DeweFRF v6.5

modal analysis software

User manual

v1.2.0

General information

Software version

DeweFRF User manual corresponds with software version 6.5.

Printing notice

Specifications subject to change without notice.

Copyright notice

Trademark notice

Disclaimer

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

Table of Contents

Part I

Welcome to DeweFRF

Welcome to **DeweFRF** - DEWESoft FRF modal analysis software that radically simplifies the analyse and determination of system characteristics.

As you would expect from a fair software, **DeweFRF** provides you with help on every aspect of working with the program:

First time users	**Tutorials**	explain *basic steps* to do online or offline and single input - single output (SISO) modal analysis, *how to work with projects and how to import / export measurement setups, geometry and FRF modal data*
Detailed function description	**User guide**	beside *four basic procedure* (**Channel** and **FRF** setup, **Measure** and **Analyse**) includes also *Offline calculation*, *DeweFRF software interface* and *Data management* (how to *load / save* measurement setups, geometry and FRF modal data)
Background information	**Reference guide**	explain *System requirements* for work whit DEWE software, appropriate *hardware* and *general* DeweFRF *System setting*; this guide also contains information about *Theory of frequency analysis* and *H1, H2 and Hv calculation algorithms*.

For details on using this help about DeweFRF software *see also* → **Getting help**.

About DeweFRF software

FRF is abbreviation for **F**requency **R**esponse **F**unction which is the base for working with DeweFRF.

Modal analysis is a procedure of determining system characteristics of:

- *mechanical* systems, this would be a response of a certain mechanical construction when a force is applied and
- *electrical* systems, this would be a response of an electrical system (for example an amplifier).

DeweFRF modal analysis software is software that helps: to generate input signal (*excitation*) to system, to measure systems *response* and analyse system characteristics → *see* **Basis of work with DeweFRFs** for getting more information about DeweFRF software.

DeweFRF software can be used with recommended system hardware and software → *see* **System requirements**.

1 Getting help

This help is designed to be used on-screen. It is extensively cross-linked so that you can find more relevant information to any subject from any location. If you prefer reading printed manuals then a PDF version of the entire help is available. This may be useful as a reference but you will probably find that the active hyperlinks, cross-references and active index make the on-screen electronic version of the help much more useful.

Getting started

Start by studying the **Basis of work with DeweFRF** and **DeweFRF software interface**.

See **DeweFRF tutorials** sections for some *basic* tutorials to get you started with using DeweFRF, eg Four step tutorial, Calculation tutorial, SISO - Single I/O modal analysis tutorial and Project management tutorial.

Working procedures

To learn how to do something consult the working procedures section, **DeweFRF User Guide** which beside four basic procedure include also Offline calculation and Data management.

Learn about

For additional information about **modal analysis** see the **DeweFRF Reference Guide** and **Analysis background**.

Using the help while you're working

As far as possible the help *separates* instructions and background information. This makes it much easier to refer to the "how-to" instructions.

All sections have extensive *links* to the other relevant sections so it doesn't really matter where you start.

Getting a printed user manual

Please don't try to print the HTML Help version of the help from the Microsoft help viewer as it would look terrible. You can get the formatted PDF version of the entire documentation designed for printing from the installation CD.

As mentioned above, however, you will probably find that the on-screen version of the help is much more useful because of the hyperlinks and cross-references.

2 Basis of work with DeweFRF

DeweFRFs base

DeweFRF modal analysis software to be based upon:

Signal generator	DeweFRF includes full-featured signal generator with 16 or more signal outputs. There are different modes and many functions for each mode.
Measure	To *measure* back excitations and systems response. There are *two ways* of making the measurement → *directly* or *offline*.
Analyzing	Analyze stored FRFs and determine system characteristics. It will also help to evaluate measurement results.

Measurement methods

Before trying to setup the measurement, let's notice that there are **two ways** (methods) of making the FRFs measurement:

1. first way is to measure the FRFs directly *while acquiring* data - From measure and the
2. second way is to measure and calculate respectively offline From stored data.

Decision what to do depends on the user (if the archive time data is needed) and on the system performance. The system benchmarks might differentiate from system to system, but with recent computers (3 GHz P4) it is possible to measure online 256 channels at few kHz bandwidth.

There are **two** method to generate input signal (*excitation*) and also measurement:

Impulse measurement	→	input signal (excitation) is generated with *modal hammer*.
Continuous measurement	→	excitation with *function generator* or *shaker.*

For detailed information → see **Measure methods**

Basic procedures

Work with *DeweFRF* compose ***four basic procedure*** *(step)*:

1. CHANNEL SETUP - to set all the *analog channels* which will be used during measurement: excitation and sensors.
2. FRF POINT & GEOMETRY SETUP - to define *geometry* and nodes of mechanical structure, assign *points* and setup *parameters* for calculation.
3. MEASURE - acquire frequency response *data*.
4. ANALYSE - animating *shapes*, calculate *damping* and export *data* to UFF and clipboard.

3 System requirements

Standard application requirements

To achieve a good performance for all standard applications, we recommend the following hardware and software:

- WINDOWS 2000 / XP [1)]
- Intel Pentium III 700 MHz processor or higher
- 256 MB RAM or higher
- Approx. 40 MB free hard disk space for DEWE Software
  Additional 20 to 200 MB for drivers and post processing software packages
- A/D board for using **real** mode (requires licence key) → for setup this hardware *see* **Hardware setup**

> *We strongly recommend to run DeweFRF with Windows Administration rights - otherwise file access or driver problems might occur!*
> [1)] *DeweFRF is not running under WINDOWS 95/98/ME/NT4 operating systems!*

High-performance application requirements

For all high-performance applications like high-speed video or high-speed data acquisition, we recommend the following hardware and software:

- WINDOWS 2000 / XP [1)]
- Intel Pentium 4 2.4 GHz processor or higher
- 512 MB RAM or higher
- Serial ATA or SCSI hard disks and/or RAID systems
- Approx. 40 MB free hard disk space for DEWESoft
  Additional 20 to 200 MB for drivers and post processing software packages
- A/D board for using **real** mode (requires licence key) → for setup this hardware *see* **Hardware setup**

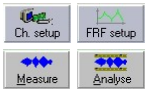
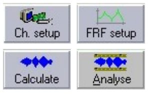
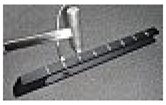
> *We strongly recommend to run DeweFRF with Windows Administration rights - otherwise file access or driver problems might occur!*
> [1)] *DeweFRF is not running under WINDOWS 95/98/ME/NT4 operating systems!*

Part II

Tutorials

Welcome to the tutorials of DeweFRF modal analysis software.

In DeweFRF software **TUTORIAL** are *four* basic tutorials available:

	**Basic four step tutorial for direct measurements**	explains *basic steps* to do online modal analysis in DeweFRF and is recommended for all new users
	**Basic tutorial for calculation from time data**	explains the *procedures* to do modal analysis from stored data
	**SISO** **Single input single output tutorial**	explains the specialities if the structure is excited with *single input* and measured with *single output* (SISO)
	**Project management tutorial**	will help to explain how to work with *projects* and how to import / export measurement *setups*, *geometry* and *FRF modal data*

1 Four step tutorial

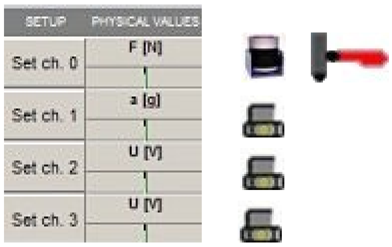
This tutorial will guide through four basic steps how to do *modal analysis*.

Please click on caption or picture for detailed explanation of the certain step

1

CHANNEL SETUP

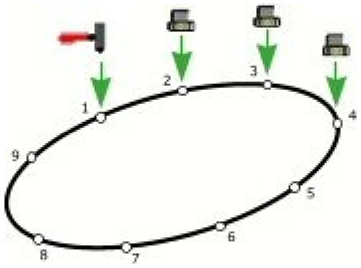
Set all the analog channels which will be used during measurement: excitation and sensors.



2

FRF POINT & GEOMETRY SETUP

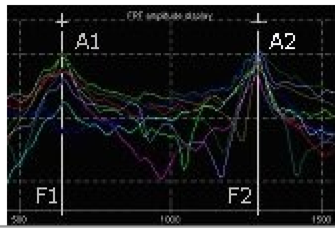
Define geometry and nodes of mechanical structure, assign points and setup parameters for calculation.



3

MEASURE

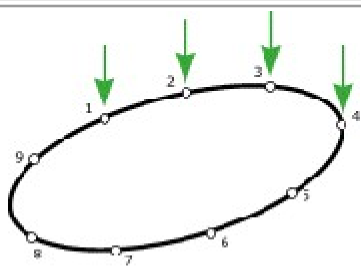
Acquire frequency response data.



4

ANALYSE

Animating shapes, calculate damping and export data to UFF and clipboard



1.1 Channel Setup

The **Channel setup** can be done by clicking the  icon on FRFs tool bar.

In **Channel setup** we can define:

Set all the channels which will be used during measurement in Analog and Analog out tab

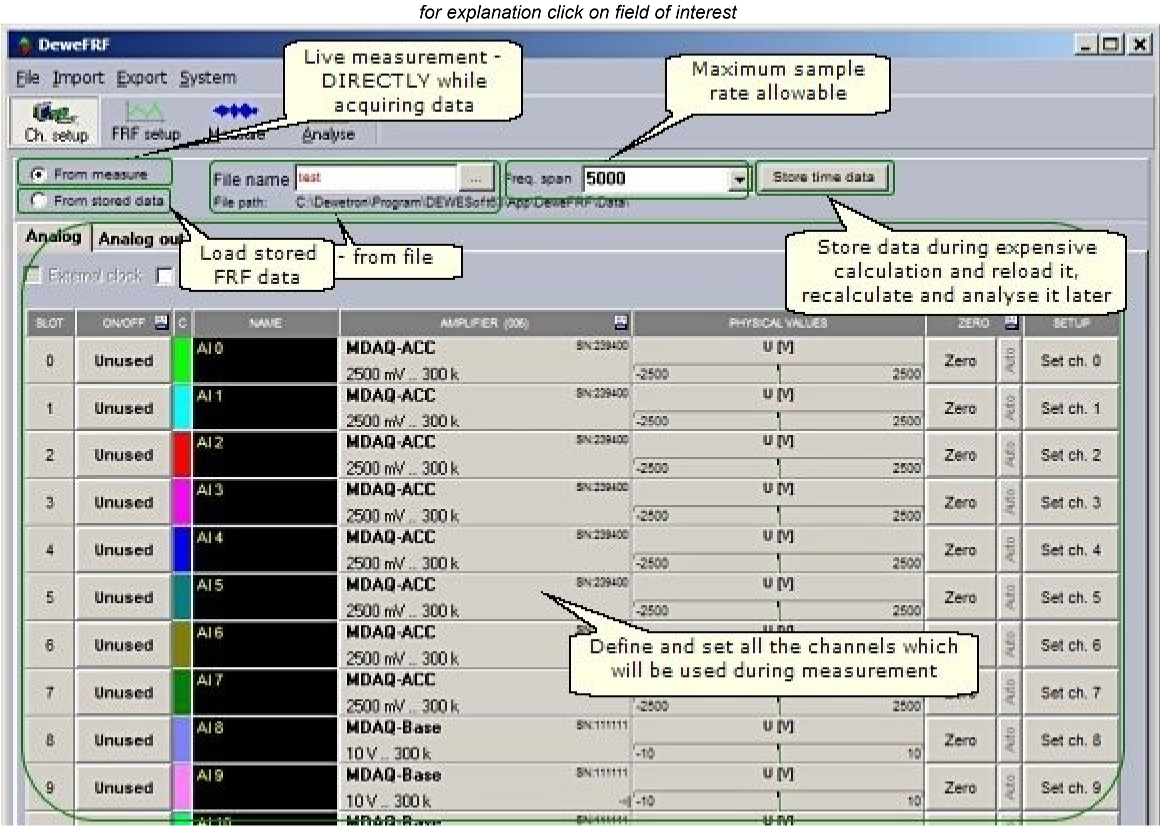
Measurement method: From measure (default) or From stored data → see Calculation tutorial for **offline calculation**

Set frequency span - maximum sample rate allowable

About Import/Export Channel setup data see → Project management tutorial

When the Channel setup are *done*, go to the *next step* → **FRF setup**

for explanation click on field of interest



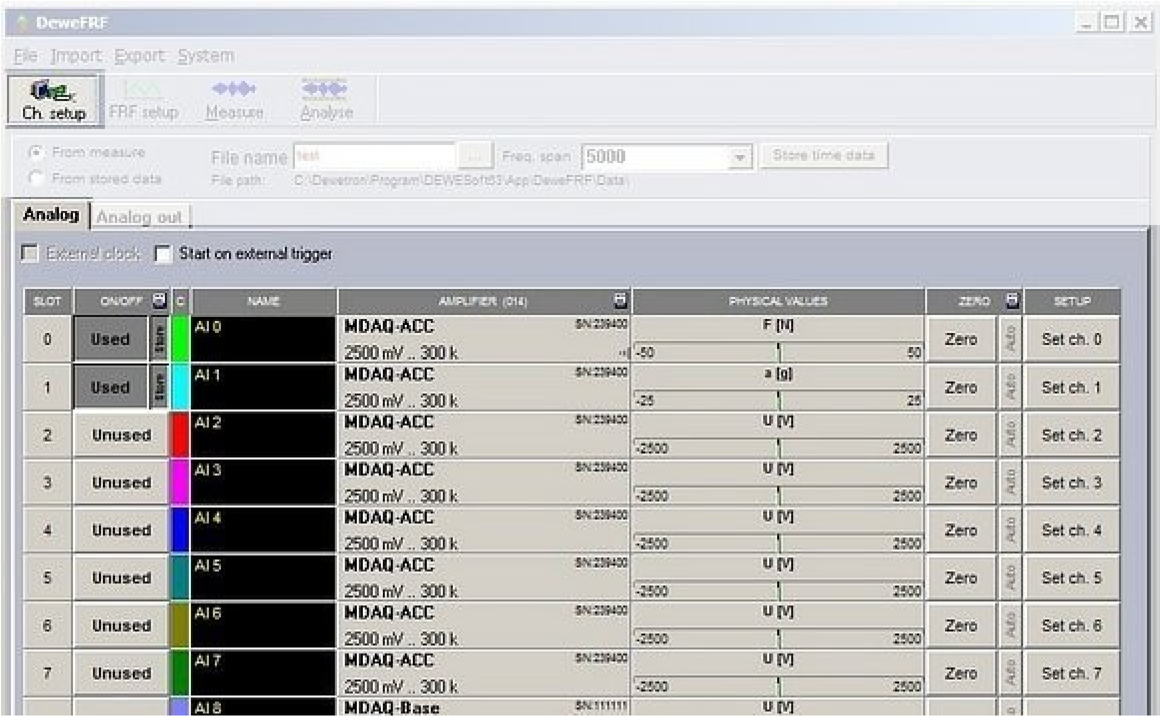
The basic channel setup is the same as in all Dewe software and can be read also in DEWESoft manuals.

Two tabs which is the most important for DeweFRF in the Ch.Setup screens are:

- **Analog** tab → define and set *all* the channels which will be used during measurement
- **Analog out** tab → define analog *output* setting.

Depending on selected choice in System menu → **Hardware setup** we can set also other selected tabs (eg *Math*, etc).

Analog channel setting



The *important* thing here is to <u>choose</u> and <u>setup</u> the channels which are used in the measurement with:

Slot	- input channel
On/Off	- activate / deactivate channels
C	- channel color selector
Name	- channel name
Amplifier	- amplifier type and ranges
Physical values	- current input values
Zero	- remove offset
Setup	- input settings and calibration

Channel naming

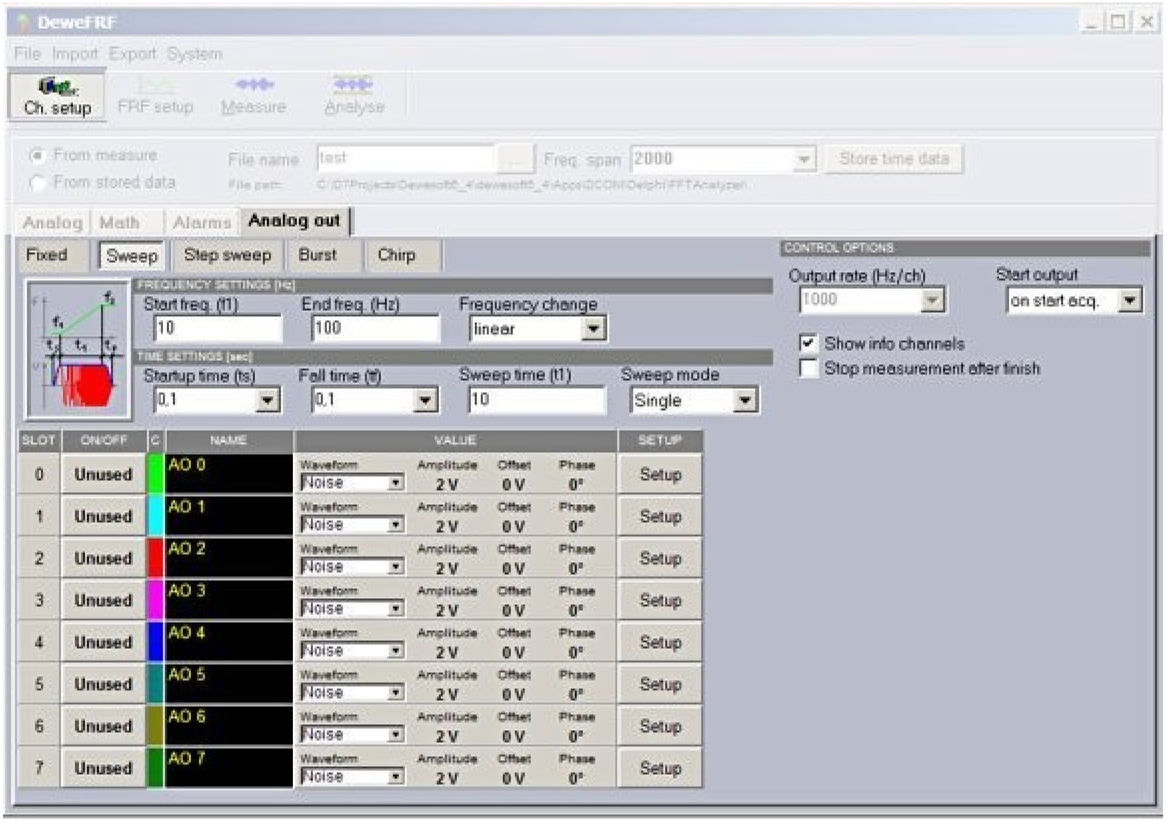
There are two *conventions* how to name the channels that the point *direction*, *sign* and point *index* will be **automatically** *deciphered* from the name. You can use:

- 10x+ convention as the channel name, where at first the number of point index is defined (10 in our case), then the direction (x, y or z) and at last the sign (+ or -);
- 10_(-1) convention as the channel description (the second line of the name), where the first number is again the node index, followed by underscore an open bracket, then the sign and the direction (1 is x, 2 is y and 3 is z and -1 is –x, -2 is –y and -3 is -z) and at last the closing bracket

If any of above conventions is used, point *indexes*, *directions* and *signs* will be **automatically** *assigned* as soon as the channels will be **used** for measurement.

For detailed information see User Guide → **Channel Setup**

Analog output channel setting



With analog output setup we define excitation signals.Few basic forms of excitation signals are useful for modal testing: *filtered noise*, *sine sweep*, *burst* or *chirp*.

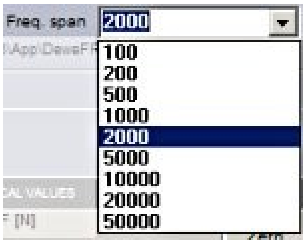
The output rate of the function generator is not related to the sampling rate and should be more than 10 times *higher* then the highest output frequency to avoid harmonic distortion.

Basic forms of excitation signals:

noise	For this 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.
sweep	*Start*, *end frequency* and *sweep time* are defined in the setup before generation. In addition to this sweep can be either linear or logarithmic to provide more cycles at lower frequencies.
burst	For this mode the *rise*, *fall* and *dead time* is defined. Bursts are useful in testing response characteristics of the systems. Within the burst we can choose to have any signal, but in practice noise is usually used as the base generated signal. Bursts are usually enhancements of the noise excitation for measuring transfer function. With burst we eliminate the need of using the window of frequency analysis and therefore eliminate the errors caused by window.
chirp	Excitation is similar to burst, except the *short* sine sweep instead of the noise. Chirp better defines the frequency range than the burst.

For detailed information see User Guide → **Channel Setup**

Frequency span



The *maximum* allowable sample rate will vary according to which A/D board you have installed, and how many channels are activated for recording.

In the drop down list select the appropriate frequency span.

WARNING 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 !
By OFFLINE CALCULATION - From stored data we can't choose the frequency span because it is defined by the sampling rate of time domain data which is loaded. !

1.2 FRF Setup

The **FRF setup** can be selected by clicking the  tool icon:

In **FRF setup** we can define:

Geometry of mechanical structure with nodes, trace lines as well as global coordinate systems

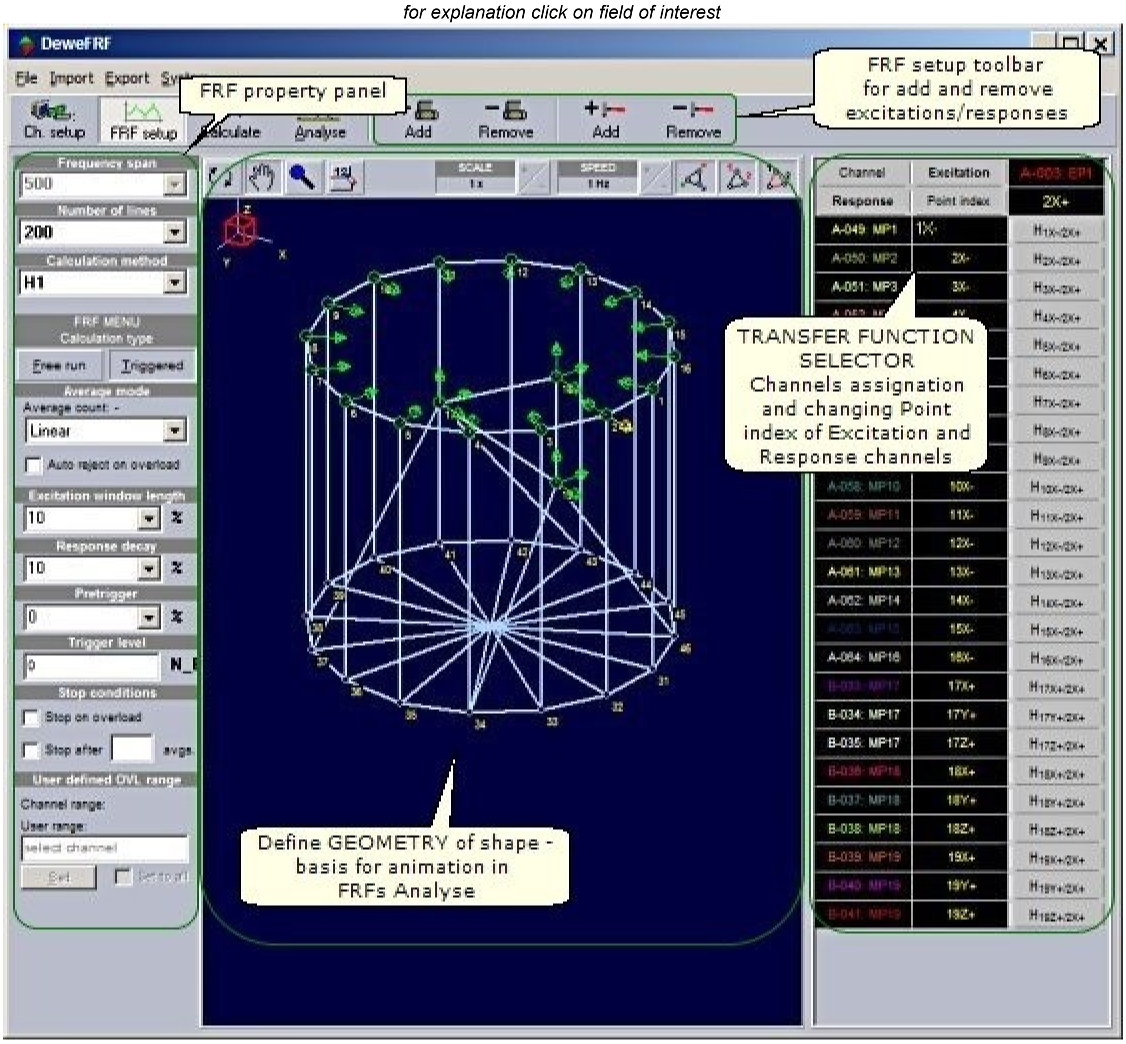
Channels assignment: excitation and responses channel, adding/removing excitations or responses and changing point index

FRF property: frequency span, number of lines, calculation method and FRF Calculation type (free run or triggered acquisition) with all of its property

About Import/Export FRF setup data see → Project management tutorial

When the Channel and FRF setup are done, go to the *next step* → **Measure**

for explanation click on field of interest



FRF property

The property screen is the same as the Measure screen.

1. You can see *general* FRF setup on the left part of this screen (in upper part of FRF property panel), where you can define *frequency span* of measurement, *number of lines* in FRF and *calculation method*.

   The **frequency span** defines the upper limit for calculation and the **number of lines** defines on how many segments this frequency span will be split. In the example above the 500 Hz span is split in 200 lines, resulting in 2.5 Hz frequency resolution.

   Next, the **calculation method** has to be chosen. H1 is commonly used function, H2 can be used only when the number of excitation matches the number of responses, the Hv is the most advanced method which reduces noise on excitations and responses, but takes more time to calculate.

2. On the left part of this screen (in lower part of FRF property panel) you can define *calculation method* in FRF MENU and of its *property* to the set of *free run* or *triggered* acquisition.

There are two main types of *excitation*:

- continuous excitation with *function generator* or *shaker*

  If *noise* or sine *sweep* is used in the function generator, the **Free run** is chosen as the calculation type. We have to define the window because of leakage. *Hanning* or *Blackman* might be a good choice. Since some samples are near to zero, *overlap* has to be defined to get all samples in the final result. **50**% or **66**% overlap is recommended.

  If *chirp* or *burst* is chosen, the **Triggered** calculation is used to acquire only bursts. The response decay can be used is the structure damping is low to reduce amplitudes at the end of calculation window.

  Another application could be measuring electrical transfer functions with direct function generator output.

- impulse excitation with modal *hammer*

  For the impulse acquisition, select **Triggered** excitation. *Rectangular* Window type is used for the output, but the values beyond "excitation window length" settings are trimmed. For responses, exponent down is used with exponent decay as set.

It is not necessary to set the detailed setup of the FRF (from the left side) since this menu will be also shown in the **Measure** screen.

For detailed information see User Guide → **FRF Setup**

Channels assignment

After Channel setup is the next step is to define FRF parameters. At first, the channels are assigned as such that the first channel is excitation channel and the following channels is auto assigned to the responses.

However, the excitation and responses can be changed in terms of *adding* and *removing* excitations/responses (with buttons on FRF toolbar), *changing point index* (always starting by index number like 12 , direction X,Y or Z and sign + or -, combined in for example 12Y+).

Then you can define the excitation channel by clicking on the first column. In this row we also define the *index*, *direction* and *sign* of the measurement point. Generator does not have any effect at this point.

The same is valid for the response channels. Number of response channels is number of all selected channels minus one (used for excitation).

For detailed information see User Guide → **FRF Setup**

Define Geometry

The geometry can be loaded from *.unv* file by selecting Import - Geometry.

On Point Maker screen you can define geometry *nodes* of mechanical structure, trace lines as well as *global coordinate systems* of the nodes are supported. When entering points, *arrow* with *indexes*, *directions* and *signs* will be shown.

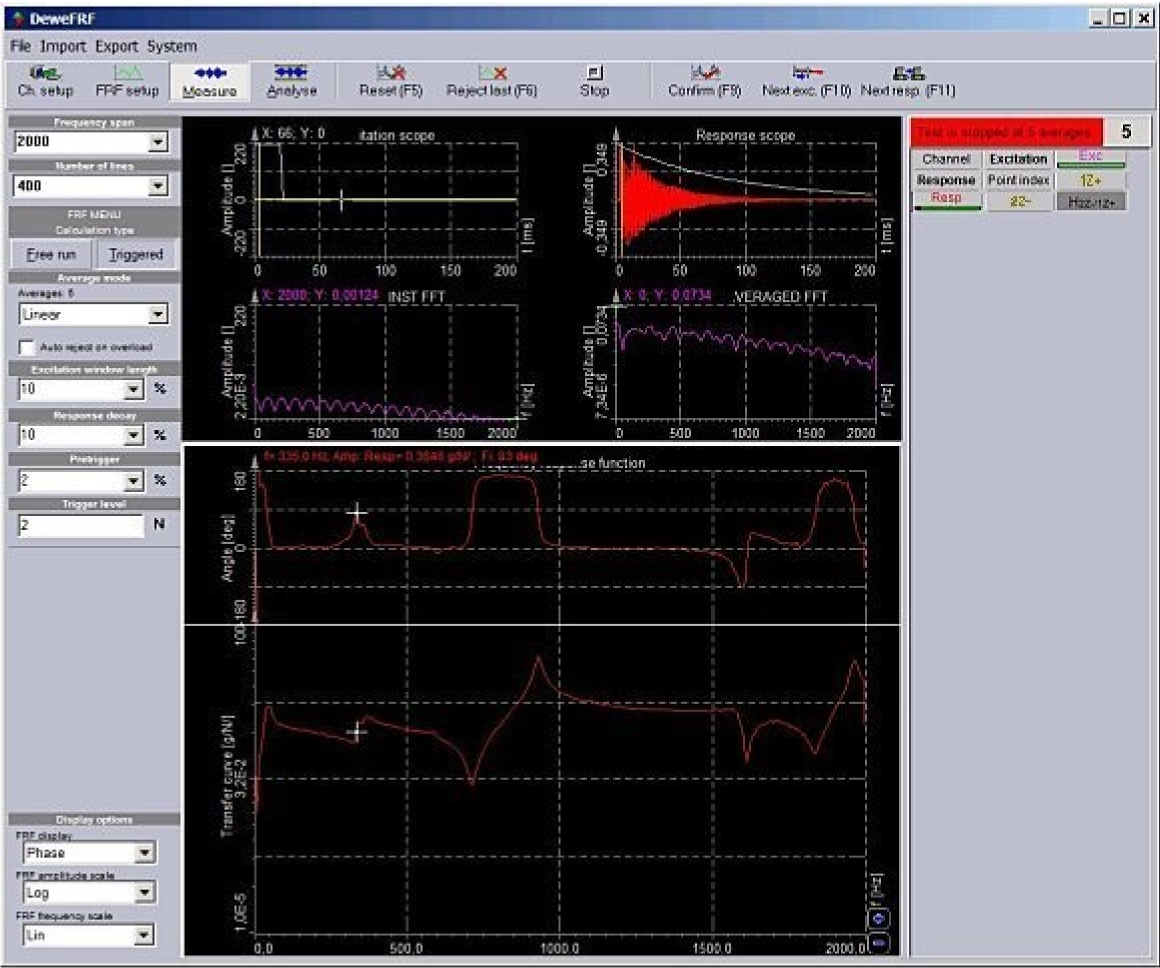
We can left-click and move the mouse inside the geometry to *rotate*, *move* the geometry and to *zoom in* and *out* the geometry.

For detailed information see User Guide → **Point Maker**

1.3 Measure

The **Measure** can be started by clicking the 

FRF tools icon.



The *Measure tool bar* items 

provides a few basic functions for:

- resetting measurements - **Reset (F5)**: to delete *all* the measurements
- rejecting *last* measurement - **Reject last (F6)**: to delete only the *last* measurement from the result
- stopping the measurement - **Stop**, which can be restarted by pressing the **Measure**
- when the data is *correctly* calculated, FRFs can be taken by pressing **Confirm (F9)** button.

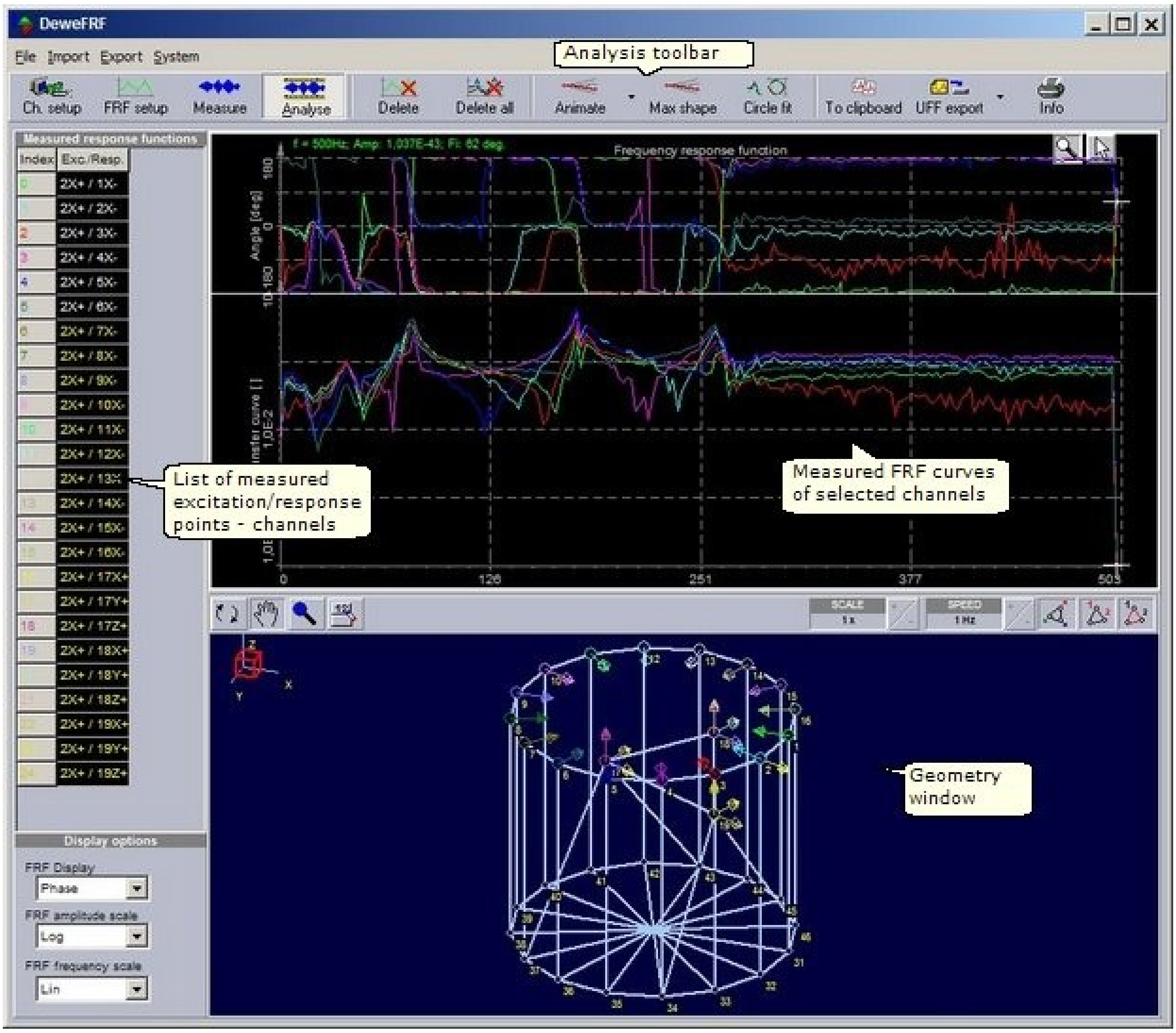
If the *impulse excitation* is chosen, make hits to acquire data. Number of averages are shown on the right side with overload indicator and there are also some controls to monitor hits: *Reset average* will reset number of averages to zero and *Reject last hit* will subtract *last* measurement from the result.

If more channels are to be measured, go back to the **FRF setup**, change the *point indexes*, do another *measurement* and **Confirm** button will add it to all measurement *result*.

For detailed information see User Guide → **Measure**

1.4 Analyse

When FRFs are stored, the **Analyse** window can be started by clicking on the  FRF tools icon or it starts *automatically* when **Stop** button is pressed in **Measure** tool:



In the Analyse window you see all the *measured data*, *FRF result graph* and *all the points* which are measured (*geometry*).

Analysis tools

The FRF *Analysis* tool bar items:

provides a few basic functions:

- When FRFs are stored, the *animation* and *shapes* can be displayed for chosen geometry with pressing on **Animation** Analyse tool icon. The frequency of animation can be chosen with clicking on the graph and the curves which are displayed can be chosen from the list with Shift-click or point and drag with the mouse.
- **Max shape** show maximal deformation of shape.
- **Circle fit** calculation (also Analyse tool) is provided to quickly validate the results and calculate the *damping* of the system.

Measured data

Measured FRF curves of channels can be:

1. EXPORTED to the *universal files* with UFF export menu as *Modal FRF data*, *Coherence*, *Excitation data* or *All data*

2. Copied To clipboard and from this to others SW

For detailed information see: User Guide → **Analyse - Export FRFs**

About Import/Export *Measured data* *see* → **Project management tutorial**

For detailed information see: User Guide → **Analyse**

2 Calculation tutorial

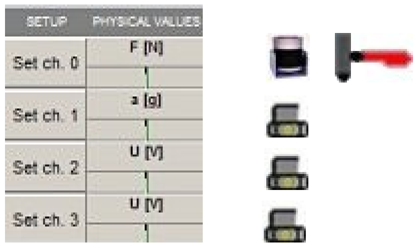
This tutorial will guide you through basic steps to do modal analysis based on *Dewesoft data files*. If number of channels or frequency span is too high to calculate the FRFs *while acquiring data*, we can **stream** the time domain data and later recalculate the FRFs from the streamed file.

This tutorial uses *EimerNoise.dsd* and *EimerGeo.unv* files which can be found on installation CD and can be used to follow the steps in this tutorial.

please click on caption or picture for detailed explanation for the certain step

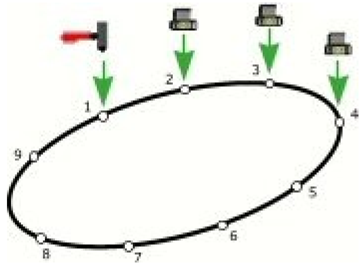
1 STORE/LOAD DEWESOFT TIME DATA

Loads Dewesoft data which will be used for calculation



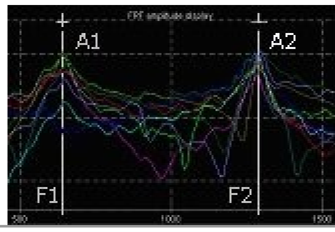
2 FRF POINT & GEOMETRY SETUP

Define geometry and nodes of mechanical structure, assign points and setup parameters for calculation.



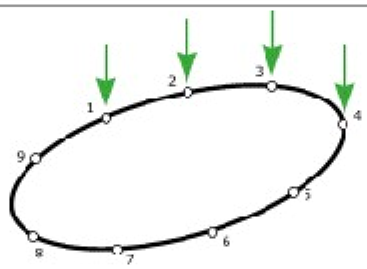
3 CALCULATE

Calculate frequency response data.



4 ANALYSE

Animating shapes, calculate damping and export data to UFF and clipboard



2.1 Channel setup

STORE/LOAD DEWESOFT TIME DATA

The channel setup can be selected by clicking the Ch. setup icon on FRFs tool bar.

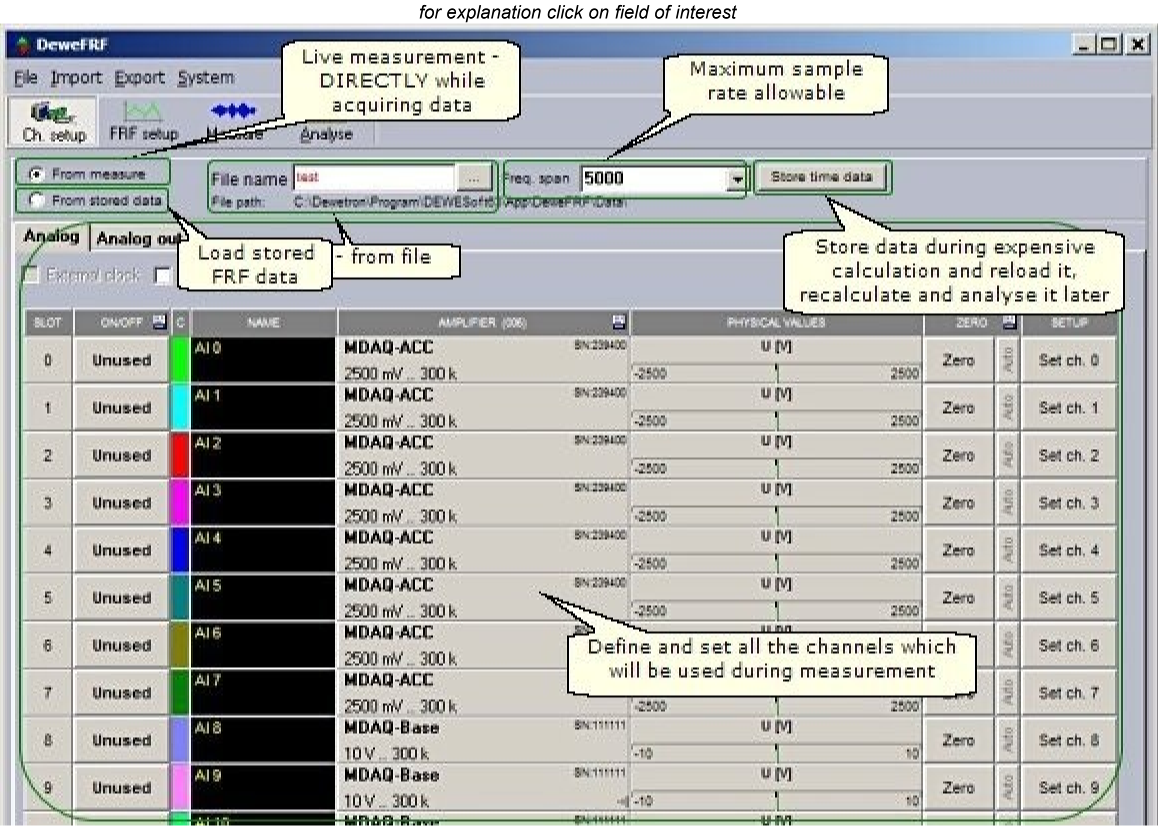
Time *domain* data is stored in Dewesoft format with extension *.*dsd*. We can either *use* - reload already stored data in Dewesoft or we can *store* *.*dsd* file in DeweFRF.

Next step → **FRF Setup from stored data**

Store time data

To store time domain data in Dewe FRF, first *set all the channels* which will be used during measurement in Analog and Analog out tab. The measurement setup can also be *imported* from Dewesoft setup file - *see* → Project management tutorial.

for explanation click on field of interest



After pressing Store time data button on Ch. setup screen we get Warning screen for <u>overwriting</u> file stated in the field File name.

In "File name" field of FRF Ch. we can enter setup screen a new file name and store data under this name!

