scale <i>i</i> * Section <i>i</i> (3)	Section <i>i</i> (6)

The coefficients for our example are calculated below:

Update	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

1.9.3.2 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**:

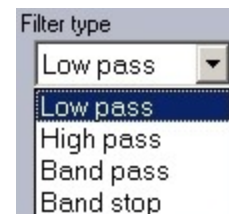
- **Filter type** and **Window** type
- **Order**
- **Scale**
- **Kaiser** window type - **Ripple**
- **Cut-off Frequency** - **FLow** and **FHigh**

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

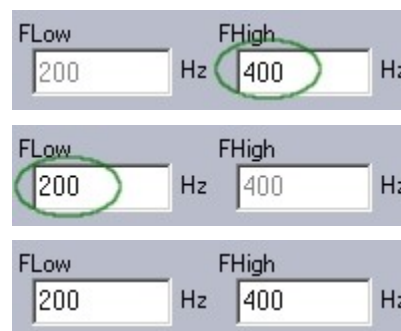
When a **Kaiser** Window type is selected, new **Ripple** field appears on the right side of the **Window** type field. In this field you can enter ripple value in **dB**. It tells the maximum allowed pass band ripple of the filter. The more this value is, the bigger will be non-linearity in the pass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



Cut-off frequency

The filter cutoff frequency *defines* the **-6 dB** point (*half* amplitude) of the filter. You can enter **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.

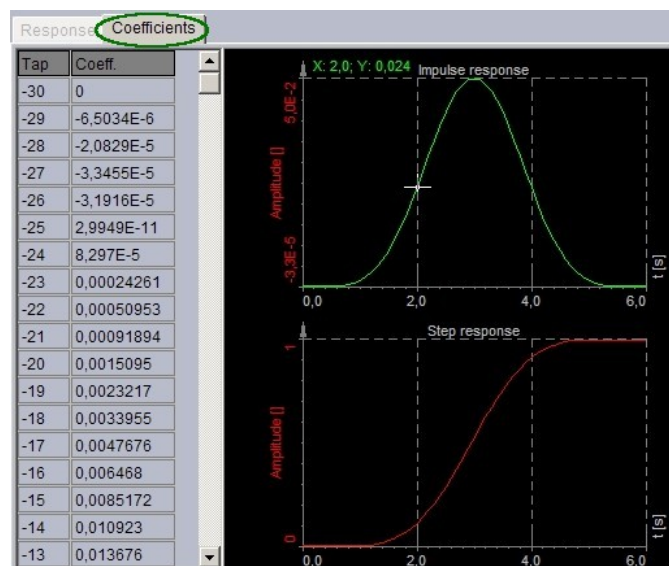
Response curve / Coefficients preview

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 **Logarithmic** and **Linear** display, you can also *edit* coordinates value and *auto scale* Y axis.

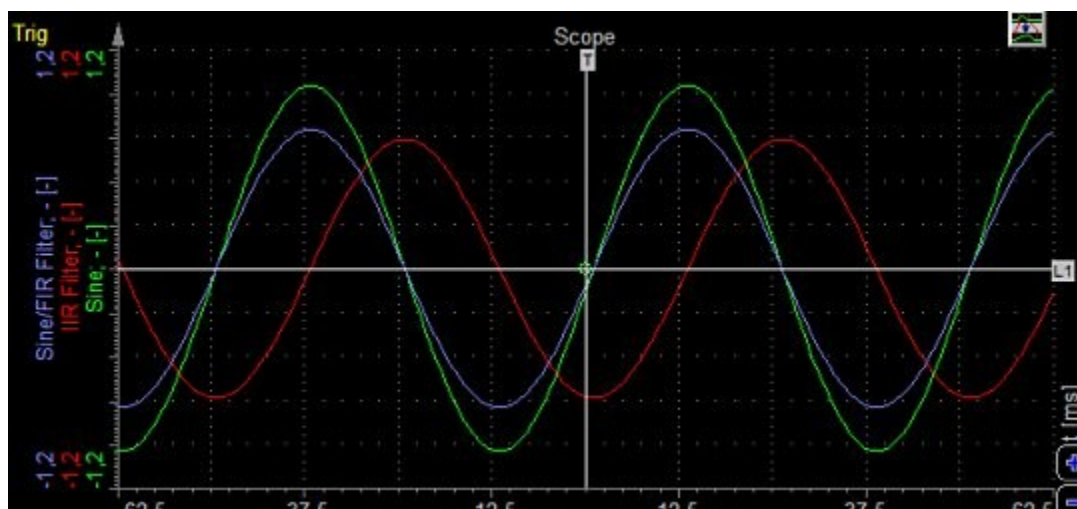


Filter comparison

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.

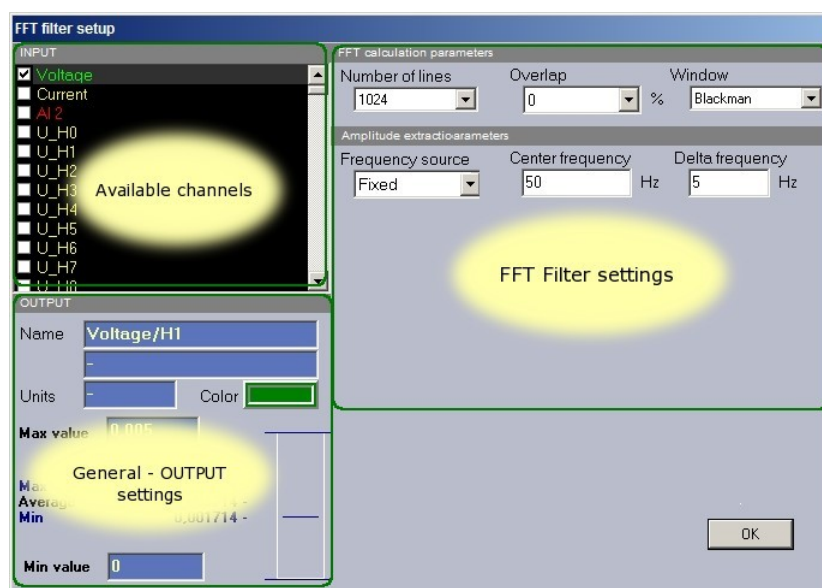


Enough of the down sides, let's look at the response graph at 20 Hz (exactly at the limit). The green curve is the *original sine* wave while the red one is *calculated* with IIR filter. We can clearly see the *phase delay* of the output.

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*.

1.9.3.3 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*.

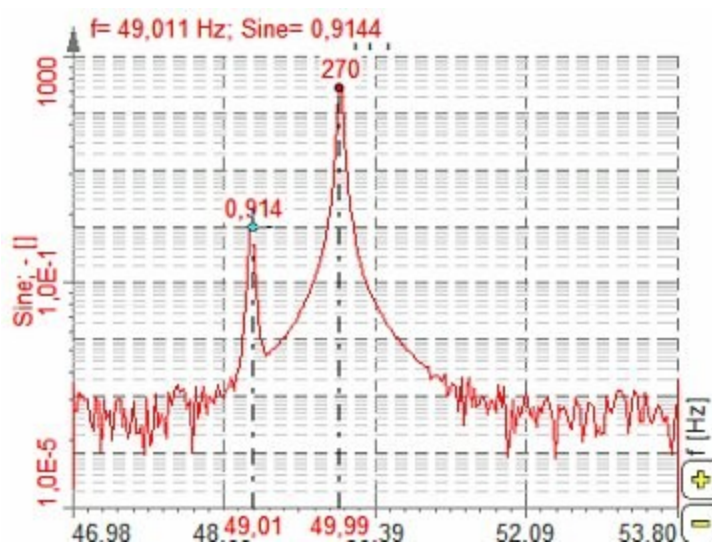
for detailed information about *basic settings* of the *input and output channels* see → **Setup screen and basic operation**

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:



FFT filter parameters

To set the **FFT** filter, it is recommended to observe the signal in the **FFT** display and choose the right **Number of lines** and **Window** which fits the best and then set the *filter parameters*.

For **FFT Filter** you can **set**:

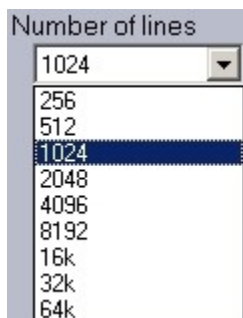
– FFT calculation parameters

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

– Amplitude extraction parameters

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

Number of lines



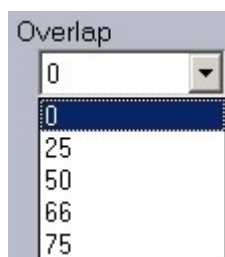
You can select **Number of lines** from list.

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



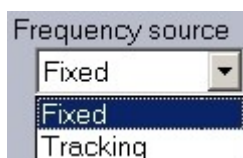
You can select **Window** type from the list.

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

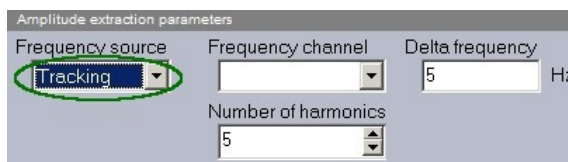
In this field you can enter **Delta frequency** in Hz. This value depends on the wanted frequency band. It also *depends* on the *window* and *number of lines* (line resolution). In our example we would choose 0.5 Hz since we don't want that 50 Hz value will appear in the result.

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

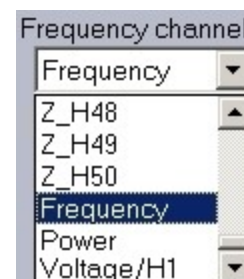
If we choose **Tracking Frequency source**, we need to enter the **Frequency channel** and **Number of harmonics** instead of *Center frequency*.



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.

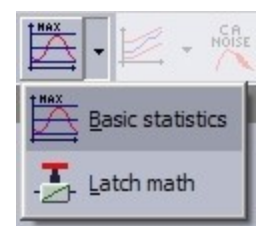
1.9.4 Statistic group

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** and **Latch math**. New **Statistics** line can be **added** by selecting:

- **basic add Statistics** button (when appropriate *Statistics* is shown on icon)
- from the **Statistics list** → for *statistics list* display select ▼ *triangle sign* on basic button



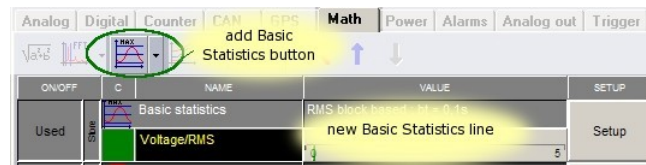
When we select *Statistics* from *list*, icon changes to *selected* one, so we can choose the *same* *Statistics* again by simply *clicking* on the basic *add Statistics* button.

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**
- **Latch math**

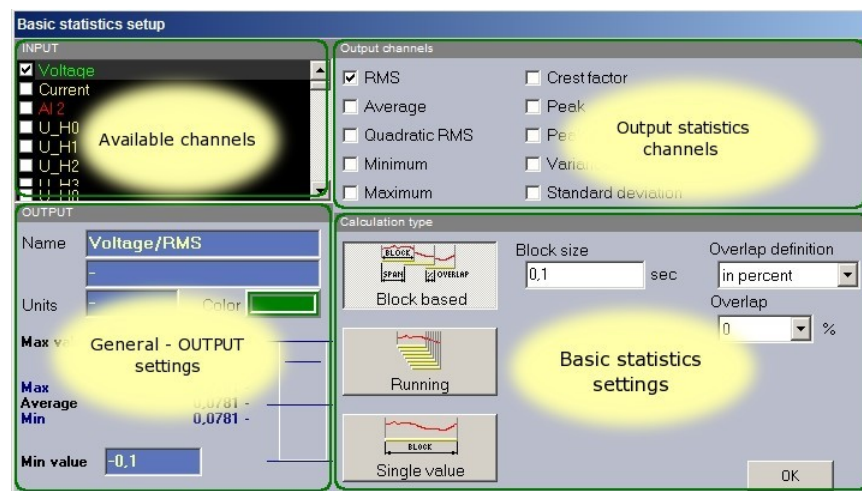
1.9.4.1 Basic statistics

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



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](#)

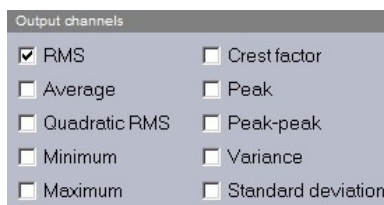
The statistic module provides basic **statistical quantities** of the *signal*. There are three basic calculation modes:

- **Block based**
- **Running**
- **total - Single value**

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.

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.

Average

Average will calculate the average or middle point.

Peak

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

Quadratic RMS

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

Peak-peak

Is the difference between the minimum and maximum.

Minimum

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

Variance

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

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.

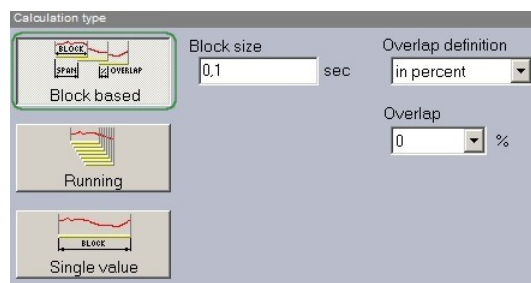
Standard deviation

The standard deviation is a measure of the spread of the values of the signal away from its mean, measuring how widely spread the values in a data set is. If the data points are close to the mean, then the standard deviation is small (if all the data values are equal, then the standard deviation is zero).

Calculation type

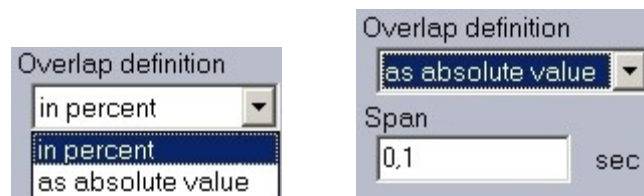
Block based

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



- **Block size** in [sec] defines the *time interval* for calculation. 0,1 second in our case means that it will calculate the statistical quantities in 0,1 second interval. Therefore the *resulting* channels will have an **update interval** of 0,1 second.
- **Overlap** - Is useful when we need a *specific* time interval, but want to have *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.



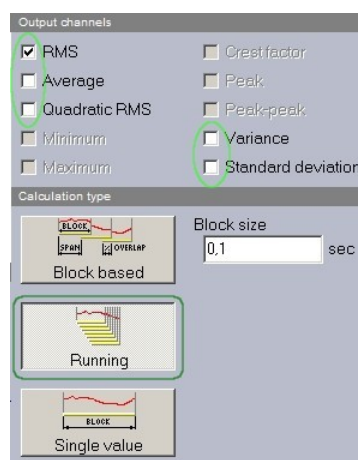
Overlap can be defined:

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

Running

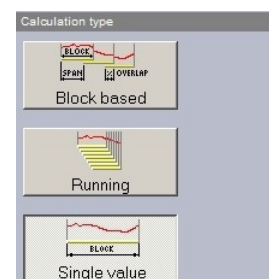
Running calculation is an 'extreme' version of overlapping. The second block is calculated *after one sample* after the first block. **Block size** has the same meaning as for *block based* calculation.

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).



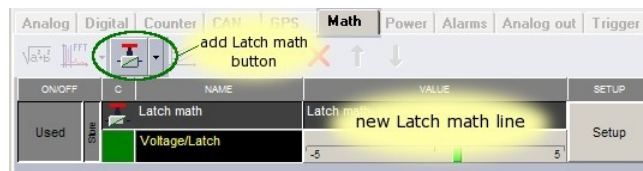
Single value

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.



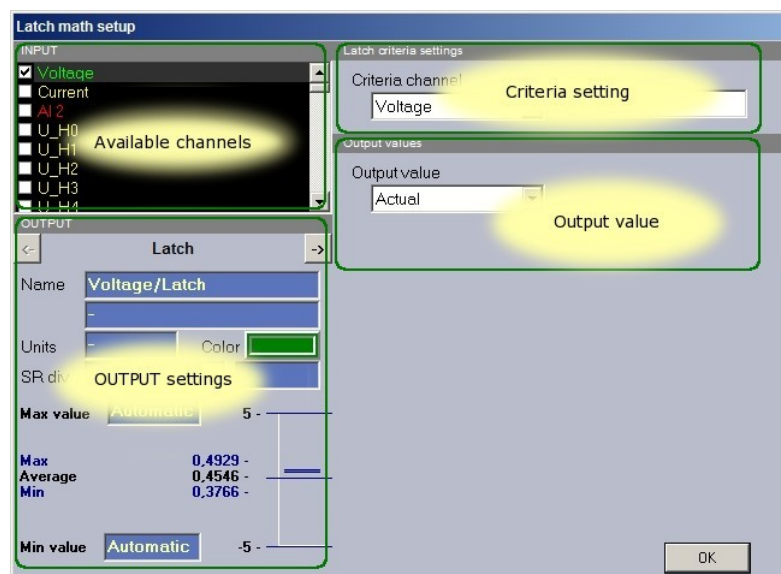
1.9.4.2 Latch math

New **Latch math** module can be *added* on the **DEWESoft Math setup** screen by selecting *add* **Latch math** button:



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](#)

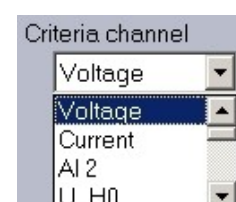
The **Latch math** will return the value of the channel when a latch condition is met. So we can use this for getting the values only at specific trigger position (for example a state of the test). These values can be nicely displayed in the tabular display.

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.

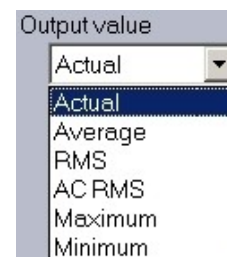
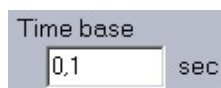
After that enter **Criteria limit value** in corresponding field.



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.



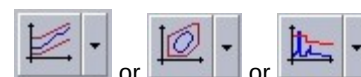
1.9.5 Reference curves

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

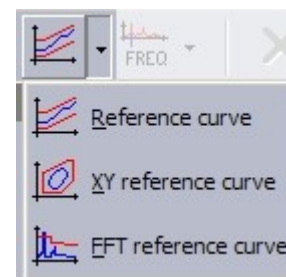
Add new Statistics

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:

- **basic add 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 basic 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.

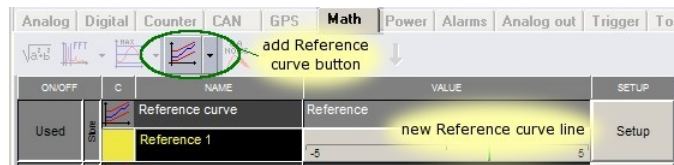
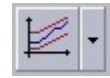


When you press the **Setup** button on activated *new Reference curve line*, the **Reference curve setup** window will open. This window depends on selected *Reference curve type* and his *name* appears on title line of window:

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

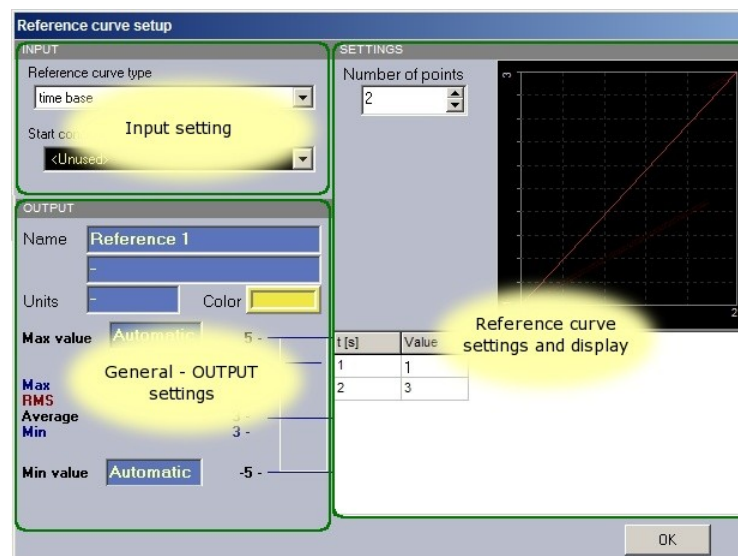
1.9.5.1 Reference curve

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:



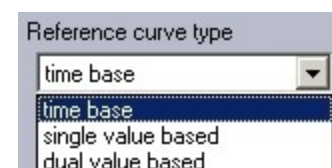
for detailed information about **basic settings** of the **output channels** see → [Setup screen and basic operation](#)

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.

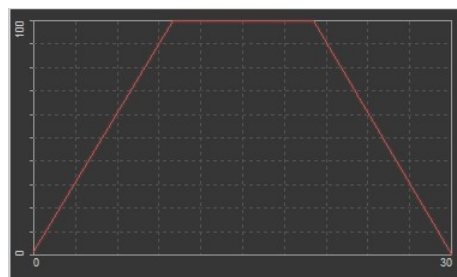
INPUT	SETTINGS
Reference curve type time base	Number of points 4
Start condition channel AI 2	Start if value above 2

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:

$$\text{abs}(\text{'REFCURVE'} - \text{VELOCITY}) > 5$$

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

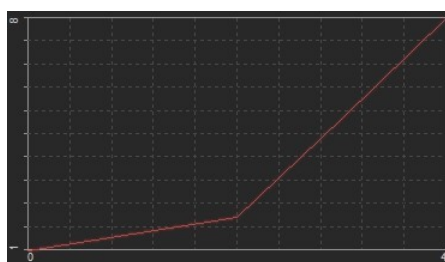
A single value based reference curve can be understood as a sort of *non-linear scaling*. We need to define a **First reference channel**, **Number of points** and a table with the *values of reference* and output channel.

INPUT		SETTINGS	
Reference curve type			
single value based			
First reference channel		Number of points	
AI 2		4	

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

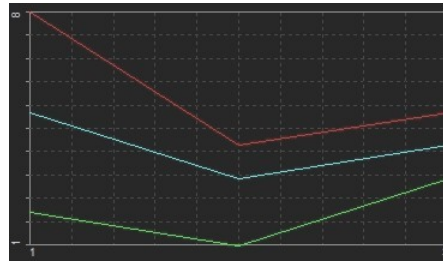
Dual value based reference curve have *two inputs*: **First reference channels** and **Second reference channels**. We can imagine as the *tridimensional surface* of reference points. *X* and *Y* are two *reference channels* while the *Z* is *defined points*. We need to define *first* the channels and the **Number of points** for each channel.

INPUT		SETTINGS	
Reference curve type			
dual value based			
First reference channel		Number of points	
AI 2		4	
Second reference channel		Number of points	
Voltage		4	

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

Then we need to enter the reference *values* in the list.

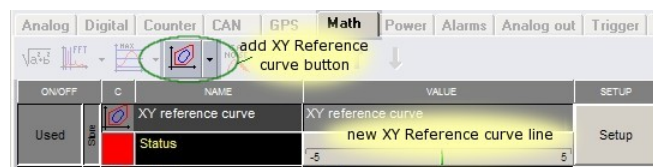
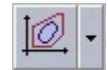
Display on right upper part *Reference curve setup* window now shows this curve:



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

1.9.5.2 XY reference curve

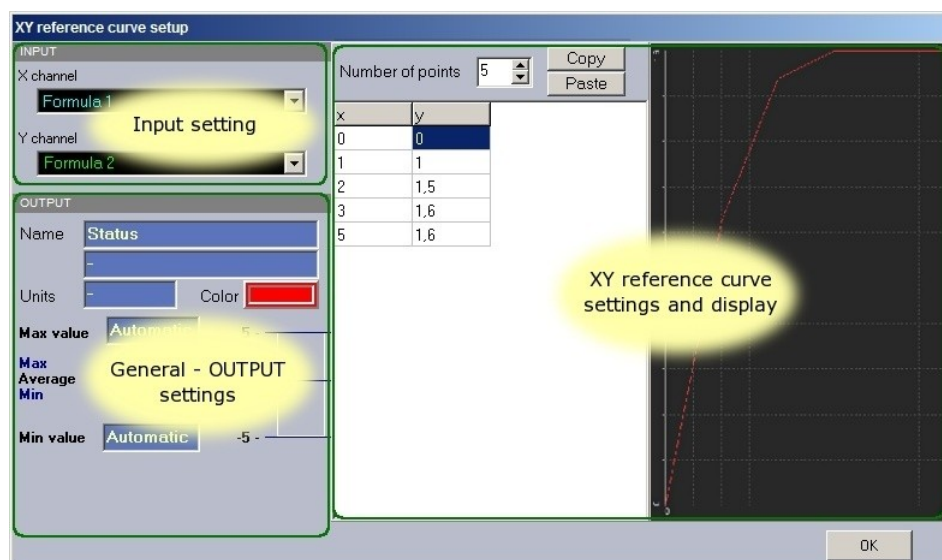
New **XY reference curve** module can be added on the **DEWESoft Math setup** screen by selecting **add XY reference curve** button:



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](#)

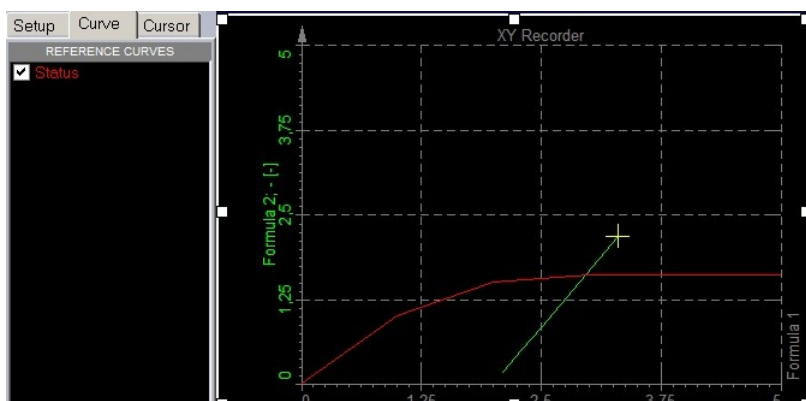
After choosing **INPUT X channel** and **Y channel**, we need to set **Number of points** for **XY reference curve**. The reference points can be entered value by value. We have also another option for easier transfer of points - *copy and paste* function. **Copy** button copies all the points to the 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 **XTABYCR 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*.

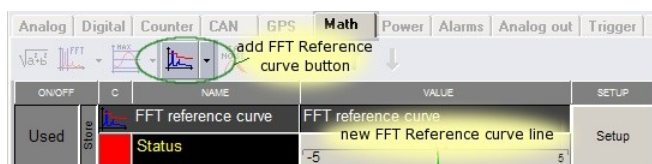
The **XY display** can show the reference curve as seen of the picture below.



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*).

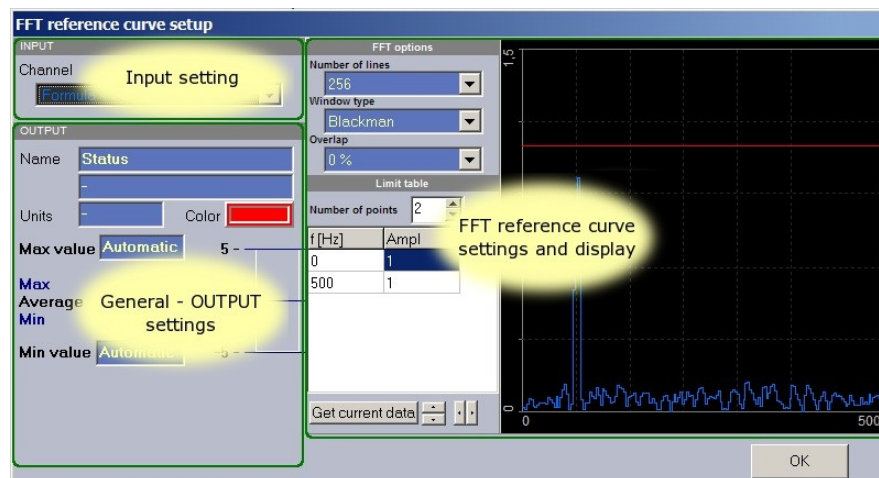
1.9.5.3 FFT reference curve

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](#)

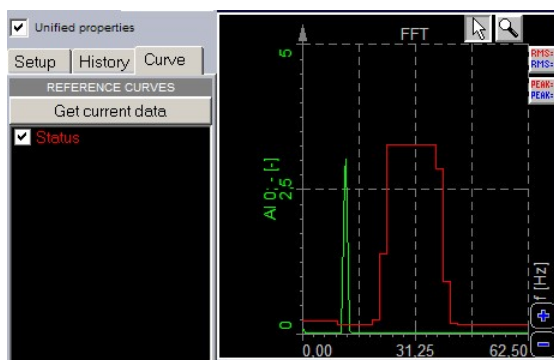
The **FFT reference curve** math module provides the **reference curve** for *FFT display*. It can be used for displaying the reference on FFT screen and also to *provide a math output channel* which goes *high* when the *levels are exceeded*.



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.



1.9.6 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

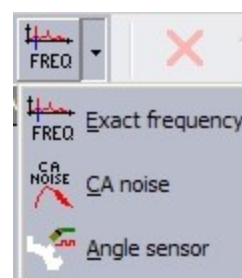
Available DSA function on **Math** setup screen are: **CA NOISE**, **Exact frequency** and **Angle sensor**.

New DSA function can be **added**:

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

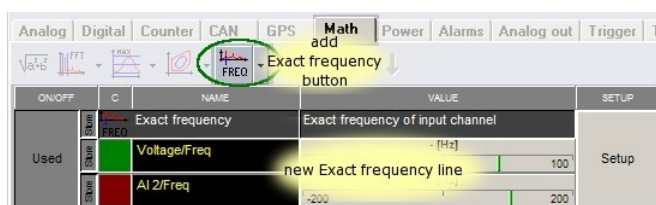


When you press the **Setup** button on activated *new CA noise / Exact frequency* line, the **setup** window will open. This window depends on selected type of function and his name appears on title line of window:

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

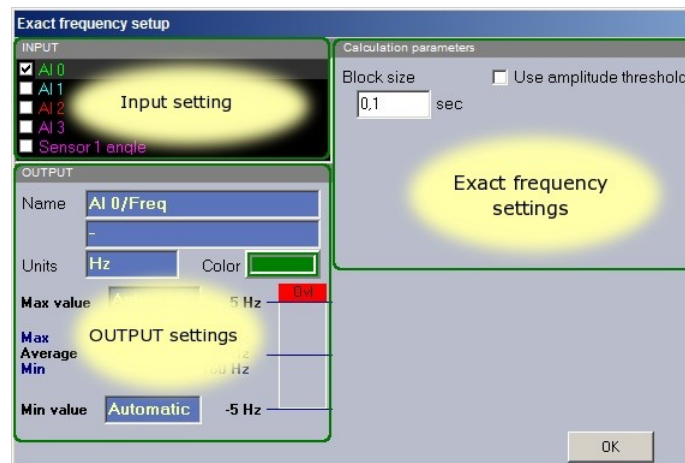
1.9.6.1 Exact frequency

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.

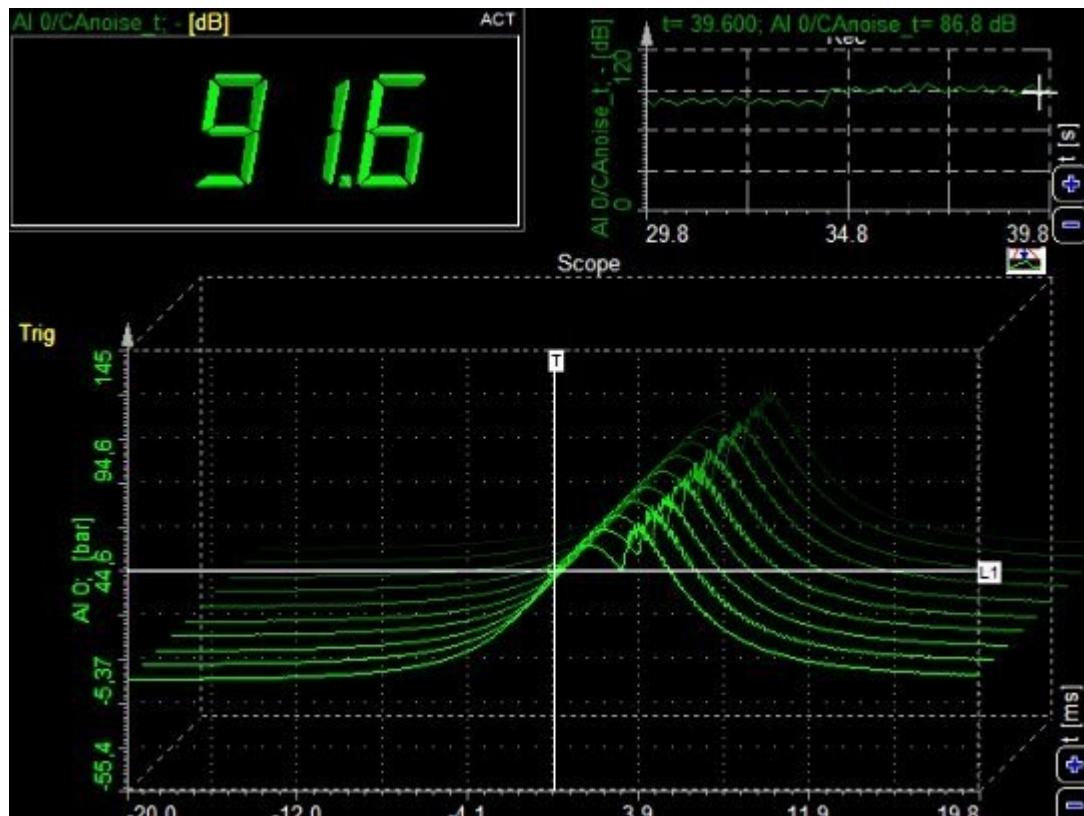
The method is based on finding *best fit* to the theoretical *sine wave*. It can measure **millihertz** accurate with sampling rates only few **kHz** and a *low* block size. The method works best on the signals close to the pure sine wave. It will search and lock to the highest harmonic component in the signal.

In the right area of the window are -**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.

1.9.6.2 CA noise

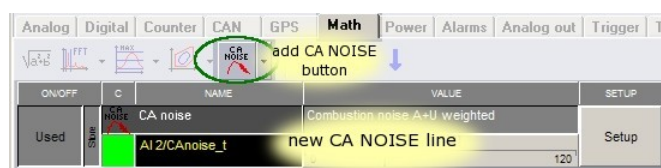
CA noise description



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

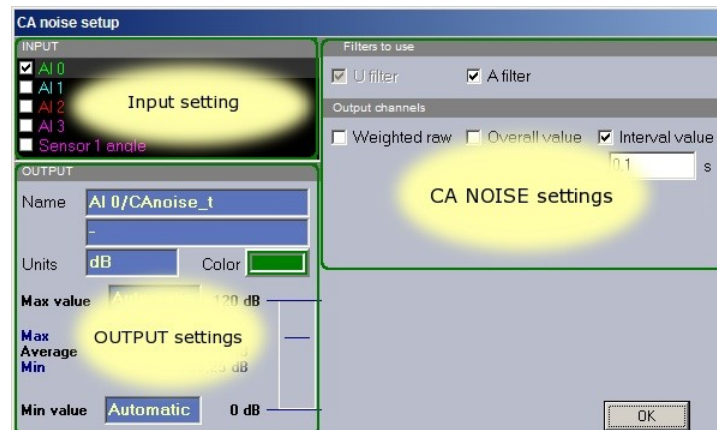
New **CA NOISE** function can be *added* on the **DEWESoft Math** setup screen by selecting *add CA NOISE* button:



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:

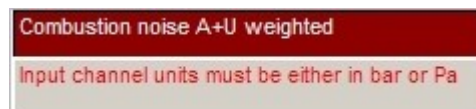


The CA NOISE supports *multiple input channels*.

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:

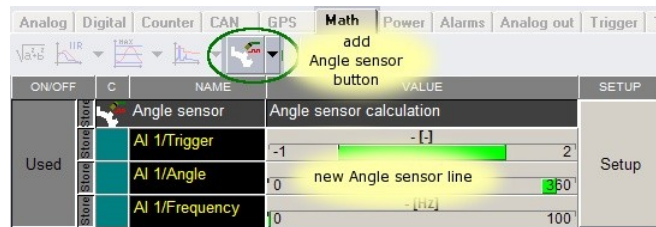


There are three types of output channels available:

- **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*.

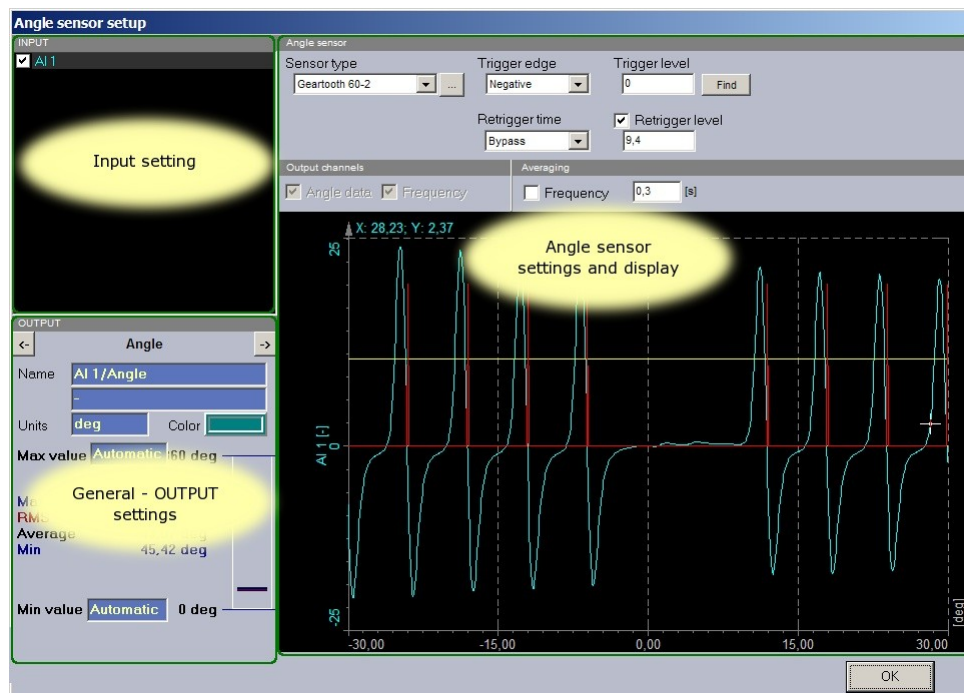
1.9.6.3 Angle sensor

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:

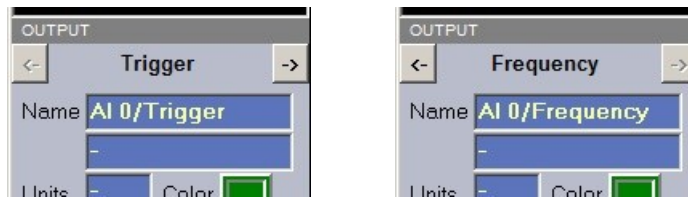


for detailed information about *basic settings* of the *input* and *output channels* see → [Setup screen and basic operation](#)

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.

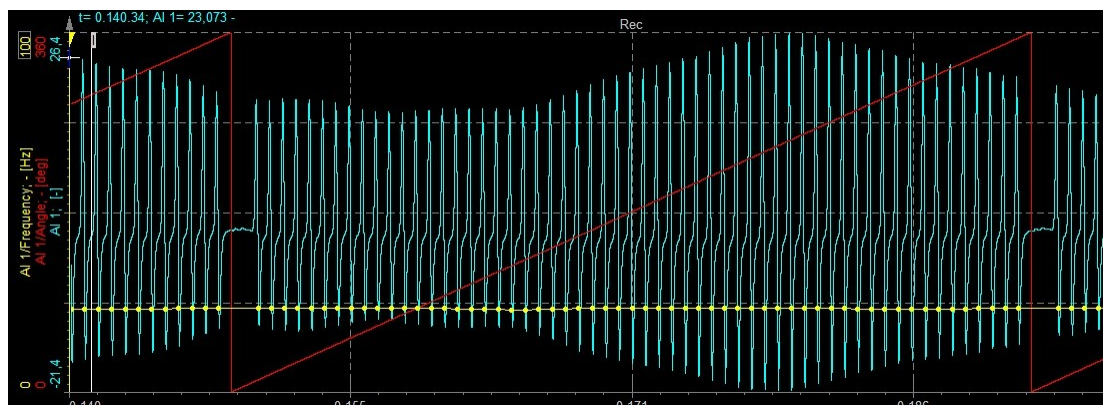
The **output** from this math module is the:

- *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*.



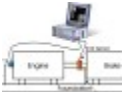
The nice feature of this module is that it *implements* calculation **delay**. For sensor like 60-2 it is not possible to determine the angle until the first gap is recognized. For *that time* the **angle sensor math** **holds** all the calculations and *with* the first gap also calculates the data for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

1.10 Dynamic signal analysis

DEWESoft Dynamic signal analysis covers a wide range of measurements in the field of **structural dynamics**, **industrial acoustics** and **machine diagnostics**.

The frequency response function measurement is a part of different manual - [FRF user's guide](#).

	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 <i>shafts</i>
	Human vibration	explains procedures for working with Human vibration module , used to evaluate effects of vibration on <i>human body</i>
	Order tracking	explains procedures for working with Order tracking module used to extract the harmonics during <i>machine run up</i> and <i>run downs</i>
	Combustion analysis	explains procedures for working with Combustion analysis module used to calculate parameters of internal combustion engines

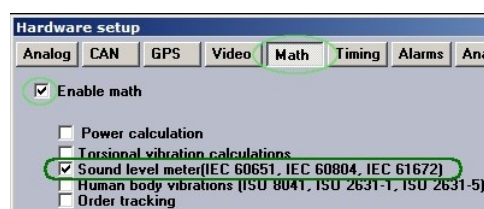
1.10.1 Sound level

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	24 bit AD card (Orion), MDAQ-ACC or DAQP-ACC
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](#)

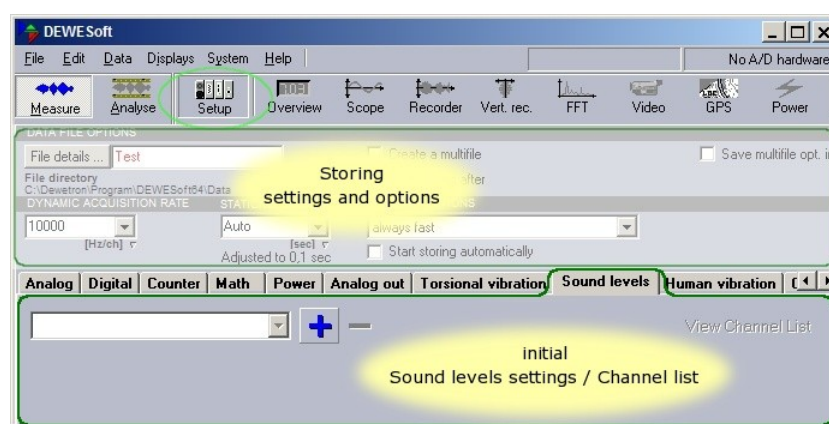
After *required hardware - microphone installation* and *setup* (this procedure is to perform *only* by installation new or changing hardware) according to manufacturer instructions, **DEWESoft Sound level** measurement is allowed by selecting the **Sound level meter** checkbox in **System → Hardware 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/out 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 deleted by *selecting* him and then using  - minus button.

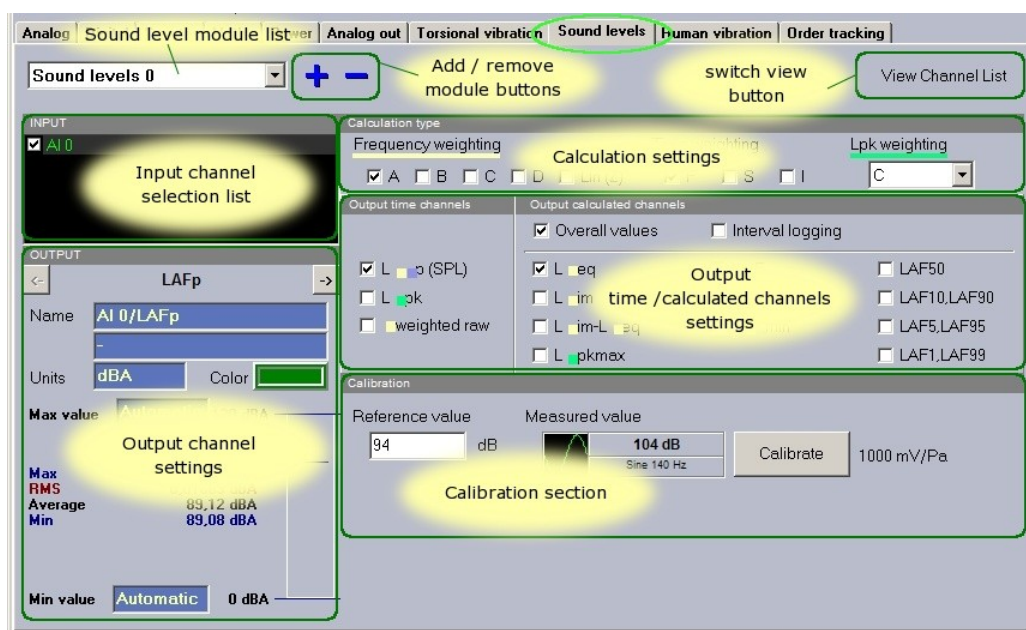
WARNING Think well before using this command, because we can't return deleted module with settings!

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 time channels** definition
- **Output channel** settings
- **Output calculated channels** definition
- **Calculation type**
- *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

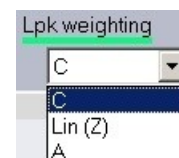
We have several options to choose in **Calculation type** section:



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.

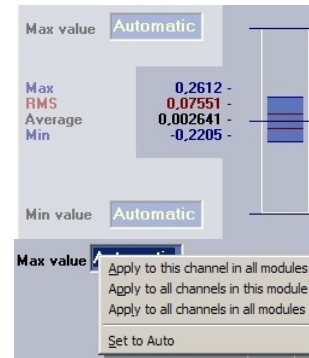
for detailed information about this **General** setting see also → [User Guide](#) → **Channel Setup**

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.

Rightclicking 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).

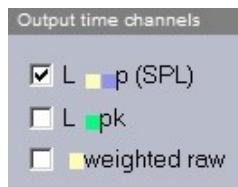
Apply to all channels in all module

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:

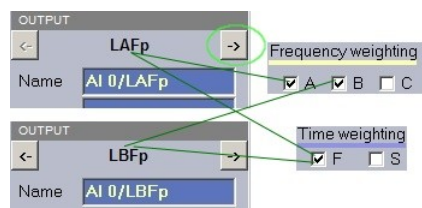
Output time channels

Select desired option with checking box beside caption.

We have three *types* of **Output time channels**:



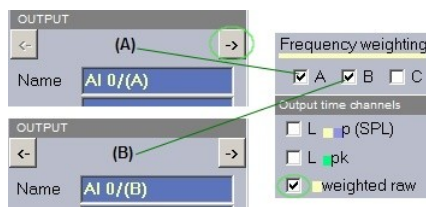
1. **L_{FT}p** - *time* and *frequency* weighted sound pressure level already scaled to **dB** which depends on **Calculation type** selection → **Frequency weighting** and / or **Time weighting**. For *each* combination of selection in this sections one **separate output channel** is created:





2. L_{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 one separate output channel is created.

3. $L_{weighted\ raw}$ value shows the *frequency weighted time curve* of sound in **Pascal**. This value depends on **Calculation type** selection → **Frequency weighting**. For *each* selected choice in this section one separate output channel is created:

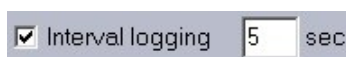


Output calculated channels



Select desired option with checking box beside caption. First choose upper calculation parameter (at least one must be selected for calculated channels):

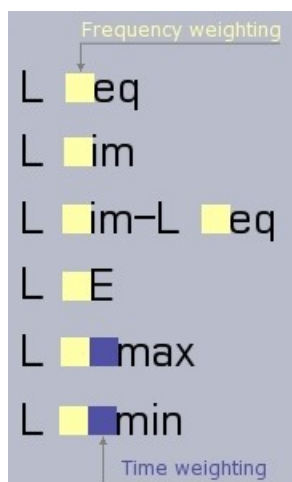
- **Overall values** - we have only *one* value at the end of the measurement
- **Interval logging** - the *time interval* for logging is defined.



For example if we select to have interval logging with **5 seconds** interval, we will get a *new value after each 5 second*. After that the value is *reset* and the calculation is *started again*.

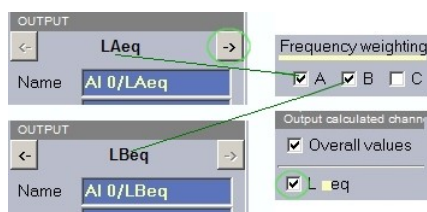
- or both

The *values* which can be **calculated** are:

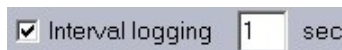


- *frequency weighted* L_{eq} value, which is equivalent sound level
- L_{im} tells the *impulsivity* of sound and is *impulse weighted* equivalent; the difference between those two values is *calculated* as $L_{im} - L_{eq}$
- L_E is *frequency weighted sound energy*
- L_{max} and L_{min} is *time and 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 one separate output channel is created.



When **Interval logging** is selected and defined:



also for these combination of selection in this sections one **separate output channel** is created:



NOTE: Interval logging is designated with t characters on end of channel name.



- L_{Lpk} **Lpkmax** is with *C* or *linear* weighted maximum *peak* value of sound; depends on **Calculation type** selection → **Lpk weighting**. For selected choice one **separate output channel** is created.



- **Classified sound levels**. 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.

View Channel List

When this button is selected, *Channellist* 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**.

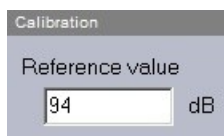
2. Calibrating the microphone with calibrator

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.

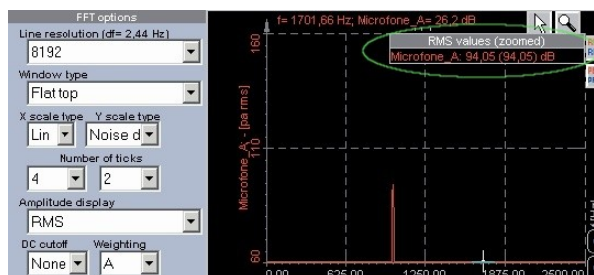
NOTE We need to use setup sample rate in DEWESoft at least 5 kHz or higher to make successful sound calibration. This can be changed in DEWESoft Tuner Utility.

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*.

1.10.2 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

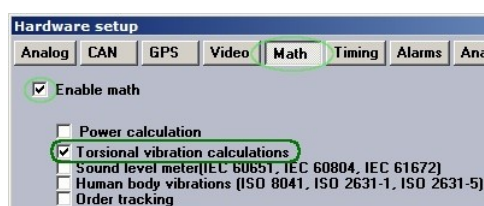
from Torsional vibration or Rotational vibration → see → **Extract order**

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	24 bit AD card (Orion) 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](#)

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

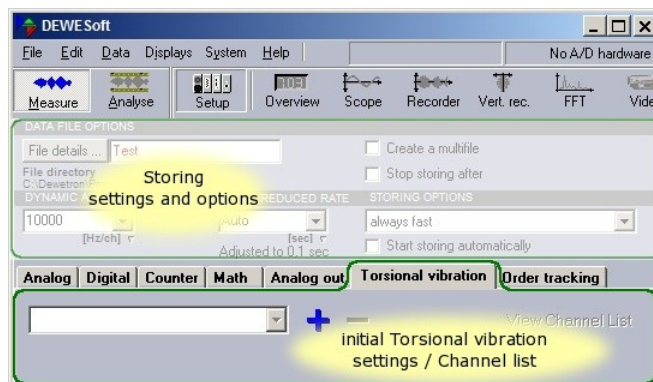


After selecting this option, a tab labeled '**Torsional vibration**' appears in the **DEWESoft Setup** screen (see picture below) to *define and setup* **Torsional vibration module(s)**.

To use *Torsional vibration* module we don't need to set anything on *analog* or *counter channels*.

Torsional and *rotational vibration* can be measured with either **encoder** (up to 3600 pulses per revolution) or *special RIE sensor* which has less resolution (up to 720 pulses per revolution) but is much less sensitive to vibrations which could damage standard encoders.

Add / New Torsional vibration module



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.

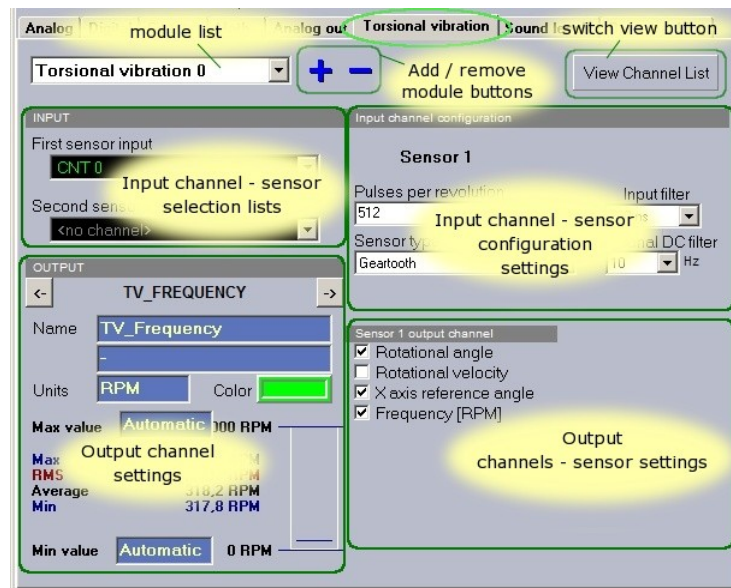
WARNING Think well before using this command, because we can't return deleted module with settings!

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
- **Input channel configuration** - for *Sensor 1*
- **Output channel** settings
- **Sensor output channel** definition for *Sensor 1*

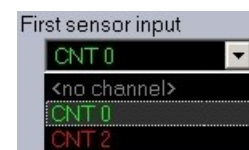
With using **View Channel List** button we can show different view on *Torsional vibration module* setup → **Channel list** with all *defined output channels*.

Input channel - sensor

First of all select the **input channel - sensor 1** (the *first connected sensor*) from the **First sensor input** drop down list.

Since we have connected in example above the first sensor to **CNT0**, we need to select it from list.

Please note that CNT1, CNT3 and CNT5 are *not available*, since they will be used internally *to calculate exact frequency*.

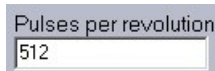


Input channel configuration

Since we have by Rotational vibration only one (first) sensor connected, on *Rotational vibration setup* screen only one for **Sensor 1** *Input channel configuration* section appear. On this section we can set:

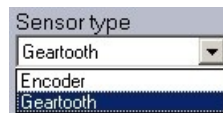
- Pulses per revolution**

enter in this field actual value pulses per revolution:



- Sensor type**

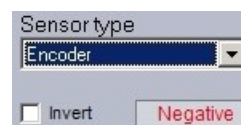
select **Geartooth** or **Encoder** sensor type from drop down list:



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}]

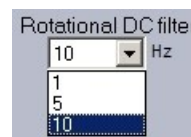
PPR pulses per revolution [-]

for information about **Input filter** see → [User Guide](#) → **Counter setup**

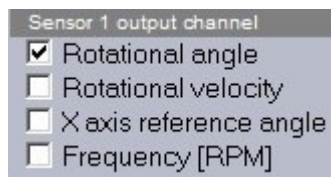


- Rotational DC filter**

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**

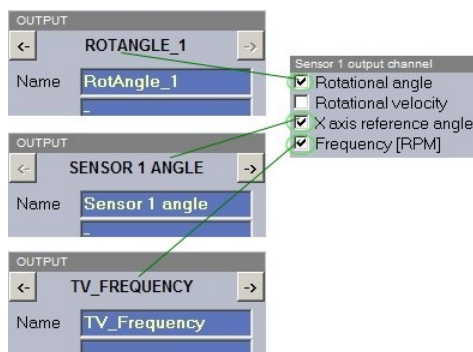
Torsional vibration module can have more **output** channels according selection in **Sensor 1 output channel**.

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 →	Name RotAngle_1	output channel is created
Rotation velocity choice →	Name RotVelocity_1	output channel is created
X axis reference angle choice →	Name Sensor 1 angle	output channel is created
Frequency [RPM] choice →	Name TV_Frequency	output channel is created

Example:



At least one **output** channels must be checked.

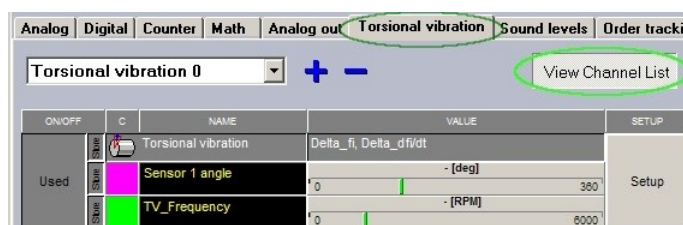
Use **<-** (Previous) and **->** (Next) buttons to navigate between the **output channels** in **OUTPUT channel settings** part of screen.

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.

View Channel List

When this button is selected on *Rotational vibration module* setup, **Channellist** 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 **View Channel List** 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**.

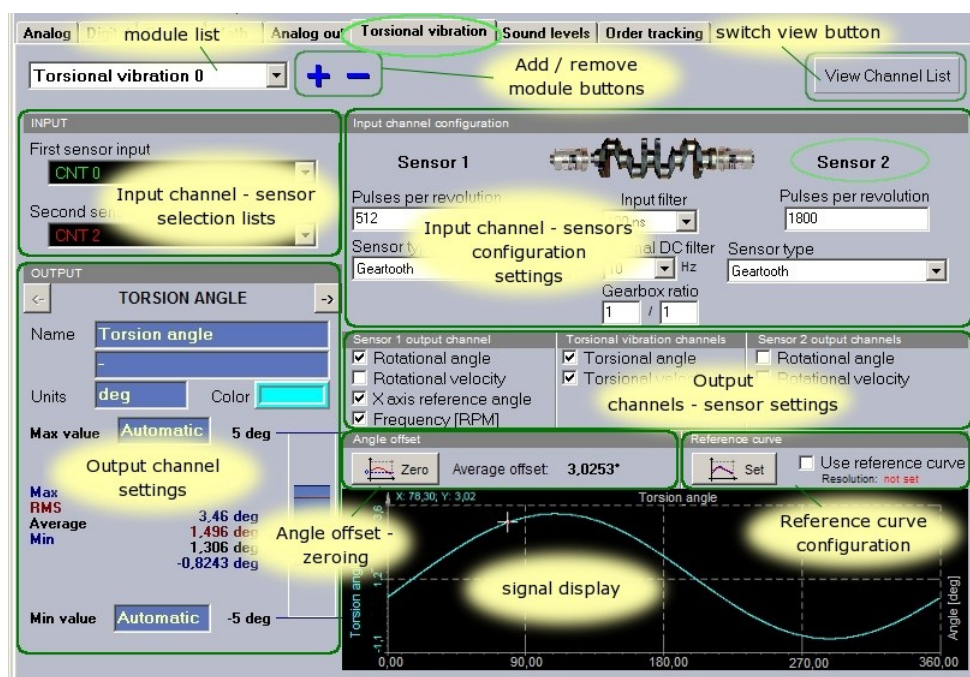
Beside **first input channel - sensor 1** also **second input channel - sensor 2** (the **second connected** sensor) from the **Second sensor input** drop down list must be **select**.

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 two (*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

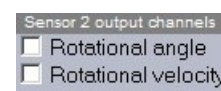
Sensor 1 output channel

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



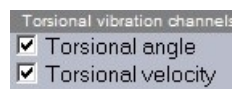
Select desired option with checking box beside caption.

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*



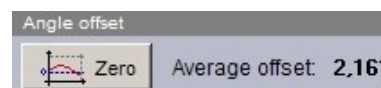
Select desired option with checking box beside caption.

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.

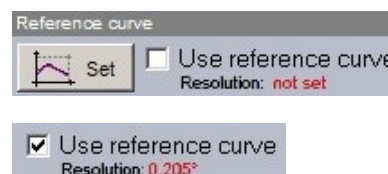
A click on the **Zero** button **removes** the *angular difference* (offset) between the two **sensors** (set angle offset to 0).



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 →

Name **Torsion angle**

output channel is created

Torsional velocity choice →

Name **Torsion velocity**

output channel is created

- **Sensor 2 output channels** section

for on **Sensor 2 output channels** section selected:

Rotational angle choice →

Name **RotAngle_2**

output channel is created

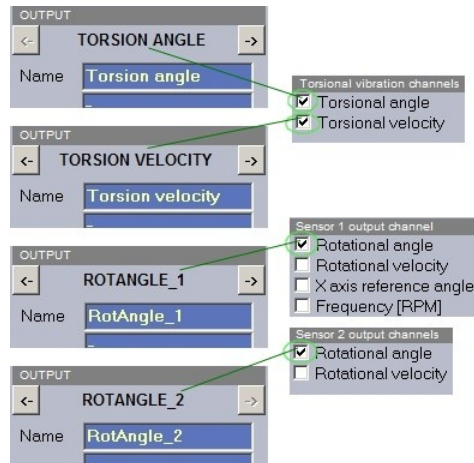
Rotational velocity choice →

Name **RotVelocity_2**

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.

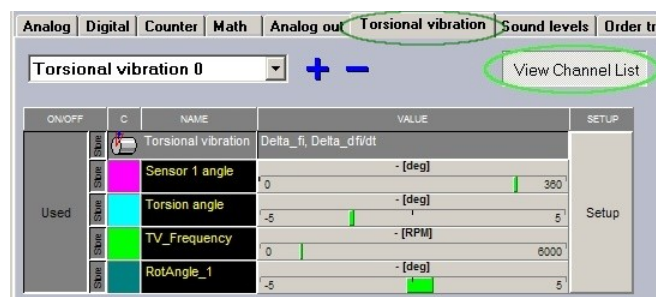
Use **<-** (Previous) and **->** (Next) buttons to navigate between the **output channels** in **OUTPUT channel settings** part of screen.

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.

View Channel List

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 **View Channel List** 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.

1.10.3 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).

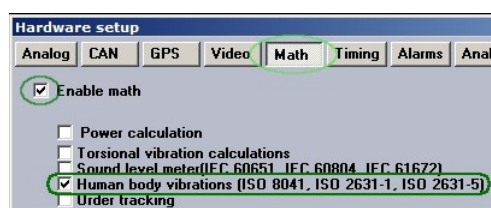
Required hardware	24 bit AD card (Orion1624 or DSA41)
Required software	SE or higher + HBV option, DSA or EE
Setup sample rate	At least 5 kHz

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** →

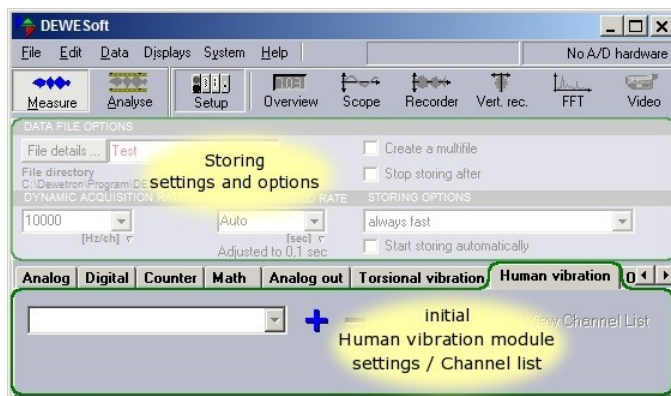
Hardware setup → **Math** tab.