After pressing **Store time data** button on Ch. setup and Yes on Warning screen we now see this screen:

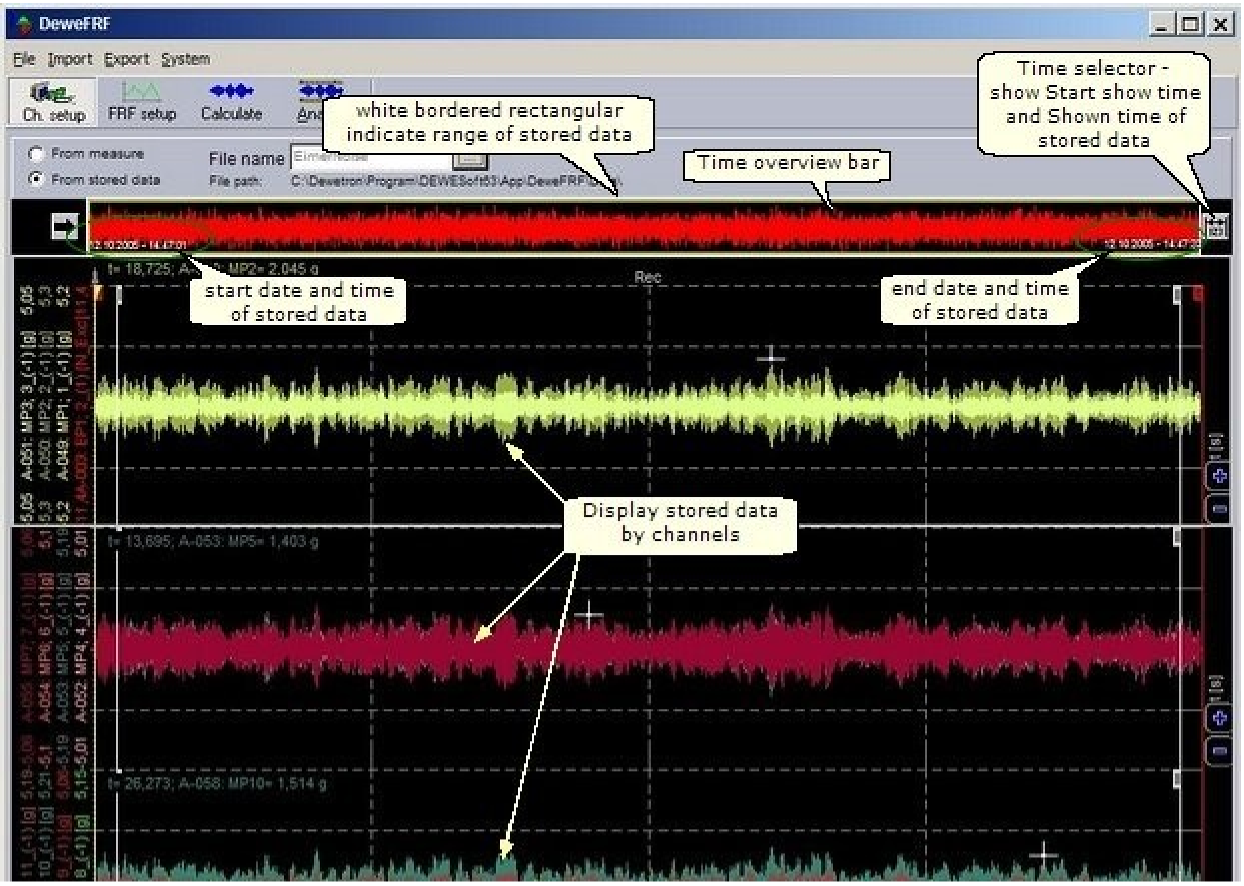
In place of "Store time data" the Stop button appears. By pressing on Stop button we get *display screen of stored data in certain time*.

For detailed information see User Guide → **Data management - Time data**

Reload Dewesoft time data

To reload data, choose From stored data and *define* the File name for loading. The Measure tool button is changed to **Calculate** to show that data will be taken from *stored* file. If the channel setup of Dewesoft includes the *recorder*, the data region of interest can also be chosen by zooming in the recorder.

To follow the steps with demo data, please open *EimerNoise.dsd*. You will get the screenshot as shown below.



For detailed information see User Guide → **Offline calculation**

Frequency span

By OFFLINE CALCULATION - From stored data we can't choose or set the frequency span because it is defined by the sampling rate of time domain data which is loaded. !

2.2 FRF setup

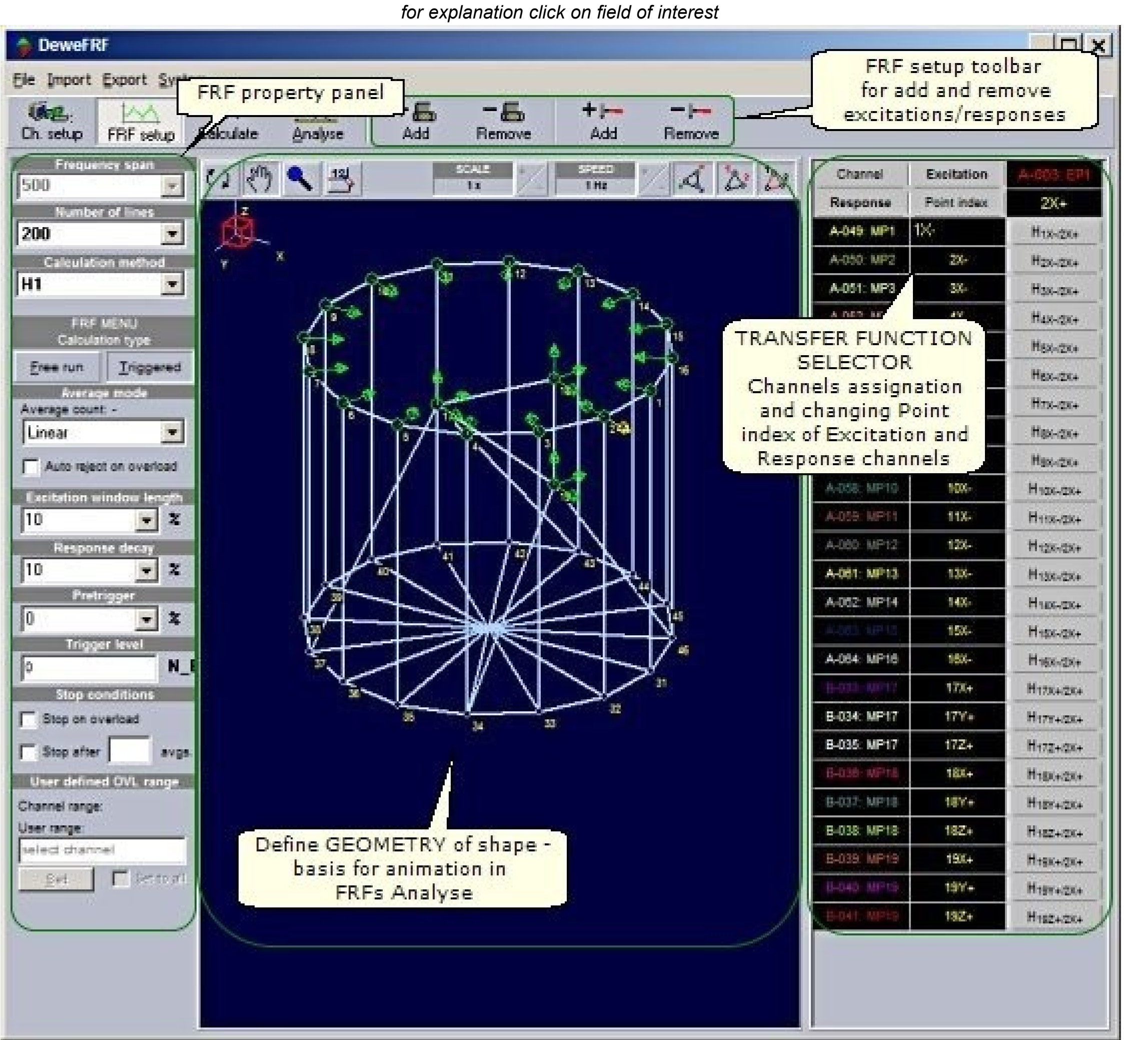
The FRF setup can be selected by clicking the 

tool button.

First it is recommended to *load* or *define* the geometry - *see* → Import/Export Geometry.

In our test example the geometry can be loaded with Import menu → **Geometry** and loading EimerGeo.unv.

for explanation click on field of interest



The geometry in above screen describes the demo structure as seen below:

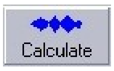
FRF properties

Are the same as with measurement - *see* → Four step tutorial - **FRF Setup**.

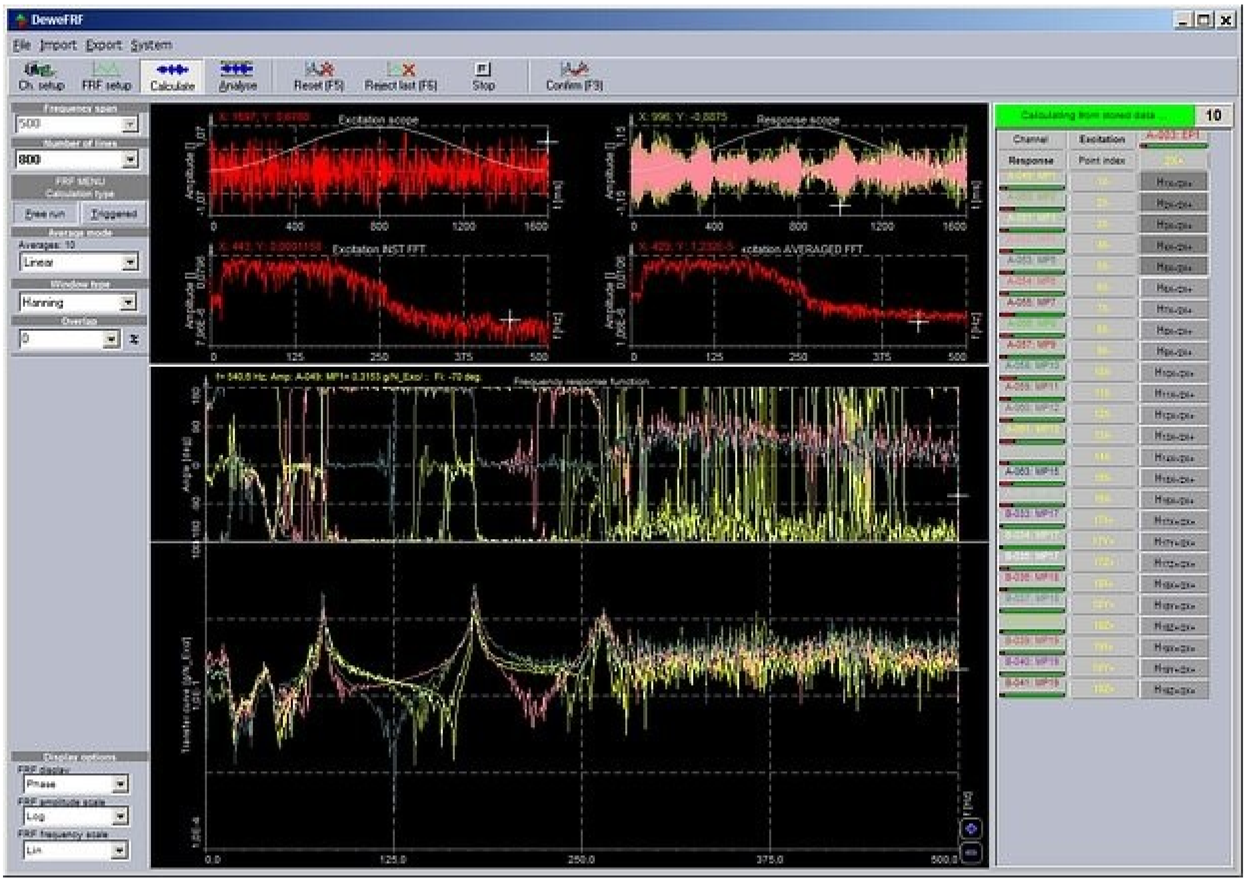
The only thing we can't choose is the *frequency span* because it is defined by the sampling rate of time domain data which is loaded.

In our example select for example 400 lines, choose H1, Blackman window and 50% overlap.

2.3 Calculate

The **Calculate** can be started by clicking the 

FRF tool icon. The Ch. setup *selected range* of the data file will be immediately *recalculated* with *parameters* chosen in FRF setup.



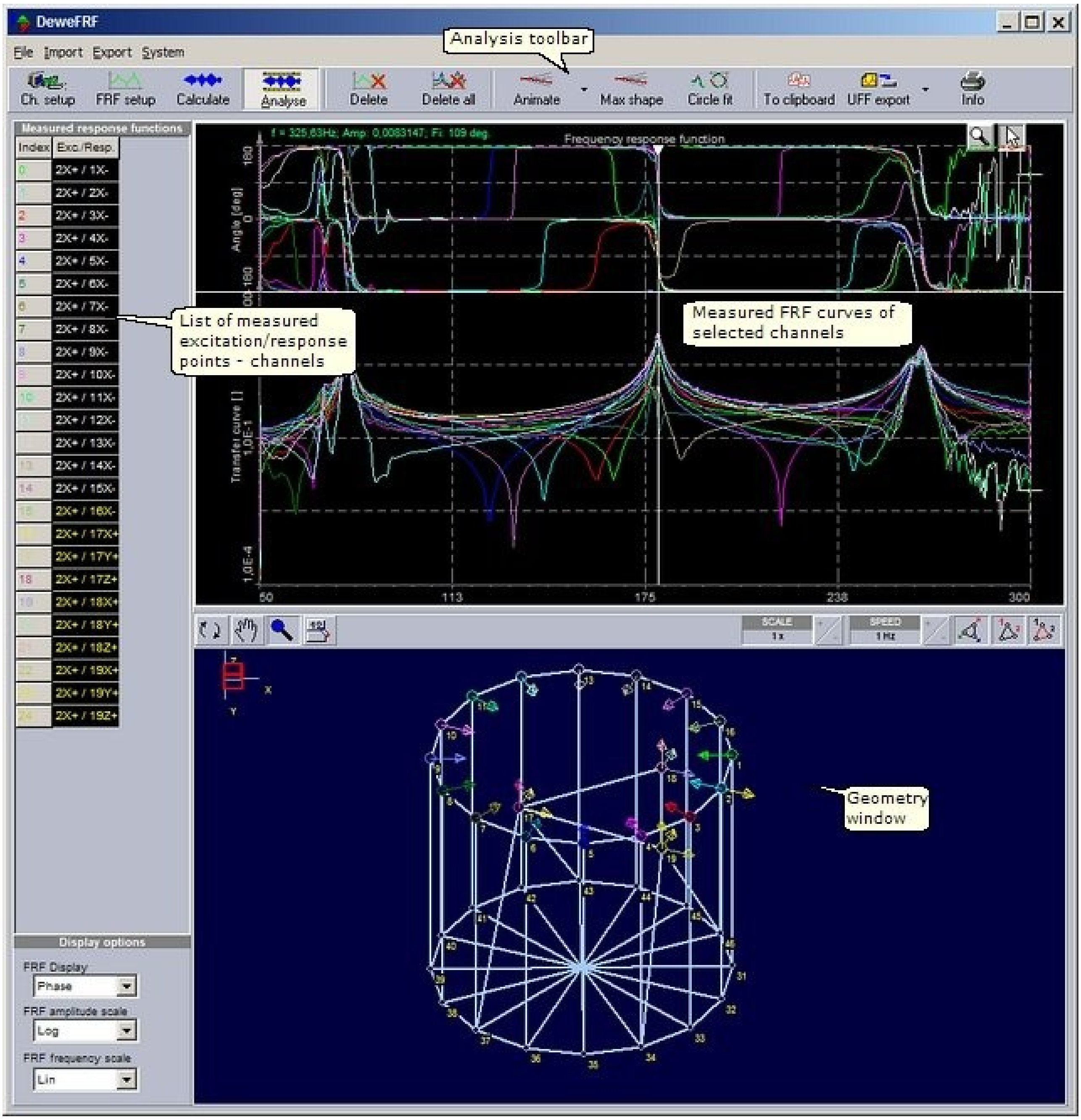
FRF parameters can be furthermore adjusted (except *Frequency span* can not be set) and *FRFs* can be *recalculated* by pressing *again* **Calculate**.

In our case you can play with settings to *optimize* the results. Choose different window (for example Rectangular) to see how important it is to use the correct window.

For detailed information see User Guide → **Offline calculation**

2.4 Analyse

When FRFs are stored, the **Analyse** window can be started by clicking on  FRF tools icon or it starts automatically when **Confirm (F9)** button is pressed in **Calculate** procedure:



In the Analyse window you see all the *measured data*, *FRF result graph* and *all the points* which are measured (*geometry*).

The FRF *Analysis* tool bar items:

provides a few basic functions:

- When FRFs are stored, the *animation* and *shapes* can be displayed for chosen geometry with pressing on **Animation** Analyse tool. The frequency of animation can be chosen with clicking on the graph and the curves which are displayed can be chosen from the list with Shift-click or point and drag with the mouse.
- **Max shape** show maximal deformation of shape.
- **Circle fit** calculation (also Analyse tool) is provided to quickly validate the results and calculate the *damping* of the system.

Measured data

Measured FRF curves of channels can be:

1. EXPORTED to the *universal files* with UFF export menu as *Modal FRF data*, *Coherence*, *Excitation data* or *All data*
2. Copied To clipboard and from this to others SW

For detailed information see: User Guide → **Analyse - Export FRFs**

About Import/Export *Measured data* *see* → **Project management tutorial**

For detailed information see: User Guide → **Analyse**

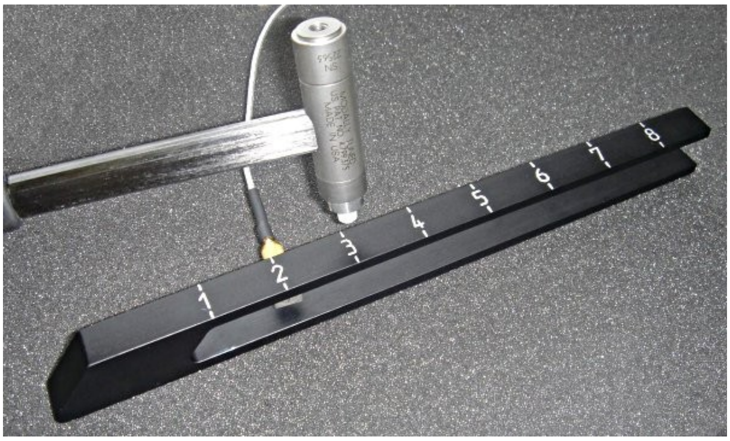
3 SISO - Single I/O modal analysis tutorial

Although providing mathematics to calculate hundreds response channels and multiple excitations, single input - single output test is quite *common* for simple low cost modal testing. It is a little bit **different** in the way that either *excitation* or *response* channel should be moved in order to measure all the points.

SISO → Single Input Single Output

There are *four* basic steps to do SISO modal analysis:

- **Channel setup**
- **FRF setup**
- **Measure**
- **Analyse**



3.1 Channel setup

In the channel setup we define exactly two measurement channel – one is excitation, another one is response. It is helpful to load the geometry to get a feeling where to locate excitation and response.

Hammer

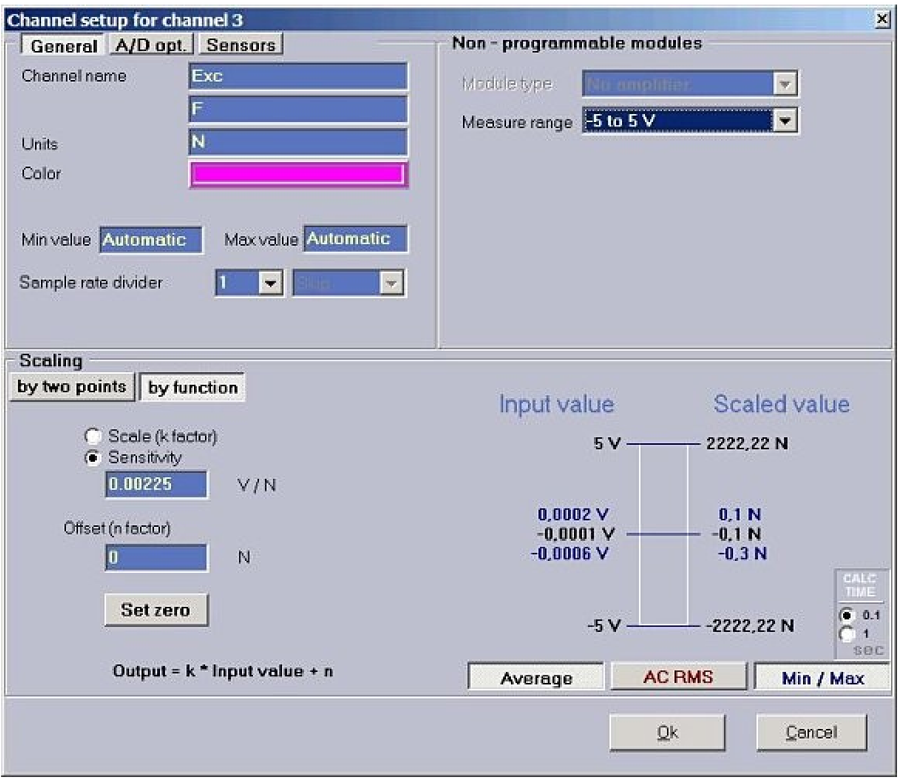
The hammer is usually the excitation for single input single output modal analysis. We have to enter the *scaling factors* for the hammer. The modal hammer comes with *calibration sheet* which looks lik this:

Model Number 086C03	ICP® IMPACT HAMMER			
Performance	**ENGLISH**	**SI**		**Optic**
Sensitivity (±15 %)	10 mV/lbf	2.25 mV/N		for st
Measurement Range	±500 lbf pk	±2200 N pk		**T -**
Frequency Range (-10 dB) (Hard Tip)	8 kHz	~~8000 Hz~~	[1][2]	IEE
Frequency Range (-10 dB) (Medium Tip)	2.5 kHz	2500 Hz	[1][2]	
Frequency Range (-10 dB) (Soft Tip)	750 Hz	750 Hz	[1][2]	**Not**
Frequency Range (-10 dB) (Super Soft Tip)	600 Hz	600 Hz	[1][2]	
Resonant Frequency	≥22 kHz	≥22 kHz		

The important field in the calibration sheet is *sensitivity*. In our example the modal hammer has 2.25 mV/N sensitivity. This means that if we hit with 1 N force, it will produce 2.25 mV at the output.

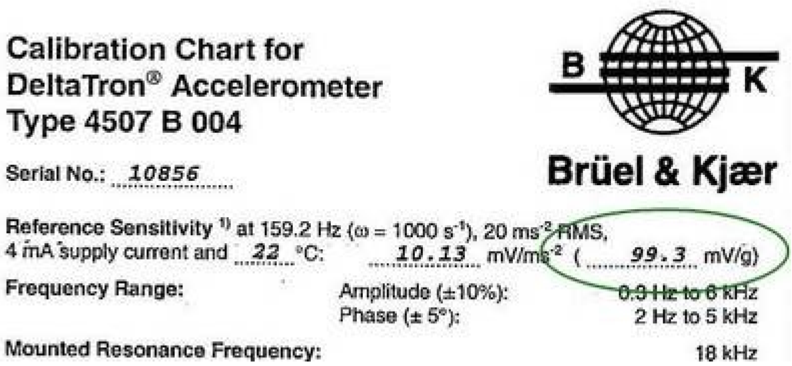
To scale the excitation channel, we first need to enter the measurement units. Since we measure the force in newtons, we enter N as the measurement *unit*. Immediately this unit will be shown in the scaled value preview of the channel setup. Next we have to enter the sensitivity. We go to "Scale by function" tab and choose the **sensitivity** radio button instead of scaling. In the entry field we need to *enter* sensitivity in V/N. Since we have the sensitivity defined in mV/N, we need to convert it to V/N.

In our case 2.25 mV/N = 2.25/1000 V/N = 0.00225mV/N. We immediately see on the right side what is our *scaled* measurement range. In our example the maximum range is 2222 N. This means if we hit with **more** than this force, the channels will *overload* and the measurement will <u>not be valid</u>. If we have amplifiers which supports different ranges (like MDAQ-ACC), we can choose *lower* ranges to make the measurements more precise.

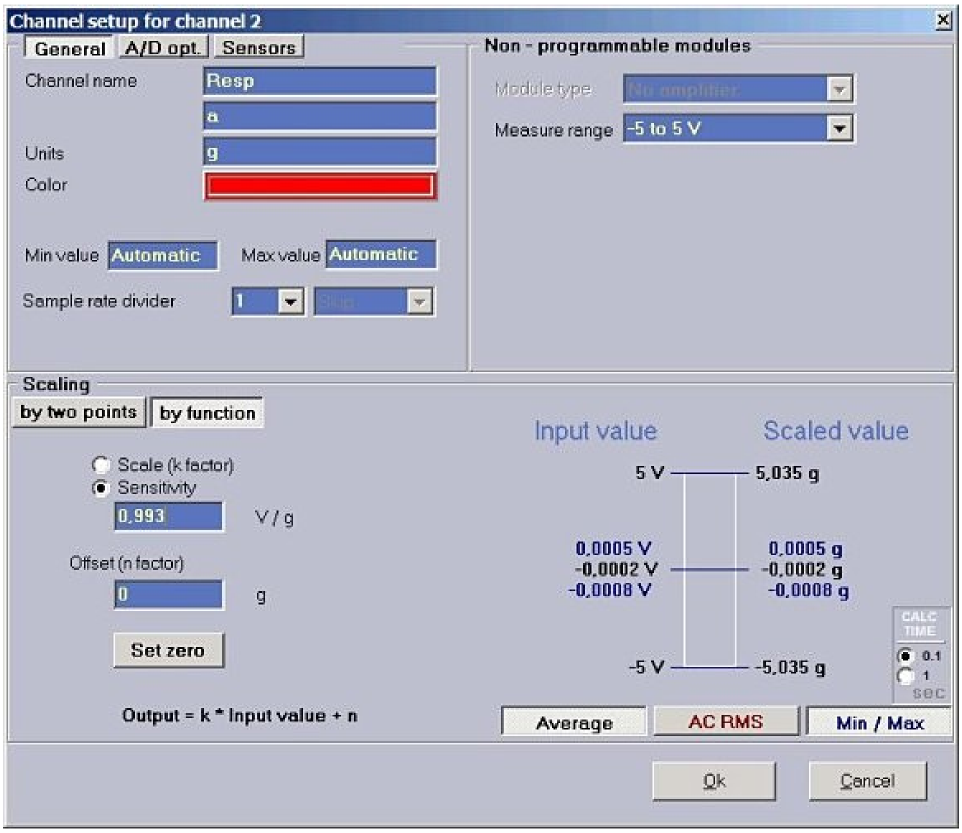


Acceleration sensor (response)

Basically we do the same for the acceleration sensor. The measurement *unit* in this case is acceleration in g, which we enter in units entry field. The sensitivity as we read from the calibration sheet is 99,3 mV/g which corresponds to the 0,993 V/g, which we enter in sensitivity field.



The 5 V measurement range corresponds to the 5,035 g.

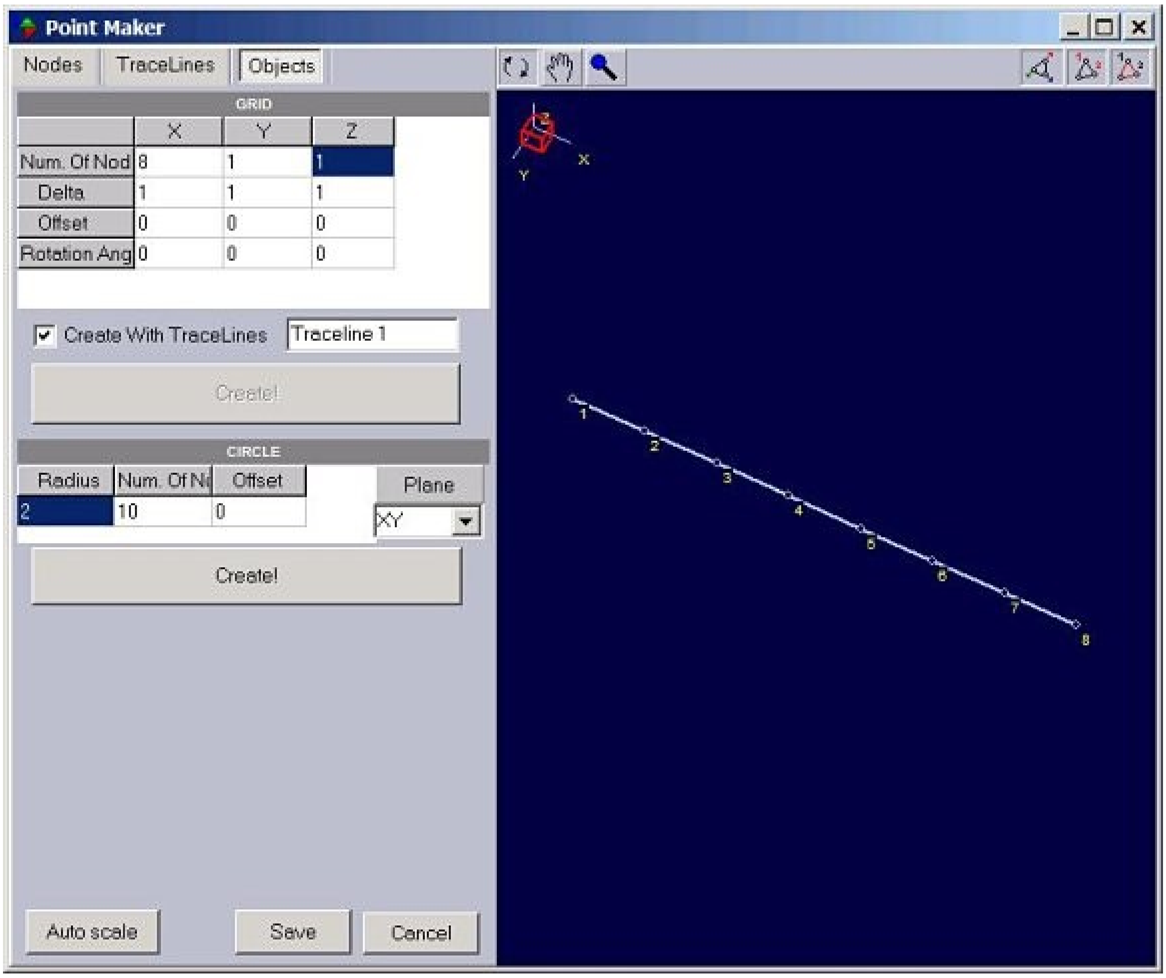


Next step is to define the **FRF setup**.

3.2 FRF setup

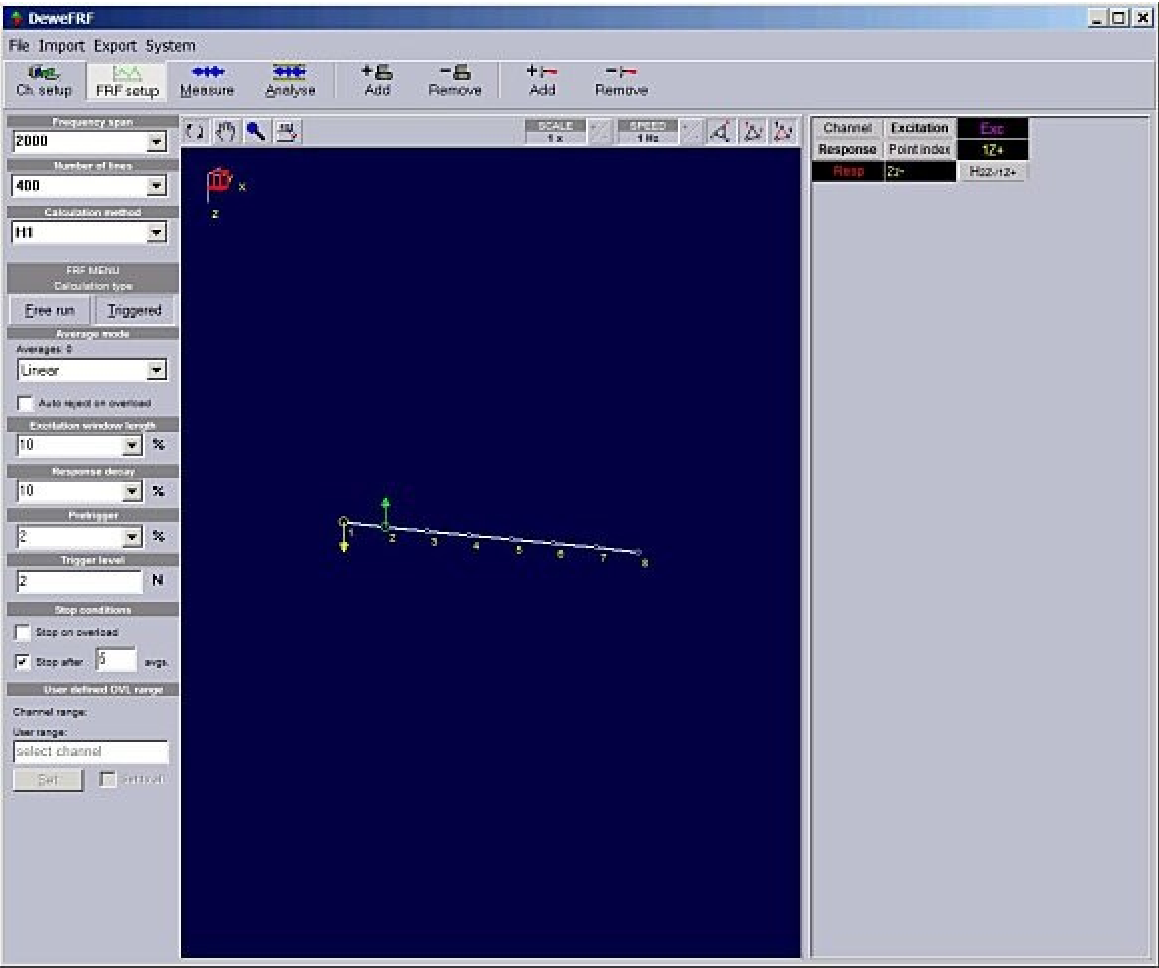
Creating Geometry

For SISO it is very helpful to first define the geometry. In our case it is a simple beam structure with eight points. We can *import* the geometry with Import menu → **Geometry** item or *create* the geometry with File menu → **New Geometry...**. In this case we can go to Objects tab sheet, create the *grid* with **8** points in x direction and 1 point in y and z direction. We check create with tracelines and press Create button. The beam structure is generated. We close the window with Save button.



FRF Setup

In the FRF setup we select the **Triggered** mode. First we define the *bandwidth* of measurement. We have to know that we can achieve only *few* kHz of bandwidth with modal hammer, so more than this is not reasonable. Then we define the *number of lines*. The bandwidth is divided by the number of lines to get the frequency resolution. In our case the *frequency resolution* will be 2000 Hz/400lines=5 Hz.

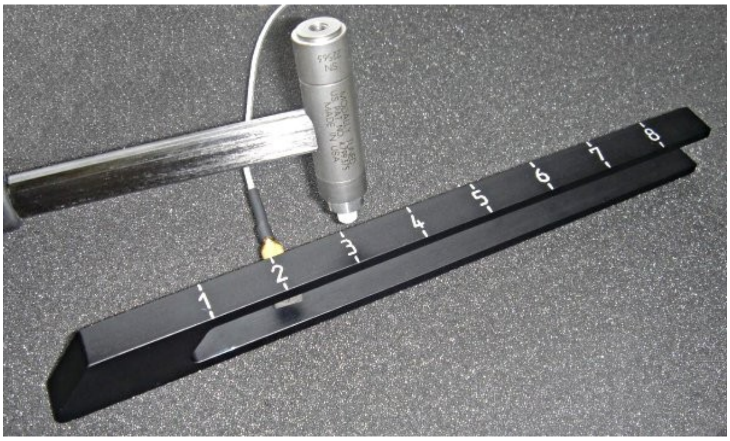


Then we define the Trigger level in N. Since the structure is very light, we need to excite it with a *low* force to prevent double hits. *Few* newtons of force is suitable. In general the *heavier* is the structure, *more* force is needed to excite it.
With **very large** structures and **complex** structures the SISO method is not suitable and we have to look for more sophisticated methods to excite the structure with one or several shakers.

Next we define the Pretrigger. *Few* percent of pretrigger is nice to get also the whole rise time of hit even before the trigger level is achieved. We can also define the number of hits for one measurement point.

In our case we have chosen to put the **sensor** on the bottom of the beam on point 2. The reason for this is that we need to measure *mobility point*.
Mobility point is the point where the excitation and response is the same. If this point is not measured, we can't calculate excitation/response matrix in full (we need this for animation and modal parameters calculation).
The point 2 is chosen because we don't want to load too much the structure with the mass of the sensor. If it would be on point 8, it would affect *more* the natural frequency of the beam.

In order to be successful, we need to provide a good *surface* to place our item under the test. Ideally the structure should be *free* in the space not to have a contact with any other element. Since we usually don't have a chance to do our modal testing in the orbit, we have to make some compromises. There are two ways of doing this: first is to put the structure on some *foam* or light springs with as low natural frequencies as possible. Second way is to tie the structure to the *heavy* structure with known natural frequencies far away from what we are interested in. What helps in this case is that we can't really excite a big table with low force needed to excite the light beam as we have.



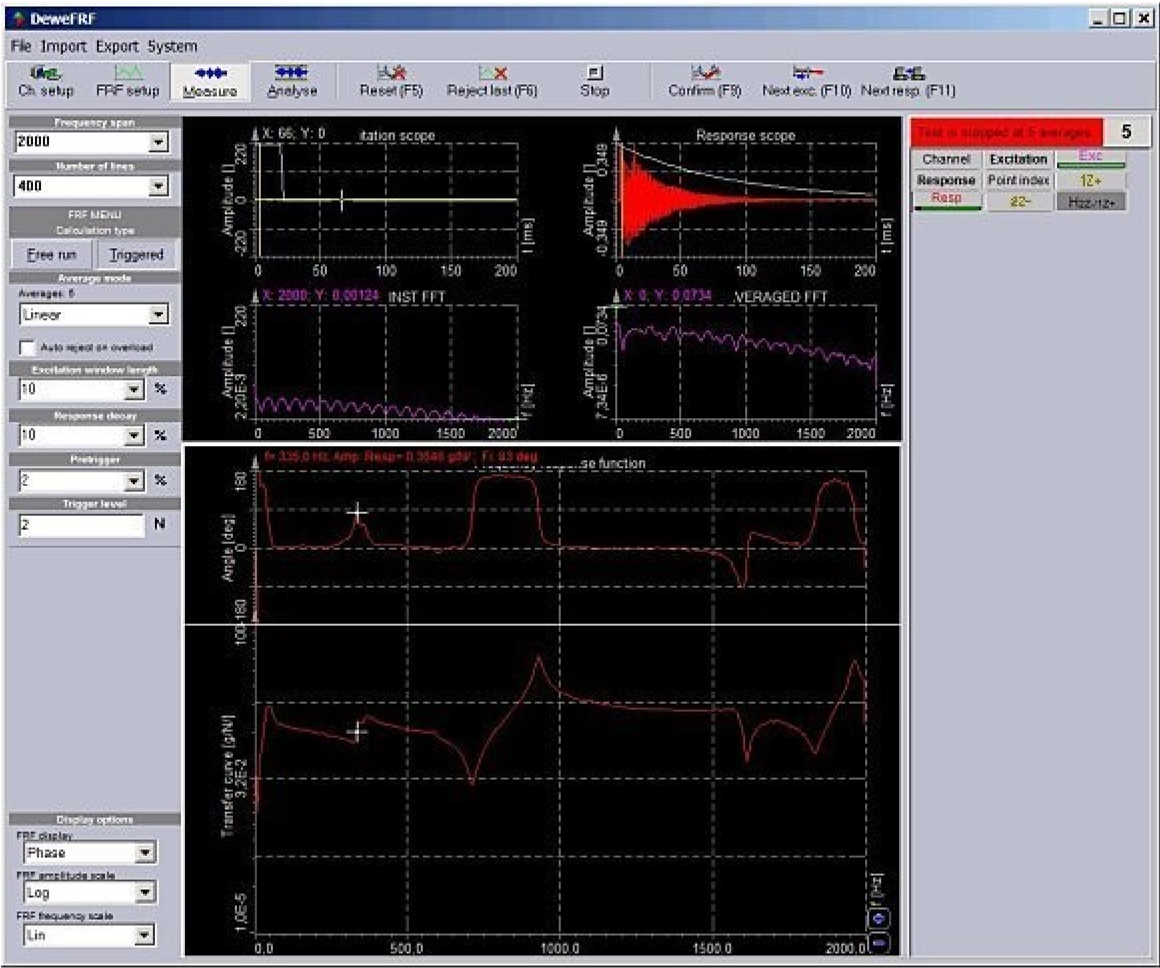
Now we have to decide for the *strategy* of doing the test. In general we could move the *excitation* points or points of measurement. In the eyes of results there is no difference as long as we measure the mobility point. But from the structure point of view it is far better to move the excitation point, because with moving the response we would change the property of the beam with mass loading. Natural frequencies would shift a little and we would never get as good result as we would move the excitation.

Our first excitation point will be 1z+ (since we have to start from beginning) and our response point will be *always* **2z-**.

Now we can proceed with **Measure**.

3.3 Measure

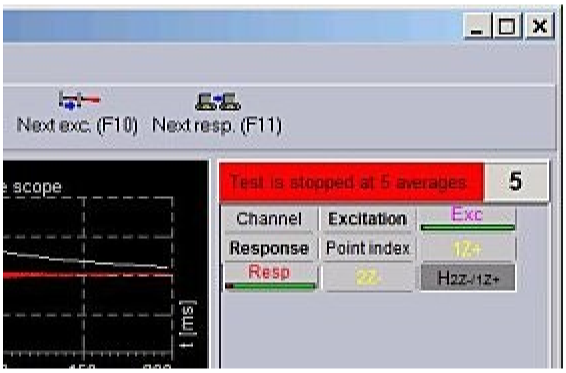
We position the modal hammer on point 1 and *make* the hits.

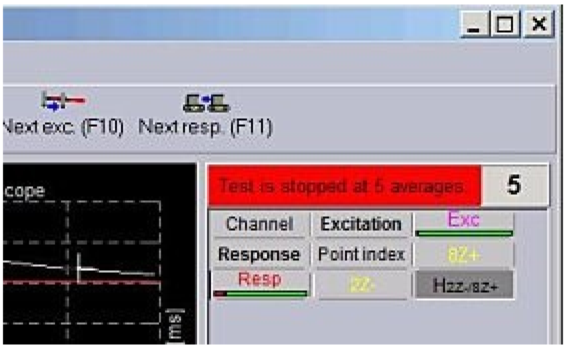


When first point is measured, we have several *controls* to continue the test – like always we have:

- **Reset (F5)** (to delete *all* the measurements),
- **Reject last (F6)** (to delete only *last* measurement from the result if the hit was inadequate),
- **Stop** (to stop measurement, which can be restarted by pressing the **Measure**),
- when the data is *correctly* calculated, FRFs can be taken by pressing **Confirm (F9)** button.