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.

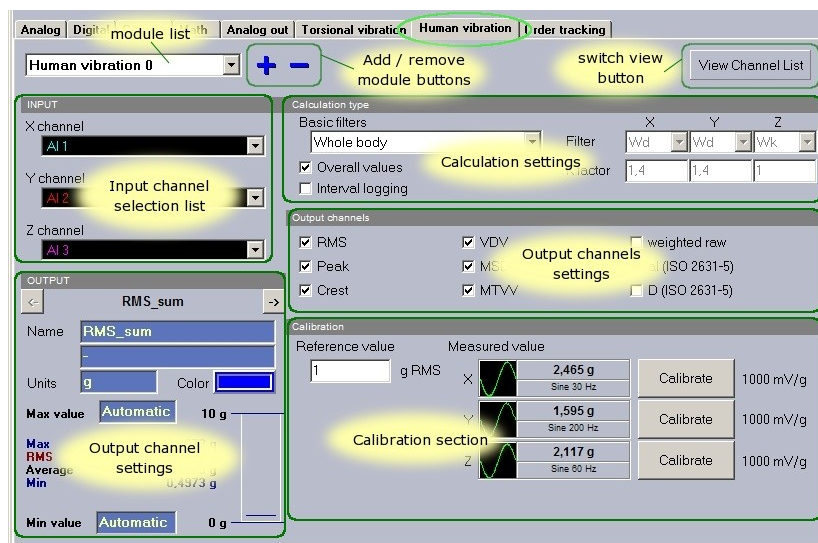
WARNING *Think well before using this command, because we can't return deleted module with settings!*

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:

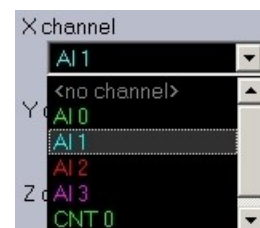
- **Input channel** selection
- **Output channels** - calculated parameters definition
- **Output channel** settings
- **triaxial accelerometer Calibration**
- **Calculation type**

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**.



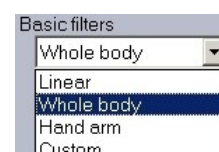
Calculation type

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.

	X	Y	Z
Filter	Wh	Wh	Wh
K factor	Lin	Lin	Lin
	Wb	Wb	Wb
	Wc	Wc	Wc
	Wd	Wd	Wd
	We	We	We
	Wf	Wf	Wf
	Wh	Wh	Wh
	Wj	Wj	Wj
	Wk	Wk	Wk
	Wm	Wm	Wm

- With custom filter we need to define also a *weighting K factor*. This is a *multiplication factor* for each axis when calculation vibration sum.

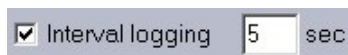
NOTE: We need to take care about the *high pass frequency limit* of the *sensor* and used *amplifier*. For hand arm mode, the *high pass frequency* is **6.4 Hz**, which is easy for any sensor. For the hand arm, the *frequency limit* is **0,4 Hz**, where we need already to choose sensor carefully. We 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.



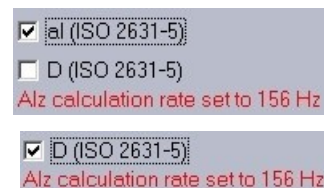
RMS	calculate the <i>root means square</i> value of the weighted <i>signal</i>
Peak	the <i>maximum deviation</i> of the signal from the from the <i>zero</i> line
Crest	the <i>ratio</i> between the <i>peak</i> value and RMS value Crest factor gives an impression about the spikes in the signal; pure sine waves have a crest factor of 1.41 .
VDV	<i>fourth power</i> vibration dose value
MSDV	motion <i>sickness</i> dose value
MTVV	the <i>maximum transient</i> vibration value, calculated in one second interval
weighted raw	<i>full speed time</i> signal weighted with chosen filter We can use those channels for calculation of FFT or CPB spectrum.
al (ISO 2631-5)	the <i>lumbar spine response</i> from excitation measured in all <i>three</i> directions
D (ISO 2631-5)	<i>acceleration dose</i> , measured from the lumbar spine response

NOTE: **al** and **D** are the values based on **ISO 2631-5** which describes the *calculations* and the *limits* for *lumbar spine response* to *vibrations*.

The base for this standard is that the professional drivers of buses or trucks are exposed to vibrations when driving on rough roads or over the bumps. Multiple shocks cause transient pressure changes at the lumbar vertebral end plates that cause damage after years of driving.

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 *Alz calculation rate* is displayed.



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**

Human vibration module can have more **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:

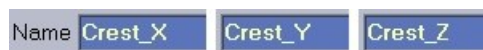


for selected **Interval logging** in **Calculated parameters** section:



Crest **output** channels

for selected **Overall values** in **Calculated parameters** section:



for selected **Interval logging** in **Calculated parameters** section:



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**

al (ISO 2631-5) output channels

independently of selection in *Calculated parameters* section:

Name **alX** **alY** **alZ**

D (ISO 2631-5) output channels

none of choices is selected in *Calculated parameters* section:

Name **alX** **alY** **alZ**

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

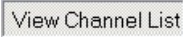
When this button is selected on *Human vibration module* setup, *Channel list* screen with all *defined output channels* appear:

ON/OFF	C	NAME	VALUE	SETUP
		Human vibration	Human body vibrations (ISO 8041, ISO 2631-1, ISO 2631-5)	
Used		RMS_sum_t	- [g]	10
		RMS_X_t	- [g]	10
		RMS_Y_t	- [g]	10
		RMS_Z_t	- [g]	10
		RMS_sum	- [g]	10
		RMS_X	- [g]	10
		RMS_Y	- [g]	10
		RMS_Z	- [g]	10
		filter_X	- []	5
		filter_Y	- []	5
		filter_Z	- []	5
		k_X	- []	5
		k_Y	- []	5
		k_Z	- []	5

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/s²** or **mV/g** (or both) for IEPE sensors and in **pC/g** for piezoelectric (charge) sensors. Therefore we need to **scale** it to *physical quantity*.

The *Reference sensitivity* is the key value to enter in the **DEWESoft** setup. Then it is 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

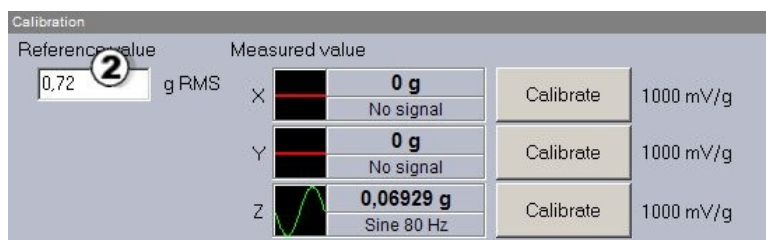
If we want to perform calibration with the *calibrator*, we can perform it here - in *Human vibration* module itself.

First enter the **Reference value** ② of vibration. This can be read from the **calibrator**.

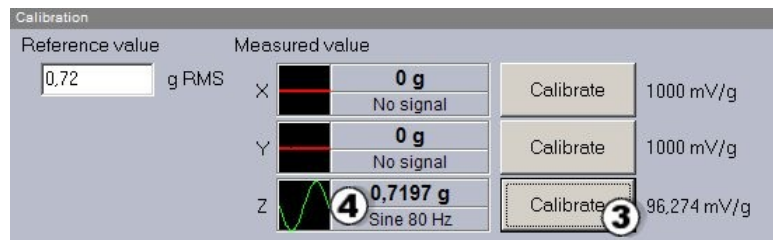
Example: Usually the calibration *levels* are 10 m/s² peak, which is 7,07 m/s² RMS or **0,72 g**. We need to enter this value in '**Reference value**' field.

As soon as we *mount* the *sensor* on the calibrator, we should see the amplitude and the frequency of the *signals*. In this case we see that we applied the sensor in **Z** direction.

The frequency of calibration is below 200 Hz (80 or 160 is typical, in our case it is **80 Hz**). This is a simple check that everything is ok.



Next simple step is to press the **Calibrate** button 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.

1.10.4 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...

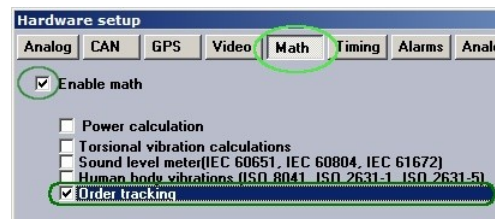
Final measured vibration The ratio between *excitation* and *system response* is defined by the system transfer curve. So the final measured vibration of the system is a product of *excitation force* and *system transfer curve*. Since the transfer curve is fixed, we get *different* responses for excitations 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	Orion
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](#)

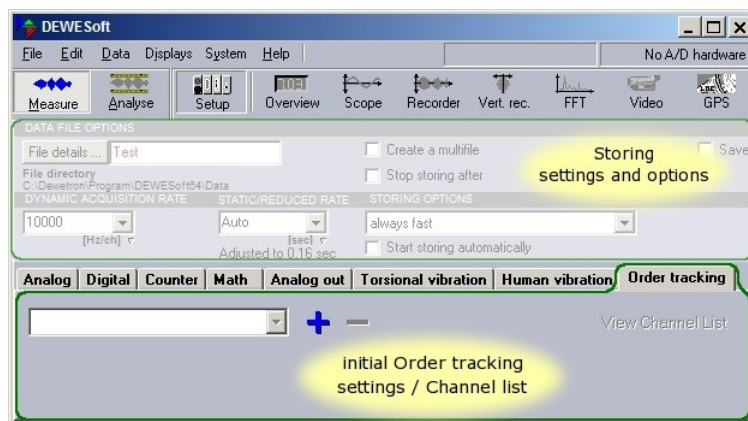
After *required hardware - accelerometer installation and setup* and **DEWESoft Counter Hardware setup** (this procedure is to perform *only* by installation new or changing hardware) according to manufacturer instructions, **DEWESoft Order tracking** measurement is allowed by *selecting* the **Order 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.

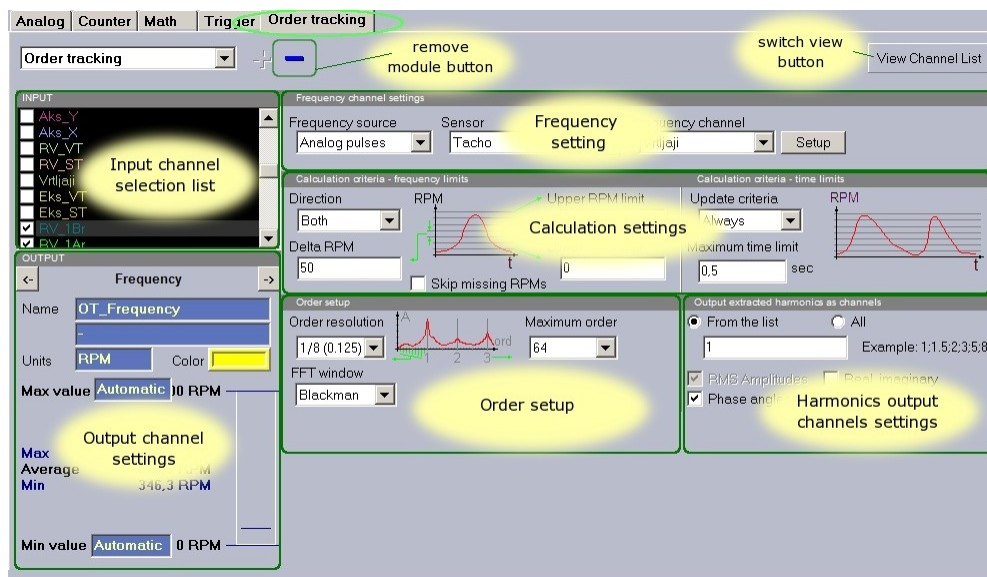
Delete Order tracking module

Module can be deleted by *selecting* him and then using - minus button.

WARNING Think well before using this command, because we can't return deleted module with settings!

Setup

When an order tracking is added, we get the following pictures with the **settings** for *order tracking*.



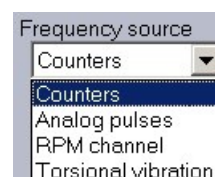
This screen has the following main sections:

- **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*.

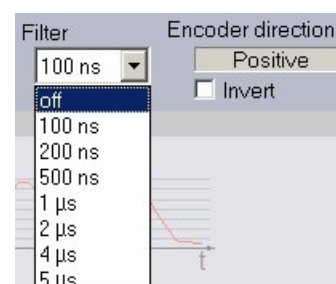


We can select any counter *sensor*.

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 right 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**

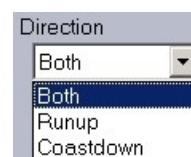
- Third option is the **RPM channel**. In this case the frequency source comes *directly* from *analog channel* which measures the *frequency* (like DAQP-FREQ-A).
- Fourth option - **Torsional vibration** is available when we have torsional vibration. The source can come *directly* from torsional vibration.

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.

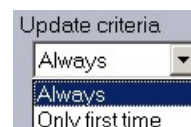
Here can we also **Skip missed RPMs** with checking this box.



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*.

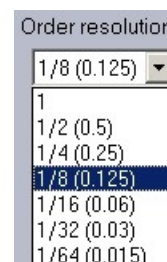


We can also define the **Maximum time limit**. If the frequency doesn't *change* within specific time, we will *still get another* point for harmonics and 3D history.

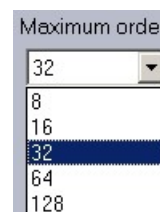
Order setup

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**



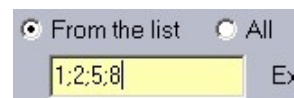
Output extracted harmonics as channels

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*.

1.10.5 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	16 bit AD card (Dewetron Orion-1616 board) and DEWE-CRANKANGLE-CPU
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:

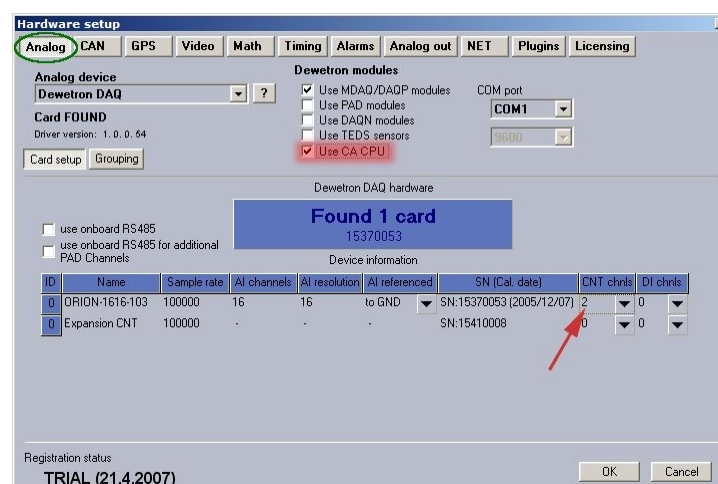
- Activate hardware** or modify hardware settings:
- *analog and counter channels* on **Hardware setup** window with **Analog setup**
 - **Counter setup**

- Combustion module**
- **Enable module**
 - **Add module**
 - **Delete module**

Combustion analysis setup select **CA type**: **Engine**, **Angle sensor**, **Calculations**, **Heat release**, **Knock detection** and then define all settings

Activate hardware

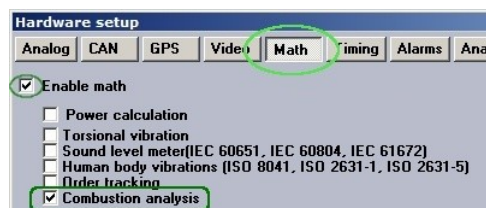
To **activate** your hardware or *modify* hardware settings, please select Dewetron DAQ (Orion – 1616) from the drop down list in the **System** menu → **Hardware setup...** → **Analog** tab:



If we want to use **Combustion analysis** in *external clock* mode, **Use CA CPU** option must be checked.

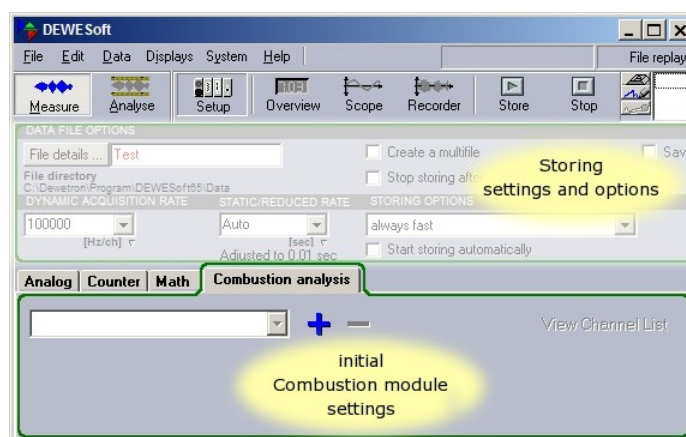
Enable Combustion analysis module

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**'.



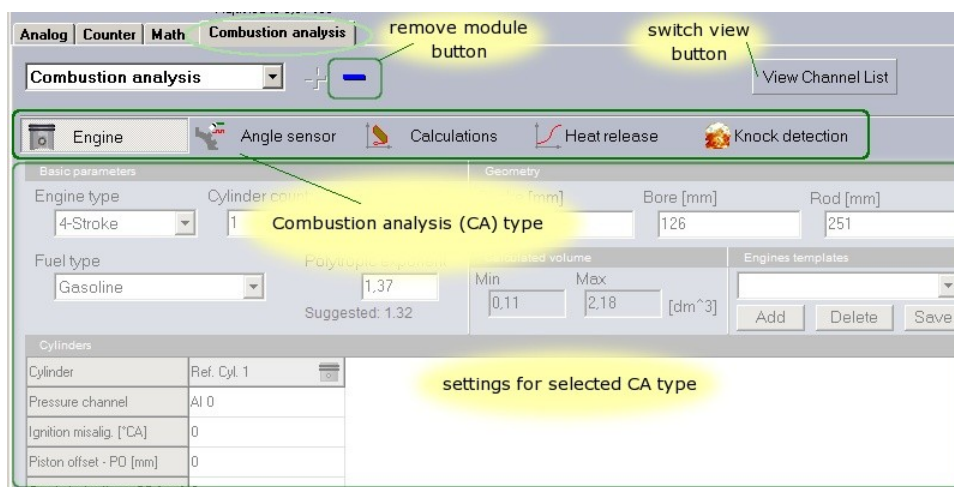
Delete Combustion analysis module

Combustion analysis module can be deleted by selecting him and then using  - minus button.

WARNING Think well before using this command, because we can't return deleted module with settings!

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

Geometry		
Stroke [mm]	Bore [mm]	Rod [mm]
166	126	251

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.

Calculated volume	
Min	Max
0,11	2,18 [dm ³]

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.

Engines templates	
Add	Delete Save

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 misalign. [°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 misalign. [°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).

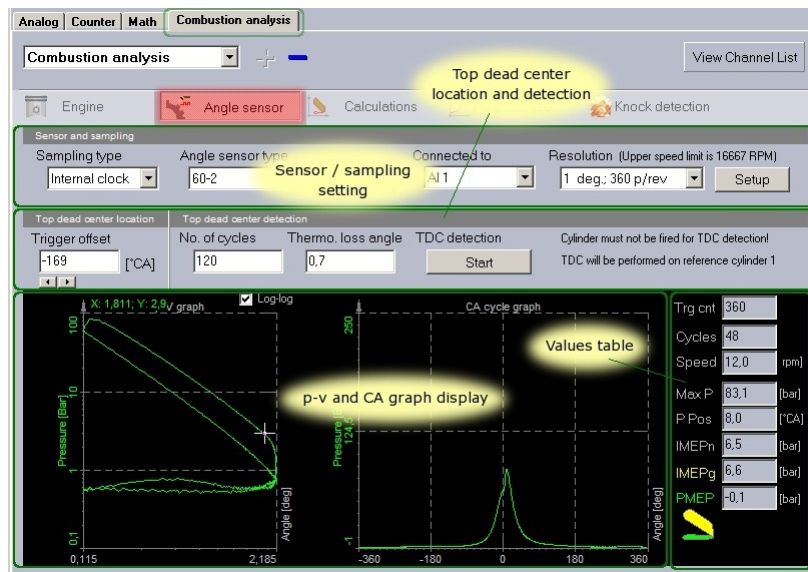
Additional channels

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.

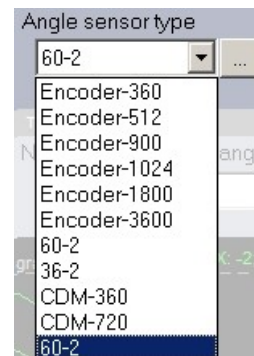
External clock means that data will be acquired in *angle domain*. We need the CA CPU hardware to *create needed number* of pulses from any sensor.

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

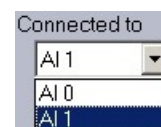
Angle sensor depends what sensor we connected.

When  button is selected, the **Counter sensor editor** window appears to define *counter sensor*.



Connected to

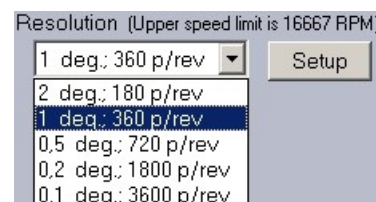
Here we define *physical connection* of the *sensor*. If use *external clock* with CA-CPU, it will show the CA-CPU *inputs*. If we use *internal clock*, it will offer the *counter inputs* for *encoder* and CDM sensors and *analog inputs* for *gear tooth with double or missing teeth*.



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  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 with  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

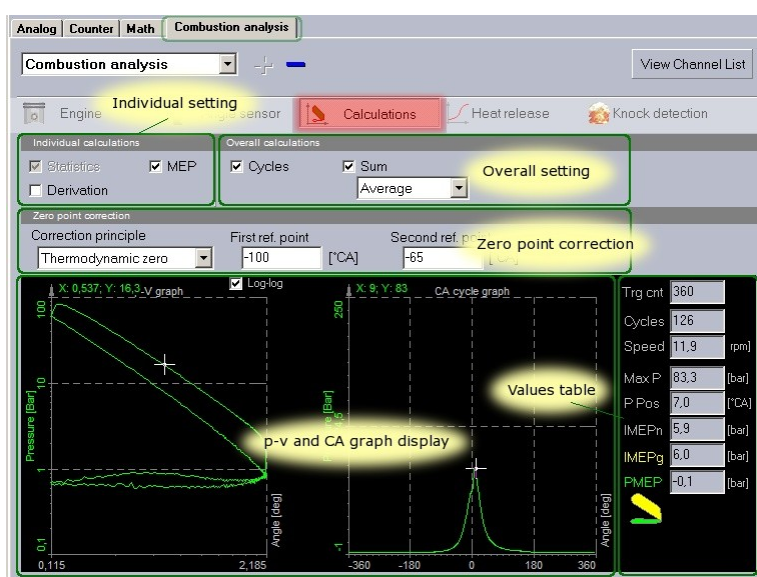
When **Start** button is selected, table with *values* appear instead on right side table:

TDC detection	Min	Ave	Max	Std	Cycle	[°CA]
Cancel	0	0	0	0	0	

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:



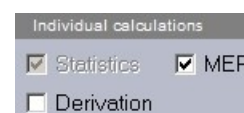
Settings and entered values for this **Calculations** CA type are divided on sections:

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.

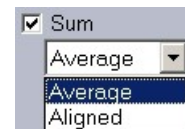
MEP option adds *net*, *gross* and *pumping* MEP values.



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).

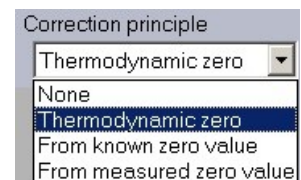


Zero point detection



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

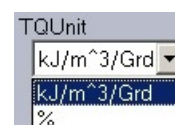
Heat release is very intensive calculation and we can *limit the range* of calculation *only around top dead center*.

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.

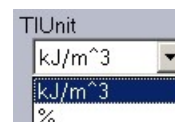
TQUnit

We can either have the *physical* unit for the heat release or have it expressed in *percentage*.



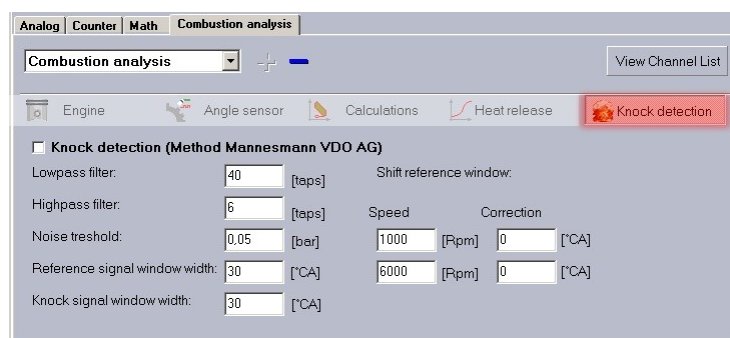
TIUnit

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.

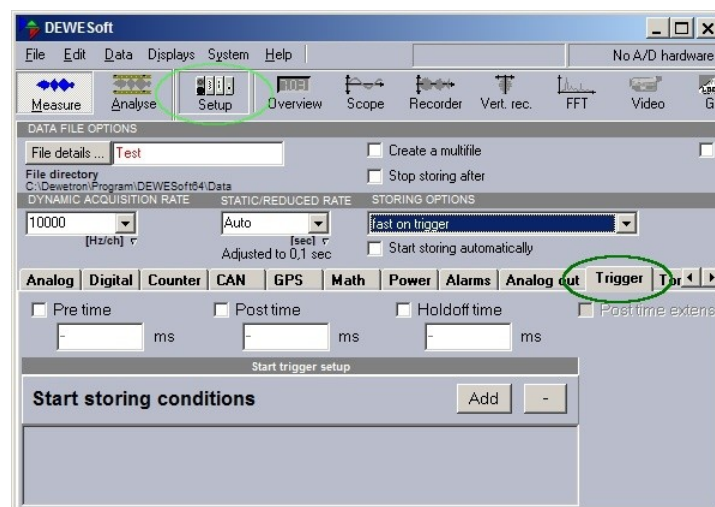
Shift reference window

Is two point linear scaling offset definition how much the calculation window is *shifted* for different *speeds*.

1.11 Trigger setup

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

When we select *trigger type storing* from the **STORING OPTIONS** drop down list on the **DEWESoft Setup** screen, a **Trigger** tab will appear and it will *automatically* open screen to **set up** triggers:



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

- **Timing 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*
- **Remove trigger**
- **Trigger controls** how *activate / deactivate* triggers

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 when the pressure grows above a certain limit. But you only want to store the data if there is really a part in the machine. Then you can connect a signal to another channel which reports if the unit is present or not and use this result as a Don't store function.

Stop trigger setup

The **Stop trigger condition** and **Setup** works in exactly the same way 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*.

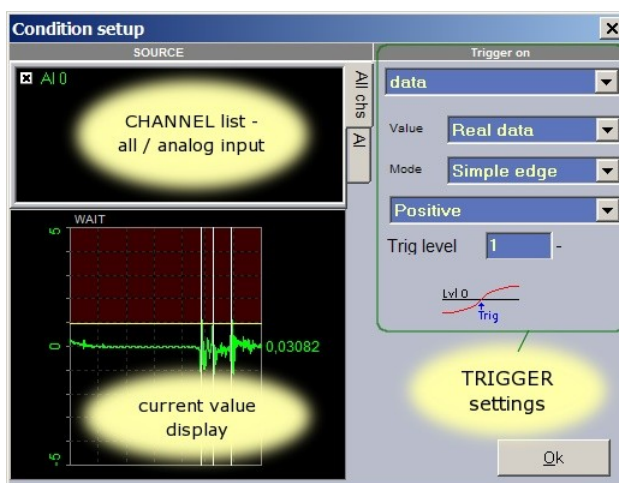
If *several* trigger events appear, all of them will be stored into the *same* file. **DEWESoft** offers a special **analysis** feature in the **Analyse mode** for an easy data analysis.

for information about **triggered data files** see → [User Guide](#) → **Reload triggered file**

1.11.1 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.

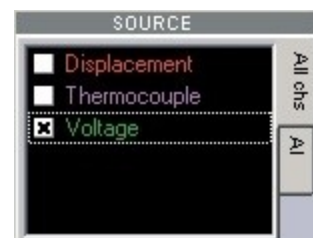
When we select the **Setup** button on the **DEWESoft Setup screen** - *trigger condition line*, the following *Condition setup* window appears:



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 *white* lines displayed indicate when 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.

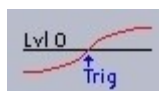
The trigger event is a *rising* or *falling* edge, which crosses a defined level.

Positive slope:

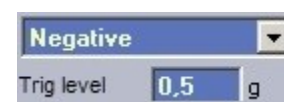


Trigger on *rising* edge when signal rises over defined **Trig level**.

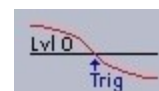
symbolic display of trigger condition
(below settings)



Negative slope:



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



• 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*.

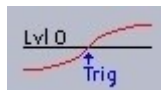
This trigger type is mostly used with very *noisy* signals.

symbolic display of trigger condition
(below settings)

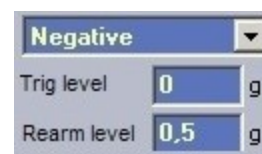
Positive slope:



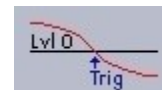
Trigger on *rising* edge when signal rises over defined **Trig level**; retriggers *only* when **Rearm level** has been crossed.



Negative slope:



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



• 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*.

symbolic display of trigger condition
(below settings)

Enter range:



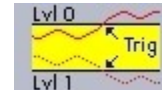
Trigger when the signal *enters* the window - signal *falls below* **Upper level** or *rises above* **Lower level**.



Leaves range:



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



• 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*

symbolic display of trigger condition
(below settings)

Positive pulse:



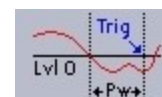
Trigger on *rising* edge when signal rises over defined **Trig level** and *stays above* this level **longer than** selected **Time**.



Negative pulse:



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



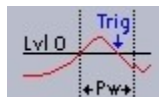
Shorter than Time

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

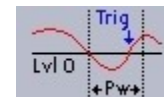
symbolic display of trigger condition
(below settings)

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 *below* defined **Trig level**, but *rises above* this level **earlier** than selected **Time**.



- 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.

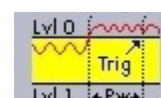
Longer than Timelevel **In range**

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**.

symbolic display of trigger condition
(below settings)

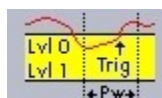
level **Out of range**

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**.

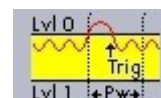
**Shorter than Time**level **In range**

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.

symbolic display of trigger condition
(below settings)

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.



- Slope

Smoother than Delta time

Positive slope

Mode: **Slope**

Positive slope

Delta level: **0.5** g

Smoother than

Delta time: **10** ms

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

Negative slope

Negative slope

Delta level: **0.5** g

Smoother than

Delta time: **10** ms

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

Any slope

Any slope

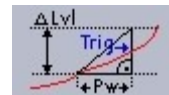
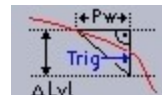
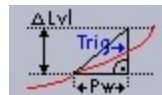
Delta level: **0.5** g

Smoother than

Delta time: **10** ms

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

symbolic display of trigger condition
(below settings)



Steeper than Delta time

Positive slope

Positive slope

Delta level: **0.5** g

Steeper than

Delta time: **10** ms

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

Negative slope

Negative slope

Delta level: **0.5** g

Steeper than

Delta time: **10** ms

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

Any slope

Any slope

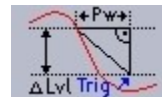
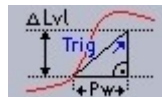
Delta level: **0.5** g

Steeper than

Delta time: **10** ms

Triggers when signal rises over or drops *below* defined **Delta level** *earlier* than defined **Delta time**.

symbolic display of trigger condition
(below settings)



Time trigger setup

time **equal to**

absolute (time only)

Trigger on: **time**

Time format: **absolute (time only)**

Time: **equal to**

15 : 12 : 45 . 500

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

displayed time

00:03:48.526

level **every**

Trigger on: **time**

Time format: **absolute (time only)**

Time: **every**

2 **m**

The system triggers **every** defined **time** x [unit], the time starts running from the *beginning* of measurement.

07:12:10

relative

time **equal to**

Trigger on

time

Time format

relative

Time

equal to

5 m

The system triggers when the defined **time** x [unit] has been *passed* since the measurement has been started.

level **every**

Trigger on

time

Time format

relative

Time

every

2 m

The system triggers every **time** x [unit], the time starts running from the *beginning* of measurement.

Time format select from drop down list:

Time format

relative

relative

absolute (time only)

Time select from drop down list:

Time

every

equal to

every

[unit] select from drop down list:

m

s

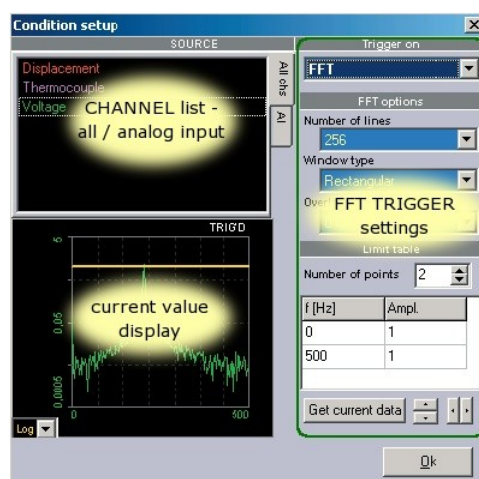
m

h

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 **Logarithm** (see display above) also **Linear** display.



Limit table

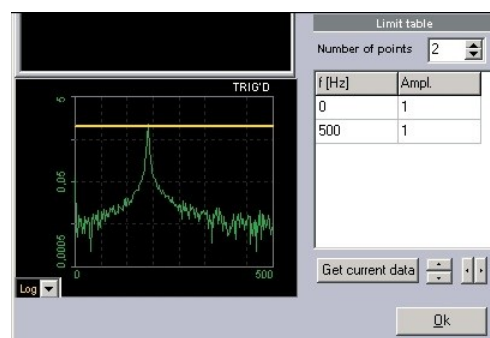
After you have done your FFT option settings you have to **define**:

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

Default (standard) **Number of points** are **2**.

The standard **Ampl. level** is **1** for **0** and **max. frequency**.

You can click on this field and change this value.

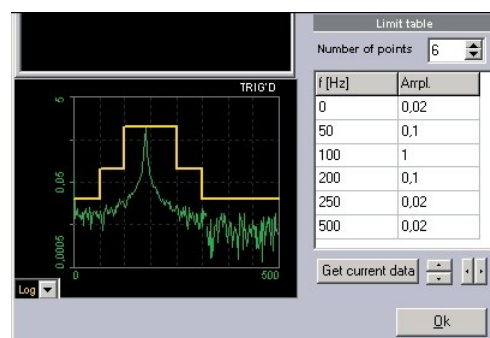


You can increase **Number of points** to expand the table (e.g. **6** in example on right).

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**.

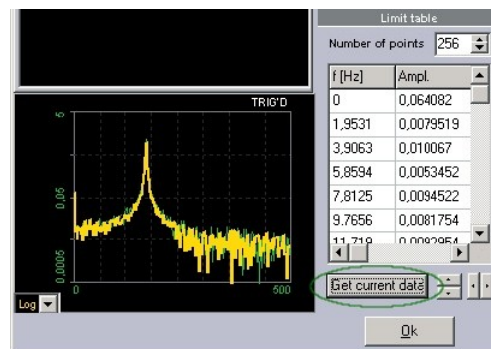


2. Take current measurement from the system

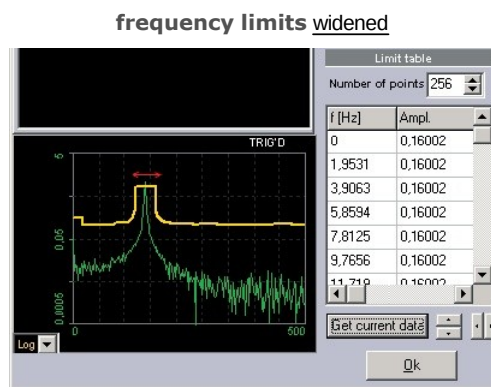
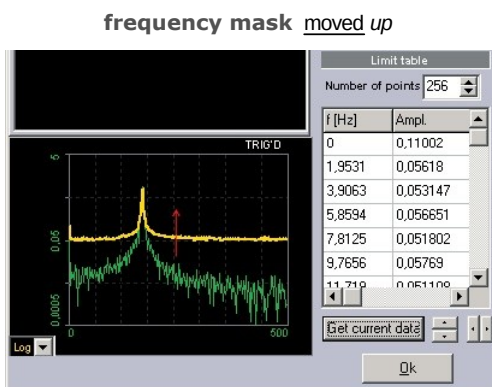
The second way is to take a **frequency mask** out of the currently displayed signal. To do that simply presses the **Get current data** button.

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 **up** / **down** button to **move** the **limit up** / **down** and the **left** / **right** button to **add** / **remove limit** in **frequency domain**.



Examples:



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

1.12 Sensor database

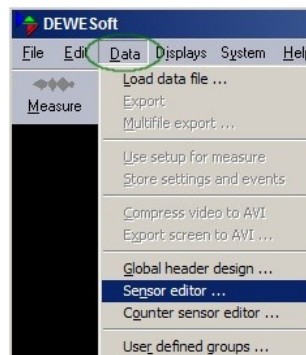
DEWESoft offers **sensor database** which holds the list and the properties of all sensors which will be used. In addition to the linear **scaling**, which can be done also in the input **Channel setup**, the sensor database offers also **scaling** by table or polynomial, even transfer curves can be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

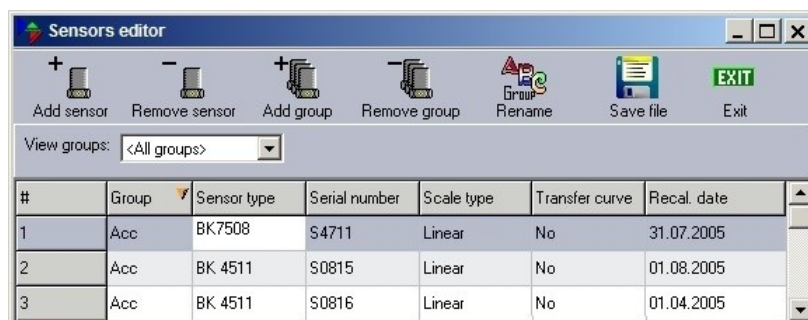
Each **sensor** is **defined** with **information** in **database**:

- **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 <i>View groups</i> selection is set to <All groups> . Use it to move a sensor into <i>another group</i> .
Sensor type	The sensor type should contain the name or type of the sensor, but can contain <i>any</i> text desired.
Serial number	The serial number has to be <u>unique</u> ! It is not allowed to use the same serial number (even in different groups), because this information is used for sensor <i>identification</i> .
Scale type	The sensors editor differs between <i>linear</i> , <i>polynomial</i> and <i>table scaling</i> .
Transfer curve	Can be used for sensors with <i>phase</i> characteristics like current clamps...
Recal. date	Enter the date when the sensor has to be <i>recalibrated</i> . When the date expires, the <i>Recal. date</i> will change to red color.

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

The screenshot shows the 'Sensors editor' window. At the top is a toolbar with icons for adding/removing sensors and groups, renaming, saving, and exiting. Below the toolbar is a 'View groups' dropdown set to 'Current'. A table lists sensors with columns: #, Sensor type, Serial number, Scale type, Transfer curve, and Recal. date. The first sensor (ID 1) is selected. Below the table are tabs for 'GENERAL', 'SCALING', and 'TRANSFER CURVE'. The 'GENERAL' tab is active, showing a table of properties and values:

Property	Value
Physical (input) unit	
Electrical (Output) unit	
Channel name	
Channel description	
Wanted range min	-5
Wanted range max	5
Wanted filter [Hz]	0

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 <i>enabled</i>

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:

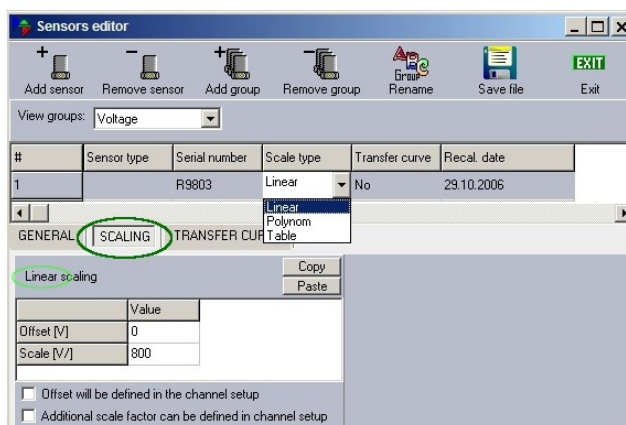
Physical (input) unit	the <i>physical</i> unit of the sensor, e.g. V , A , °C , mm , psi , % ,...
Electrical (output) unit	the <i>electrical</i> output unit of the sensor, most times V or A
Channel name	use this field to <i>pre-define</i> the channel name for the <i>setup</i>
Channel description	additional information about the sensor, helps to select the right sensor in the input setup
Wanted range min	<i>Minimum</i> electrical input range
Wanted range max	<i>Maximum</i> electrical input range
Wanted Filter [Hz]	desired <i>lowpass</i> input filter range

Example of general sensor setting:

Property	Value
Physical (input) unit	A
Electrical (Output) unit	V
Channel name	Current
Channel description	Current measurement
Wanted range min	-10
Wanted range max	10
Wanted filter [Hz]	10000

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

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

... enter this value in **Scaling** tab - **Linear scaling**

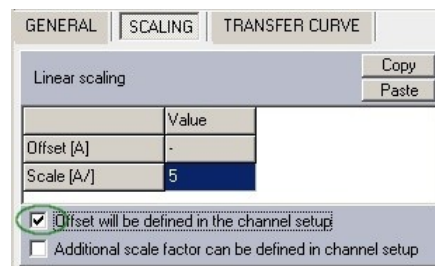
Example → see above

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

$$y = (a_0 + a_1 * x + a_2 * x^2 + \dots + a_n * x^n)$$

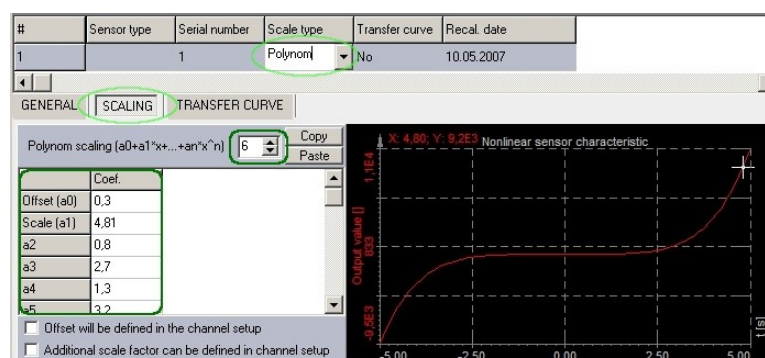
Option:

- **Polynom scaling** ($a_0 + a_1 * x + \dots + a_n * x^n$)

in this field enter **n** and **coefficients** in **Coef.** column (coefficient **a0** = offset)

- other fields: same as for *Linear* (see above)

- example:

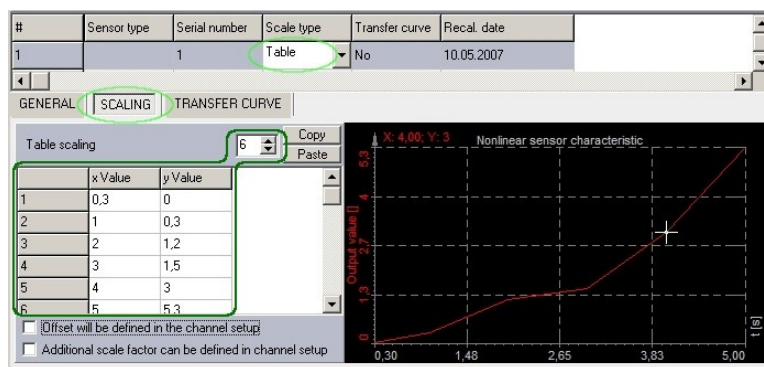


- **Table** also used for **nonlinear** sensors, but normally *easier* to enter because most *calibration information* contains *several calibration points*

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:



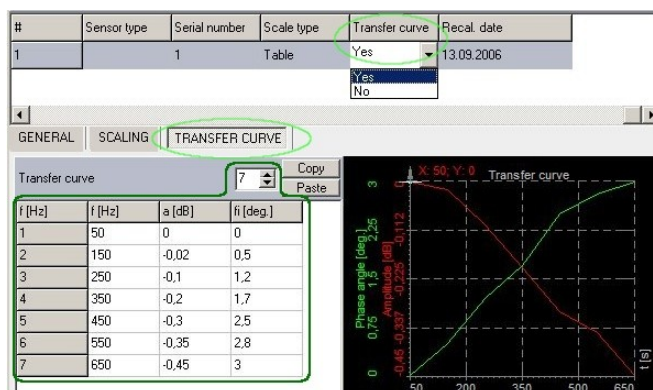
TRANSFER CURVE

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:

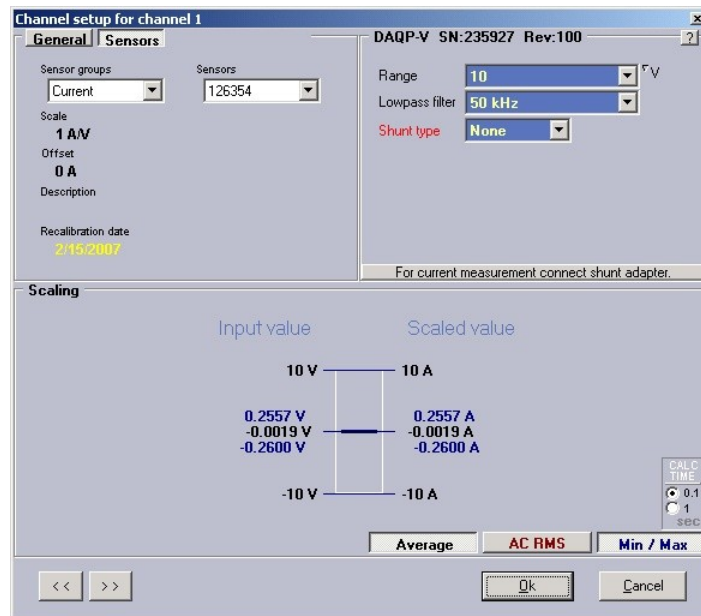
- **Transfer curve** in this field enter *number of points* (rows of table) and in in table below **f[Hz]** - signal frequency, **a[dB]** - amplitude deviation in dB and the **fi[deg.]** - phase angle in *degrees*
- **Copy, Paste** usually *Windows* operation to **copy** / **paste** value to / from another 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**.

When we go back to **analog setup** 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. We only 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**.



Manage sensors and sensor group

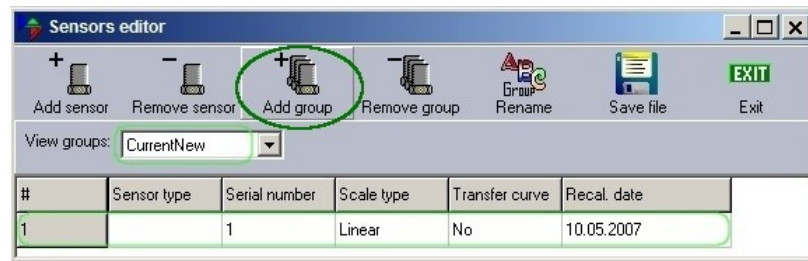
Add group

Before you start editing sensors you should think a few seconds about **grouping** 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*:



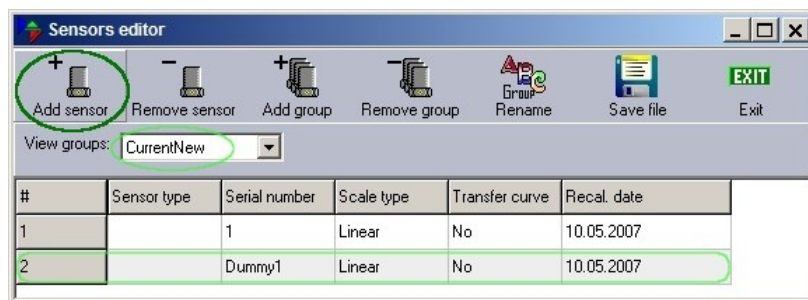
When you press the **OK** button, the **new group** will be **created**, the **View groups** selection will change to the **new group name** and **new sensors** will be **created automatically**:



NOTE *Sensor groups which contain no sensor will be automatically deleted.*

Add sensor

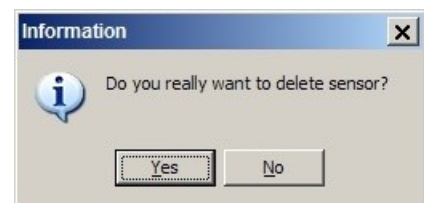
If you already have an existing sensor group and just want to **add new sensors**, select the *desired group* from the **View groups** selection and press the **Add sensor** button and *new row - sensor* is added to table:



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.

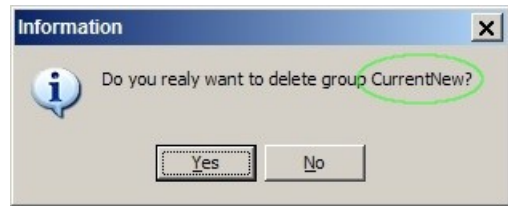
A warning will appear and ask if you really want to delete the sensor:



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:



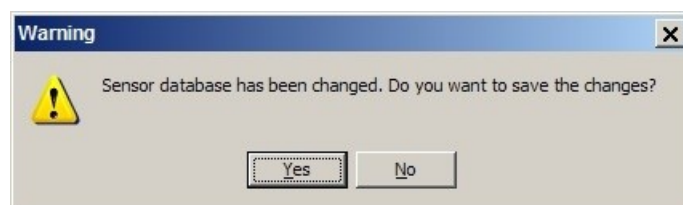
Save file (sensor database)

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.



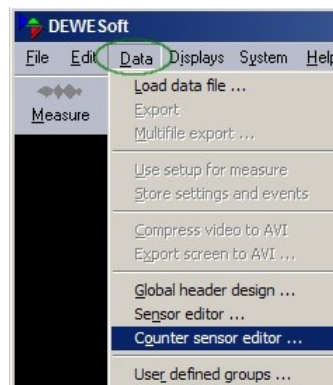
Yes → **save** changes or **No** → *leave without* changes

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

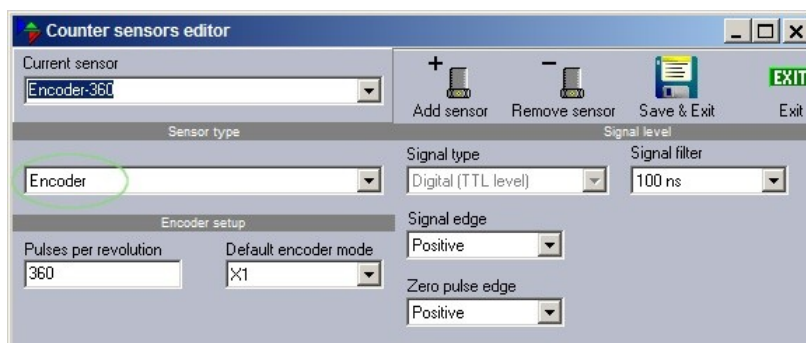
1.13 Counter sensor editor

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*, *gearthooth* and other *angle sensors*.

The *Counter sensors editor* can be accessed via **Data** main menu → **Counter sensor editor ...** item.



When this menu item is selected, the *Counter sensor editor* window appears to define *counter sensor*.

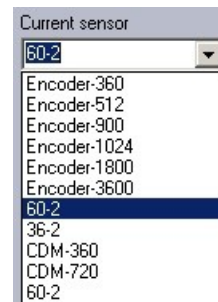


There are *several predefined sensors* already added when first starting **DEWESoft**, but we can always **add** new ones, **modify** existing or **delete** the sensors. On upper right side of *Counter sensor editor* window common command icons appear to:

- 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:

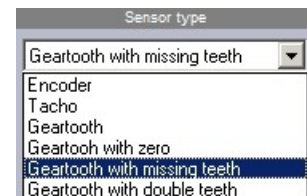
Sensor type

Signal level

Encoder setup or **Geartooth setup** (depend on selected *Sensor type*)

Sensor type

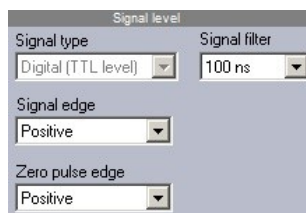
There are several basic *sensor types* available which can be selected from *Sensor type* drop down list :



Encoder	classic angle encoder with A, B and Z signals, can be <i>only digital</i> and used with Orion counter
Tacho	<i>one pulse</i> per revolution, can be either <i>analog</i> level or <i>digital</i>
Geartooth	sensor with <i>defined number of pulses</i> per revolution, but <i>without any zero pulse</i>
Geartooth with zero	sensor with <i>defined number of pulses</i> per revolution <i>with zero pulse</i> , sometimes also called CDM sensor
Geartooth with missing teeth	a classic in-vehicle sensor with <i>any number of pulses</i> where some teeth are <i>missing</i> for <i>zero pulse</i> recognition. A typical example is the geartooth with 60 teeth where <i>two</i> 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 <i>any number of pulses</i> per revolution with <i>some double teeth</i> . A typical example is 36+1

Signal level

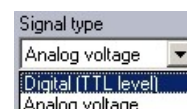
There are several signal level settings.



From drop down list can be selected:

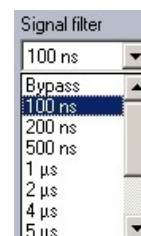
Signal type

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*.



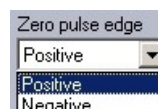
Signal edge

can be either *positive* or *negative*.

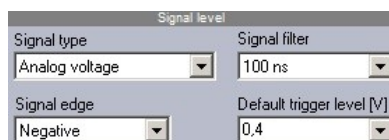


Signal edge

For *encoders* and *geartooth* with *zero* we also define the **Zero pulse edge** which can and can be either *positive* or *negative*.

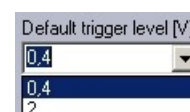


For *analog sensors* we also define the **trigger level**.

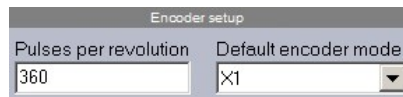


Default trigger level

Please note that these are *default* values and can be *adjusted* when the sensor is used in *each specific* application.



Encoder setup



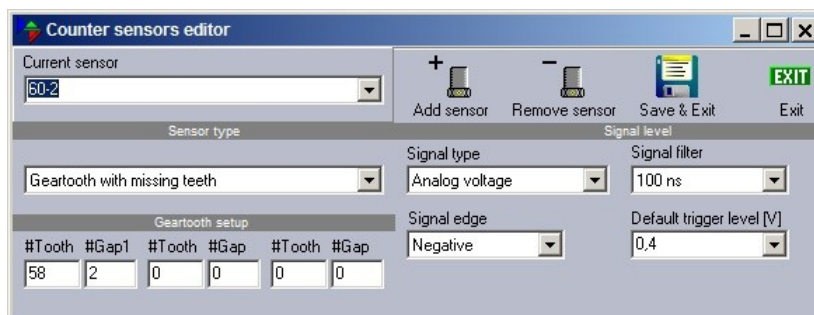
For *encoder sensor* we define:

- 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 → **Encoder**.

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 setup					
#Tooth	#Gap1	#Tooth	#Gap	#Tooth	#Gap
58	2	0	0	0	0

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.

Geartooth setup	
#Tooth	#Double teeth
36	1

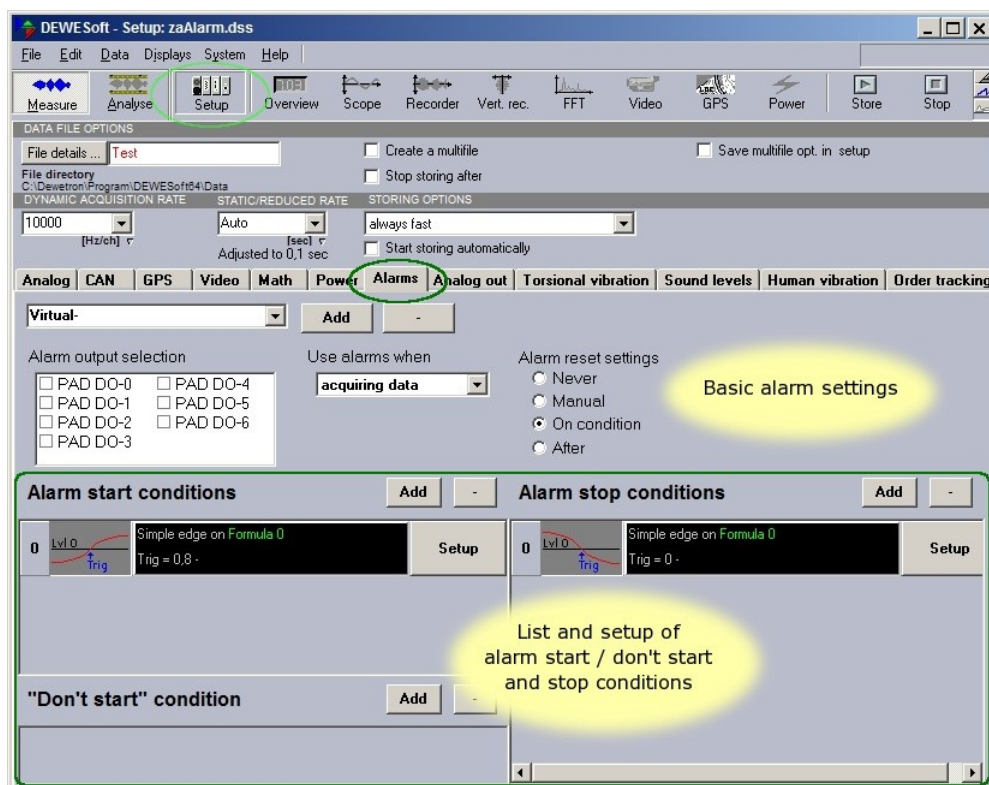
1.14 Alarm Monitoring

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* or *PAD-DO7* modules.

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*)

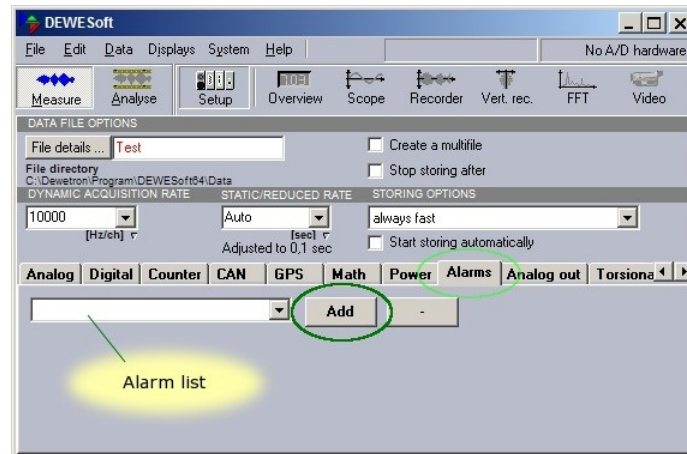
Alarm conditions to **Add (New) / Change / Remove** and define **Alarm start condition** or **"Don't start" condition** and eventual **Alarm stop conditions**



Basic settings

Add (New) / Delete alarm monitoring channel

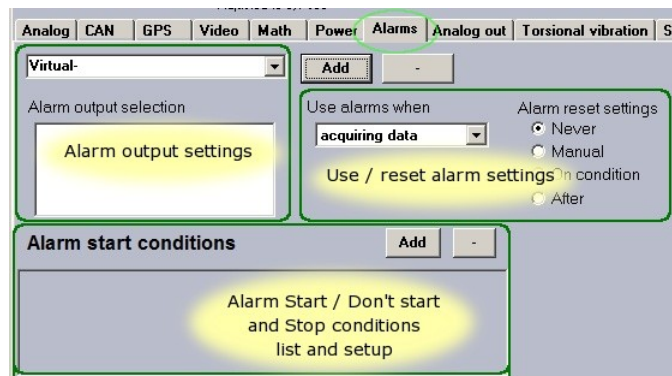
When you select **Alarms** tab to define **new Alarm**, on lower part of screen an *empty* alarm list appear:



with *two icons*:

- **Add** 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

after checking one *DAQ output* on
Alarm output selection drop down list
change to:

after checking two *DAQ outputs* on
Alarm output selection drop down list
changes to:
this will output same channel on two
physical outputs

3. the **Enable alarm monitoring** and **Use PAD DO-7 on address** box are checked

if no alarm is checked, the alarm will be
virtual - only available for display

after checking one *PAD output* the alarms
will be output on selected channel

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

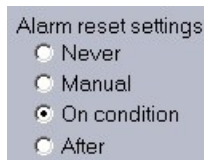
Select from **Use alarms when** drop down list:

- alarm while **acquiring data**
- alarm during **storing**



Alarm reset settings

Select appropriate choice from displayed list:



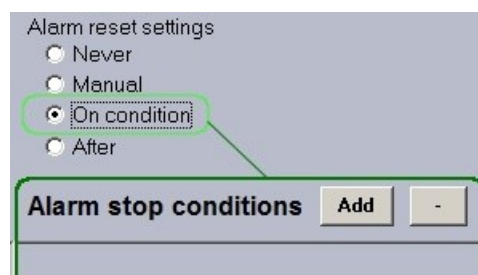
- Never** when *Start condition* carry out, **Alarm** remain **on** to end of acquiring or storing data
- Manual** when *Start condition* carry out, **Alarm** go **on** and on first row of display screen red **Alarm ON** button appear:



To *manual* alarm **reset** alarm click on this button and display change to:

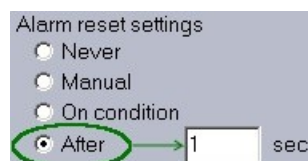


- On condition** next to **Alarm start condition** and "Don't start" condition on right lower part of screen also **Alarm stop conditions** section appear:



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:



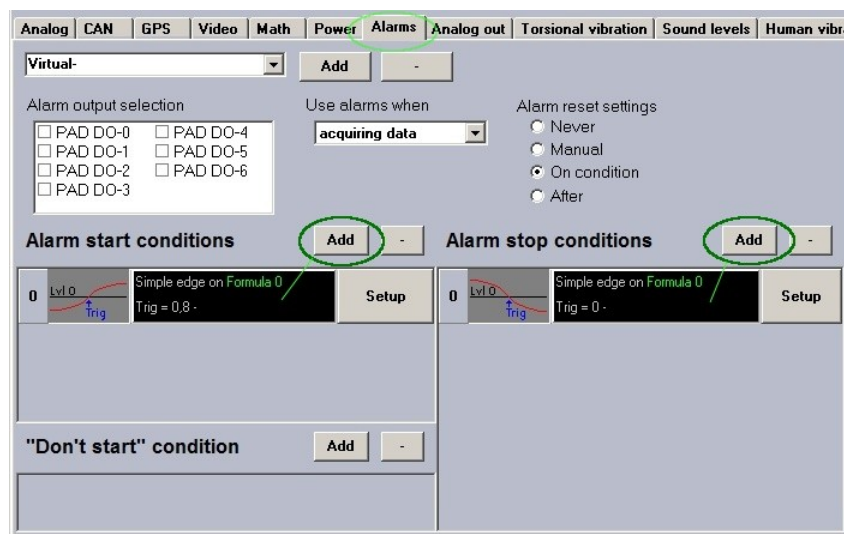
Set Alarm conditions

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** - **Add** alarm button beside *Alarm start / stop condition* or "Don't start" condition, a new alarm condition line is displayed:

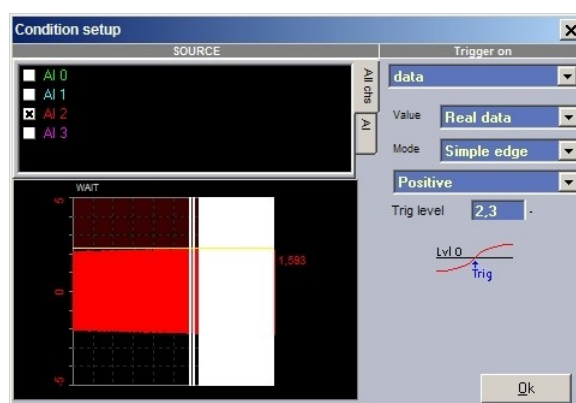


The **alarm monitoring settings** are similar to *trigger* conditions → see also → [User Guide](#) → **Setup - Triggers**

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 to 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.