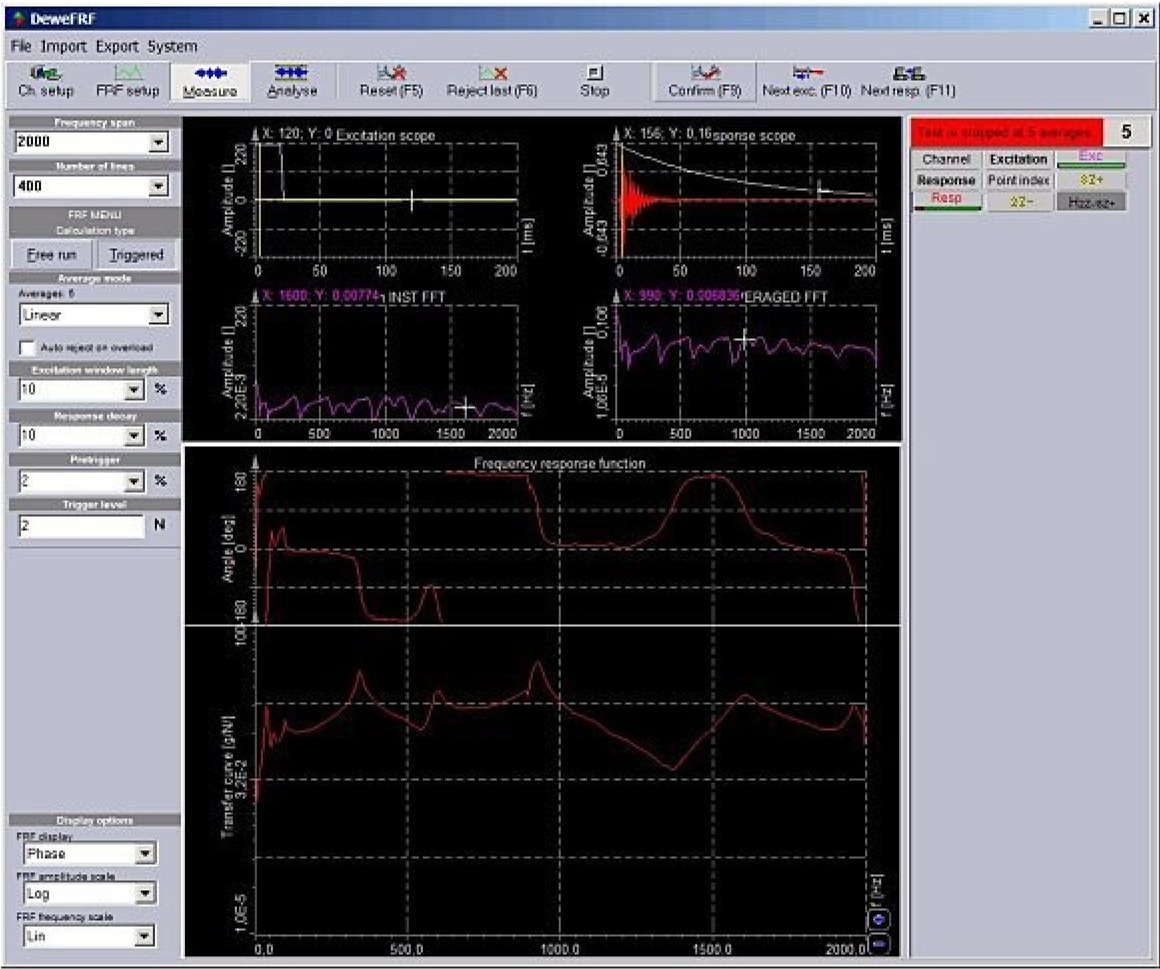
On the right side of the screen where the *list of measurement points* is shown, we can immediately see which points are currently valid. If we define the number of hits, the bar will be shown in red to notice that the measurement is finished.





There are two more controls which appear only if just one *excitation* and *response* channels are defined:

- first option is **Next exc. (F10)** , which confirms the measurement and *moves* the point index so that *excitation* point is incremented by one. This allows us to *immediately* measure *next* point on the structure.
- **Next resp. (F11)** does the same, but it will *move* to the next *response* point.



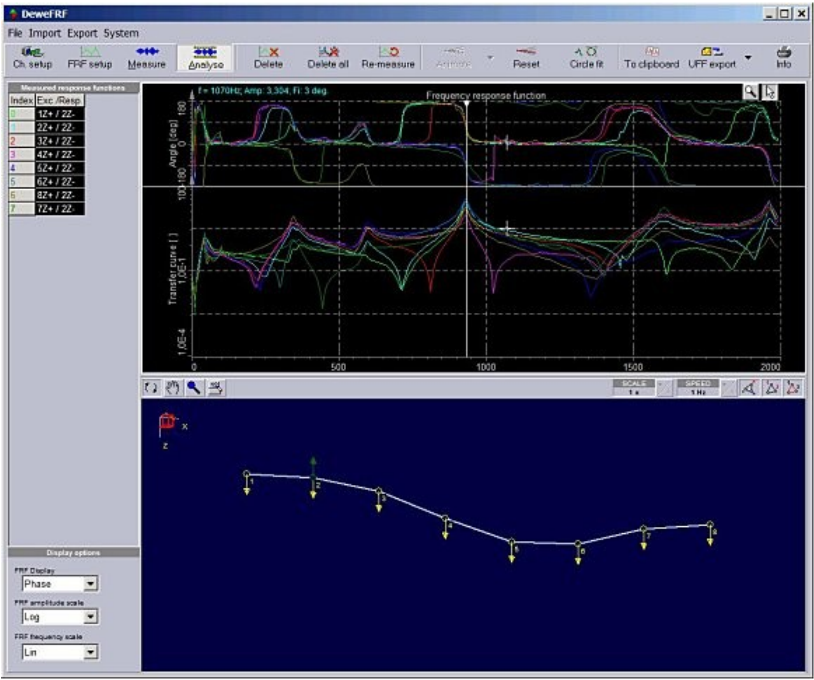
In our case we decided to move the *excitation* point. We press **Next resp. (F11)**. The table on the right side will show 2z+ as the next measurement point indicating that we should *next* hit on point 2. We do all the measurement in the same fashion up to point 8 to get all the FRFs. For the last measurement we press **Confirm** to go to **Analyse** mode.

3.4 Analyse

The analysis is the same as described in basic tutorial Analyse.

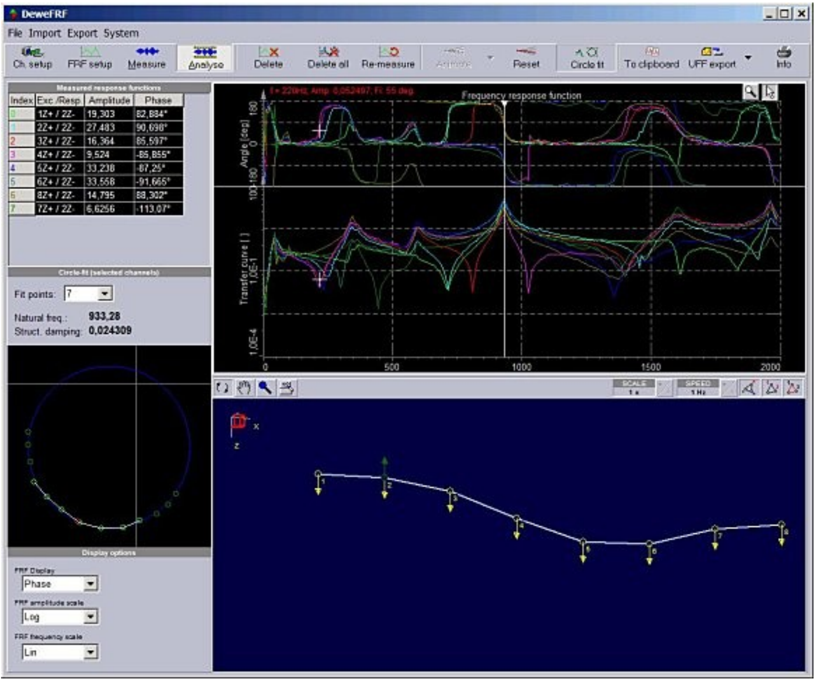
The FRF Analysis *tool bar* has additional possibility for SISO measure model. Single FRFs which have inadequate results can be removed and measured *again* by pressing **Re-measure** button.

In this **animation** it is clearly seen that point the point **7** is not measured perfectly. We have to choose the faulty FRF from the list on the left side and *choose* **Re-measure** to go back to Measure mode. After taking *new* measurement on point 7 we press Confirm to *add* this measurement to the results. The incorrect measurement will be *replaced* with the new one.



The **Animation** helps us to immediately see the quality of measurement results.

Each natural frequency can be further examined with **Circle fit**. For modes which are spaced *far* from each other, circle fit will yield quite good idea of the *exact natural frequency* and the *structural damping* of the certain modes.



As with all other tests, the FRFs can be *exported* to the universal files or *copied to clipboard* - *see:* User Guide → **Analyse - Export FRFs**.

4 Project management tutorial

In Dewesoft FRF there are many ways to Import - Open or Export - Save setup, geometry and measured data.

type of file	type of data	action
Project *.dfs (*Dewesoft FRF project)* data file	- **Channel Setup** Setting of all analog channels for excitation and responses - **FRF Setup** Assigned points of structure to channels and setup parameters for calculation - **Geometry** Defined geometry and nodes of mechanical structure *! Project doesn't include measured data*	**New / Open** **Save / Save as**
Measurement Setup *.dss (*Dewesoft setup data file*) or *Dewesoft text setup* file	- **Channel Setup** Setting of all analog channels for excitation and responses	**Import** **Export**
Geometry *.unv (*Universal file format*) data file	- **Geometry** Defined geometry and nodes of mechanical structure	**New / Import** **Export**
Modal FRF Data *.unv (*Universal file format*) data file	- **Measured data**	**Import** **Export**

4.1 Project file

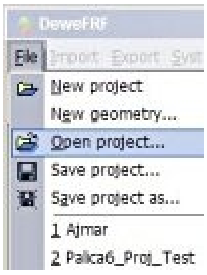
In " Project ", Channel setup data, FRF Setup data and Geometry data are stored.

New project

We can open ***New*** *Project - Channel setup* by selecting:

File menu → **New project**

Open project



We can directly import *existing Project - Channel setup* data by selecting:

File menu → **Open project...** .

(in standard Open window we choose name of existing project)

or by click on:

File menu → *name of recent project*
(by this option project opens *automatically*)

Basic Channel setup screen is *displayed* for defining data and setting of analog channels which will be used during measurement: excitation and sensors (in Analog and Analog out tab)

Define *new Project* data.

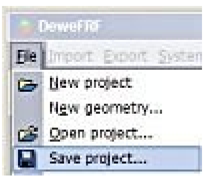
Modify *existing Project* data.

For detailed information see User Guide → **Import / Export**

Project file → **Save Project**

In " Project " file Channel setup data, FRF Setup data and Geometry data are stored. After define/modify *Project data* we can save this data in "project".

Save project...



We can save existing or new *Project - Channel setup data* by selecting:

File menu → **Save project...**

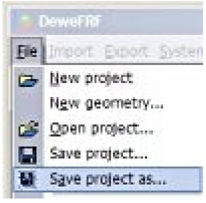
by New project:

By *new* project Standard Save As *window* is displayed - *see "Save project as..."* below.

by Open existent project:

Warning: *existing project is saved without warning under same project name and overwrite its data.*

Save project as...



We can save the *existing* (modified) *Project - Channel setup data* under a new name by selecting:

File menu → **Save project as...**

Standard Save As *window* is displayed for save *Project data* to *.*dfs* (*Dewesoft FRF project)* data file.

For detailed information see User Guide → **Import / Export**

4.2 Measurement setup file

In " Measurement setup " file Channel setup data are stored.

Import

We can also directly Import - Load Measurement setup data from *.*dss* (*Dewesoft setup file*) or *Dewesoft text setup* data file by selecting:

Import menu → **Measurement setup**

Export

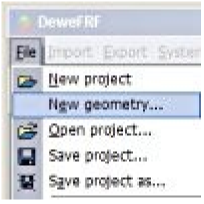
We can Export -Save *existing* (modified) or *new* *Measurement setup data* by selecting:

Export menu → **Measurement setup**

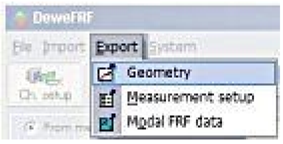
For detailed information see User Guide → **Import / Export**

4.3 Geometry

You can define the **Geometry** of mechanical structure for FRF testing as:

New Geometry	**Import** Geometry
 we can define ***New*** *Geometry* by selecting: File menu → **New Geometry...** and Point Maker screen in Ch. setup tool is displayed for define *new* geometry.	 to load ***existing*** (previous defined and saved geometry of mechanical structure) from *universal files* into Point Maker screen select: Import menu → **Geometry**

Export Geometry



After defining new or modified existent we can Export - Save *Geometry* by selecting:

Export menu → **Geometry**

Standard Save As window is displayed for save *new* or *modified* Geometry to *.*unv* (*Universal file format*) data file.

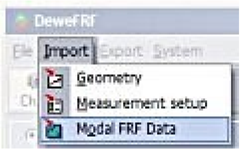
WARNING *because* Geometry *and* Modal FRF Data *is stored in same* *.*unv* (*Universal format*) *file it is strongly recommended to use separate filename for new or modified* Geometry.

For detailed information see User Guide → **Import / Export**

4.4 Modal FRF Data file

In " Modal FRF data" measured data are stored.

Import



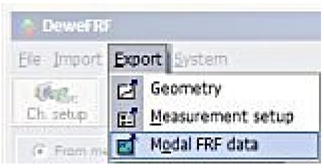
to load *existing* (previous saved) measured data from *.*unv* (*Universal file format*) data file into Analyse screen select:

Import menu → **Modal FRF Data**

In both case **Save FRF data As** window is displayed for saving *new* or *modified* measured data to *.*unv* (*Universal file format*) data file.

Export

for saving *new* or *modified* measured data file we have two options:



Export menu → **Modal FRF data**

UFF export on Analyse toolbar → **FRF data**

(here we have feasibilities for selecting another sort of data - eg *Coherence*, *Excitation data*, *All data*)

In both case **Save FRF data As** window is displayed for saving *new* or *modified* measured data to *.*unv* (*Universal file format*) data file.

WARNING *because* Geometry *and* Modal FRF Data *is stored in same* *.*unv* *(Universal format)* *file it is strongly recommended to use separate filename for new or modified* Geometry.

For detailed information see User Guide → **Import / Export**

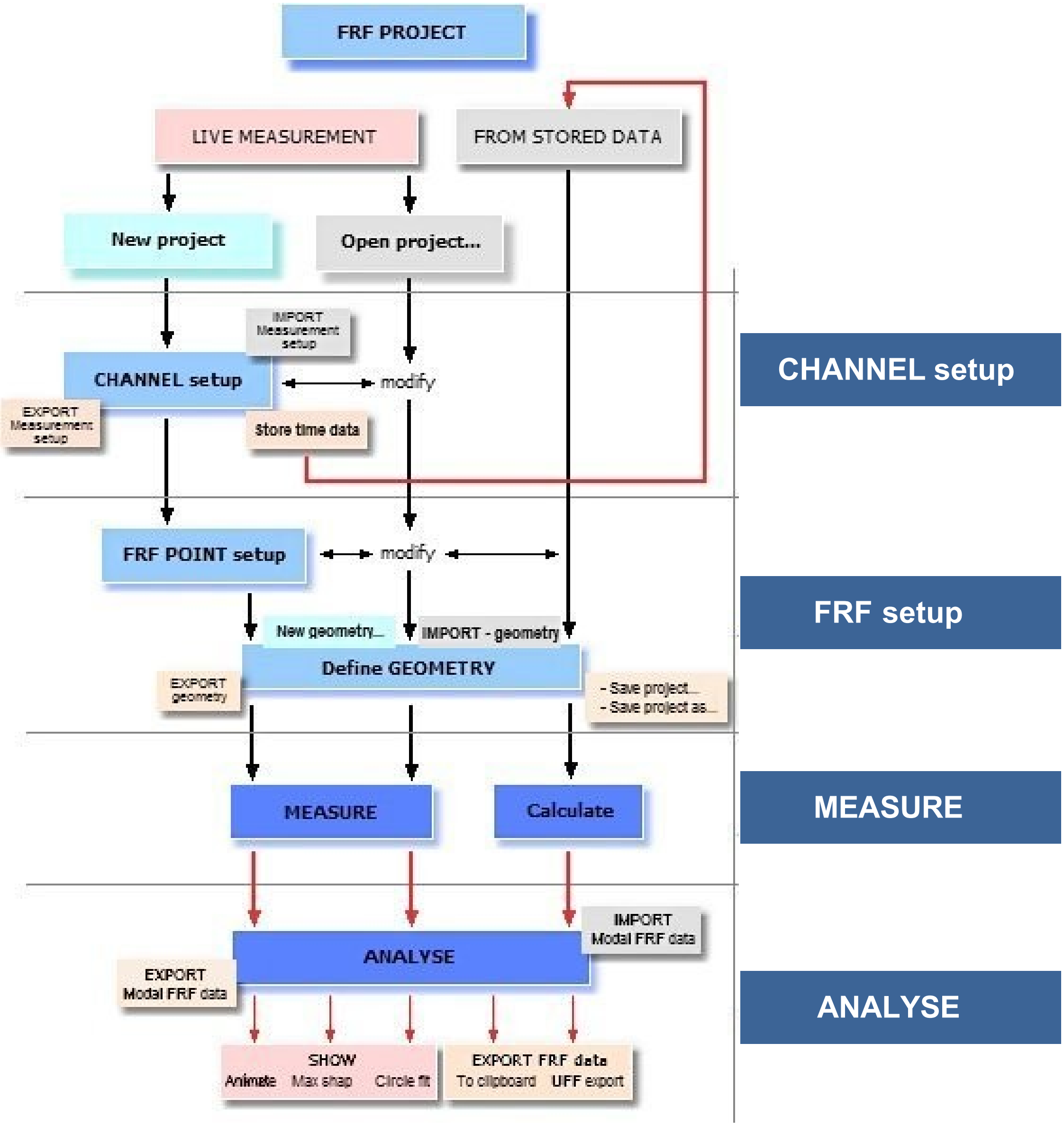
Part III

User Guide

Welcome to the **USER GUIDE** of **DeweFRF** modal analysis software. Before trying to setup the measurement, let's notice that there are **two methods** (ways) of making the FRFs measurement.

Process of each FRF measure consist of **four basic steps** and is introduced in following flowchart as **Project**:

For explanation move mouse on area / activity of interest and click



This guide also contains detailed information about:

Offline calculation - reload, calculate and analyse *stored data*

DeweFRF software interface

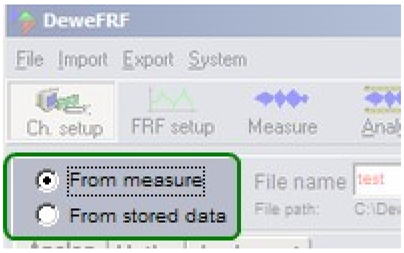
Data management - DeweFRF have many ways to Import / Export *setups*, *geometry* and *measured* data.

1 Measure methods

Let's notice that DeweFRF software enables **two methods** (ways) of making the FRFs measurement:

- first way is to measure the FRFs *directly* while acquiring data and the
- second way is to measure and calculate respectively *offline* from stored data.

From measure operation mode → **direct** measurement

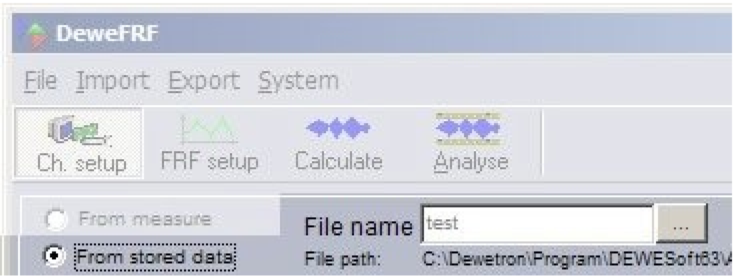


In this mode FRFs measurement is done directly while acquiring data. This is most commonly used operation mode and is selected by default and doesn't require any additional action.

There are two main types of **measurement** depended on type of *excitation*:

- continuous excitation with *function generator* or *shaker*
- impulse excitation with *modal hammer*.

From stored data operation mode → **offline calculation**



Another way for calculation is to load the Time data file. The way to calculate is exactly the same, except that the data is not *acquired*, but loaded from file.

Loading stored FRFs data can be done on Ch. setup window by select of From stored data choice.

For detailed information see: **Offline calculation**

Decision what to do depends on the user (if the archive time data is needed) and on the system performance. The system benchmarks might differentiate from system to system, but with recent computers (3 GHz P4) it is possible to measure online 256 channels at few kHz bandwidth.

2 Data management

In Dewesoft FRF there are many ways to Import - Open or Export - Save setup, geometry and measured data.

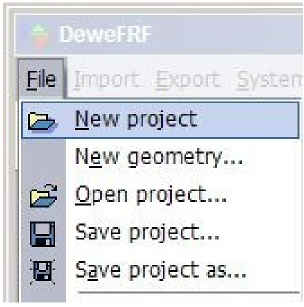
type of file	type of data	action
Project *.dfs (*Dewesoft FRF project)* data file	- **Channel Setup** Setting of all analog channels for excitation and responses - **FRF Setup** Assigned points of structure to channels and setup parameters for calculation - **Geometry** Defined geometry and nodes of mechanical structure *! Project doesn't include measured data*	**New / Open** **Save / Save as**
Measurement Setup *.dss (*Dewesoft setup* data *file*) or *Dewesoft text setup* file	- **Channel Setup** Setting of all analog channels for excitation and responses	**Import** **Export**
Geometry *.unv (*Universal file format*) data file	- **Geometry** Defined geometry and nodes of mechanical structure	**New / Import** **Export**
Modal FRF Data *.unv (*Universal file format*) data file	- **measured data**	**Import** **Export**
Time data *.dsd (*Dewesoft* data *file*)	- **measured data**	**Load time data** **Store time data**

WARNING *because* Geometry *and* Modal FRF Data *is stored in same* *.unv *(Universal format) file is strongly recommended to use different filename for this new or modified data file -* Geometry/Modal FRF Data

2.1 Project file

In **Project** *.*dfs* (*Dewesoft FRF project)* file **Channel setup** data, **FRF Setup** data and **Geometry** data are <u>stored</u>.
! does not include *time* or FRFs - *measured data*

New project



We can <u>start</u> with a *new* project by selecting:

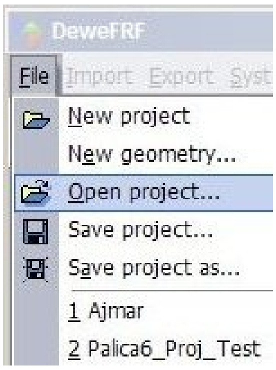
File menu → **New project...**

and Channel setup screen is *displayed*.

You can define all *new* Project data (Channel setup data, FRF Setup data and Geometry data).

Don't forget to save Project after making a setup.

Open project



We can directly <u>import</u> an *existing* project by selecting:

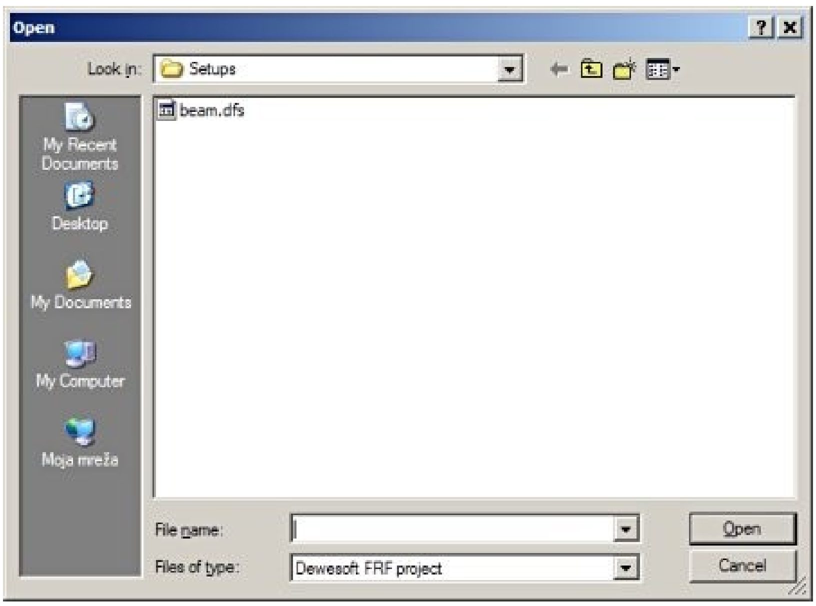
File menu → **Open project...**

or by click on:

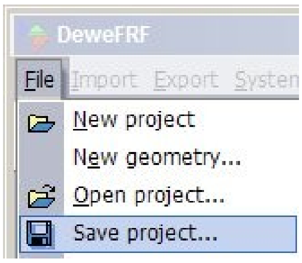
File menu → *name of recent project*

(by this option project opens *automatically*).

By selecting the first option, the standard *Open window* is displayed to <u>load</u> *existing* Project project from *.*dfs* *Dewesoft FRF project* data file:



Save project... (Export)



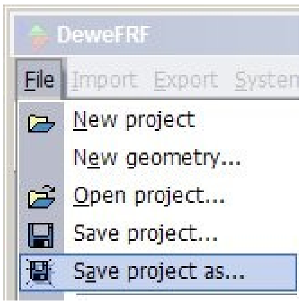
We can save *existing* or *new* project by selecting:

File menu → **Save project...**

By *new* project Standard *Save As window* is displayed - *see "Save as".*

Warning: existing project is saved under same project name and overwrite previous data.

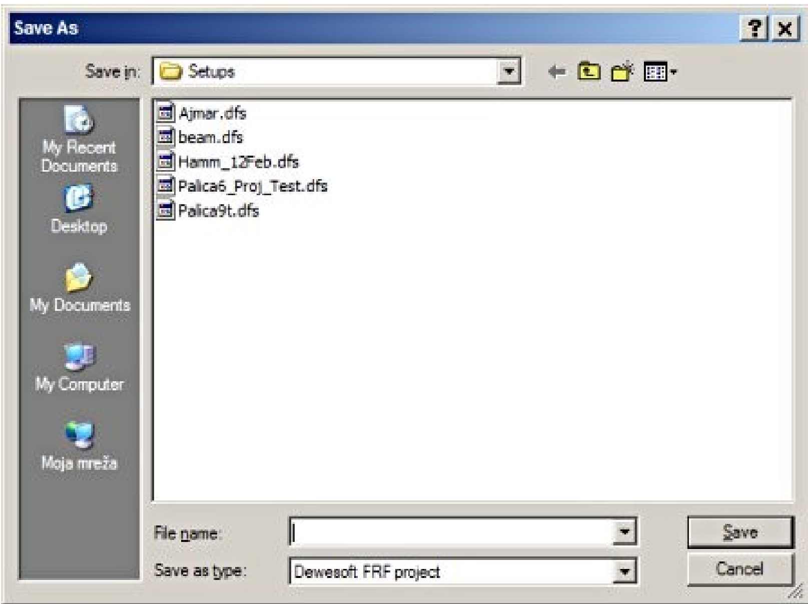
Save project as... (Export)



We can save *existing* (modified) project to the new setup file by selecting:

File menu → **Save project as...**

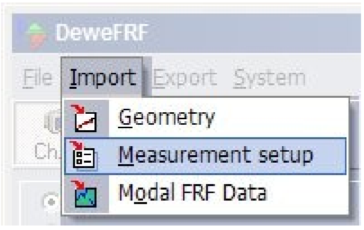
Standard *Save As window* is displayed for save *new* or *modified* Project to *.*dfs Dewesoft FRF project* data file:



2.2 Measure setup file

In **Measurement setup** *.*dss* (*Dewesoft setup file*) or *Dewesoft text setup* data file Channel setup data are stored.

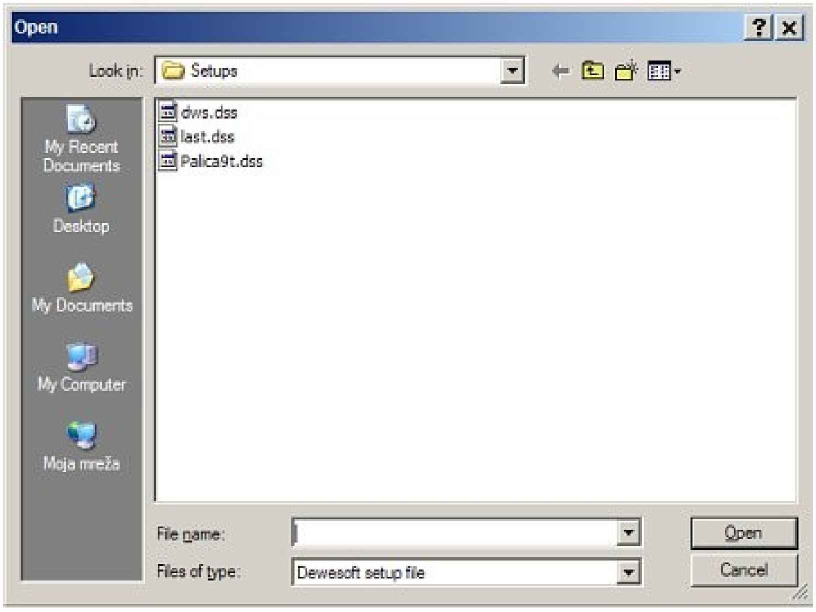
Import



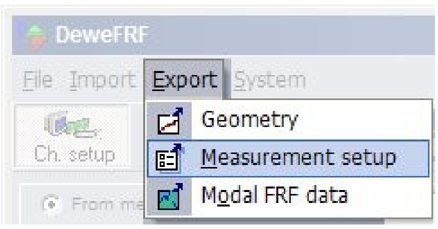
We can also directly Import - Load Measurement setup data from *.*dss* (*Dewesoft setup file*) or *Dewesoft text setup* data file by selecting:

Import menu → **Measurement setup**

The standard *Open window* is displayed for loading *existing* Measurement setup from *.*dss* file:



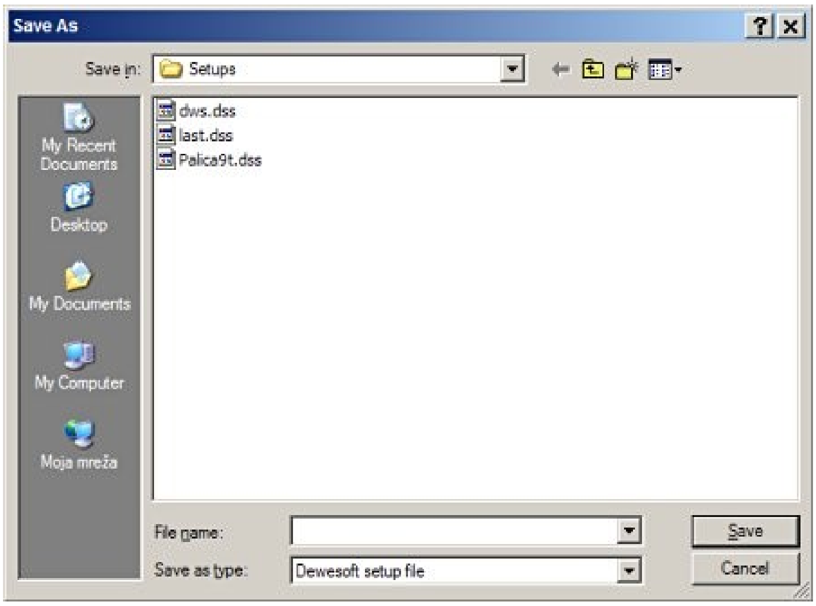
Export



To export - save *existing* (modified) or *new* Measurement setup data select:

Export menu → **Measurement setup**

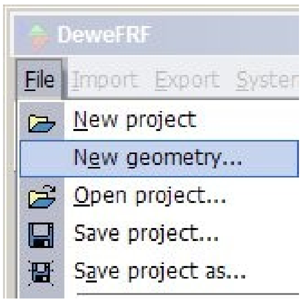
The standard *Save As window* is displayed for saving *new* or *modified* Measurement setup to *.*dss* file:



2.3 Geometry

We can define **Geometry** of mechanical structure for FRF testing in *.*unv* (*Universal file format*) data file as:

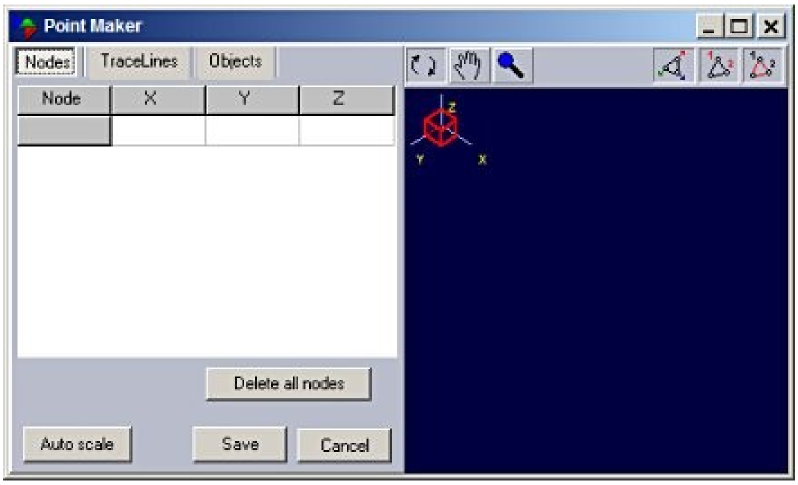
New geometry



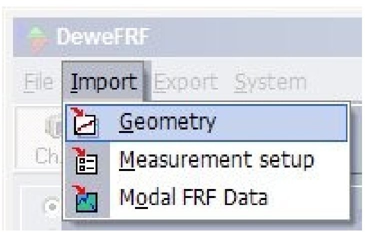
To define new geometry of mechanical structure for FRF testing choose:

File menu → **New Geometry...**

Point Maker screen is displayed. In this screen we can define geometry - *measured points* of *excitation* and *response* and *trace lines* of mechanical structure.



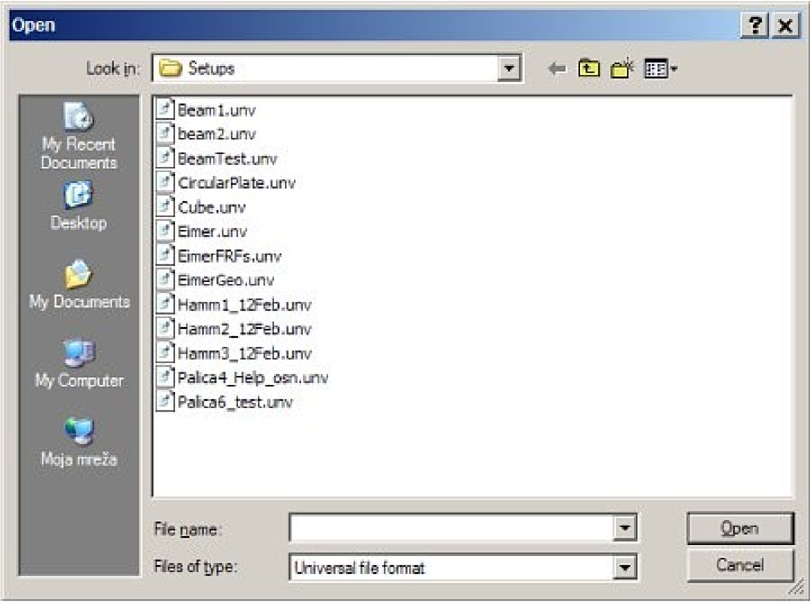
Import geometry



We can also load *existing* (previous defined and saved) geometry of mechanical structure from *universal files* into Point Maker screen by selecting:

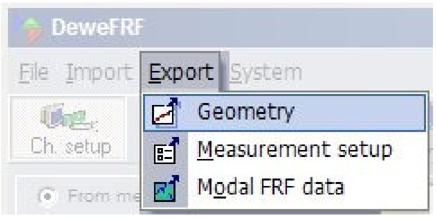
Import menu → **Geometry**

Standard *Open window* is displayed for load *existing* geometry from *.*unv* data file:



Eventually modify *existing* data and afterward Export geometry.

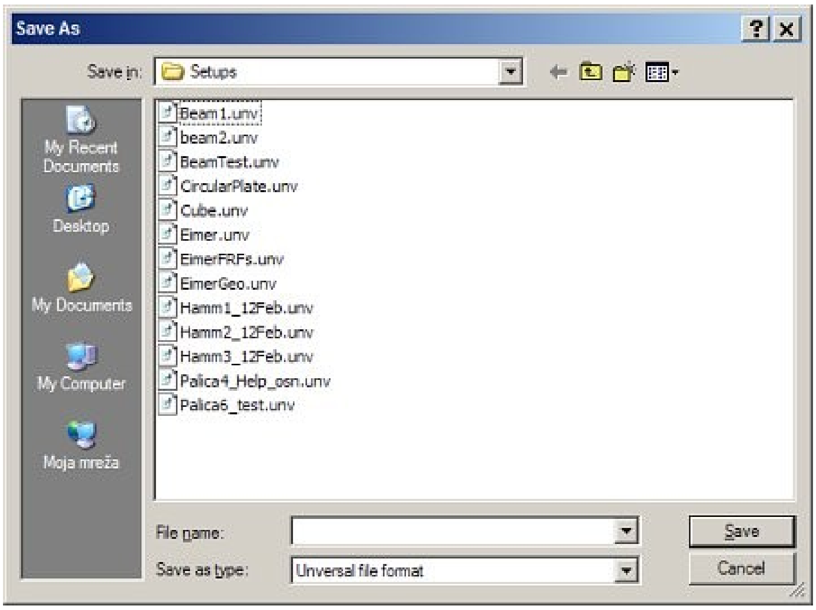
Export geometry



After defining new or modified geometry we can export - save geometry by selecting:

Export menu → **Geometry**

Standard *Save As window* is displayed for save *new* or *modified* geometry to *.*unv* data file:

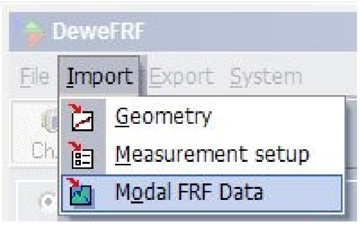


WARNING *because* Geometry *and* Modal FRF Data *are stored in same* *.*unv* *(Universal format)* *file it is strongly recommended to use separate filename for new or modified* Geometry.

2.4 Modal FRF data

In **Modal FRF data** *.*unv* (*Universal file format*) data file measured data are stored.

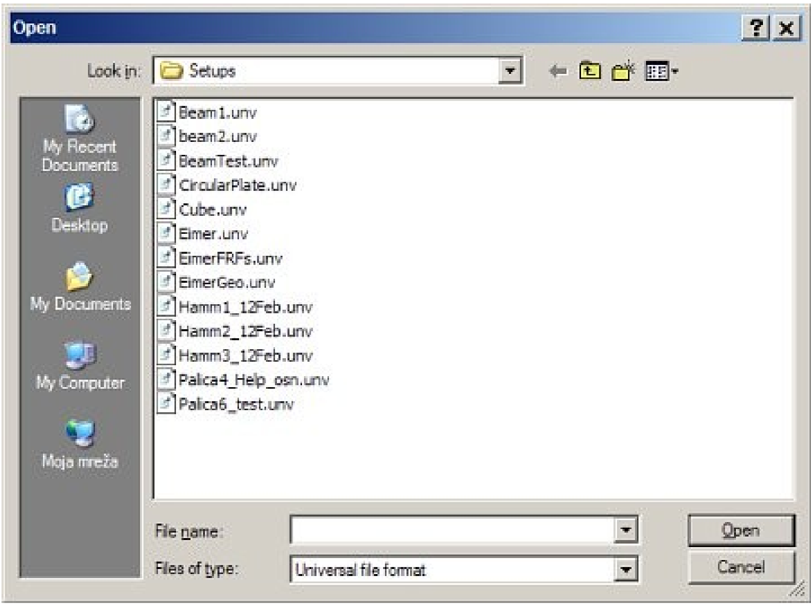
Import Modal FRF Data



We can directly import - load *existing* (previously saved) Modal FRF Data - measured data from *.*unv* data file into Analyse screen by selecting:

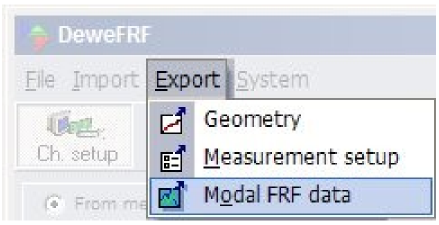
Import menu → **Modal FRF Data**

Standard *Open window* is displayed to load *existing* Modal FRF Data - measured data from *.*unv* data file:



Export Modal FRF Data

to save *new* or *modified* measured data to *.*unv* data file we have two feasibility:



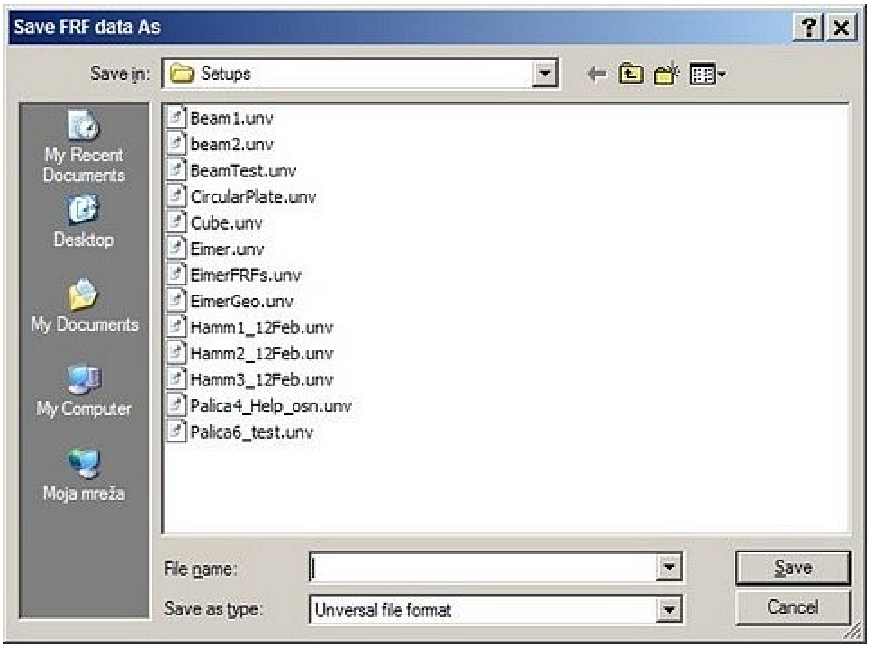
For export - save Modal FRF Data - measured data select:

Export menu → **Modal FRF Data**

UFF export on Analyse toolbar → **FRF data**

(here we have optiions for selecting another sort of data to save - eg *Coherence*, *Excitation data*, *All data*)

In both case **Save FRF data As** *window* is displayed for save *new* or *modified* Modal FRF Data - measured data to *.*unv* (*Universal file format*) data file:

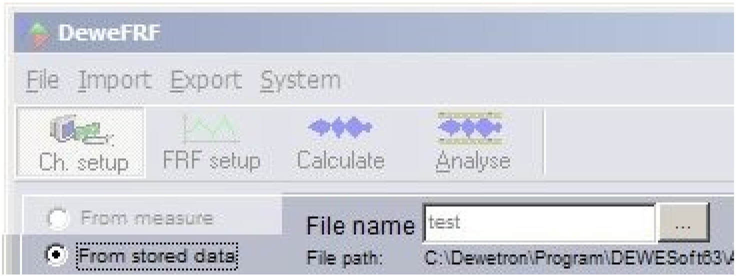


WARNING *because* Geometry *and* Modal FRF Data *are stored in same* *.*unv* *(Universal format)* *file it is strongly recommended to use separate filename for new or modified* Geometry.

2.5 Time data

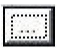
In **Time data** *.*dsd* (*Dewesoft* data *file*) measured time domain data is stored.

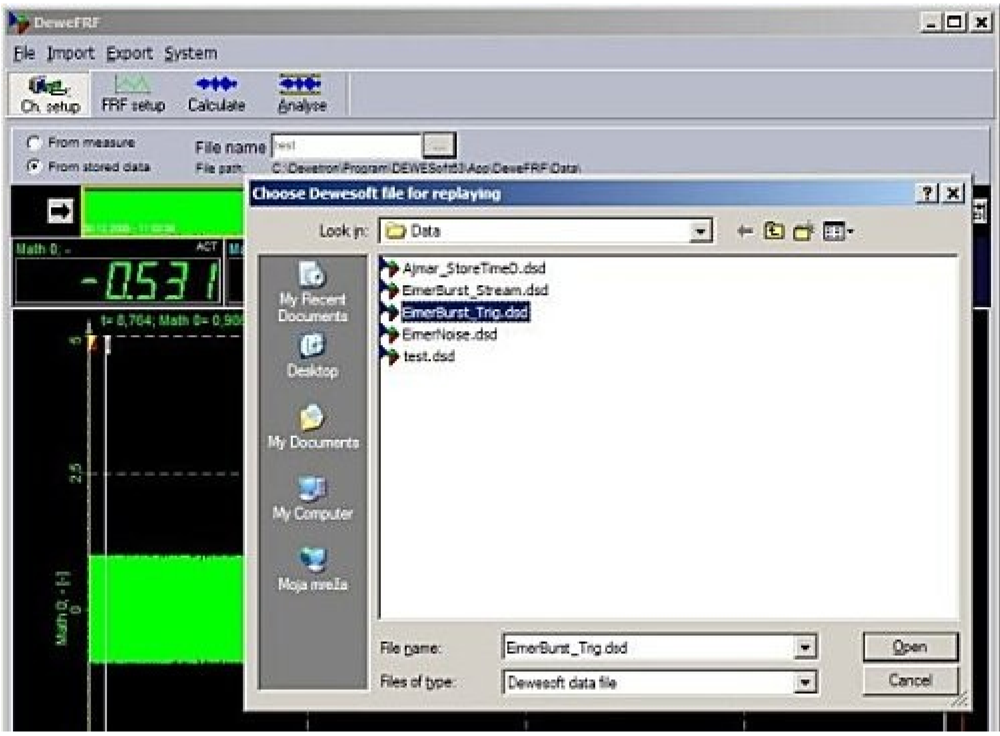
Import Time data → From stored data



Loading *Store time data* - measured data can be done by selecting:

From stored data choice on Ch. setup window

After pressing on  button *Choose Dewesoft file window* is displayed for loading *existing* measured data from *.*dsd* data file:



If we confirm desired file in list with Open we get a little bit different **Ch. setup** window.