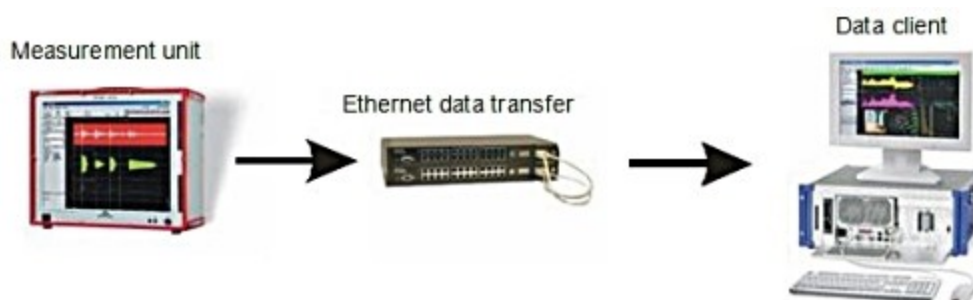
1.15 NET acquisition

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 Dewetron **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**

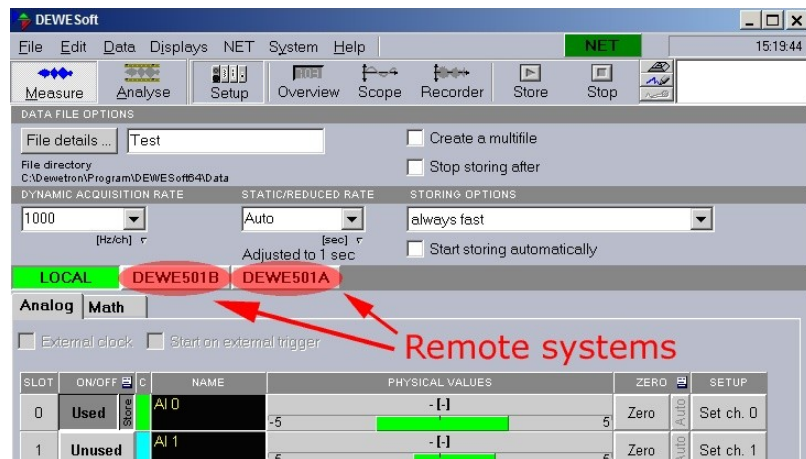
In this *Remotely controlling a Slave MU* section we are using only the **master client** computer. The Dewetron **MU** is set to **Slave MU** mode, and we have already *connected* to it using the steps from the **Measure-Connecting to the remote** units section.

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 subtle 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**, **DEWE501B** and **DEWE501A**. 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

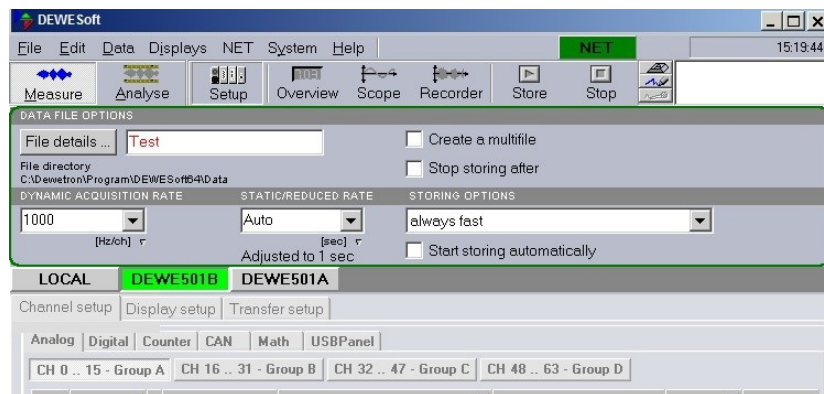
However, **LOCAL** computer still has a **Math** tab. It is interesting to note that you can *perform math functions* in *real time* on this **client** using any channels that are *transferred* from the **MU's**!

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.

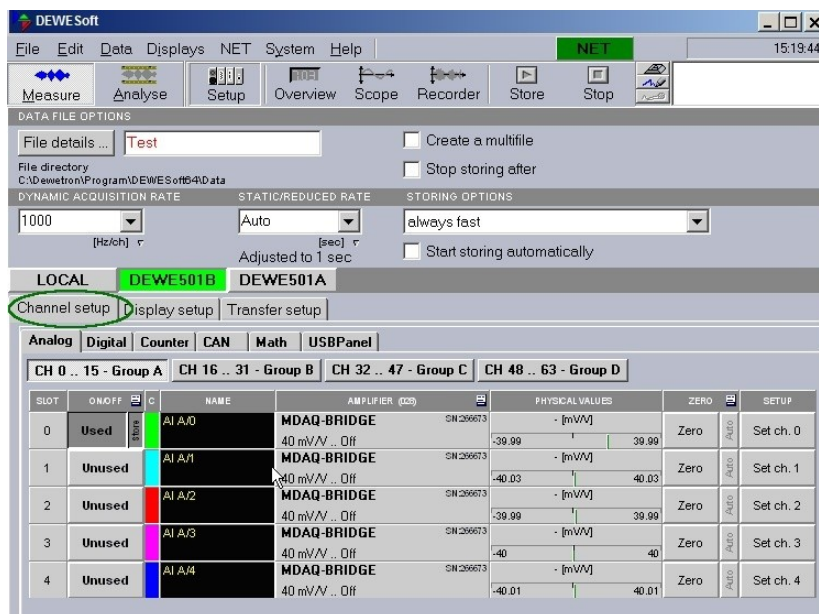
On central part of this screen you are doing exactly what you would be doing if **DEWE501B** was a *stand-alone MU*: set the *dynamic* and *reduced sample rates*, choose a *filename*, set *storing option*... → see [User Guide](#) → **Recording Setup**

Note the *three tabs* under **DEWE501B**:

- **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 **DEWE501B** 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, **DEWE501B** 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 means **transfer 'which channels** will be **sent** across the **network** during recording, for **display** on the **client**'.

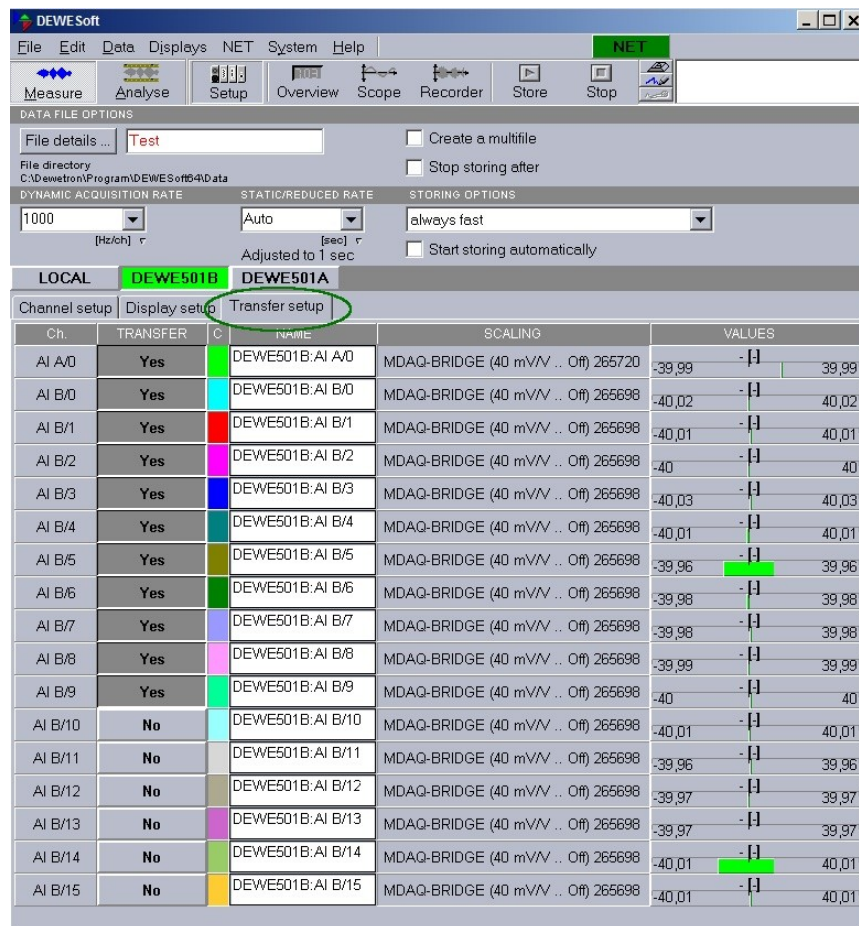
That is the entire scope of what transfer means. It has no effect on the actual **Storage of data on the client** (assuming that **local** storage is *enabled* – the default and highly *recommended* mode in **System** menu → **Hardware setup...** → **NET** tab).

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 DEWE501B 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

2 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... acquisition, Power, Torsional vibration, Sound level,...)

see → [User Guide](#) → **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 on what conditions *data* are *stored*

see also → [User Guide](#) → **Recording Setup**

3. STEP

Storing data

start measure and *store data*: Manually, Triggered, Remotely or Automatically

see also → [User Guide](#) → **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

2.1 Displays design

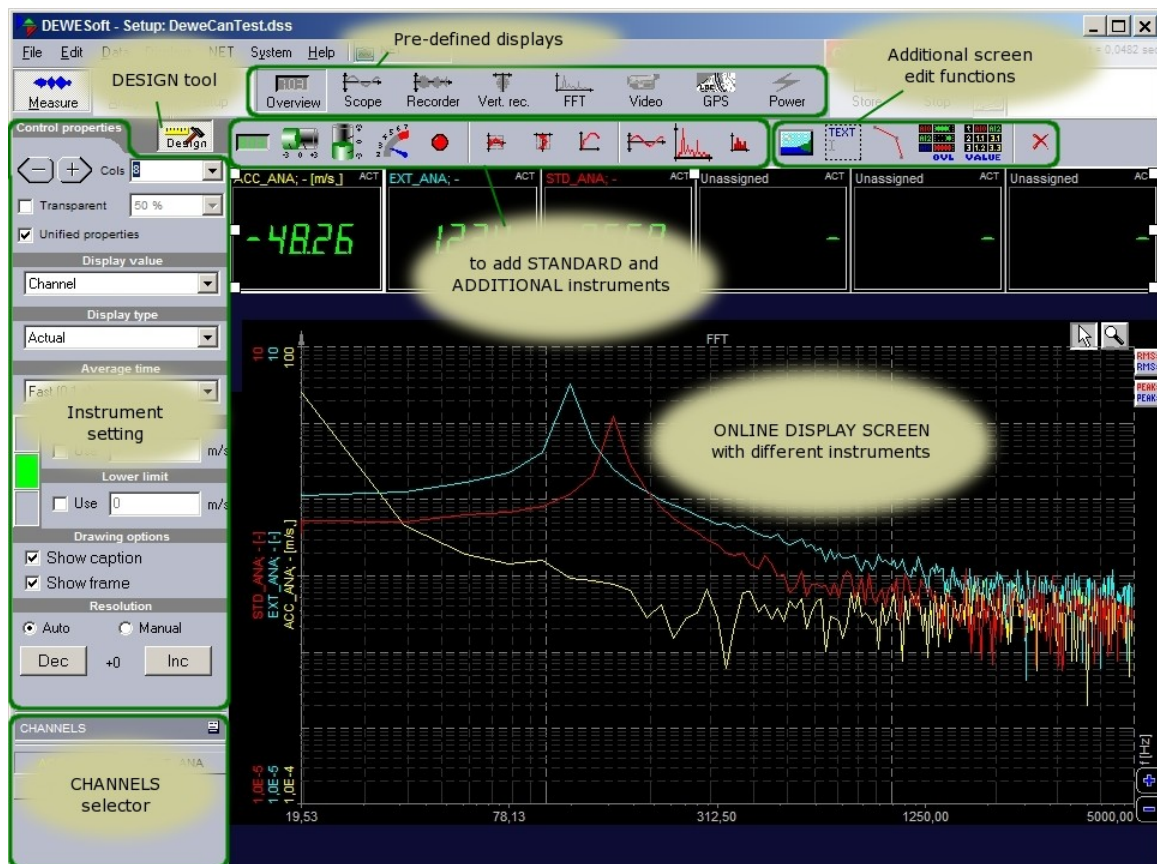
Primary goal of **DEWESoft design** online **display screen** is to **create** *clear* and *intelligible appearance* of acquired and calculated data in different instruments on screen for *review* and *analyzing* of these data. **DEWESoft** allows you to **set up** *different instruments* for each input, 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.

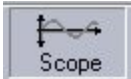
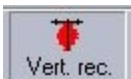
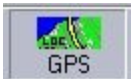
DEWESoft has few **pre-defined** displays but these screens can be altered and **own specific displays** can be **created** with different appearance. The picture below shows typical display with standard elements for designing the display.



2.1.1 Pre-defined displays

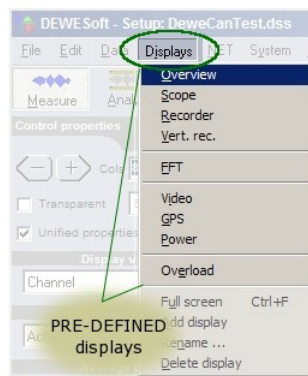
DEWESoft provide some default screens with **pre-defined displays** of different *instruments* for each input, for example *scope*, *recorder*, *FFT analyzer*,...

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:

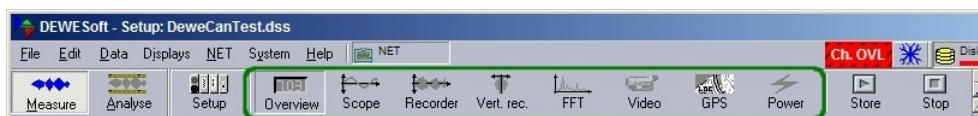
Name	Icon	Description
Overview		Overview screen include <i>one meter</i> for all <i>active channels</i> by default. An Overview display is intended to be <i>defined</i> and <i>supplemented</i> by the user with <i>adding</i> and <i>arranging meters</i> and other <i>instruments</i> .
Scope		Scope - oscilloscope screen has <i>predefined one Scope instrument</i> , which is usually used for <i>displaying fast, short-time events</i> . Like in a traditional scope you can define <i>trigger</i> conditions.
Recorder		Recorder screen has <i>predefined the recorder instrument</i> displays for <i>all</i> of the <i>active channels</i> by default. This screen is meant to display the time-history <i>plotting</i> capabilities and to get an idea of the <i>acquired signals</i> over a <i>long time - trend</i> .
Vertical recorder		Vertical recorder screen has <i>predefined the vertical recorder instrument</i> displays for <i>all</i> of the <i>active channels</i> 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 <i>predefined FFT</i> and one <i>scope instrument</i> displays for the <i>all</i> of <i>active channels</i> 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 <i>acquired data</i> . <i>Scope</i> helps you to interpret the FFT data.
Power analysis		Power display screen has <i>predefined Vector scope</i> of <i>selected signals</i> and one <i>scope instrument</i> displays for the <i>all</i> of <i>active channels</i> 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 <i>acquired data</i> immediately. The <i>time domain</i> based graph helps you to interpret the Power data.
Video		Video screen has <i>predefined Video display</i> and one <i>recorder instrument</i> display for the <i>all</i> of <i>active channels</i> by default (recorder graph can display up to 4 signals). The basic idea of the Video is to observe a video together with a <i>recorder</i> view for <i>time domains</i> display your <i>acquired data</i> .
GPS		GPS screen has <i>predefined GPS display, digital meters</i> for GPS channels and one <i>recorder instrument</i> display. The basic idea of the GPS screen is to observe a graphical representation position data together with a display acquired values in <i>digital meters</i> and a <i>recorder</i> view for <i>time domain</i> display of <i>acquired data</i> .

We can select **pre-defined online display** with:

1. using the **Displays menu**



2. selecting **icon** on **DEWESoft** toolbar



On **DEWESoft** toolbar shown icons can be *changed* easily to the most used displays.

for detailed information about **Shown displays** see → [Reference Guide](#) → **General setup - Shown displays**

3. using **shortcuts** (see above).

	shortcut		shortcut
Overview	<F3> OR <SHIFT>+<F1>	FFT analysis	<SHIFT>+<F5>
Scope	<SHIFT>+<F2>	Power analysis	<SHIFT>+<F6>
Recorder	<SHIFT>+<F3>	Video	<SHIFT>+<F10>
Vertical recorder	<SHIFT>+<F4>	GPS	<SHIFT>+<F11>

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*, but efficient *overview* of your **signals** and **measurement**. Please look at the following section **Creating own display - Display settings** for more information on this topic.

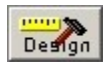
2.1.2 Display settings

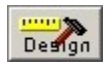
All **DEWESoft pre-defined** and **added** displays can be **adapted** to meet the user's requirements and to have a *simple*, but efficient *overview* of your **signals** and **measurement**.

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, **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** and **channel grouping** on lower left 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 maybe you don't want these channels to be in the new instrument or 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** near the bottom 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.

2.1.2.1 Channels 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
- **Resizing the CHANNEL selector** - *resize* the CHANNEL selector
- **Channel grouping** - *create and edit* user defined channel *groups*

Examples of the CHANNEL selector:



CHANNEL list view



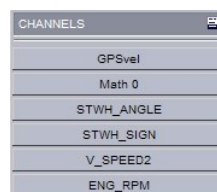
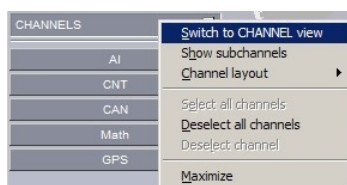
GROUP view



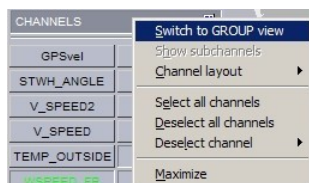
GROUP view with subchannels list

CHANNEL (List) view / Group view

When you *right-click* anywhere on the CHANNELS selector or its icon, you can choose between the 'old styled' CHANNEL view (list) and the GROUP view (and vice versa):



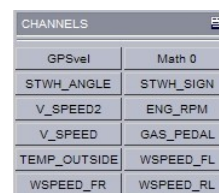
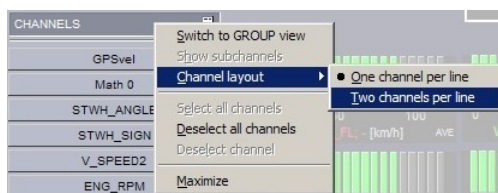
Channel view



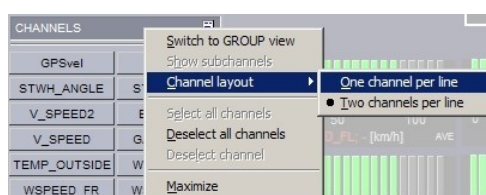
Group view

With Channel layout menu option can be on **CHANNEL** view (list) selected:

1. 'Two channel per line' Channel view:



2. or 'One channel per line' Channel view:

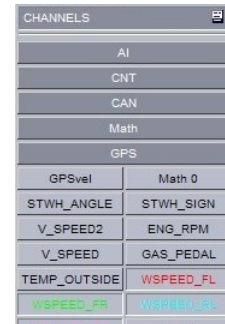
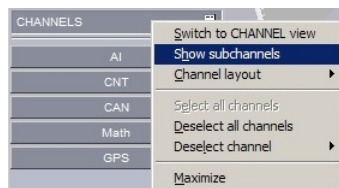


The **GROUP** view is a *hierarchical tree*: it starts with the *base input* type (for example **AI**, **Math**, **CAN**,...). When you click on one of these, the **CHANNELS** selector *opens* this area and displays the *next level* - in most cases already the channels.

NOTE: *If there is only one hardware group available (e.g. AI) the channels are always displayed in List view!*

The **groups** are pre-defined, but you can also **define groups** by yourself.

In the GROUP view you can select **Show subchannels**. In this case *all* available group and channels within the hierarchy will be displayed.



With click on group also *all* available subgroup and channels below this *current level* within the hierarchy will be displayed.

On upper example on right group 'CAN' is clicked and display shows also all his subgroup.



On lower example on right subgroup 'STEERING_WHEEL' is clicked and display shows also all his subgroup and channels.

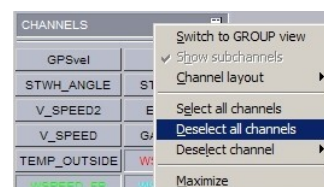
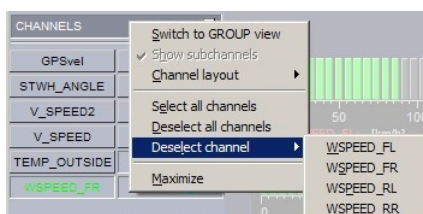


When subgroup(s) (see pictures above right) are displayed, with click on  icon basic Group view can be restored.



Select /deselect channels

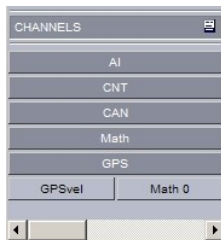
Channels can be **assigned** or **reassigned** to **instruments** simply by *clicking* on them in the **CHANNELS** selector. But there is also another way. *Right-click* on the **CHANNELS** selector (or the  icon) and choose **Select all channels** to assign *all* channels to instrument choose or **Deselect channel** (choose also the *desired* channel) or **Deselect all channels**:



Resizing the CHANNEL selector

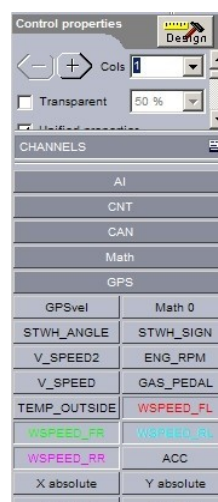
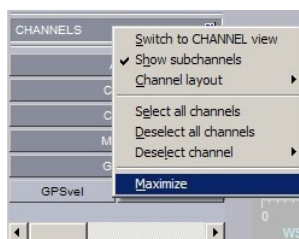
Sometimes when you have hundreds of channels, the default space for the **CHANNELS** selector may be too small for you. In this case you can see all channels in two ways:

1. moving between channels with standard Windows *slider*:

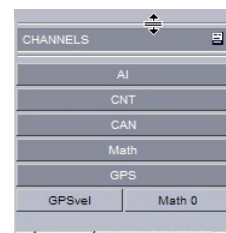


2. resizing **CHANNELS** selector space:

right-click on the **CHANNELS** selector (or the icon) and choose **Maximize** (DEWESoft decrease space for the **Control properties** → see right picture)



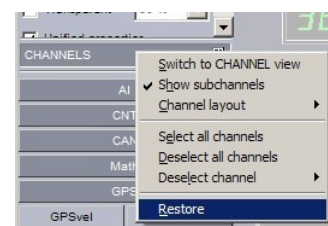
or *move* the mouse cursor over the top line, *left-click* (and keep mouse button pressed) and *move* upper boundary of the **CHANNELS** selector upwards to the desired position within the **Control properties**:



Move CHANNEL selector border manually

Maximize CHANNEL selector

As the **CHANNELS** selector overlays the whole instrument control (see above) when you use the **Maximize** function, you can *right-click* on the **CHANNELS** selector (or the icon) and select **Restore** to move *back* to the *original* **CHANNELS** selector position (see right) or *move* **CHANNEL** selector border manually (see picture above right).

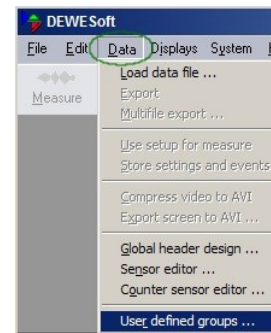
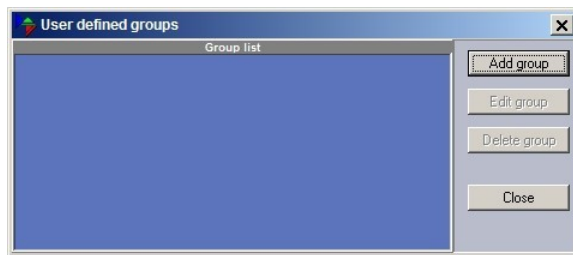


Channel grouping

The **channel groups** are pre-defined, but you can **define** channel *groups* also by *yourself*. These groups can *contain* any available channels, *independent* from their signal source. Therefore it is *possible* to **mix** for example **AI** channels with **PAD** and **CAN** channels.

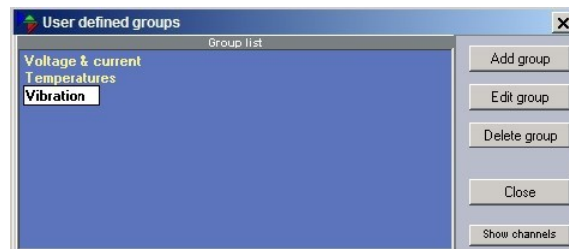
To define your own groups you have to switch back to the **Measure** → **Setup**.

Now you can select **User defined groups** from the **Data** menu. The following window appears:



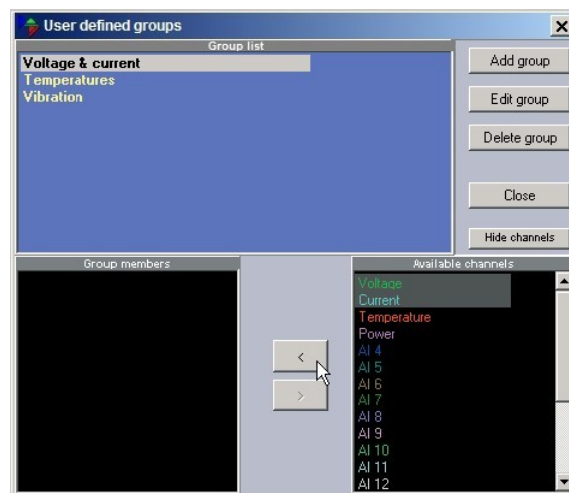
Now you can **add** groups according to your requirements - simply click the **Add group** button.

Example:



If you have done a typo or want to **rename** a group just **select** the group again and press the **Edit group** button to **change** it.

The next step is to **assign** channels to the **new** groups (or existing **old** groups). Therefore **select** the group, in our example **Voltage & current** and press the **Show channels** button:



Now choose from the **Available channels** list on the right side - be aware that also **unused** channels are mentioned in the list - and press the **<** arrow button to **assign** your selection to the **Group members** list.

Repeat that for all groups. If you have assigned a **wrong** channel to your **Group members** list, select it and press the **>** arrow button to **de assign** it.

For the case that you **don't need** a group any more you can select it and press the **Delete group** button.

When you have finished your work on your **User defined groups**, press the **Close** button and change back to your instrument.

The result may look like that:



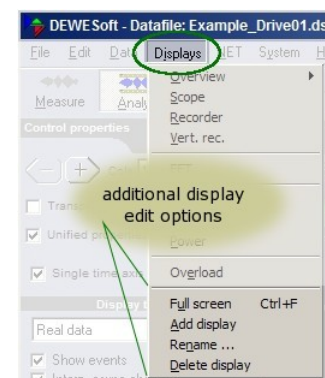
The user defined groups appear below the pre-defined physical groups

At any time you can go back to the **User defined groups** menu and **change** the assigned channels. All these settings will be **stored** together with your measurement setup and will be **lost** when you have **not stored** your **setup** before closing **DEWESoft**.

2.1.2.2 Displays menu functions

DEWESoft enable some general **display** function:

- **Full screen** active online display screen *enlarge* on full **DEWESoft** window
To escape from this mode press **Esc** button on your keyboard.
- **Add display** add another display
- **Rename ...** Rename display
- **Delete display** Delete display



Additional display screens

Within all online **display** screens, you can:

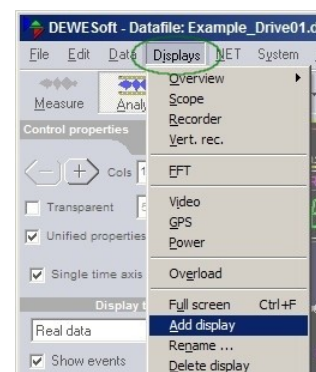
- **add** several further **displays** of the same kind *pre-defined display* using the **Add Display** function from **Displays** menu.

The advantage of this feature is to have *different, independent viewings*.

This is very helpful when you have to *compare* signals or display *dozen* of channels.

New display appear with none predefined instrument and with name of format: **'NameofPreDefDisplay n'**

and you are allowed to **design** your own **display** completely.

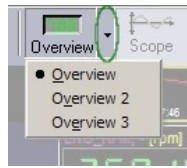


When on whichever pre-defined display is *more than one display*, **DEWESoft** tool icon change to (example for *Overview* display see on right).



With this control you have access to list of all created **displays** for pre-defined display.

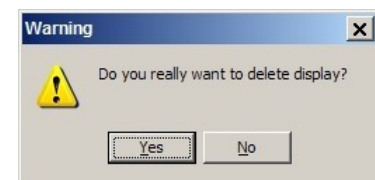
When you have added additional displays, the you can *select* them directly with the *down arrow* next to the appropriate *display icon*:



or from the **Display** menu:



- **rename** according to your requirements; *selected* display can you rename in **Name of display window**, which is called from the **Display** menu → **Rename ...**
- **delete display**; *selected* display can you delete with call the **Display** menu → **Delete display ...** function; after selecting this option **Warning** window



2.1.2.3 Screen edit functions

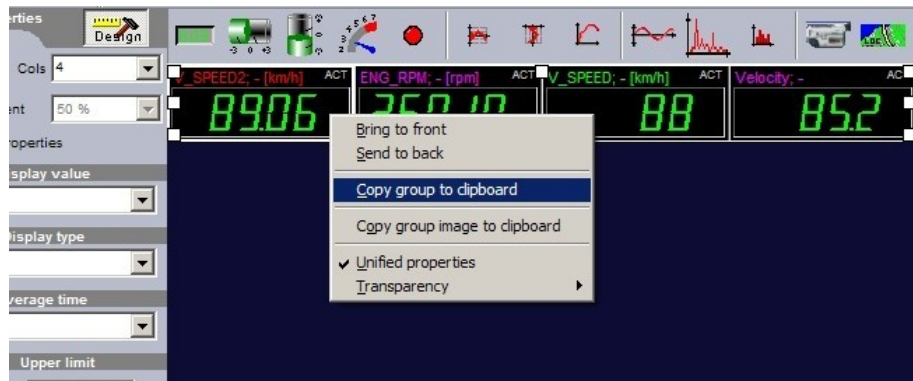
DEWESoft offers an additional **screen edit** function to create *intelligible display* all channels data on the screen.

- **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 same **settings** and can be edited as all other.

To copy the element, *right-click* on the element and select **Copy group to Clipboard** from the menu list:



or select **Copy group to Clipboard** from the **Edit** menu.

Then *move* the cursor to the desired *position* (must be on screen *outside* group), *right-click* again and select **Paste** button which appear on screen. That's all - the new element is *available*:



Delete instruments

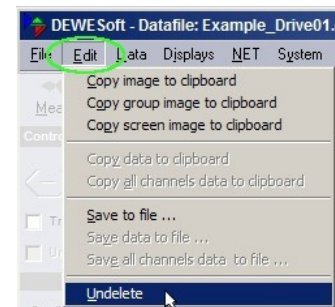
You may want to **delete** instruments from the display screen. To delete any instrument, just *click* it once to *select* it, and then click:

-  - 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!

2.1.2.4 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**

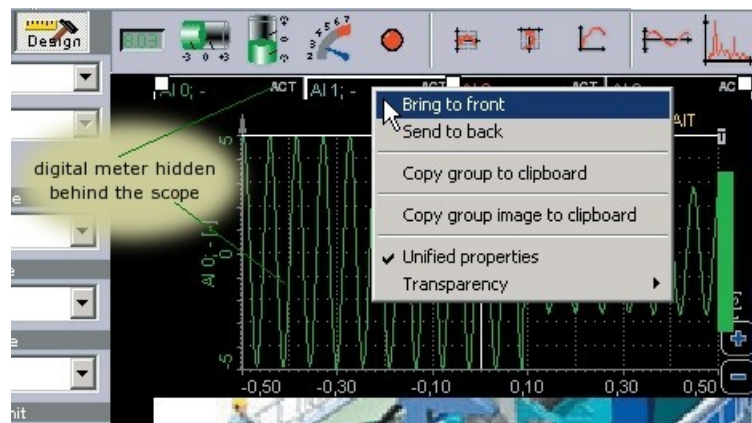
Position

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.

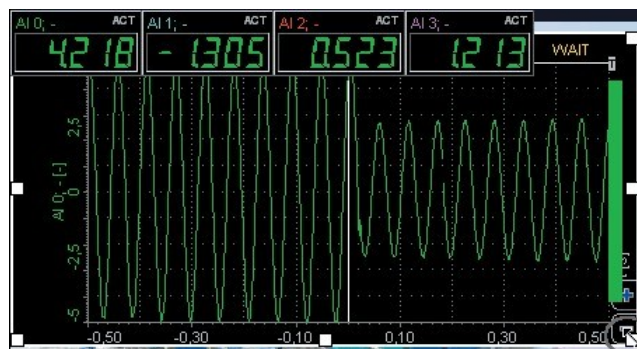
If the element comes in conflict with other elements, you can **bring** the element in **front** of the other one or **behind** it; therefore *right-click* on the element and then select **Bring to front** / **Send to back** from the appearing menu list.



The **Design** mode offers a very need fully 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:

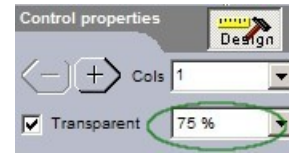
- use the **Transparent** checkbox in the **Control properties** bar and *select* the transparency *value* from the drop down list

for detailed information about **Control properties** → see → **Instrument setup**

- **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%:



2.1.3 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.



2.1.3.1 Group properties

DEWESoft offers a **Design Control properties** to create *complex view* all acquired and calculated data on the online display screen.

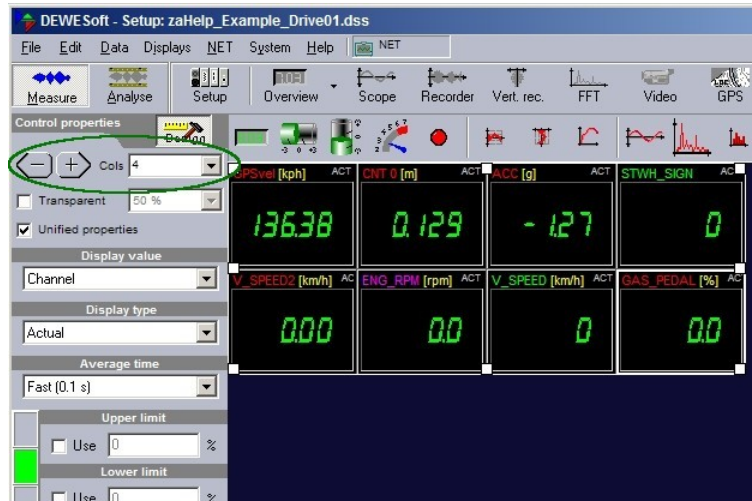
With **Control properties** **DEWESoft** allows to *create* and *arrange* elements - instruments in box - Group of instruments with:

- 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

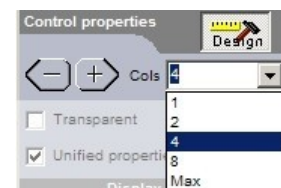
Placing **instrument** by instrument on the display can *take some time*, especially when you have to display *dozen* of channels. Therefore **DEWESoft** allows **creating instrument groups**, which are *boxes* with the same sort of instruments.

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**:



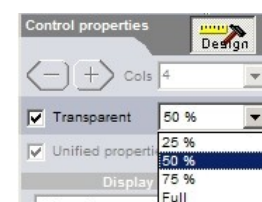
button to **add** same **instrument** or



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 all digital meters within an array are set to 50% *transparency*, show the **peak-peak** value with very slow (5s) *average time*.

Use the **Unified properties** checkbox to allow **different** settings for each instrument *within* the selected *group*.

In our upper example now you can set e.g. **peak-peak** value for the *first* instrument, **RMS** for the *second*, **Max** for the *third*...

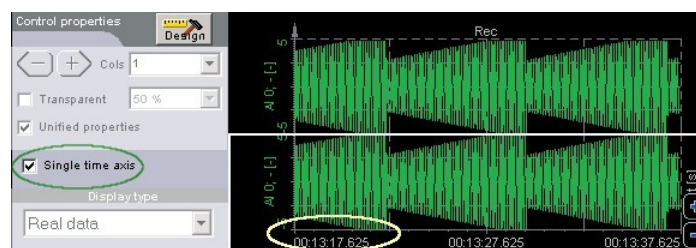
2.1.3.2 Common graph settings

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.

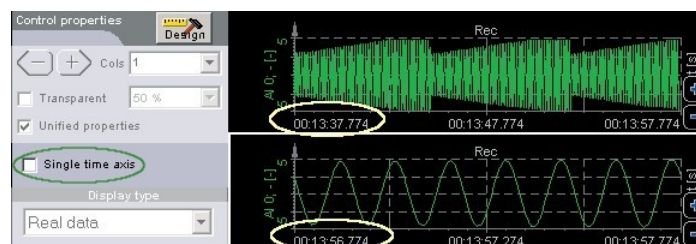
- **single** time axis → **Single time axis** box **checked**

The *advantage* of the **single** time axis is to have more space to *display* the *data*.



- **multiple** time axis → **Single time axis** box **unchecked**

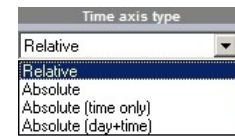
The *advantage* of the **multiple** time axes is to make it easier to *read* time related *information*.



Appearance by *vertical instruments* is same, except Time axis is vertical.

Time axis type

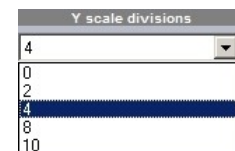
You can show either **absolute** or *relative* time in the **Time axis** (in case of *Vertical recorder* in vertical direction) for *each* graph. Click on any graph to select it and then use the selector to make your choice.



- **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**

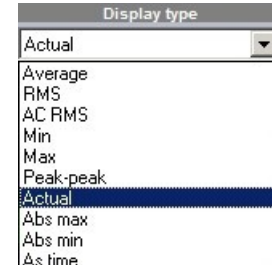
Grid resolution

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

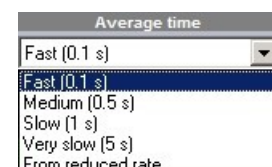
Your acquired signal may *not* be only a static signal; if you have ever tried to display a *high-dynamic* signal with *digital* values, you know that you have to make some kind of **statistic** to get representative values. Therefore all *instruments* offer *different* display types, which can be selected 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 on right).

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.

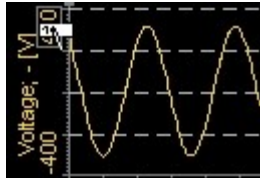
Appearance and functionality by *vertical instruments* is same, except Time axis is vertical.

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:

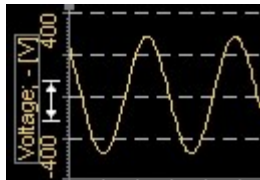
- **Enter values**



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 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 <i>selected</i> channel (see also above).
<CTRL> + Left mouse button	Selects <i>all available</i> channels and scales all channels to the <i>same</i> maximum and minimum (highest and lowest value within all channels).
<CTRL> + <ALT> + Left mouse button	Selects <i>all available</i> channels and scales <i>each</i> channel to its <i>own</i> maximum and minimum.
<SHIFT> + Left mouse button	Scales the <i>selected</i> channel symmetrical around <i>zero</i> ($\pm$ values are the same).
<CTRL> + <SHIFT> + Left mouse button	Scales <i>all available</i> channels symmetrical around <i>zero</i> (all $\pm$ values are the same)
<CTRL> + <SHIFT>+ <ALT> + Left mouse button	Selects <i>all available</i> channels and scales <i>each</i> channel symmetrical around <i>zero</i> (each channel to its own $\pm$ value).
Right mouse button	<i>Undo</i> auto scale for the <i>selected</i> channel (see also above).
<CTRL> + Right mouse button	Selects <i>all available</i> channels for <i>undo</i> auto scale.

Simple measurement cursor functions

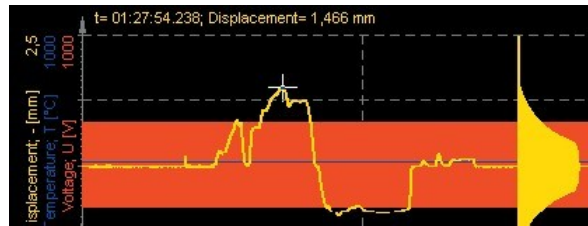
WARNING: This way displaying X- and Y-axis information can't be done in Design mode → available only in Run mode.

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.

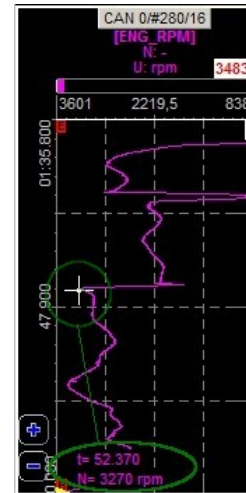
The next display gives an impression of the *cursor functionality*:



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.*

2.1.3.3 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 part of the screen:

- **Control properties** for detailed information about **digital meter** **Control properties**: grouping, number of column, Add / Remove, 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 → **Creating own display**



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**

Example caption and frame ON:



Example caption and frame OFF:



As a standard, the caption is *displayed*.

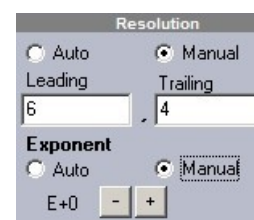
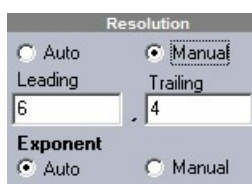
Resolution

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**.



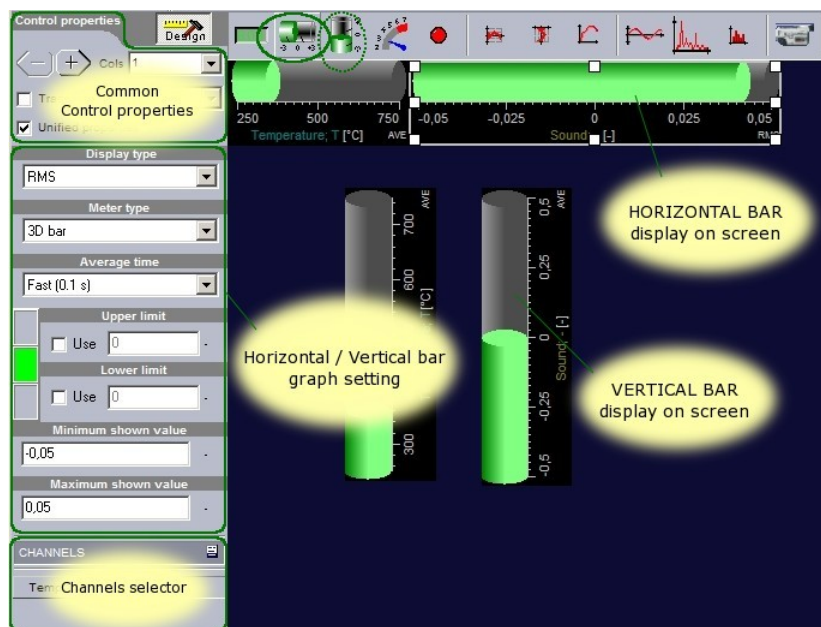
2.1.3.4 Horizontal / Vertical bar graph



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 part of the screen:

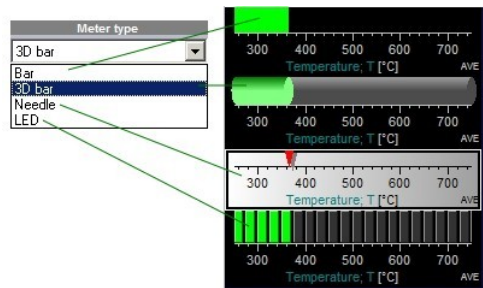
- **Control properties** for detailed information about **Horizontal / Vertical bar graph Control properties**: grouping, number of column, Add / Remove, transparency,...
see → **Control properties**
- **Horizontal / Vertical bar settings** typical settings (same for **Horizontal** and **Vertical bar**) 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 → **Creating own display**



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.

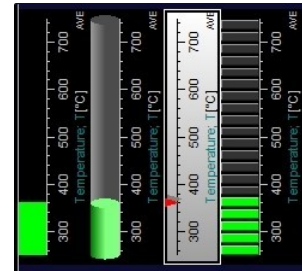
They all contain the **channel name**, **display type** and **unit** beside the *measurement values*.



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**

2.1.3.5 Analog meter



The **Analog meter** is designed to give a *graphical indication* of the **current value** of a signal within the defined display range.

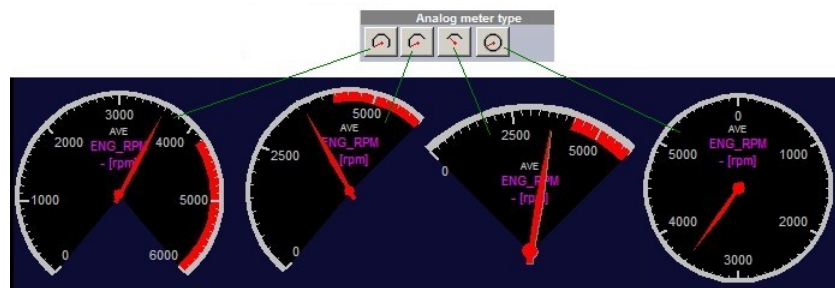
When you select an *Analog meter* in the **design** or **run** mode, following settings will appear on left part of the screen:

- **Control properties** for detailed information about **Analog meter Control properties**: *grouping, number of column, Add / Remove, transparency,...*
see → **Control properties**
- **Analog meter settings** typical settings for *Analog meter* are:
 - **Display type**
 - **Upper / Lower limit**
 - **Analog meter type**
 - **Minimum / Maximum shown value**
 - **Average time**
- **Channels selector** for detailed information about *assigning / reassigning channels* to **Analog meter** → see → **Creating own display**



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**

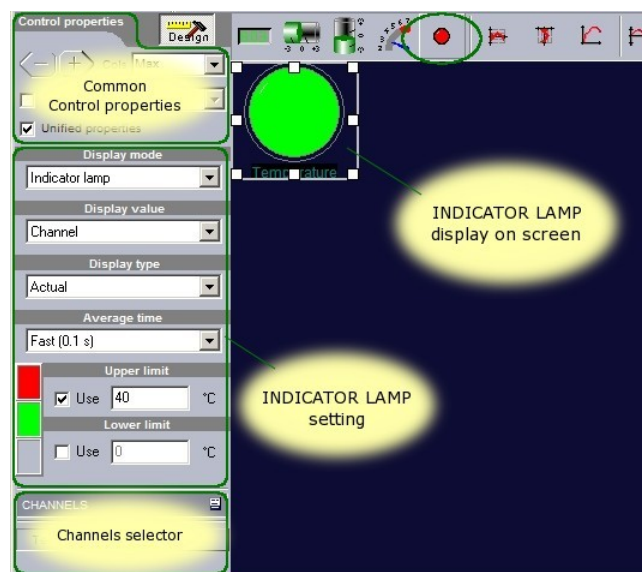
2.1.3.6 Indicator lamp



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 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** typical setting for **Discrete Display** *Display mode* are:
 - **Display mode**
 - **Options**
 - **Discrete values**
- **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 → **Creating own display**



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**.

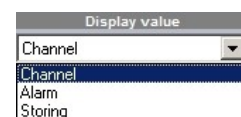


Indicator lamp Display mode setting

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.

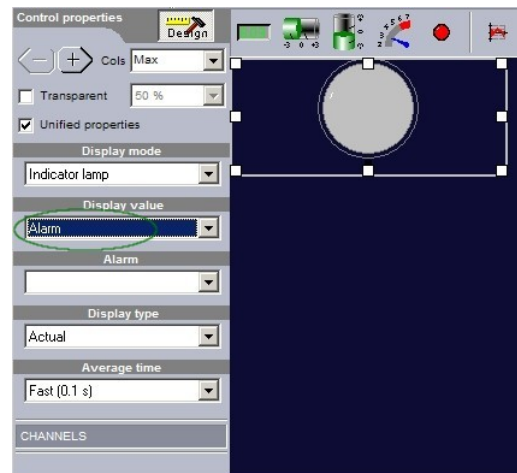
For **Channel** option you be able to set (see picture above):

- **Display type**
- **Average time**
- **Upper / Lower limit**

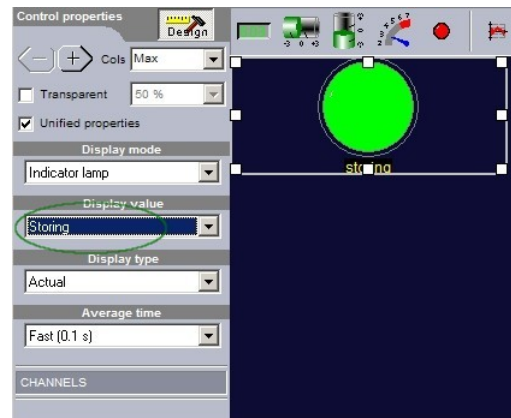
2. **Alarm** will show the *state* of alarm signals

For **Alarm** option you be able to set:

- **Alarm** from list; alarms are defined in channel setup in 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.

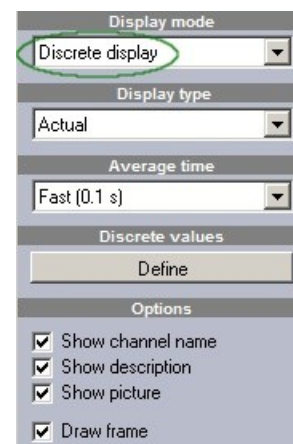


Discrete Display Display mode setting

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**

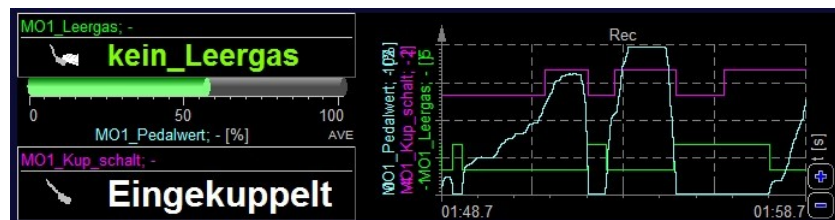


Let's take few CAN channels which have defined *discrete channels*. There are few channels which describes the bit value if the car is running in idle (**MO1_Leergas**) and if the clutch is pressed (**MO1_Kup_schalt**). The channels have *defined discrete values* which are loaded from the DBC library.

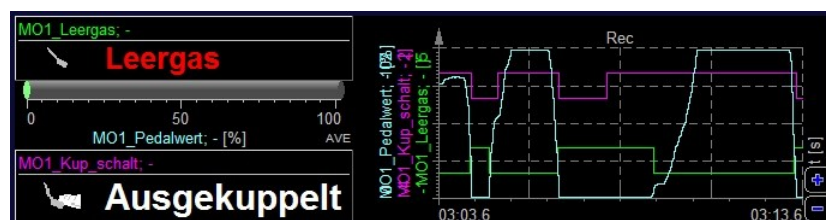
Used	Store	mMotor_1	000000 0 0000 1 000000 0 1 0 000 0000 1 0 00 1 0
Used	Store	MO1_Leergas	kein_Leergas
Unused		MO1_Sta_Pedal	Fahrpedal_iO
Unused		MO1_Kickdown	kein_Kickdown
Used	Store	MO1_Kup_schalt	Schalter_sagt ausgekuppelt
Unused		MO1_TiOut_Br	Empfang_iO
Unused		MO1_Sta_Bremse	erfuellbar
Unused		MO1_Sta_Getr	dynamisch_nicht_erfuellbar
Unused		MO1_Sta_MotMo	nein_Werte_iO
Unused		MO1_Mo_m_ex	37.83 [%MDI] "0 99.06
Unused		MO1_Drehzahl	1488 [1/min] "0 16256
Unused		MO1_Mo_o_ex	37.83 [%MDI] "0 16256

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:



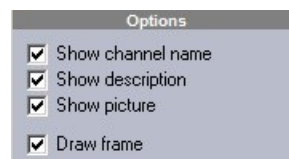
In this window you can **define**:

- **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.



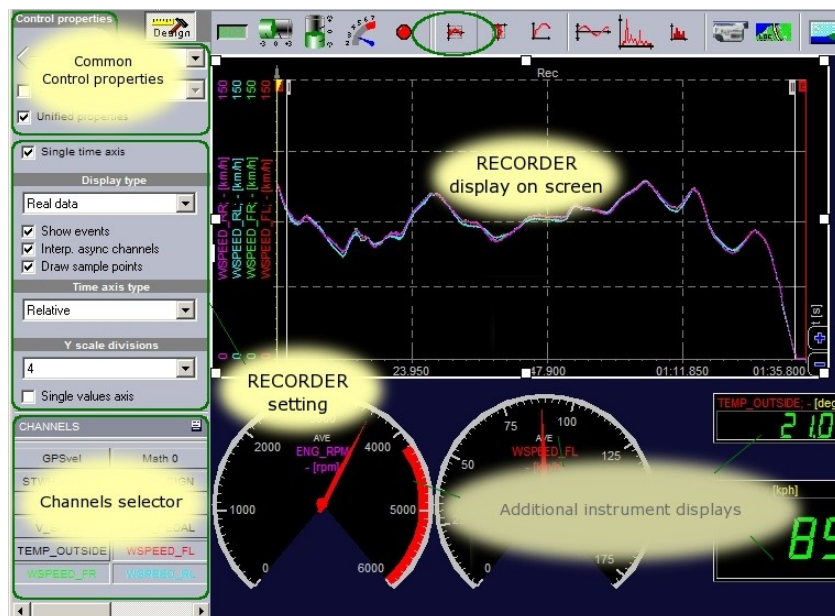
2.1.3.7 Recorder



The **Recorder** instrument is meant designed to **emulate** the **time-history** plotting capabilities of 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 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 **assigning / reassigning** channels to/from **Recorder** → see → **Creating own display**



Appearance on screen

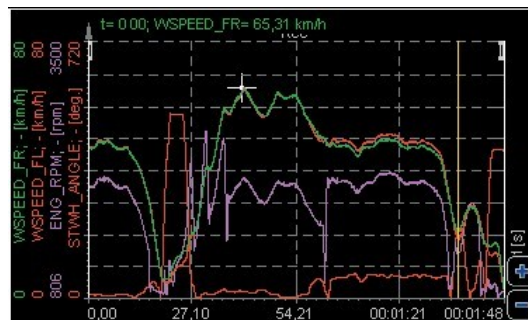
The **Recorder** element offers all *important* information:

channel name(s)

unit(s)

time information

zoom functions...

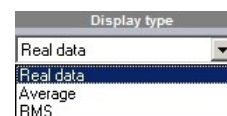


for detailed information about *instruments* **positioning**, **size** and **transparency** see → [Screen edit functions](#)

Setting

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**.



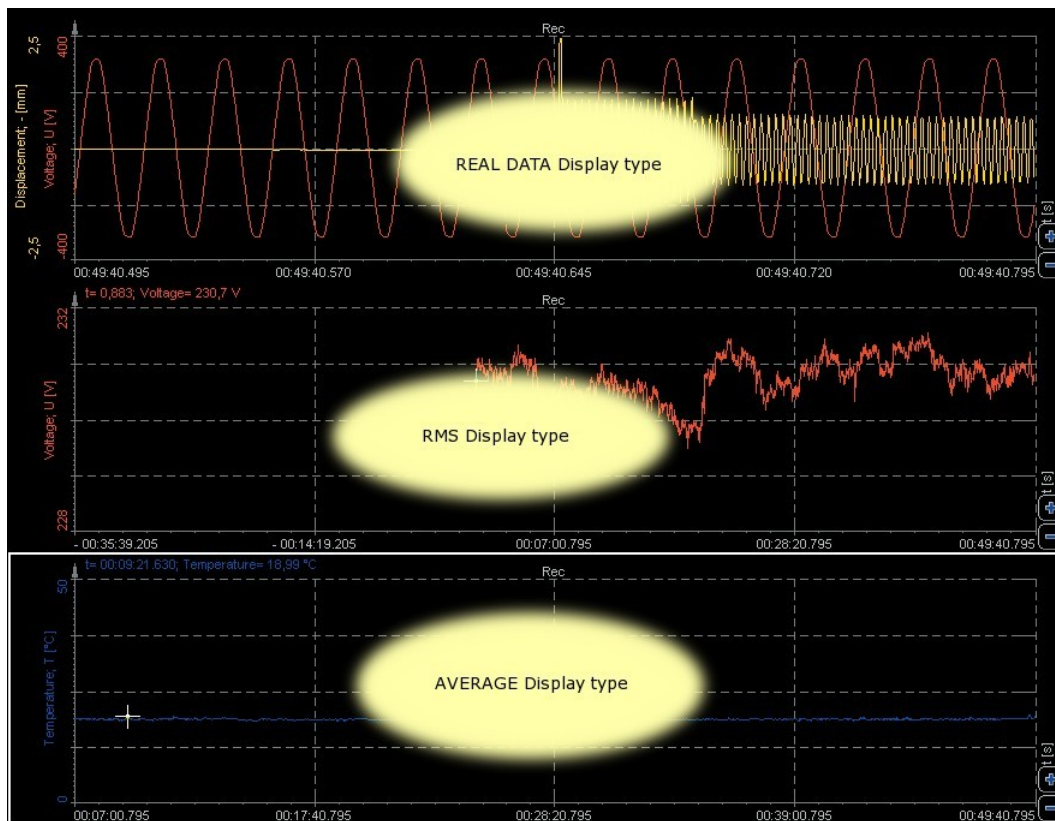
This is useful when monitoring **AC** signals, which are going to look like a *solid band* when a *long* duration is shown - not very useful to look at (unless you're just looking for overall amplitude envelopes, or obvious drop-outs). Changing the display type for this graph to **RMS** will show a more *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.

☒ Show events

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*.

☒ Interp. async channels

Use the **Draw sample points** checkbox to display *individual sample points* in **analyse** mode.

☒ Draw sample points

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.

2.1.3.8 Vertical recorder



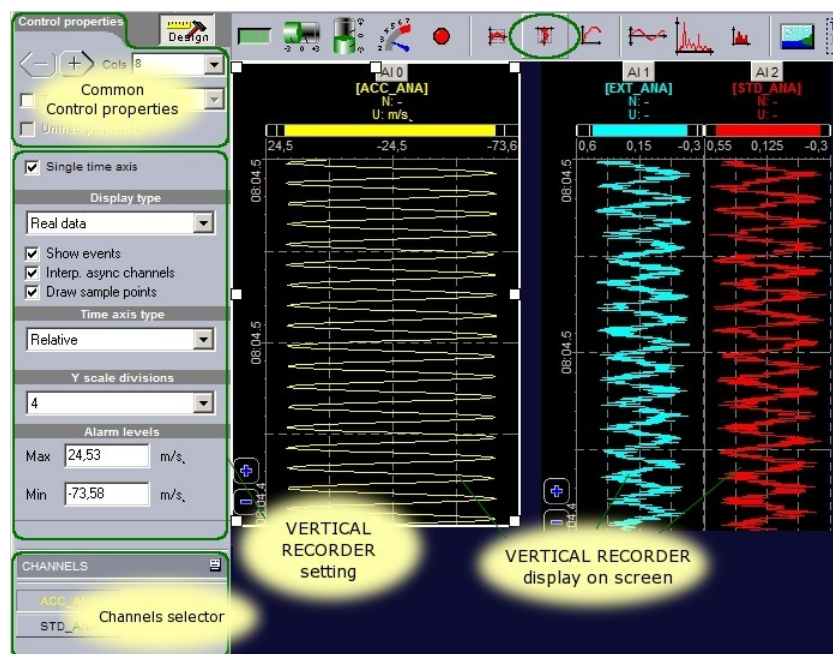
The **Vertical recorder** instrument is meant designed to **emulate** the **time-history** plotting capabilities of the traditional *strip chart* or *recorder*, but with *vertical* Time axis orientation and *enhanced display*. On each *vertical recorder* only *one* channel can be displayed.

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-axis scale/auto scale**
 - **X-/Y-axis information**
- **Channels selector** for detailed information about **assigning / reassigning** channels to/from **Vertical recorder** → see → **Creating own display**

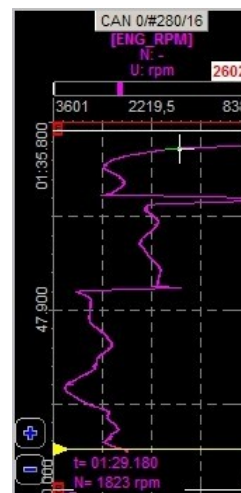


Appearance on screen

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.

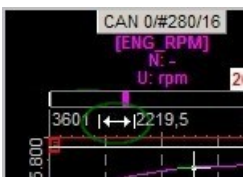
If you want to **change** the **X-axis** for *any* input, the recorder offers *two* possibilities:

- **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 ($\pm$ values are the same).