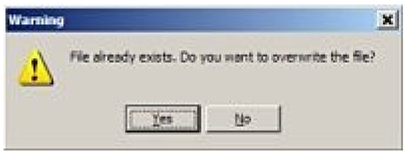
Export Time data → Store time data

to <u>save</u> measured data during extensive calculations to *.*dsd* data:

Saving measured data can be done by selecting:

Store time data button on Ch. setup window.

After pressing this button we get **Warning** screen for <u>overwriting</u> file stated in the field File name:



By pressing Yes we <u>overwrite</u> this file, by pressing No data was <u>not</u> stored and we return to Ch. setup window.

In "File name" we can enter a <u>new</u> file name and store data under this name!

After pressing Store time data button on Ch. setup (and Yes on Warning screen) we get screen like this:

Start / Stop Store process

In place of "*Store time data*" the "Stop" button is now displayed. Store process *starts* immediately. We can stop this process by pressing on Stop button and we get *display screen of stored data in certain time*.

Creating own display

On this *Online Overview* Display screens window are *predefined* display of *standard Digital meters* for *all* used channels and Control properties for this type of display. When you press the Design button, you are allowed to create your *own* display.

Designing of displays is the same as in Dewesoft. Please refer to Dewesoft user's manual for help on designing the displays.

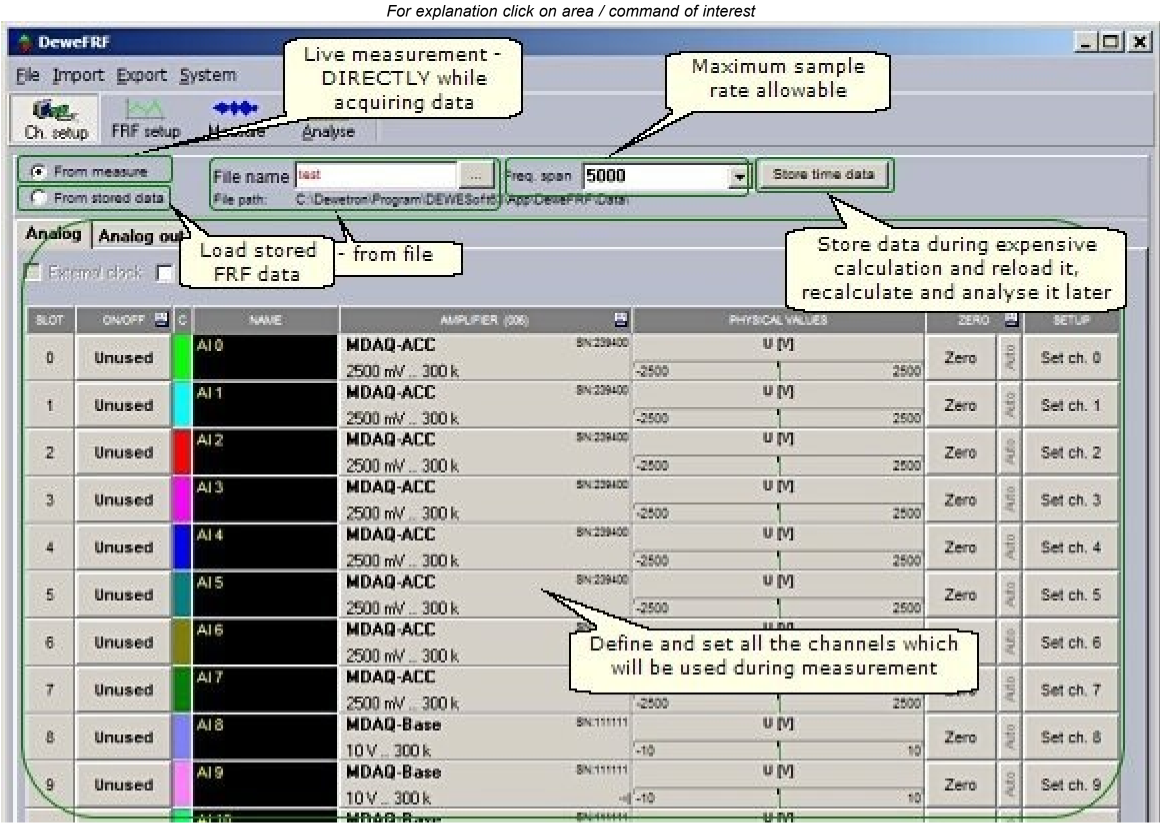
3 FRF Channel Setup

Channel setup is the *second* step of *four basic steps* how to do a **modal analysis**. In the **FRF** Channel setup we can define and set *all* the channels which will be used during measurement using applied hardware.

Channel setup is relevant to measuring (*see* → Measure methods):	
directly while acquiring data with continuous or impulse excitation in **new** or **existent** project:	or for **offline calculation** - From stored data:
Set all the channels which will be used during measurement in **Analog** and **Analog out** tab **Store time data** **store** segment of data for reloading and recalculating in **offline calculation** **Set frequency span** maximum sample rate allowable	Set Time selector (Show and Start show Time) About **Channel setup** for **offline calculation** *see*: User Guide → **Offline calculation**
About New/Open/Import/Save/Export Channel setup data *see*: User Guide → **Data management**	
When the Channel setup are **done**, go to the *next step* → **FRF setup**	

For entering Channel setup select **New Project** from File menu or click the  icon and

For explanation click on area / command of interest

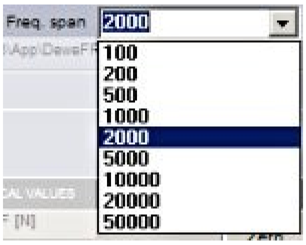


common *Channel Setup Screen* is displayed. Displayed **tabs** in central part of screen depend on selection in System → Hardware setup menu and consecutive on measurement mode PROF or DEMO.

Note: For FRF analysis it is reasonable setting only *Analog* and *Analog out* in Hardware setup.

In this User guide we can get help about the **Analog** tab and **Analog out** tab (in *Analog out* tab also Function generator can be set).

Set frequency span



Frequency span defines the upper frequency limit of the FRF. This defines what will be the *highest frequency* which can be measured.

The maximum sample rate allowable will vary according to *which* **A/D** board you have installed, and *how many* channels are activated for recording.

In the drop down-list choose appropriate frequency span.

Note:

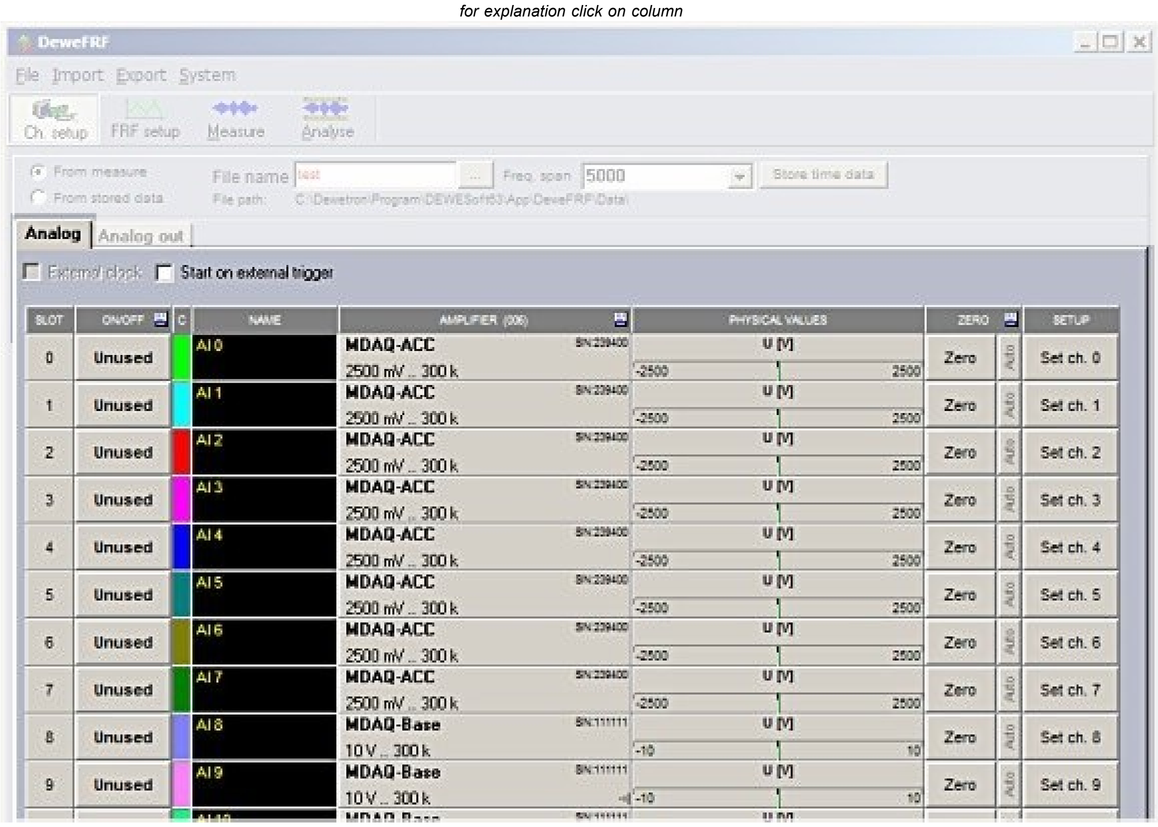
The sampling rate is set to 2*1.28*Frequency span. The factor 2 is the Nyquist factor for sampling and 1.28 is used to assure that the results will be *alias free*.
So if the frequency span is 2000 Hz, the sampling rate will be set to 5120 Hz.

WARNING	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 !

3.1 Analog

In **Ch. setup** → **Analog** tab we can **set all the channels** which will be *used* during measurement.

for explanation click on column



Above the channel list we can select:

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.

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*.

Columns in the channel list explained

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

Slot - *input* channel

On/Off - *activate* / *deactivate* channels

C - channel *color* selector

Name - channel *name*

Amplifier - *amplifier* type and ranges

Physical values - current *input values*

Zero - remove *offset*

Setup - *input settings* and *calibration*

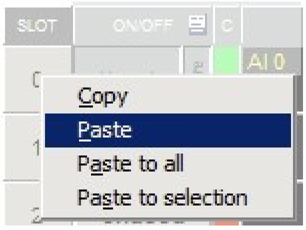
3.1.1 Input channel - Slot

This column indicates the number of the channel. It always *starts* with **0** and counts up to the maximum number of *available* output.

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 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.

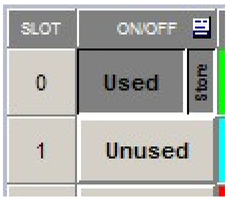
But you can copy the settings also to a channel selection or to all the channels.

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

Back to 'Analog' channel list   Back to 'Analog out' channel list

3.1.2 Activating inputs - On/Off

Input 0 is already *active* (marked Used beside it). Click *on* the input slot to activate them and the screen is displayed like this:

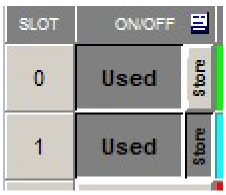


In this column there is a button that you click to *toggle* this input on/off (Used / Unused). If it says Used, then it will be available. Use the button to *activate* / *deactivate used* channels.

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

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

Beside the Used / Unused 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.

When you press the button, 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 *unrequired* data (for example you need the data *only* for mathematics and *store* the math result).

Back to 'Analog' channel list

3.1.3 Channel color - C

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

Choose from any displayed color or *create* a user defined one. 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 Channel setup** or
- *Analog out* channel - see → **Analog out Channel setup**

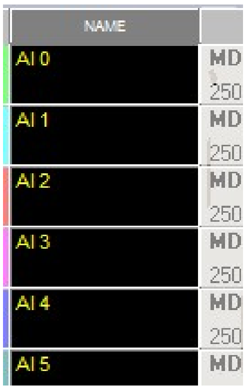
This color will carry through the text and graph representations of *this* channel *throughout* all DEWE software procedure screens.

Back to 'Analog' channel list

Back to 'Analog out' channel list

3.1.4 Channel name - Name

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



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

Navigation keys are same as in other Windows application.

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

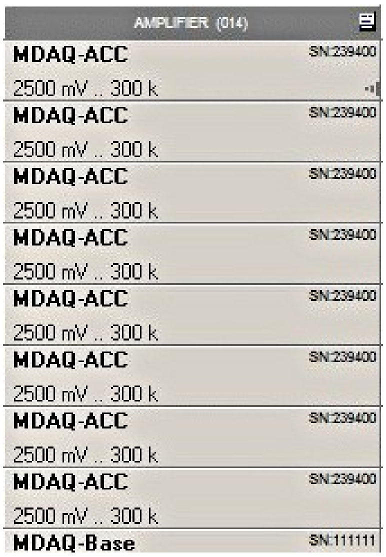
- *Analog in* channel - see → **Analog Channel setup** or
- *Analog out* channel - see → **Analog out Channel setup**

Back to 'Analog' channel list

Back to 'Analog out' channel list

3.1.5 Amplifier - type and ranges

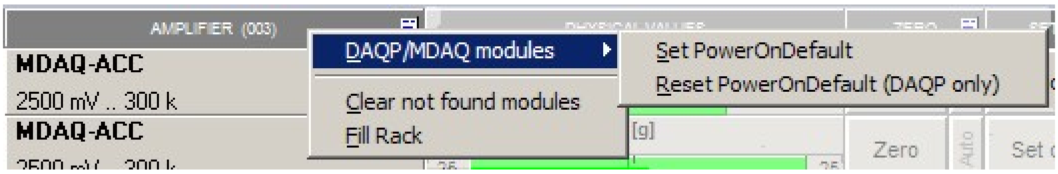
Shows the DEWE-Module currently **installed** in this slot:



- 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.

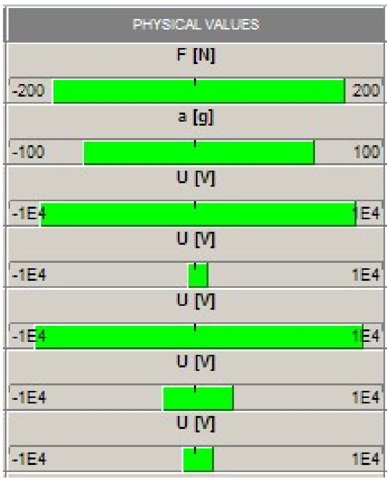
A small arrow on the right side of the field is running through all fields, *indicating* that all channels are **scanned** for *new amplifiers* or *settings*.

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:



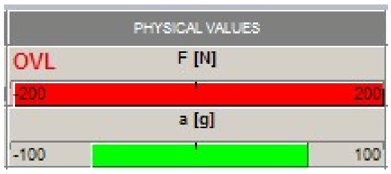
Back to 'Analog' channel list

3.1.6 Physical values



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

All this can be set in Channel setup column, which can be reached by *double click* on channel cell in this column. When the input signal **exceeds** the possible range, a *red* indicator **OVL** - *overload* will be displayed:

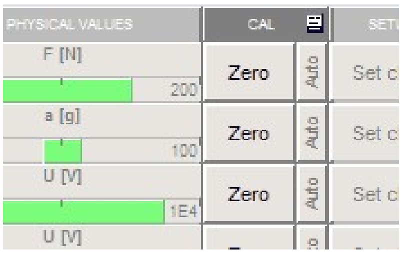


If this happens, *check* your *sensor* and / or *select* another *input range* (in Channel setup).

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 **digital representation** of your *inputs*:

3.1.7 Input zeroing - Zero

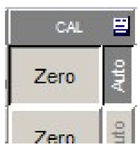
This is a button that you can click to *perform* a **mathematical zeroing** of this input, to *offset* small variations in the *zero position* of the input.



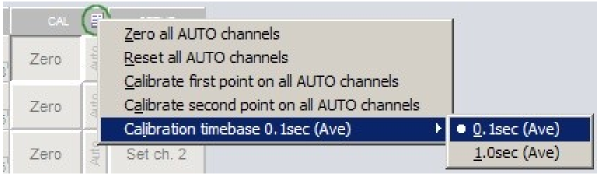
Press the *left* mouse button to *activate* **zeroing**, and the *right* mouse button to *deactivate* it (reset to *default* input range).

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**.

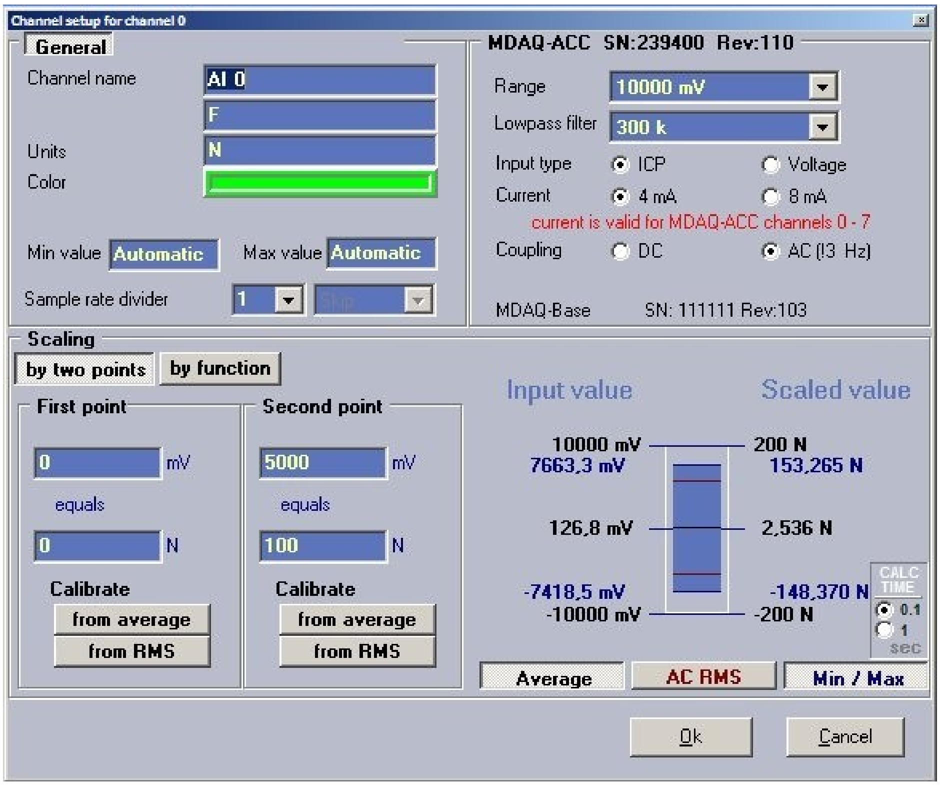
- We can also choose the **Calibration time base as 0.1 sec (Ave)** or **1.0 sec (Ave)** (same as in Channel setup). 0.1 second will give *faster response* time while 1 second will provide *more averaging time* to perform calibration.

3.1.8 Channel Setup

To **calibrate** *any input*, just click the Setup button for that input, eg for set channel 0 press Set ch. 0 button.

This is a button that *calls* up the **Channel setup** dialog box for this *input channel* and this setup dialog will appear:

For explanation click on field of interest

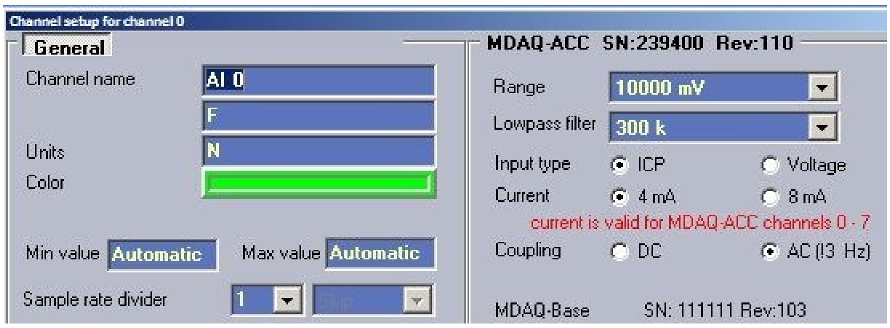


Back to 'Analog' channel list

3.1.8.1 General

In this "General" part of Channel setup screen you can *enter* the Channel name, the measurement value, Units of measurement, *define* Color, display range, Sample rate divider and Module selection for this channel.

For explanation click on field of interest



Measurement value and unit

Measurement value can be **F** or **Force**, **Load**, etc and are entered as *text*.

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

Navigation keys are same as in other 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). This fields is 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 Windows application, Enter key also closes Channel setup window.
  To set the values back to Automatic range detection, just right-click in the desired field and select **Set to Automatic** from the list:

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,
- **Average** selected number of values,
- **4th**, **6th** or **8th** order **Filter**.

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

Module selection

All boxes and fields at the top-right of screen depend from the built-in **module**. If you have actual DEWE-Modules (DAQP or PAD series) in this computer, they will *appear* here, and you can *control* their **ranges** and **filters** directly from this screen.

- you could select *any* non-programmable DEWE-Module and **Module type** combo box and **Measure range** will be shown on screen; older DAQN series are supported, making this software backwardly compatible with all modules that we have made.
- if a programmable module is installed in this slot, the Module type combo box will be *replaced* with a **Range selector**, and the Measure range will be replaced with a **Lowpass Filter selector** and other **module specific fields** are shown.

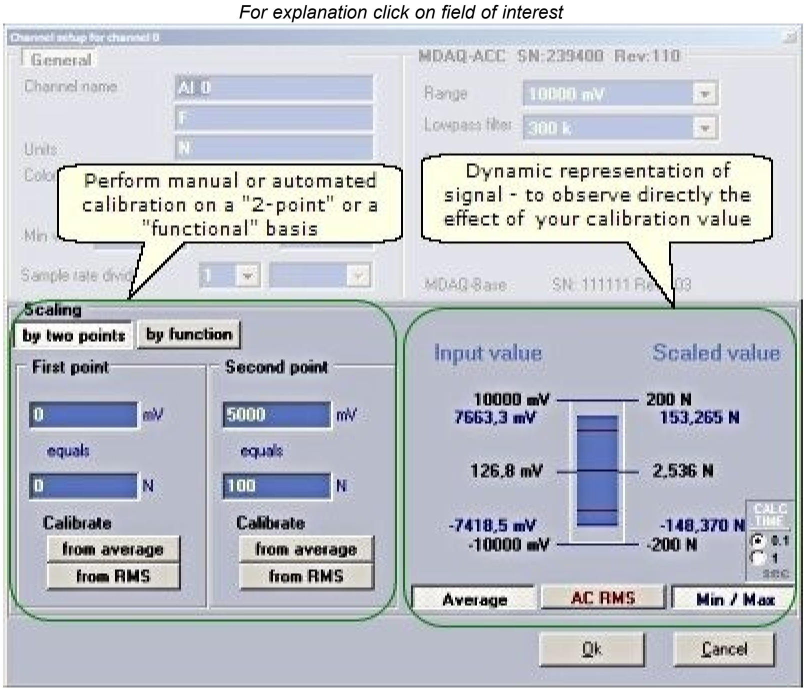
for details see A/D Board Specifications

Back to 'Analog' channel list

Back to 'Analog out' channel list

3.1.8.2 Scaling

For explanation click on field of interest



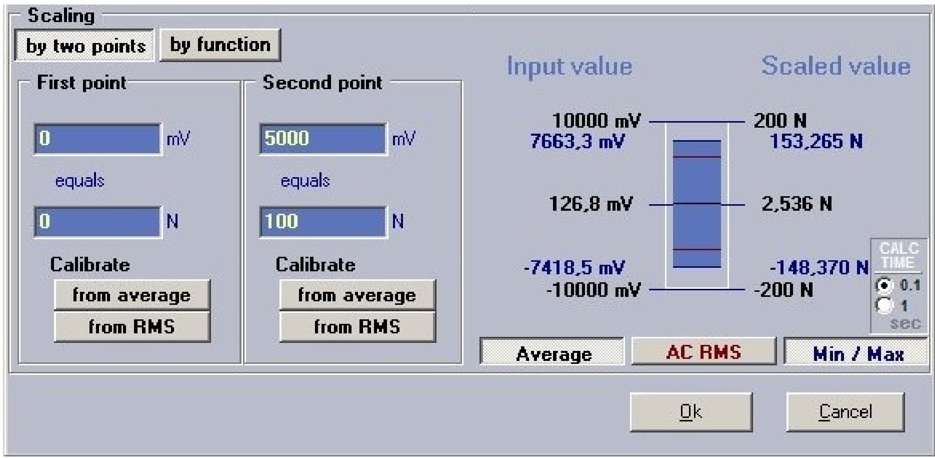
The *bottom-left* corner of Channel setup screen is where you can *perform* scaling, either on a: **2-point** or **functional basis** and *manual / automated* **calibration**.

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

Back to 'Analog' tab - Channel setup

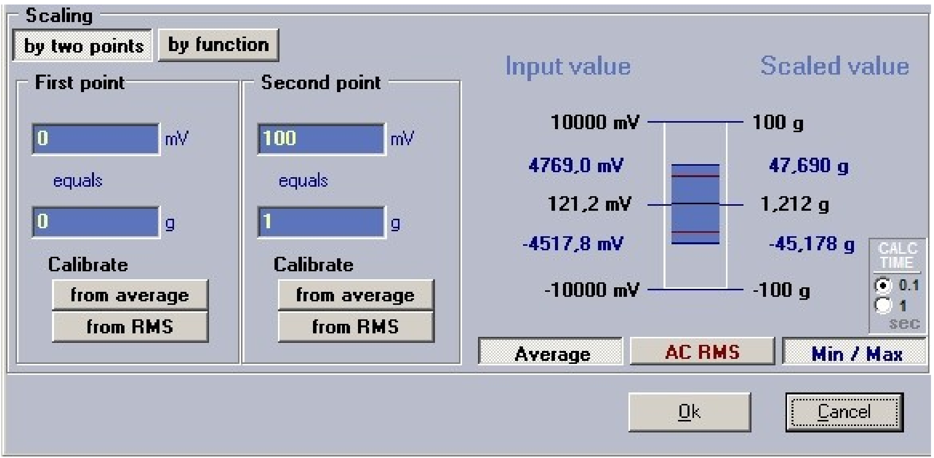
Back to 'Analog' channel list

3.1.8.2.1 by two points



In this example we calibrate the system so that the channel AI 0 (Channel name in General part of channel setup screen): 0 real volts *equals* 0 N and 5000 mV (electrical units) *equals* 100 N (engineering units - **Force**).

For **other** channel AI 1 (Channel name in General part of channel setup screen) of this example: 0 real volts *equals* 0 g , and 100 mV (electrical units) *equals* 1 g (engineering units - **Acceleration**):

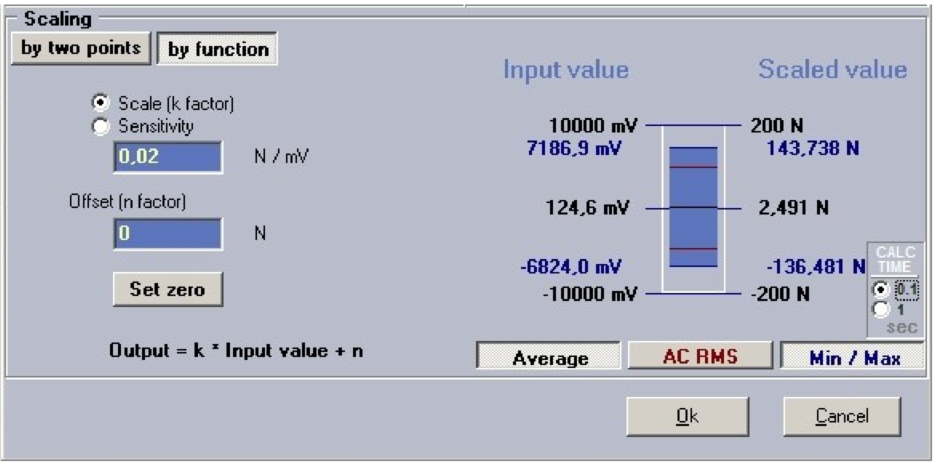


Back to 'Channel setup' - Scaling

Back to 'Analog' channel list

3.1.8.2.2 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:

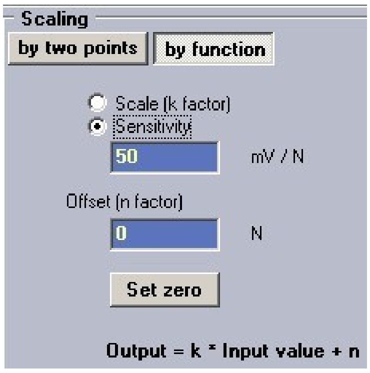


In our example we calibrate the system so that for channel AI 0 (Channel name in General part of channel setup screen) this factor is **0.02**.

Above *shows* your basic y = mx + b algebraic formula for a general linear function, which makes it easy to *input* strain gage k-factors and offsets (n-factor), as well as to perform simple offset calls like the *conversion* from Celsius to Fahrenheit (enter 1.8 in the top box and 32 below).

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

- **Scale (k factor)**; example above: k = 0.02 N / mV
  Scale factor is the factor which defines how to scale the electrical signal to deduce the physical values.
- **Sensitivity**; same example: k = 50 mV / N
  Sensitivity is the factor which is usually used in sensor calibration sheets and defines the electrical output per physical quantity measured by the sensor.



Use the **Set Zero** button in the main setup to conveniently offset the signal to 'force' it to **zero**, when it is not exactly at that value (but should be), due only to *small* sensor offsets and perhaps *long* cable lengths causing capacitive coupling and/or line loss.

Back to 'Channel setup' - Scaling

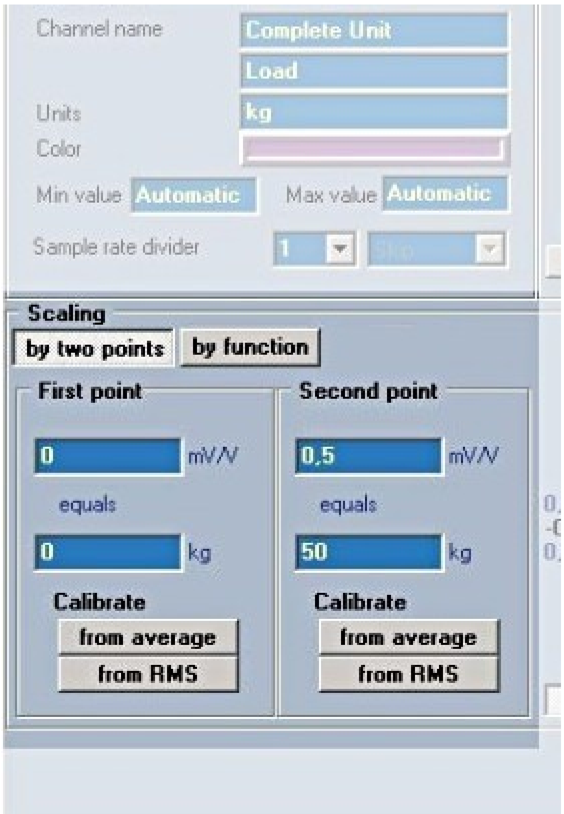
Back to 'Analog' channel list

3.1.8.2.3 calibration

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

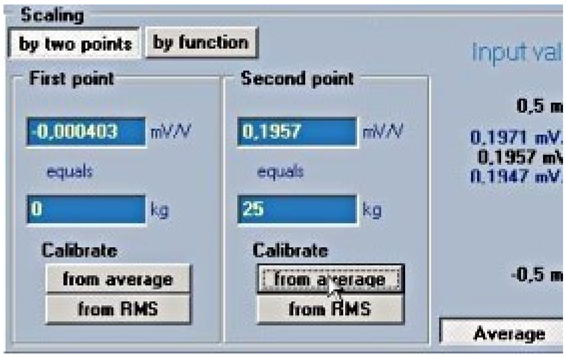
In 2-point calibration, there are *two* other sets of buttons that will allow you to perform a more accurate calibration when you have *access* to a known signal *source* or *load*. Let's say that we have a load *sensor* that we want to *calibrate* to a known load. We want to *enter two* points - say 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:

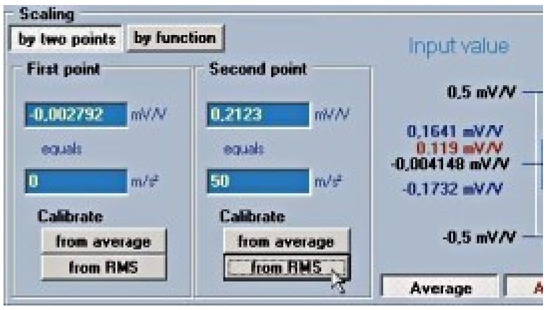


If you have an AC signal source, you should not use the *from average* buttons - *use* the **from RMS** buttons instead! Also, press the *red*-labeled RMS button below the graph to *add* the RMS values to the *dynamic display*.

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, 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. DEWE software is now *perfectly* calibrated to your *signal* based on the known loads that you input. Note in the screen picture above that DEWE software has *automatically* input the two values in the upper two boxes that it needed in order to achieve this calibration.

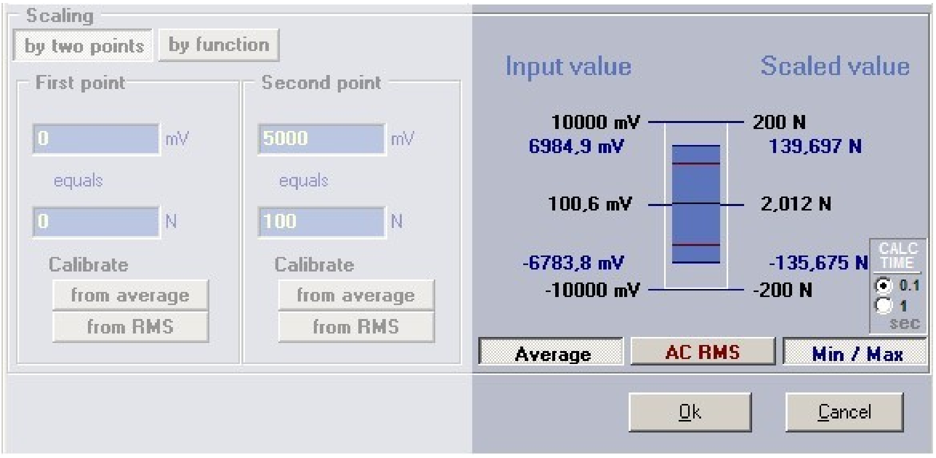


Back to 'Channel setup' - Scaling
Back to 'Analog' channel list

3.1.8.2.4 dynamic representation

The bottom-right portion of the dialog contains a dynamic representation 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 **calibration values**:

for explanation click on field of interest



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

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*.

Dynamic graph

On a dynamic graph your **signal** is 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.

Calculation time

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

Predefined choices are:

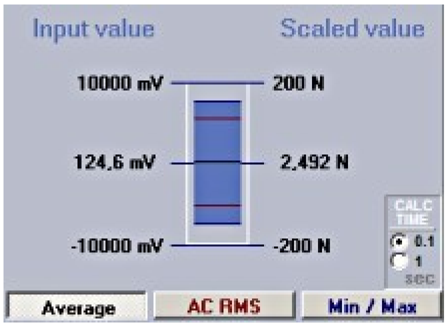
- **0.1** second
- **1** second.

Back to 'Channel setup' - Scaling
Back to 'Analog' channel list

3.1.8.2.5 dynamic graph

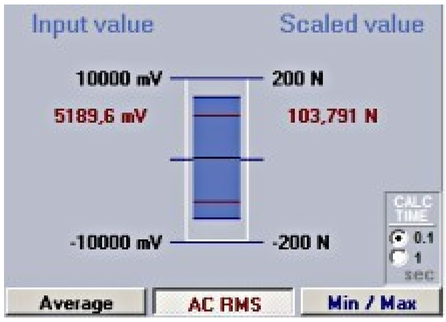
Depending on *pressed* button(s) in this area **Average** **AC RMS** **Min / Max**, different Input and Scaled values are *represent* on dynamic graph:

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

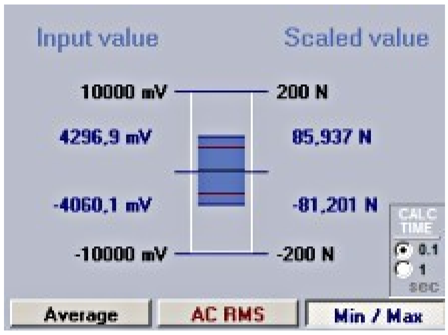


When button **AC RMS** is pressed, **AC RM** value of your *Input* and *Scaled* **signal** is *represented* on dynamic graph.

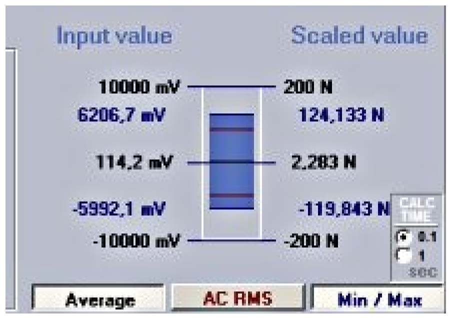
If you have an AC signal source, you <u>should</u> press the AC RMS button.



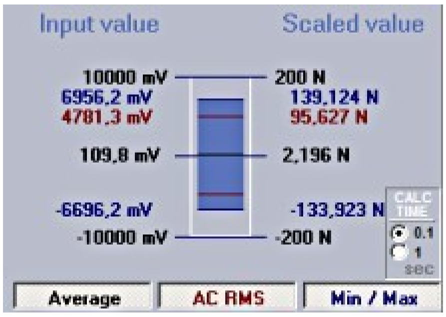
When button: **Min / Max** is pressed, **Minimum** / **Maximum** value of your *Input* and *Scaled* **signal** is *represented* on dynamic graph.



Usually the: **Average** and **Min / Max** button 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 **Min / Max** and **AC RMS**

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

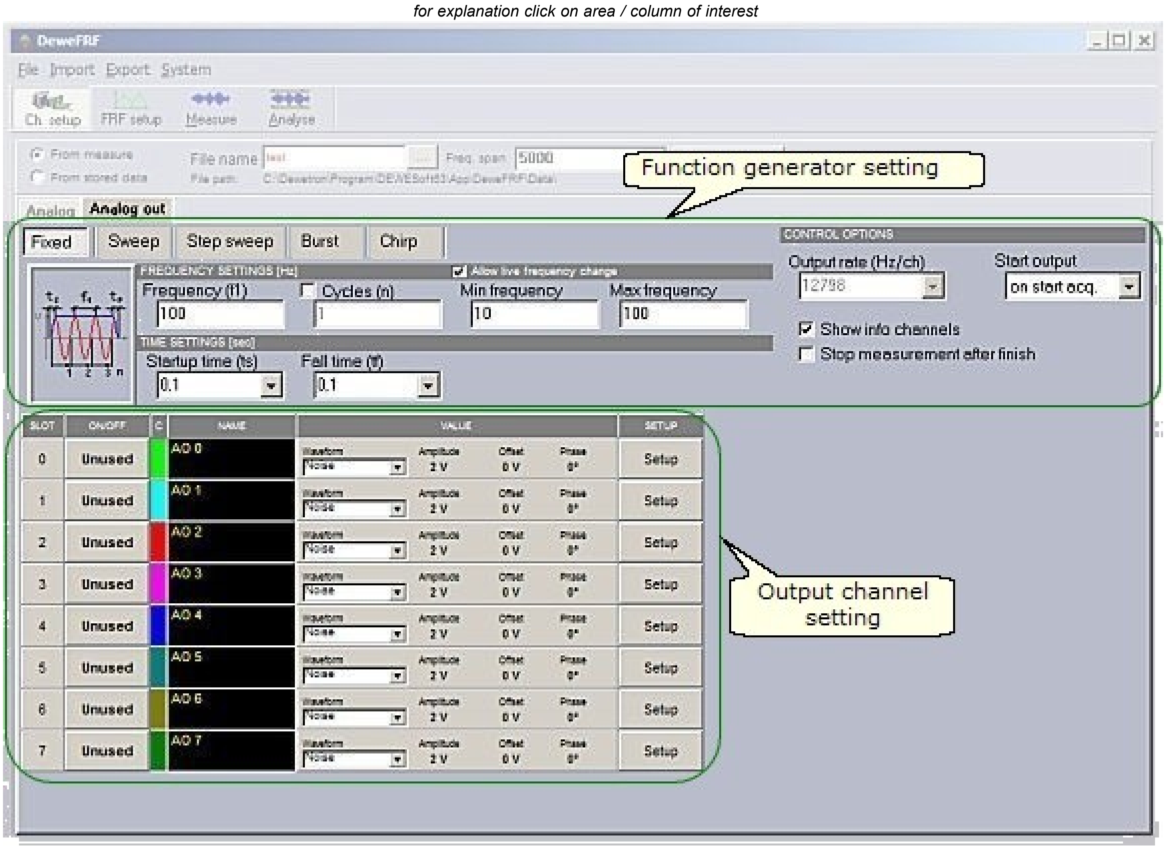
Back to 'Channel setup' - Scaling
Back to 'Analog' channel list

3.2 Analog out

DEWESoft offers a full-featured **signal-function generator** with **16** or more *analog signal* outputs as an option.

One of the available tabs In **Ch. setup** screen next to *Analog* (and eventual others, eg *Math*) has to be labeled **Analog out**. In this tab in the *upper* part there are the definitions of a **Function generator** and at *bottom* part table the **Channel list** in which we can *set* all the ***analog out*** channels which will be used during measurement. The handling of this settings is already well-known from the *Analog input* section and very similar.

for explanation click on area / column of interest



Columns in the channel list explained

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

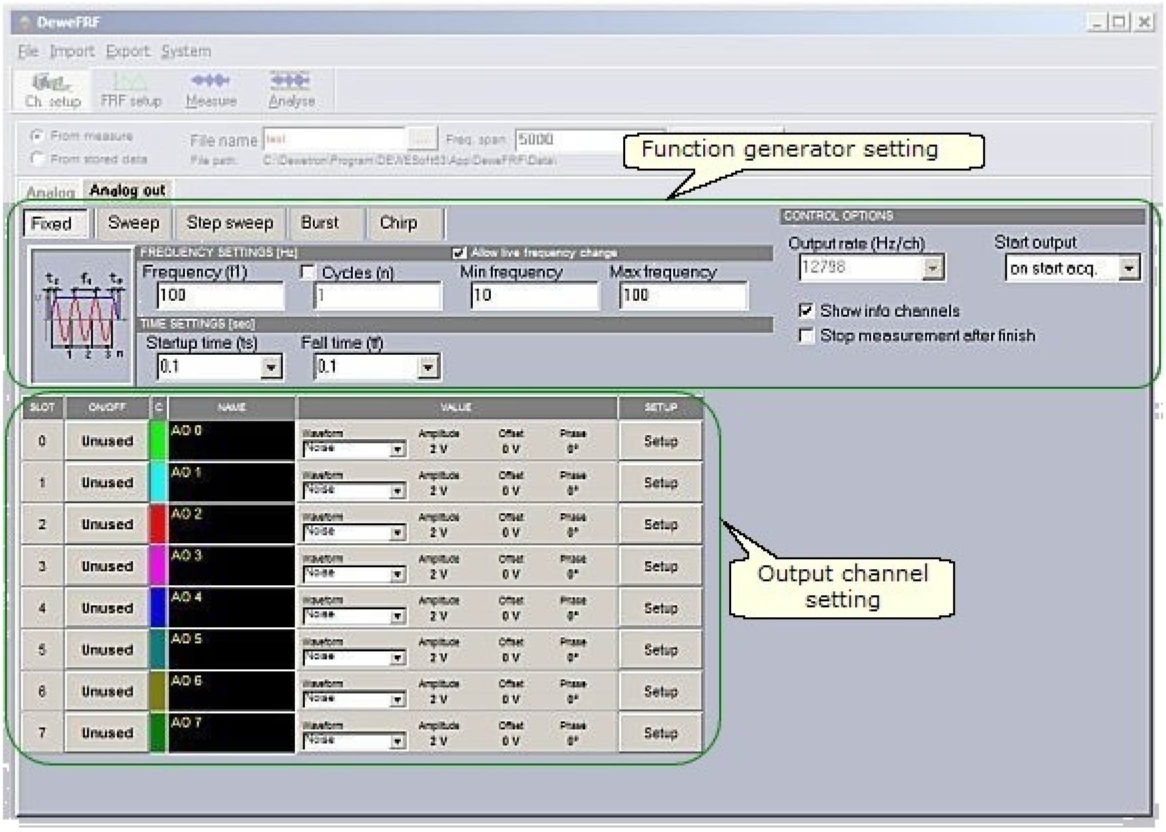
Slot	- *output* channel
On/Off	- *activate* / *deactivate* channels
C	- channel *color* selector
Name	- channel *name*
Value	- *amplifier type* and ranges
Setup	- channel *settings*

3.2.1 Function Generator

DeweFRF software offers a full-featured **signal generator** with 16 or more *analog* **signal outputs**, any analog output card from *National instrument* can be used as the hardware with *different*:

- **operation modes**
- **signals Waveform**

Analog output channels, which will be *used* during measurement, are *set* in **Ch. setup** on **Analog out** tab.