Right mouse button

Undo auto scale for the *selected* channel (see also above).

Setting

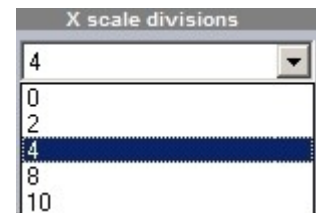
Display type

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**

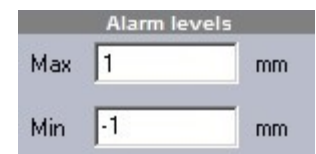
X scale divisions

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** above the grid.



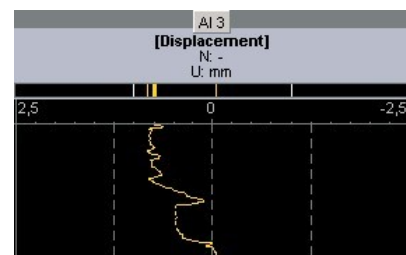
As a *standard*, the alarm values are set to the *minimum* and *maximum range* for the appropriate channel. If you want to use own limits, just **enter** the alarm level according to your *requirements*.

For example, if we want to supervise the signal within ± 1 mm, we enter the following settings: → see above 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.

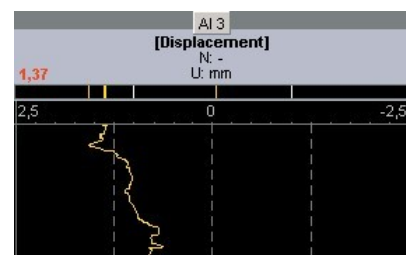
Example 1

The current signal is *within* the defined *limits* of ± 1 mm, indicated by the two **white** markers.



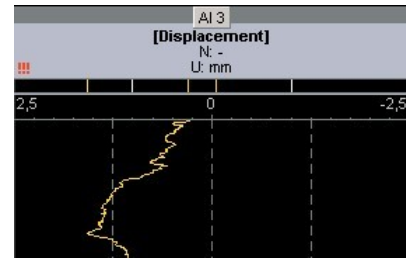
Example 2

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.



2.1.3.9 X-Y recorder



The **X-Y recorder** is designed to **display channels versus channels**; there is *no direct time relation*.

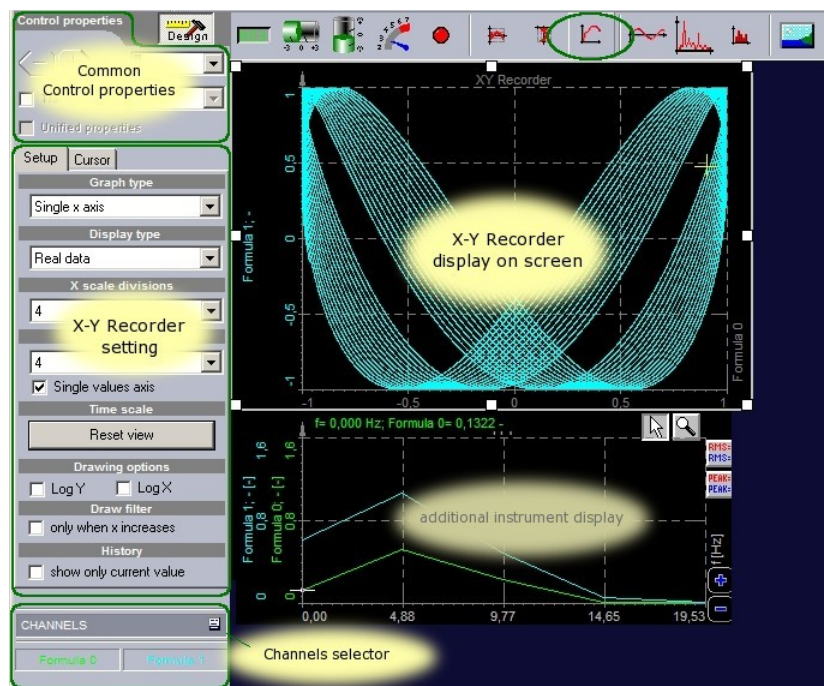
When you select *X-Y recorder instrument* in the **design** or **run** mode, following settings will appear on left part of the screen:

- **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 → **Creating own display**



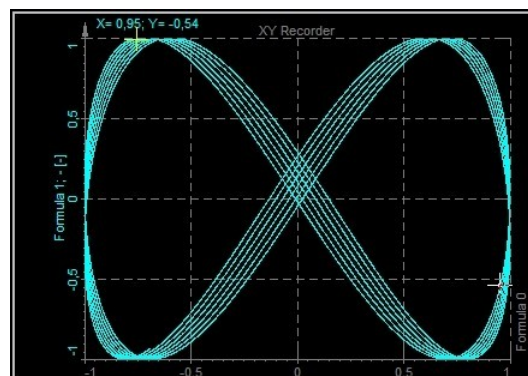
Appearance on screen

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*.

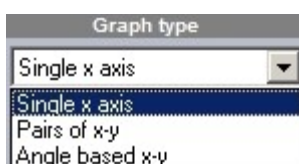
X-Y recorder appearance with *only current value* see → **History** (below) .



for detailed information about *instruments positioning, size and transparency* see → **Screen edit functions**

XY Recorder Setup

Graph type



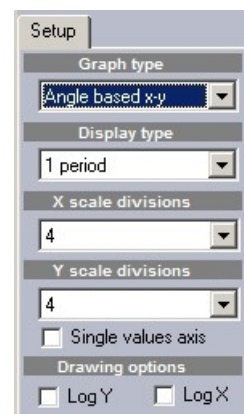
The **Graph type** knows *three* modes:

- **Single x-axis** Allows only one channel **x-axis**. The **first assigned** channel is *used* for the **x-axis**, up to **4 following** channels the **y-axis**.
- **Pairs of x-y** 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**

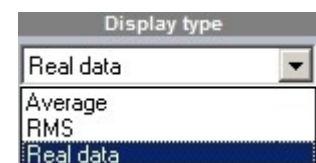


Display type

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**.



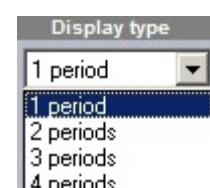
Select any graph by *clicking* on it, and then select this value from the **Display type** drop down list.

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.*

Angle based x-y graph

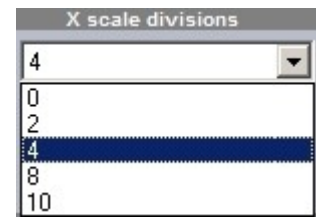
The **Display type** for this Graph type is different as for other *Graph types*.

You can display signals over a **1 period** or **2, 3** or **4 periods** which can be selected from 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**.

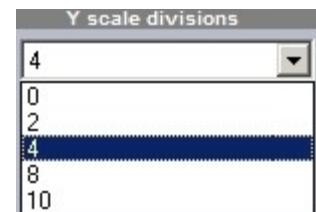


X scale divisions

4

0
2
4
8
10

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**.



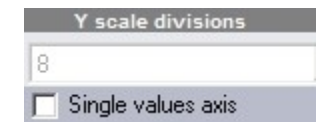
Y scale divisions

4

0
2
4
8
10

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**.



Y scale divisions

8

☐ Single values axis

Time scale

Recorder or **scope** displays show their content **only** for a **limited time**. But what about the **X-Y graph**? How **long should** it display data? **Decide** it by yourself. Simply **click** on the **Reset view** button in **Time scale** section to **clear** the graph.



Time scale

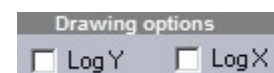
Reset view

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**



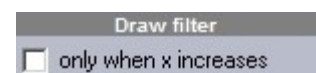
Drawing options

☐ Log Y ☐ Log X

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.

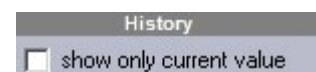


Draw filter

☐ only when x increases

History

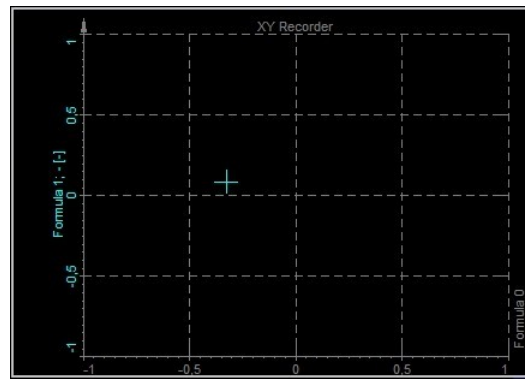
When the signal is changing very **strong**, it may be helpful to click the **Show only current value** checkbox in **History** section; this **removes** all displayed values from the **X-Y graph** and **shows only** the **current value**. **Unselect** this feature to receive the **whole** signal history again.



History

☐ show only current value

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.



2.1.3.10 Scope

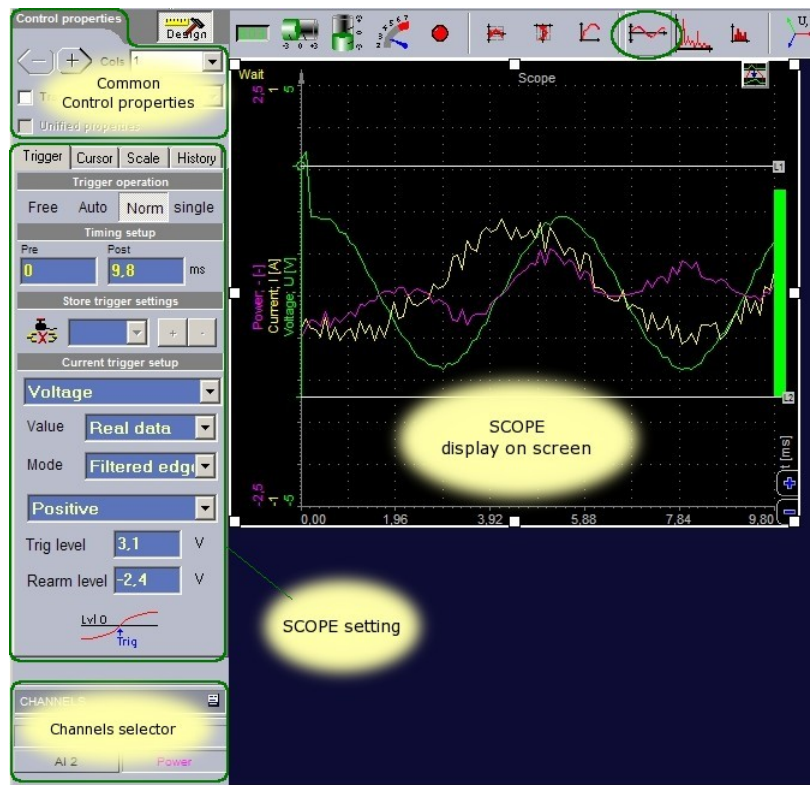


The **Scope** instrument is used for **displaying fast, short-time events**. Like in a traditional *scope* you can **define trigger conditions**. Up to 16 inputs can be displayed at once in each graph.

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 → **Creating own display**



Appearance on screen

The **Scope** element in the overview offers all important information:

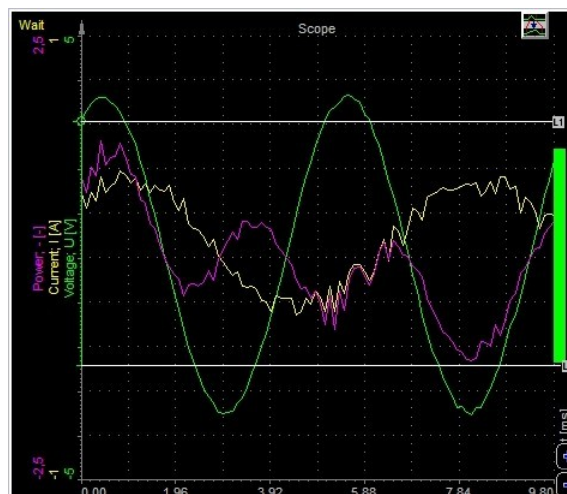
channel name(s)

unit(s)

time information

zoom functions...

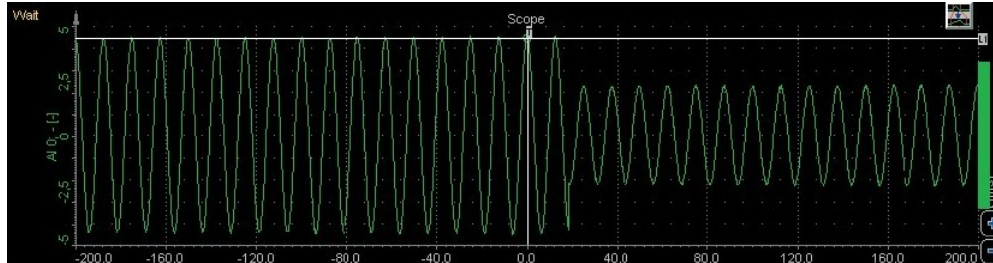
When the *scope* is not triggering, the bar on the right side shows the *current levels* of the signal so we can *optimize* the trigger level according the normal values (we can also use *Auto trigger* mode). When the trigger is lost for some *seconds*, data will be shown *none* triggered.



for detailed information about *instruments positioning, size and transparency* see → [Screen edit functions](#)

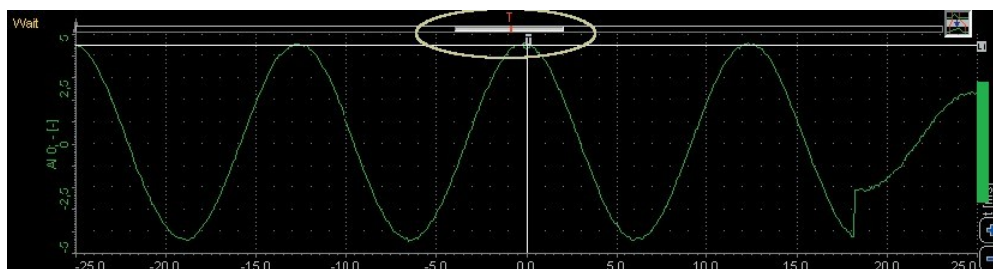
Run mode **Zoom** (additional appearance setting)

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.

2.1.3.10.1 Setting - display tools

The **Scope** instrument typical **settings** include three main groups:

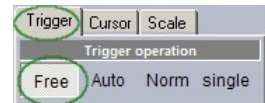
- **Trigger** → **Free run** - settings of triggered acquisition
Auto
Norm
single
- **Cursor** - cursor measurement to show the **cursor readouts** for each channel within the selected scope; with **Reference curves** possibility
- **Scale** - to change displayed *offset* and *scaling* of signals
- and 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.

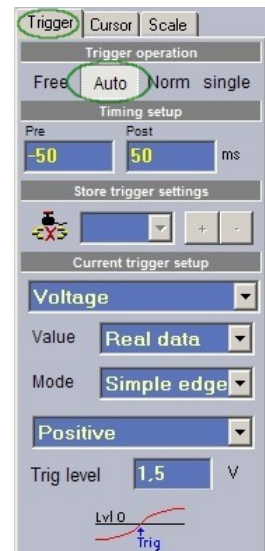


- **Auto** The **auto trigger** **displays** values if the *trigger condition* is **true**; when there is *no* trigger within some time, it displays the *current* value.

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:

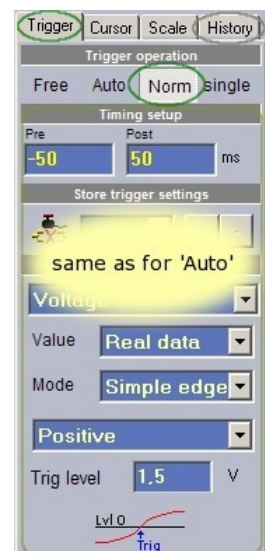
Simple edge	Pulse-Width
Filtered edge	Window and pulse-width
Window	Slope
- **Store trigger settings**



- **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**.

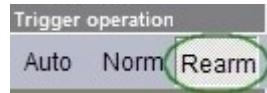


- **Single** This function can be used to **acquire single events**.

After selecting **single** button:



and event appear this button changes to **Rearm** (see also right):



Press it to **get another** single shot event.

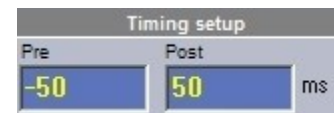


For this type of **Trigger** operation can be set the **same** setting as for **Auto** trigger → see above.

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](#)

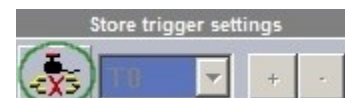
Hint The trigger **level** can also be changed by *moving* the **white vertical line** in the scope graph. Simply *click* on the line, *keep* mouse button pressed and *move* the line to the desired position.

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.



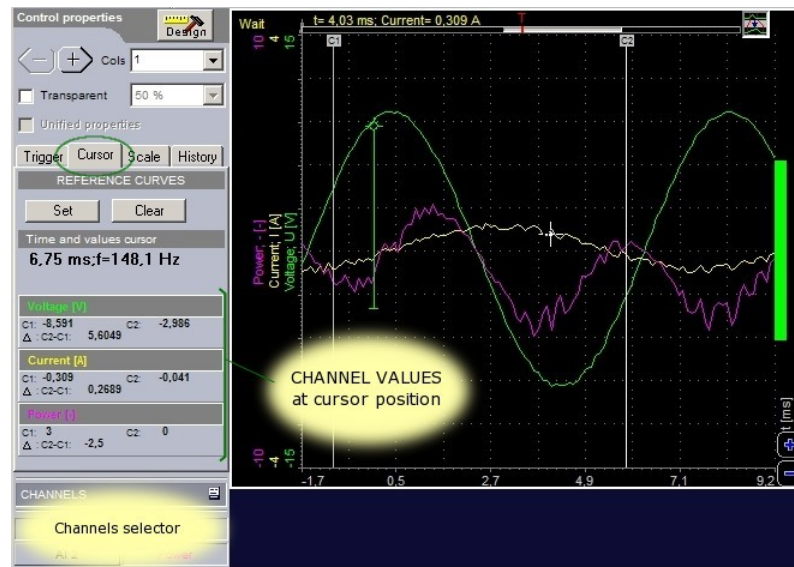
The  buttons can be used to **delete** selected *additional* conditions.

WARNING: As long as the **Link storage trigger** button is not pressed, the data is only displayed - not stored!

Cursor

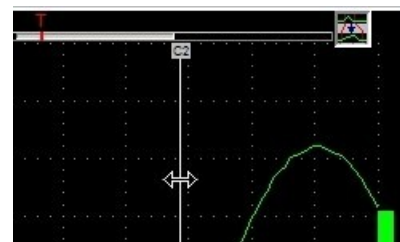
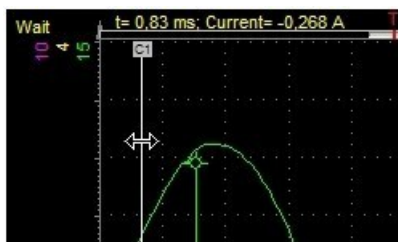
Cursor measurements

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* 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**

Voltage [V]		
C1: -8,591	C2: -2,986	
Δ : C2-C1: 5,6049		
Current [A]		
C1: -0,309	C2: -0,041	
Δ : C2-C1: 0,2689		
Power [J]		
C1: 3	C2: 0	
Δ : C2-C1: -2,5		

- Time and values cursor between **C1** and **C2**

Time and values cursor
6,75 ms; f=148,1 Hz

WARNING: The freeze function is not working in the scope!

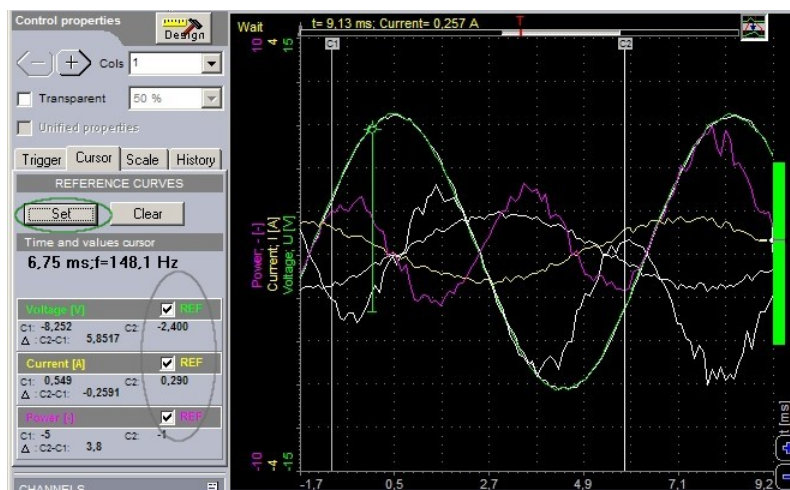
Reference Curves

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:

Voltage [V]		
C1: -8,252	C2: -2,400	☒ REF
Δ : C2-C1: 5,8517		

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**.

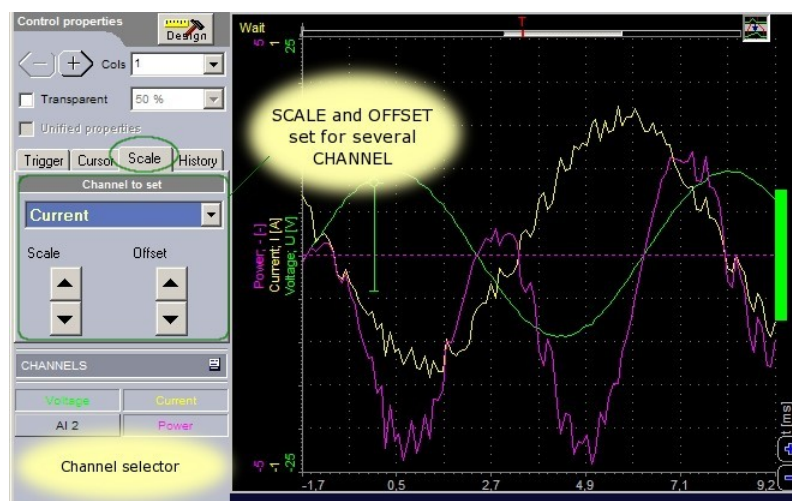
Scale

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:

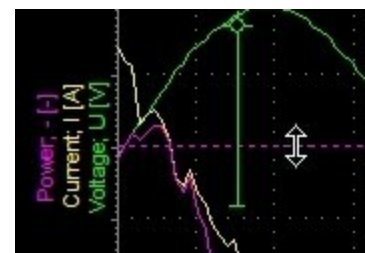


NOTE: *These settings have no influence on the acquisition resolution or accuracy. It's just a display feature!*

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

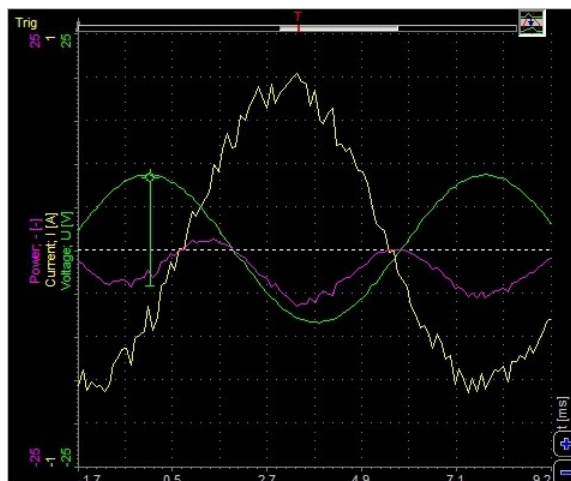


Now click (and keep mouse button pressed) and move the dashed line to the **desired offset** position.

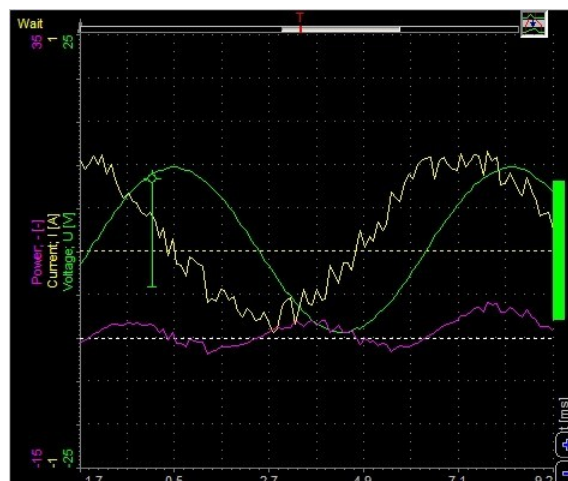


Examples for *scaling* and *offset* functions:

Scope Scale view after scale down *Power* channel:



and after set negative offset for *Power* channel:

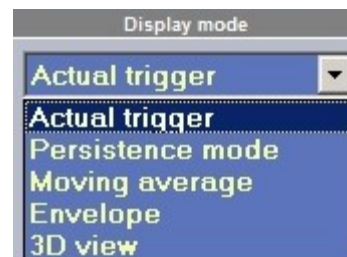


History

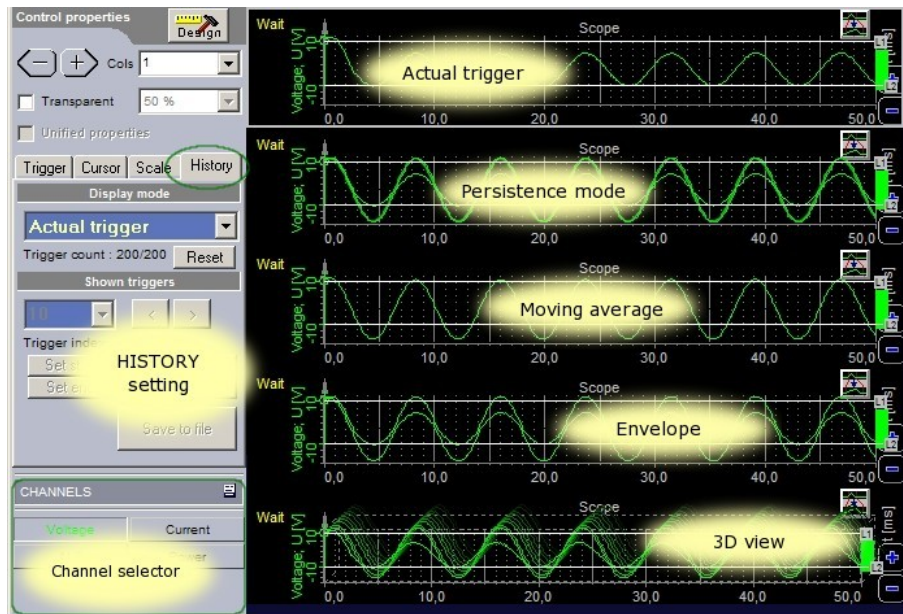
When the **Norm** or **Single** trigger is selected, another tab appears: the **History**. This feature allows you to **display** the following **information - history type**:

- 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*.

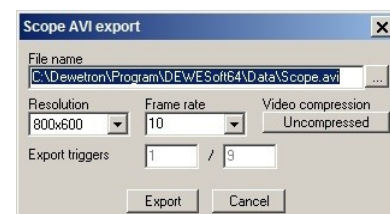


In *stopped* condition it's also possible to **export** the **acquired data** by pressing the **Export to AVI** or **Save to file** button.

When **Export to AVI** is selected **Scope AVI export** window appear to set: **File name**, **Resolution**, **Frame rate**, **Video compression**

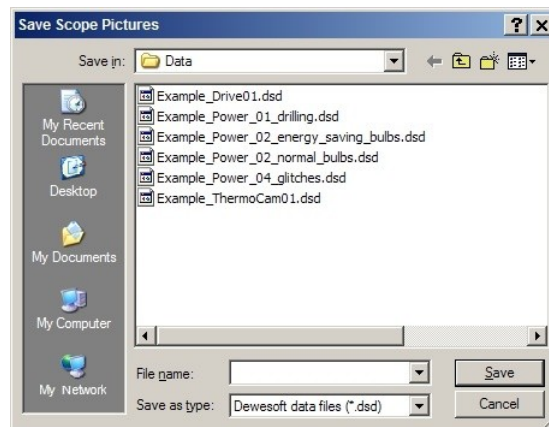
for information about this settings see → **Export screen to AVI**

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.

When **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.

2.1.3.11 FFT Analysis

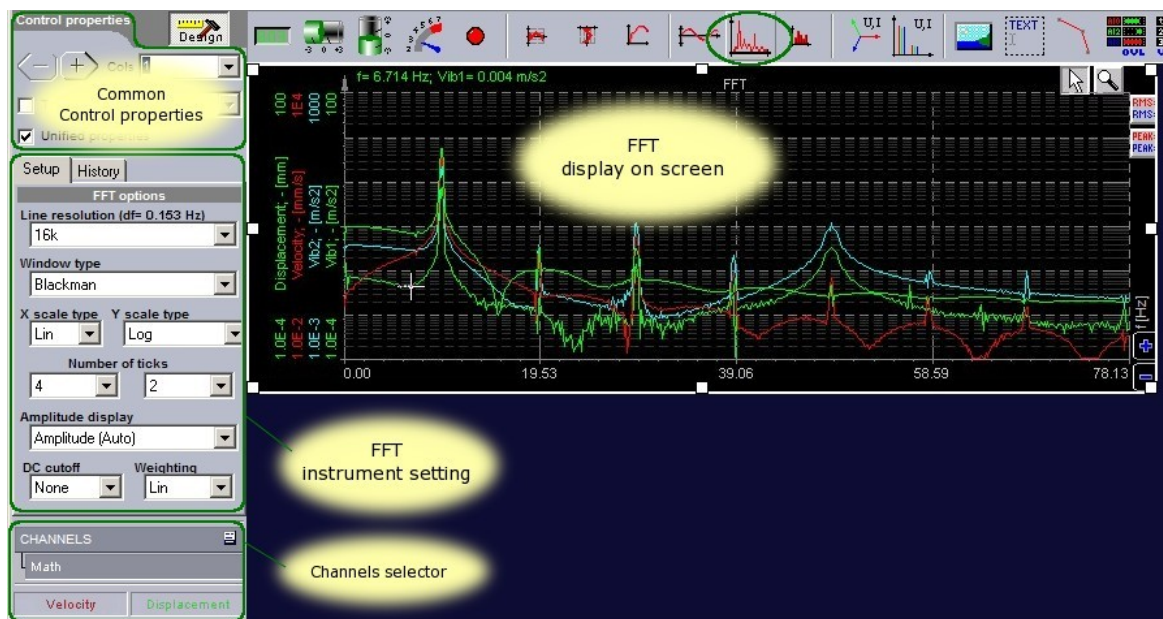


The **DEWESoft FFT (Fast Fourier transformation)** instrument shows the **frequency** components of acquired signals in **amplitude** and **frequency**.

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:

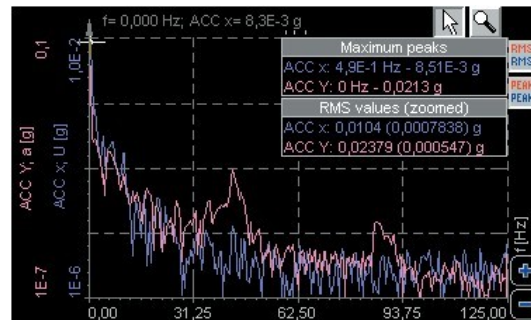
- **Control properties** for detailed information about **FFT instrument** **Control properties**: grouping, number of column, Add / Remove, transparency,...
see → **Control properties**
- **FFT settings** typical setting for **FFT instrument** are combined in two tabs:
 - **Setup** with Line resolution, 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 → **Creating own display**



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** →



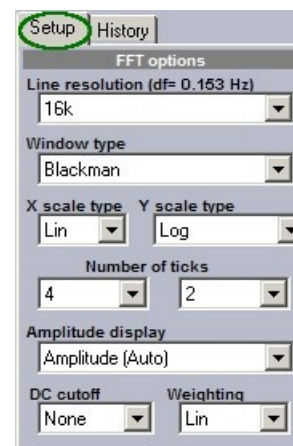
FFT Settings

Serious **FFT analysis** requires several settings to get a useful result. All these settings **DEWESoft** combine in two tabs:

1. Setup

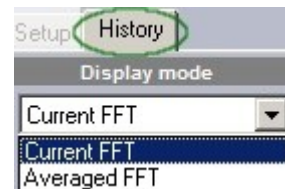
In **FFT Option** can be set:

- **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**

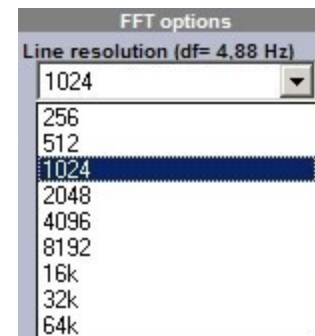


FFT Setup settings

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 / 1024 lines). The result is **4.88 Hz per line resolution** (mentioned in the selection line).

To change the **FFT lines**, just click in the **Line resolution** field and select from the drop down list.

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**

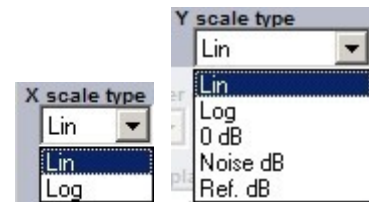
→ see → [Reference Guide](#) → **Theory of frequency analysis**



X and Y scale type

DEWESoft allows:

- two different **X** (frequencies) **axis** types (**Linear** and **Logarithmic**)
- four **Y** (amplitudes) **axis** scaling types (**Linear**, **Logarithmic**, **0 dB** and **Noise dB** and **Ref. dB** scaled)

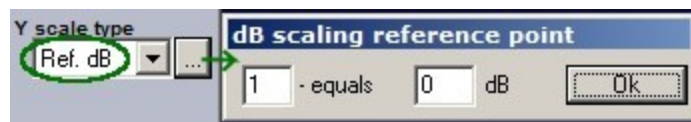


Select the *axis type* from drop down list according to your application.

for detailed information and using hints about **X and Y scale type**

→ see → [Reference Guide](#) → **Properties of the Fourier transform**

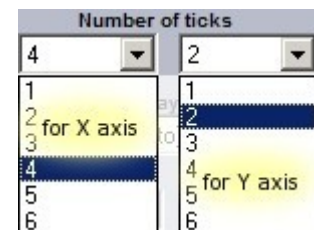
When in **Y scale type Ref. dB** is selected, 'Db scaling reference point' window appear to *enter* this reference point and confirming that with **OK**.



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**.

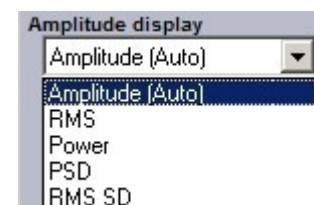
The first field in row is to select the **X axis** ticks, the second row for the **Y axis** ticks.



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*.



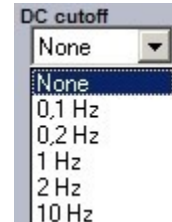
If we have the sine wave with **2 V** peak *amplitude* (**4 V** peak-peaks), we would have in the FFT **2 V** *amplitude*. For the table let's assume that the original signal unit is **Volts**. If the units are *different*, it will appear in the FFT.

Amplitude type	Units	Description
Amplitude (Auto)	V	is the <i>pure</i> signal amplitude
RMS	V rms	is the RMS amplitude, calculated as Amplitude/sqrt(2)
Power	V * V	calculated as RMS value squared

PSD	$V * V / \text{Hz}$	calculated as RMS squared, <i>divided</i> by the line resolution and <i>sqrt(2)</i> - used for checking the <i>noise</i>
RMS SD	$V / \text{sqrt}(\text{Hz})$	calculated as RMS value, <i>divided</i> by the <i>square root</i> of line resolution - also used for checking the <i>noise</i>

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 **Linear** **Weighting**. For *sound analysis*, special FFT weighting can be set. As opposed to the *sound module* in *math*, where the weightings will be calculated in *time* domain, this will calculate the sound weighting in *frequency* domain.



FFT History settings

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.



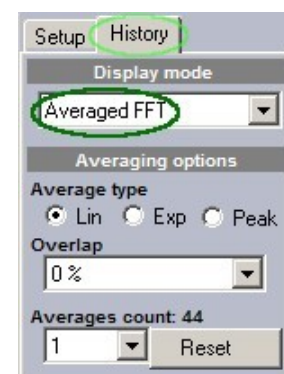
2. Averaged FFT

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.

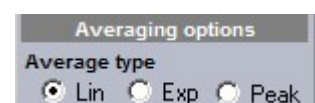
for detailed information about **Averaging**

→ see → [Reference Guide](#) → **Theory of frequency analysis**



Average type

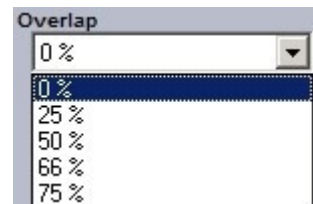
From **Averaging options** select **Average type**: **Linear**, **Exponential** or **Peak**. As a standard, **linear** is selected.



Overlap

Depending on the application, it may be necessary to define a **data overlap**.

When the *window type* is used, we have to use **overlap** otherwise some of the data will be *ignored*. Therefore the use of overlap is highly recommended.

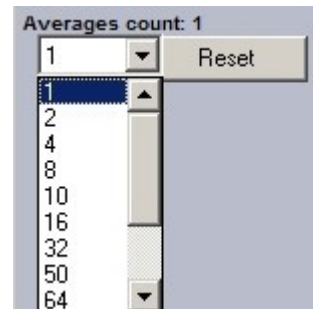


for detailed information and hints about **Overlap** → see → [Reference Guide](#) → [Theory of frequency analysis](#)

Averages number

In **Averaging options** select the **maximum averaging values** from drop down list. On row above the **Averages count: n.** - **Averages number** is displayed. Numbers run from 0 to selected **maximum averaging values** (100 on drop down list).

When **Reset** button is selected, running number is set to 0.



2.1.3.12 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.

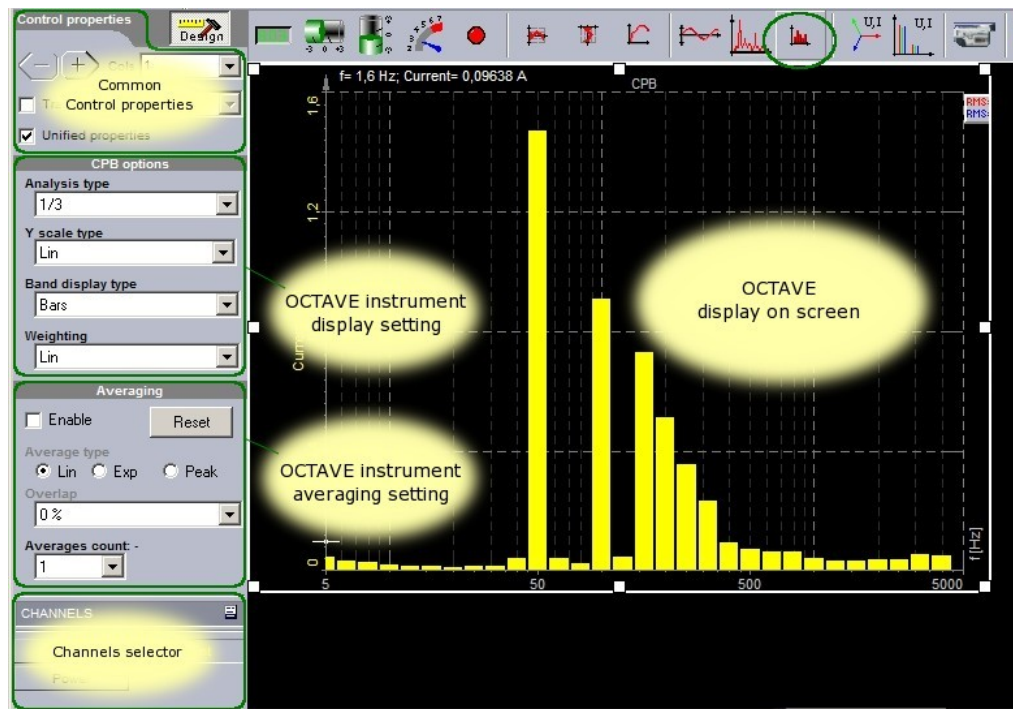
for background information about **FFT analysis** → see → [Reference Guide](#) → [Theory of frequency analysis](#)

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:

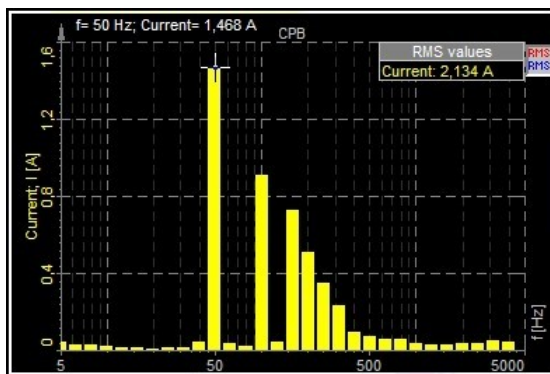
 - **Y-axis scale/auto scale** (Common instruments tools)
 - **Online calculation**
- **Channels selector** for detailed information about **assigning / reassigning** channels to/from **Octave**
→ see → [Creating own display](#)



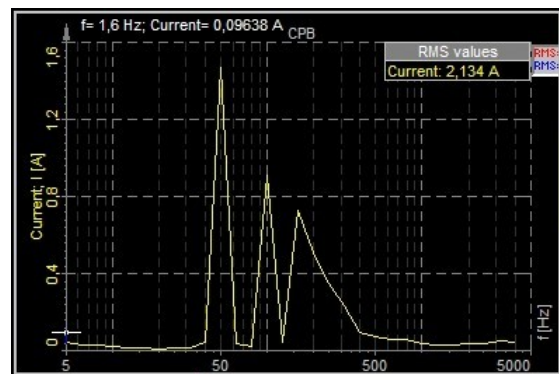
Appearance on screen

The **Octave** element offers *all important* information: **channel name(s)**, **unit(s)**, **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:



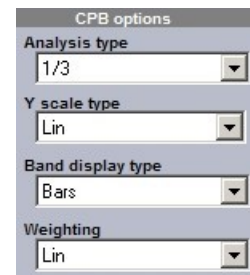
for the **RMS values** →



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](#)

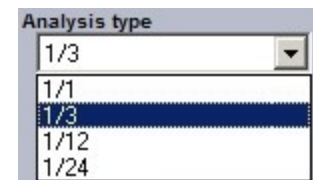


Analysis type

DEWESoft knows four different **Analysis types** for FFT analysis. The analysis type defines the *width of each band*.

The next band is calculated as $2^{(AnalysisType)}$ 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.

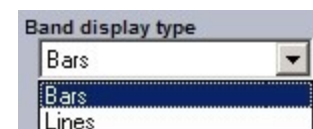


Band display type

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)

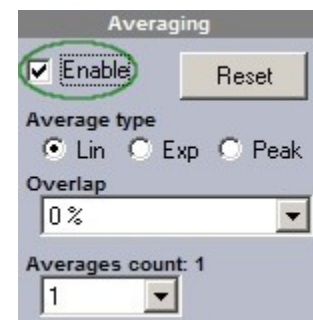


Octave Averaging settings

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](#)



2.1.3.13 3D FFT



The **DEWESoft 3D FFT** instrument **shows** the **frequency** analysis of **acquired** signals in **amplitude** and **frequency**, but this time **also** with either **time** or **RPM history**.

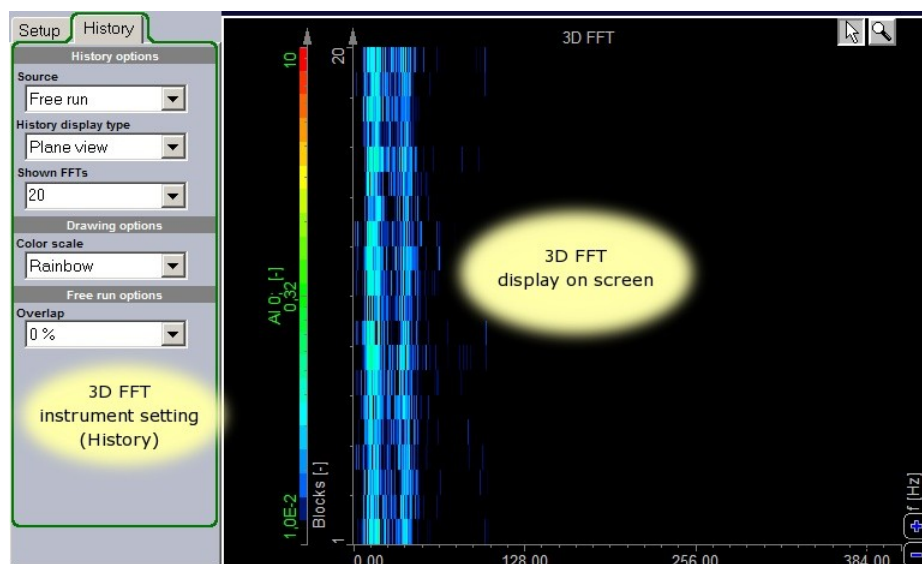
for background information about **FFT analysis** → see → [Reference Guide](#) → **Theory of frequency analysis**

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 **3D FFT instrument** **Control properties**: *grouping, number of column, Add / Remove, transparency,...*
see → **Control properties**
- **3D FFT settings** typical setting for **3D 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* - is same as for **FFT instruments**
 - **History** *display options* for *Current FFT* and *Averaged FFT*

Available **appearance** setting for **3D FFT instrument** in **Run mode** are (same as for **FFT instruments**):

 - **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)
- **Channels selector** for detailed information about **assigning / reassigning** channels to/from **3D FFT**
→ see → **Creating own display**



The screen appearance, frequency and value scaling as well as the basic (**Setup** tab) **3D FFT** settings are virtually the same as with the normal FFT, so please consult the **FFT display** section for these settings.

3D FFT History settings

Totally different is the **History** tab section where can be set:

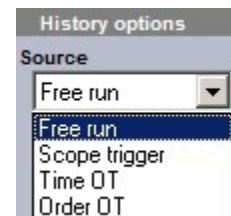
- **History options** - Source, History display type and Shown FFTs
- **Drawing options** - Color scale
- **Free run options** - Overlap (only for *Free run* Source)
- **Scope history options** - Pretime (only for *Scope trigger* Source)
- **Order history options** (only for *Time* and *Order OT* Source)

History options

Source

From **Source** option drop down list different *mode* can be selected:

- | | |
|----------------------|--|
| Free run | displays <i>non triggered</i> FFT shots. The history Z axis in this case is the time. |
| Scope trigger | displays an FFT shots which relates to the <i>scope trigger</i> - when a scope <i>triggers</i> , a new FFT is added to the FFT history. |
| Time OT | the FFT history relates to the <i>order tracking</i> - one order tracking math module is needed that this mode works. The <i>history axis</i> in this mode is the machine RPM and the X axis is the <i>frequency</i> in Hz. |
| Order OT | the FFT history relates to the <i>order tracking</i> - one order tracking math module is needed that this mode works. The <i>history axis</i> in this mode is the machine RPM and the X axis are <i>machine orders</i> . |



History display type

For all Source option we can select from **History display type** drop down list:

- | | |
|--------------------|---|
| Plane view | Plane view is the view of the FFTs from the <i>top</i> where the <i>amplitudes</i> are shown like colors. |
| Waterfall | or 3D view - displays the FFTs in a <i>waterfall</i> -like type. Very helpful to show <i>signal abnormalities</i> or <i>changes</i> . |
| Persistence | Events will be displayed in an <i>overlay</i> technique; the <i>older</i> the events are, the <i>lighter</i> the colors will be. |

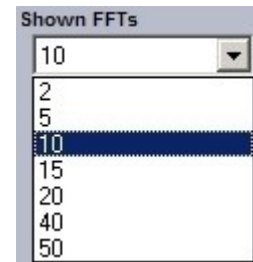


Shown FFTs

For all **Source** option except **Order tracking** we can select from **Shown FFTs** drop down list.

Please be aware that choosing *high* number of 3D FFT shots might load the PC, so *check* the *maximum* value before storing the data.

Note also that in **Time** or **Order OT** mode the *number* of FFTs is defined by the maximum, minimum and delta RPM in the **Order tracking** math module.



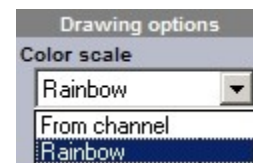
Drawing options

Color scale

From **Color scale** option drop down list *color mode* can be selected:

From channel Channel color will be used.

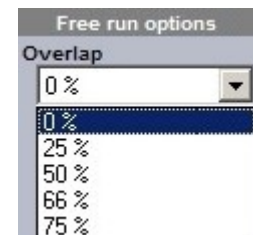
Rainbow default



Free run options - Free run Source

Overlap

same as for *FFT display* → see → **Averaged FFT - Overlap**



Scope history options - Scope trigger Source

Pretime

Can be used to **define** the 3D FFT shot **pre-time**. At the point of triggering of the scope, it determines *how many points* are taken *before* the *trigger shot* to *calculate* the *FFT*.

If for example 512 points are defined for FFT and the overlap is 25%, it will take 128 points *before* the trigger shot and 384 *after* the trigger shot.



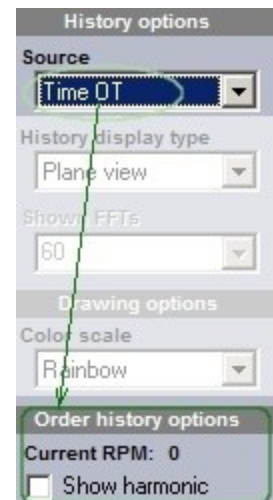
Order history options - Time OT Source

On this section **Current RPM** is displayed .

With selection **Show harmonic** checkbox on the 3D FFT graph H-1 harmonic is displayed.

Order OT Source

When this option for *Source* is selected, **Show harmonic** option is not available:



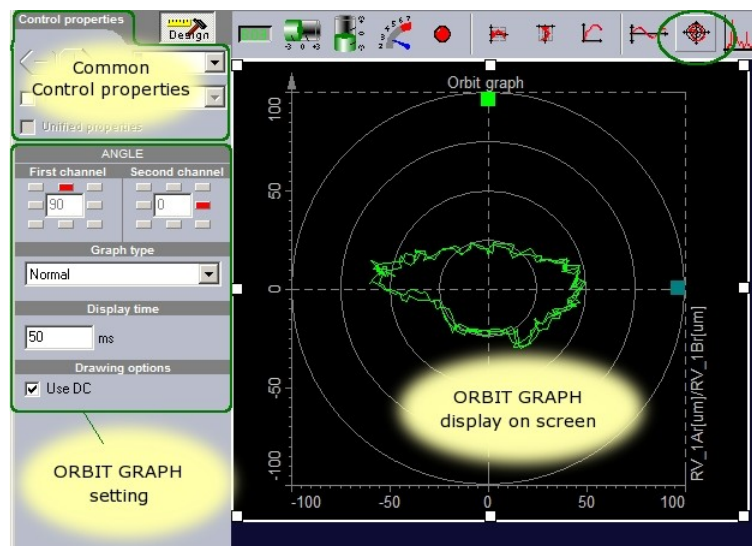
2.1.3.14 Orbit plot



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 → **Creating own display**

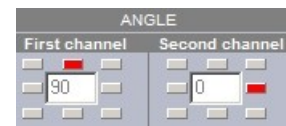


Mounting angle

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.

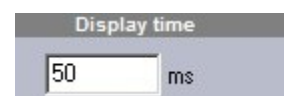
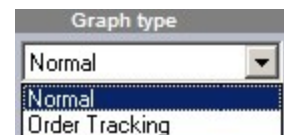
Please note that the sensors should not be mounted in a *straight line* - there should be an angle offset between them.



Graph type

There are two possible graph types - **Normal** 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.

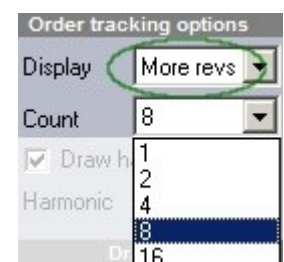
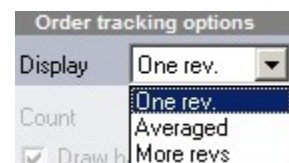


- In **Order tracking** mode the signal sources can only be *channel* used in the **order tracking**.

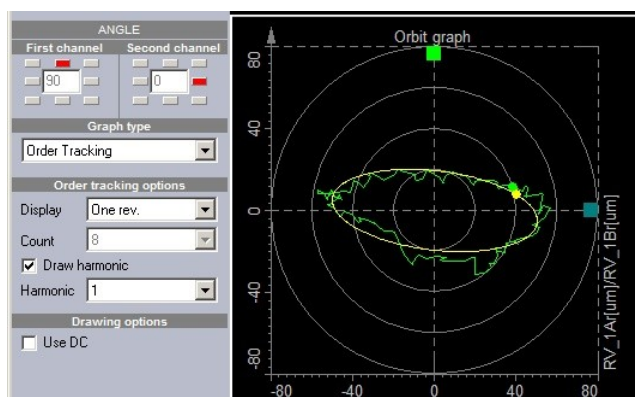
Since order tracking defines the *rotation frequency*, we can display *current* rotation - **One rev.** mode, **Averaged** number of cycles or **More revs** (cycles). For last two modes we need to define the number of cycles to average or display.

For last two modes we need to define **Count** - the *number of cycles* to average or display.

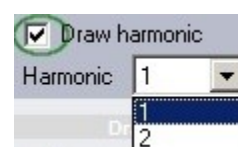
When for Display **One rev.** is selected, Count field is not available (value 1 is defined).



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.



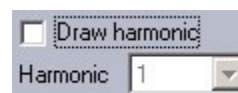
These harmonics must be defined in list of *Output extracted harmonics as channels* section of **Order tracking** module *setup* screen, otherwise *only first* harmonics is available on list:



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.



2.1.3.15 Vector scope



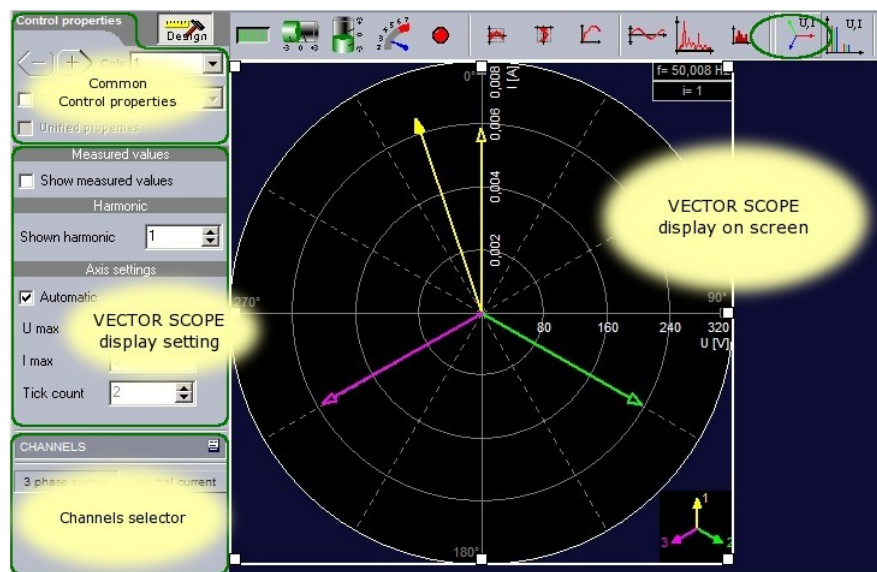
The **DEWESoft Vector scope** instrument is used for displaying the **amplitudes** and **phase angle** between the voltage and current power module **channels** and additional to the vector also the most important **measurement values** for each phase: **U_i**, **I_i**, **Phi**, **cos Phi**, **P_i**, **Q_i** and **Si**.

for background information about **Vector scope** → see book → **Power quality the new challenge** by W. Schoeffer, published by Dewetron

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 → **Creating own display**



Appearance on screen

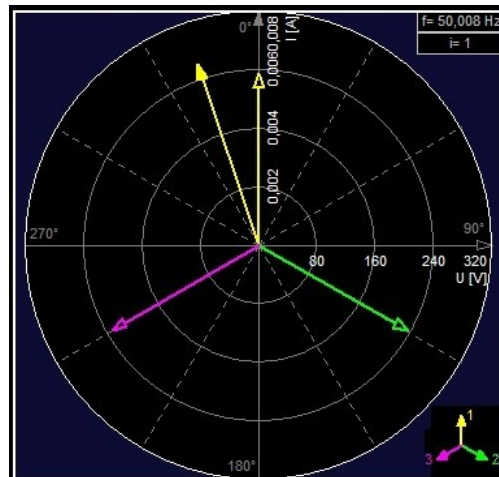
The **Vector scope** displays the **phase angle** between the channels and:

channel names

unit(s)

frequency information

typical values



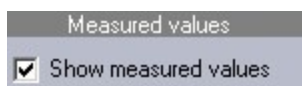
for detailed information about *instruments positioning, size and transparency* see → [Screen edit functions](#)

Measured values

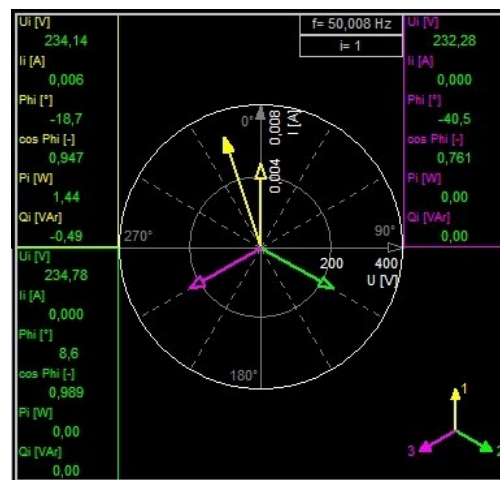
Show measured values

This function **shows** in addition to the vectors also the most important **measurement values** for each phase: U_i , I_i , Φ_i , $\cos \Phi_i$, P_i , Q_i and S_i (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

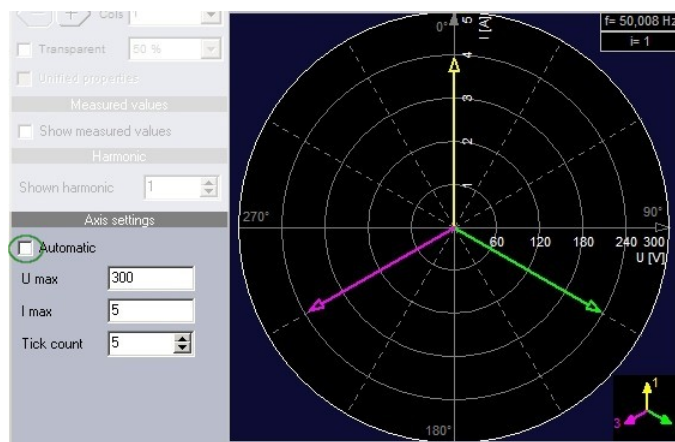
With **Automatic** enabled, the vector scope always **scales** to the *maximum* of all **displayed channels**.

Manual set

When **Automatic** is disabled, you can *enter* value for:

- **Umax** and
- **I_{max}**

Use the **Up / Down** arrow to *increase /decrease* **number of ticks** -**Tick count**.



2.1.3.16 Harmonics analysis

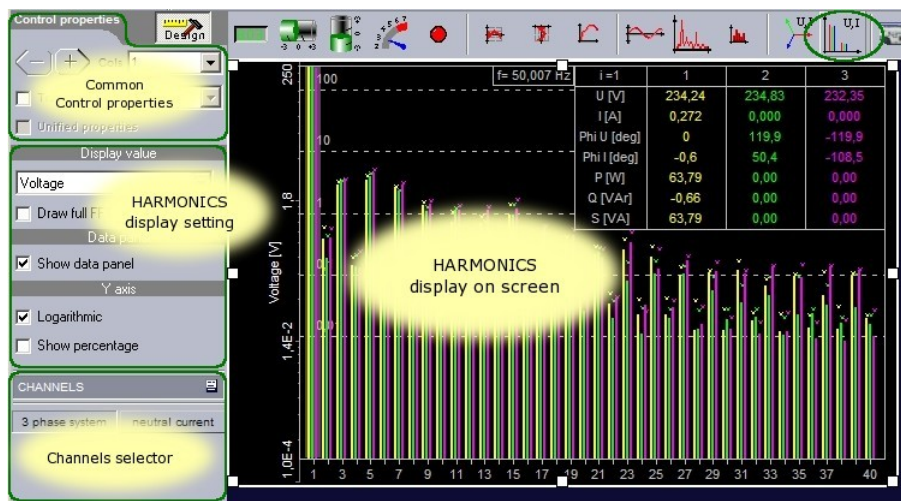


The **DEWESoft Harmonics** display shows **frequency components** of the **input signals**. **Harmonics** analysis is similar to the **FFT** analysis, the main difference is the type of calculation and displaying: the harmonics display refers to a **base frequency** (e.g. **50** or **60 Hz**) and displays **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:

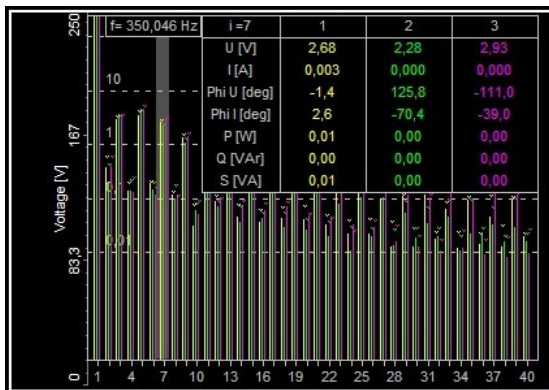
- **Control properties** for detailed information about **Harmonics display** **Control properties**: grouping, number of column, Add / Remove, transparency,...
see → **Control properties**
- **Harmonics settings** available **appearance** setting for **Harmonics display** instrument:
 - **Display value**
 - **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 → **Creating own display**



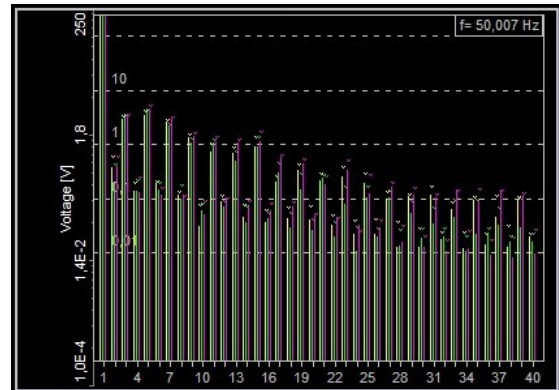
Appearance on screen

The **Harmonics** display shows: **base** and **harmonic frequencies**, **channel name(s)**, **unit(s)**...

Harmonics display with Datapanel and Logarithmic Y axis



Harmonics display with Logarithmic Y axis



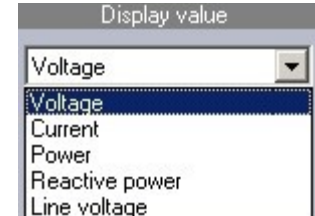
for detailed information about *instruments positioning, size and transparency* see → [Screen edit functions](#)

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:

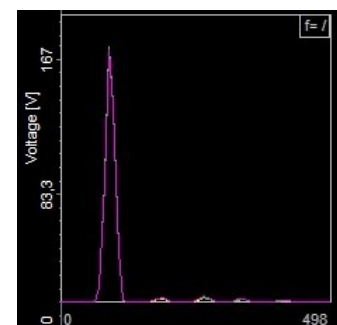
- **Voltage**
- **Current**
- **Power** - active power
- **Reactive power** - this is *wasted energy*
- **Line voltage**



Select appropriate *option* from drop down list according to your needs.

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

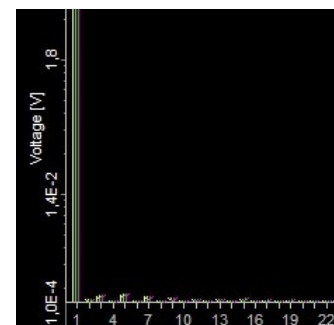
In this part of **Harmonics** display settings you can choose with *check / uncheck* appropriate box different **Y axis scaling**:



Logarithmic

Logarithmic checkbox in **Y axis** section is:

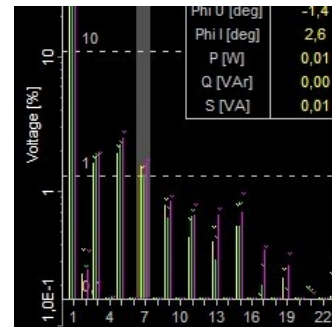
- *Selected* → **logarithmic** Y axis scaling (example see right)
- *Unselected* → **linear** Y axis scaling (example picture see above)



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)



2.1.3.17 Video display

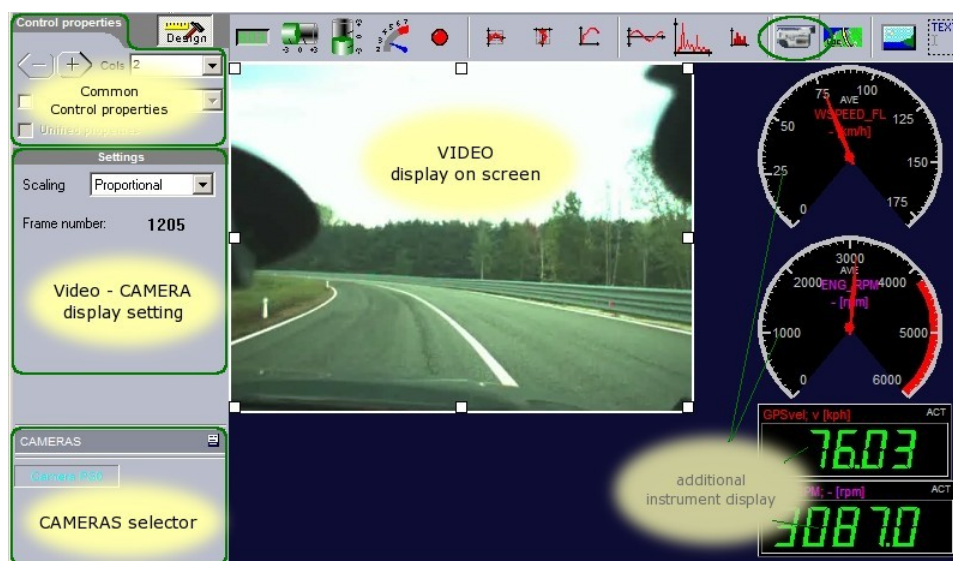


DEWESoft Video display is provided to **show** content sensitive **acquired videos** together with other different data of measurement in various instruments. This video information can help to *interpret* and to *document* these data and measurement.

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.

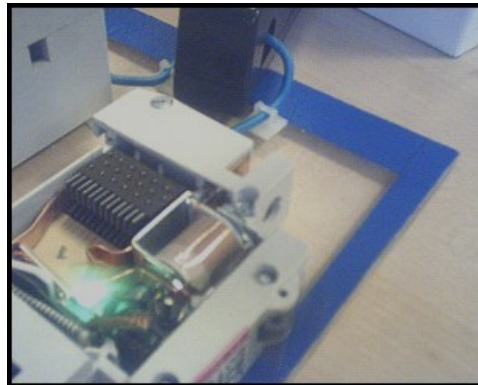
When you select **Video display** in the **design** or **run** mode, following settings will appear on left part of the screen:

- **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

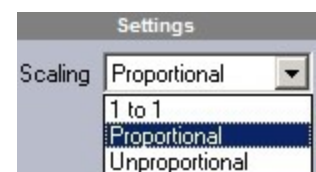
Camera information

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 can display the *available cameras* to select the right one.



2.1.3.18 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 → **Creating own display**



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

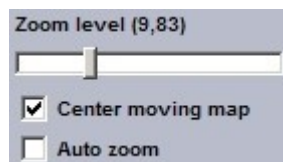
The **GPS** instrument can be switched in *two modes* using the tabs:

- [View](#)
- [Calibrate map](#)

View setting

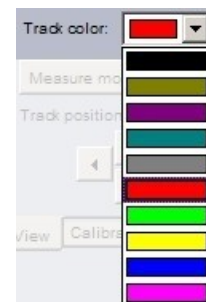
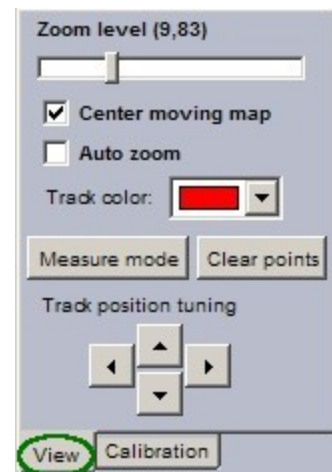
- **Zoom**

DEWESoft offers two ways to scale the track: *Auto zoom* or *zoom manually*.

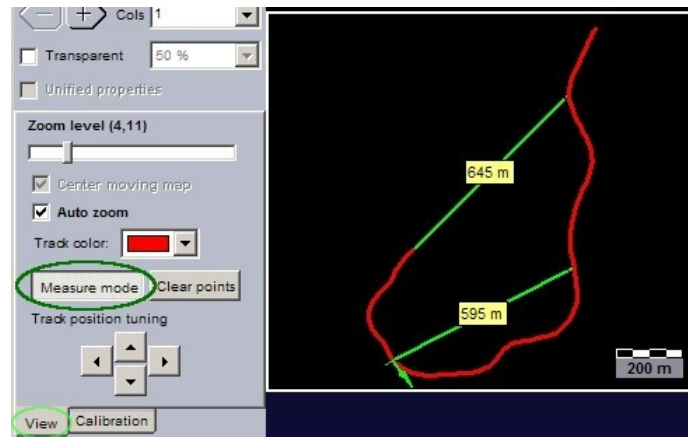


Use the **Zoom level** slider to *zoom manually*; above the slider after caption, you can see the **zoom factor**.

- 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.



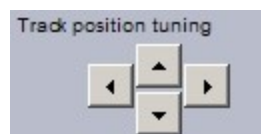
- When you select the **Measure mode** you can **measure distances** within the map. Simply click a point within the map for the *starting* point. Move the cursor to the second *desired* position and click again - a *line* will be *drawn* with the distance *labeled*:



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.

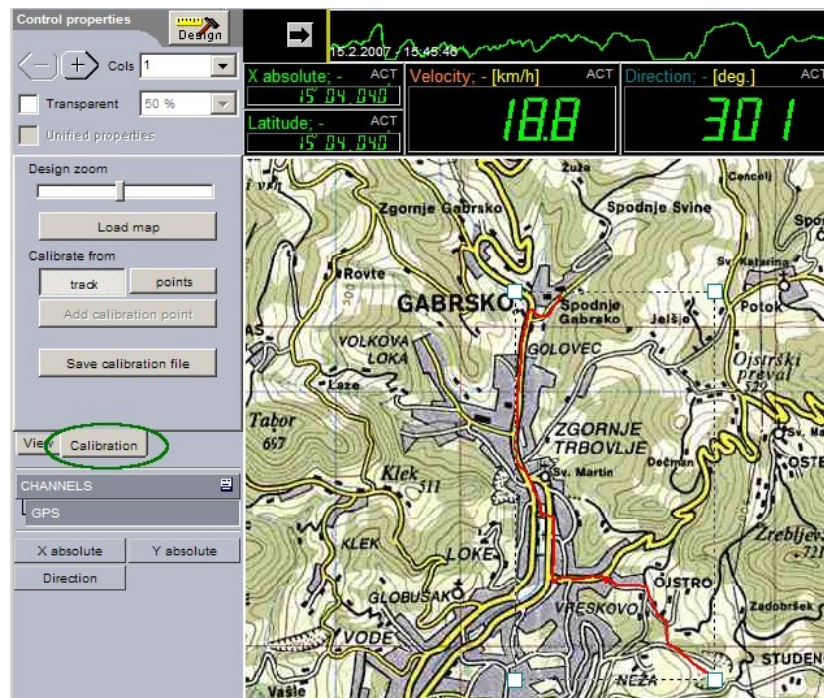
Calibrate Map

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.

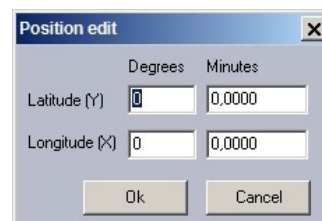
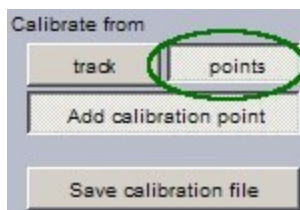


If we *don't know* the coordinates of the map, we can take a *short tour* in the area storing the measurement. Then we go to GPS screen, go to **Calibration** and **Load map**. The map is shown in behind and the traveled path gets four handles around it. Now we can *drag* and *adjust* those handles to 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.

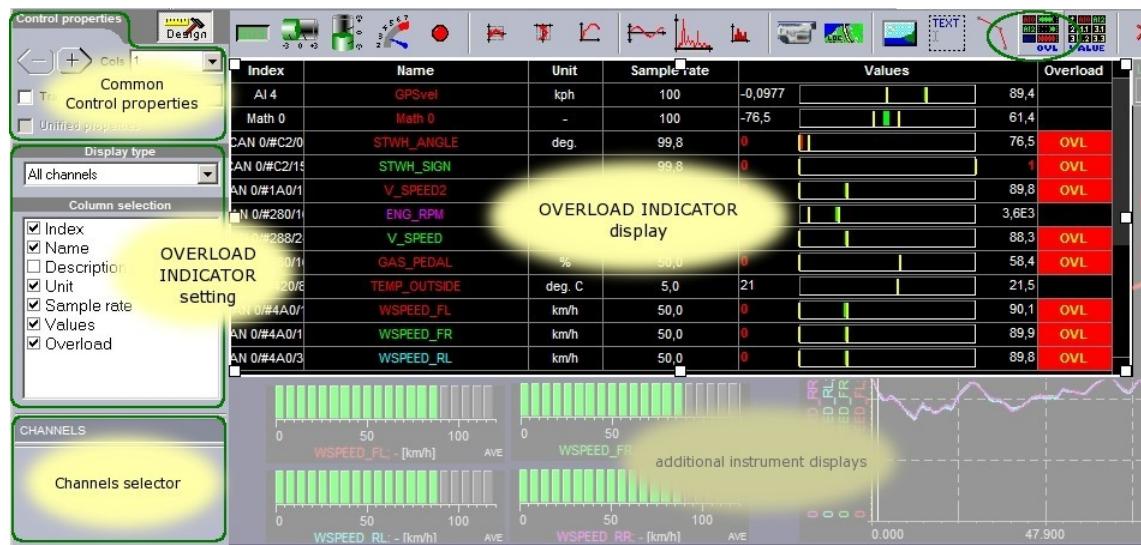
2.1.3.19 Overload indicator



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 → **Creating own display**



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
AJ 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
CAN 0/#C2/15	STWH_SIGN		99,8	0	1
CAN 0/#1A0/17	V_SPEED2	km/h	50,0	0	89,8
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

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:

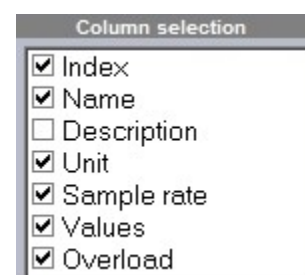
Index	Name	Unit	Sample rate	Values	Overload

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).



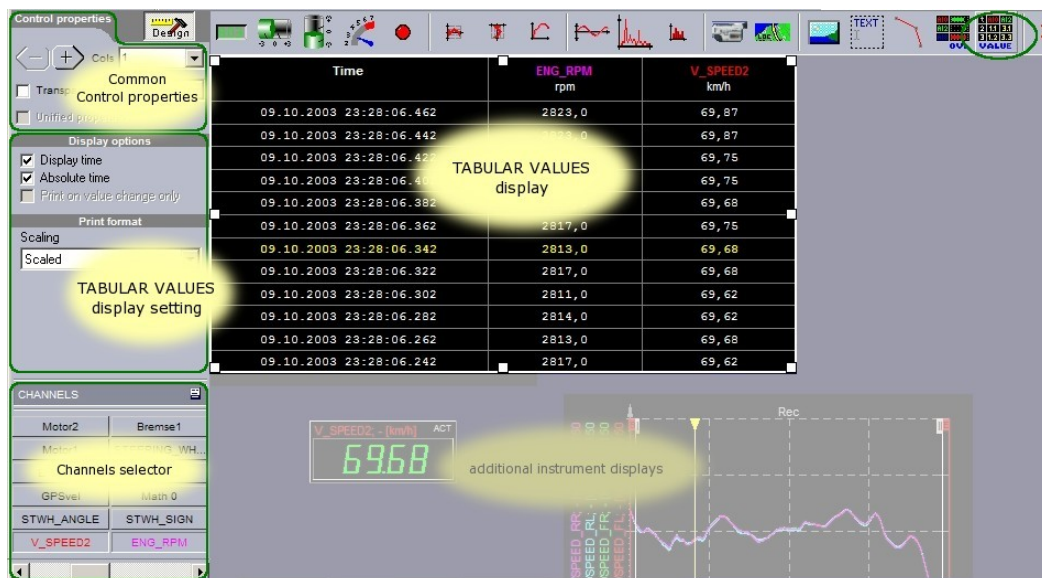
2.1.3.20 Tabular display



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 → **Creating own display**



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

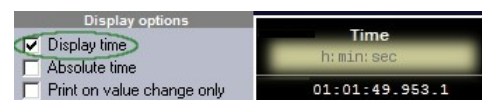
The **Tabular values** display offer three possibilities:

- Display time
- Absolute time
- Print on value change only

Simply check box in the **Display option** to:

1. Display time

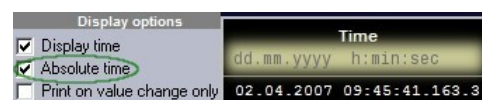
When we select this field check box, **time data** from measurement start in *predefined* format is **displayed**, if it is not checked, the **Time** column is *hidden*.



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*.

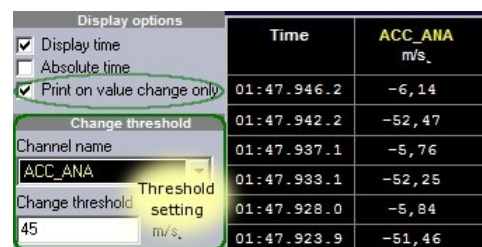


All **values** of selected channels are **displayed**.

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 to 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.

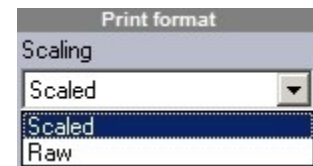


Print format

The **Tabular values** *display* offer settings for displayed *channels values*:

- **Scaling**

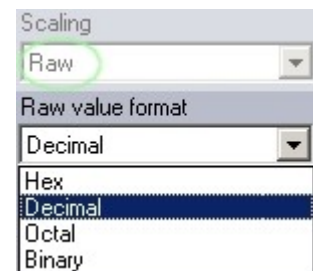
Select the **Scaled** or **Raw** type from drop down list according to your requirements. Usually the scaled value is displayed, but sometimes it is nice to see the raw channel values, especially when data is transmitted digitally (CAN, PCM or other digital buses).



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	
Print format	V_SPEED2	Print format	V_SPEED2	Print format	V_SPEED2
Scaling	km/h	Scaling	km/h	Scaling	km/h
Scaled	76,62	Raw	7662	Raw	1DEE
	76,62	Raw value format	7662	Raw value format	1DEE
	76,68	Decimal	7668	Hex	1DF4

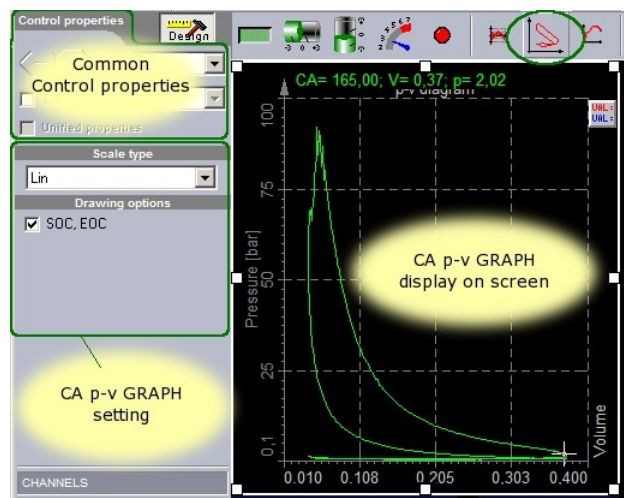
2.1.3.21 CA p-v



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:

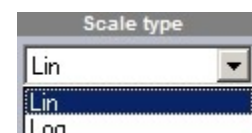
- **Control properties** for detailed information about **Control properties**: *grouping, number of column, Add / Remove, transparency,...*
see → **Control properties**
- **Scale type** describes the orientation of the sensor
- **Drawing options** There are two basic graph types - **Normal**, where 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 → **Creating own display**



Scale type

The scale can be displayed either as **linear** or **logarithmic** (for *volume* and *pressure*).

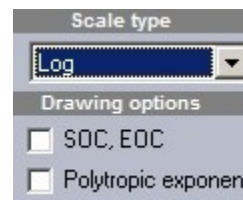
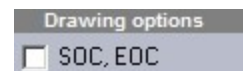
Linear axis is common used type for seeing the *real* value of the pressure, while logarithmic have two advantages: the *pumping cycle* can be seen very nicely and also the *polytropic expansion* and *compression* are *linear* in log-log scale.



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*).

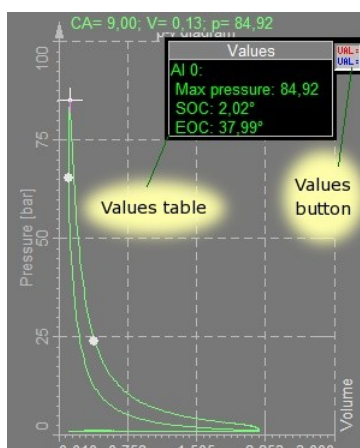
Additional option with *logarithmic* scale is to display the **Polytropic exponent**. This is used to see how good the entered *polytropic coefficient* fits to the *measurement data*.



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  button and Values table appear:



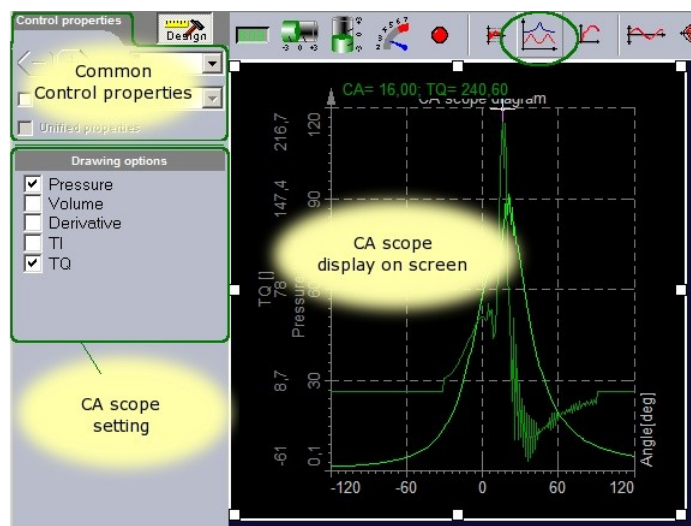
2.1.3.22 CA scope



The **DEWESoft combustion scope** shows the *x-y scope* of the **crank angle** and the typical **combustion curves** aligned to the angle offset of each cylinder.

When you select *combustion scope plot* in the **design** or **run** mode, following settings will appear on left part of the screen:

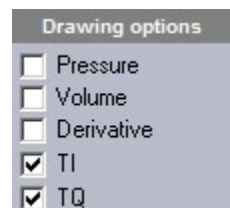
- **Control properties** for detailed information about **Control properties**: *grouping, number of column, Add / Remove, transparency,...*
see → **Control properties**
- **Drawing options** selects which parameters will be shown on the graph.
Available **appearance** setting for **CA scope** in *Run mode* is:
 - **Axis scale/auto scale** (Common instruments tools)
- **Channels selector** for detailed information about **assigning / reassigning** channels
→ see → **Creating own display**



The input to the *CA scope* can *only* be combustion pressure or additional channels. It will show them *aligned* to the *reference cylinder*.

Drawing options

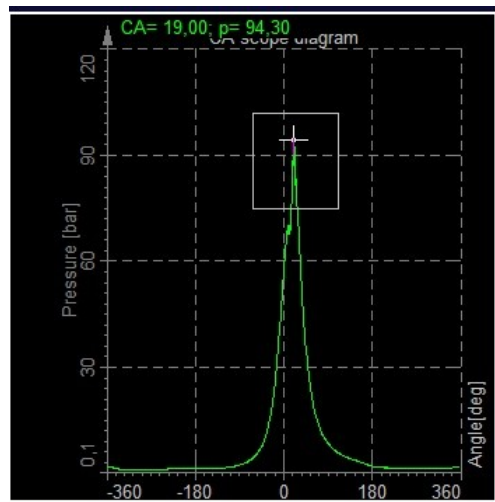
For each channel we can display cylinder **Pressure** (measured data), calculated **Volume**, **Derivative** of the pressure, **TI** (integrated heat release) or **TQ** (heat release curve).



Each individual property will be having *each own axis* and the axis scale can be *modified to fit* the wanted range.

The angle **x** axis extends from **-360** to **360** for *four stroke* engine and from **-180** to **180** for *two stroke* engine but we can enter the value to show *only a limited range*.

Another way of zooming is simply to click on one point of the graph, then choosing the *region* with *dragging* the mouse while holding the mouse button and releasing the mouse button on the end point of the zoom. The **box** with a *new zoomed region* will be displayed and when a mouse button is released, it will zoom in the selected region in *both* axes.



2.1.3.23 Additional controls

DEWESoft offers an additional **data displays** and **controls** to create *instrument appearance* on the online display screen.

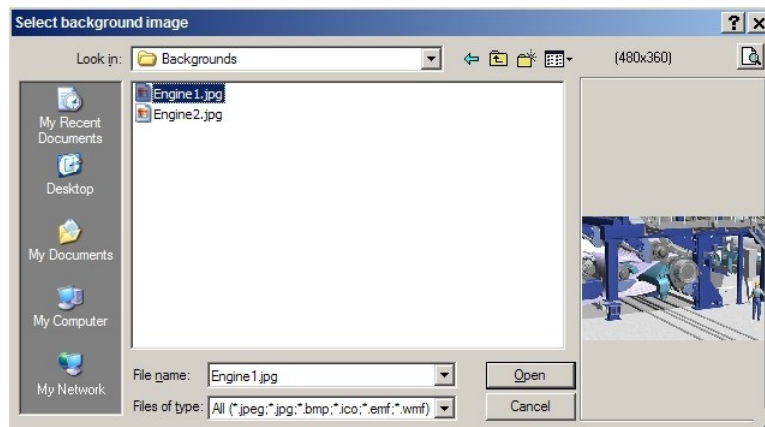
- **Background picture** used to *illustrate* your measurement
- **Text element** used to *write text* on the screens
- **Line element** used to *draw* lines, connect different elements,...

Background picture



The **background image** can be used to *illustrate* your measurement or as *base picture* to place different *instrument* displays with measured data on it.

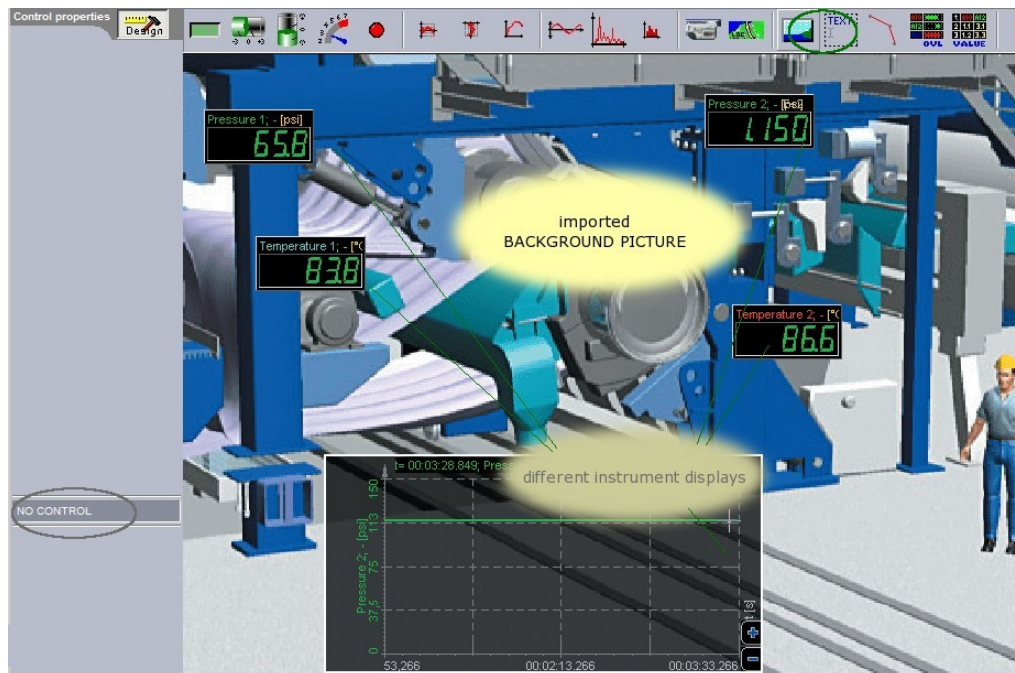
When you select the **Background picture** control in the **design** mode, the **Windows Explorer** window appears to *select* the desired graphic file:



Supported background file *formats* are: **jpg, jpeg, bmp, ico, emf** and **wmf**.

Now press **Open** and *arrange* the other instruments as required.

Example: Overview screen with background image



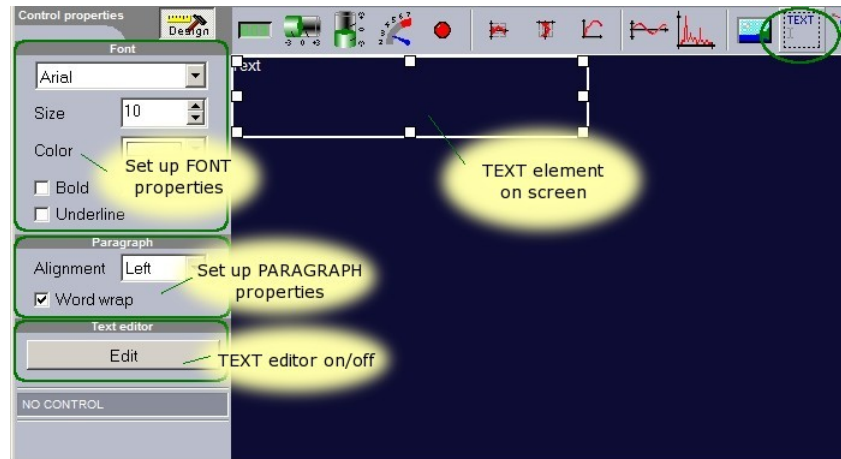
Note that the **background images** don't have any *control setting*.

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 wax 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).