Available modes are *fixed* frequency, *sweep*, *burst* or *stepped sweep*. There are many functions available for each mode like frequency, operation mode, timings, control and so on. 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.

The **start** and **stop** of generation can be either on *start* of acquisition 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

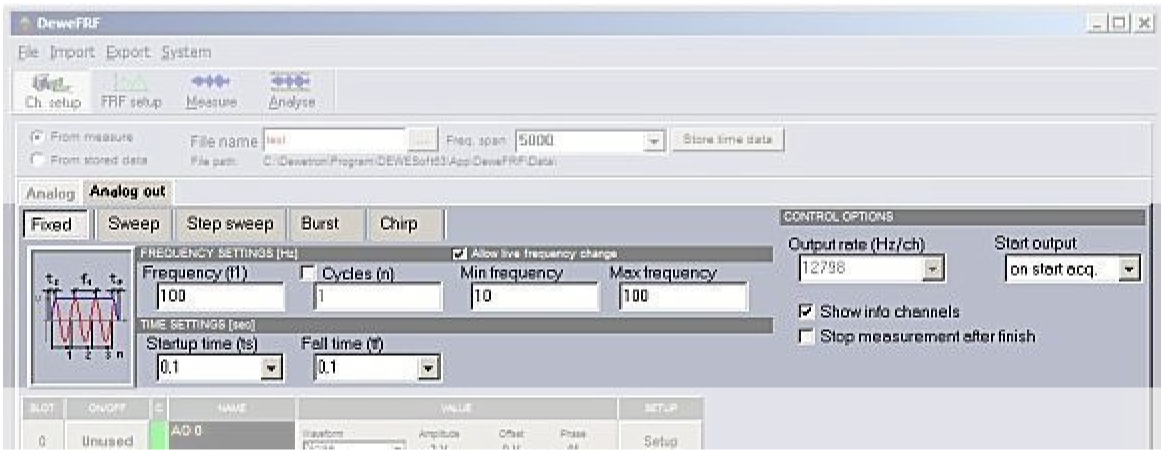
The start and stop of generation can be with a *half sine window*, so the amplitude will not rise from **0** to **full** amplitude *immediately*, but within a certain time which ensures that the systems under test don't suffer from *impact shock*.

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.

3.2.1.1 Operation modes

In **Analog out** tab of DeweFRF *Channel Setup Screen* can be *set* for particular **operation mode**:

- **Frequency setting [Hz]** and **Time setting [sec]** - *for explanation click on operations mode name bellow*
- **Control Options**.



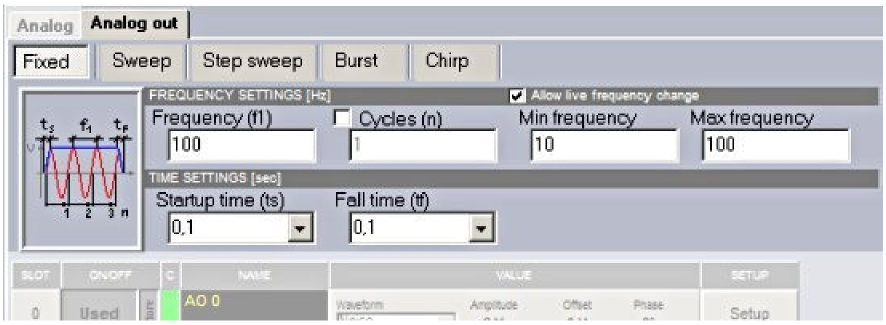
The function generator offers five *different* **operation modes**, which are *set* for all analog output channels:

- **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*.

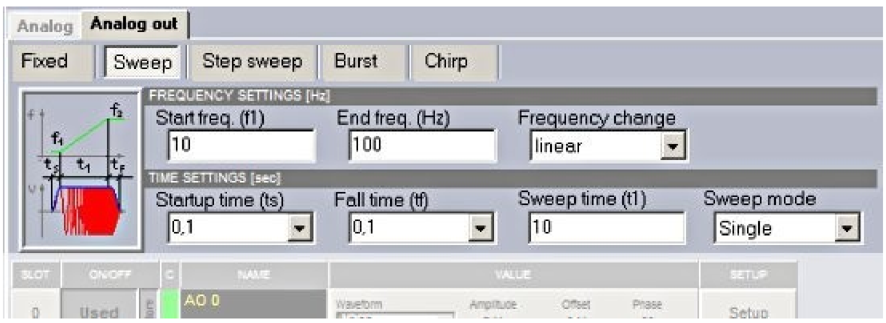
Applications: fixed mode is used if a defined waveform should be *output without changing* the *frequency* or *amplitude automatically*. Such function generator is valuable in laboratory testing of *electronic circuits*, *power amplifiers*, *filters* and so on. Also it provides a great base for *calibration* 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). Additionally the Frequency change of the sweep can be either linear or logarithmic to provide more cycles at lower frequencies.

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

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



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

Step sweet

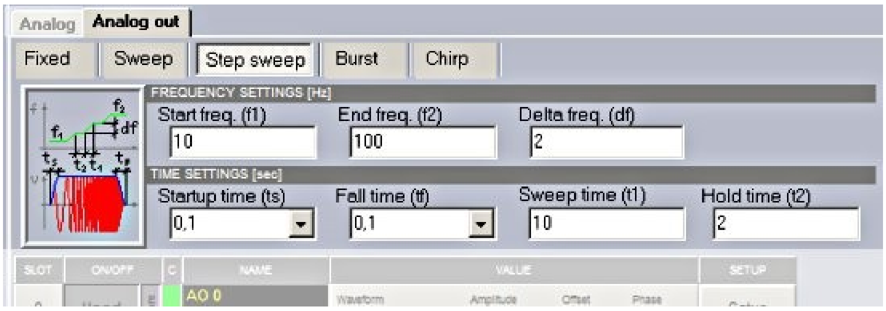
The step sweep mode is a special procedure for modal testing, where the *signal* remains at *certain* **fixed frequencies** to allow the structure to settle to that frequency. In addition to this more *exact* frequency resolution of *transfer function* can be achieved with smaller steps. Even though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

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

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

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

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



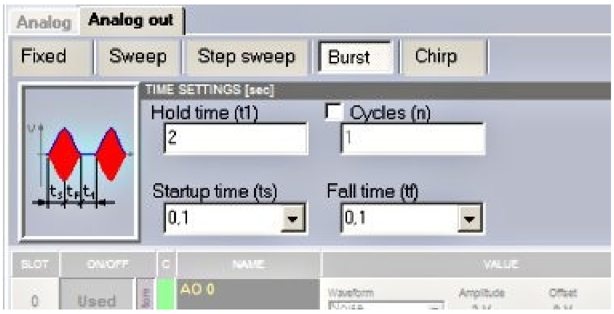
Applications: this mode is mainly used for modal *structural tests*.

Burst

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

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

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



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

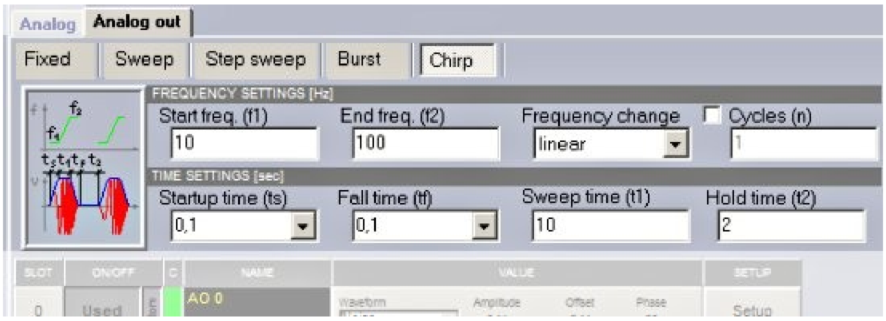
Chirp

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

Start freq. (f1) and End freq. (f2) *define* the frequency *range* of the chirp. The Frequency change can be either linear or logarithmic.

Additionally it is possible to *define* the **number** of chirp Cycles (n) to *output*.

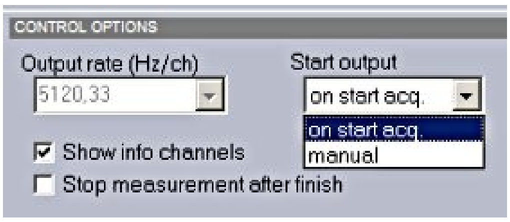
The *time settings* also contain Startup time (ts) and Fall time (tf) which *defines* the **ramp** at the *beginning* and the *end* of the chirp, Sweep time (t1) - the chirp **duration** - and the Hold time (t2) *between* two chirps.



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

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

3.2.1.2 Control Options



With the Control Options Output rate and Start output condition are *set* for the analog output.

The **Start output** condition can be set either to:

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

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

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

If **Stop measurement after finish** is *enabled* the data *acquisition* will be *stopped* after the signal output has been finished. This *requires* a defined *cycle number* for output in the *operation mode settings*.

3.2.2 Activating outputs - On/Off

This column displays the state of the channel:

- **Unused** → *disabled*, no signal output
- **Used** → *active*, signal output).

Click on the button to activate or deactivate channels.

If it reads **USED**, then it will be *available*.

Back to 'Analog out' channel list

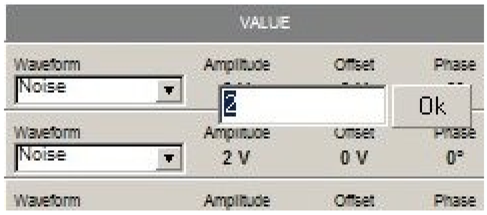
3.2.3 Channel value

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

Waveform type

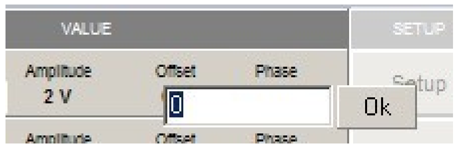
The waveform type can be set *individually* for *each* channel:

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

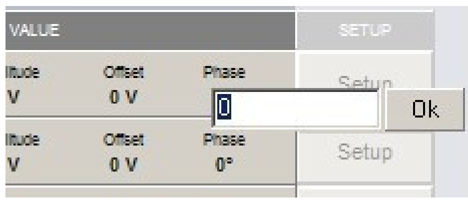


Also **Amplitude**, **Offset** and **Phase** can be set *individually* for *each* channel as well.

If you need *precise* entries simply click in the numeric field in appropriate column 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.

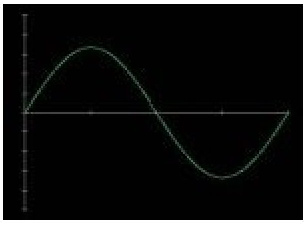


Back to 'Analog out' channel list

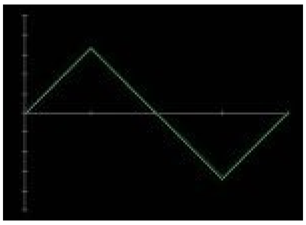
Back to Function Generator

3.2.4 Waveform type

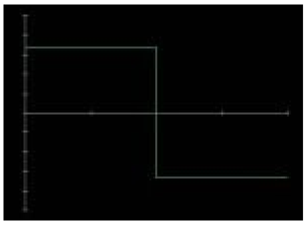
In DeweFRF software the **waveform types** are:



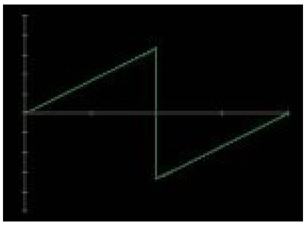
SINEWAVE



TRIANGLE

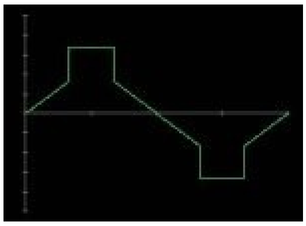


RECTANGLE



SAW

NOISE



ARBITRARY

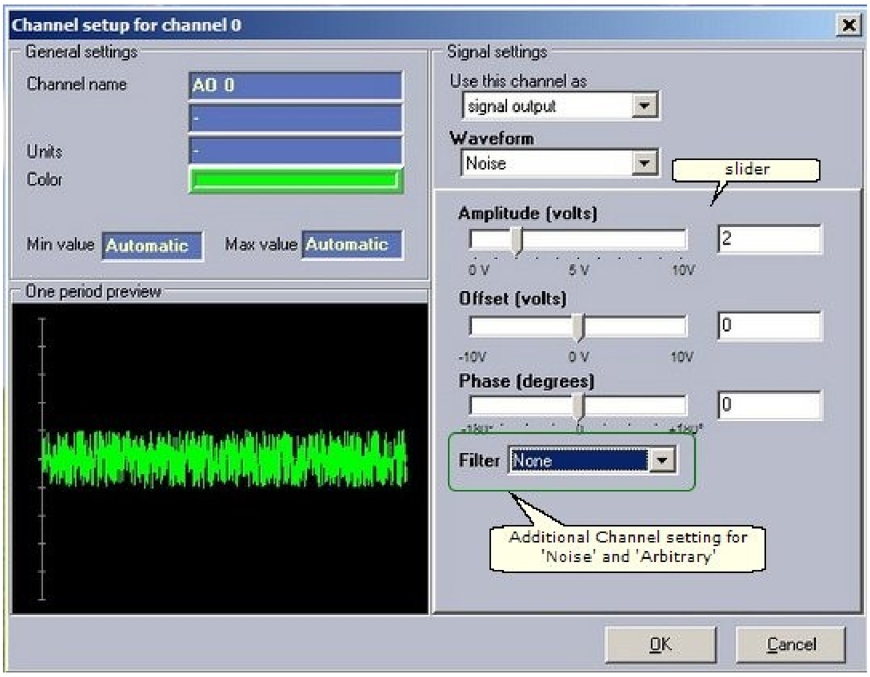
Waveform types can be *set* in:

- **Waveform** of Value column in Analog out tab or
- **Signal setting** of analog *output* Channel setup screen in Analog out tab.

3.2.5 Channel setup

Press the **Setup** button in SETUP column to *enter* the **setup screen** for <u>each</u> *channel*.

for explanation click on field of interest



The *upper left* area of the analog *output* channel setup screen contains **General setting**:

- the *base* **channel information Channel name**, measurement value, **Unit** and **Color**. Like in the analog input setup screen,
- the **Min value** and **Max value** fields can be used to *set* a minimum and maximum **display value**.

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

- the **signal usage** selection in **Use this channel output as**.
- the **Waveform type**. The Waveform drop down selection has a direct *influence* on the settings displayed in the lower right area.

All generated channels themselves are available as DeweFRF *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 <u>directly</u> with your keyboard.

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

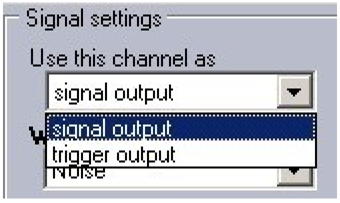
The lower left area shows a nice One period preview of the *selected* signal.

Signal setting

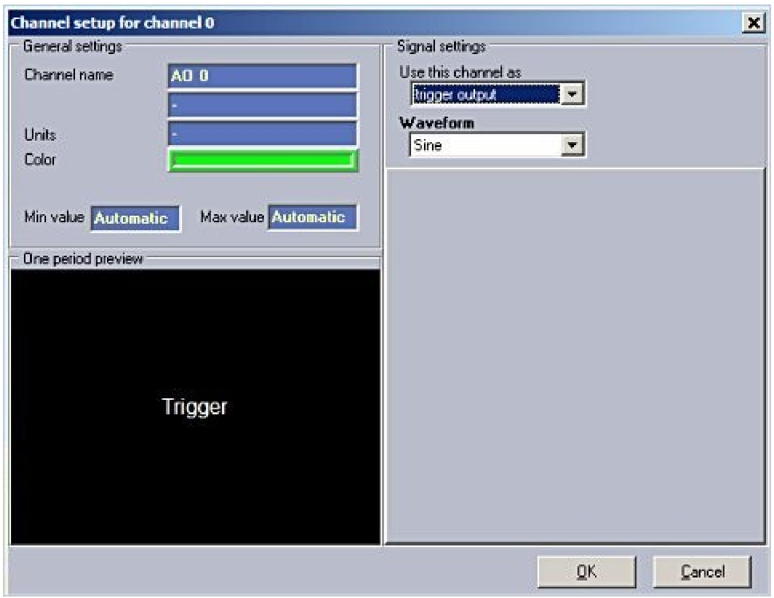
Use this channel output as

The *channels* can be used as:

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



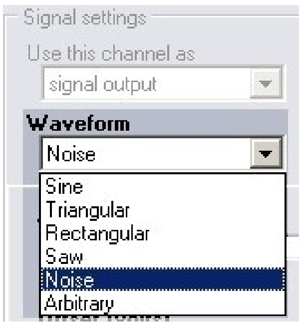
Often it is intended to trigger the *acquisition* on *measurement* side when *certain criteria* is *met*. When the sweep is active, the trigger value *goes up*. It is up at step sweep, at fixed frequency and during the bursts.



Waveform

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.



Additional signal setting

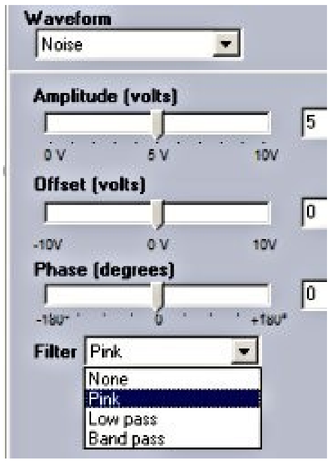
For '**Arbitrary**' type of waveform there are sliders and numeric fields in 'Signal' setting tab, *additional* is also tab **Points** for this **signal setting**.

For '**Noise**' type of waveform bellow the sliders there is *also* an additional **Filter signal setting**.

see also → **Noise - Additional settings**
see also → **Arbitrary - Additional settings**

3.2.5.1 Noise - Additional settings

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

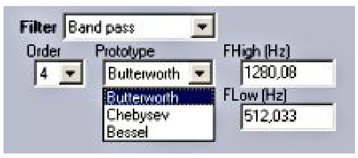


- None — no filter is used - the output signal is white noise
- Pink — a filter is used to obtain pink noise
- Low pass

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.

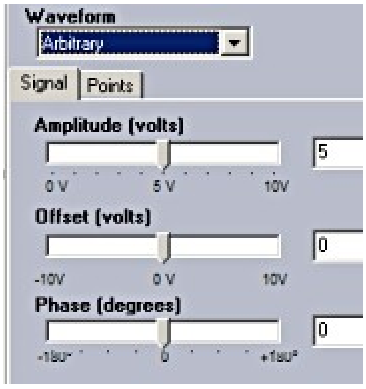
For noise excitation it is common to define the *upper* and *lower* frequency limit. The lower limit prevents DC excitation and upper limit prevents too high frequency noise which is higher than the bandwidth of the measurement.

Back to 'Analog out' channel list   Back to Function Generator

3.2.5.2 Arbitrary - Additional settings

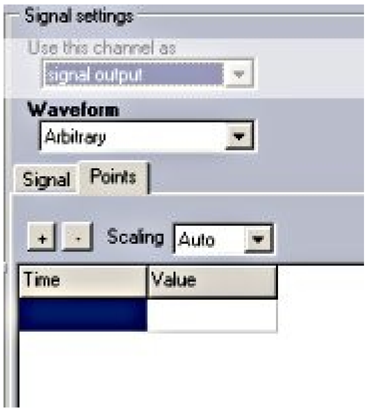
The **arbitrary** waveform makes definition of special waveforms very easy through two tabs.

Signal tab

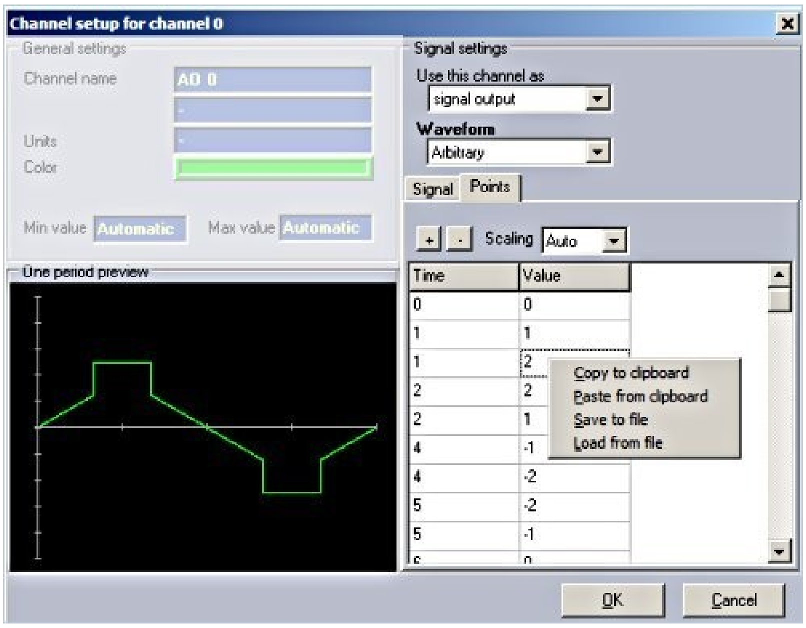


All parameters, like **Amplitude**, **Offset** and **Phase**, can be set up as already known.

Points tab:



The *waveform* itself is defined in a table, which can be displayed in the Points tab:

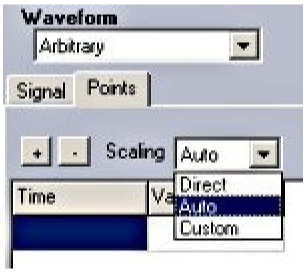


There are *three* ways to *get* a **signal**:

- 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 DeweFRF. 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 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 !

Back to 'Analog out' channel list

Back to Function Generator

4 FRF Setup

In FRF Point & Geometry setup - FRF setup we can define DeweFRF modal analysis software and applied mechanical structure:

FRF setup is relevant for measuring **directly while acquiring data** with continuous or impulse excitation or for **offline calculation** → *see* Measure methods.

FRF property: frequency span, number of lines, calculation method and FRF Calculation type (free run or triggered acquisition) with all of its properties

Channels assignment: point index of excitation and response channels, adding/removing excitations or responses with help of FRF setup tools

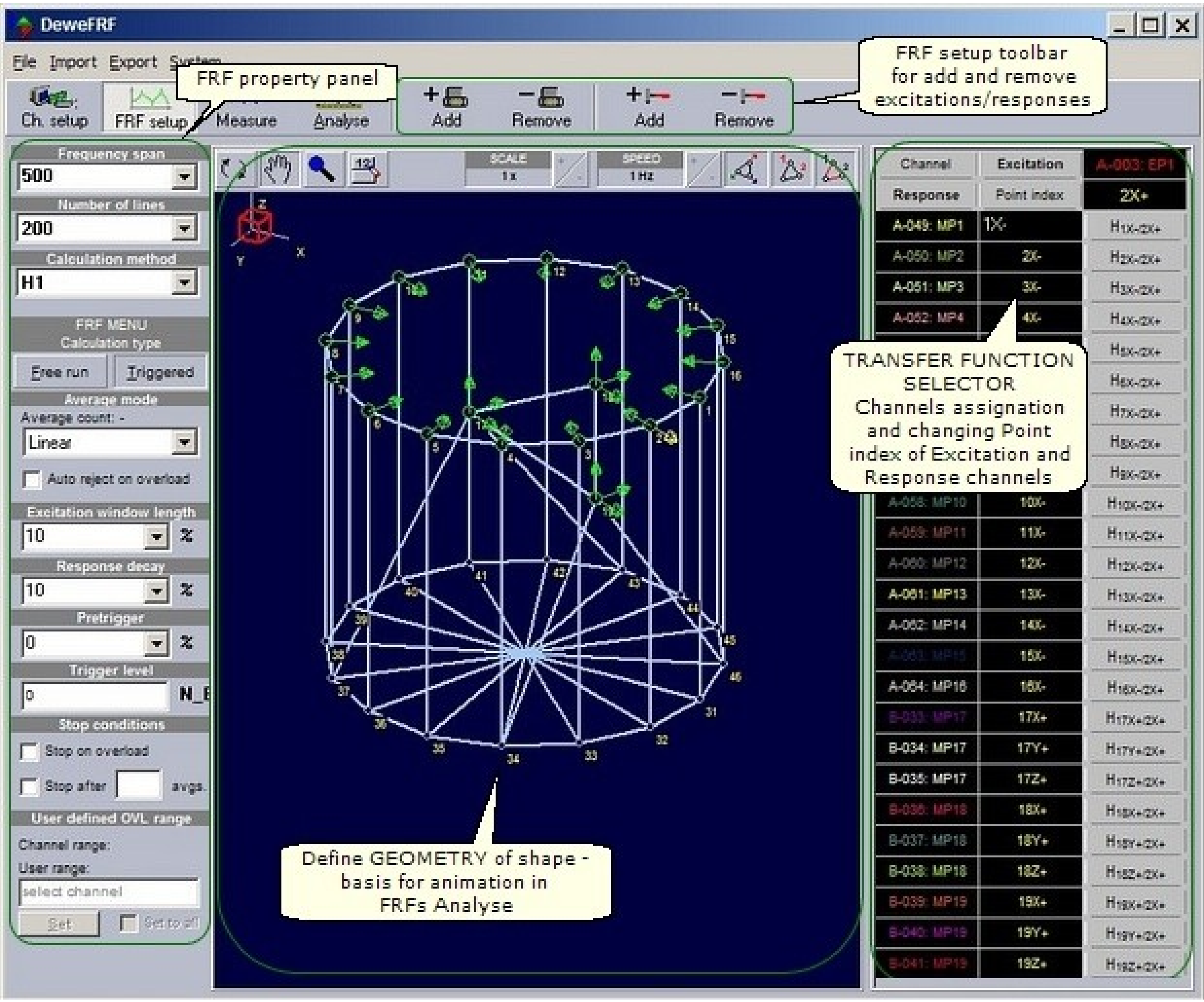
Geometry of mechanical structure with nodes, tracelines as well as global coordinate systems

More about Import/Export FRF setup data as Project data file *see:* User Guide → **Data management**

When the Channel and FRF setup are **done**, go to the *next step* → **Measure**

For entering FRF setup, click the  icon.

for explanation click on field of interest



4.1 FRF property panel setting

Some settings are set in FRF property panel:

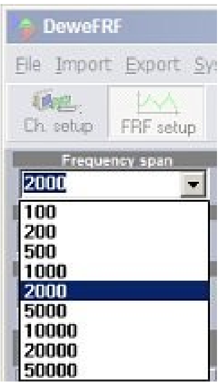
Common settings	• **Frequency span** • **Number of lines** • **Calculation method**	• **Averaging mode** • **Stop conditions** • **User defined OVL range**
Continuous (shaker) excitation	• **Calculation type** • **Window type** • **Overlap**	
Impulse (hammer) excitation	• **Calculation type** • **Window type** • **Excitation window length**	• **Response decay** • **Pretrigger** • **Trigger level**

4.1.1 Common

Few common settings are set regardless of *excitation* type:

- **Frequency span**
- **Number of lines**
- **Calculation method**
- **Averaging mode**
- **Stop conditions**
- **User defined OVL range**

Frequency span



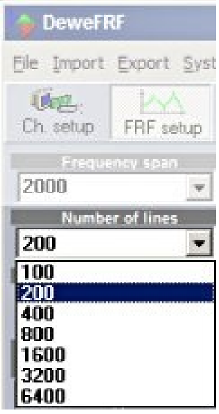
The **frequency span** defines the upper limit for calculation and the **number of lines** defines on how many segments this frequency span will be split. In the example above the 500 Hz span is split in 200 lines, resulting in 2.5 Hz frequency resolution.
The maximum sample rate allowable will vary according to *which* **A/D** board you have installed, and *how many* channels are activated for recording.

In the drop down-list choose appropriate frequency span.

Note:

The sampling rate is set to 2*1.28*Frequency span. The factor 2 is the Nyquist factor for sampling and 1.28 is used to assure that the results will be *alias free*.
So if the frequency span is 2000 Hz, the sampling rate will be set to 5120 Hz.

Number of lines



Number of lines combined with frequency span defines the *resolution of FRF*. The span frequency is equally divided in number of lines.

If we have 200 lines and the frequency span is 2000, the line resolution will be 2000/200=10 Hz.

Calculation method

There are three calculation methods available: **H1**, **H2** and **Hv**.

H1 minimizes the *noise on input* and is quite commonly used procedure.

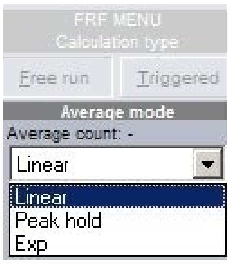
H2 minimizes the *noise on output*, but can only be applied if the number of input and number of outputs are *equal*.

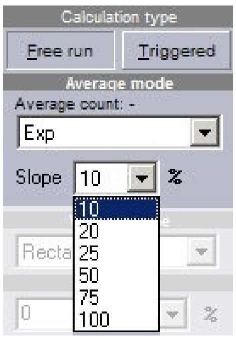
Hv tries to minimize the *noise on input* as well as the *noise on output*, so it's the best procedure. But it takes more calculation power than H1.

For further reading please see:

Reference Guide → **Theory of frequency analysis**

Averaging mode





The average type can be **Linear**, **Peak hold** or **Exponential**.

Linear average means that each FRF will have *equal weight* in the final result.

Peak hold average will take the result for each individual frequency line where the excitation will be of the *highest* value.

With Exponential averaging the FRFs will have *less* weight with time.

When **Exp** average type is selected the slope defines the weight of the *last* average.

Stop conditions

Stop on overload will **stop** the measurement and the function generator if the input or output channels will go *over* the limit of physical measurement range or the user defined range.

Stop after option will **stop** the measurement *after* a predefined number of averages .

User defined OVL range

If we select to *automatically stop* on overload or to *ignore* overloaded trigger shots, the user defined OVL range defines the limits of the forces and responses which the structure allows.

First of all we have to select one channel (either excitation or response). If the channel doesn't have the user defined range, then the entry field shows "Not set".

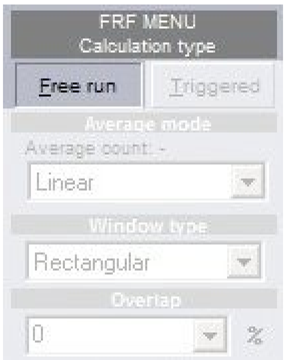
If we want to enter the *range*, we have to type in the value and press **Set** button. If we want to set the values for all the channels, the **Set to all** checkbox have to be set **prior** to clicking on the "Set" button.

4.1.2 Continuous (shaker) excitation

Several settings are set in FRF property panel → FRF MENU for **Continuous (shaker) excitation** type:

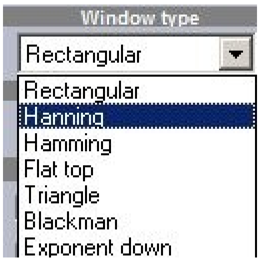
- **Calculation type**
- **Window type**
- **Overlap**

Calculation type



If the excitation is *shaker* or *electrical* signal of any kind of *noise* or *sine* sweep, the **Free run** method is used to calculate the FRF.

Window type

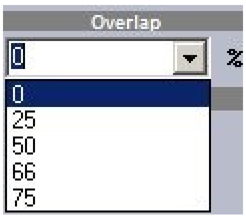


The window type defines the FFT window which will be the *same* for *all* excitation and response.

The window adds the *damping of frequency* modes, but they are necessary to avoid *leakage* (on sweeps or noise).

The rectangular window is therefore not recommended, but **Hanning** or **Blackman** would be a good choice.

Overlap



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

66,6% overlap is the minimum to ensure *same weight* for *each* sample.

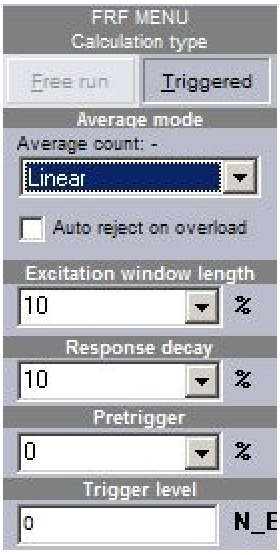
For further reading please see:

Reference Guide → **Theory of frequency analysis**

4.1.3 Impulse (hammer) excitation

Several settings are set in FRF property panel → FRF MENU for ***Impulse (hammer) excitation*** type:

- **Calculation type**
- **Window type**
- **Excitation window length**
- **Response decay**
- **Pretrigger**
- **Trigger level**



Calculation type

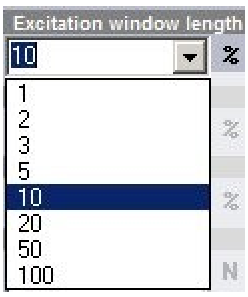
Triggered acquisition is used if the excitation is *impulse* (like with *modal hammer*), *chirp* or *burst*.

Averaging mode

The *additional* option common in averaging is to automatically reject the *overloaded* FRF. In this case if any of the signals (either excitation or response) goes *to overload*, this FRF will not be taken into the *final averaged result* but will be automatically rejected.

Window type

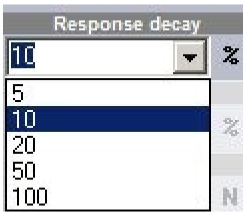
The window for excitation channels is **automatically Rectangular** with a predefined window length.



Excitation window length

The window will have a *factor* of **1** for the *defined percentage of values* and a value of **0** for the *rest* of the signal.

This can **reject** the *double hits* on excitation signal, however the use of it is doubtful since the response to a double hit can't be rejected.

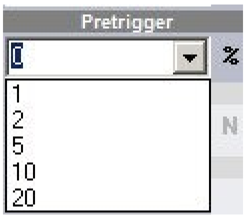


Response decay

The window for response is **automatically** an exponent *down* with a defined decay. Decay is the *attenuation factor* at the *final sample* of the signal.

So if we define 5 % response decay, the final *window value* will be **0.05**.

The response decay **increases** the *damping of the structure*. The response decay can be used if the structure damping is low to reduce amplitudes at the end of calculation window.



Pretrigger

Pretrigger defines the *number of values* which are taken before the *actual* trigger **occurs**.

Trigger level

Is the *level* of the *first excitation channel* where the FRFs are acquired. It *always triggers* on the **positive** edge of this level.

The trigger level and pretrigger value can also be chosen *visually* in excitation scope.

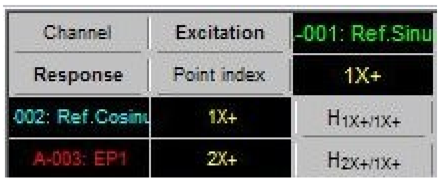
The **trigger level** is visually represented by the yellow horizontal line. The cursor changes to *arrow* when the mouse is moved over this line. Then we can move the line with left mouse *click and drag*.

The **pretrigger** is shown as the vertical yellow line and the value can be defined by *clicking and dragging* this line as well.

4.2 Transfer function selector

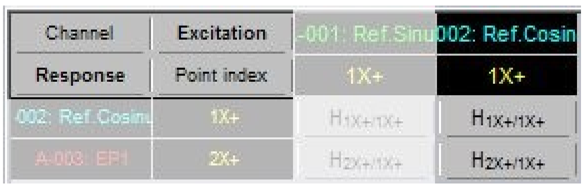
The transfer function selector shows the matrix of selected *excitation* and *response channels*.

When the excitation and responses are *selected*, the table *shows* **all the transfer functions** which will be *calculated*.



Add excitation and response channels

By pressing the  (Add excitation) button on the FRF toolbar we **add** the *excitation* channels:



By pressing the  (Add response) button on the FRF toolbar we **add** *response* channels.

Define the channels

Then we can define the channels by **selecting** them from the channel list for each excitation and response.

Point index

The point index can be **directly obtained** from the *channel name* if the correct convention is used (see Channel naming section in **Channel setup**) or it can be **written in by hand** (by pressing the first column) in form of:

1x+ (first the *point index*, then the *direction* x, y or z and finally the *sign* + or -).

Arrow

With *direction* and *sign* of point index is also defined arrow in this point. When in the Geometry window of **FRF Setup** "*Display arrow*" switch is **On** the arrows display color is:

- *yellow* for excitation
- *green* for response.

Single input single output modal analysis – SISO

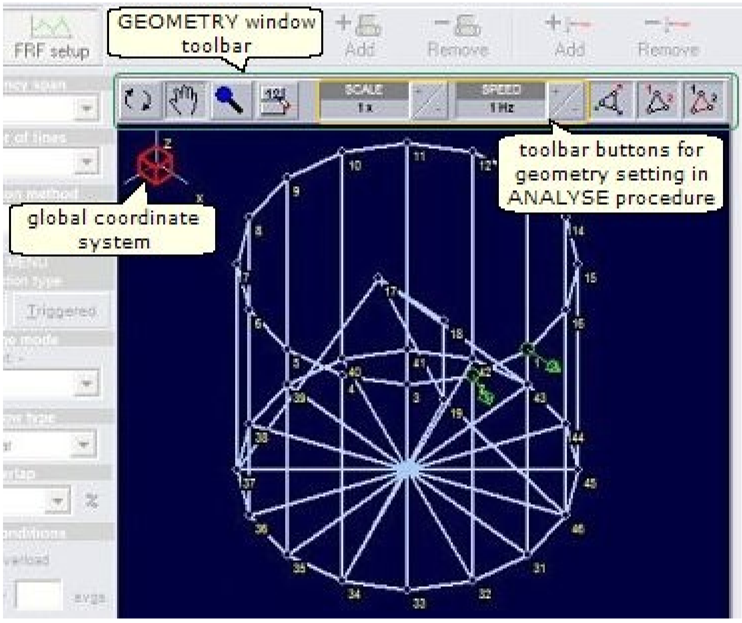
In the FRF setup only the *starting* points are **defined**. Then we can define all the parameters for the testing (bandwidth, number of lines, excitation type etc.). For details *see* DeweFRF Tutorial → SISO - single I/O modal analysis.

4.3 Geometry window

In the Geometry window we can see the geometry *nodes* (points*)* of mechanical structure to which **channels** for **excitation** and **response** are assigned → *see also* Transfer function selector, as well as *trace lines, coordinate system* and when entering, *arrow* with *indexes*, *directions* and *signs* will be shown.

It is very helpful to load the geometry to get a feeling *where* to locate **excitation** and **response**. About Import/Export Geometry data *see*: User Guide → Data management → **Geometry**.

for explanation click on field of interest



The main part of the geometry window shows GEOMETRY of construction with *points* - *nodes*, *trace lines*, *arrow* and *directions*.

On left side of geometry display there a is symbolic picture of GLOBAL COORDINATE SYSTEM.

Geometry window **TOOLBAR** is divided in three parts:

1. MANIPULATE with geometry

2. Geometry SETTING FOR ANALYSE procedure - SCALE and SPEED (not relevant for FRF setup)

3. Geometry DISPLAY SETTING

Geometry of mechanical structure for FRF testing is usually defined in **Point Maker** or in another appropriate software.

For detailed information see:

FRF setup → **Point Maker**

To set up **Point Maker** click the icon.

The geometry in above screen describes the demo structure as seen below:

Geometry is stored in **.unv* - *Universal file format* data file and can be imported from this in **DeweFRF**s **FRF setup** file with Import menu → **Geometry**.

for information about Import/Export Geometry see → User Guide → Data management → **Geometry**

Arrows are defined in **Transfer function selector** (Channels assignation).

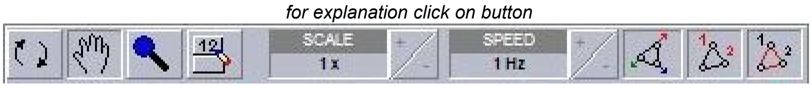
SUGGESTION

because Geometry *and* Modal FRF Data *are stored in* __*same*__ **.unv (Universal format) file it is strongly recommended to use separate filename for new or modified* Geometry*.*

4.3.1 Geometry window toolbar

Some tools are accessible on FRF Geometry window toolbar:

for explanation click on button



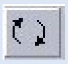
MANIPULATE with geometry	• **Pan** • **Rotate**	• **Zoom** • **Point Maker**
SETTING FOR ANALYSE	Scale and Speed → *see* Analyse - **Geometry window**	
DISPLAY SETTING	• **Arrow** • **Nodes number**	• **Trace line**

4.3.1.1 Manipulate geometry

All tools are accessible with press on appropriate button - icon:

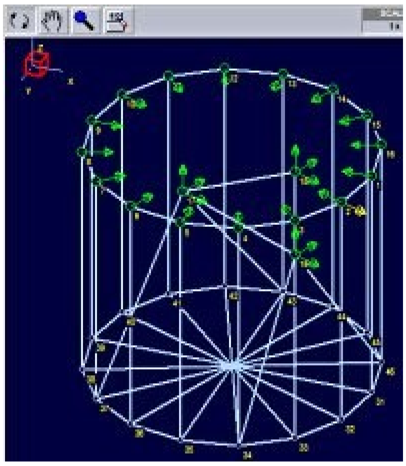
Pan

When this button is pressed, we can left-click and drag the mouse inside the geometry window to *move* the geometry on geometry window. Even if rotate is chosen option, we can pan with Shift-click on geometry display.

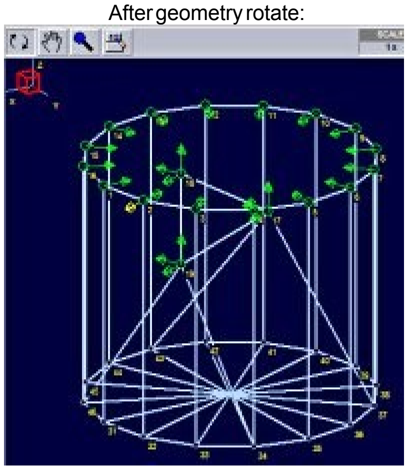


Rotate

When this button is pressed, we can left-click and drag the mouse inside the geometry window to *rotate* the geometry on geometry window.



After geometry rotate:



Zoom

When this button is pressed, we can left-click and drag the mouse anywhere inside the geometry widow from outside *to center* to *extend* (zoom in) the geometry or drag the mouse from inside outward to *contract* (zoom out) the geometry. Even if rotate is the default option, we can zoom with Control-click on geometry.

Point Maker

With this button we can set up Point Maker for define *new* or modify *existent* geometry.