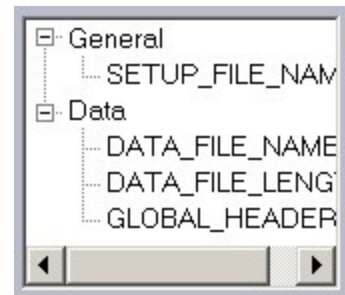
Text editor

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>
```



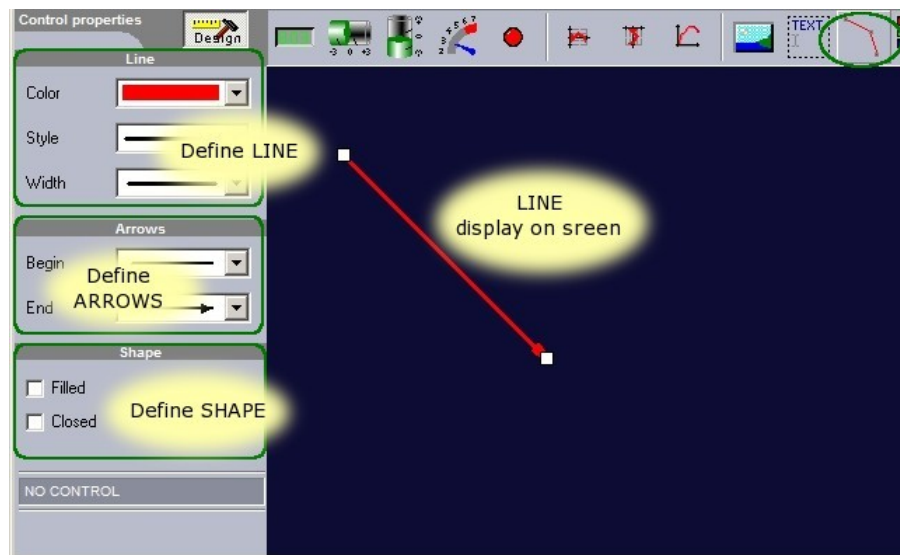
When you have finished, simply **click** anywhere *outside* of the text element or **press** once again **Edit** button to **confirm** the changes.

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 between two Text elements



Draw shapes

In **Shape** section you must *first* check **Closed** (and in case of need **Filled**) field, then:

1. you draw *line* with two points (first two corners of shape) as described above
2. move mouse cursor to *third* corner of shape, **press** and **hold Shift** key on keyboard and when you **left click** the shape appear; while **pressed left mouse** button, you can *move* this corner on desired location; with *release* left mouse button (and consecutive **Shift** key) shape is drawn
3. on this way you can add *fourth, fifth,...* corner

Positioning and size

- To **modify** a **line**, simply **click** once on it to *select* element and then **move** the *end* points to the desired *new* location.
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.

2.2 Recording Acquired Data

Measured data are stored in **DEWESoft** *.dsd (DEWESoft data file). We **predefine** the filename to be used for **data recording** and some other recording features on the **DEWESoft Setup** screen:



- **DATA FILE OPTIONS**
 - **Setting the filename** – setting **filename** and **Data header**
 - **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
- **DYNAMIC ACQUISITION RATE** setting a **sample rate** to be used for *dynamic* acquisition
- **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** - *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.

2.2.1 Basic Recording Setup

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*.

for detailed information about setting **filename** see → [User Guide](#) → **File Name Setup**

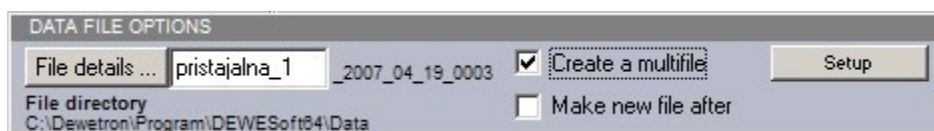
for detailed information about **Global data header** see → [User Guide](#) → **Data Header Design**

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 'pristajalna_1') 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

With checking **Save multifile opt. in setup** field all multifile settings (see above) are stored in setup file (otherwise it is stored as global setting).

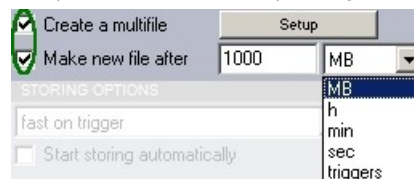
Make new file after

When the **Create a multifile** box is *checked*, the **Setup** screen has an additional 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*.

in this example **DEWESoft** will stop storing after 1000 MB

Please *select* between *file size* values like

MB (Megabytes) and **triggers** (number of trigger events)



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

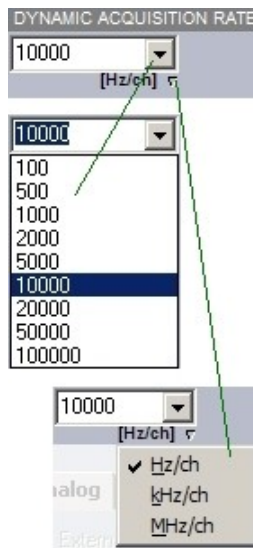


If we select *time* relates value, we get additional **Absolute time** field. When this box is *checked* the *absolute time* is selected. This is very useful when acquiring data for longer time periods. If we choose to switch the file *each hour* with absolute time, then switching will be done *exactly 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 not 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).

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

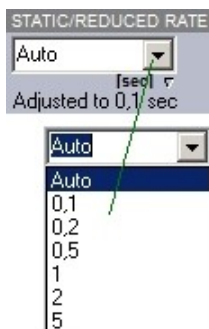
Sometimes it is required to *change* the **units** for the sampling rate (**Hz/ch**, **kHz/ch**, **MHz/ch**). Simply click on the *small* arrow below the drop down list to **select** another **scaling**.

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* 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 important to achieve a useful scaling!

STATIC / REDUCED RATE

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



You can select the value from the drop down list, like **0.1 seconds**.

This means that every 0.1 seconds the *reduced* data will be available.

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

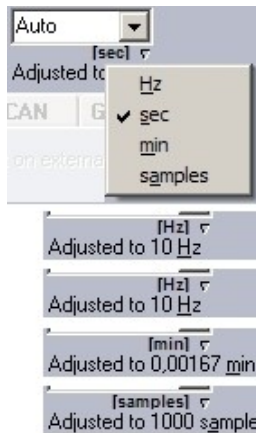
You are also allowed to enter your *own* values, but be aware that not all values will be accepted - if so the *real value* will be mentioned below the drop down list.

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

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

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

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



CHANGE 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.

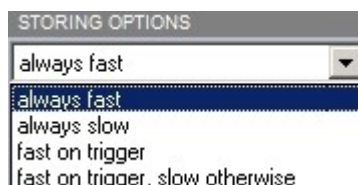
When you switch to the **Analyse** mode, and *load* one of these files, you can **export** any set of values:

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

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 though 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:

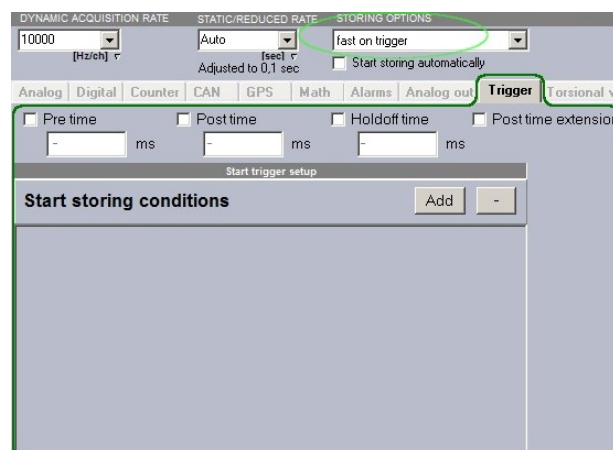


for information about **start / pause / stop storing** see → [User Guide](#) → **Start / Stop Recording - Manually**

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.

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



for detailed information about **set up trigger** see → [User Guide](#) → **Setup - Trigger**

for information about **start / stop storing** see → [User Guide](#) → **Start / Stop Recording - Triggered**

Start storing data

For **starting storing automatically** check **Start storing automatically** option.

Generally, you can *activate recording* in four ways:

- **Manually**, using the **STORE** and **STOP** buttons on the screen
- **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*
- Start storing **automatically**

for detailed information about **start / stop storing data** see → [User Guide](#) → **Start and Stop Recording**

2.2.2 File Name Setup

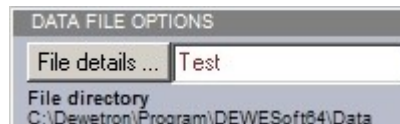
The **file name** is **name** of *data file(s)* to be used for **data recording**.

We can define the **file name** for each measurement *separately* by entering it in the file name edit. We can *predefine* the **file name** on the **DEWESoft Setup** screen. By default, this field will be set to 'Test' as the filename.

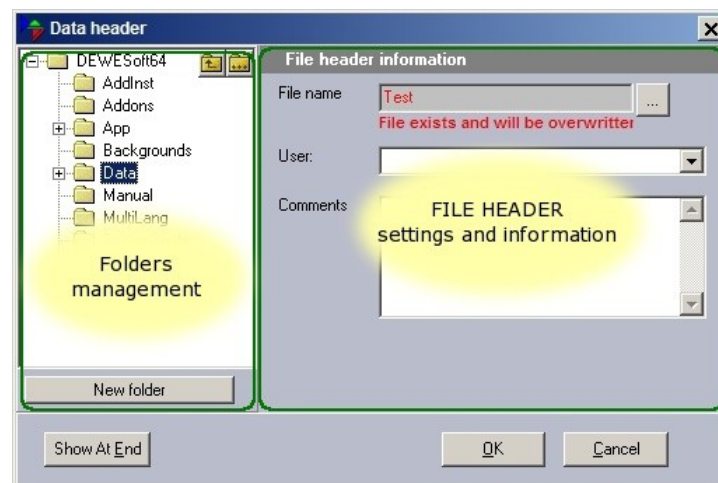
If whatever files you type in here already *exists* in current **data directory**, its name will be colored **red** here.

Normally default data directory is **C:\Dewetron\Program\DEWESoft65\Data**. The default folder where data is stored can be changed by clicking on **File details** button.

Information about defined **File directory** for *data recording* can be seen below filename:



More information can be *entered* using the **File details ...** button on **DEWESoft Setup** screen. When you press this button, the **Data header** window will appear:



The left part of the window is designed to do the *Data directory management*. The right section will be used for *file details*. You can enter the file name, User information, comments,... When all settings are done, *close* the window (also with **Show At End** button).

To give you more *flexibility*, the window can be **designed** according to your own *requirements*.

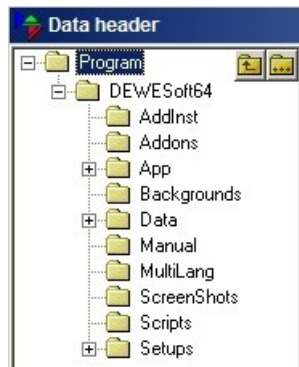
for more information how to design **Data header** see → [User Guide](#) → **Data Header Design**

Folder management

At the top of the window, you see the current *standard* folder for your data files and all sub folders. After the *first* system setup, the standard path is **C:\Dewetron\Program\DEWESoft65\Data**. Sub directories are *not* available.

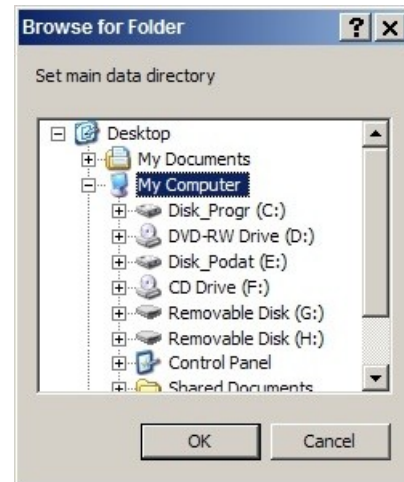
Use the button  to *change* to the higher directory levels - **Up One Level** on directory tree.

When we press one time this button in example screen above, directory tree change:



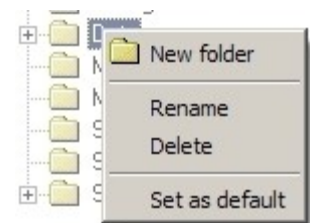
Or use the  button (**Browse for Folder**) to get the general *Browse for Folder* window.

Within the folders list, you can also to manage your data
→ **set main data directory:**



Move the cursor over the *desired* folder (e.g. ...**Data**) and press the *right mouse* button. The window like that on right will appear. It allows you to:

- **create** a **new sub directory** - **New folder**
- **Rename** a **folder** (not working with the *main* directory)
- **Delete** a **folder with all data** (not working with the *main* directory)
- **Set as default** (a folder)
- for *main* directory (e.g. **C:**) **Change folder** option is also available; with selection this option we call *Browse for Folder* window for setting *main data directory*

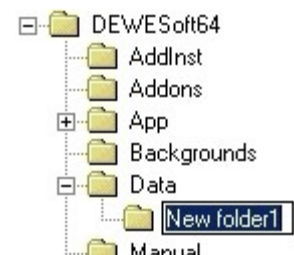


Creating new sub folders

To **create new sub folders**, just use the method mentioned above or the **New folder** button at the bottom of the folders selection list.

When you press **New folder** button, a **new sub folder** of the *selected* directory (e.g. ...**Data**) will appear.

The standard folder name is **New folderxx** and can be *changed* at any time. The **xx** in the folder name is used for a continuous *number*.



When you use the create folder function *several* times *without* selecting a *new* base folder, you will get each time one new *sub* folder within the previous sub folder.



As a standard, *any* new folder will get a *continuous number*. When you have desired to create several new sub folders from the *Data directory*, it may look like this on the right picture.

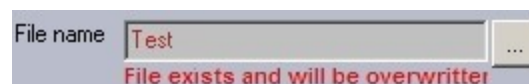


This is already a very powerful data management function. To *improve* it according to your requirements, please refer also:

→ [User Guide](#) → [Data Header Design](#) → [Work with folder levels](#)

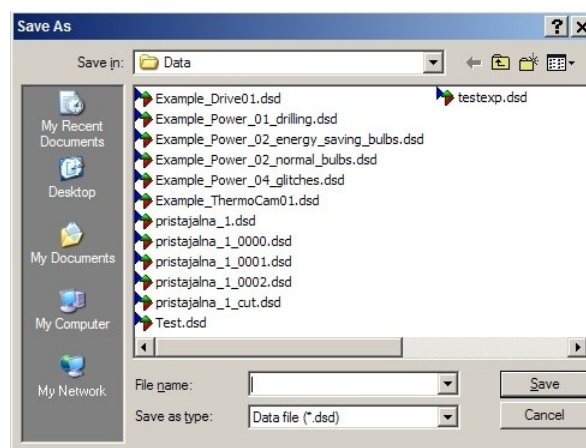
File name

After you have selected the file *destination* in the left section of the window, you can *enter* the **File name**.



If the file already *exists*, the name will be displayed in red color and a **warning message** will appear. If you want to overwrite the file, *keep* the file name. When you *start storing*, a window will appear and ask you if you really want to *overwrite* or *change* the filename.

Use the  button to get an overview of the already existing files in the *selected folder* - standard **Save As** window:



The **Comments** field can be used to *note any information* about the *measurement* which can be important for the *analysis* or to *find data* again. It can be a location, user names, sensor types, and many more.

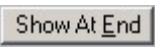
Example:



Closing data header window

When all settings are done, leave the window with the **OK** button or press **Cancel** to reject all changes.

Show At End

There is also another possibility - press the  button to **close** the window. Then you *can start* your **acquisition** and do your **measurement**. When you **stop** the acquisition, the window will *automatically appear again* and you can do the *final changes* for this file, for example *changing* the comments according to your measurement.

CAUTION: *location and file name can't be changed - you have to define them before you start your measurement!*

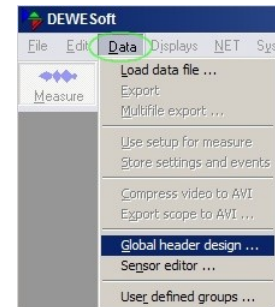
2.2.3 Data Header Design

The **data header** (also called **global** header) is used to define the text fields which can be entered to better *define* the measurement, like *comments*, *operator* and so on.

The **global data header** can be **adapted** according to your *requirements*.

To enter the edit mode, please select:

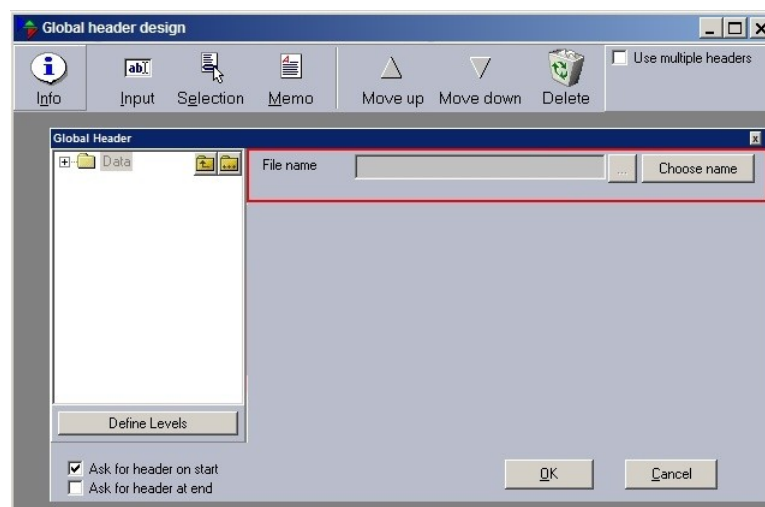
Data menu → **Global header design ...**



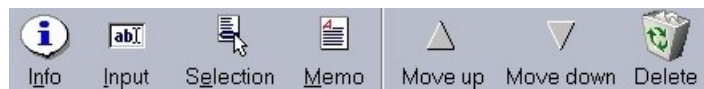
The *Global header design* window has several tools:

- **Icon bar**
create and manage various fields
 - **Info field**
 - **Input field**
 - **Selection field**
 - **Memo field**
 - **Move up**
 - **Move down**
 - **Delete**
- **File name field**
rename and manage data file name field
- **Data directory management**
define data directory level and manage data directory
 - **Ask for header**
 - **Define folder levels**
 - **Work with the folder levels**
- **Use multiple headers**
- **Save Global header design**

When a single global header is not enough, we can use multiple headers and then *select in setup file*, which data header will be used.



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.

The screenshot shows the 'Data Header Design' window. It has a 'File name' field with a 'Choose name' button, a 'User' dropdown menu, and a 'Comments' text area with a line counter showing '5'. At the bottom of the right section, there is a red-bordered box labeled 'Info'.

Just **click** in the new line and **enter** the **desired text**, for example 'File header information'.

File header information

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.

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:

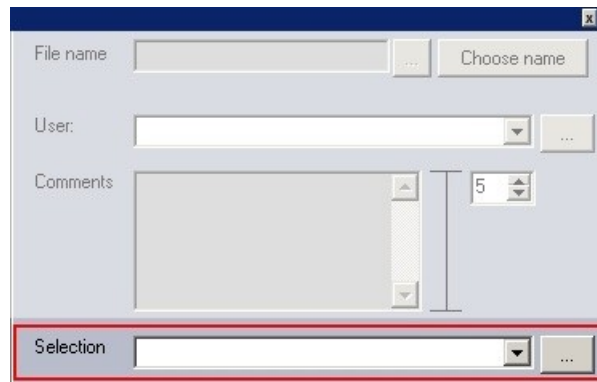


Selection field



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:



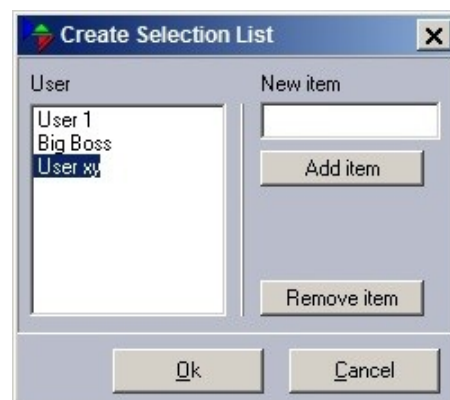
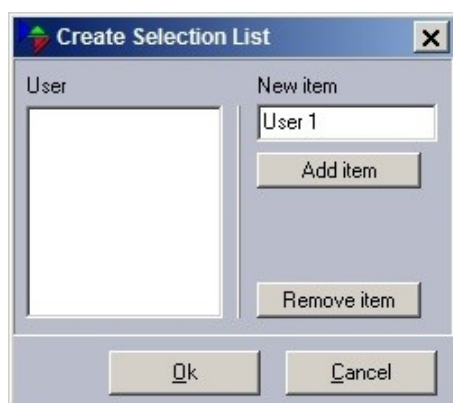
and *label* it (for example '*User*' - click on '*Selection*' and *overwrite* it):



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:

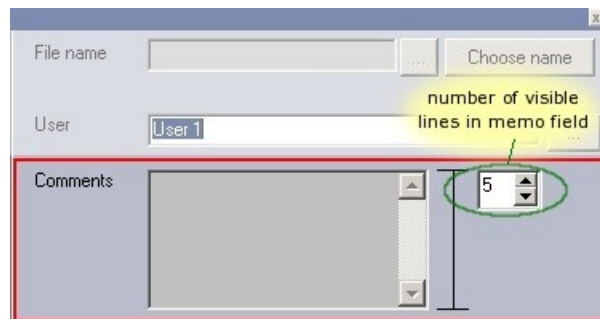


Memo field



The memo field can be *used* to enter any **comment** in more than *one* line. This field is very helpful for any measurement **remarks**, for example connected *sensors* (serial numbers) or *measurement conditions* ('dry weather', 'all machines powered down',...).

Let us **create** a new field with press the **Memo** icon:

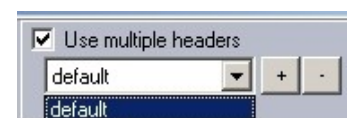


On the right side of the memo field, you can **select** the *number of visible lines* in the memo *field*. This is no limitation in the number or characters, just a limitation of *place* on the display. As a *standard*, **5 lines** are selected.

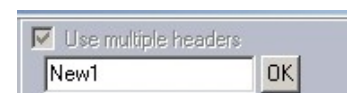
Use multiple headers

Sometimes a single data header is not enough. We can define several headers for specific measurements and relate a specific data header to each setup.

With checking this box you can create and use *different headers*. By default there is one header named '**default**' and two buttons.



By pressing the **+** button we can **enter** *new name* for the header. '**plus**' and '**minus**' buttons are substituted with **OK** button to *confirm* new name.



After accepting new name these appear on the list. The header which we will use, can be now selected from this list.

'**plus**' and '**minus**' buttons are again here.



With the **-** button we can **delete** selected header and warning window appear to confirm delete action.

Now on *Data header* window new **Used header** field is displayed to select proper header from drop down list:

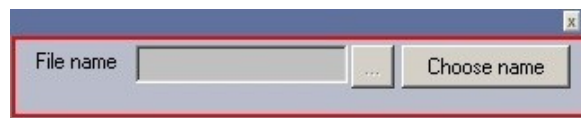


and also **Header entries** section with two buttons:

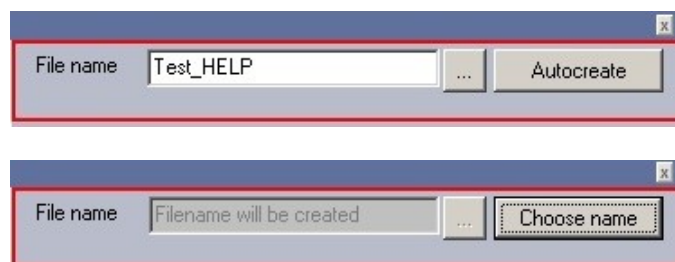
- **Save** to save default header entries
- **Clear** to clear default entries of the data header

File name field

We can **select** the **File name** field with click on it. The currently *selected* field will have a **red box** around:



To **rename** the *field*, just *click* on the '**File name**' (this is *predefined standard* name) and *edit* it. The entry fields have two additional buttons: **Autocreate** and **Choose name**.



All the entries made in the design are overridden by the settings made in **Channel setup** (file name and multifile option), so it can be left blank.

Data directory management

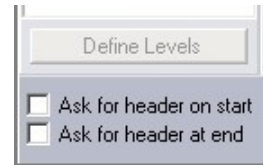
On the left section of the window are setting for **Data directory** management.

Ask for header

The **header** window can **appear** at *two* different *times*:

- at **start** of *measurement*
- at **end** of *measurement*

Just *select* (check) your preferred setting at the left bottom of the window.



If file name already *exist*, on *start* of *storing data*:

- the *Data header* windows appear, if **Ask for header on start** is selected → see also → **File Name Setup**
- otherwise or if **Ask for header at end** is selected, the *Warning* window appear → see also → **Start and stop recording**

Define folder levels

As we have already seen in procedure *File Name Setup* → **Data directory management**, the **global header** can be used to **create folders** easily.

To **adapt** the structure to your *requirements*, you can use the **Define Levels** button. When you press it, the following *Design Levels* window appears.



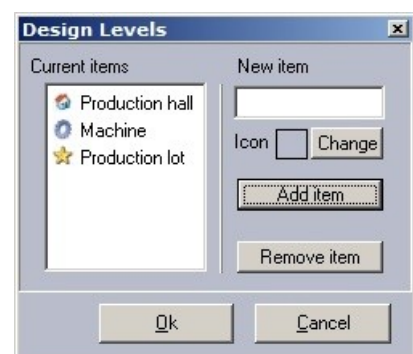
Enter the desired **names** and press the **Add item** button.

It is also *possible* to **add icons** to find your data easier - but not necessary. The icons have to be bitmap files (*.BMP) with 16 x 16 pixel size. Just press the **Change** button in the icon section and we get standard *Open* window for *searching* a matching file.

In our example on right, we enter the three *levels*:

- **Production hall** (*highest level*)
- **Machine** (*first sublevel*)
- **Production lot** (*second sublevel*)

with matching icons.



If you *select* one of the values, the **Add item** button will change to **Edit Item**, you can **edit** item and than confirm changes with this button.

To **remove** a value *select* it and press the **Remove item** button.

To **accept** the entries press the **OK** button or **Cancel** to *leave without changes*.

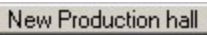
End global header design

To **accept** all *changes* in the **global header design**, just press the **OK** button or **Cancel** to *leave without changes*.

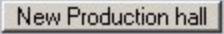
Work with the folder levels

Click  - **Setup** button on **DEWESoft** tool bar and on **Setup** screen press the  button to go back to our new **Data header** window.

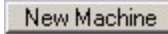
NOTE: following description assume levels entries in upper *Define directory levels* chapter.

You will see now that the button at the bottom of the **folders list** has now changed from  to .

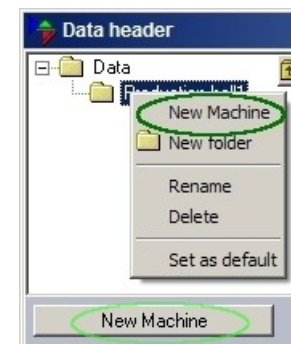
The currently *available folder* can be also selected when you press the *right mouse button*.

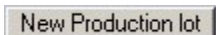
Click on  button or select this new folder from list after pressing the *right mouse button* on sub ordered folder will now *automatically create* a new folder called **Production hall1**.



If you have done that, the *name* of button will change to .

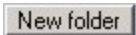
Click on this button or select this new folder from list after pressing the *right mouse button* in the **... \Production hall1** folder, will now *automatically create* a new folder called **'Machine1'**.



If you have done that, the *name* of button will change to .

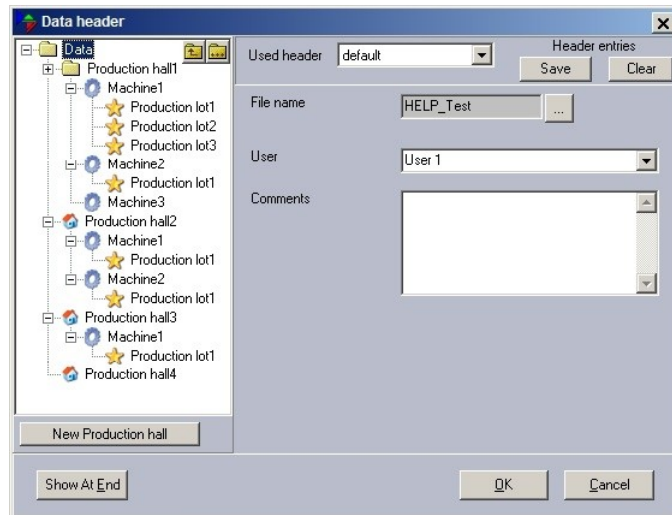
Click on this button or select this new folder from list after pressing the *right mouse button* in the **... \Production hall1 \Machine1** folder will now *automatically create* a new folder called **Production lot1**.



As you can see, **New folder** button *change* according too selected directory *level*. Name of button on *lowest* directory levels change again to .

This is a very *simple* way to create *complex* folder structures very *quickly*.

In our example, the structure can look like that:

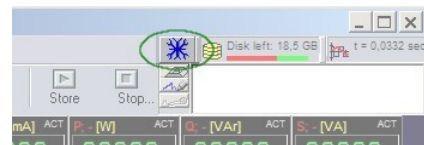


As you can see, our *selected* icons have been *used*. This helps to keep the overview.

2.2.4 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.

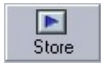
2.2.5 Start and stop recording

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

Start data recording

It is as simple as pressing  - the **Store** button on the main toolbar or press <F5> on the keyboard.

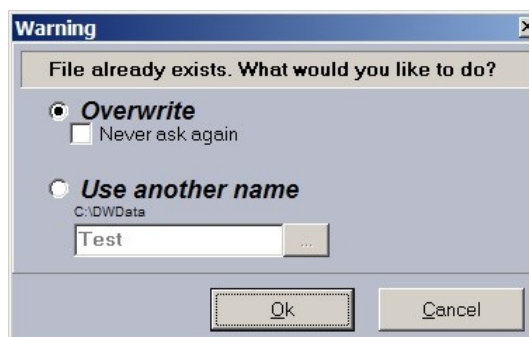


On the right top part of this screen you will see the currently **available space** on your *hard disk drive*.

As you can see in the example screen, the hard disk is nearly full - better free some hard disk memory before you start the acquisition or select another file location.

If a **file** *already exists* of the *same name* as the one that you are trying to record, the following **Warning** window will appear (the **Data header** windows appear, if on the **Global header design** window **Ask for header on start** option is selected

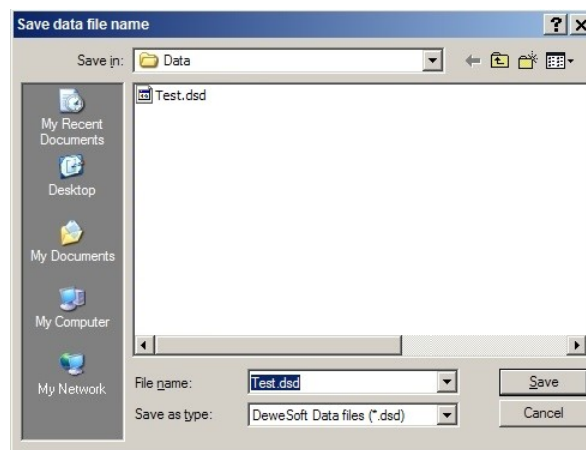
→ see also → **File Name Setup** and → **Data Header Design**):



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.

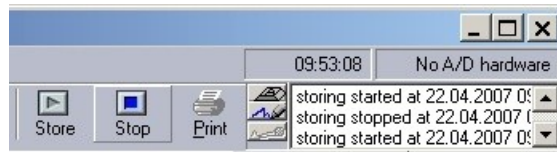
Pause recording

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 same **data file**.



Stop recording

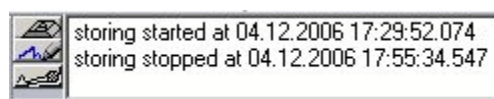
To **stop** recording click the  - **Stop** button or press <F7> on the keyboard.



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 **K** symbol, and a notice is added to the scrolling **EVENT** box.
- **Notice Events** - press 'n' key on your keyboard or the  icon and then **type** some *text* in. The **text input** dialog box will appear:



When you press the **Enter** key, the text is *added* to the record *at that point*. You can also click the **OK** icon on the screen to enter a notice event. The event is also marked with thin **gray** vertical line and an **N** icon on *data* at location when we started entering the event.

- **Voice Events** - If you have a *DirectX sound card* installed, and it has been *set up* under the **General Setup** section of the **System** menu, you can use a *microphone* to make *spoken notices* to your recorded data. Simply press and hold down the 'V' key on your keyboard, or click the microphone icon  and then speak into the microphone. *Release* the key when done. These voice events can be *replayed* through your speakers later on, and are a great way to add a rich layer of information to your data. This event is marked with the **V** icon and a thin vertical **gray** line.

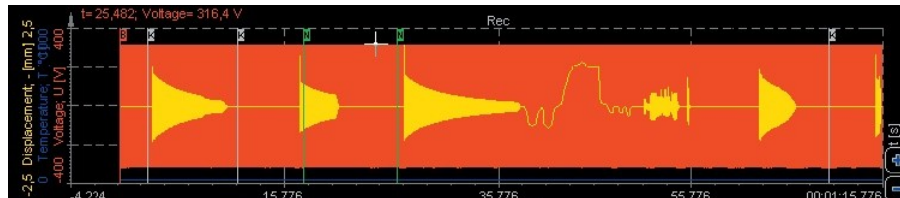
Events can be **added**, **edited** or **deleted** also in **Analyse** mode → see → [User Guide](#) → **Signal analysis**

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 **E** 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

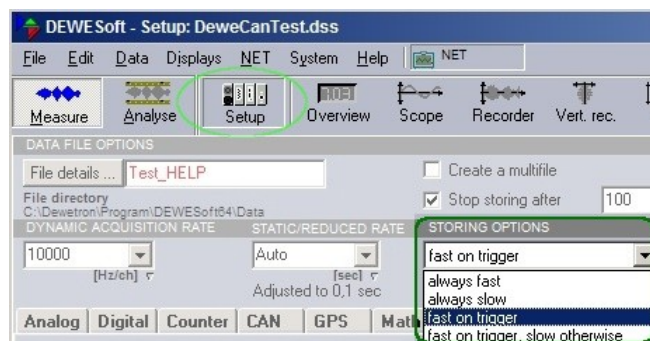


Triggered

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*.

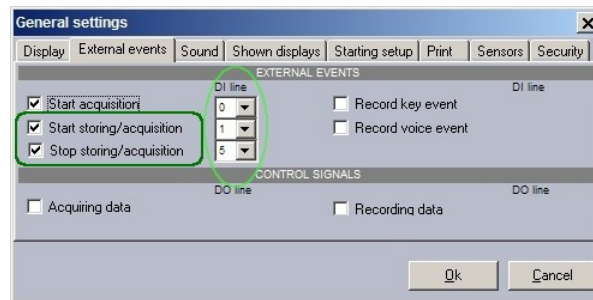
If the *post time* is not defined, data will be stored until a *manual Stop* button is pressed or **Stop storing condition** occurs.

Remotely

You can use the **digital I/O lines** installed on most *National Instruments A/D boards* to **start** and **stop recording**.

NOTE: *Currently is this option only available with National Instruments hardware!*

This option should be set up under the **System** menu, on the *General Settings* 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 → [Reference Guide](#) → **General setting**

2.3 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 Dewetron **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):

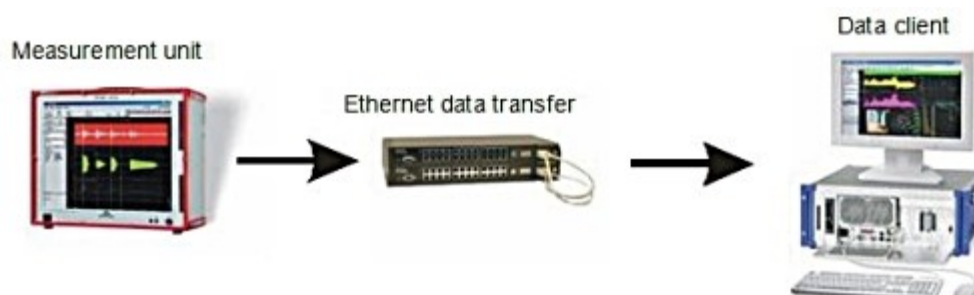
NET Setup	Network <i>configurations</i> , appropriate hardware and DEWESoft-NET setup: • Hardware setup → see Installation Guide → NET module • DEWESoft NET Setup → see User Guide → Setup - networked data acquisition
Measurement	Manage measurement units with NET menu option including <i>Measure transfer speed</i> ; Creating a display , <i>measure-acquire</i> data and store this data on net
Analyse	Analyze acquired and stored data on net, <i>export</i> 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:

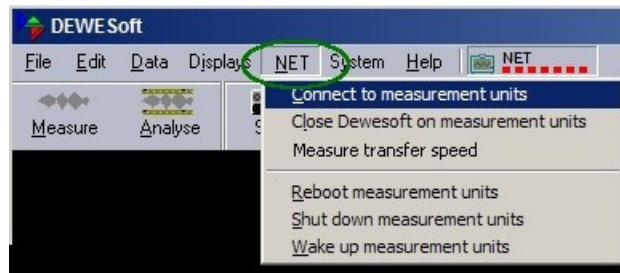
- the *required computing power* is too high for a single measurement unit (e.g. many channels sampled with a high sample rate)
- there is too much *distance* between the units for analog data transfer
- the measurement unit is *not accessible* (e.g.: dangerous measurements, test rig measurements,...)
- 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:

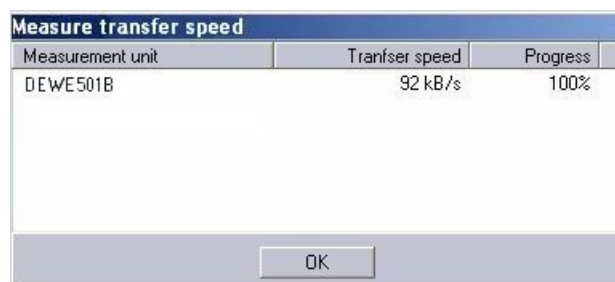


Notice from the menu that you have several other useful capabilities:

- **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

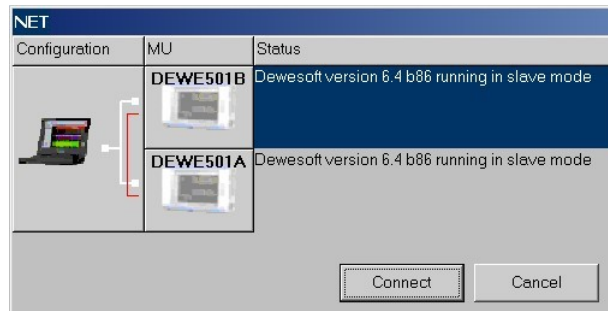
Click the **Measure transfer speed** item and DEWESoft will perform a *test*, *measuring* the **bandwidth** (transfer speed) between the MU(s) and this *client*:



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 Dewetron **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**!

2.3.1 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, '**WEBSTER**' is *added* in front of the transfer channels *from* that **MU** by default, as you can see above.

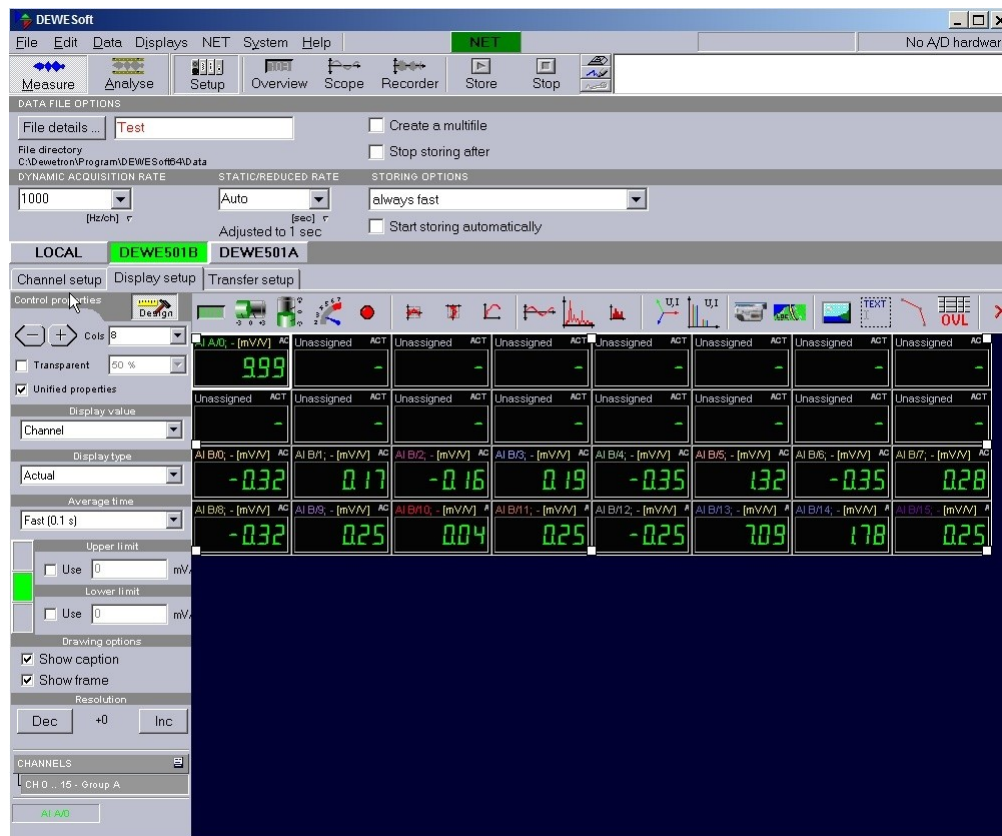
The *channels* from each **MU** will be shown this way *automatically*. This is the only thing that is different from setting up a screen in the standalone mode of **DEWESoft**, for years now.

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 **DEWE501B**! It is not the display that you will see here on the **client**.

Storing data

The **live channel data** is transferred via **Ethernet** to the **client(s)** with the drawback that the *Ethernet interface* is the *bottleneck* for the transfer. Therefore the live channel data can be **stored** in *different ways* to avoid excessive network traffic and optimize the *calculation* power of unit:

- **Local**

Measurement data is stored *locally* (i.e. on the **client** side). The *live data* of the *selected channels* is *transferred* to the **client** where it is stored.

The maximum **throughput** for local storage is **12 MByte/sec**.

- **Remote**

Measurement data is stored on the **measurement unit** (i.e. on the *server* side). The data is *transferred* to the **client** *manually* (on *user request*), when the measurement is *stopped*.

The maximum **throughput** for remote storage is **8 MByte/sec**.

- **Local & Remote**

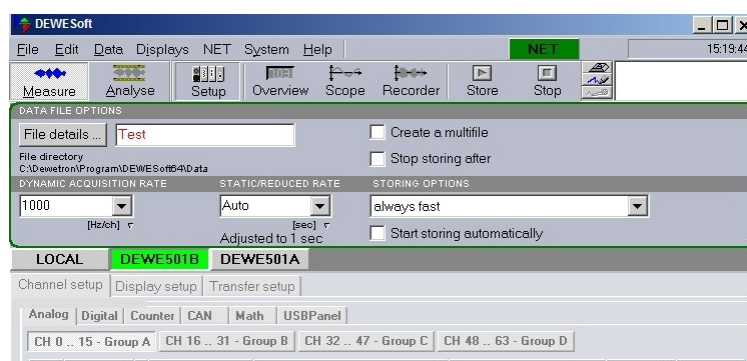
It is often *not possible* to **transfer all** possible channels in *real time* to the client for storage even with a gigabit Ethernet interface.

Imagine even one DEWE system with **32 channels**, being sampled at **200 kB/s** each at 24-bit mode. This is already 25.6 MB/sec, which is more than **200 Mb per second** (where each Byte = 8 bits). It does not take long for the network to be completely *overloaded* with data and be overwhelmed with packet *loss*.

As it is mentioned in [User Guide](#) → **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 **DEWE501B** or **DEWE501A**) so that we can proceed to set up this **Slave MU**:

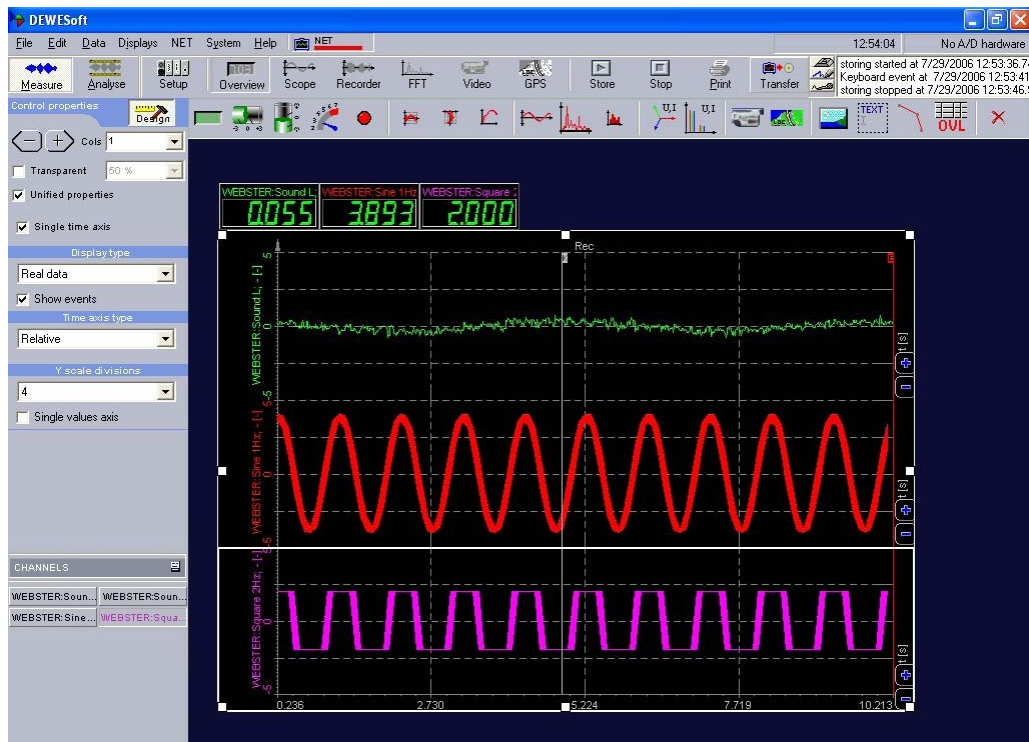


Now click the **Setup** button and on central part of this screen you are doing exactly what you would be doing by *stand-alone MU*: set the *dynamic and reduced sample rates*, choose a *filename*, set *storing option*,...

for information about **storing settings** → see [User Guide](#) → **Recording Setup**

Storing data on the MU(s)

With the **client** and **MU** properly configured, we can now **store data**. Just press **Store** from the toolbar, in the normal way.



You can see from the **event log** (see right) that we **started** storing, added a **notice** event by pressing the spacebar during the recording, then **stopped** it by press **Stop** button from the toolbar.



for details see → [User Guide](#) → **Recording Data**

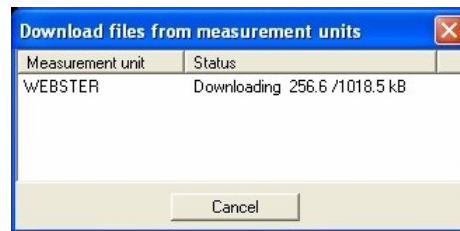
Uploading stored data to the client

Immediately after the **acquisition** is *stopped*, a button called **Transfer** appears on the controlling **client** allowing the **data file(s)** to be **immediately uploaded** from the **MU's** for viewing on the **client** computer.

When **Store data on measurement units** has been *checked* during **DEWESoft NET configuration** (**System** menu → **Hardware setup...** → **NET** tab), the data is **transferred** from the **local MU** for viewing on the **client** computer by *clicking* the **Transfer** button *after* the measurement.



Please click it, and the *data file(s)* from *all MU(s)* that we just *used*, will be **downloaded** to the *client* for you. A **transfer box** appears to show the *progress*, and eventual *completion* of the download:



In this case we only had *one MU* named '**WEBSTER**', so *only one* file needed to be downloaded.

Note: This is a software option, which must meet **DEWESoft NET Requirements**.

3 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*
- **Store settings and events**

2. STEP

Publishing the data

printout of instruments, copy Channel setup and Display elements to Clipboard

3. 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 **enter** the **DEWESoft Analyse** mode press the  - **Analyse** button on main tool bar.

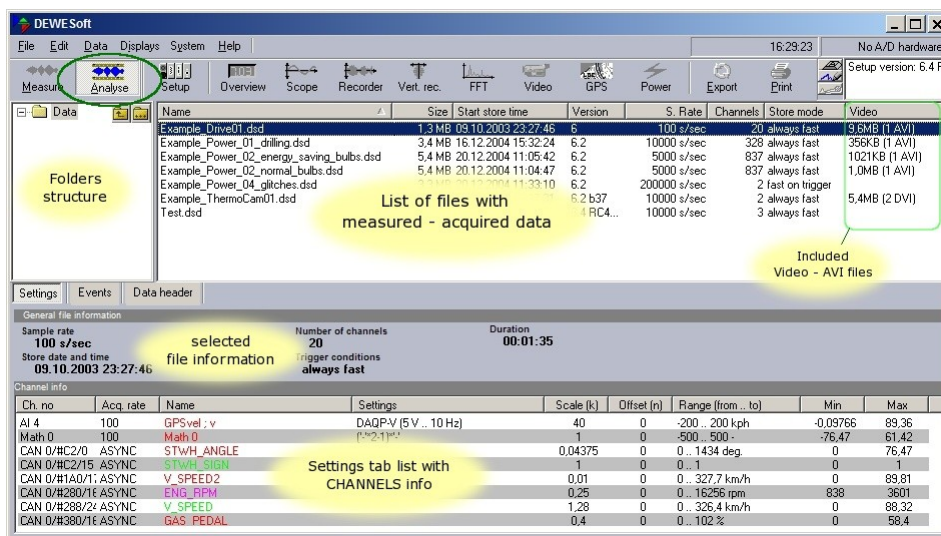
3.1 Reviewing data files

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

3.1.1 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](#) → **File Name Setup**

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*:

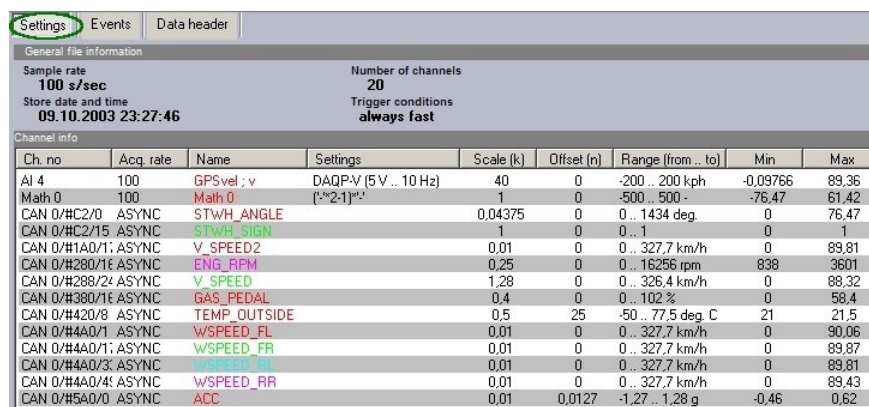
Name	Name of the file
Size	Size of the file in kB (= 1024 Bytes)
Start store time	Date and time when the file has been modified
Version	The <i>version</i> of DEWESoft used to acquire the data file
Sample rate	The used <i>sampling rate</i> ; it will also show the reduced rate if the data was stored slow and fast on trigger
Channels	Number of active channels
Store mode	One of the four <i>storage modes</i> : 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)

Name	Size	Start store time	Version	S. Rate	Channels	Store mode	Video
Example_Drive01.dsd	1,3 MB	09.10.2003 23:27:46	6	100 s/sec	20	always fast	9,6MB (1 AVI)
Example_Power_01_drilling.dsd	3,4 MB	16.12.2004 15:32:24	6.2	10000 s/sec	328	always fast	356KB (1 AVI)
Example_Power_02_energy_saving_b...	5,4 MB	20.12.2004 11:05:42	6.2	5000 s/sec	837	always fast	1021KB (1 AVI)
Example_Power_02_normal_bulbs.dsd	5,4 MB	20.12.2004 11:04:47	6.2	5000 s/sec	837	always fast	1,0MB (1 AVI)
Example_Power_04_glitches.dsd	3,3 MB	20.12.2004 11:33:10	6.2	200000 s/sec	2	fast on trigger	
Example_ThermoCam01.dsd	115,3 kB	17.06.2004 13:37:31	6.2 b37	10000 s/sec	2	always fast	5,4MB (2 DVI)
Test.dsd	89,4 MB	04.12.2006 17:29:52	6.4 RC4...	10000 s/sec	3	always fast	

NOTE: *DEWESoft is not able to analyze data from old versions below 6.0.*

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*.



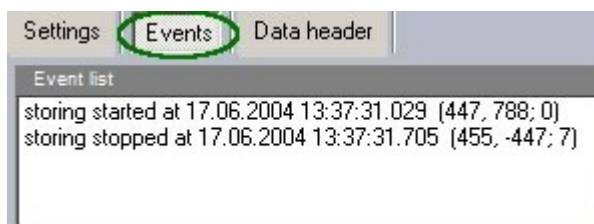
General file information								
Sample rate				Number of channels				
100 s/sec				20				
Store date and time				Trigger conditions				
09.10.2003 23:27:46				always fast				
Channel info								
Ch. no	Acq. rate	Name	Settings	Scale (k)	Offset (n)	Range (from...to)	Min	Max
AI 4	100	GPSvel ; v	DAQPV (5 V .. 10 Hz)	40	0	-200 .. 200 kph	-0,09766	89,36
Math 0	100	Math 0	[*2-1]"	1	0	-500 .. 500	-76,47	61,42
CAN 0/#C2/0	ASYN	STWH_ANGLE		0,04375	0	0 .. 1434 deg.	0	76,47
CAN 0/#C2/15	ASYN	STWH_SIGN		1	0	0 .. 1	0	1
CAN 0/#1A0/1	ASYN	V_SPEED2		0,01	0	0 .. 327,7 km/h	0	89,81
CAN 0/#280/1E	ASYN	ENG_RPM		0,25	0	0 .. 16256 rpm	838	3601
CAN 0/#288/2E	ASYN	V_SPEED		1,28	0	0 .. 326,4 km/h	0	88,32
CAN 0/#380/1E	ASYN	GAS_PEDAL		0,4	0	0 .. 102 %	0	58,4
CAN 0/#420/8	ASYN	TEMP_OUTSIDE		0,5	25	-50 .. 77,5 deg. C	21	21,5
CAN 0/#4A0/1	ASYN	WSPEED_FL		0,01	0	0 .. 327,7 km/h	0	90,06
CAN 0/#4A0/1	ASYN	WSPEED_FR		0,01	0	0 .. 327,7 km/h	0	89,87
CAN 0/#4A0/3	ASYN	WSPEED_RL		0,01	0	0 .. 327,7 km/h	0	89,81
CAN 0/#4A0/4	ASYN	WSPEED_RR		0,01	0	0 .. 327,7 km/h	0	89,43
CAN 0/#5A0/0	ASYN	ACC		0,01	0,0127	-1,27 .. 1,28 g	-0,46	0,62

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 <i>number</i>
Acq. rate	Acquisition rate will show the <i>acquisition rate</i> of <i>each channel</i> . 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 ASYN with old file versions)
Name	Channel <i>name</i> and <i>color</i> of the channel
Settings	<i>Type</i> , <i>input range</i> and <i>filter range</i> of the amplifier. For newer amplifiers also the <i>serial number</i> is mentioned
Scale (k)	<i>Scaling factor k</i> ($y = kx + n$)
Offset (n)	<i>Scaling offset n</i> ($y = kx + n$)
Range (from..to)	<i>Scaled input range</i> (min. to max. value)
Min	<i>Minimum value</i> within the whole file for the channel
Max	<i>Maximum value</i> 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.

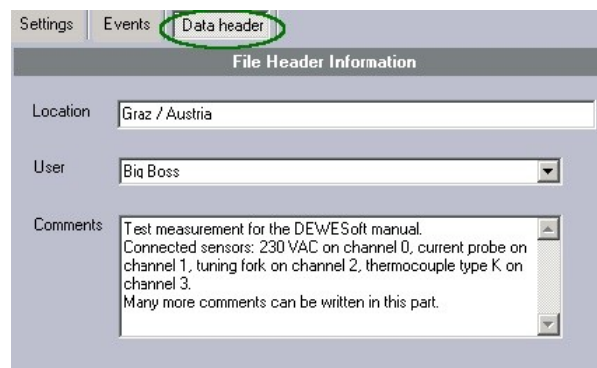


Event list	
storing started at	17.06.2004 13:37:31.029 (447, 788; 0)
storing stopped at	17.06.2004 13:37:31.705 (455, -447; 7)

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 → **File settings**.

To continue with **Analyse** mode the  - **Analyse** button on main tool bar must be selected *again*.

3.1.2 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*.

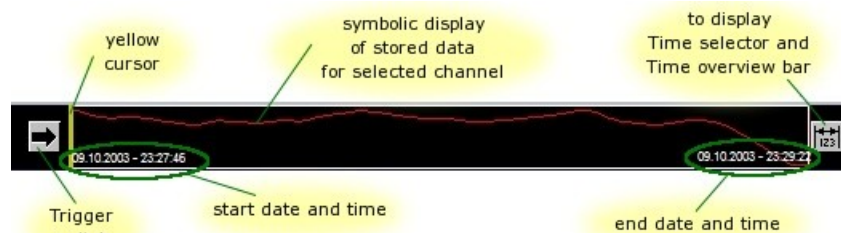
The **display settings** can be done in **Measure** and **Analyse** mode. You can use in **Analyse** mode all of the *features* available in the **Measure** mode: to *add / delete graphs* and *inputs* from each graph, *change* the *T-axis* and *Y-axis scales*, and more. If we *alter* the *display settings* 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 **Channel selector** see → **CHANNEL 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 .

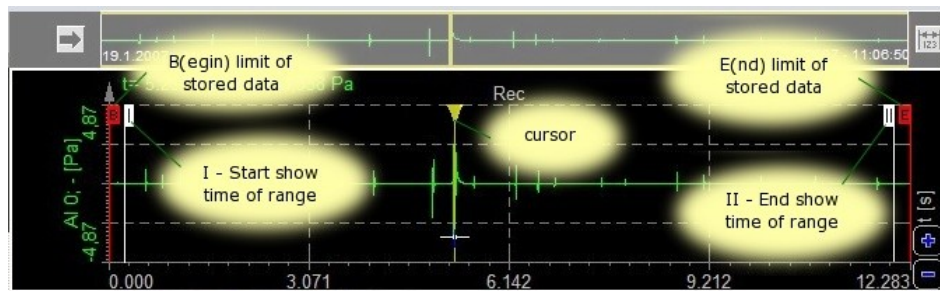
3.1.3 Selecting data

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*

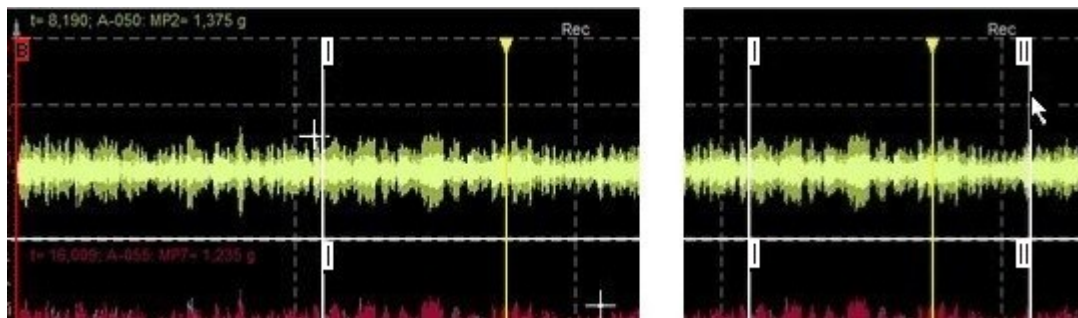
Selection range of stored data

1. Measurement cursor

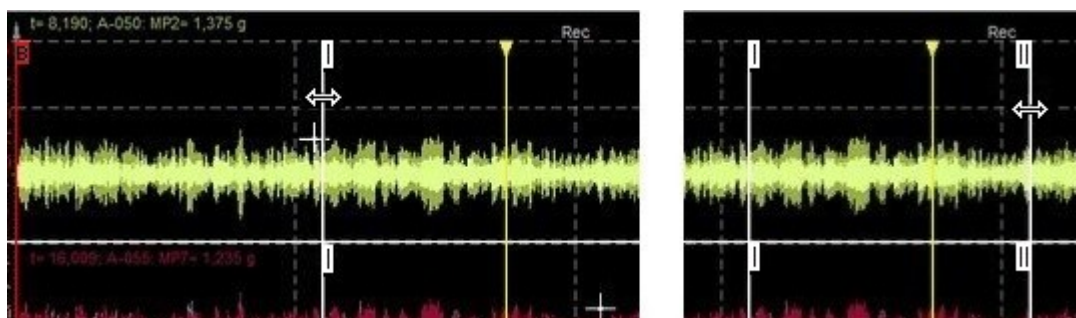


The *recorder* and *vertical recorder* offer two **measurement cursors** for the active graph on top labeled with **I** for *Start show time* and **II** for *End show time*. You can *drag* these cursors to **select a certain region** in two ways:

- *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 with him) to the *second* position of interest

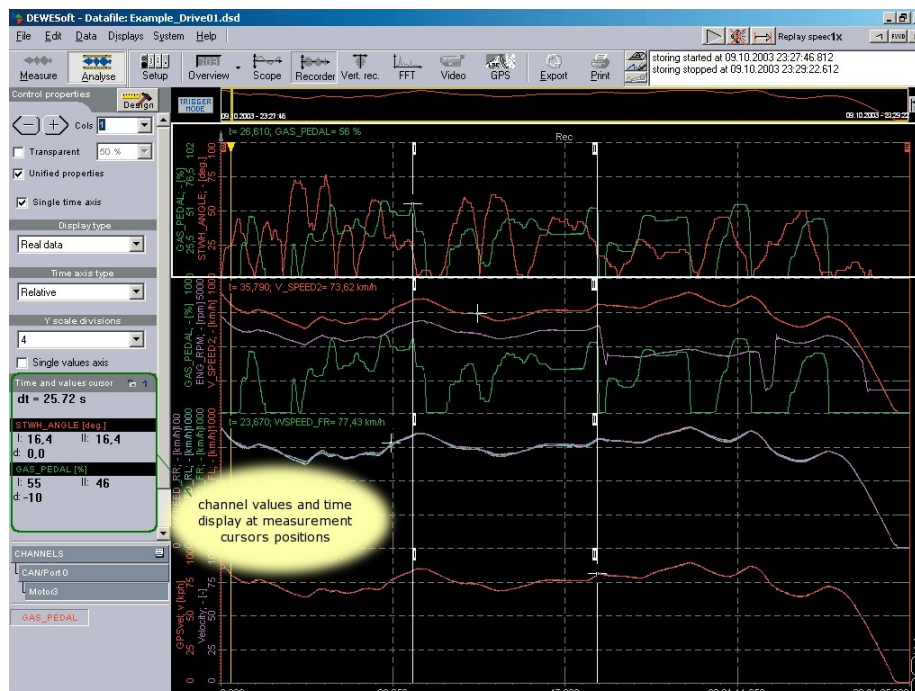


- *drag* the *first* cursor **I** from *leftmost* side of the graph to the *position* of interest and the *second* cursor **II** from the *rightmost* side of the graph (when mouse point on vertical line of measurement cursor mouse cursor change to two-sided arrow which indicate feasibility to move measurement cursor)



It's no matter in *which* recorder you use the cursor; it will *automatically* move in all displayed recorders and *vertical* recorders *simultaneously*.

On the left part of the display, the **readout values** of the currently selected graph *at cursor position* will be displayed. To show values from another graph, simply click on it and the values will change. As there is *only one* time base available, the cursor position is the *same* for all displayed graphs:



As we can see above, in **Analyse** mode on left part of screen below *recorder* settings (same also by *vertical recorder*) appear new section to **display** readout **values of signals** and **time** at cursor position - **Time and values cursor** with icon to **lock cursor** (see also **Lock Cursor 1 to selected position**).

Time and values cursor	
dt = 3,64 s	
Lock cursor button	
WSPEED_FL [km/h]	
l: 75,50	ll: 84,81
d: 9,31	
WSPEED_FR [km/h]	
l: 75,87	ll: 85,00
d: 9,13	
WSPEED_RL [km/h]	
l: 74,87	ll: 84,31
d: 9,44	
WSPEED_RR [km/h]	
l: 75,81	ll: 84,87
d: 9,06	

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 $\frac{1}{25}$ 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 $\frac{3}{4}$ 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.

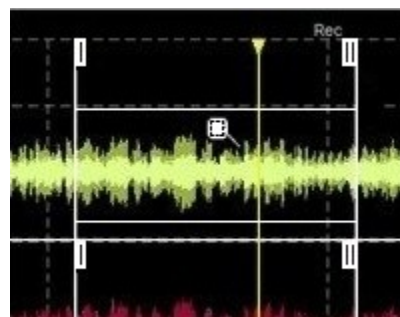
You can do that action *several* times to see the information of real interest.

3. Window zoom

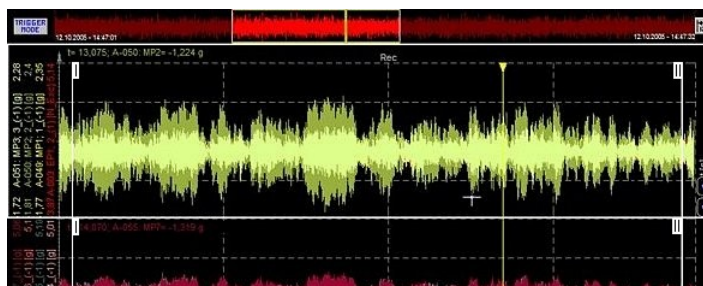
Additionally to the described method you can also **zoom in both axes** using the *window zoom* (also known as *boxed zoom*, because you open a 'box'). To do that *place* the mouse cursor at the *first* point (normally the upper left corner of the area of interest), press the <SHIFT> key and keep it 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.



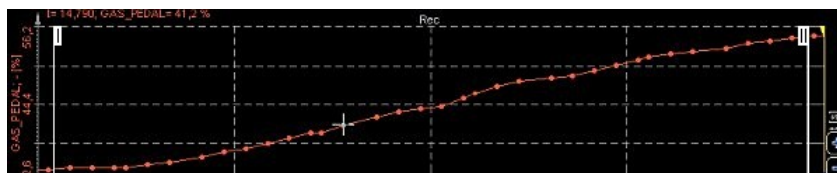
Release the mouse button (and afterwards also the <SHIFT> key) to zoom now into the selected window:



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.

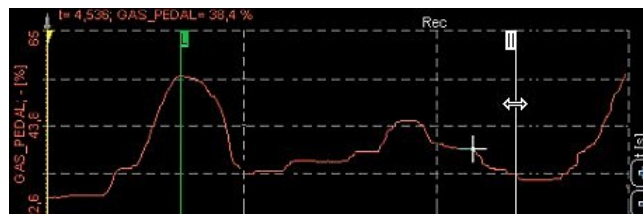


Lock Cursor 1 to selected position

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 **L** (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 **L** disappears and **cursor 1** is again available for *new* locks.

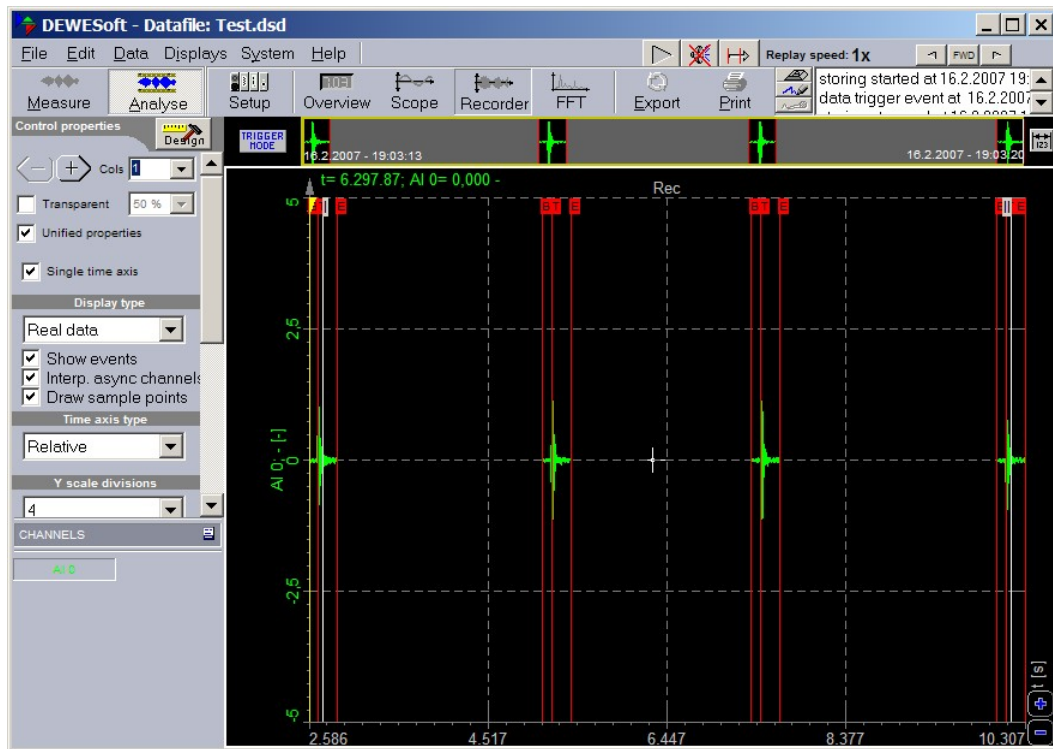
3.1.4 Selecting triggered data

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:

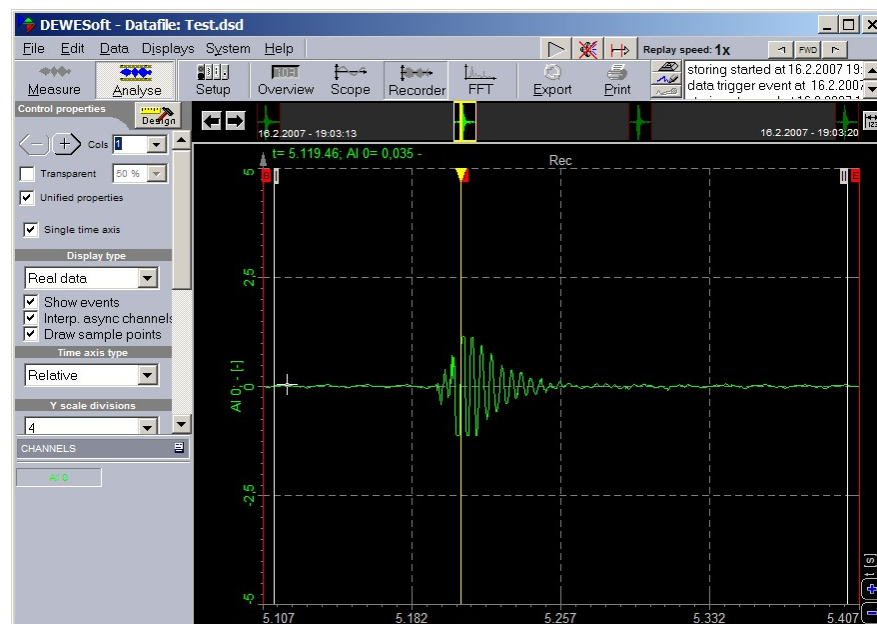
- store **only the portion** of interest by choosing '**fast on trigger**' storing option
- acquire data with **two speeds** (to have reduced data *also* for the regions *without* trigger event) - we need to use a different strategy '**fast on trigger, slow otherwise**'.

'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.



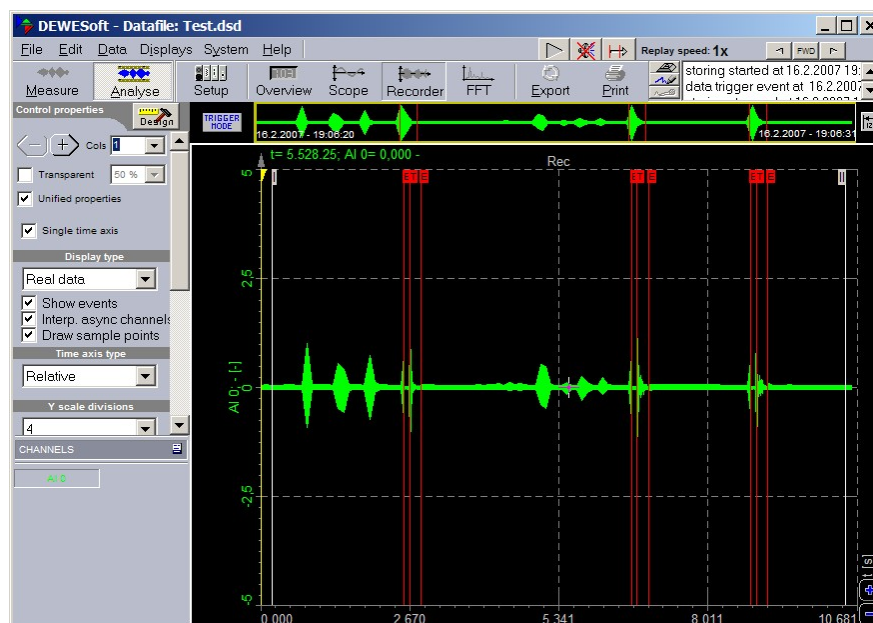
To display each event *separately*, just press the **TRIGGER MODE** button and after that use the   arrows to **move forward** and **backward** between the events - select event. On *first* trigger only  arrow is available - to navigate *forward* to second trigger and on last trigger only  arrow is available - 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](#).

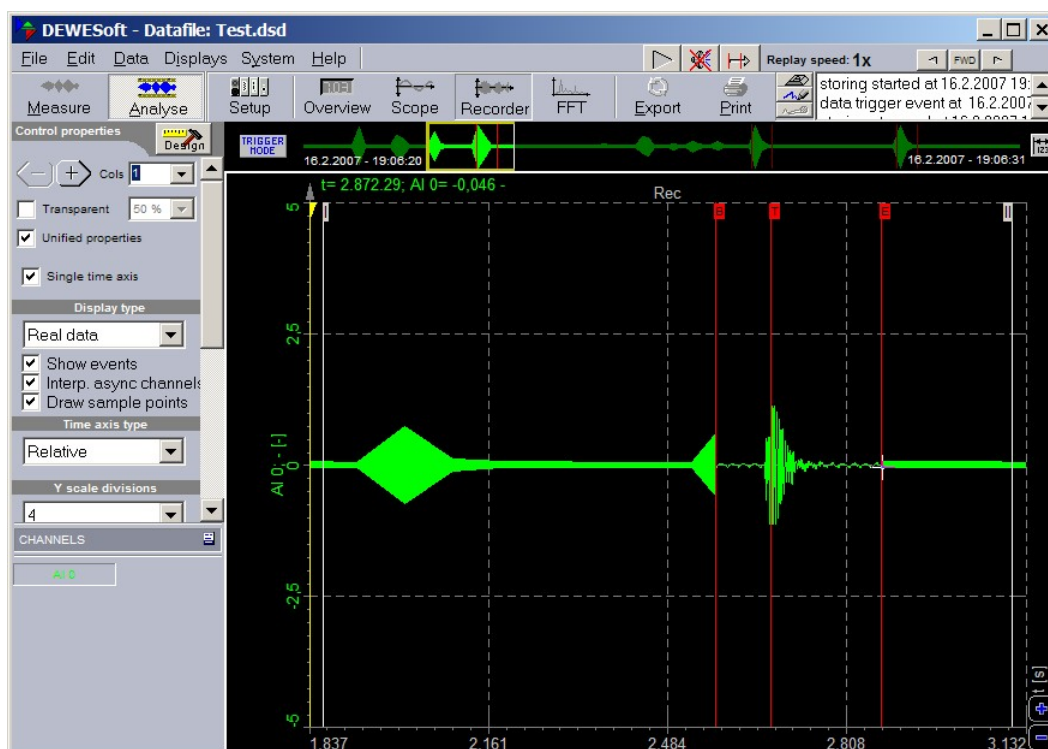
'fast on trigger, slow otherwise' triggered data file

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.



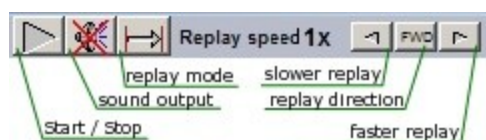
All other **Analyse** action and procedure on *triggered* signals are *same* as described → [Reviewing data files](#).

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.



3.1.5 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*

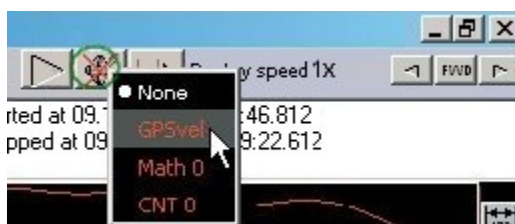
Start / Stop replay

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.

3.1.6 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

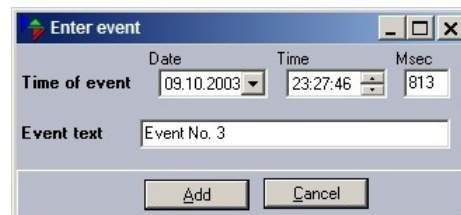
for information about **marking recording data with events** see → [User Guide](#) → **Manually Start / Stop Recording**

Manage events

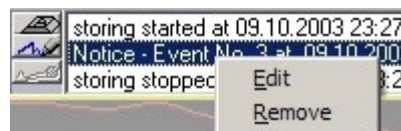
As mentioned before, all **events** are *displayed* in the *recorder*.

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.

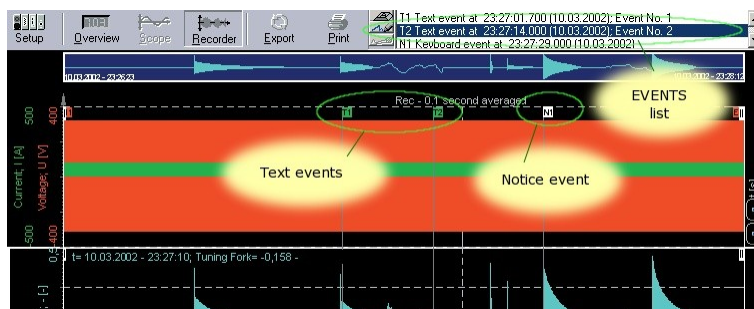


To **change** previous entered notice *right click* on this notice in the list, from displayed menu choose **Edit** and **Enter event** window will pop up. In this window now **Add** change to **Update** button, with which we accept changes to carry out in **Event text** field.



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*.



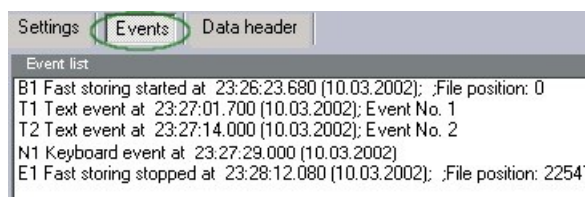
After all your new events are defined, select **Store settings and events** from the **Data** main menu (see also → [User Guide](#) → [Settings and Events](#)). All events will now be **stored** into the already *open* data file.

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**:



3.1.7 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 aware to have the same channels to avoid any errors or misinterpretations.*

3.1.8 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.
In most cases this will be **C:\Dewetron\Program\DEWESoft65\Data**
2. Please **rename** video file to this specification: **xxxxx.cam0.avi**,
where 'xxxxx' is *name* of **DEWESoft** file to be synchronized. If we have more video files, we can name then **xxxx.cam0.avi**, **xxxx.cam1.avi** and so on.



The picture above shows the example how this looks like. Original **DEWESoft** file is **test.dat** and therefore a video clip should be named **test.cam0.avi**.

3. After this open **DEWESoft**, press the **Analyse** button to enter the **Analyse** mode and you should see already *video* file as part of **DEWESoft** data file.

Name	Size	Start store time	Version	S. Rate	Channels	Store mode	Video
Testdsd	2,6 MB	7.4.2004 11:09:16	6.1b13	10000 s/sec	5	always fast	56,4MB

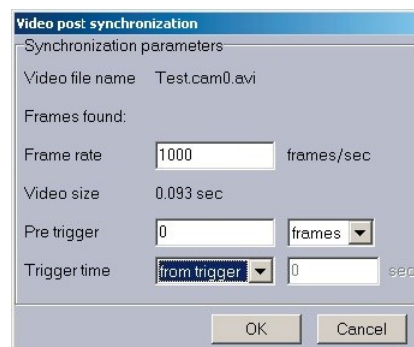
Synchronization settings

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**).



The dialog box titled "Video post synchronization" contains the following fields and controls:

- Synchronization parameters** (Section Header)
- Video file name**: Test.cam0.avi
- Frames found**: (Label)
- Frame rate**: 1000 frames/sec
- Video size**: 0.093 sec
- Pre trigger**: 0 frames (dropdown menu)
- Trigger time**: from trigger (dropdown menu), 0 sec
- OK** and **Cancel** buttons at the bottom right.

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.

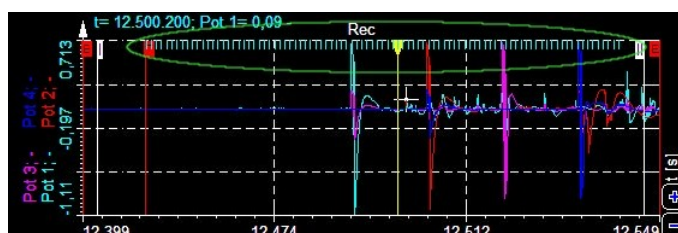
The second way is to define video start **from relative time**. In this case, the mentioned time *counts* from the *beginning* of measurement in **seconds**.

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. You will also see **frame ticks** at the top of the *recorder*: these marks *indicate* the **exact assigned time position** of each *frame* (*zoom in* for precise view). To *show / hide* this markers press <CTRL>+<ALT>+<T>.



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*.

3.2 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

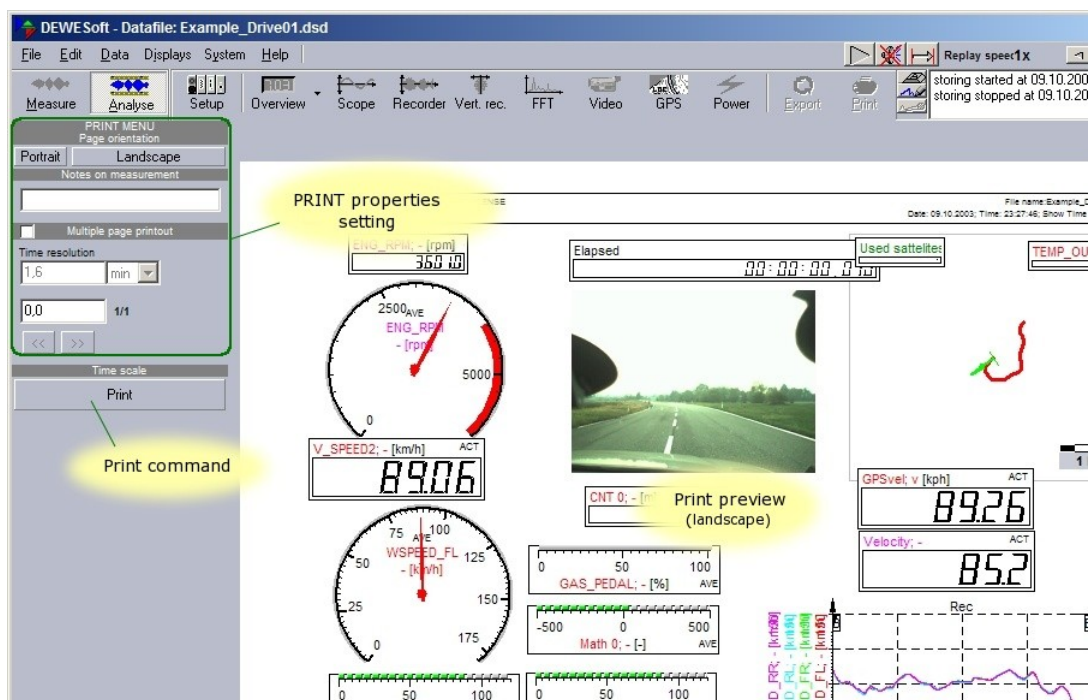
3.2.1 Screen printout

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



- **Print** button.

The following screen will appear:

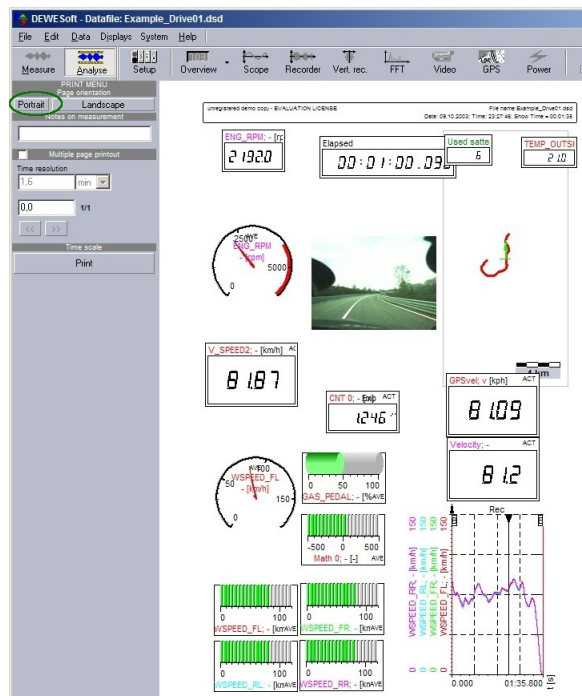


To **leave** the *printout* window simply **click** on any other instrument to change directly, for example the recorder.

On **PRINT MENU** can be set different **Print properties**:

Page orientation

Select between **Landscape** (see above) and **Portrait** format:



Notes on measurement and page header

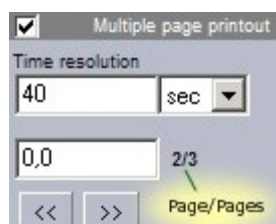
In **Notes on measurement** field enter a **comment**:



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 $0 \div 40\text{sec}$, 2nd for $40 \div 80\text{sec}$ and 3rd for $80 \div 120\text{sec}$.

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 $0 \div 40\text{sec}$, 2nd for $25 \div 65\text{sec}$, 3rd for $50 \div 90\text{sec}$ and 4th for $75 \div 115\text{sec}$.

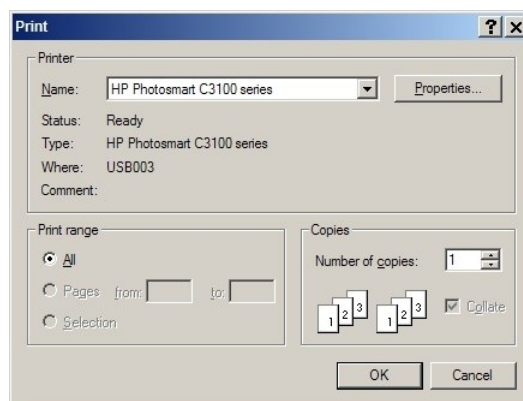
We can *display preview* of every page in multi page printout → use **>>** button for navigation *up* and **<<** button for navigation *down* between pages.

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.

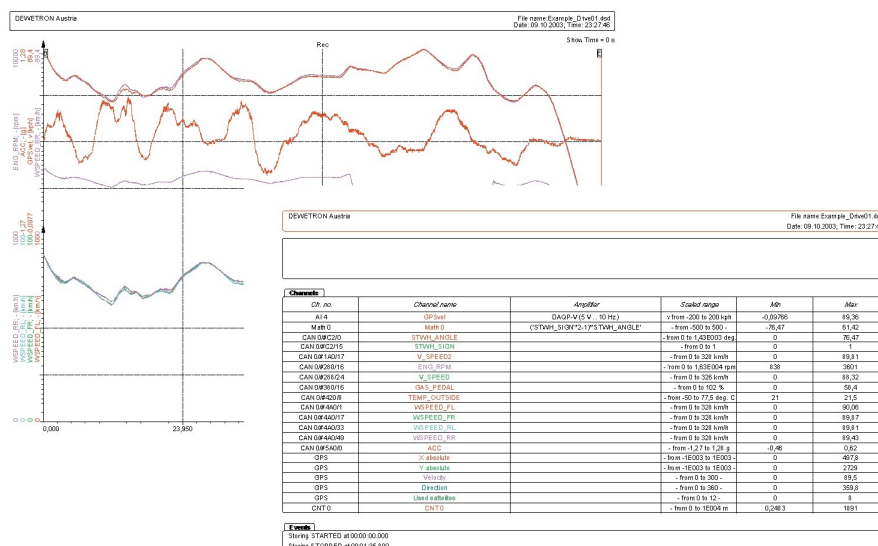
DEWESoft uses the complete *Windows®* printer support.

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.



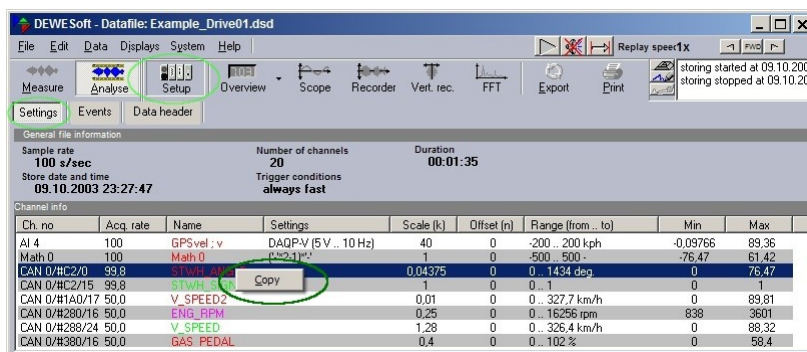
After starting the printout, you can start *immediately continue working* with **DEWESoft**, even if the printer is still running.

Example: typical **Setup** and **Recorder** printouts in landscape format:

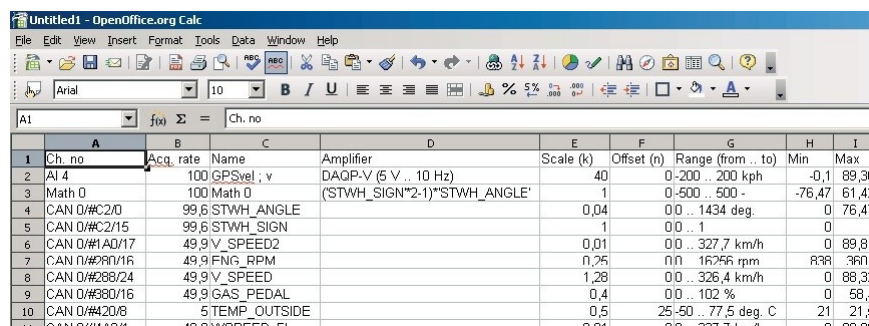


3.2.2 Copy channel setup

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.



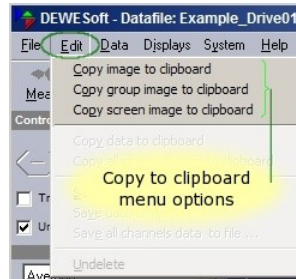
Now open and change to the other application and **paste** it.



NOTE: This feature is also available in the settings during file selection.

3.2.3 Copy visual elements

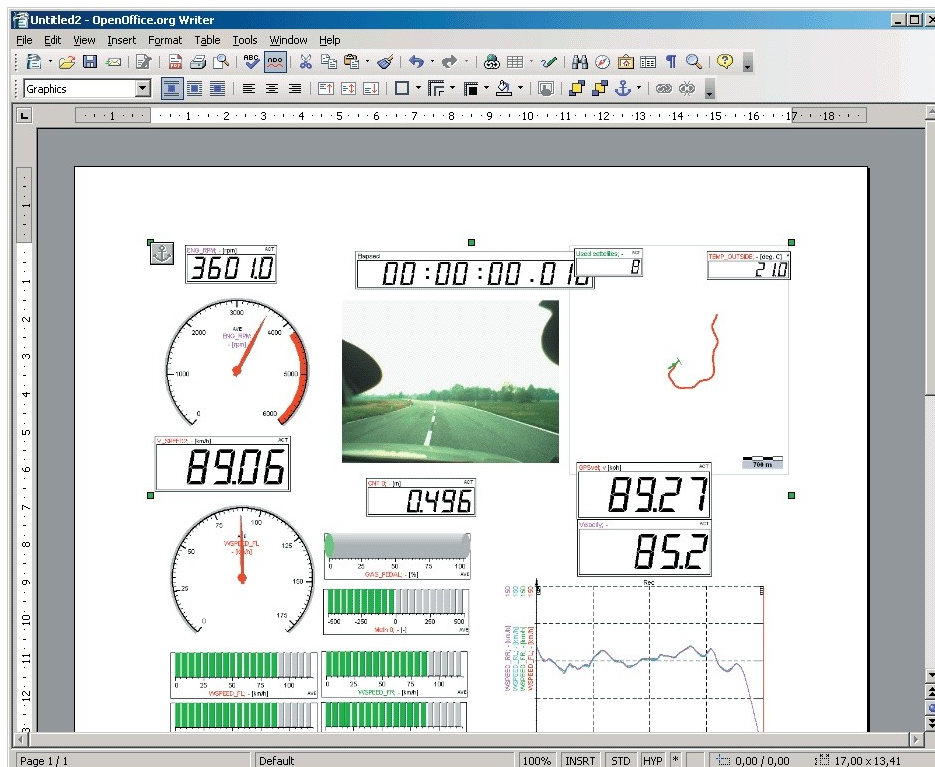
For documentation in *external* applications, you can **copy** the **display elements** also into the clipboard.



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



3.3 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**

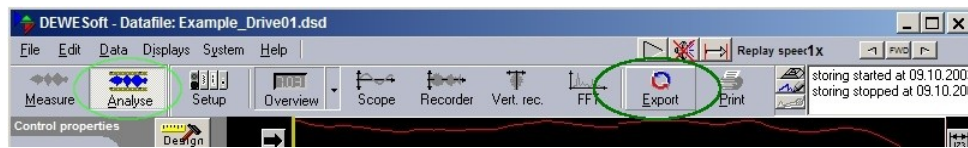
Time range of data

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.*

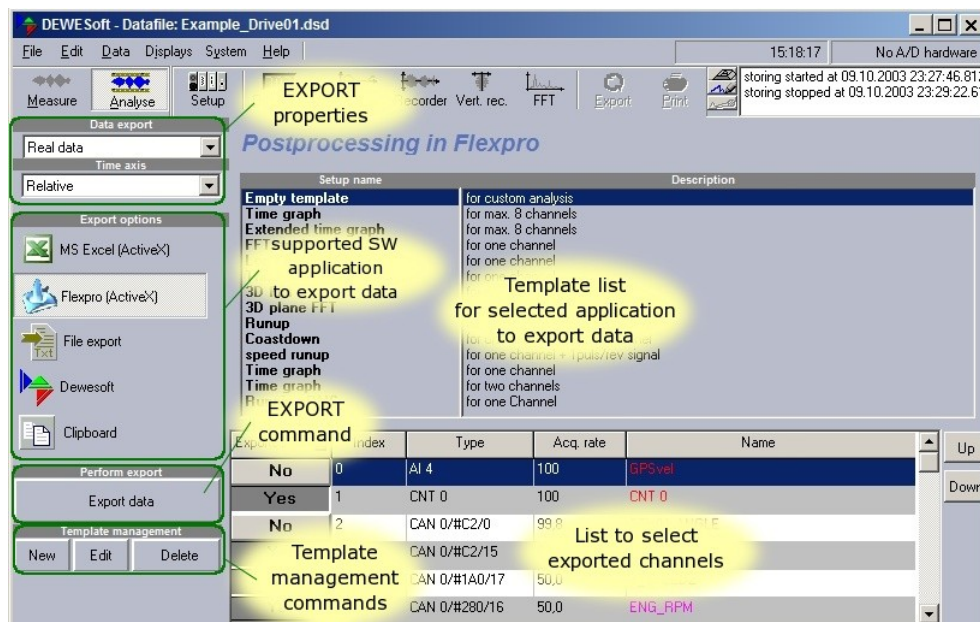
To **export** data, press the  - **Export** button on main **DEWESoft** tool bar, which is visible *only* when you are in the **Analyse** mode:



or select **Export** from the **Data** menu:



Once you press **Export**, the following screen will appear, ready as standard to export data directly into *FlexPro*:

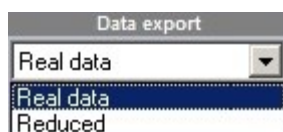


Set export data properties

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

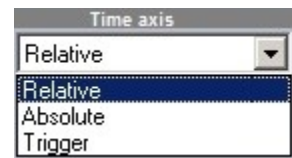
Select **type** of time scale from **Time axis** drop down list:

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).

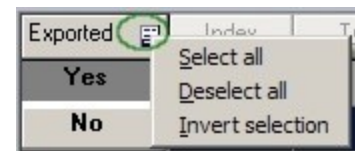


Exported channels

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  button *up* and on  button *down* within *Channels* list. This will change the *export order*.

The *export list options* will be stored when we choose **Store settings and events** from main **Data** menu.

Export option - formats

DEWESoft currently 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 *past*ing 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.

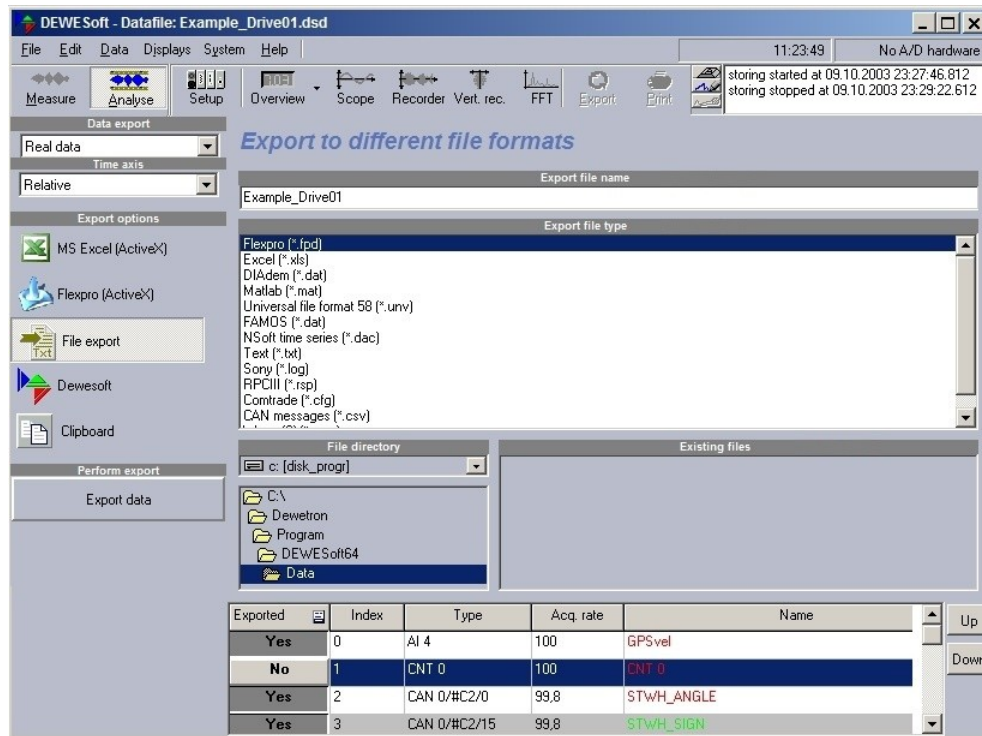
The other file formats except *FlexPro* and *MS Excel* just prompt you for a **filename**, which you supply using a standard Windows file dialog box. Click **OK** and the software will do the rest. Be sure to supply the proper filename **extension**.

File export

The supported data **file formats** are:

<i>FlexPro</i>	* .fpd	Powerful, easy-to-use data analysis software
<i>Microsoft Excel</i>	* .xls	Standard spreadsheet software (not useful for large amounts of data)
<i>DIAdem</i>	* .dat	Powerful data analysis package for automotive industry
<i>Matlab</i>	* .mat	Common analysis and mathematics package
<i>Universal file format 58</i>	* .unv	For import to 3-D modeling and structural analysis software from different vendors
<i>FAMOS</i>	* .dat	FAMOS file format export
<i>NSoft time series</i>	* .dac	NSoft file format
<i>Text File</i>	* .txt	Delimited ASCII text file
<i>Sony</i>	* .log	Sony DAT recorder data format
<i>RPCIII</i>	* .rsp	RPC III data format used for road load data analysis
<i>Comtrade</i>	* .cfg	Comtrade data format for power analysis
<i>Wave</i>	* .wav	Wave audio data format (available as custom export)
<i>BWF</i>	* .bwf	Multi channel audio data format (available as custom export)
<i>ATI</i>	* .ati	For direct import in iDEAS analysis package (available as custom export)
<i>SDF</i>	* .dat	For direct import in Prosig analysis package (available as custom export)
<i>WFT</i>	* .wft	Nicolet file format (available as custom export)
<i>Google earth</i>	* .kml	Export of GPS path to Google earth (available as custom export)
<i>CAN messages</i>	* .csv	tab delimited export of CAN messages for replay in demo mode

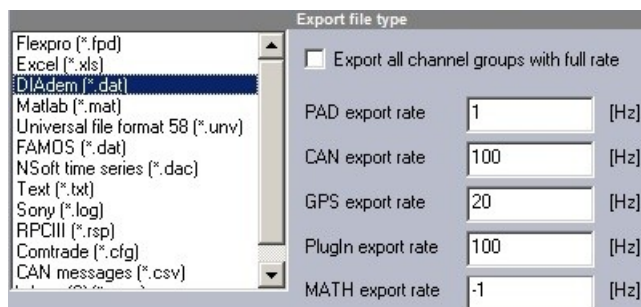
If you select the **File Export**, following display appears:



On central part following *Export to different file formats* sections is displayed:

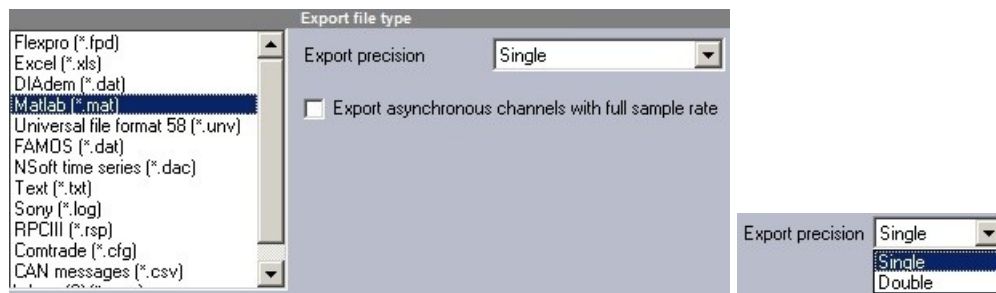
- Export file name** *name of exported file; enter name according to OS name convention*
- File directory** *name of directory where exported file to be saved; select with double click from drop down list*
- Existing files** *list of existing files same type in selected directory*
- Export file type** *list of available type exported file with optional file settings for:*

- *DIAdem* *.dat file type



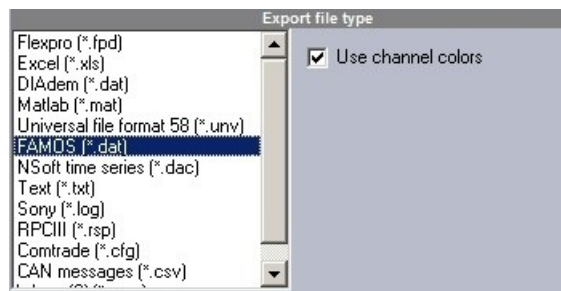
Diadem has an option to export certain groups of channels at specific sample rate, so the resulting data file is shorter. We can achieve this by de selecting **Export all channel groups with full rate** and defining the *rate* for each channel group (PAD, CAN, GPS, Plugin and MATH). If -1 is entered, the data will be exported at **full rate**.

- **Matlab** *.mat file type



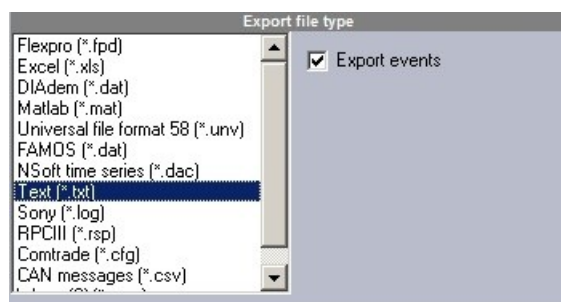
Export precision can be selected from drop down list between **Single** and **Double**. Usually single precision is enough, but sometimes Matlab functions expect data in double precision. Please be aware that the amount of exported data will be *doubled* if this option is chosen and Matlab files can't exceed 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



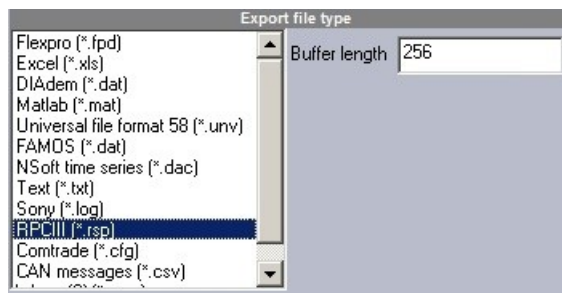
Use channel colors can be checked to use **DEWESoft** channel colors otherwise *default Famos* color is chosen.

- **Text** *.txt file type



With checked **Export events** also **events** will be *exported* in text file.

- *RPCIII* *.rsp file type

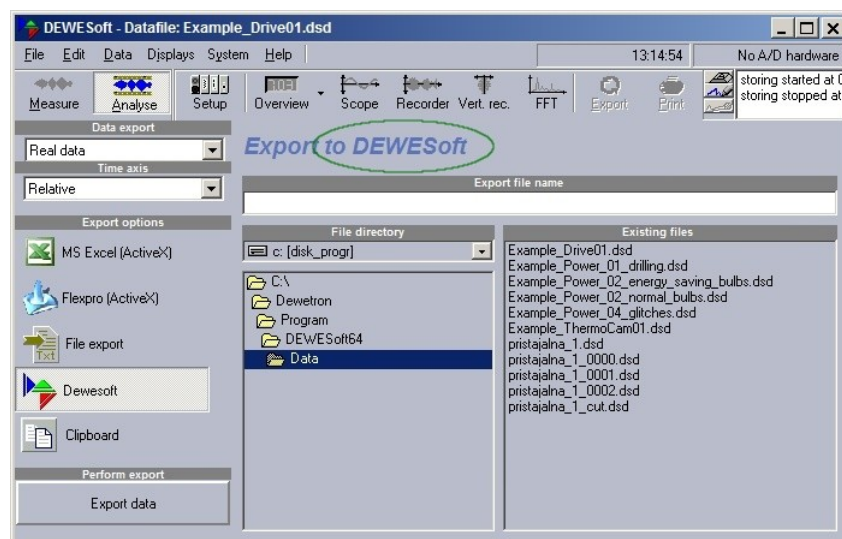


Buffer length field can be set to *match* the analysis capabilities. With *large* files *larger buffer length* is recommended. The buffer length should be multiple of 2, i.e. 256, 512, 1024 and so on.

NOTE: For all other file type Export file type section will be without any additional setting (see screen above - File export).

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:

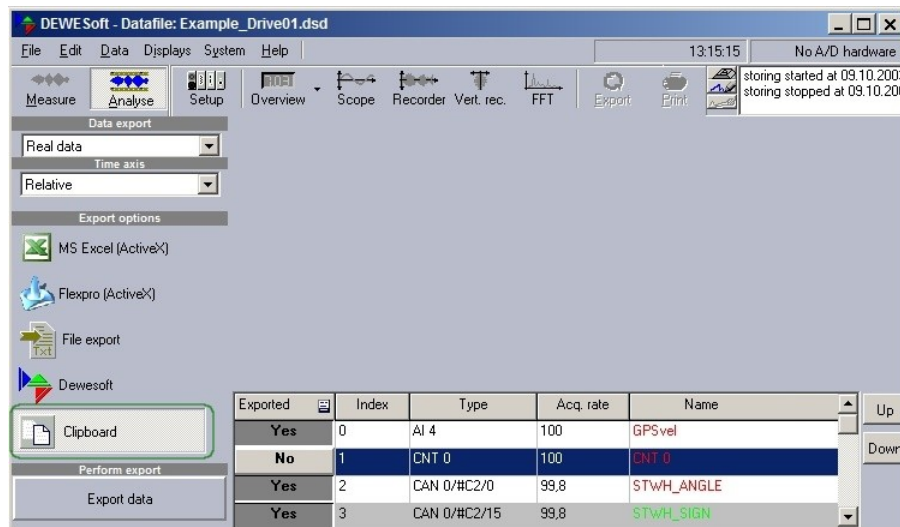
- Export file name** name of exported file; enter new file name according to OS name convention
- File directory** name of directory where exported file to be saved; select with double click from drop down list
- Existing files** list of existing files same type in selected directory

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).

3.3.1 to FlexPro and MS Excel

To enhance **export data** to *Microsoft Excel* and *FlexPro*, following procedures can be used:

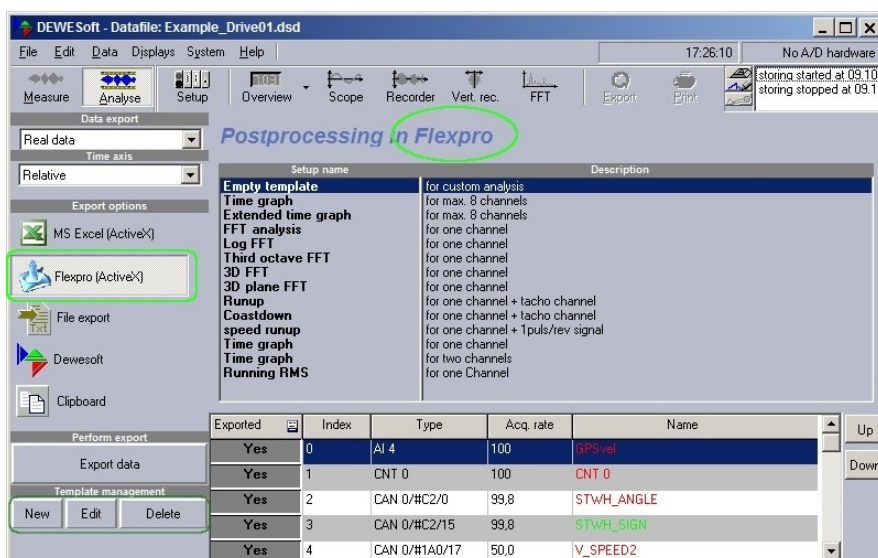
1. STEP
Select application click on either **FlexPro (ActiveX)** or **MS Excel (ActiveX)** on **Export option**
2. STEP
Template Management **Template Management controls** are only available for *Microsoft Excel* and *FlexPro* export option to manage **template - scripts**, which are listed on central **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 C:\Dewetron\Program\DEWESoft65\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.*

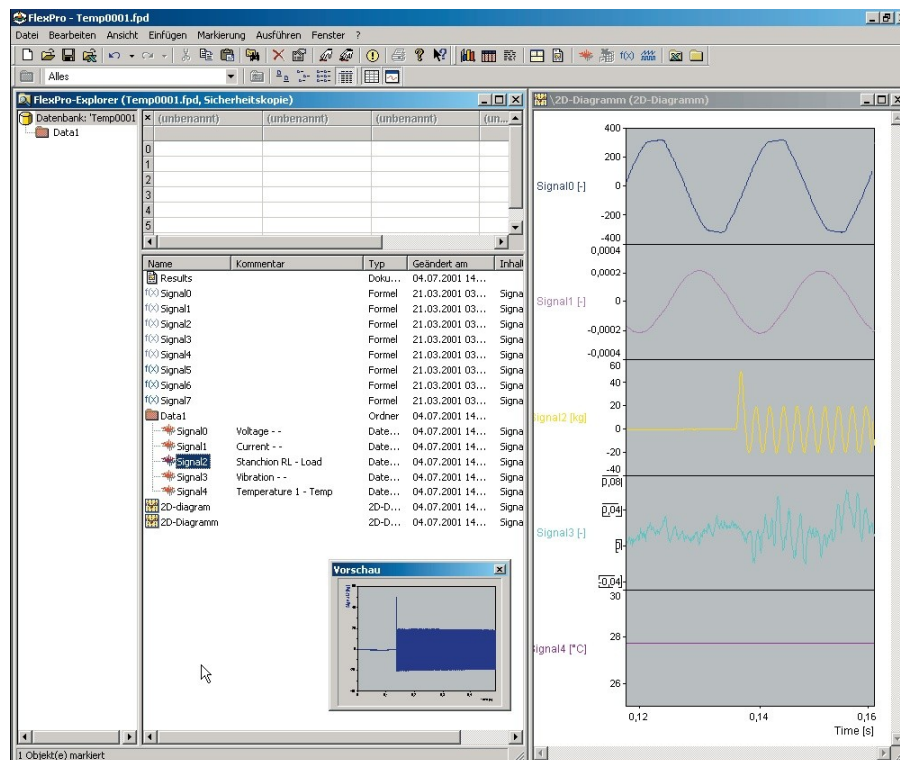
The *concept* is a nice one: once the script is created, just **load** any **data file** and **run** the **script**. What happens next is a big time-saver. **DEWESoft** runs this script:

- *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.

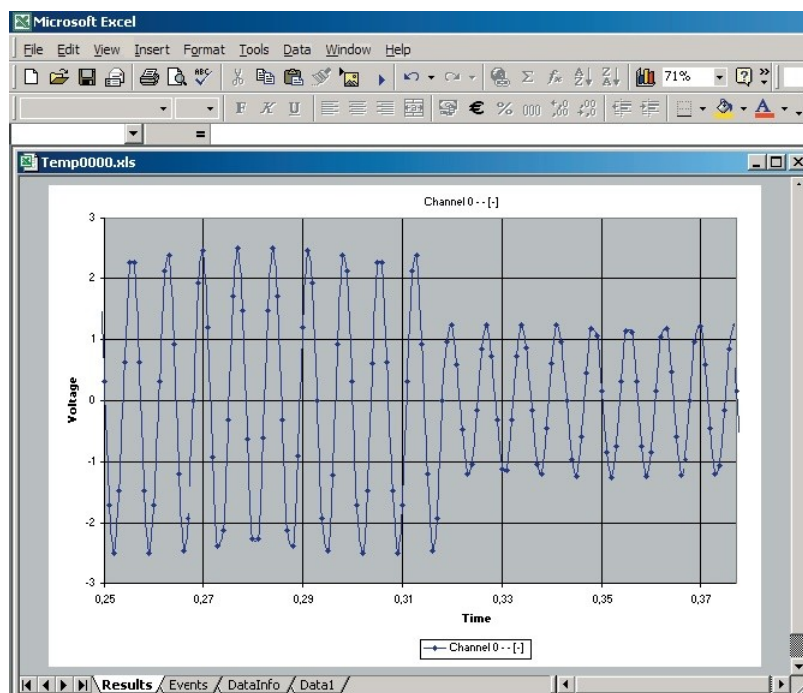
Let's say that you have a script that takes four incoming signals, runs 3rd octave analysis on them, creates a polished looking report, complete with your company's title bar and logo, automatically puts in the data and other variables from the data file, and formats it for your color printer. After 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

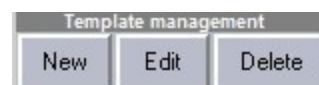


NOTE: our *MS Excel* export function knows about *Excel's* built-in limitation of 65536 rows per worksheet. If your data is longer than that, it simply creates **multiple**-worksheets within the workbook that it creates! It also makes a simple **chart** with the first several thousand data points for 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** button to *create* a *new* one or **Edit** to *change* an *existing* script.



To *delete* an *existing* script simply *select* it and press **Delete** button.

When you press the **New** 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**

3.3.1.1 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!*

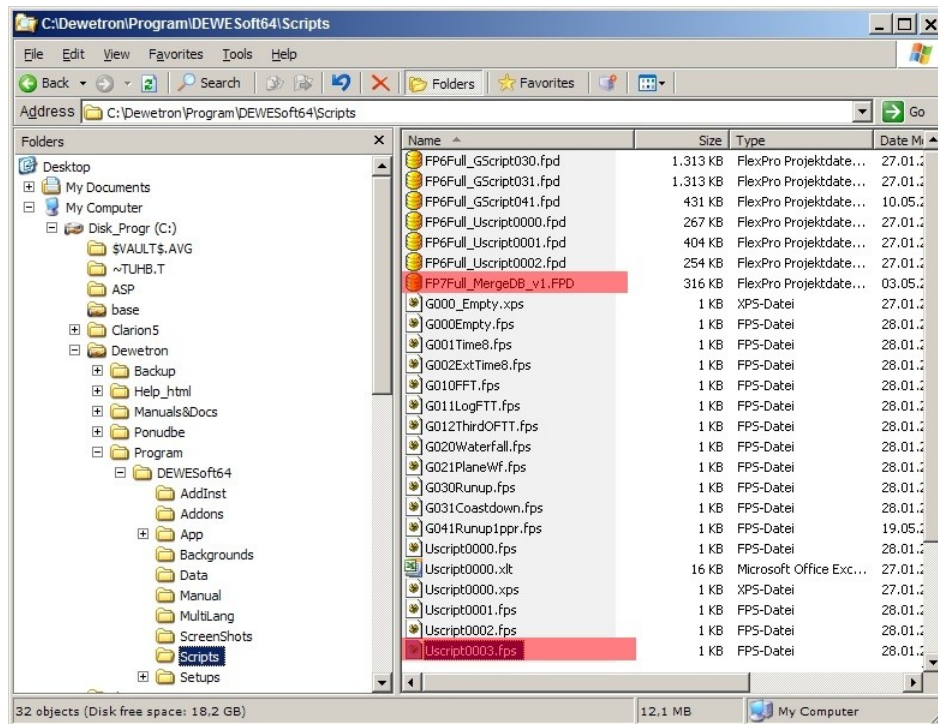
Installation

To make the *MergeDB-Script* available in the *DEWESoft export* screen, two files must be in the **DEWESoft** script folder (usually **C:\Dewetron\Program\DEWESoft65\Scripts**):

- Copy the file **FP7Full_MergeDB_v1.fpd** to the *DEWESoft script folder*
- Copy the file **Uscript000X.fps** to the *DEWESoft script folder*

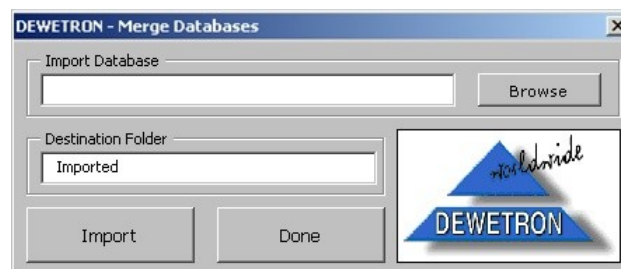
NOTE: *There are several Uscript-files in the DEWESoft script folder (e.g.: Uscript0001.fps, Uscript0002.fps ...). The X in the filename Uscript000X.fps must be replaced by a number that is not assigned yet! For example if the last file is called Uscript0002.fps, name the created file Uscript0003.fps!*

DEWESoft script folder with the files *needed* for MergeDB:



Usage of MergeDB

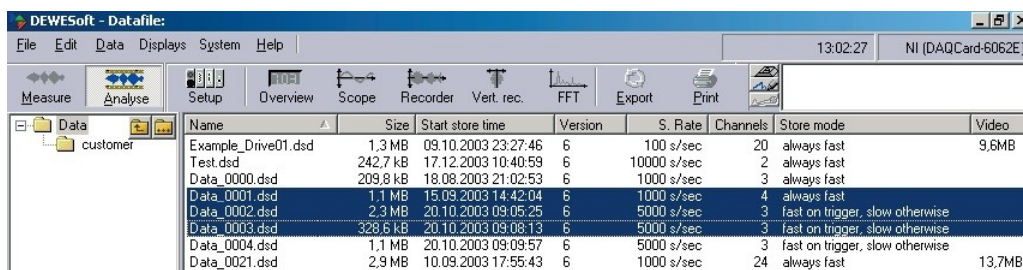
After selecting *MergeDB-Script* on the *DEWESoft export* screen **DEWETRON - Merge Database** window appear:



- 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*.

3.3.2 Export multiple files

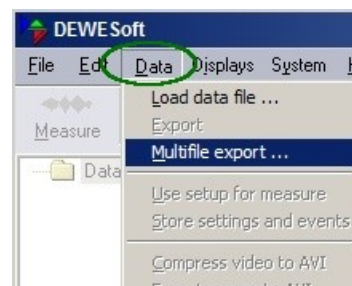
If you want to **export several files** at *once*, there is a nice function available. Press the **Analyse** button to enter the *file explorer*. *Select all files* you want to *export*.



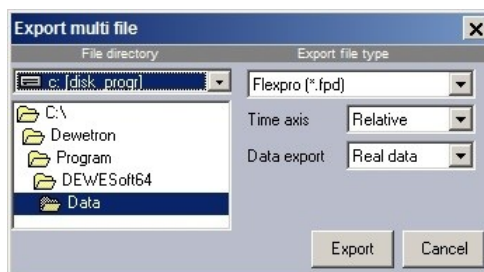
How can we *mark* several files? There are two possibilities:

- to mark several files which are listed near to each other, keep the SHIFT button pressed *during* selection
- to mark several files picked out of the list, keep the Ctrl button pressed *during* selection

After all files to export are *marked*, select **Multifile export** from the **Data** menu.



Now the **Export multi file** window appears.

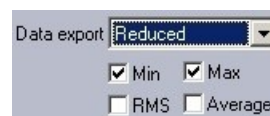


This window allows you to select:

- **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:



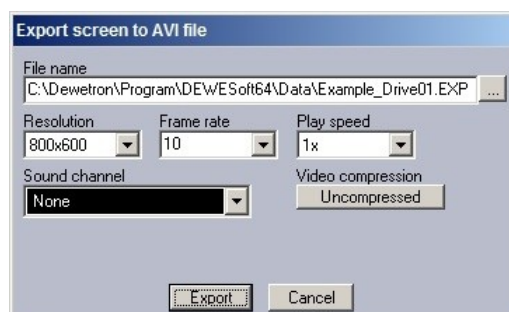
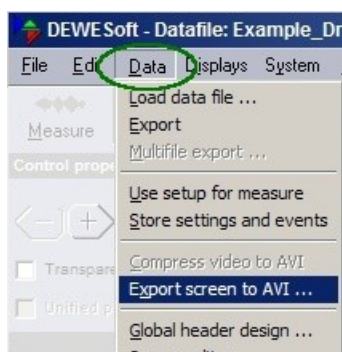
Press the **Export** button to start the *export* process or **Cancel** to leave *without* exporting the files.

NOTE: *The export keeps the original file names for the exported files.*

3.3.3 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*.

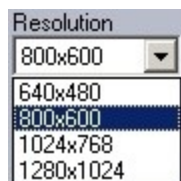
Select **Export screen to AVI** from the **Data** menu and the following **Export screen to AVI file** window will appear:



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 → **File Name Setup**

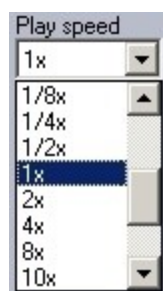
- **Resolution:** from 640 x 480 to 1280 x 1024 selectable from drop down list - high resolutions will create larger files!



- **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. 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**

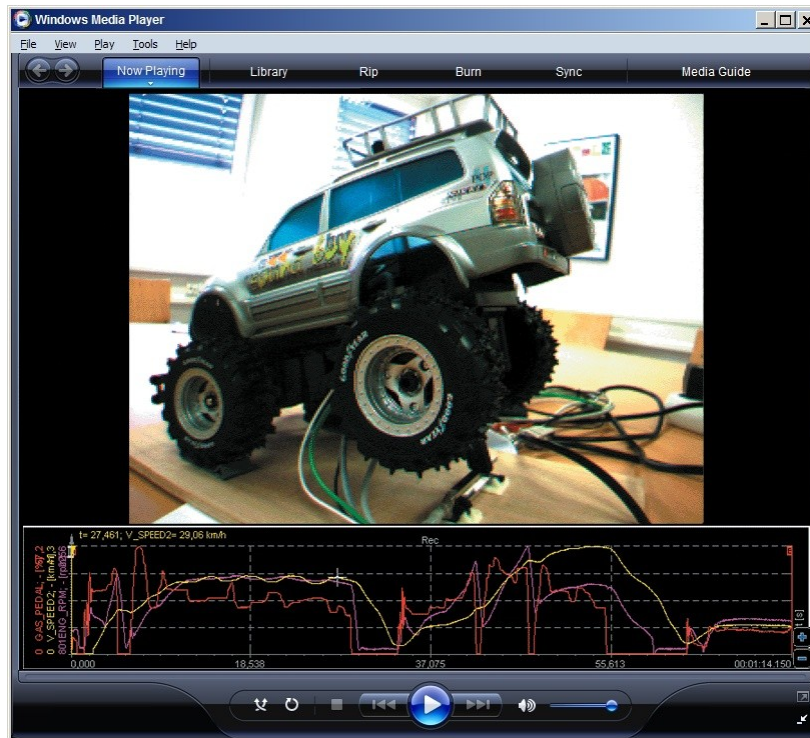
When you have done all changes press the **Export** button to start the *export* or **Cancel** to leave *without* exporting the video.

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*:



4 Data management

In **DEWESoft** there are many ways to **Import** (Open) or **Export** (Save) *setups*, *measured* and *other data*.

type of file	type of data	action
Setup data *. dss (DEWESoft setup data file) or *. txt (DEWESoft text setup file) - for detailed information about this file see → Structure of text setup file	Channel setup setting of all <i>analog channels</i> , <i>application modules</i> , instruments appearance, storing options, multifile setting ...	New Load and Load last Load display setup Save and Save As
Acquired data, settings and events *. dss (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
Video files *. dvi file and *. avi file	Video files online video data from <i>cameras</i> or <i>exported instrument screen</i> like <i>Overview</i> , <i>Scope</i> , <i>Recorder</i> ,...	
XML files sensors.xml , cntsensors.xml , modules.xml , CAengines.xml files	Sensors database, counter sensor database, amplifier transfer curves and typical combustion engines	Sensor database Counter sensor Engine templates

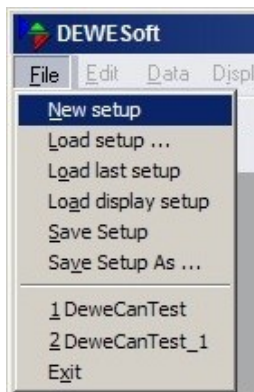
4.1 Setup data

In **DEWESoft** *.**dss** (DEWESoft **setup data**) file or *.**txt** (DEWESoft **text** setup) file Channel setup data and **DEWESoft** Application modules setup data are stored.

for detailed information about the *.**txt** (DEWESoft **text** setup) file see → **Structure of text setup file**

WARNING: DEWESoft setup data file does not include any measured data!

New Setup data



We can start with a **new Setup** by selecting:

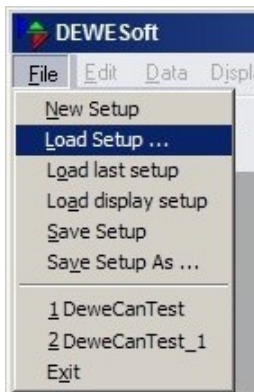
File menu → **New Setup**

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**

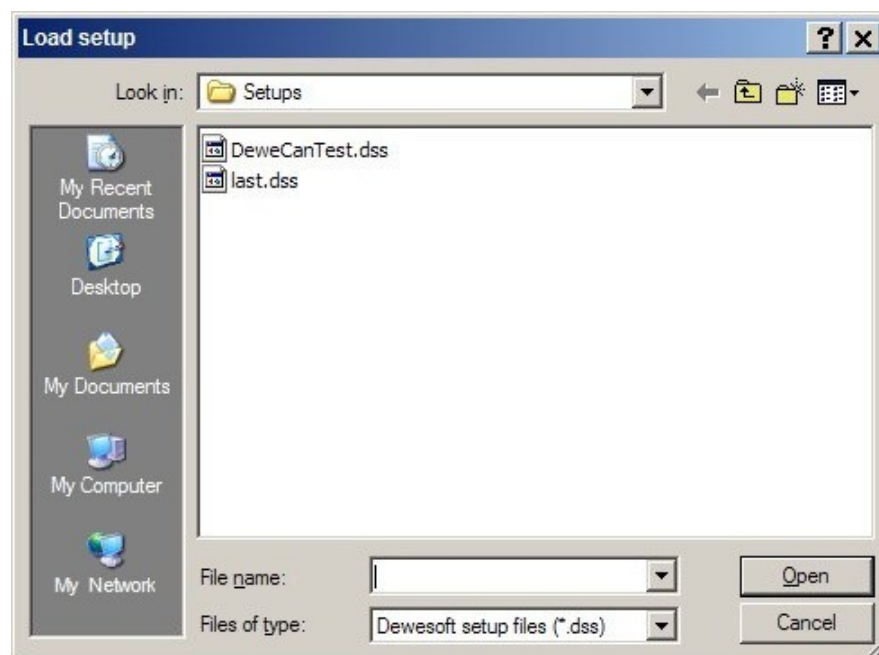
and *choose* from the file list,

or by click on:

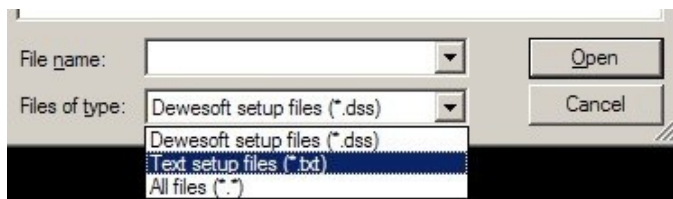
File menu → *name of recent project* on lower part of this menu

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 ***.dss** (DEWE Soft **setup data**) file or ***.txt** (DEWE Soft **text setup**) file:



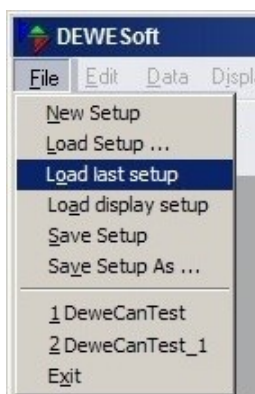
NOTE: *In order to see available text setups within the file dialog, it is necessary to switch the file type to Text setup files (*.txt).*



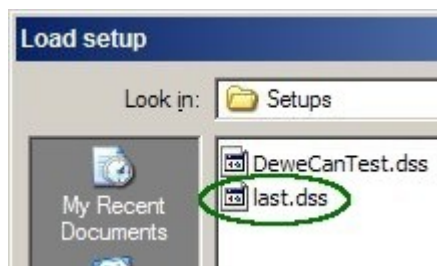
Load Last Setup data

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**

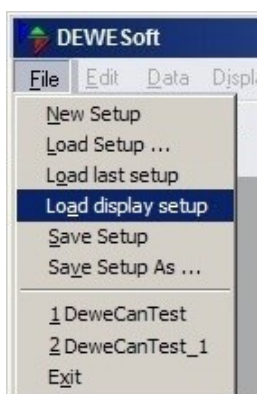


and look for a file called **last.dss**:



WARNING: *Be aware that this file will be overwritten immediately when you change anything or load another setup.*

Load display Setup data



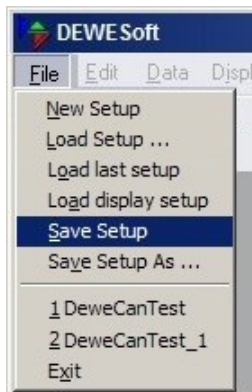
Load display setup options *keeps* the *channel settings*, but overwrites the instrument appearance with the appearance *found in the setup* file:

File menu → **Load Display Setup**

and the standard **Load setup** window (see above) is displayed to load existing **Display Setup** from *.dss (*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



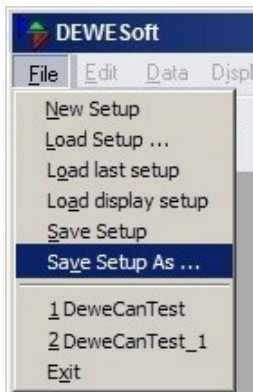
After setting all *channels* and *Application* modules data is reasonable to save this *new* or *changed* **Setup** by selecting:

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 ...



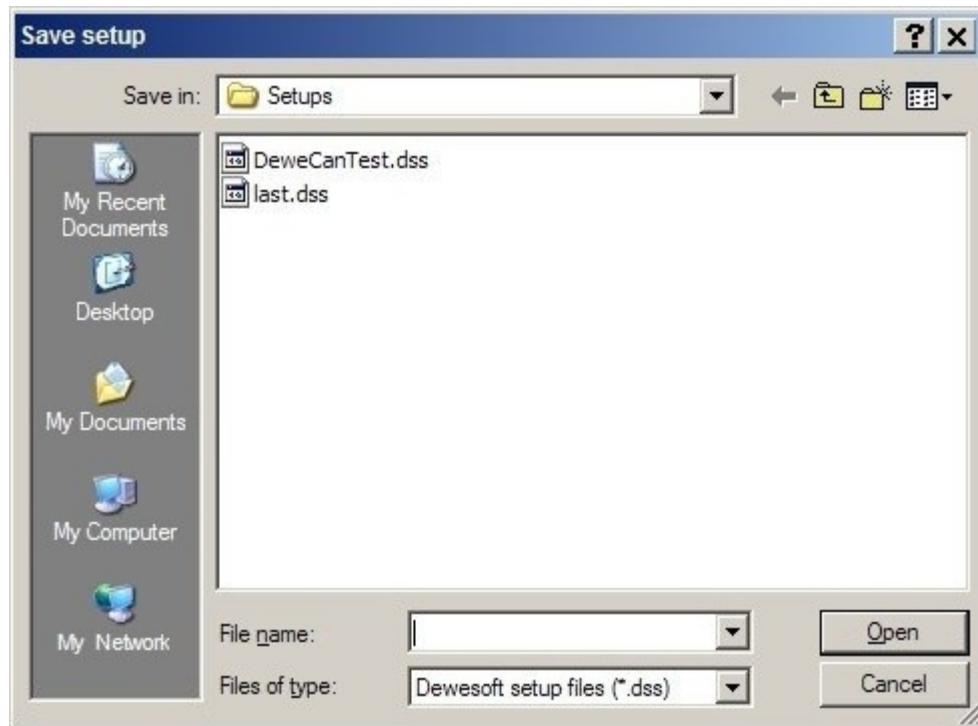
We can save *existing* (modified) **Setup** data under the **new name** by selecting:

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 *.dss (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 *.dss - 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 → [Reference Guide](#) → **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**

4.2 Structure of Text Setup file

The so-called **BigList** is a kind of **setup file** for configuring the channels of **DEWESoft**. It is also called '**Text setup**' and has the file-extension ***.txt**. It is an ASCII file which consists of *three* different sections, marked by the corresponding *start-* and *end-tags*. The *separator* between the elements within the file is the tabulator (#9).

A text setup can be loaded by **DEWESoft** similar to a *DEWESoft setup file* (*.dss).

see → [User Guide](#) → [Load Setup data](#)

NOTE: *It is not possible to save a text setup by means of DEWESoft. 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.*

Text setup file consists of *three* different sections:

- **General Information**
- **Channel List**
- **Sensor List** (optional)

In this section are also described **Module settings** and show **Example of Text Setup file**.

1. section: **General Information**

The general information section of the **Text setup file** (BigList) is surrounded by the following two *tags*: **<INFOLIST_V1>** and **</INFOLIST_V1>**. In between the following information can be provided:

SampleRate	- the analog <i>acquisition rate</i>
CAN0dbc	- the <i>path</i> of a DBC-file being applied to CAN port 0
CAN1dbc	- the <i>path</i> of a DBC-file being applied to CAN port 1
DisplaySetup	- the <i>path</i> of a display-setup-file

Next to the keyword SampleRate and the required **<Tab>** separator character, the *sample rate* can be provided in *samples per second*.

Similar, as mentioned above, *.dbc-file(s) or **display setups** can be *referenced* here. The reference can be an absolute file path including the **file name** or *just* the file name. If only the file name is provided, the file has to be located *within* the sub-directory Setups of the **DEWESoft** *installation* directory.

2. section: Channel List

The channel list section of the **Text setup file** (BigList) is surrounded by `<CHANNELIST_V1>` and `</CHANNELIST_V1>` tags. In between the settings of the different *channels* can be provided. The **order** of the channels for **exporting** to *different data formats* will be equal to the *order* of the channels within this list.

For each channel the following *parameters* can be specified:

Column Heading	Description	Comment
ID	is a consecutive unique index starting at 1	required
TYPE	specifies the kind of a channel, which can be AI (analog), PAD, CAN or Math	required
USERGROUP	specifies a user defined channel group	optional
CHANNEL	is the number corresponding to the channel-number of an analog channel or the number of a PAD-channel where the module number and the channel of the PAD-module have to be defined separated by ' _ '	required for AI-channels, for PAD-channels and CAN-channels; For CAN-channels it denotes the CAN-port
SHORT_NAME	is a short name for the channel, corresponding to the name-field of a channel in DEWESoft's channel-setup screen	
LONG_NAME	corresponds to the description of a channel for AI-, PAD- and CAN-channels. For Math-channels this parameter contains the mathematical formula of the channel	required for Math-channels; optional for others
SENSOR_ID	links the channel to a specific sensor defined within the sensor list having the same sensor-ID	optional
RANGE_MIN	is the minimum range value of the channel	optional
RANGE_MAX	is the maximum range value of the channel	optional
UNITS	is the physical unit of the channel	optional
SAMPLE_RATE	can be equal to the SampleRate specified in the INFOLIST-section or less in order to set a sample rate-divider	optional
FILTER	Can be either FILTBU for Butterworth or FILTBE for Bessel which can be set for certain modules. See also table 3.	optional
SCALE	Defines the scaling factor of a channel. Should be left empty, if a sensor of the sensor list is referenced by its SENSOR_ID.	optional
OFFSET	Defines the offset of a channel. Should be left empty, if a sensor of the sensor list is referenced by its SENSOR_ID.	optional
AMPL_SETTINGS	Certain strings for setting up the different types of amplifiers can be combined separated by comma (','). See table 3 for the possible settings matching the modules. Should be left empty, if a sensor of the sensor list is referenced by its SENSOR_ID.	optional

TABLE 1: THE COLUMNS OF THE CHANNEL LIST

Different Types of Channels within the Channel List

The channel list may consist of different types of channels. These can be of the types **AI**, **PAD**, **CAN** or **MATH**.

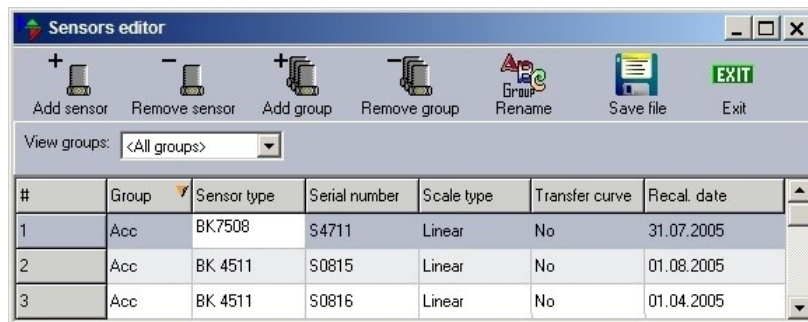
Channels of the type **PAD** must have a *channel number* (**CHANNEL**) which consists of the *slot-number* and the *channel number* of the module separated by ' _ '.

Channels of the type **CAN** must be defined in a *.dbc-file. The CAN-channels in the channel list having a **SHORT_NAME** *matching* to a channel *defined* in the .dbc - file will be set to used.

Channels of the type **MATH** must have the *calculation* in the column **SENSOR_ID**. The *syntax* of the calculation has to be equal to **DEWESoft Math**-channels. For MATH channels the field **CHANNEL** can be left *blank*.

3. section: **Sensor List** (optional)

The sensor list section is an *optional* part of the **Text setup file** (BigList). It is surrounded by the following two tags: **<SENSORLIST_V1>** and **</SENSORLIST_V1>**. In between a *number* of sensors can be defined. After loading a BigList in **DEWESoft**, the sensors defined *here* can be found within **DEWESoft Sensors editor**.



for detailed information about the **Sensors editor** see → **Sensor database**

To *match* a sensor to a *certain channel* of the channel list, the **SENSOR_ID** is used. If this ID *corresponds* to the one within the channel list, the **parameters** of a sensor are *applied* to the channel.

Column Heading	Description	Comment
GROUP	The group to which the sensor belongs to	
TYPE	Can be any string denoting the type of a sensor	
SENSOR_ID	The sensor ID allows to reference a sensor to a certain channel of the channel list	
ELEC_UNIT	The electrical unit (e.g. [V], [mV], etc.)	
PHYS_UNIT	The physical unit (e.g. [N], [m/s ²], etc.)	
SCALE	The scaling factor	
OFFSET	The offset	
DESCRIPTION	A description of the sensor	
RECALDATE (DD.MM.YYYY)	The recalibration date of a sensor. The syntax of the date has to match to the one specified in brackets.	