For detailed information see: FRF setup → **Point Maker**

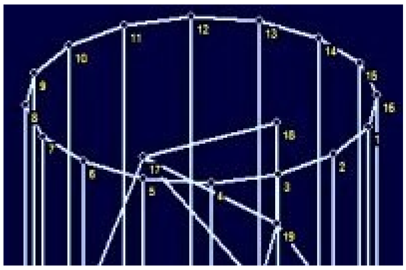
4.3.1.2 Display setting

All geometry display tools are accessible with pressing on appropriate button - icon. When you choose these tools it is very important what part you wish to see *clearly*.

Arrow

When this button is pressed, we can observe in the Geometry window *arrows* and *directions* of excitation or response in constructions points. This is an **on**/**off** switch button.

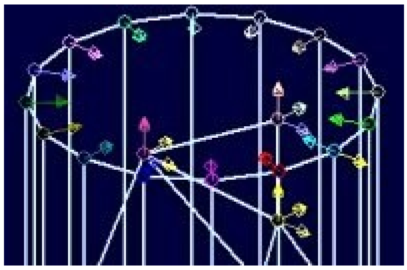
Arrow switch is off:



Nodes number

When this button is pressed, we can observe all *numbers* of *nodes* assigned to construction. This is **on**/**off** switch button.

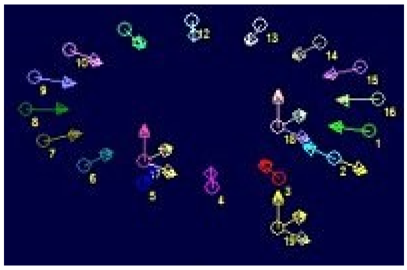
Nodes number switch is off:



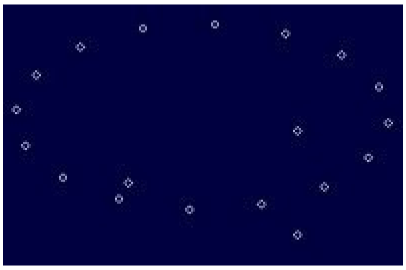
Trace line

When this button is pressed, we can observe all *trace lines* between *nodes* assigned to construction. This is **on**/**off** switch button.

Trace line switch is off:



When you switch **all** buttons *off*, you can see a screen like this:



4.4 Point Maker

Point Maker is tool to *graphically* define and represent measured nodes and objects (*grids* and *circles*)- points of mechanical structure for which *tracelines* and **channels** are defined for each **excitation** and **response.**

Point Maker can be opened with:

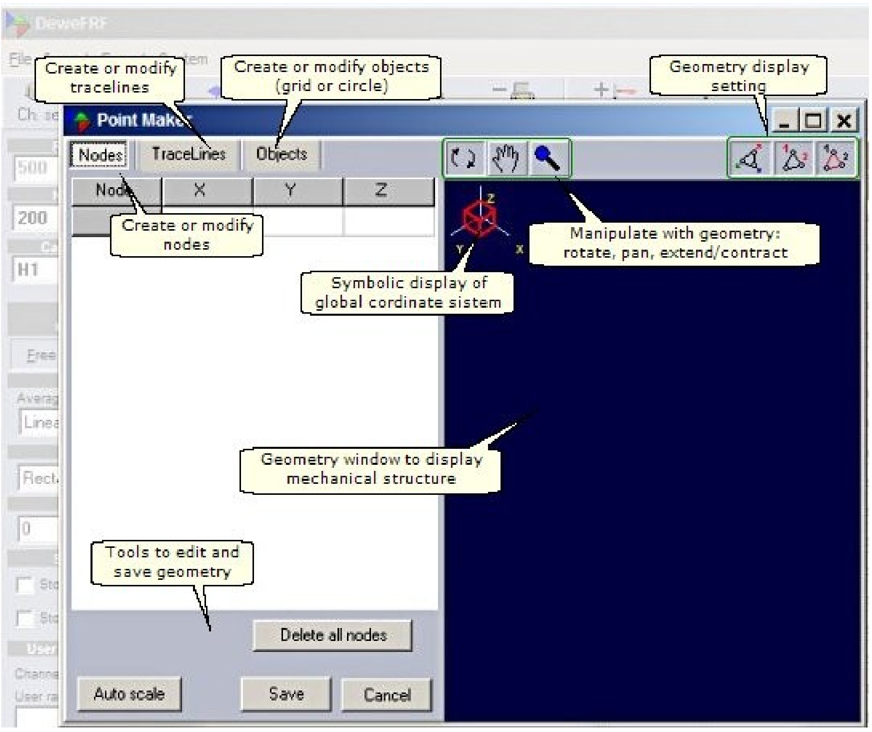
- click tool of Geometry window toolbar in **FRF setup** or **Analyse** procedure
- selecting **New Geometry...** from FRF File menu

With **Point Maker** we can define new or modify:

- **Nodes**
- **Trace Lines**
- **Objects**

Point Maker has several **Tools** for your assistance.

for explanation click on field of interest



After defining *new* Geometry of structure in **Point Maker** export (save) it from **DeweFRFs FRF setup**.

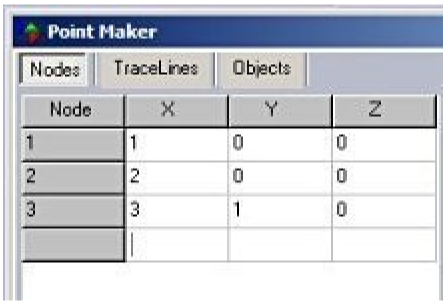
For information about Import/Export Geometry see: User Guide → Data management → **Geometry**

SUGGESTION

because Geometry *and* Modal FRF Data *are stored in* *<u>same</u>* *.*unv* *(Universal format)* *file it is strongly recommended to use separate filename for new or modified* Geometry.

4.4.1 Nodes

Node (point) has to be defined with an X, Y and Z *coordinate* input in the "Node" table. Each coordinate needs to have a value (even the 0 value).



In order to enter a coordinate value the cell for the value must be left-clicked and the value entered. Coordinate value can be only a number.

The coordinate is chosen either by the Tab key or again by left clicking the cell.

If the X and Y coordinate are already defined as soon as the first number for the Z coordinate is entered a new coordinate line for a new node to be entered is displayed automatically. The current node is enumerated (Node column) with a consecutive number.

If the entry of the Z coordinate is followed by pressing the Tab key then the cursors automatically goes to the X coordinate of the next node. Of course, to edit an existing node coordinate or to create a new node, we can always left click on the appropriate cell.

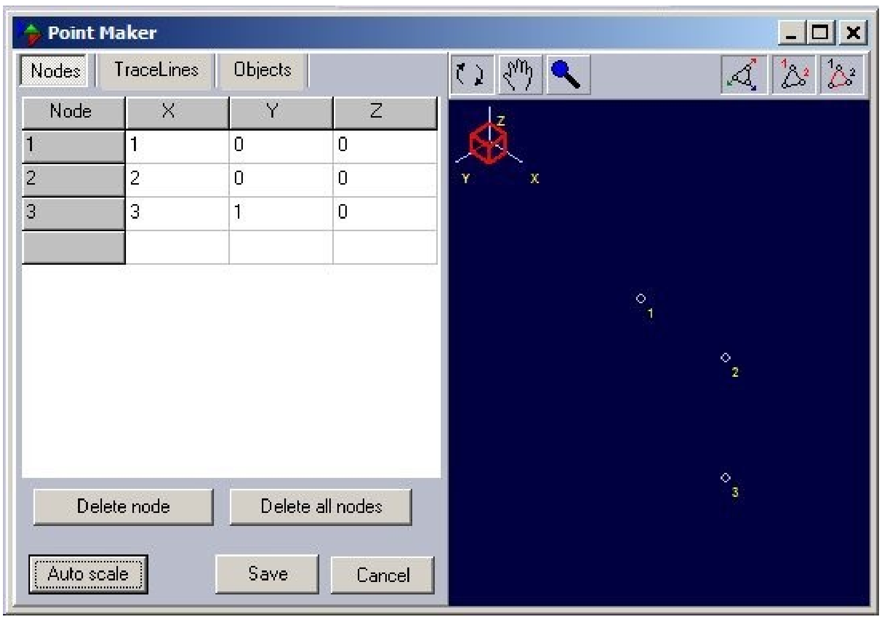
For detailed description action with keyboard keys:

see → **Navigation keys**

First the **Nodes** has to be selected by clicking from the Point Maker menu.

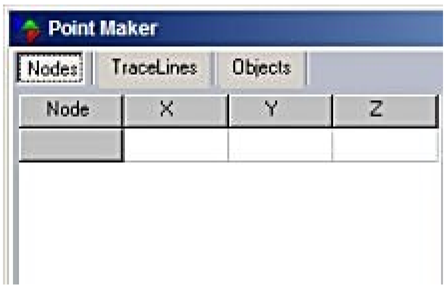
There are three following basic procedures in **Point Maker** for nodes manipulation:

- **New node**
- **Edit node**
- **Delete node**



New node

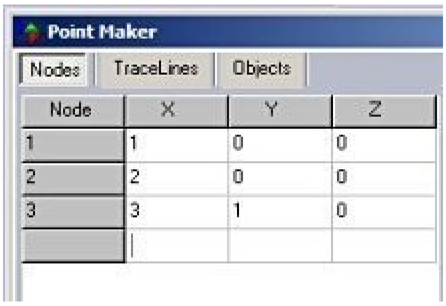
When we select File menu → **New Geometry...** or when Delete all nodes the following table is created:



In order to enter a coordinate value the cell for the value must be left-clicked and the value entered.

Input for X coordinate can also be initiated by double clicking the header of column (Node, X, Y, Z).

If the entry of the Z coordinate is followed by pressing the Tab key then the cursors automatically goes to the X coordinate of the next node.

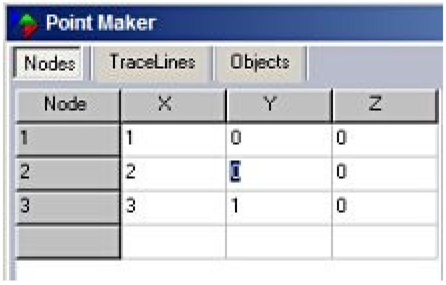


We select the coordinate to be entered by left clicking on the coordinate cell and entering the given value.

A value must be entered for all the coordinates of each Node (even 0) !

Edit node

The field of the coordinate we would like edit (modify) is selected by left clicking and entering the desired value. In the example below this is the Y coordinate of node 2.

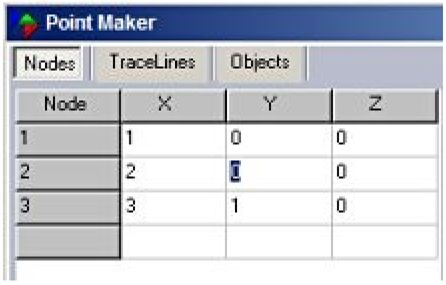


If the entry of the Z coordinate is followed by pressing the Tab key then the cursors automatically goes to the X coordinate of the next node.

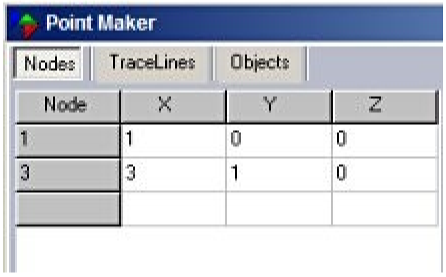
A value must be entered for all the coordinates of each Node (even 0) !

Delete node

By left clicking on the cell we select the coordinate of the point we would like to delete. In the example below we would like to delete node 2 so the Y coordinate of node 2 is selected (we could also select X or Z coordinate of the same node).



In **Point Maker** tools for deleting a node the Delete node is clicked:



To delete ALL the nodes at the same time in **Point Maker** tools Delete all nodes button is clicked.

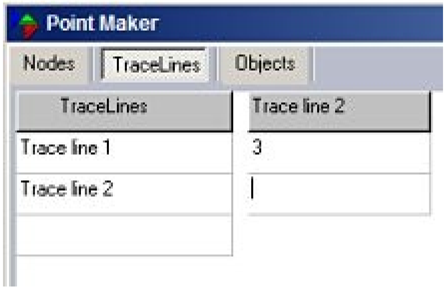
Nodes do not renumerate (above right table shows node 2 is deleted) !
All TraceLines attached to the deleted node are **also** deleted !

Navigation keys

←	to the previous cell (in the same line); in number one digit left and from first to the previous cell
→	to the next cell (in the same line); in number one digit right and from last to the next cell
Tab	to the next cell (from Z also to X of the next line)
Shift+Tab	to the previous cell (from X also to Z of previous line)
↑	up one cell in the same column (up one node)
↓	down one cell in the same column (down one node)
Home	to the X coordinate of the same node; in number to leftmost digit and then to the X coordinate
End	to the Z coordinate of the same node; in number to rightmost digit and then to the Z coordinate
PgUp	first node in the column
PgDn	last node in the column
Enter	select the entire number in the cell
Backspace	deleting the left digit in number while editing the coordinate or entire number when this is selected
Delete	deleting the right digit in number while editing the coordinate or entire number when this is selected

4.4.2 Trace Lines

Trace line is defined with the input of the *start*, *end* and all the nodes *between* in the "TraceLines" table.



The column "TraceLines" is selected using the left click. In this column the name for the trace line is entered.

During the input for the name of the trace line an empty cell for the entry of the next trace line is displayed in the left column. We can select it by left clicking.
The name for the trace line is also displayed in the header of the right column which also contains the first empty cell for the first node input.

The cell for the node input is selected using the left click. During the entry of the node the cell for the next node is created below. Value of node is only a number.

The next cell is selected either by left clicking or using the Tab key.

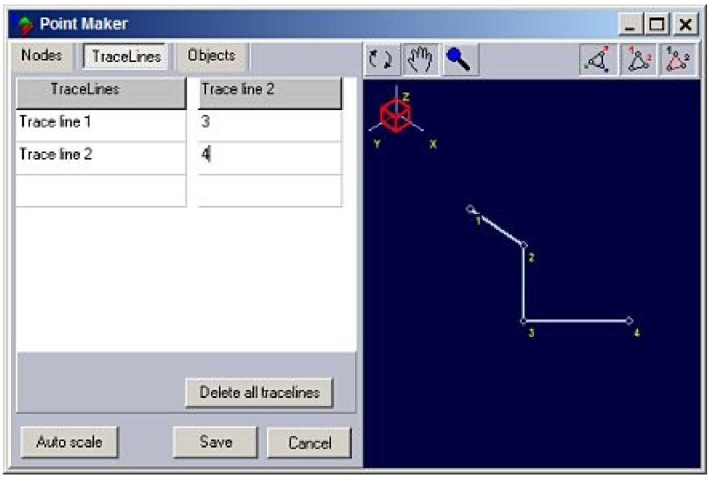
For detailed description action with keyboard keys

see → **Navigation keys**

First the **TraceLines** has to be selected by clicking from the Point Maker menu.

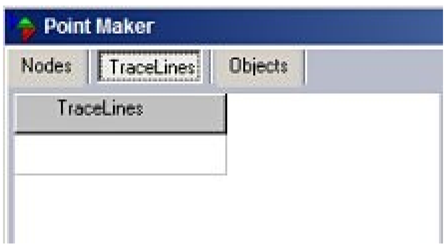
In the **Point Maker** the following basic procedures for trace lines exist:

- **New trace line**
- **Edit trace line**
- **Delete trace line**



New trace line

When we choose File menu → **New Geometry...** or when in geometry there is no defined trace line, then selecting **TraceLines** on toolbar creates the following table:

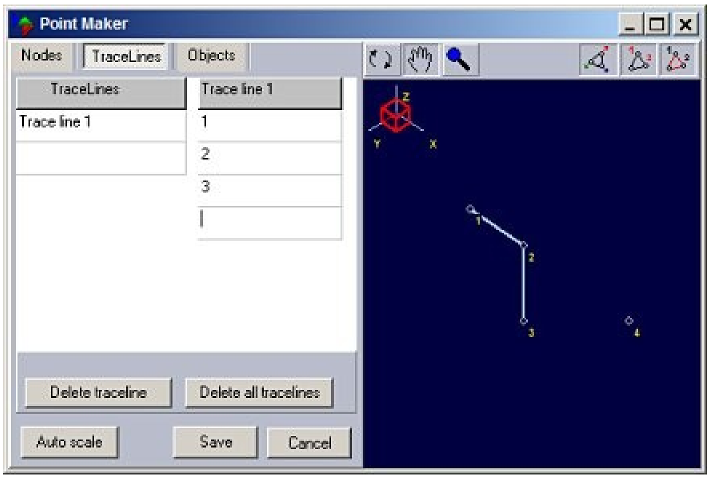


The column "TraceLines" is selected using the left click. In this column the name for the trace line is entered.

The name for the trace line is also displayed in the header of the right column which also contains the first empty cell for the first node input.

The cell for the node input is selected using the left click. During the entry of the node the cell for the next node is created below.

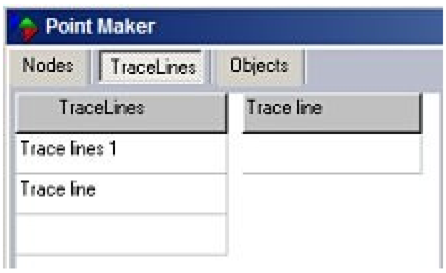
The next cell of the node list is selected using the Tab key or left clicking. At the same time the trace line is displayed in the Geometry window.



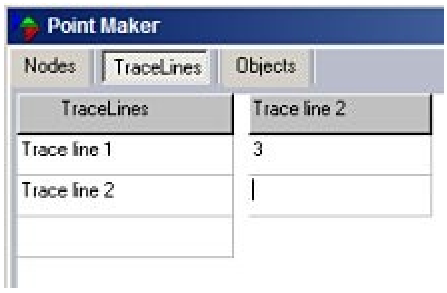
The name for the trace line is also displayed in the header of the right column which also contains the first empty cell for the first node input.

The cell for the node input is selected using the left click. During the entry of the node the cell for the next node is created below.

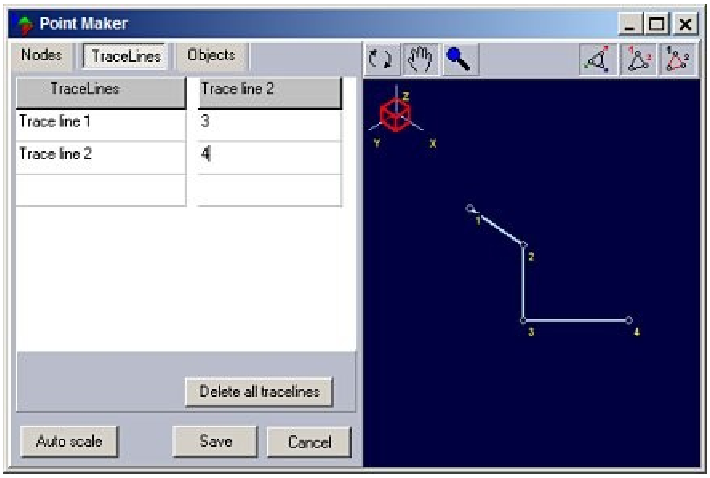
During the input for the name of the trace line an empty cell for the entry of the *next* trace line is displayed in the left column. We can select it by left clicking.



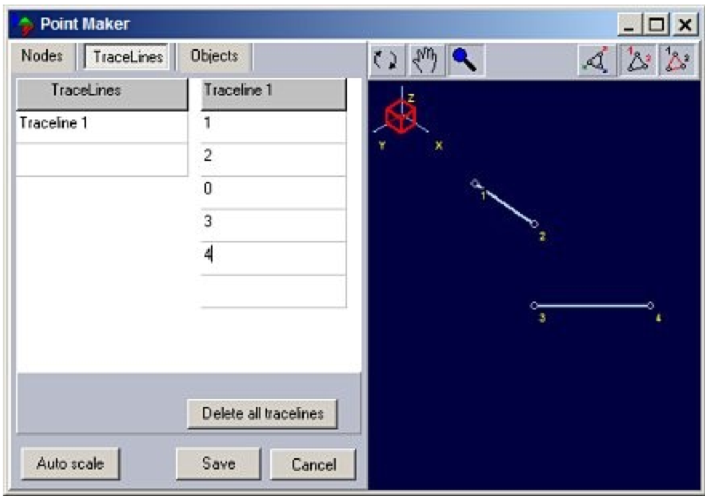
The cell for the node input is selected using the left click. During the entry of the node the cell for the next node is created below.



The next cell of the node list is selected using the Tab key or left clicking. At the same time the trace line is displayed in the Geometry window.



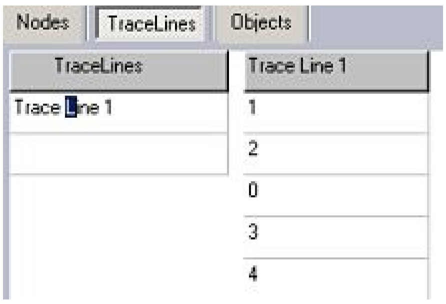
Trace line can be interrupted by entering the value of **0** in the node field. We can proceed with the next node after:



Edit trace line

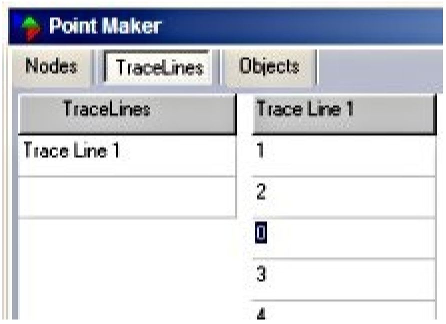
Name

Using the left click we select the cell with the trace line to be changed - at the same time the header for the right column is being modified.



Node

Using the left click we select the cell with the node marking to be changed.



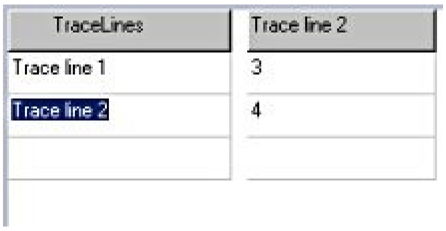
In the Geometry window the trace line display is also accordingly modified.

Delete trace line

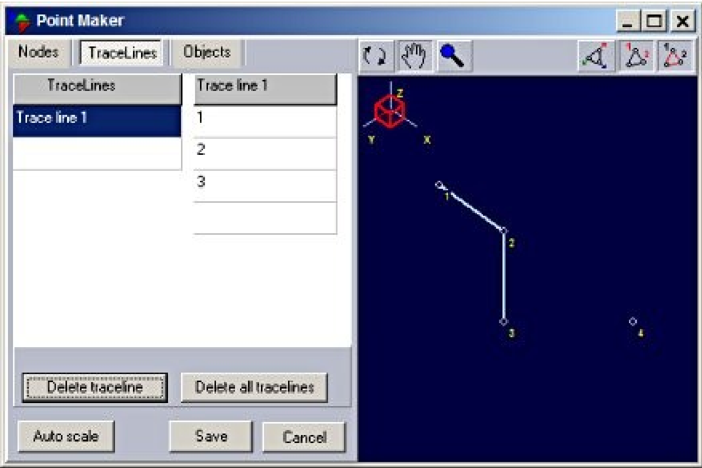
Deleting trace lines **Point Maker** can be accomplished in two ways:

1. With deleting the *trace line*:

   Using the left click we select the cell with the name of the trace line to be deleted.



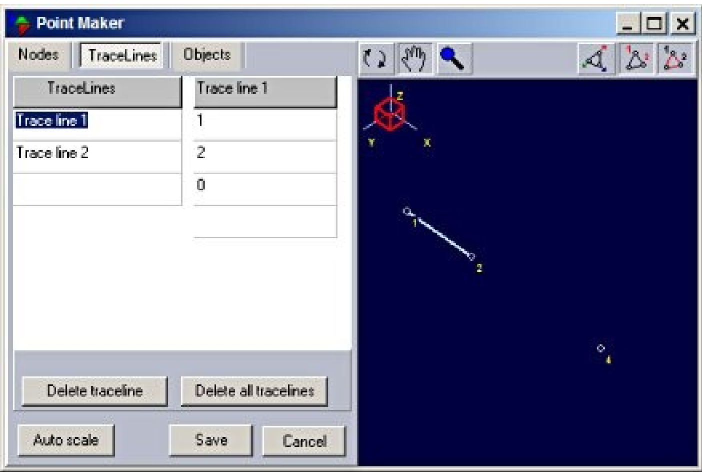
In **Point Maker** tools  can be chosen to delete selected trace line (Trace line 2 above):



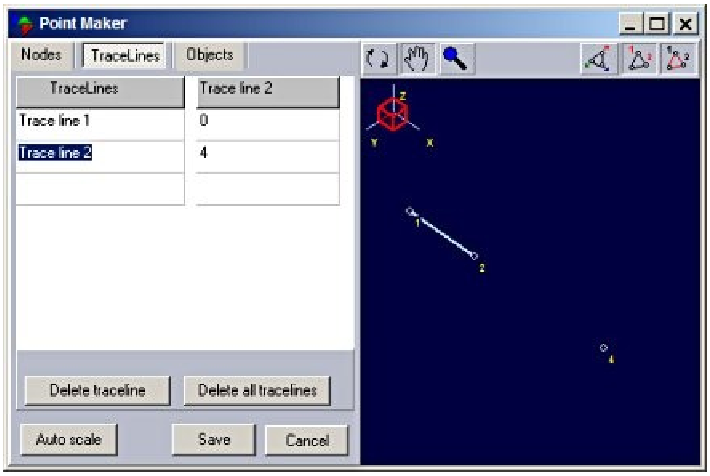
If in **Point Maker** tools  is clicked, ALL trace lines are deleted !

2. With deleting the *node*:

   With deleting the node (for example Node 3) as described in the procedure **Delete node** leaves trace line 1 as:



and trace line 2 as:



ALL trace lines connected to the deleted node are also deleted !

Navigation keys

Navigation keys are in "TraceLine" column to use in same manner as in another text processors. In right column to be worth:

←	in number one digit left
→	in number one digit right
Tab	to the next cell in the column
Shift+Tab	to the previous cell in the column
↑	up one cell in the same column (up one node)
↓	down one cell in the same column (down one node)
Home	in number to leftmost digit
End	in number to rightmost digit
PgUp	first node in the column
PgDn	last node in the column
Enter	select the entire number in the cell
Backspace	deleting the left digit in number while editing the coordinate or entire number when this is selected
Delete	deleting the right digit in number while editing the coordinate or entire number when this is selected

4.4.3 Objects

Point Maker can create the following objects faster:

- **Grid**
- **Circle**

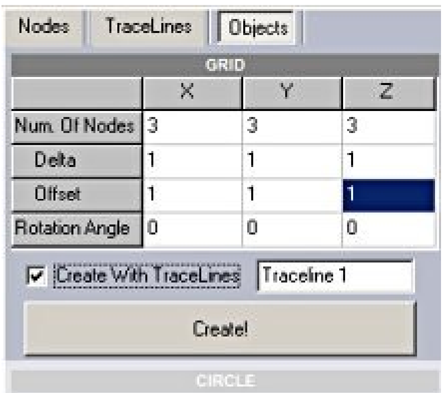
First we select by clicking **Objects** in the menu.

Grid

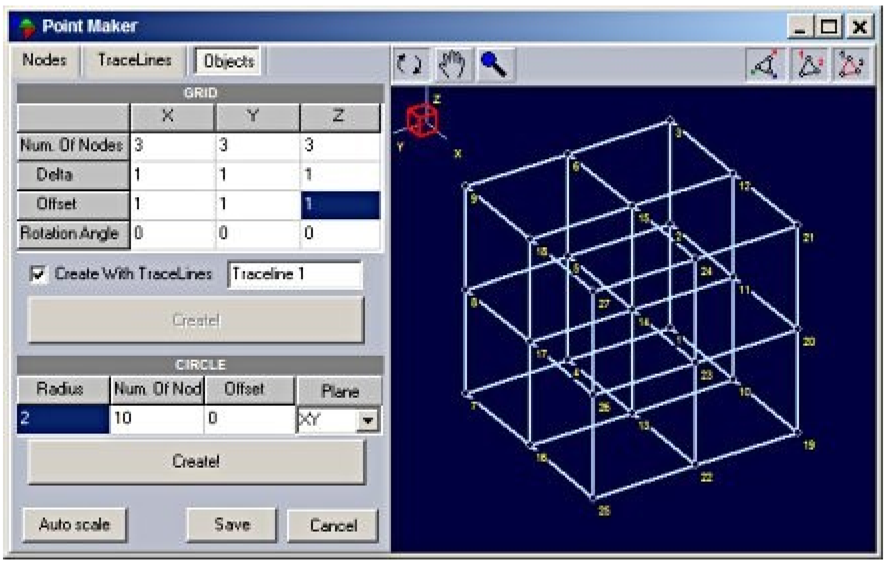
In the table of parameters the values Number of Nodes, Delta (distance between nodes), Offset (distance of first node from 0 value of coordinate) and Rotation Angle (turn around) for any of the particular coordinates X, Y in Z are entered. The choice Create with TraceLines is checked if the trace line between the nodes is desired.

Value of parameters is only a *number*. In Delta, Offset and Rotation Angle a *negative* number is allowed too.

Using the left click the given cell is selected and a value can be entered. The cells can be selected also using the Tab key, for detailed description action with keyboard keys refer to Navigation Keys.



After clicking **Create!** (under Grid parameters) a grid is created using the above parameters:

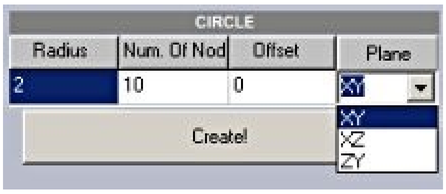


Circle

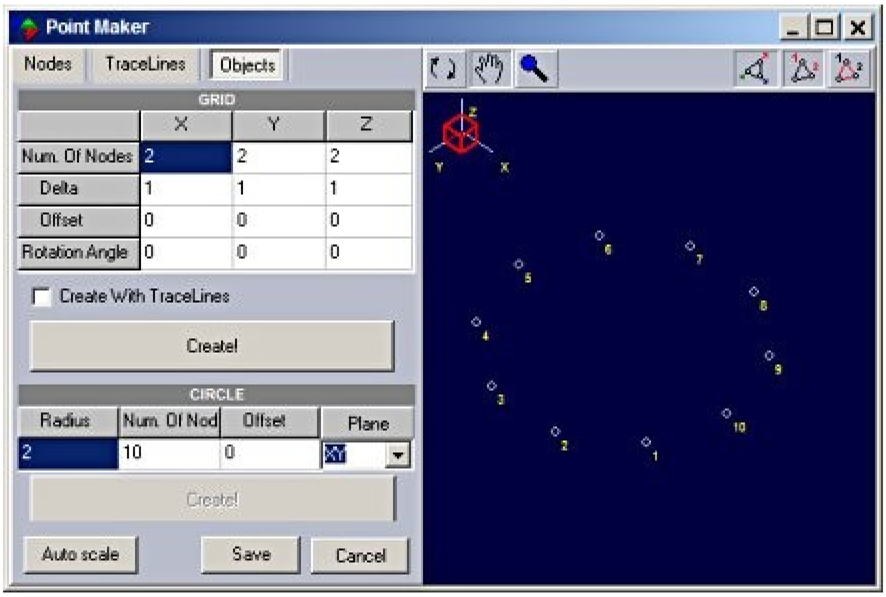
In the table of parameters values for the cells for the Radius, Number of Nods, Offset (distance from selected Plane along the coordinate perpendicular to this Plane) have to be entered and the Plane selected.

Value of parameters is only a *number*. *Negative number* is reasonable only in Offset parameter; in Radius it is not meaningful, in Number of Nods it causes unpredictable results (deletes nodes of previous created circle).

Using the left click the given cell is selected and a value can be entered. The cells can be selected also using the Tab key, for detailed description action with keyboard keys refer to Navigation Keys.



After clicking **Create!** (under Circle parameters) a circle is created using the above parameters:



Trace lines have to be defined separately.

Navigation keys

←	to the previous cell (in the same line); in number one digit left and from first to the previous cell
→	to the next cell (in the same line); in number one digit right and from last to the next cell
Tab	to the next cell (from Z also to X of the next line for Grid; from last to first cell for Circle)
Shift+Tab	to the previous cell (from X also to Z of previous line for Grid; from first to last cell for Circle)
↑	up one cell in the same column (up one parameter)
↓	down one cell in the same column (down one parameter)
Home	to the X coordinate of the same parameter; in number to leftmost digit and then to the X coordinate
End	to the Z coordinate of the same parameter; in number to rightmost digit and then to the Z coordinate
PgUp	first parameter in the column
PgDn	last parameter in the column
Enter	select the entire number in the cell
Backspace	deleting the left digit in number while editing the coordinate or entire number when this is selected
Delete	deleting the right digit in number while editing the coordinate or entire number when this is selected

4.4.4 Tools

Point Maker has the following tools for your help :

- **Common and 'Objects' tools**
- **'Nodes' tools**
- **'TraceLines' tools**

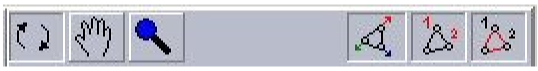
Common and **'Objects'** tools

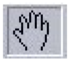
- **Auto scale** Scale display to fit in actual size of Geometry window.

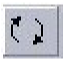
- **Save** Saves actual geometry in *Project* (not to the hard disk!) - later with particularly action together with other projects data in *.dfs -Dewesoft FRF project data file.

- **Cancel** Leaves Point Maker without any change to actual geometry.

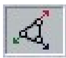
Manipulate and display geometry



- **Pan**  When this button is pressed, we can left-click and drag the mouse inside the geometry widow to *move* the geometry on geometry window.

- **Rotate**  When this button is pressed, we can left-click and drag the mouse inside the geometry window to *rotate* the geometry on geometry window.
When this button is pressed, the same is achieved with *rotating* Global coordinate system. We can left-click and drag the mouse inside the Local coordinate system to *rotate* the geometry on geometry window.

- **Zoom**  When this button is pressed, we can left-click and drag the mouse anywhere inside the geometry widow from outside *to center* to *extend* (zoom in) the geometry or drag the mouse from inside outward to *contract* (zoom out) the geometry.

- **Arrow**  When this button is pressed, we can observe in Geometry window *arrows* and *directions* of excitation or response in constructions points. This is **on**/**off** switch button.

- **Point number**  When this button is pressed, we can observe in Geometry window all *numbers* of *nodes* assigned to construction. This is an **on**/**off** switch button.

- **Trace line**  When this button is pressed, we can observe in the Geometry window all the *trace lines* between the nodes assigned to construction. This is an **on**/**off** switch button.

'**Nodes**' tool

- **Delete node** Delete *selected* node.

- **Delete all nodes** Delete *all* nodes.

Nodes do not renumerate (by 'Delete node') !
All TraceLines attached to the deleted node are **also** deleted !

- Common tools see above

'**TraceLines**' tool

- **Delete traceline** Delete *selected* traceline.

- **Delete all tracelines** Delete *all* tracelines.

- Common tools see above

5 Measure

Before trying to do the measurement, let's notice that there are *two ways* - Measure methods of making the FRFs measure.

In **FRF Measure** we can *acquire* frequency response data using applied hardware:

Measure is relevant to measuring ***directly*** (live) while **acquiring data** with continuous or impulse excitation and is represented in this chapter.	Calculate is procedure for **offline calculation** - in Ch. setup *selected* range of the stored data will be immediately recalculated with parameters selected in FRF setup → *see* Offline calculation

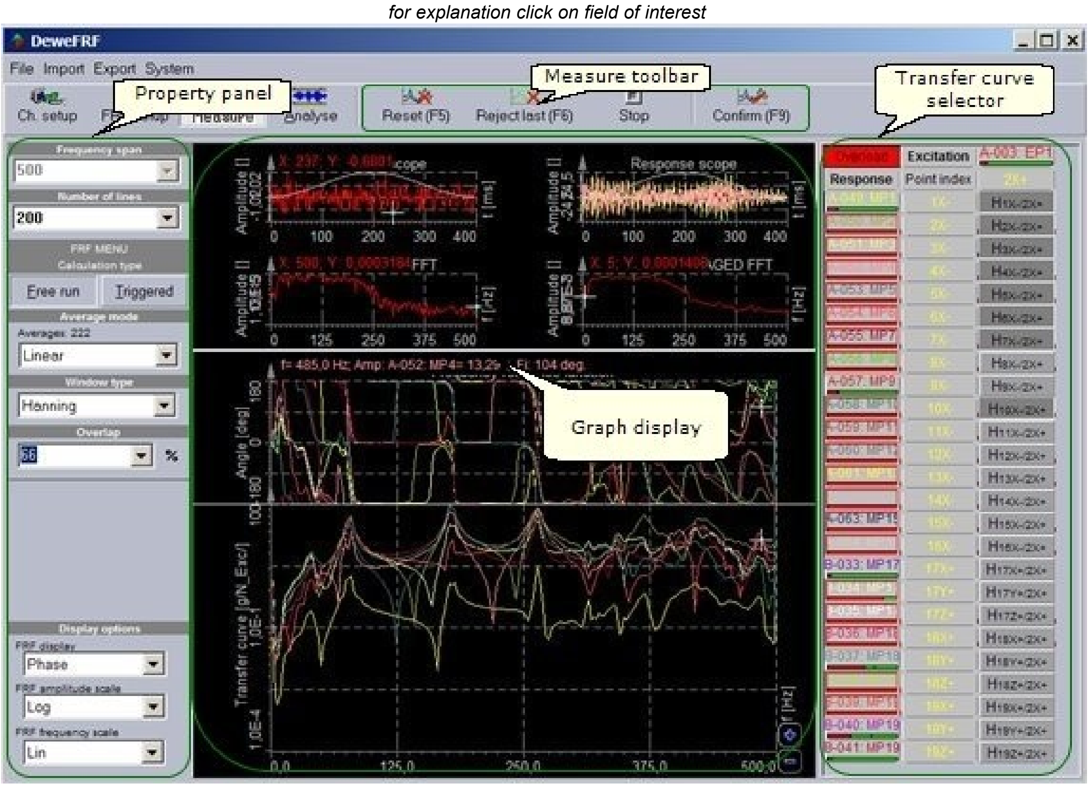
Property panel	frequency span, number of lines, calculation method and FRF Calculation type (free run or triggered acquisition) with all of its properties; same as and in fact part of FRF setup *see* → **FRF property panel setting**
Graph display	set Excitation scope, excitation FFT (current and averaged), the response scope and the frequency response function display
Transfer curve selector	transfer function list (selector what is shown on selected graph), information about the amplitudes, number of taken averages, information about last trigger
Measure	measure performing
Measure tools	help to manage and control measure

About Import/Export measured data as Modal FRF data *see:* User Guide → **Data management**

When the Channel, FRF setup and Measure are **done**, go to the *next step* → **Analyse**

The Measure can be started by clicking the  FRF tools icon.

for explanation click on field of interest



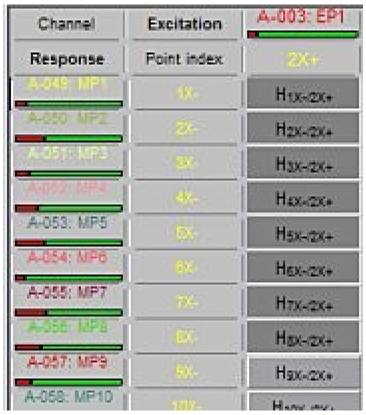
The measurement consist of three major parts of the screen: property panel, **graph display** and **transfer curve selector**.

The **property panel** is the *same* as in FRF setup - *see:* User Guide → **FRF property panel setting**.

Transfer curve selector

The right side shows some information about the **acquisition** procedure.

It displays the information about *last trigger* shot (if it's ok, it is green, in case of overload it is red). The number on the right side shows the number of *taken averages*.



The lower transfer function list is the **selector** which *transfer function* or *channel* is shown on selected graph, but also provides the information about the *amplitudes* of each input and output.

In case of **overload**, the bar below the channel names turns out in *red*. If the overload is gone, the bar turns back to *green*, but still it shows the *red border* around the channel indicating that the channel was overloaded *once* during the measurement.

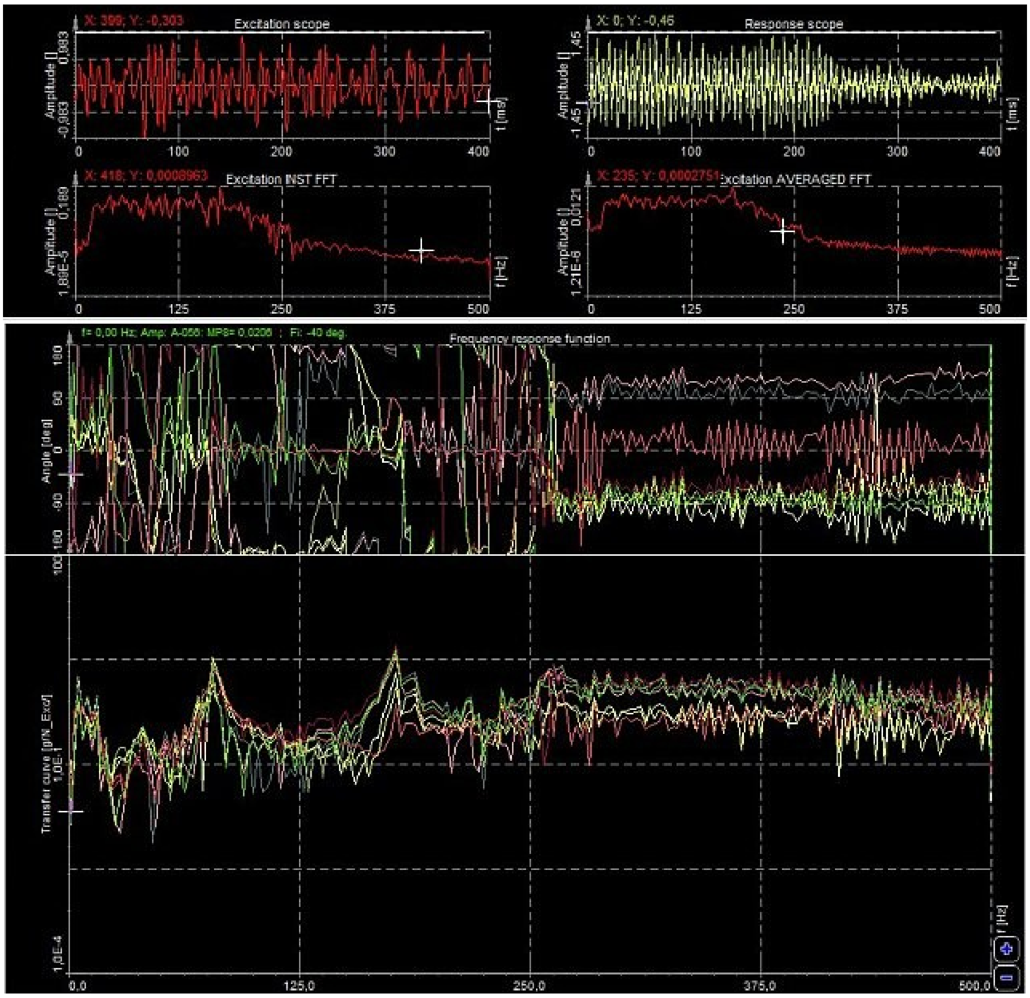
5.1 Graph Display

Central part of Measure screen *displays* graphs. We can:

- **select channels** which are displayed
- **set display properties** of particular graph.

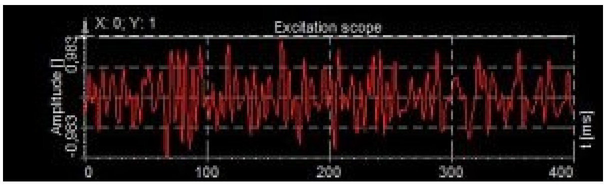
The mouse moved over the graph will show cursor *values* for nearest point and if we click on this point, the amplitude and frequency will *remain* shown.

The graph display:

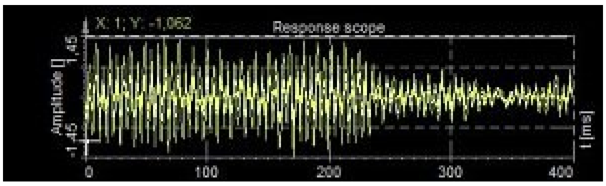


consists of the displays:

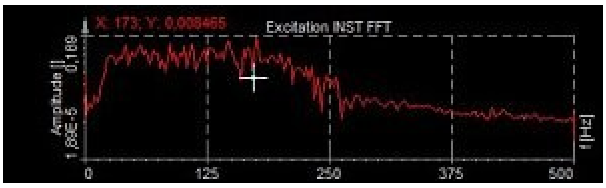
- Excitation scope

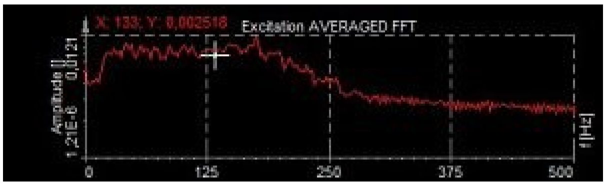


- Response scope

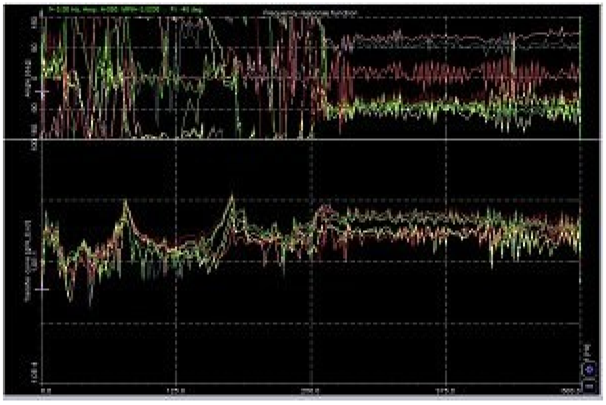


- Excitation FFT *current* and *averaged*



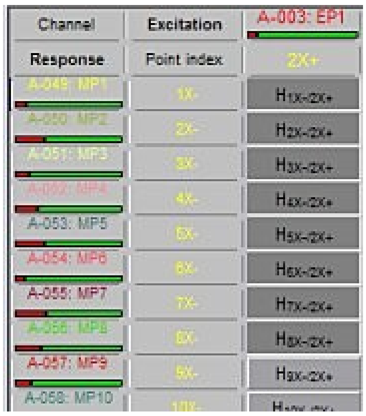


- Frequency response function



Selection of channels

To select *channels* or *properties* for each display, click on it and it will get a *white border* around it which shows that this graph is selected:



The items *which* are displayed on each graph can be chosen from the right side - transfer function list (table) with left click on channel:

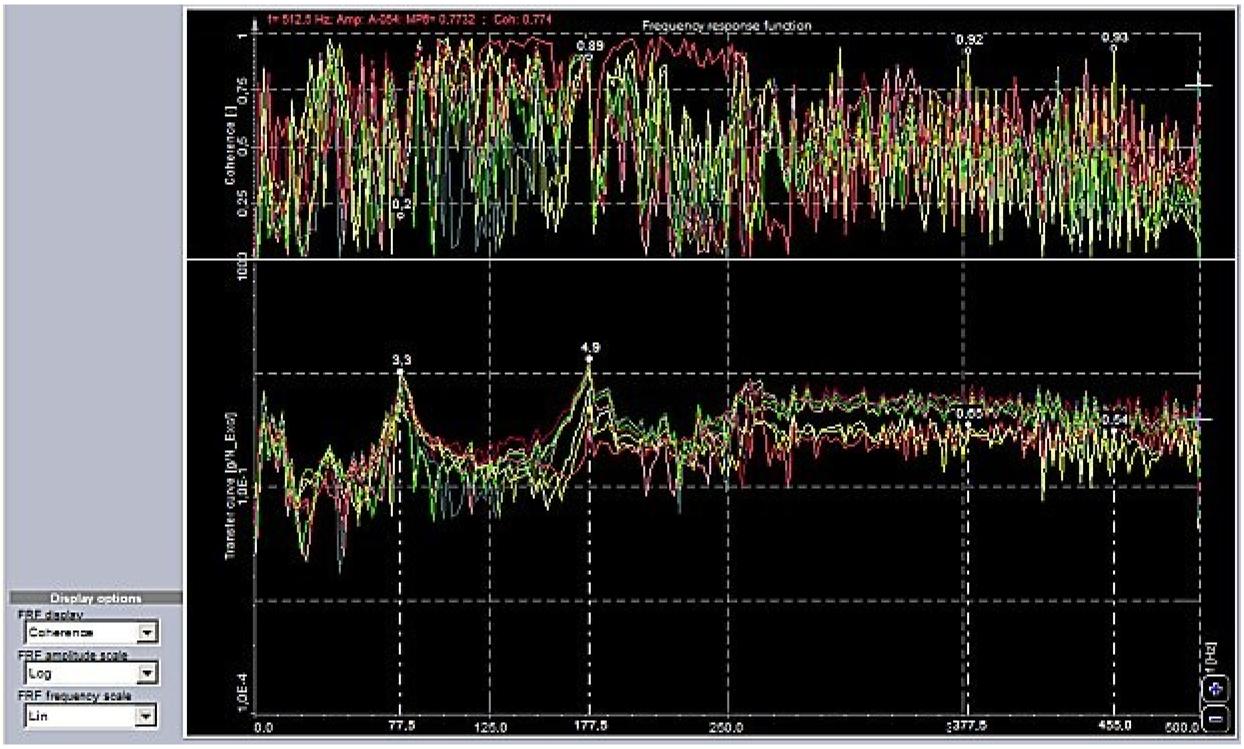
- for *excitation scope* and *FFTs* the **excitation** channels can be selected (*see* example on left),
- for *response scope* and for *FRF transfer functions* the **response** channels can be selected.

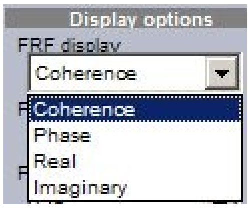
NOTE *screen display selection is limited to maximum* 8 **channels**.

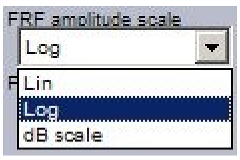
Display properties

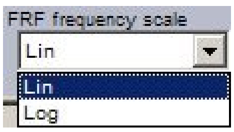
Display option

Frequency response function









For FRF we have several choices:

- on the upper - FRF display we can show *coherence*, *phase*, *real* or *imaginary* display.

  The **coherence** is the ratio between vector and scalar sum of single FRF shots and shows *how well* the FRF shots are correlated to each other.
  The coherence of **1** shows *perfect* correlation, coherence of 0 shows *no* correlation.
  When we select coherence and if we click on coherence curve the value of coherence in this point is displayed (*see* graph above).
  The coherence is used when *multiple* data is acquired for single FRF.

- The other two selections are *linear* or *logarithmic* amplitude and frequency plots.

Excitation / Response scope

For Excitation / Response scope we can select auto scale y coordinate.

Excitation current / averaged FFT

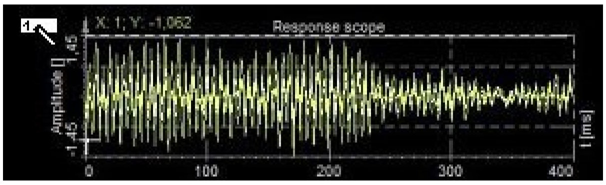
For Excitation current / averaged FFT we can select:
- auto scale y coordinate
- logarithmic x axis .

Coordinates Scaling

Each graph has x and y *coordinates scaling* possibilities.

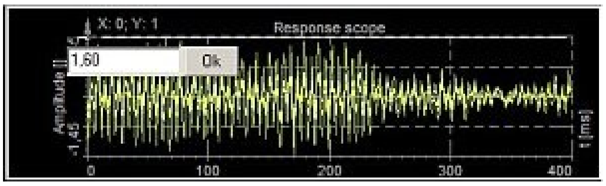
1. Edit minimum or maximum value for x and y coordinate

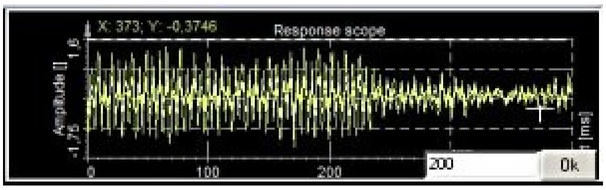
This works for *all* graphs except with only one exception - upper one of Frequency response function display.



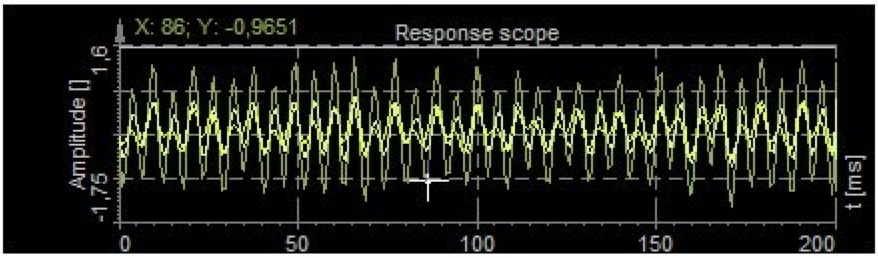
If we move with the mouse over the minimum or maximum value for x and y, we can enter custom value by clicking with left mouse button and entering the value.

Edit minimum or maximum value for x and y coordinate is presented on followed example of Response scope graph:

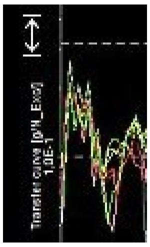




After editing we can see Response graph like bottom:

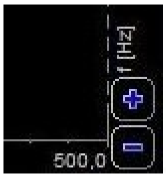


2. **Coordinates scale of FRF graph**



For FRF graph we also have the **Auto scale** y axis. If we move with the mouse left to y coordinate of FRF graph the pointer change to and we can by:

- *left* click → **Auto scale** y axis
- *right* click → **Full scale** y axis.



Scale x axis can be done by mouse click on:

+ icon *enlarge* / zoom in

- icon *contract* / *zoom out* x axis.

5.2 Measurement

When we set the *Property panel* (in fact in *FRF setup*), *Graph display* and *Transfer curve selector* we can start the measurement:

- **measure** the FRFs *directly* while acquiring data and the
- to **Calculate** *offline* from stored data - *see* → Offline calculation.

Directly measure the FRFs while acquiring data

There are *two* methods to generate input signal (*excitation*) and also measurement:

impulse measurement → input signal (excitation) is generated with *modal hammer*,
continuous measurement → excitation with *function generator* or *shaker.*

1. Impulse measurement

Typical impulse measurement is Single input single output modal analysis – SISO (for details *see* → SISO - single I/O modal analysis). There we define **two** measurement channel – one is *excitation*, another one is *response* and help to get faster results.

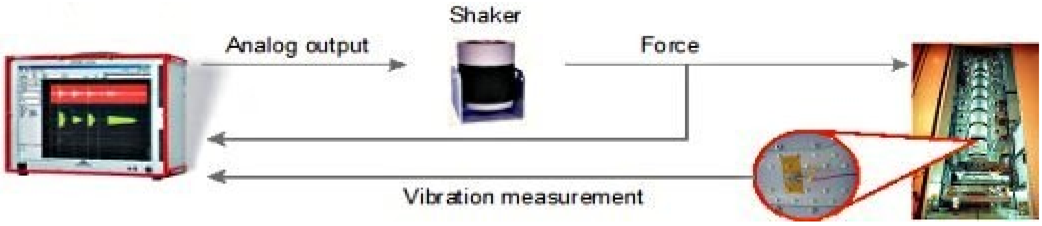
When confirming the measurement either excitation (hammer) or response (sensor) has to be moved to the *next* point. However it is **advisable** to **always** move the *excitation* point through the structure especially if it is a light structure – otherwise the natural frequencies could change due to the mass loading effect from the sensor.

If the excitation point is moved, it is important to measure the ***mobility point*** (the same point of excitation and response) that the animation can be shown correctly.

2. Continuous measurement

A single or multiple shakers are driven by either analog output of DeweFRF (*see* → **Channel setup**) or by *external* shaker system. The force is induced in the structure and the *response* of the structure is measured back by the *accelerometers* or another *vibration transducers*.

If the shaker is used for excitation of structures, the system has the following configuration:



Then the measurement can be started by pressing **Measure**. If the time data is loaded, the menu item button is called Calculate (if the data is stored, parameters can be adjusted and FRFs can be *recalculated* by pressing again Calculate).

Transfer curve selector will show chosen excitation and response channels in gray color.

When the data is correctly calculated, FRFs can be taken by pressing Confirm button.

If more channels are to be measured, go back to the **FRF setup**, change the *point indexes*, do another *measurement* and **Confirm** button will add it to all measurement *result*.

5.3 Measure tools

The *Measure tool bar* in upper part of screen hold some action button for basic operation:

Measure tools

- **Reset (F5)** Delete all the measurement and will reset number of averages to zero.
- **Reject last (F6)** Rejecting last measurement - to delete only last measurement from the result. In case of bad hit with low coherence it's *not* necessary to start from beginning, but we can reject the last average and **keep all** previous values.
- **Stop** The measurement is stopped, it can be *restarted* by pressing Measure button (or Calculate if we are calculating from already stored data).
- **Confirm (F9)** When the data is *correctly* calculated, the currently measured FRFs are stored in calculation results.

Extra SISO buttons

There are two more buttons which appear only if **just one** excitation and response channels are defined and help to get faster results. This is typical by Single input single output modal analysis – SISO, for details *see* SISO - single I/O modal analysis.

- **Next exc. (F10)** Confirms the measurement and moves the point index that *excitation* point is incremented by one. This allows us to immediately measure *next* point on the structure.
- **Next resp. (F10)** Does the same as above "Next exc.", but it will move to *next response* point. However it is **advisable** to **always** move the *excitation* point through the structure.

In Calculate procedure is this tool bar little different - *see* → Offline calculation.

6 Analyse

In **FRF Analyse** we can *analyse* frequency measured response data:

Analyse procedure is relevant to **analyse** measured data which are acquiring **directly** with continuous or impulse excitation or for **offline calculation** → *see* Measure methods.

FRF graph: displays measured FRFs with different Display options

List of measured FRFs: the list of measured FRFs is the selector for the display these on the graph

Geometry of mechanical structure with nodes, tracelines as well as local coordinate system

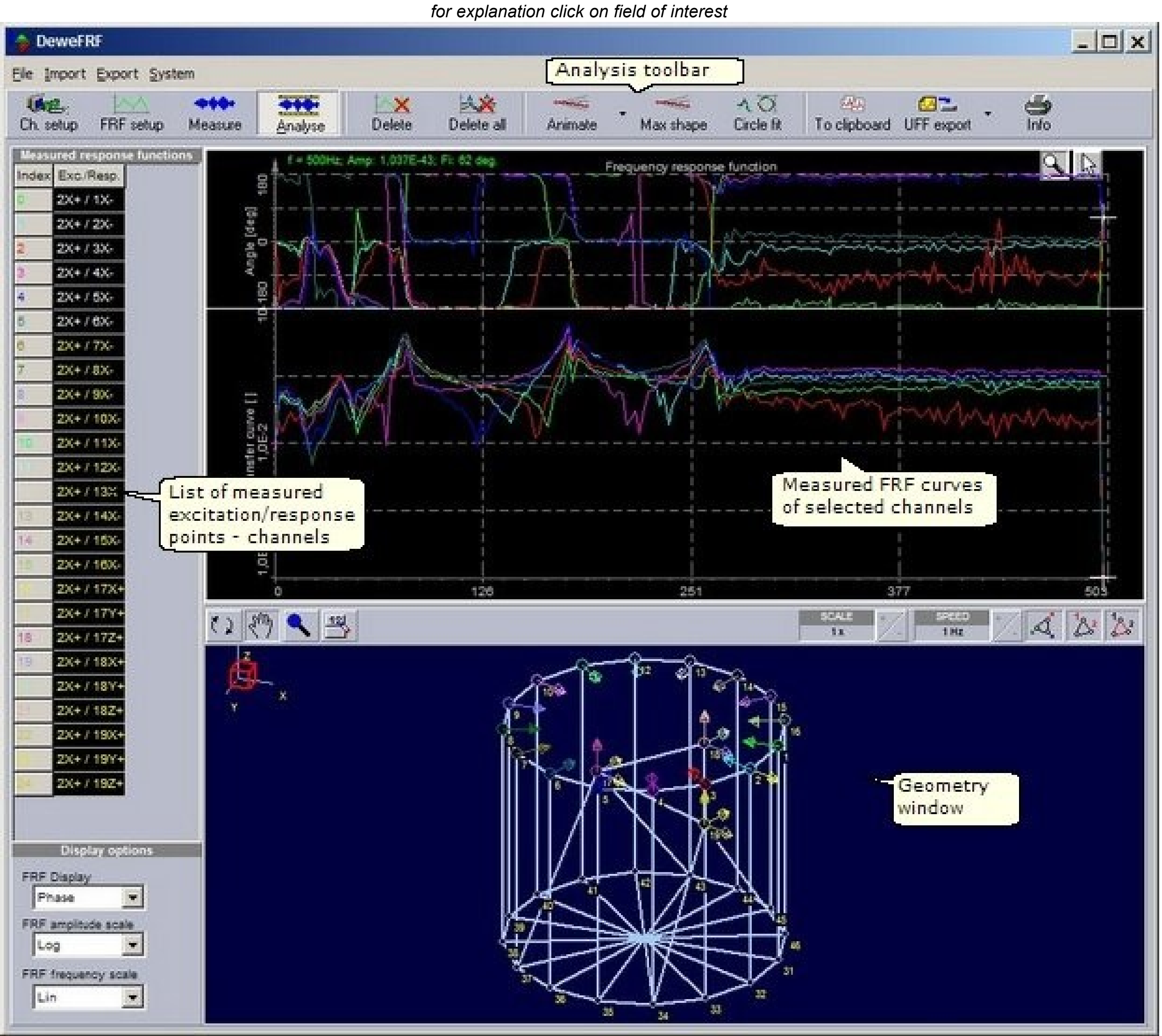
Analysis tools: for observing the deflection of shapes with animating the points on the structure or to show the maximum deflection shapes (**Animation**, **Max shape** or **Circle fit**), tools for **export FRFs** and **information** about the settings of the project used to take FRFs

About Import/Export FRF setup data as Project data file *see:* User Guide → **Data management**

About **Analyse** for **offline calculation** *see:* User Guide → **Offline calculation**

When FRFs are stored, the **Analyse** window can be started by click on  FRF tools button or starts automatically when **Stop** button is pressed in **Measure** tool:

for explanation click on field of interest



Analysis screen consists of two major parts:

- The **list** of measured FRFs. Here you see all the measurement data on the left.
- By *selecting* the items in the *list*, on the right top will be shown: the **display** of FRF result graph in *FRF graph* and geometry in *Geometry window*.

If more channels are to be measured, go *back* to the FRF setup, *change* the point indexes, *do* another measurement and confirm will *add* it to all measurement result.

The entire **setup** (not the data files) can be stored with File → Save project... as a *project file* to reuse it later.

6.1 List of measured FRFs

Graph with measured FRFs displays only FRFs which are selected in list of Measured response functions.

Displayed FRFs

The list of measured FRFs is the selector for the display – shown function are designated with *white ink* color in Exc./Resp column of the list.

Individual FRF can be *selected* by:

- dragging with mouse from the FRF in Exc./Resp list to graph
- selecting items with left-click

More FRFs can be shown by selecting items with Shift-Left click or dragging on the list with Left click.

Delete FRF from list

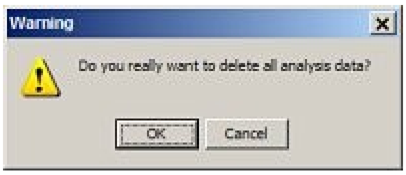
By this action FRF is removed from graph and from list of Measured response functions.

Individual FRF can be *deleted* by clicking the 

icon of Analyse tool bar.

All FRFs are deleted by clicking the 

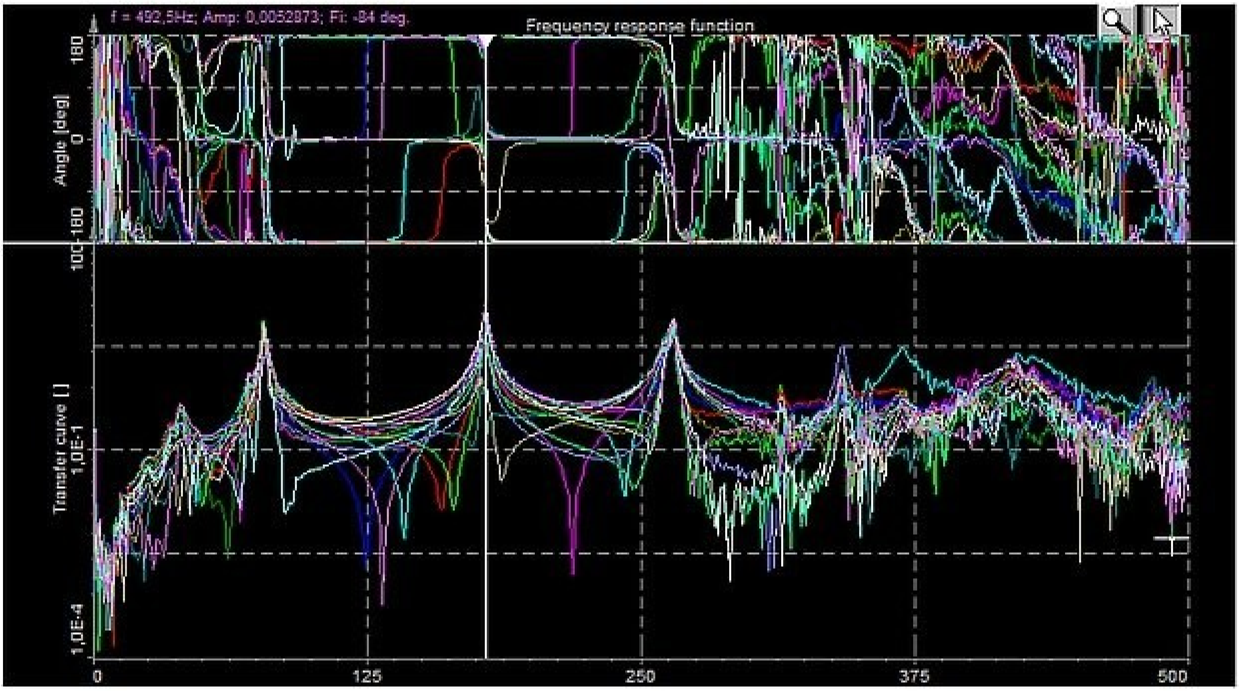
icon. After pressing this button we get the following Warning screen:

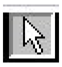


to delete all analysis data. By pressing Yes we delete all analysis data, by pressing Cancel FRFs are not deleted.

6.2 FRF graph

Central part of Analyse screen displays **graph** with measured FRFs which are *selected* in **List of Measured FRFs**.



By selecting  (default) the mouse moved over the graph will show cursor *values* for nearest point in channel color.

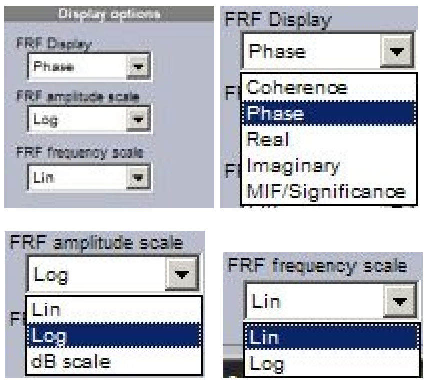
To do any **analysis** (Animation, Max shape or Circle fit) the *mode* has to be chosen by *clicking* on the FRF graph. A *white* vertical line will appear indicating the chosen frequency.

We can set display properties of graph in two ways:

Display properties

Display option

In left bottom part of Analyse screen we have several choices for FRFs Display:



- on the 'FRF display' we can show *coherence*, *phase*, *real*, *imaginary* or MIF/Significance display.

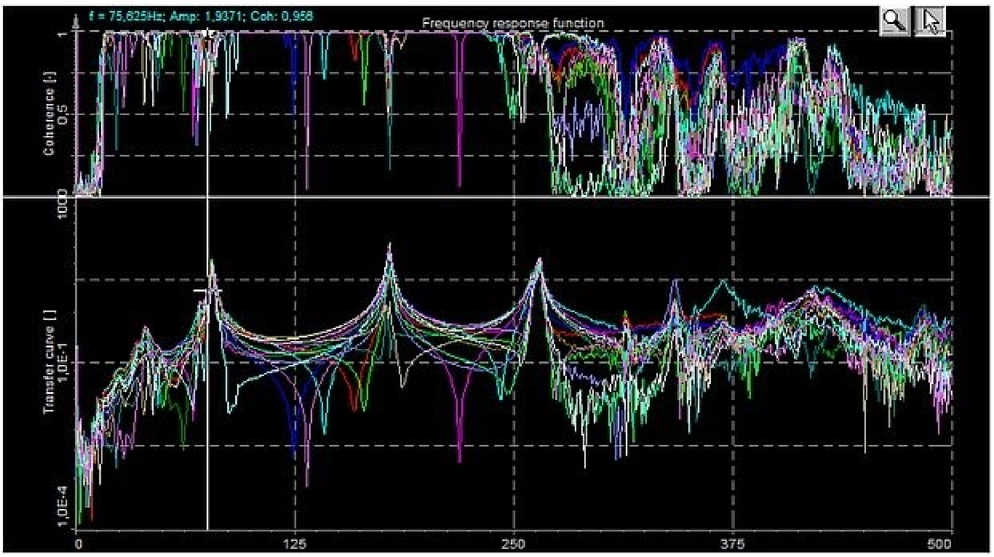
  The **coherence** is the ratio between vector and scalar sum of single FRF shots and shows *how well* the FRF shots are <u>correlated</u> to each other. The coherence of 1 shows *perfect* correlation, coherence of 0 shows *no* correlation.

  When we select coherence and if we click on coherence curve the value of coherence in this point is displayed (*see* graph above).

  The coherence is used when *multiple* data is acquired for single FRF.

- The other two selections are *linear* or *logarithmic* amplitude and frequency plots.

Typical coherence FRFs display:



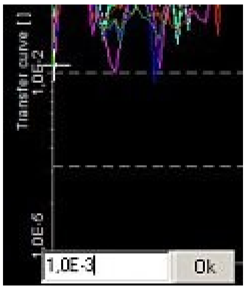
Coordinates Scaling

Transfer curve of FRF graph has x and y *coordinates scaling* possibilities.

1. Edit minimum or maximum value for x and y coordinate of Transfer curve

If we move with the mouse over the minimum or maximum value for x and y the cursor change  , we can enter custom value by clicking with left mouse button and entering the value.

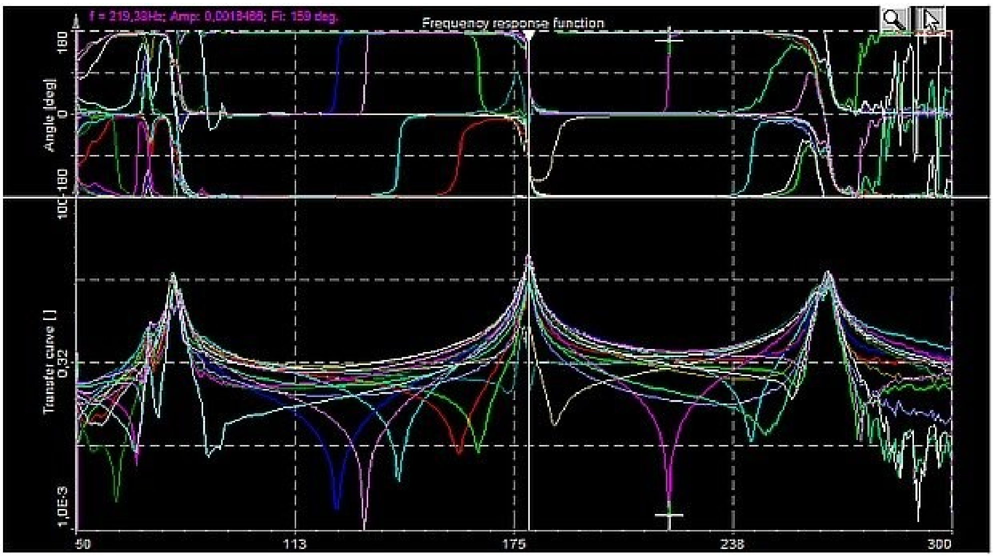
for $\mathbf{y}_{\min}$:



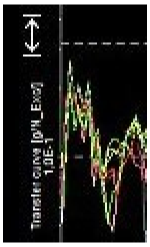
for $\mathbf{x}_{\min}$:

for $\mathbf{x}_{\max}$:

After editing we can see FRF graph like below:



2. Coordinates scale of FRF graph



For FRF graph we have also the **Auto scale y** axis. If we move with the mouse left to y coordinate of FRF graph the pointer change to ↕ and we can by:

- *left* click → **Auto scale** y axis
- *right* click → **Full scale** y axis.

With selected 🔍, cursor changes in yellow vertical line and we can with *left* mouse click and then *drag* on desired region therefore expanding this region on whole scale of **x** coordinates.

With *right* mouse click we return the default size of **x** coordinates.

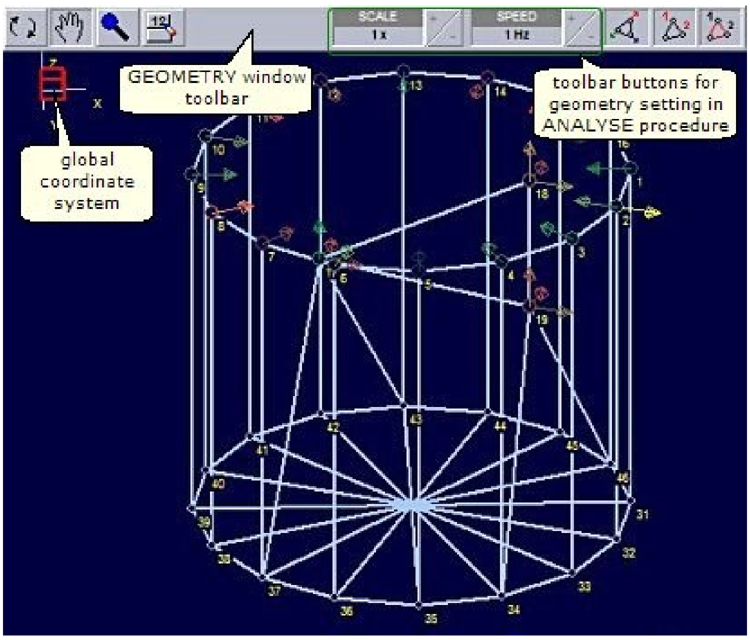
6.3 Geometry window

On Geometry window we can see geometry *nodes* (points) of mechanical structure to which a channel for each **excitation** and **response** is defined, as well as *trace lines, global coordinate system* and when entering , *arrow* with *indexes*, *directions* and *signs* will be shown.

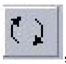
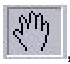
It is very helpful to load the geometry to get a feeling *where* to locate **excitation** and **response**. About Import/Export Geometry data *see*: User Guide → Data management → **Geometry**.

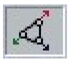
If the geometry has been created or loaded, the deflection shapes can be observed with animating the *points* on the structure or showing the maximum deflection shapes - *see* **Analysis tools** chapter.

for explanation click on field of interest



Geometry window **TOOLBAR** is divided in three parts:

1. MANIPULATE with geometry , , ,  are the same as in FRF setup - *see* → **Manipulate geometry**

2. Geometry DISPLAY SETTING , ,  are the same as in FRF setup - *see* → **Display setting**

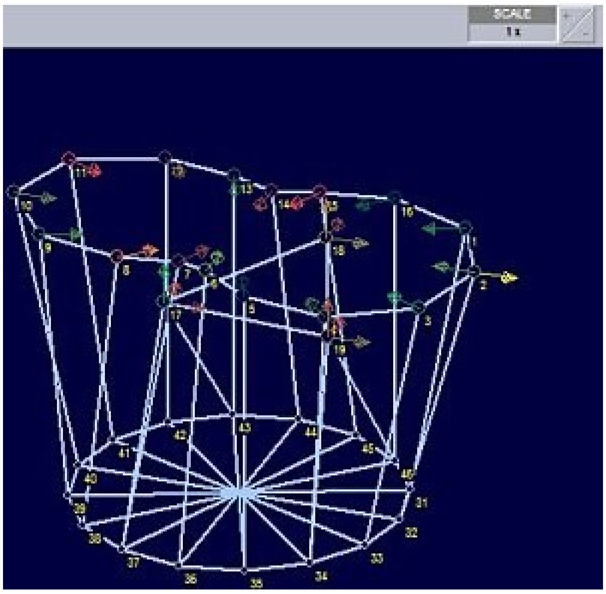
3. *Particularly* for Analyse procedure are on Geometry window toolbar *two* button:

- **SCALE**
- **SPEED**

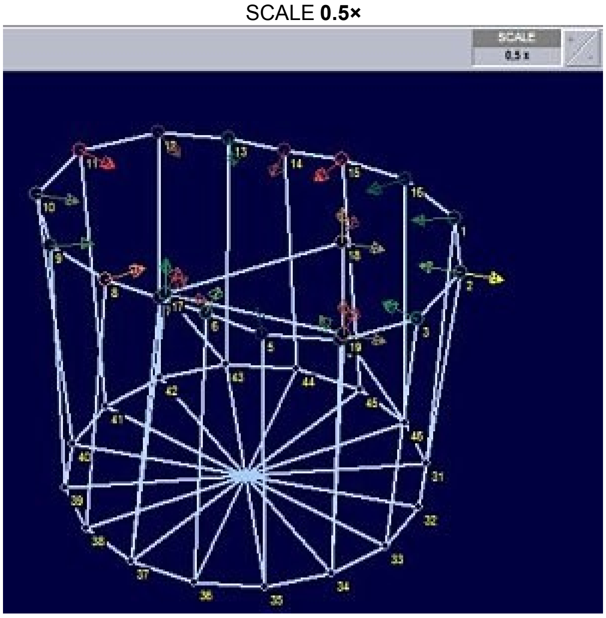
SCALE

When triangle button **+** is pressed, shapes deflections size increase for **0,1×**; When triangle button **-** is pressed, shapes deflections size decrease for **0,1×**.

SCALE **1×**



SCALE **0.5×**



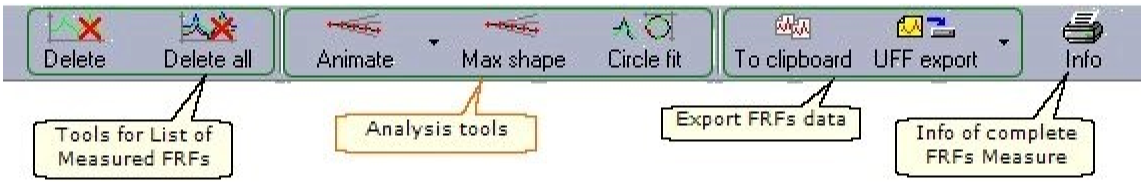
SPEED

When triangle button **+** is pressed, shapes deflection *animations speed* increase by step of **1Hz**; when triangle button **-** is pressed, shapes deflection *animations speed* decrease by step of **1Hz**.

6.4 Analysis tools

If the geometry has been created or loaded, the deflection shapes can be observed with animating the *points* on the structure or showing the maximum deflection shapes. This actions will be taken by buttons on main Analysis tool bar:

for explanation click on field of interest



Analyses tool bar is divided in three parts:

1. On left are tools for *List of Measured FRFs* - *see* → **Delete FRF from list**

2. Right most is Info tool for *complete information* of FRFs - *see* → **Info**

3. On right side are tools for manipulating FRF data - *see* → **Export FRF data**

4. On central part of toolbar are *real* Analyse tools:

- **Animate**
- **Max shape**
- **Circle fit**

Note: When a **SISO (single input, single output)** model is used we have on toolbar *additional* button:

In single FRFs which have *inadequate* results can be **measured** *again* by pressing  button,

also *see* → DeweFRF Tutorial → **SISO - single I/O modal analysis**

6.4.1 Animate

If the **geometry** has been created or loaded, the deflection shapes can be observed with animating the *points* on the structure.

By choosing **Animate** button currently *selected* operation deflection shape will be animated.

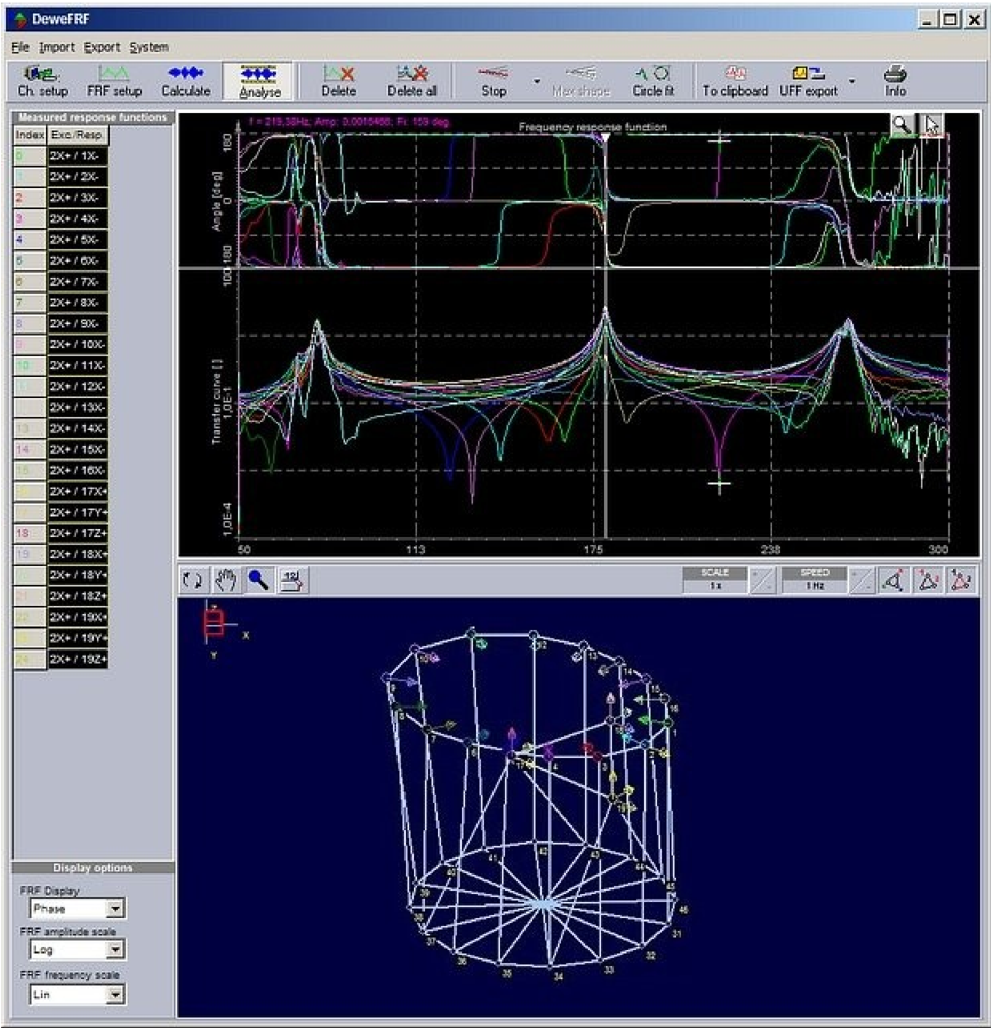
The **speed** and **amplitude** can be *changed* with the buttons on the Geometry display - *see*: Geometry window → **SCALE** and **SPEED**.

The drop down menu gives the choices to animate:

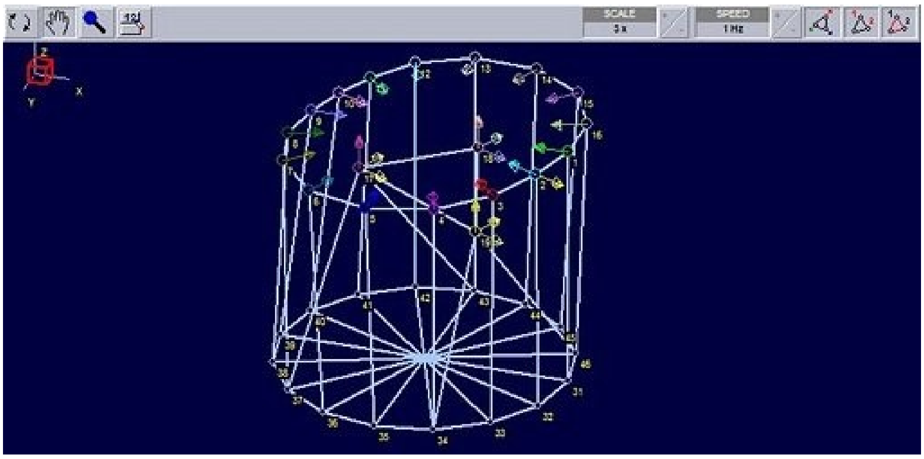
- complex,
- real or
- imaginary

parts of the currently selected mode. *Usually* complex animation is used, but often it is *useful* to animate only *imaginary* part of the results, because with well separated modes the real part of the results is only the measurement error.

After selecting Animate button instead of this we get **Stop** button to stop *animation*, however Max shape button is *disabled*.



In the above Geometry windows we can see a screen shot of bending animation of shapes in *left* most position, on next screen shot however in *right* most position.

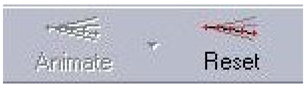


If by animation a *static* picture of *maximum deflection* shape is requested, **Max shape** option can be chosen. It is like *freezing* the animation at the time where the structure would be maximally *bended*.

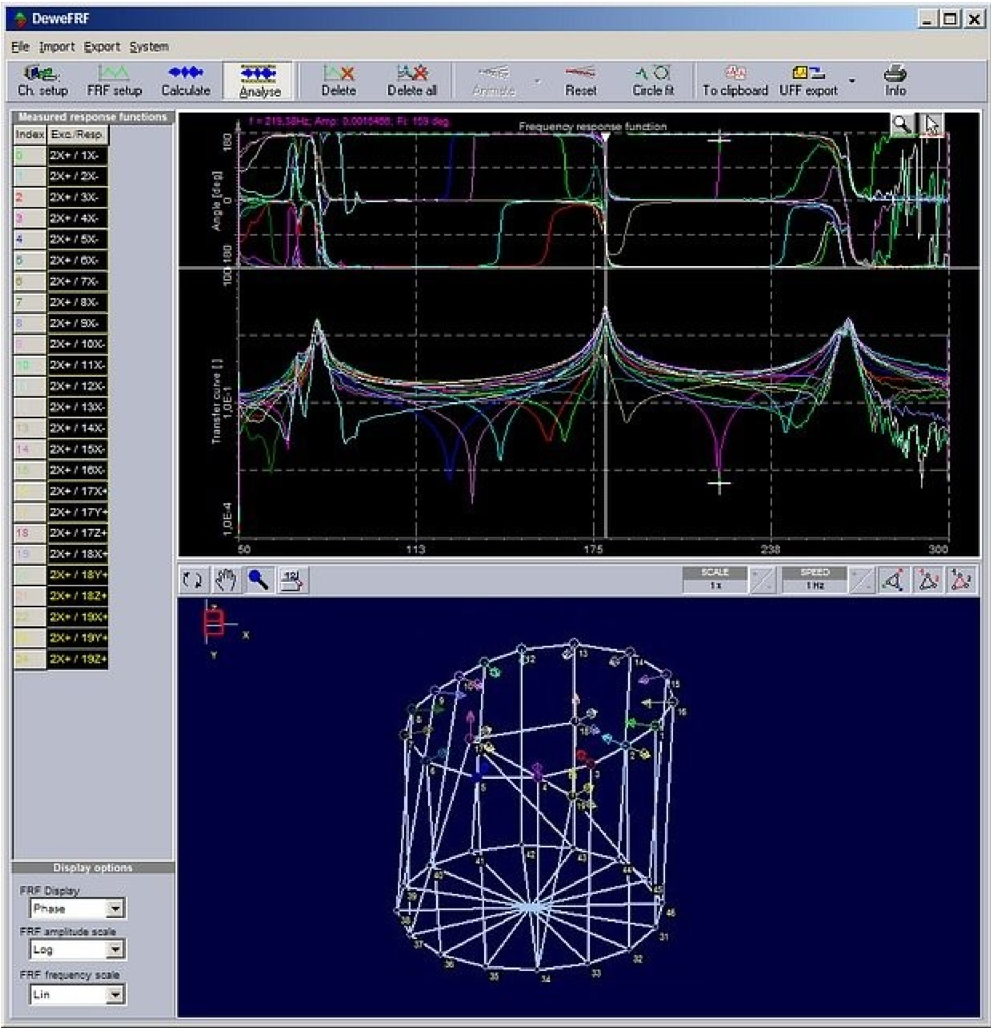
6.4.2 Max shape

If the **geometry** has been created or loaded, the *deflection shapes* can be observed with showing the maximum deflection of shapes or if by *animation* a *static* picture of *maximum deflection* shape is requested, **Max shape** option can be chosen. It is like *freezing* the animation at the time where the structure would be maximally *bended*.