AMPL_SETTINGS	Certain strings for setting up the different types of amplifiers can be combined separated by comma (","). See Table 3 for the possible settings matching the modules.	

TABLE 2: THE COLUMNS OF THE SENSOR LIST

Module Settings

The following table matches the possible strings for **setting up** the modules to the different *module types*:

Command String	Comment	Module														
		DAQP-ACC	DAQP-ACC-A	DAQP-BRIDGE-A	DAQP-BRIDGE-B	DAQP-CHARGE	DAQP-CHARGE-A	DAQP-CHARGE-B	DAQP-FREQ	DAQP-FREQ-A	DAQP-V-A	DAQP-V-B	MDAQ-ACC	MDAQ-ACC-A	MDAQ-FILT	
AC	AC coupling							•		•	•	•	•			
DC	DC coupling							•		•	•	•	•			
DIFF	Differential inputs												•			
SE	Single ended inputs												•			
ICPON	ICP excitation on (no influence on the current)					•	•						•			
ICPOFF	ICP excitation off					•	•						•			
BIPOLAR	Bipolar mode										•	•				
UNIPOLAR	Unipolar mode										•	•				
FILTBE	Bessel-Filter			•				•			•	•			•	
FILTBU	Butterworth-Filter			•				•			•	•			•	
EXC0V	0 V excitation			•	•						•	•				
EXC0V25	0.25 V excitation			•	•											
EXC0V5	0.5 V excitation			•	•											
EXC1V	1 V excitation			•	•											
EXC2V5	2.5 V excitation			•	•											
EXC5V	5 V excitation			•	•											
EXC10V	10 V excitation			•	•											
EXCUST	Custom excitation *)			•	•											
BRFULL	Full bridge			•	•											
BRHALF	Half bridge			•	•											
BR1QU3WIRE	Quarter bridge 120 Ohm, 3-wire			•	•											
BR2QU3WIRE	Quarter bridge 350 Ohm, 3-wire			•	•											
BR1QU4WIRE	Quarter bridge 120 Ohm, 4-wire				•											
BR2QU4WIRE	Quarter bridge 350 Ohm, 4-wire				•											
SHUNTOFF	Switching off the shunt				•											
SHUNT1	Switching on the 1 st shunt				•											
SHUNT2	Switching on the 2 nd shunt				•											
SHUNTEXT	Switching to external shunt				•											
SHUNT9V	Apply 9V to shunt pin				•											
ICP4MA	4mA ICP excitation on	•	•				•						•			
ICP8MA	8mA ICP excitation on	•	•				•						•			
HPFILTSLow	Slow high-pass filter	•	•				•									
HPFILTFast	Fast high-pass filter	•	•				•									
OUTRESPONSELow	Slow output filter									•						
OUTRESPONSEFast	Fast output filter									•						
TRIGGERLEVEL	To set a trigger level *)									•						
BRHALF2	Half bridge mode; shunt 2 active			•												
BRFULL2	Full bridge mode; shunt 2 active			•												
INTOFF	Integration off (acceleration)						•									
INTSINGLE	Single integration (velocity)						•									
INTDOUBLE	Double integration (displacement)						•									
HPFILTMid	Middle high-pass filter						•									
TRIGRANGESMALL	0 to 3 V trigger range								•							
TRIGRANGELARGE	0 to 300 V trigger range								•							
SETCUSTOMRANGE	Activate custom range			•	•		•	•								
ACHP1	Activate high-pass filter 1													•		
ACHP2	Activate high-pass filter 2													•		

*) the value has to be added directly after the string without adding the unit

*) the value has to be added directly after the string without adding the unit.

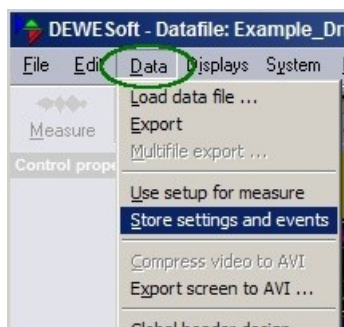
4.3 Settings and Events

Display appearance settings and **Events** (entered information about *measurement*) are stored in **DEWESoft** *.dsd (DEWESoft *data*) file.

Display appearance settings can be also stored in *.dss (DEWESoft *setup* data) file (in *Measure* mode).

for detailed information about **New / Load / Save** the *.dss (DEWESoft *setup* data) file see → **Setup Data**

Store Display appearance settings and Events



All defined events can be **stored** into the already open *.dsd (DEWESoft *data*) file by selecting:

Data menu → Store settings and events

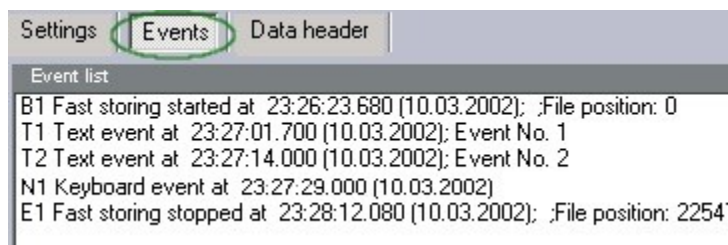
NOTE: This function stores also the current display settings into the file!

If you have changed anything in your **display appearance**, it will now be *stored also*.

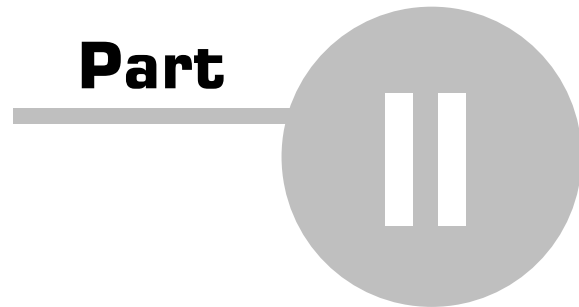
Load Display appearance settings and Events

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**:



Part



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 coming 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

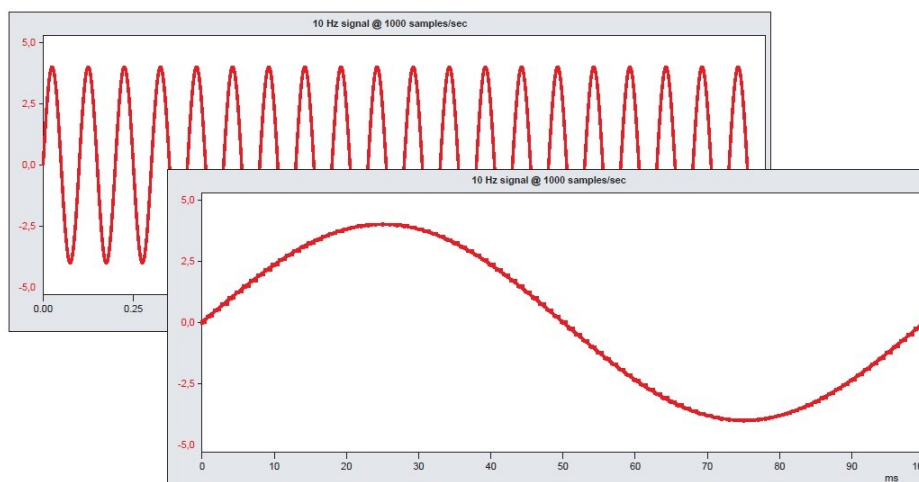
Selecting the **correct sampling rate** is very important to obtain good measurement results. When the sampling rate is *too high*, you will see a perfect signal, but you will also receive plenty of *unnecessary* data - waste of hard disk memory.

A sampling rate *below* the *signal frequency* will save hard disk memory - 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.

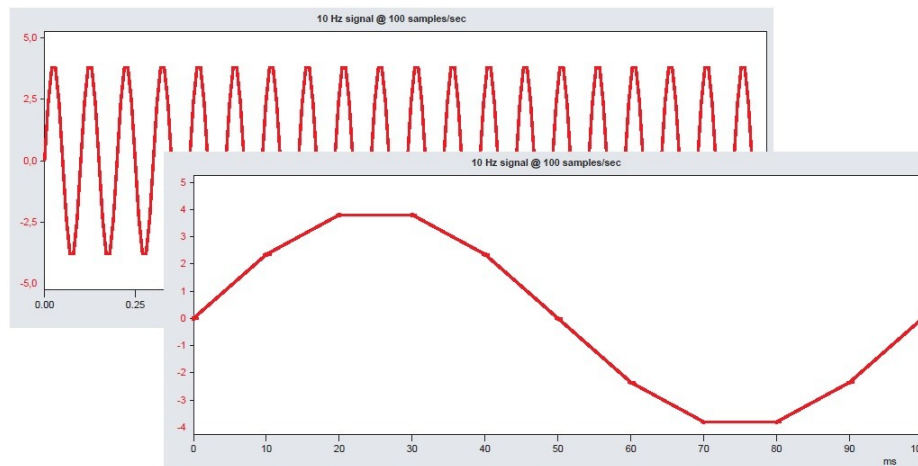
- **10 Hz sine wave, 1000 samples/sec** => **100 sampling points per period**

shows a wonderful *precise* sine wave signal - but requires *plenty* of hard disk memory



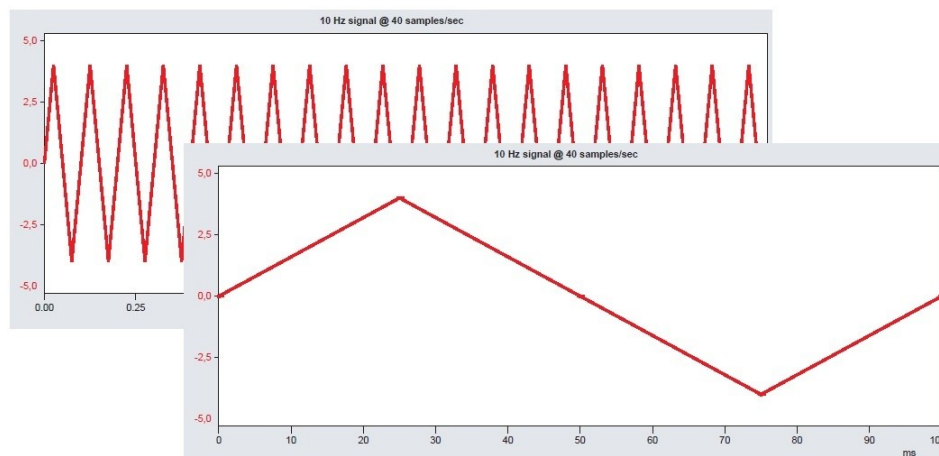
- **10 Hz sine wave, 100 samples/sec** => 10 sampling points per period

delivers a quite *useful* sine wave with *moderate* memory requirements



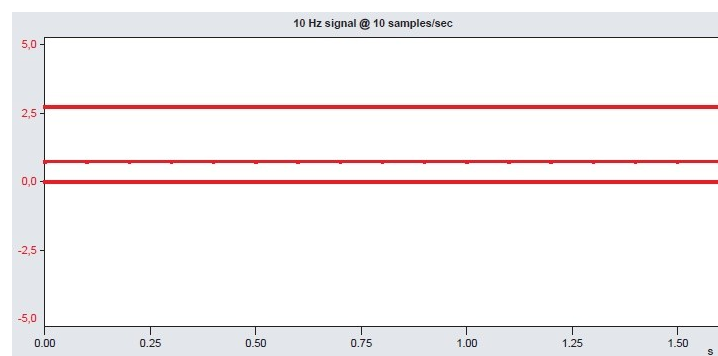
- **10 Hz sine wave, 40 samples/sec** => 4 sampling points per period

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



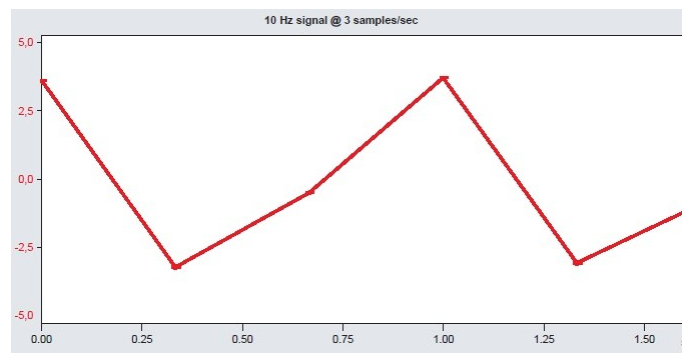
- **10 Hz sine wave, 10 samples/sec** => 1 sampling point per period

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:

$$\text{DiscreteSteps} = 2^{\text{NumberOfBits}}$$

Then divide your input range by the number of discrete steps:

$$\text{InputResolution} = \frac{\text{InputRange}}{\text{DiscreteSteps}}$$

or calculate directly:

$$\text{InputResolution} = \frac{\text{InputRange}}{2^{\text{NumberOfBits}}}$$

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!

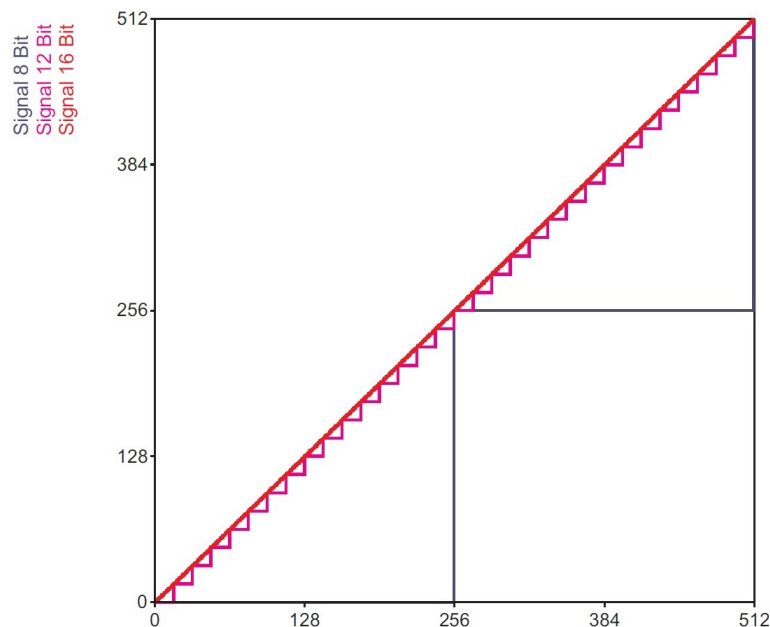
But high resolution has also other disadvantages: To get a stable signal, the A/D converter requires more time. The higher the resolution, the more time is required for the conversion:

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.

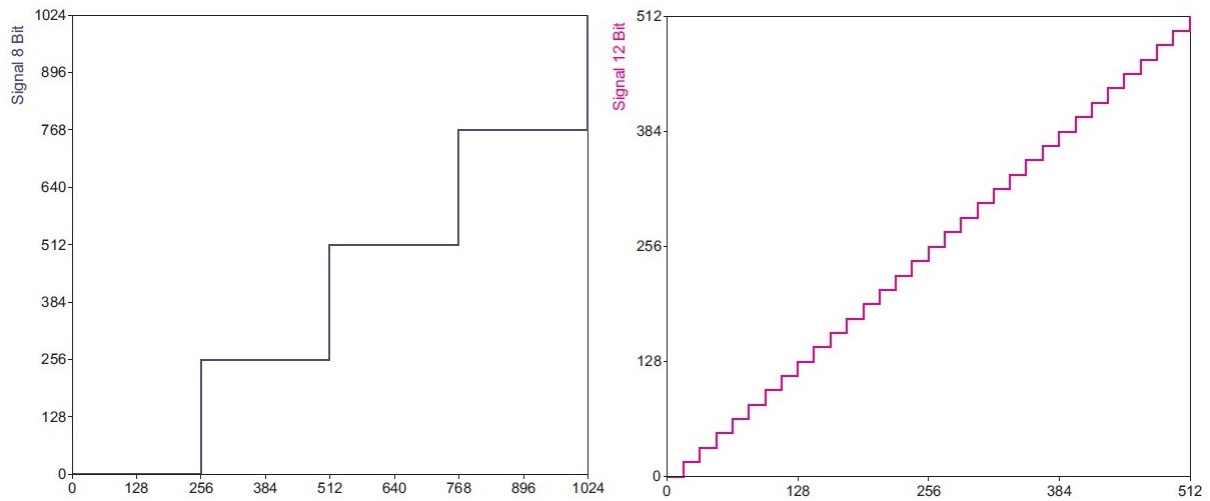
The special A/D boards are normally transient recorder boards with their own fast acquisition memory on-board. They are working 'block oriented', which means you *can't do a continuous* measurement. The technology is the same which is used in modern digital storage oscilloscopes.

When we compare now the same input signal sampled with 8, 12 and 16 bits, the signal will look like that:

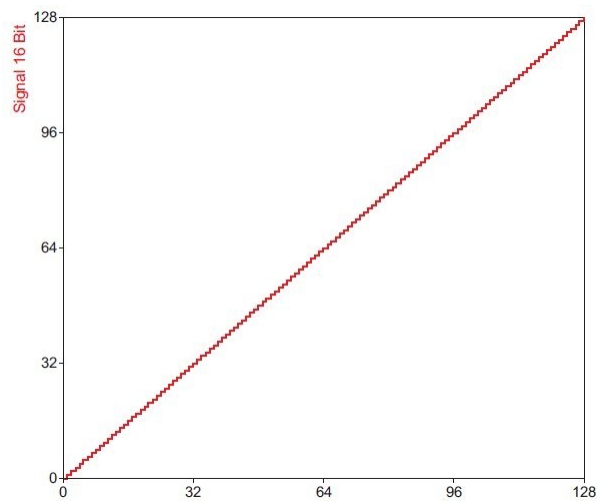


With three A/D boards (8, 12 and 16 bit resolution) set to 1 kS/s, we have acquired 512 ms of a signal, which is continuously growing with 1000 per second (ideal signal without units). While the 16 bit board has an own discrete amplitude value for each sampling point, the 12 bit board always shows discrete 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*. This way is also used for DEWETRON amplifiers and systems. The formula is simple:

$$\text{Accuracy}[\%] = \frac{\text{InputSignal} - \text{ExpectedInputSignal}}{\text{FullScaleInputRange}} \times 100$$

Three examples for a 10 V input range:

$$\begin{aligned} \text{Accuracy} &= \frac{9.505\text{V} - 9.5\text{V}}{10\text{V}} \times 100 = \frac{0.005\text{V}}{10\text{V}} \times 100 = 0.05\% \\ \text{Accuracy} &= \frac{4.505\text{V} - 4.5\text{V}}{10\text{V}} \times 100 = \frac{0.005\text{V}}{10\text{V}} \times 100 = 0.05\% \\ \text{Accuracy} &= \frac{0.505\text{V} - 0.5\text{V}}{10\text{V}} \times 100 = \frac{0.005\text{V}}{10\text{V}} \times 100 = 0.05\% \end{aligned}$$

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 not used 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

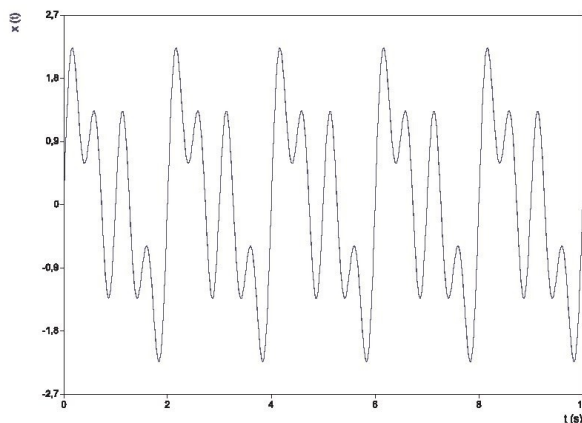
Frequency analysis is widely used tool because of its power to **reveal** the **real nature** of *dynamic signals*. Working for many years in practice doing *measurements* and lately also developing *FFT analyzers*, it became very obvious: knowing the theory of frequency analysis helps to reveal the 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.

- **What is frequency analysis**
- **Theory of frequency analysis**
- **Properties of the Fourier transform**
- **Where we can get in trouble with the FFT**
- **How can we at least try to avoid it**
- **Averaging - how to further enhance the results**
- **Representation of different signals in the FFT**
- **Conclusion**

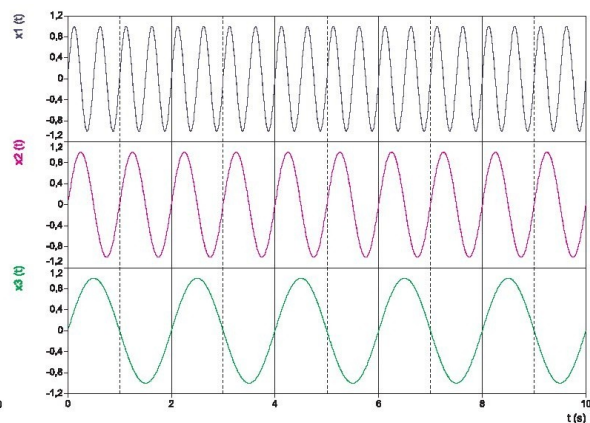
2.1 What is frequency analysis

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).

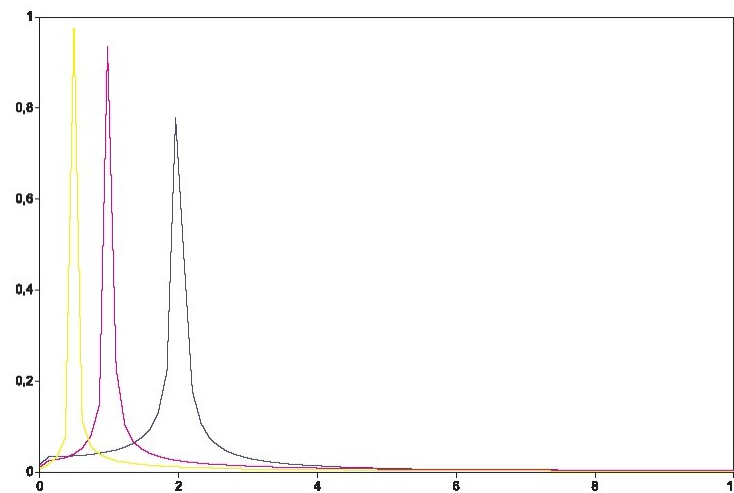


Picture 1: ORIGINAL SIGNAL



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

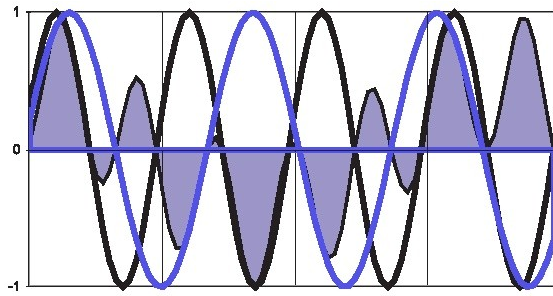
Frequency analysis is **converting** *time domain* data to the **set of sinus waves**. So the theory says. But I like more if we say that frequency analysis *checks the presence of certain fixed frequencies*. Let's look at the equation of discrete Fourier's transform to figure out why:

$$(1) \quad X(k\omega_0) = \sum_0^{N-1} x[n] \cdot (\cos(2 \cdot \pi \cdot k \cdot n / N) + j \cdot \sin(2 \cdot \pi \cdot k \cdot n / N))$$

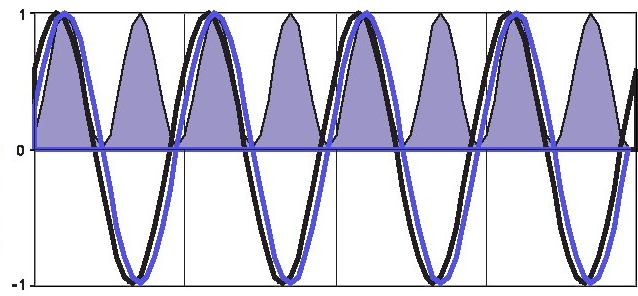
Let's think about how does it work. To check the presence of certain sine wave in a data sample, the equation does the following:

- 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

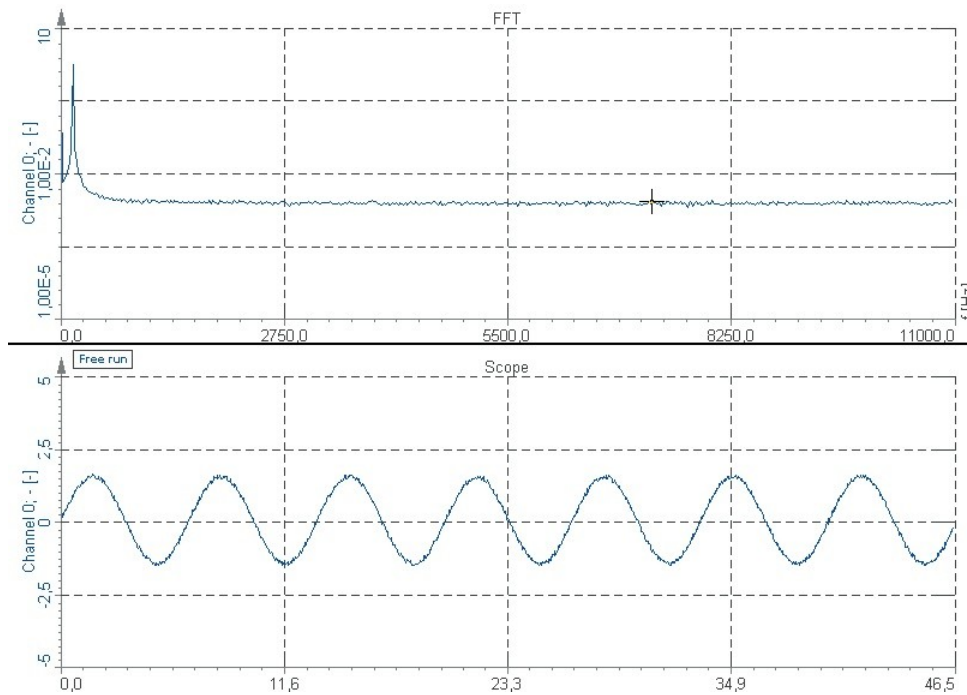
The principle shown above can extract basically any frequency from the sine wave, but it has one disadvantage. It is awfully *slow*. The next important step in the usage of DFT was the FFT algorithm. Because there is more than enough theory, let's just say that this analysis reduces amount of calculations 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:

$$(2) \quad \text{SampleRate} = \text{MaximumSignalFrequency} \cdot 2 \cdot 1.25$$

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:

$$(3) \quad \text{LineResolution} = \frac{\text{SampleRate} / 2}{\text{NumberOfLines}}$$

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.

$$(4) \quad \text{TimeToCalculate} = \frac{\text{NumberOfLines} \cdot 2}{\text{SampleRate}}$$

Just for fun we can also combine the equations above and we get:

$$(5) \quad \text{LineResolution} = \frac{1}{\text{TimeToCalculate}}$$

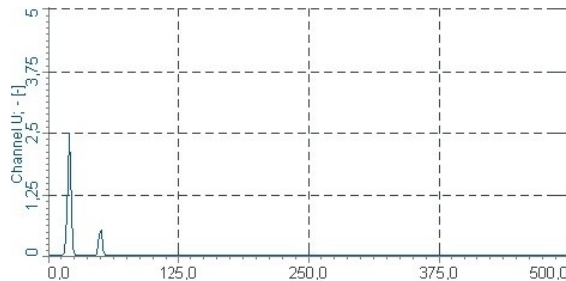
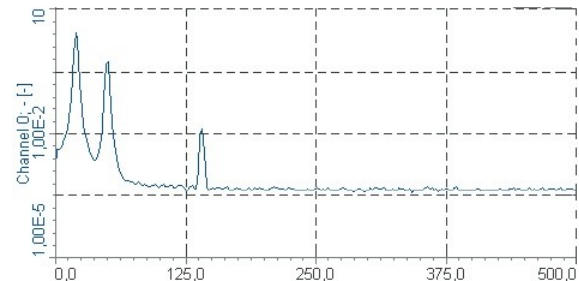
Let's look to the equations above and make a list for 22 kHz sample rate.

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

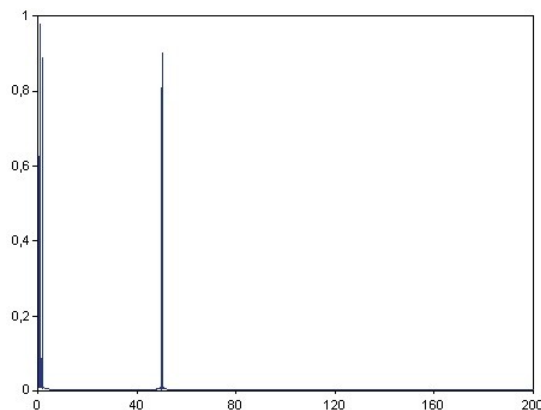
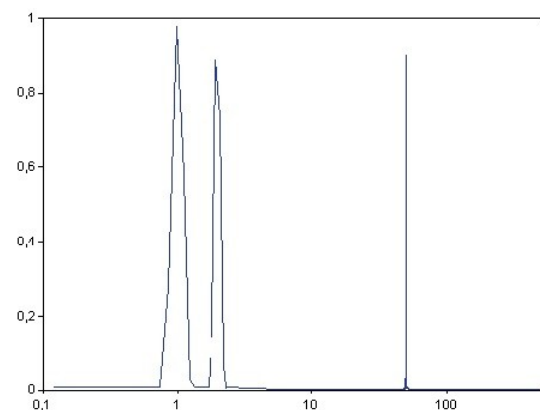
Table 1: EXAMPLE OF FFT SETTING

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:

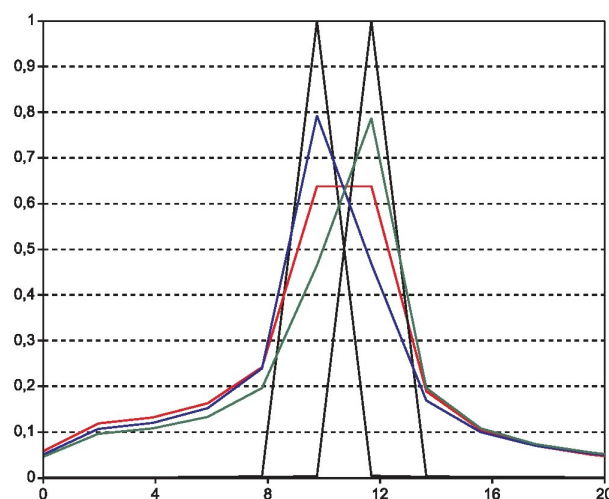
$$(6) \quad X(j\omega) = \sum_{n=-\infty}^{n=\infty} X[n] \cdot e^{-j\omega nT}$$

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 (pickett - 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 'pickett 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.



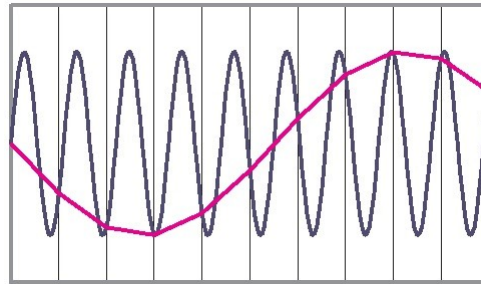
Picture 11: AMPLITUDE ERROR AND LEAKAGE OF THE FFT (NO WINDOW)

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

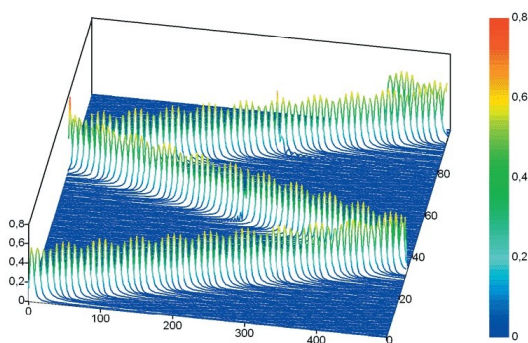
The other problem comes from the fact of the signal conditioning. If *simple* A/D converters are used, the sampling frequency must be at least twice higher than the maximum frequency of the signal. This effect is called aliasing effect. *Picture 12* shows the reason for it. Vertical lines represent samples 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.



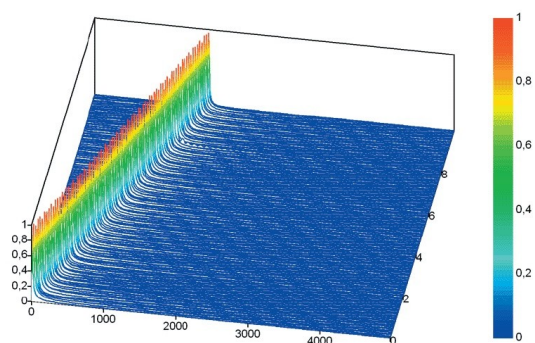
Picture 12: ALIASING

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



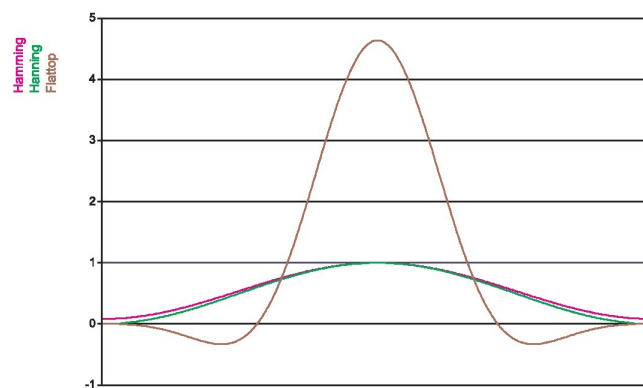
Picture 14: CORRECTLY SAMPLED SIGNAL

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

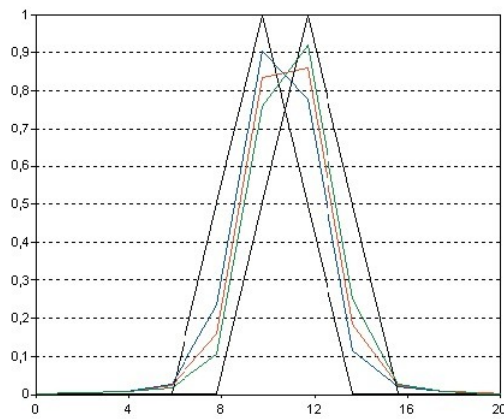
And here is the most common question to FFT: what are the differences between windows and when to use certain windows?

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*).

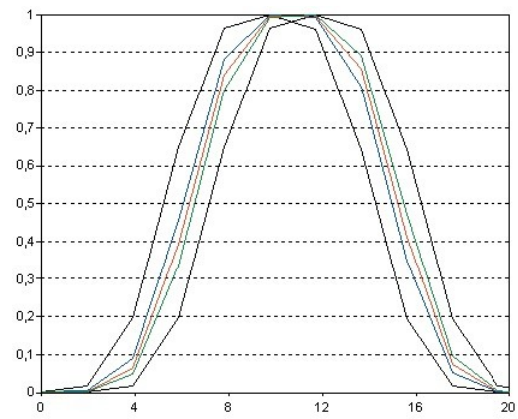
For **general** purpose, **Hanning** or **Hamming** are commonly used because they provide a good compromise between fall-off and amplitude error (maximum of 15%). This comes from the fact that old frequency analyzers didn't have that many possibilities in terms of frequency lines and these two windows 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.

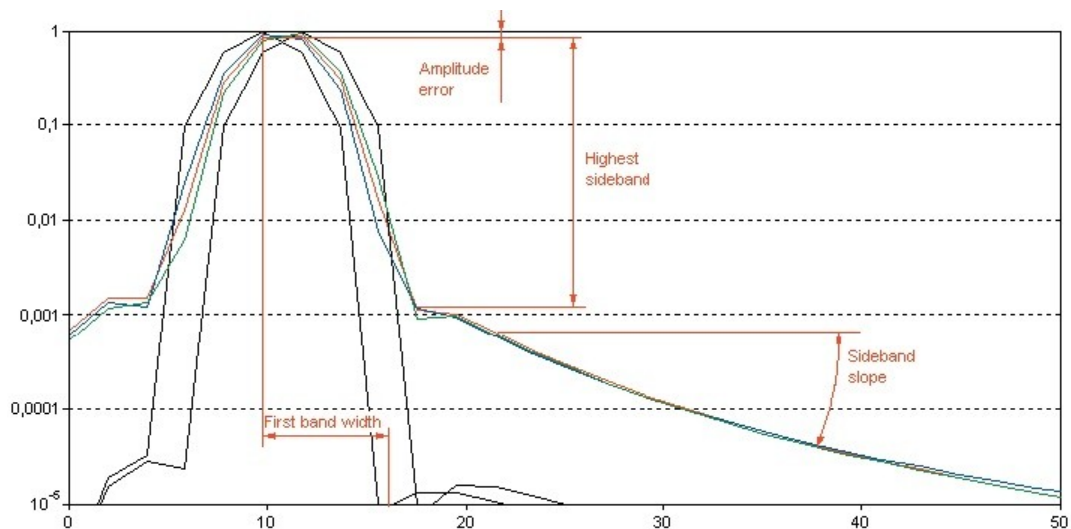


Picture 16: HANNING WINDOW



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.



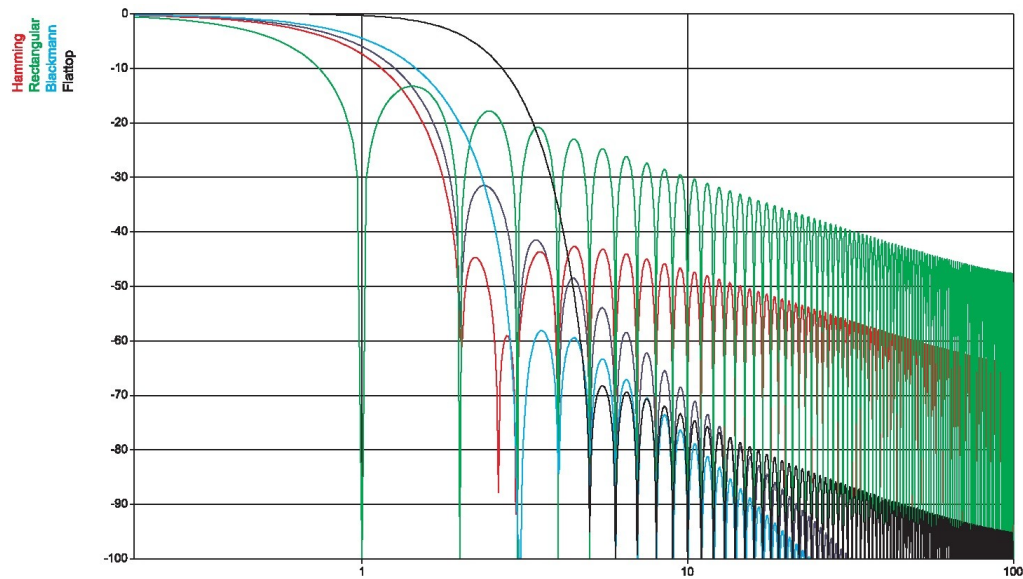
Picture 18: WINDOW PROPERTIES DESCRIPTION

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.

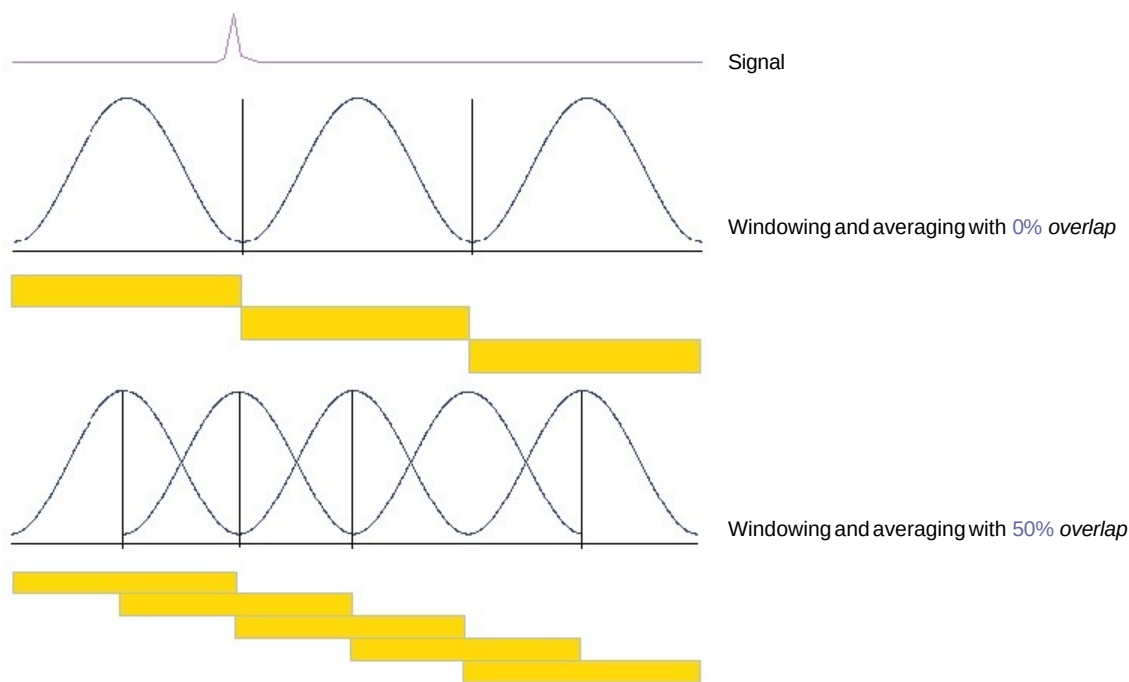
There is one more trick with windows: if we are sure that *all the frequencies will fall on its frequency lines*, **rectangular** window will give us the best result. For example to measure the *harmonics* of the *power line* (50 Hz in Europe or higher), choose 6400 or 9600 sample/sec sampling rate, 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 (FFT's become *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 *lose 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.

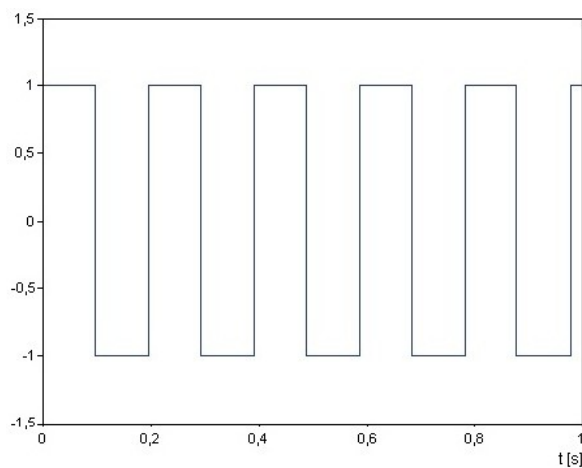
By the way, do you know what '**real-time**' frequency analyzer means? It means that it is able to calculate and show data with **66.7%** (or 50 % in some literature) *overlapping* and therefore *has no data loss*.

2.7 Representation of different signals in the FFT

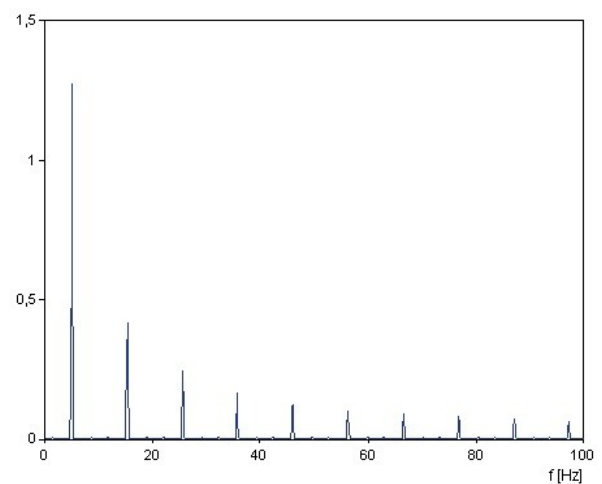
For now, we have only seen how the sine waves looks like in the FFT, but let's observe how other types of the signals look like:

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.



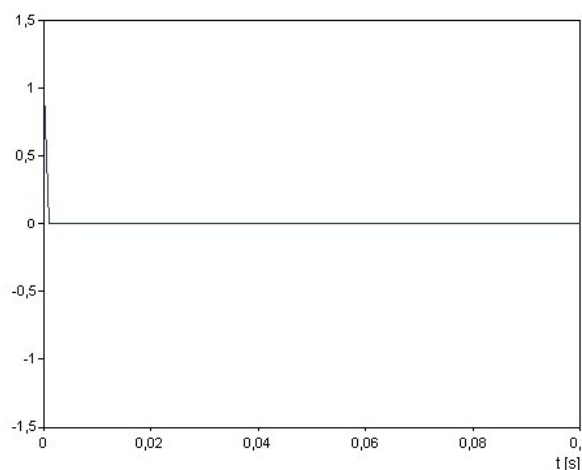
Picture 21: TIME DOMAIN SQUARE WAVE



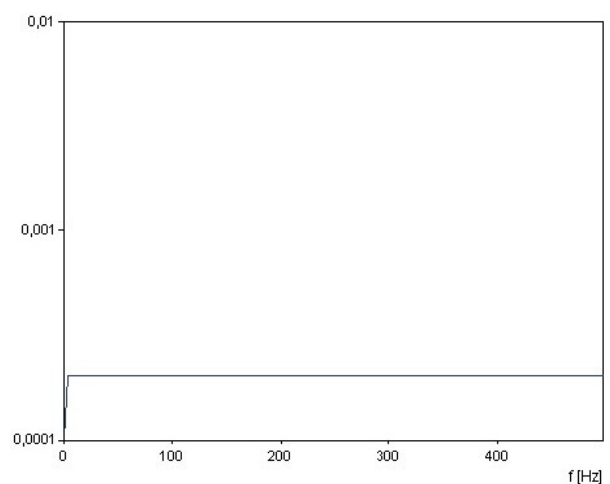
Picture 22: FREQUENCY DOMAIN SQUARE WAVE

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.



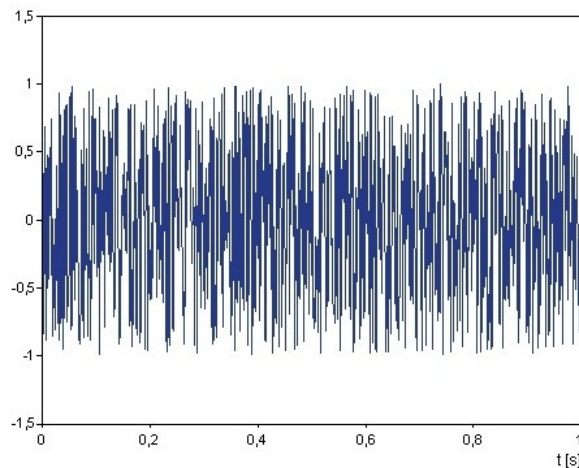
Picture 23: TIME DOMAIN IMPULSE



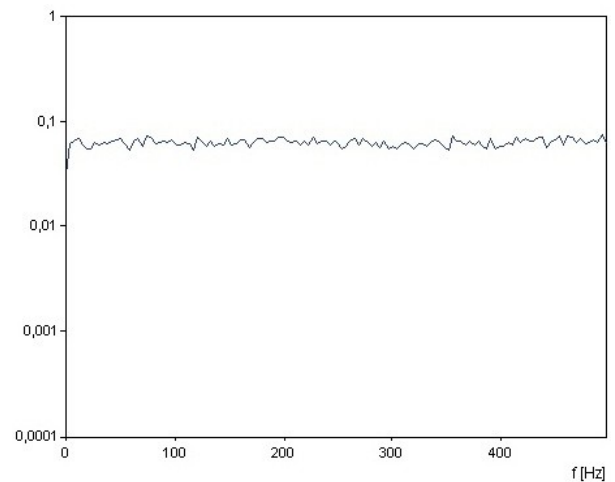
Picture 24: FREQUENCY DOMAIN IMPULSE

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.



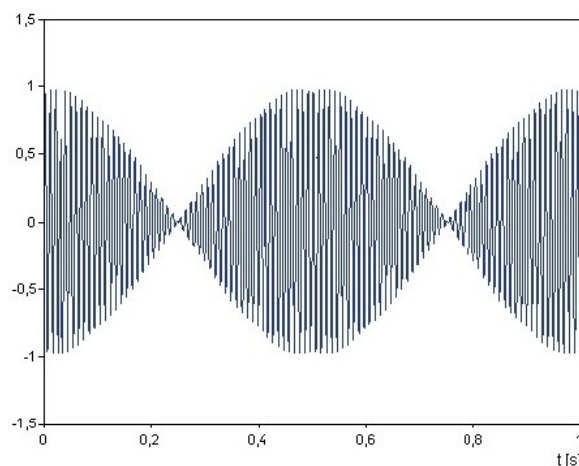
Picture 25: TIME DOMAIN NOISE



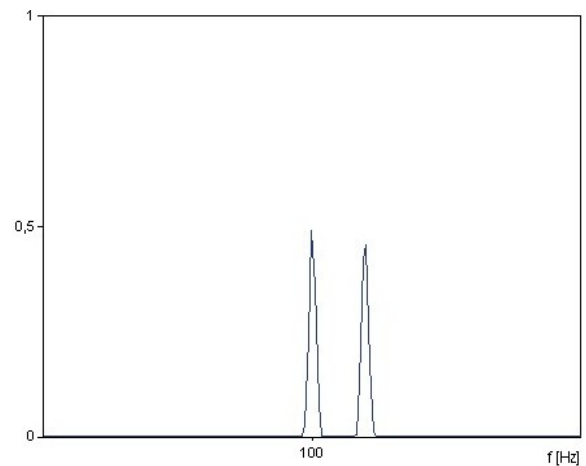
Picture 26: FREQUENCY DOMAIN NOISE

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.



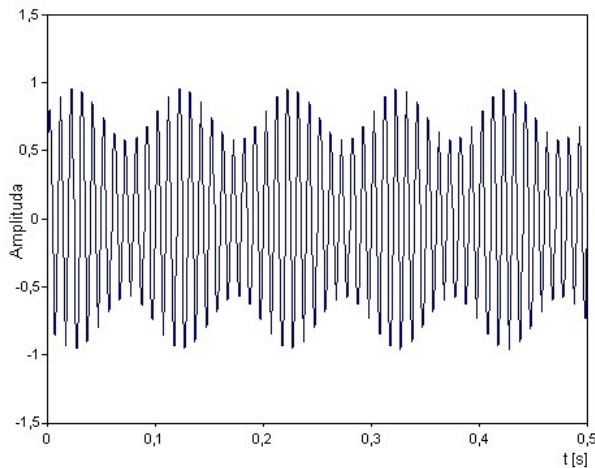
Picture 27: BEATING IN TIME DOMAIN



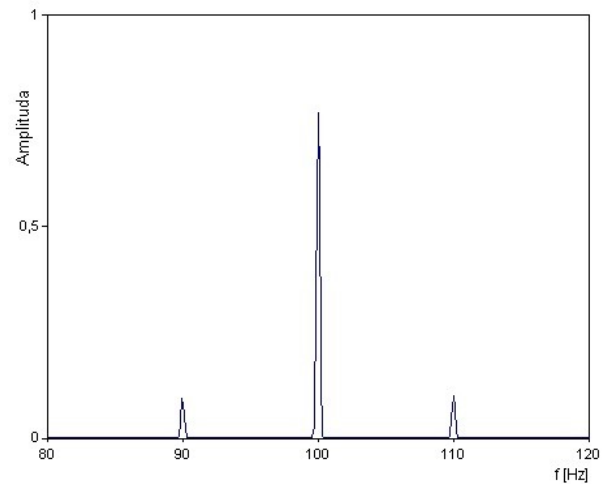
Picture 28: BEATING IN FREQUENCY DOMAIN

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

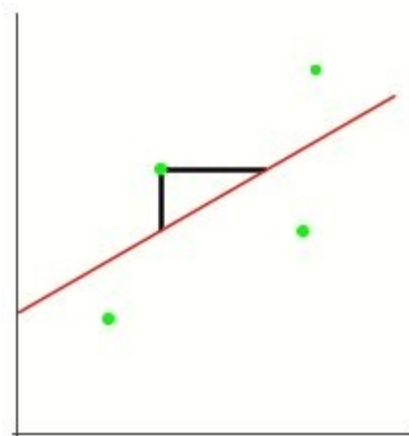
H1 algorithm expects **noise** on output.

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.

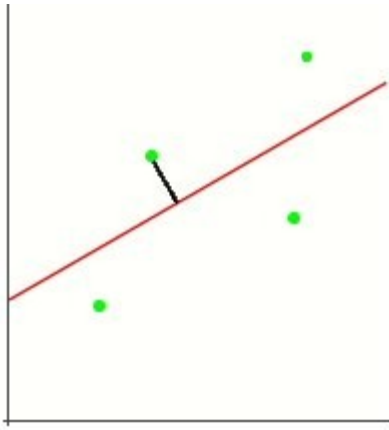


Vertical offset - leas squares (minimizes distance in x or y direction).

Hv algorithm

Hv algorithm expects **noise** on input and output.

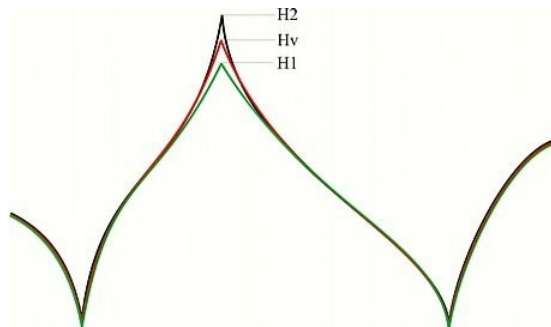
Hv algorithm tends to *minimize the noise on input and output*. It is always bounded by H1 and H2. The only cut down is that because of it's complexity it uses a lot of computing power.



Perpendicular offset - total least 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.



4 FlexPro Scripts

FlexPro is recommended software for **off-line analysis** acquired data. This software also enable creating **Scripts** to automate lot of routine procedures of particular work (e.g. for export data from **DEWESoft**).

Experienced *FlexPro* users can examine the *preformatted* **scripts** and **macros** that are loaded as **templates** in the C:\Dewetron\Program\DEWESoft63\Scripts subdirectory, and learn how to **create** their own scripts (see also **introductive example** below).

NOTE: *Please note that FlexPro scripts REQUIRE that this application be properly already installed on this computer.*

The *concept* is a nice one: once the script is created, **DEWESoft** runs the 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)
- *perform* his own job and
- *runs* the script

Export data example : 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 example show the power of *FlexPro* 'hot script' ActiveX with **DEWESoft**.

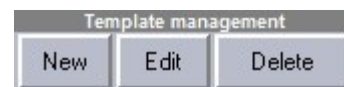
for information about **Export data** see → [User Guide](#) → **Exporting Acquired Data**

Template management controls

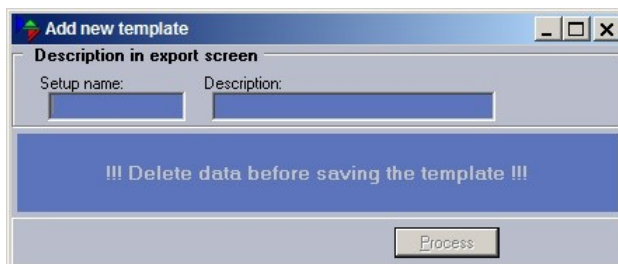
In **DEWESoft** you can easily *create* your own scripts in *FlexPro*, or *edit* the existing ones.

Just press the **New** button to **create** a *new* one or **Edit** to **change** a selected *existing* script.

To **delete** an *existing* script simply *select* it and press the **Delete** button.



When you press the **New** button, you have to *enter* a **Setup name**, **Description** and a **comment** for the *new* script:



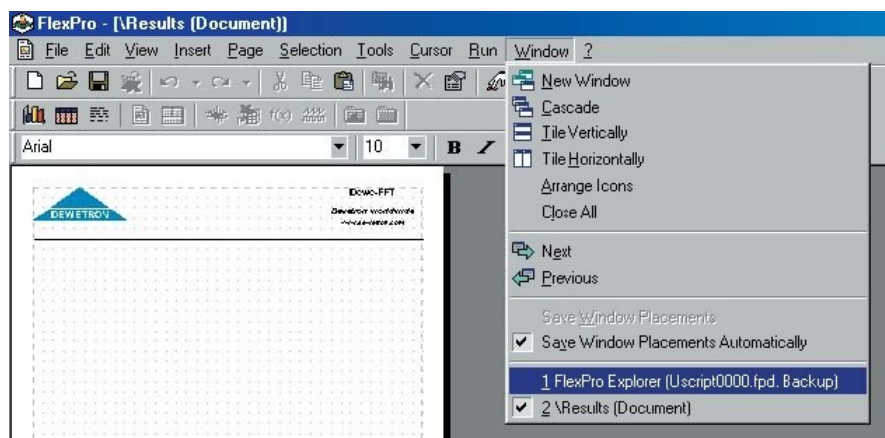
When you confirm your entries click on **Process** button, *FlexPro* will automatically start with a default **template**.

When we select **Edit** button to change an existing script, *FlexPro* will automatically start with a selected **template**.

NOTE: For more details about creating and changing scripts please refer to the original *FlexPro* documentation.

Creating FlexPro script - export data example

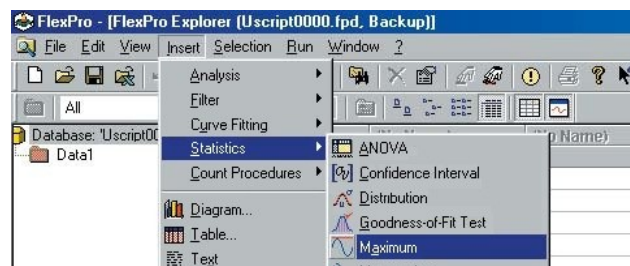
You can easily create your own scripts in *FlexPro*. After you confirm your entries in *Add new template* window with click on **Process** button, *FlexPro* will automatically start with a default **template**. Enter the *Window* menu and select the *FlexPro Explorer* window.



The *FlexPro Explorer* window allows you to create your script. Please refer to the original *FlexPro* documentation for more details. We just want to show you a simple script, which should be a first help.

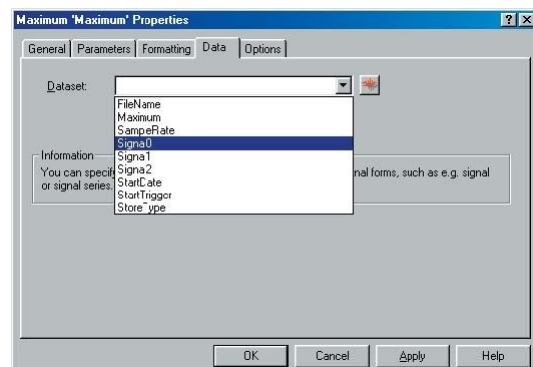
1. STEP

Select *Insert* → *Statistics* → *Maximum*



2. STEP

Select the *desired signal*

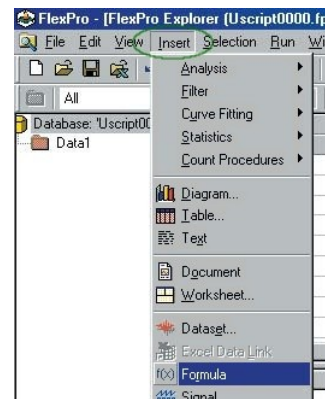


3. STEP

Repeat STEP 1 and 2 with *Statistics* → *Minimum* and - *Statistics* → *RMS*

4. STEP

Select *Insert* → *Formula*

**5. STEP**

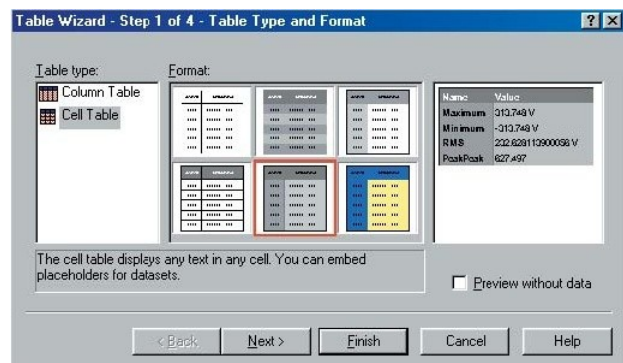
Enter the text '*Maximum - Minimum*' and close the formula window

**6. STEP**

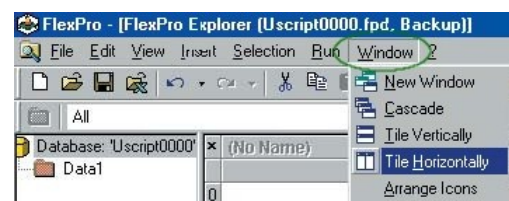
Select the formula with the right mouse button and select *Rename* from the list. Rename the formula to '*peakpeak*'

7. STEP

Select the signal **Minimum**, **Maximum**, **RMS** and **peakpeak** from *FlexPro Explorer* and select *Insert* → *Table*

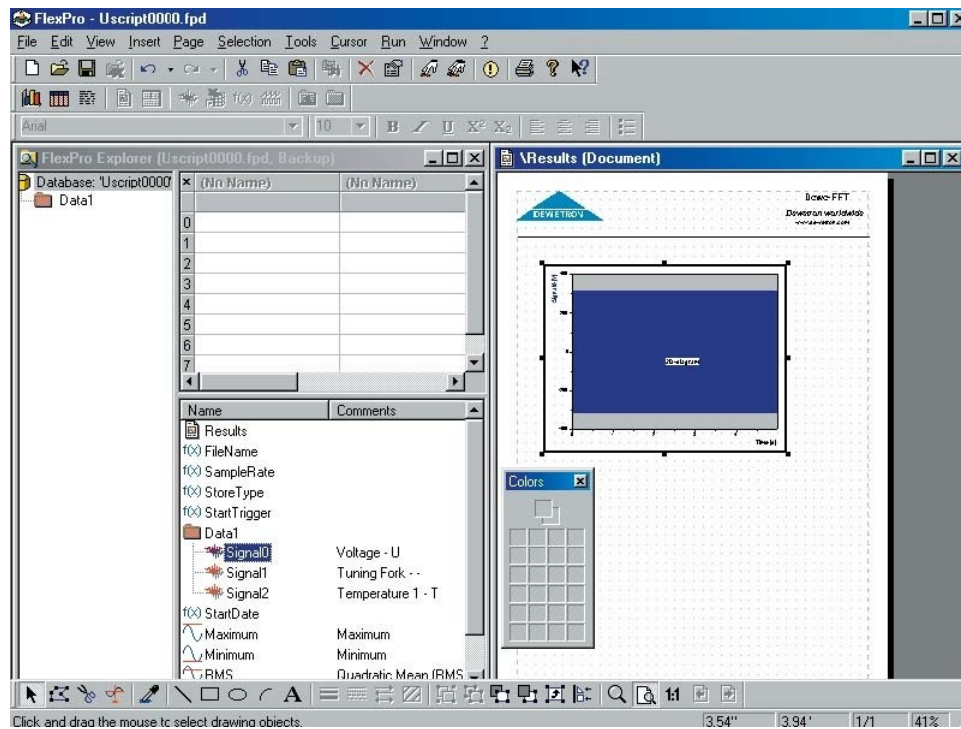
**8. STEP**

Select *Window* → *Tile horizontally*



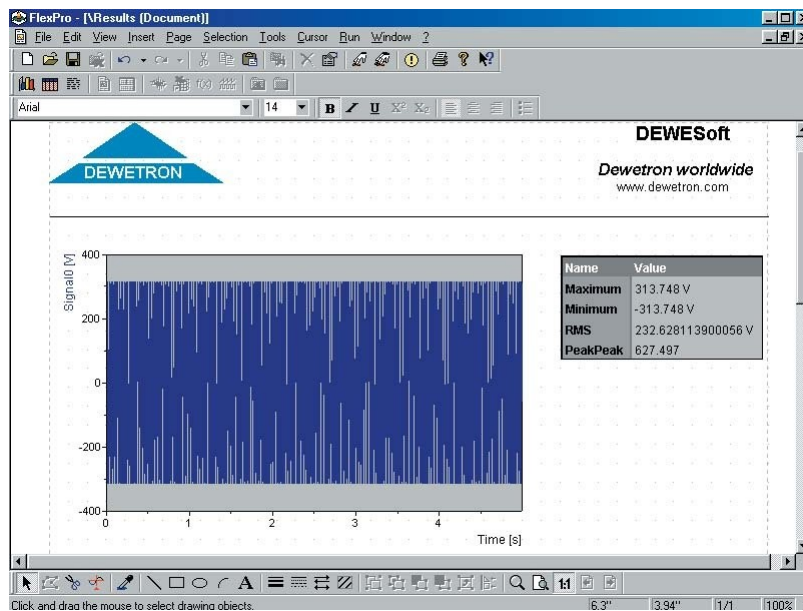
9. STEP

Click on signal and move it on the *Report* - a chart will be automatically created



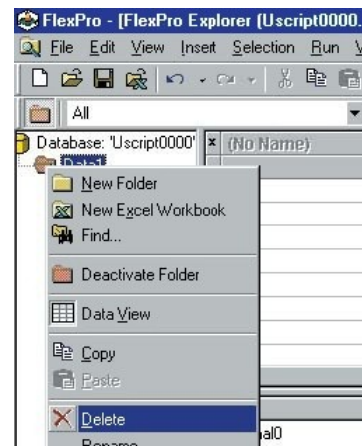
10. STEP

Move also the Table on the Report - your first script is finished:

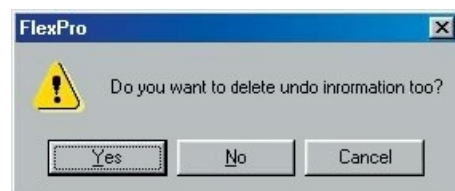
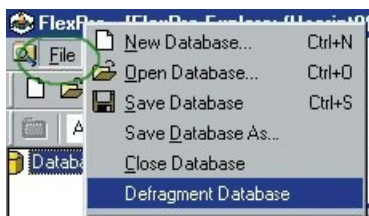


After your script has been defined, the most important procedure follows:

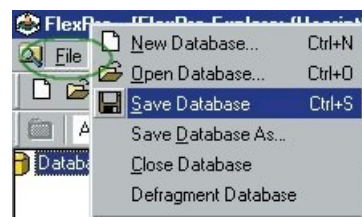
Delete all data and the data folders (right click on folder and select **Delete** from displayed menu).



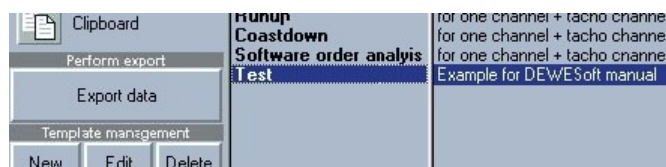
Defragment the *FlexPro* database and confirm to delete the *undo* information:



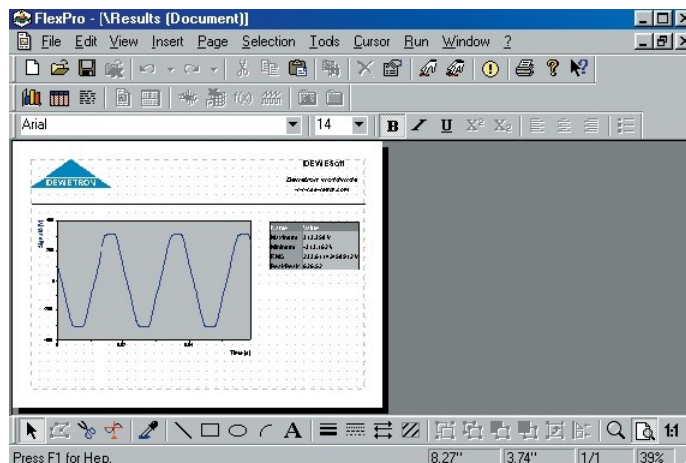
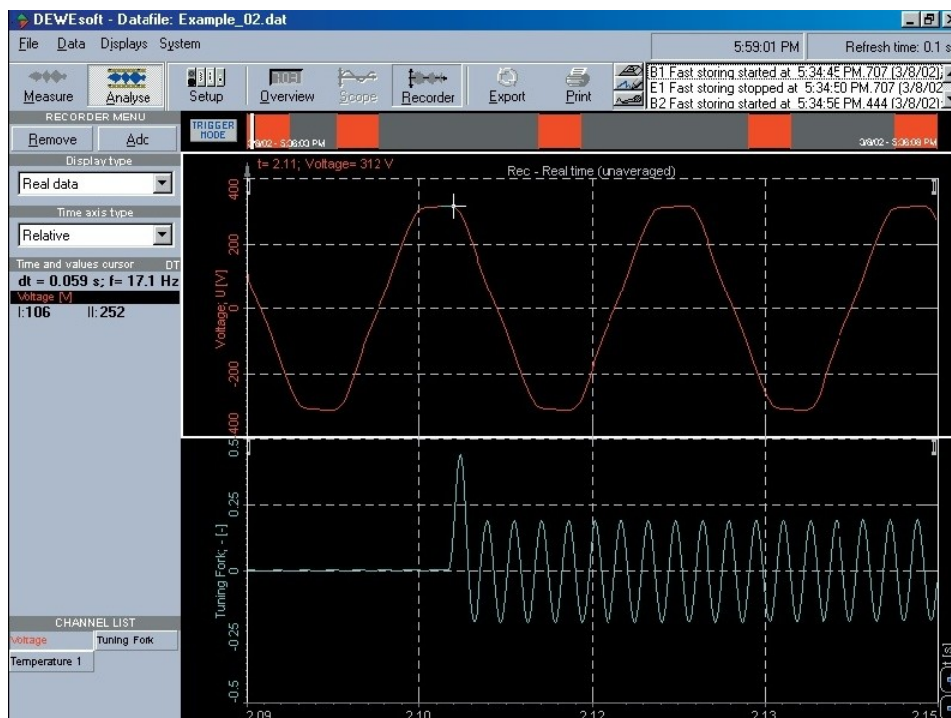
Save the database and *exit FlexPro*.



DEWESoft will now display the *new script* in the *FlexPro* scripts list:



Let's try it out: Change to the recorder, select a file and an area you want to display, go back to the export and select our new export script:



Index

- . -

.avi file 396
 .dsd file 396
 .dss file 396
 .txt file 396
 .xml file 396

- 3 -

3D FFT
 3D FFT instrument 289
 3D FFT instrument 289

- A -

Accuracy 413
 Additional display screens 234
 Alarm monitoring 212
 basic settings 213
 Measurement setup 212
 output selection 214
 set conditions 216
 Amplitude resolution 410
 Analog in Channel setup 5
 activate / deactivate 7
 Amplifier column - type and ranges 9
 channel list 6
 channel setup 12
 color - C column 8
 Input zeroing - CAL column 11
 Name 9
 On / Off 7
 Physical values column 11
 Slot 7
 Analog IN channels
 channel setup 5
 Analog meter 251
 type & appearance 252

Analog out Channel setup 27
 Arbitrary - Additional settings 36
 channel setup 33
 Channel value - value column 32
 Control Options 32
 Noise - Additional settings 37
 Operation modes 28
 Waveform type 35

Analog OUT channels
 channel setup 27
 Setting during Measurement 38

Analyse 352
 channel selection 358
 display settings information 355
 displaying data 357
 export instrument display to Video 393
 export multiple files 392
 exporting acquired data 379
 loading data files 353
 Publishing data 374
 replaying data 367
 Reviewing data files 353
 selecting data 360
 selecting triggered data 364
 selection range of stored data 360
 Signal overview window 358
 storing settings and events 370
 Time selector 359
 Video post synchronization 371
 working with Events 368

Angle sensor 141

- B -

Basic Statistics 126
 Basics of A/D conversion 408
 Accuracy 413
 Amplitude resolution 410
 Sampling rate 408

- C -

CA noise 139
 CA p-v 312
 CA scope 314

- Calibrate in channel 22
- CAN bus Acquisition Module
 - Channel setup 56
 - Measurement setup 53
- Channel setup
 - Analog input 12
 - Analog output 33
- Channels selector 229
 - channel grouping 232
 - list / Group view 230
 - resizing 232
 - select /deselect 231
- Combustion analysis 176
 - CA p-v 312
 - CA scope 314
 - Measurement setup 176
- Common graph settings and tools 242
- Control properties 240
- Counter setup
 - Encoder 46
 - Event 44
 - Frequency 50
 - Gated 44
 - Period 48
 - Pulsewidth 48
 - Two pulse edge separation 49
 - Up/down counting 44
- Counters 41
 - Counter setup 41
- Create own display 227
 - Channels selector 229
 - Displays menu functions 234
 - Instrument setup 240
- D -**
- Data acquisition 223
 - display and replay data 353
 - NET data 344
 - Recording acquired data 320
- Data displays and controls 315
 - Background picture 316
 - Line element 318
 - Text element 316
- Data Header design 330
 - data directory management 336
 - field position 331
 - File name field 335
 - folder levels 336
 - icon bar 330
 - Input field 332
 - Memo field 334
 - multiple headers 334
 - Selection field 333
 - work with folder levels 337
- Data management 396
 - Exporting acquired data 379
 - Recording acquired data 320
 - Settings and Events 405
 - Setup data 396
 - structure of Text Setup file 401
- DEWESoft
 - general mathematics 86
- Digital input 51
- Digital meter 248
 - display value 248
 - drawing options 249
 - resolution 249
- Display and replay data
 - Signal overview window 358
 - Time selector 359
- Display Settings
 - load 405
 - store 405
- Displays menu functions
 - Additional display screens 234
 - delete display 235
 - Full screen mode 234
 - rename display 235
- DSA analysis
 - Orbit plot 292
- Dynamic acquisition rate 322
- Dynamic signal analysis 137, 143
 - Angle sensor 141
 - CA noise 139
 - exact frequency 137

- E -**Events**

- load 405
- store 405

Exact frequency 137

Export instrument display to Video 393

Export multiple files 392

Export option - formats

- copy to Clipboard 386
- DEWESoft export 385
- File export 382
- to FlexPro 386
- to MS Excel 386

export Template management 387

- FlexPro and MS Excel scripts 387

Exporting Acquired Data 379

- export 379
- export formats 381
- export multiple files 392
- exported Channels 381
- MergeDB - FlexPro 390
- set export data properties 380
- to FlexPro 386
- to MS Excel 386

Exporting to FlexPro 386

- MergeDB - FlexPro 390
- Template management 387

Exporting to MS Excel 386

- Template management 387

- F -

FFT Filter 121

FFT instrument 280

- Amplitude display 284
- Averaged FFT 285
- averages count 286
- change frequency axis scale 281
- Current FFT 285
- Cursor function 282
- DC cutoff filter 285
- free run overlap 291
- Line resolution 283

Number of ticks 284

Online calculation 282

Order history options 291

Scope history options 291

settings 282

Weighting 285

Window type 283

X and Y scale type 284

FFT reference curve 135

File

- export 396
- import 396
- new 396
- save 396

File name Setup 326

- data directory management 326
- file name 328

Filters 105

- FFT Filter setup 121
- FIR Filter setup 117
- IIR Filter setup 106

FIR Filter 117

FlexPro and MS Excel scripts 387

FlexPro Scripts 431

Formula editor 92

- example of input events 101
- example of logic functions 100
- example of measure functions 103
- example of signals 102
- example of stopwatch 104
- mathematical function 95
- operators 95

Fourier transform - properties 417

Frequency analysis

- Aliasing 420
- Amplitude error 420
- averaging of the signal 425
- how to avoid trouble 422
- Leakage 420
- overlapping 425
- representation of signals in the FFT 426
- theory 416
- what is 415

- G -**GPS Acquisition module**

- channel setup 64
- GPS display 303
- GPS information 64
- Measurement setup 63

GPS display 303

- calibrate Map 305
- View setting 304

- H -**H1, H2 and HV algorithms 429****Harmonics display 298**

- data panel 299
- display value 299
- Y axis display 300

Horizontal / Vertical bar graph 250

- type & appearance 251

Human vibration 162

- calibration 169
- Measurement setup 162
- output channels 166

- I -**IIR Filter 106**

- custom filter 113
- custom filter import from Matlab 116
- derivation and double derivation 112
- integration and double integration 110
- Standard filter 107

in Channel setup form 12

- Amplifier settings 17
- calibration 22
- General 14
- live data preview 24
- Scaling 19
- Sensors setting 15

Indicator lamp 253

- Discrete display 255
- display value 254
- options of Discrete display 257

Instrument appearance

- grid resolution 243
- position 237
- size 238
- Time axis scaling 244
- transparency - Control properties 241
- transparency - on screen menu 238
- unified properties 242
- Y-axis scale / auto scale 245

Instrument group

- add / remove instrument 241
- columns number 241
- copy & paste 236
- create 240
- transparency 241
- unified properties 242

Instrument settings

- Average time 243
- common graph settings and tools 242
- cursor functions 246
- display type 243
- grid resolution 243
- minimum / maximum shown value 244
- single / multiple Time axis 242
- Time axis scaling 244
- time axis type 243
- upper & lower limit 244
- Y-axis scale / auto scale 245

Instrument setup 240

- 3D FFT Analysis 289
- Analog meter 251
- appearance 237
- Background picture 315
- CA p-v 312
- CA scope 314
- Control properties 240
- delete instruments 236
- Digital meter 248
- FFT Analysis 280
- GPS 303
- Harmonics 298
- Horizontal / Vertical bar graph 250
- Indicator lamp 253
- Line element 315
- Octave 286
- Orbit plot 292

Instrument setup 240
 Overload indicator 307
 Recorder 258
 Scope 268
 Screen edit functions 235
 Tabular display 309
 Text element 315
 undelete instruments 236
 unified properties 242
 Vector scope 295
 Vertical recorder 261
 Video display 301
 X-Y recorder 264

- L -

Latch math 129

- M -

Matematics 86
 Angle sensor 141
 Basic Statistics 126
 CA noise 139
 Dynamic signal analysis 137
 exact frequency 137
 FFT reference curve 135
 Filters modules 105
 Formula editor 92
 Latch math 129
 module setup 88
 modules 86
 Reference curve 131
 reference curves 130
 statistics modules 125
 XY reference curve 134
 Mathematical Function 95
 Measure 223
 basic recording setup 321
 NET data 344
 recording acquired data 320
 Measurement Setup 3
 Alarm monitoring 212
 Analog in Channel setup 5
 Analog out Channel setup 27
 CAN bus Acquisition 53

Combustion analysis 176
 Counter 41
 Counter sensor editor 207
 Digital input 51
 Dynamic signal analysis 143
 GPS Acquisition 63
 Human vibration 162
 NET data acquisition 218
 Order tracking 171
 Power Acquisition 70
 Sensor database 199
 Sound level 144
 Torsional vibration 152
 Triggered Acquisition 187
 Video Acquisition 65

MergeDB - FlexPro 390

- N -

NET acquisition 218
 application setup 218
 channel setup 220
 connect and manage measurement unit 345
 creating display 347
 local NET setup 219
 measurement 347
 measurement setup 344
 remote NET setup 220
 storing data 349
 transfer setup 221
 uploading data to client 350

- O -

Octave instrument 286
 analysis type 288
 averaging 288
 band display type 288
 display settings 287
 online calculation 287
 Operators 95
 Orbit plot instrument 292
 Order tracking 171
 Measurement setup 171
 output channel settings 175
 Overload indicator display 307

Overload indicator display 307
 settings 308

- P -

Power Acquisition Module 70
 calibration current input 84
 input channels 75
 Measurement setup 70
 output channels - 3 phase 81
 output channels - single phase 77
 wiring scheme 75

Pre-defined displays 225

Publishing data 374
 copy channel setup 377
 copy visual elements 378
 screen printout 374

- R -

Recorder
 display type 259
 instrument setup 258
 single value axis 260

Recording Acquired Data 320
 basic recording setup 321
 data directory management 326
 Data Header design 330
 Dynamic acquisition rate 322
 filename 321
 filename Setup 326
 marking with events 341
 multifile 321
 multifile - make new after 322
 multifile - stop storing 322
 start / stop Recording 339
 start storing 325
 static / reduced rate 323
 storing options 324

Reference curve 131

Reference curve modules 130
 FFT reference curve 135
 Reference curve 131
 XY reference curve 134

Reference Guide 407

- S -

Sampling rate 408

Scaling in channel 19

Scope
 current trigger setup 272
 Cursor tab 274
 History tab 277
 instrument setup 268
 reference curves 275
 Scale tab 276
 setting and tools 270
 store trigger settings 273
 trigger setting 270
 trigger timing setup 272
 zoom in Run mode 270

Screen printout
 multi page 375
 notes 375
 page orientation 375
 printing 376

Selecting data 360
 lock cursor 1 364
 selecting triggered data 364
 selection range of stored data 360
 window zoom 363
 zoom in time axis 362

Sensors
 Counter sensor editor 207
 database 199
 definition and editing 199
 general information 201
 group 204
 scaling 201
 transfer curve 203

Setup data 396

Setup data file
 load 397
 load display setup data 398
 load last 398
 new 396
 save 399
 save multifile 400

Signals out operation modes

Signals out operation modes

- Burst 31
- Chirp 31
- Fixed 29
- Step sweep 30
- Sweep 30

Single / Multiple Time axis 242

Sound level 144

- Measurement setup 144
- microphone calibration 150
- output channels 146

Start / Stop Recording 339

- automatically 343
- manually 339
- remotely 343
- triggered 342

Start storing data 325

Statistics modules 125

- Basic Statistics 126
- Latch math 129

Storing options 324

- T -

Tabular display 309

- print format 311
- settings 310

Theory of frequency analysis 415

Time axis type 243

Torsional vibration 152

- extract order 162
- Measurement setup 152
- output channels 159
- rotational input channel 155
- rotational output channels 156
- setup 158

Triggered Acquisition 187

- Condition setup 191
- Measurement setup 187
- start trigger setup 188
- stop trigger setup 190
- timing setup 187

- U -

User Guide 2

- V -

Vector scope 295

- axis settings 297
- harmonic selection 296
- measured values 296

Vertical recorder

- alarm levels 263
- display type 262
- instrument setup 261
- X scale divisions 263
- X-axis scale / auto scale 262

Video Acquisition Module 65

- camera setup 66
- Measurement setup 65
- Video display 301

Video data

- file name conventions 371
- remove from data file 373
- storing synchronization data 373
- synchronization 371, 372

Video display 301

- camera setting 302
- cameras selector 302
- scaling 302

- W -

Window functions

- Blackmann 422
- flat-top 422
- Hamming 422
- Hanning 422
- Kaiser 422
- Rectangular 422

- X -

X-Y recorder 264

- cursor 268
- display type 266

X-Y recorder 264

- draw filter 267
- draw option 267
- graph type 265
- history 267
- time scale 267
- X and Y scale divisions 267

XY reference curve 134