With selecting this button we can see maximum shapes deformation.



After selecting Max shape button instead of this we get **Reset** button to return of geometry in initial state (without deflection), however Animate button is disabled.

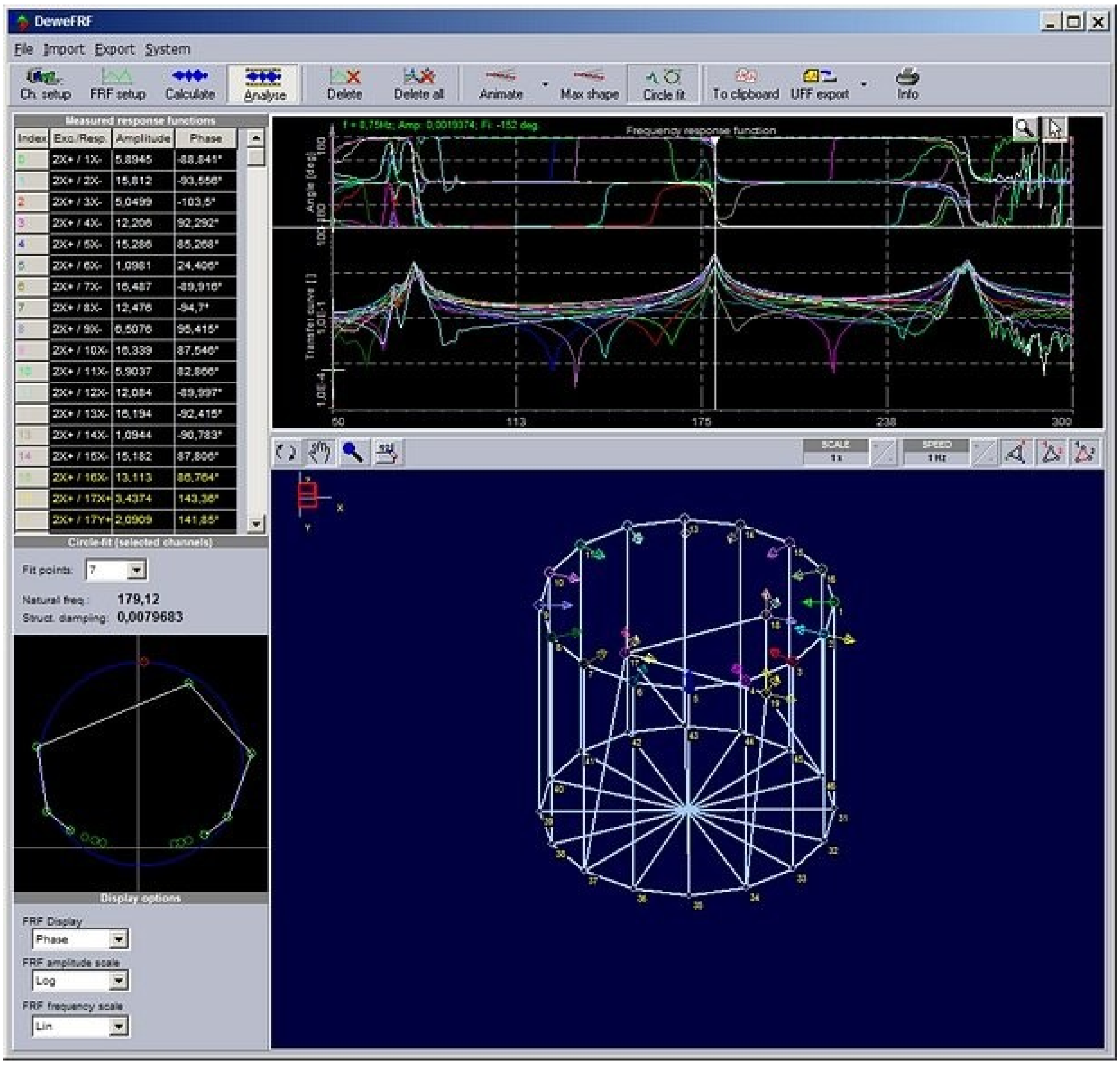


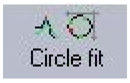
6.4.3 Circle fit

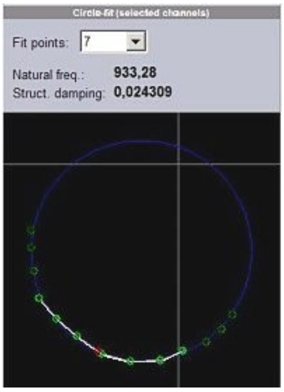
For a *quick* and *simple validation* of results, circle fit is provided for **calculation**:

- of exact *frequency* of mode and
- *damping* of the structure.

It has to be noted that circle fit is single degree of freedom fit, so it calculates the results only for a *single* mode and gives good results only if *modes* are well separated.







When circle fit is selected, the FRF list is expanded with **Circle fit (selected channels)** panel to show:

- *maximum amplitude* and
- *phase*

of the certain **FRF point**. The amplitude and phase is calculated from the mode circle as the *highest value*.

Below the *number* of Fit points can be chosen. **Number** of fit point tells the *width* of the calculation – we have to be careful not to choose too *high* number, otherwise neighbor modes might *affect* the results. This number has to be a maximum value that the points **still lay** on the circle.

In order to help to define the number of points, the circle graphics shows the points which are **selected** for calculation in *light green* and next three points which are **not** taken are in *dark* green. If it seen that dark green points would **still** be *on* the circle, it is advisable to choose **more** points for calculation.

The Natural freq. label shows the *calculated* **natural frequency**, which is calculated from the *mode circle*.

The Struct. damping calculates the **damping** from the *angles of points* in the *mode circle*.

Mode circle shows the points for calculation, the **circle** is the *result* of fit and the *red* point shows the *amplitude* and *phase* of estimated **natural frequency**.

6.4.4 Export FRFs

On *Analyse* toolbar we have *two* tools to export FRFs.

Copy 'to clipboard'

- copy 'To clipboard' button **copies** the *currently selected* and *active* FRF to the clipboard.
It is in tab delimited format suitable to be imported in *spreadsheet* applications. The *first* column is the **frequency** in Hz, the *second* column is the **real** part and the *third* column is the **imaginary** part.

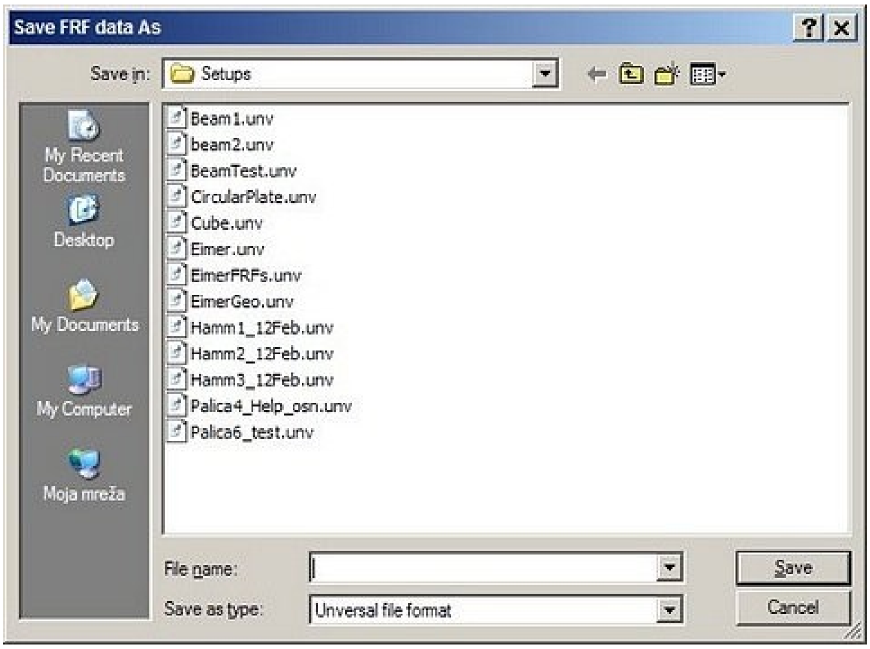
UFF export

The 'UFF export' *exports* the data in **universal file format type 58**.

We have several options to export:

- *FRF data*
- *coherence*
- *excitation FFTs* (exports only the FFTs of the *last run*) or
- *all* the results at once.

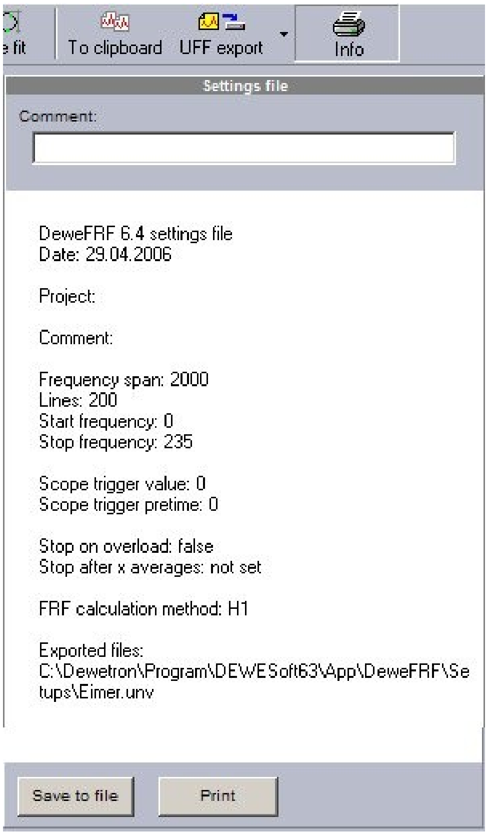
After selecting *UFF export* tool **Save FRF data As** *window* is displayed to save *new* or *modified* FRF Data - measured data to *.*unv* (*Universal file format*) data file - also *see*: Data management → **Modal FRF data**.



6.4.5 Info

The 

- 'Info' button shows the information about the **settings** of the *project* used to take FRFs. The **comment** can be *added* to the project and this project info can be *saved* to file or *printed* out.



7 Offline calculation

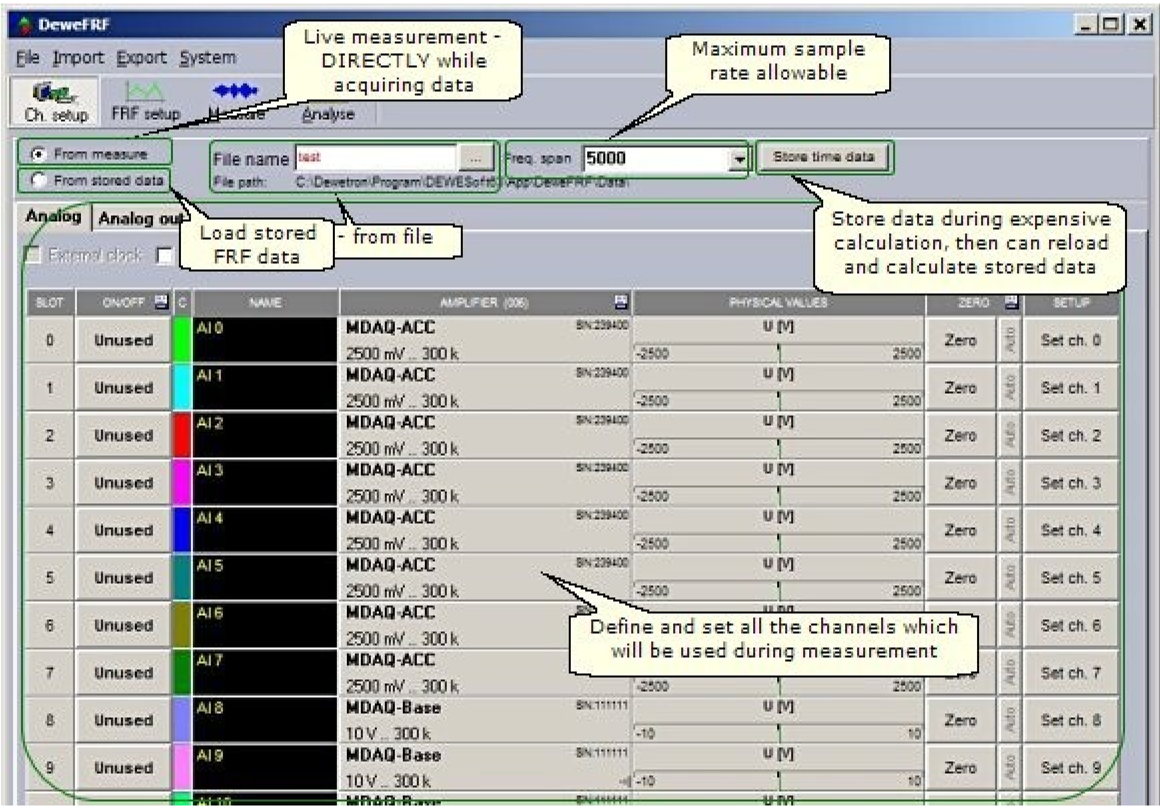
During extensive calculations we can **Store time data** in file. Later we can *reload*, *calculate* and *analyse* these **From stored data**. The way to calculate is exactly the same, except that the data is not acquired, but *loaded* from file.

Offline calculation and analyse *include* following steps - procedures:

- **Store time data** - *store* segment of measured data in Channel setup procedure of *directly* measurement
- **From stored data** - for *reload* stored data in Channel setup procedure of *directly* measurement
- **Channel setup** *From stored data* - define *range* of stored and reloaded data - *Show* and *Start show Time* for recalculation
- **FRF setup** for stored *Time data* - define DeweFRF *software* (same as for *directly* measurement, except Frequency span)
- **Calculate** stored *Time data* - *recalculating* in Channel setup defined range stored and reloaded data
- **Analyse** calculated *Time data* - *analyzing* recalculated data (same as for *directly* measurement

On Ch. setup procedures window of *directly* measurement:

for explanation click on field of interest



we can choose:

- **Store time data** for store segment of measure data
- **From stored data** for reload stored data.

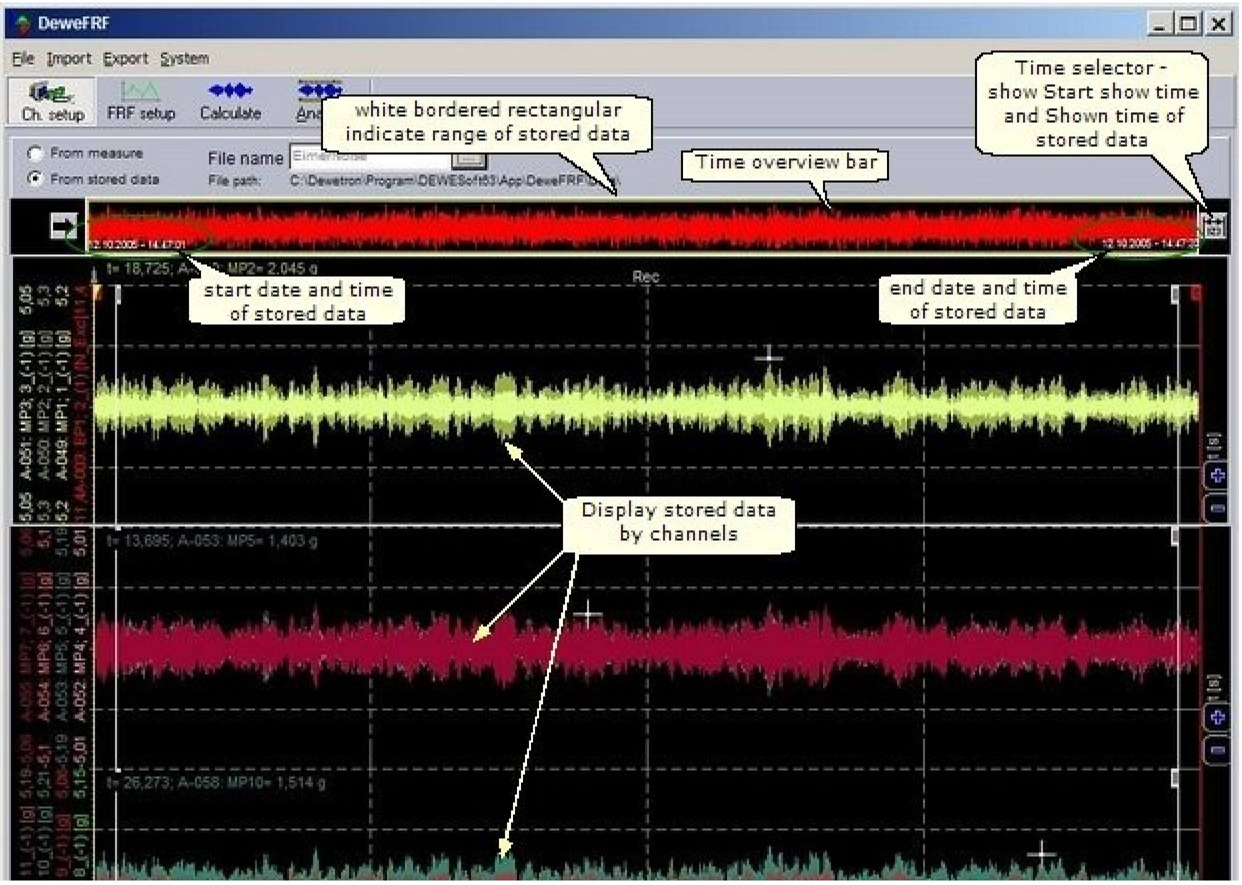
7.1 Channel setup

After selecting From stored data and entering required file in File name we **reload** *stored* data and get **Ch. setup** From stored data window with **Time overview bar** with **Time selector** of stored data on upper part of window and **Display of stored data** with complete stored signals in the recorder by channels; on this display we have also feasibility to **select** certain range of stored data for calculation.

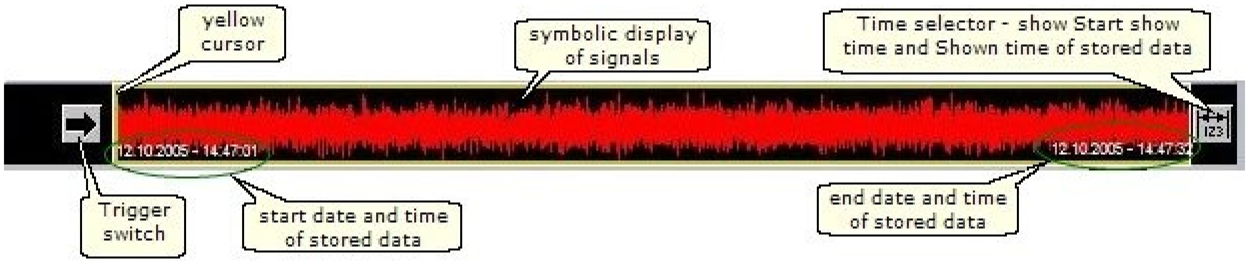
In one of the next steps of DeweFRF - Calculate procedure we can recalculate *whole* range (volume) of stored data or *only* certain (smaller) range of these. The **selection** of stored data for recalculation can be done in Ch. setup From stored data procedure where there are some ways to **select** certain (smaller) range of *whole* volume stored data for recalculation on:

- **Time overview bar** with **Time selector** or
- **Display of stored data**.

The selection of this range is the primary goal of this procedure. If we are to recalculate *whole* range of stored data, no action in Ch. setup after reloading data are needed and we can proceed with **FRF setup** procedure for **Offline Calculation**. The setup of the display as seen below depends on the display setup of Overview screen during acquisition of data (either in Dewesoft or DeweFRF).



7.1.1 Time overview bar



Stored data offers at the top of the screen a small bar - **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,
- **Trigger switch** and
- **Time selector**.

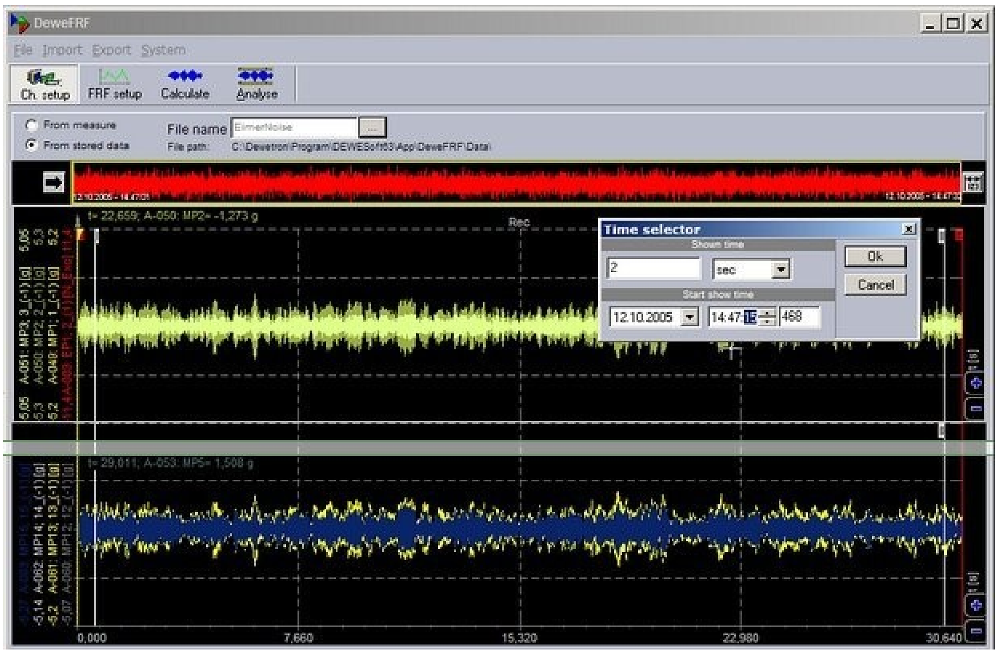
Within the Time overview bar there is also a *cursor* available and as a *standard* it is located at the left side of white bordered rectangular. If you look carefully at this area, you will see a yellow vertical line and this is a cursor.

As you can see on picture bellow time scale on bottom of Display of stored data include is whole range stored data (in our example from **0** sec ÷ **30,64** sec).

Selection range of stored data

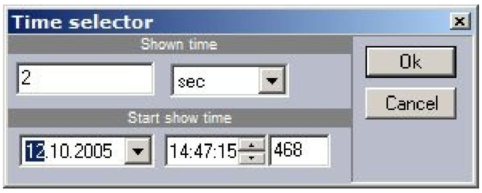
1. Time selector

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 Ch. setup From stored data screen below Time overview bar:

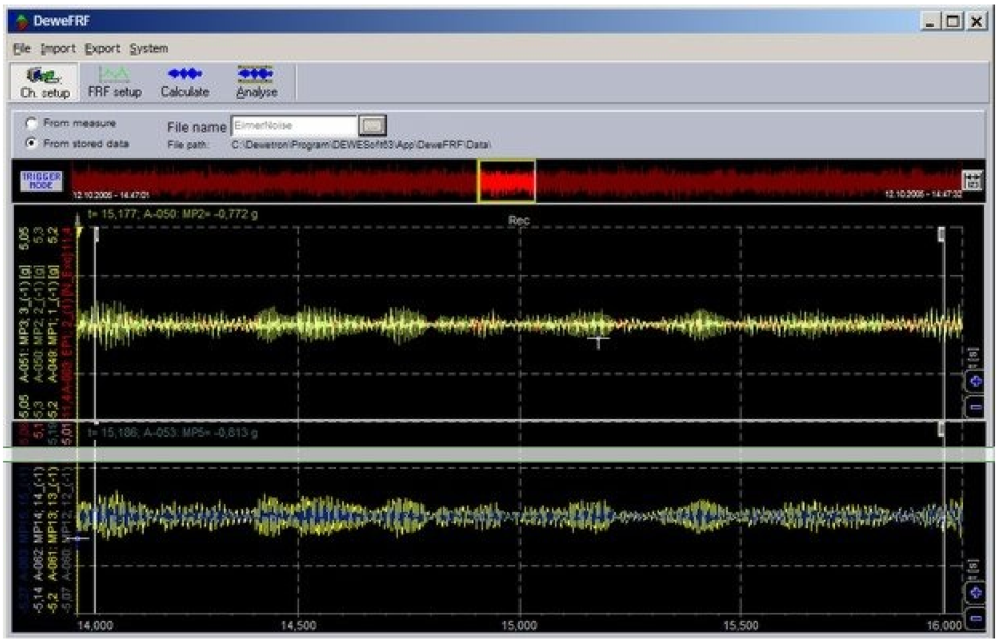


Now you can enter the Shown time and the exact Start show time position within the file. This is first way to narrow range of displayed data (and later recalculating only these).

By defining **Time selector** of measurement (define *Show* and *Start show Time*) we can select only part of the time scale (*smaller* **range** of stored data). In example bellow we set *Show time* on **2** sec and *Start show Time* we increase for **14** sec (from initial start time of stored data):



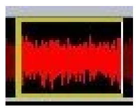
and we get screen like below:



The result depends on the used screen. For DeweFRF offline calculation multiple value element like the recorder, scope, FFT are predefined and that will **display** the signal *only* within the selected time window (= like a *zoom* function).

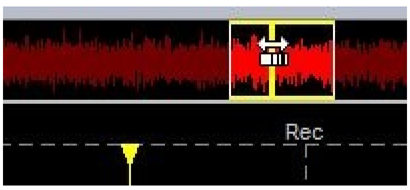
We can see the time scale on the bottom of Display of stored data include only selected range of stored data (in our example from **14,00** sec ÷ **16,00** sec i.e. entered *Shown time* **2,00** sec).

2. 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**. Symbolic display of signals outside of this box is dimmed.

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:

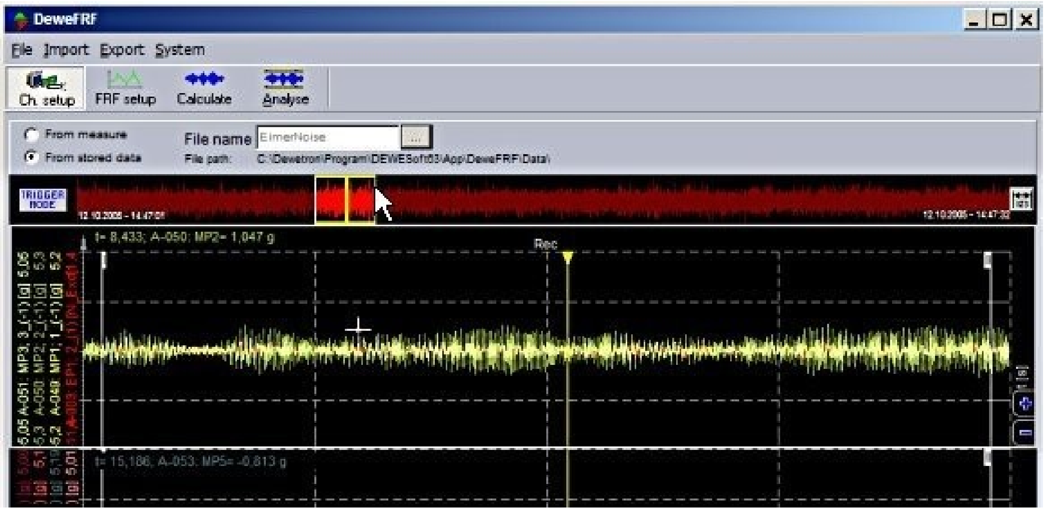


At the same time move *also* yellow cursor relatively proportional in the Display of stored data.

In case of selecting only *range* of stored data on Time overview bar in Trigger switch instead appear . Whit press this switch we *enlarge* selection on initial - *whole* range of stored data and Trigger switch return to .

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:

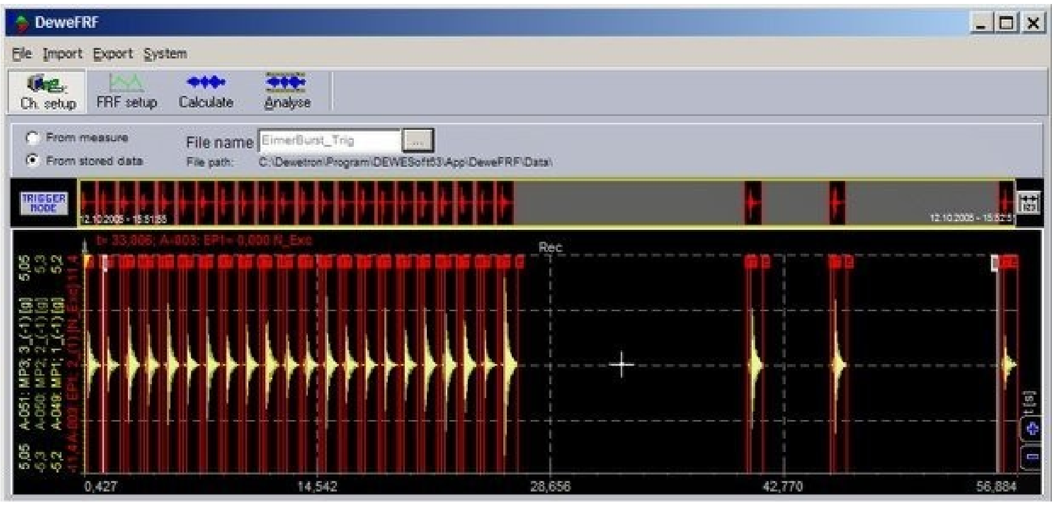
- *enter* a new start time in the Start show time window as described above (by this option we can also change size of range - Show time)
- *click* on the selection window - Time overview box and **move it by mouse**



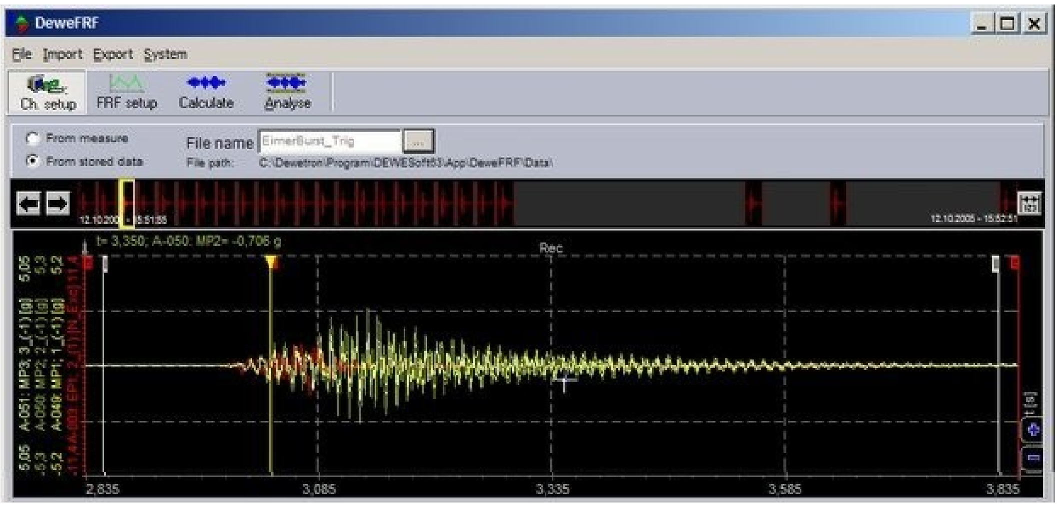
- simply use the **cursor keys** (this will move the selection window each time by the half size of the window itself).

Selecting in reloaded triggered data files

The reload of multiple trigger events within one file shows a different display on-screen.



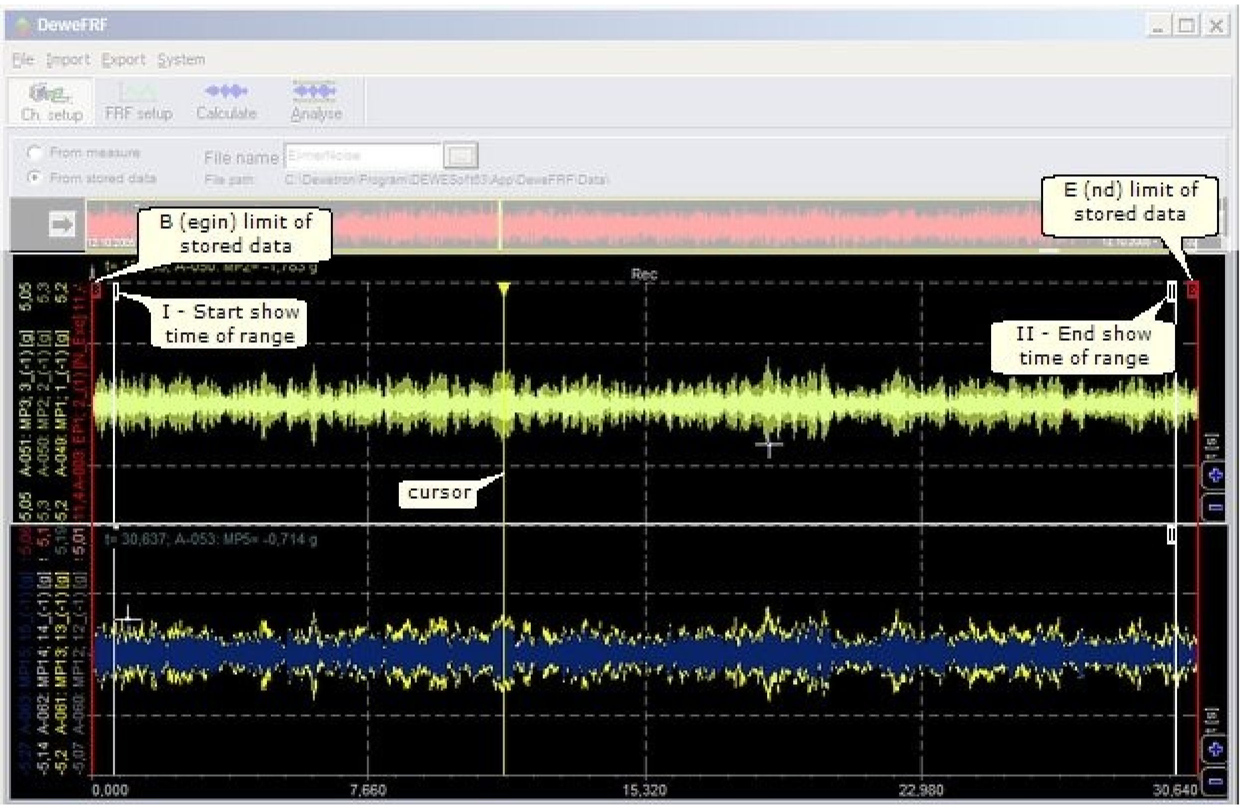
To display each event separately, just press the button and use the arrows to move forward and backward *between* the events - select event.



All other action and procedure is same as are described above.

7.1.2 Display of stored data

After selecting From stored data and entering required file in File name we **reload** *stored* data and get **Ch. setup** From stored data window with **Display of** *whole range* **stored data** like bellow (we have to move yellow cursor from beginning of graph):



On the left part of the display, the readout **values** of the *currently* selected graph 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.

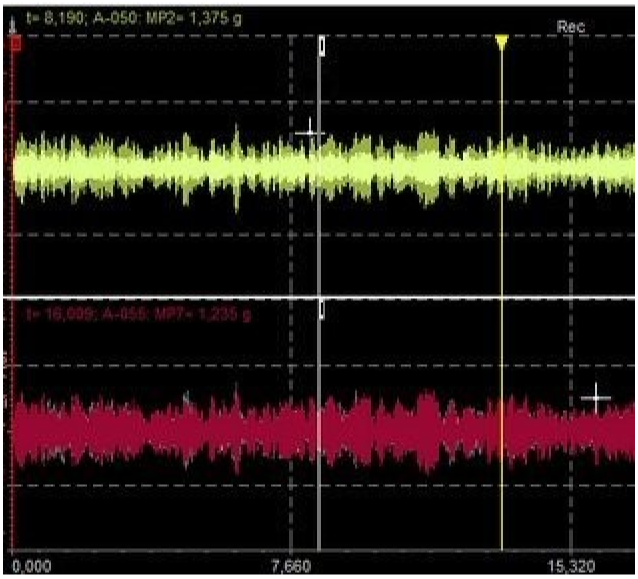
Selection range of stored data

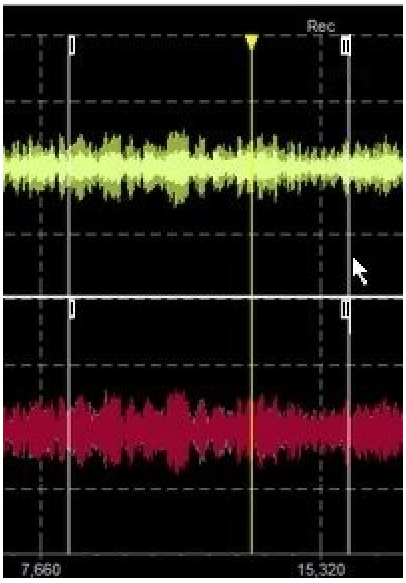
On Display of stored data we have another option to narrow range of displayed data (and later recalculating only these) and select only part of time scale (*smaller* **range** of stored data).

1. Measurement cursor

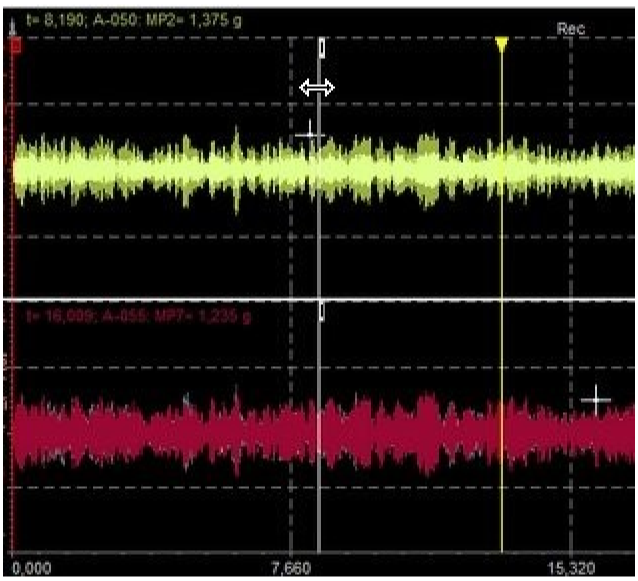
The recorder offer two **measurement cursors** for the active graph on top labeled with **I** for *Start* show time and **II** for *End* show time, which can be positioned in two ways:

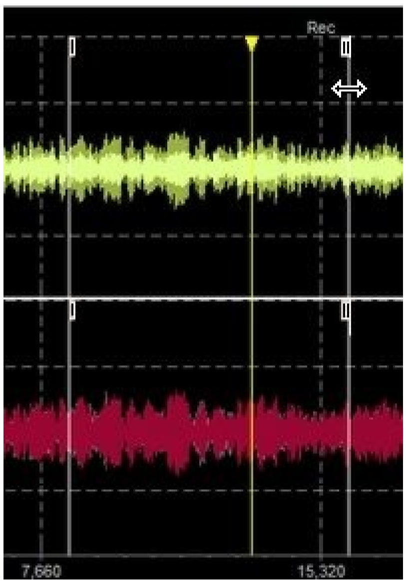
- *Move* mouse cursor to the *first* position of interest. *Click* (measurement cursor **I** appear in this point) and *hold* left mouse button and *move* (on mouse cursor appear measurement cursor **II** and move whit him) to the *second* position of interest.





- *Drag* the first cursor **I** from *left* side of the graph to the position of interest and the second cursor from the *right* 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).



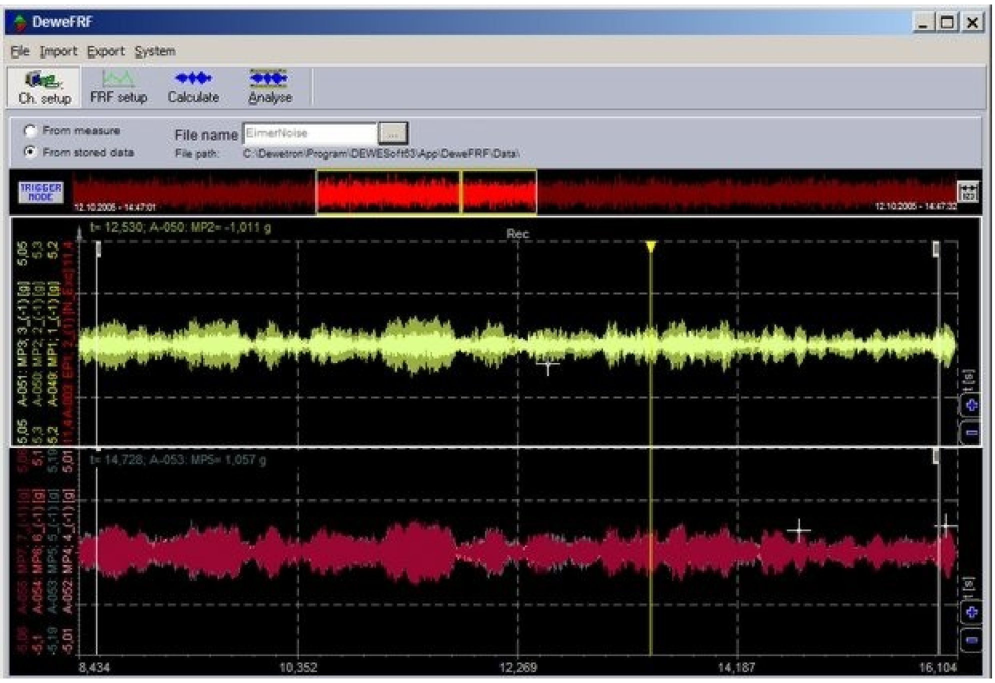


In both case we can zoom into the selected area with:

- Move the mouse *between* these two measurement cursors - it will change the appearance immediately to . Then simply do a left mouse *click* to **zoom into** the selected area. To *undo* the zoom just use the right mouse button and you will **zoom out** step by step.

- 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.

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.

2. Zoom in / out button

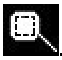
We can select *another* range of stored data within the whole signal also with **zoom in** or **zoom out** button on right side of each graph. As there is *only* one time base available all graph are zoomed in (or out) together, independent which button is selected.

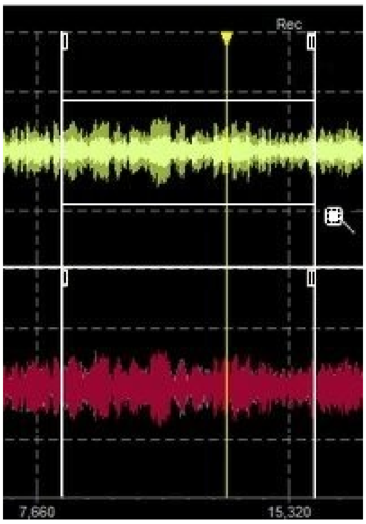
You can do that action *several* times to **zoom into** the selected area step by step (by each step is decrease of range about 1/25 prior range) and is meaningless in what place cursor there is.

By first **zoom out** graph return to previous zoom (range) and after that step by step. If we use before multiple zoom (with **+** *button* or *measurement cursor*), in first step displayed range return to about 3/4 its whole range and after that step by step.

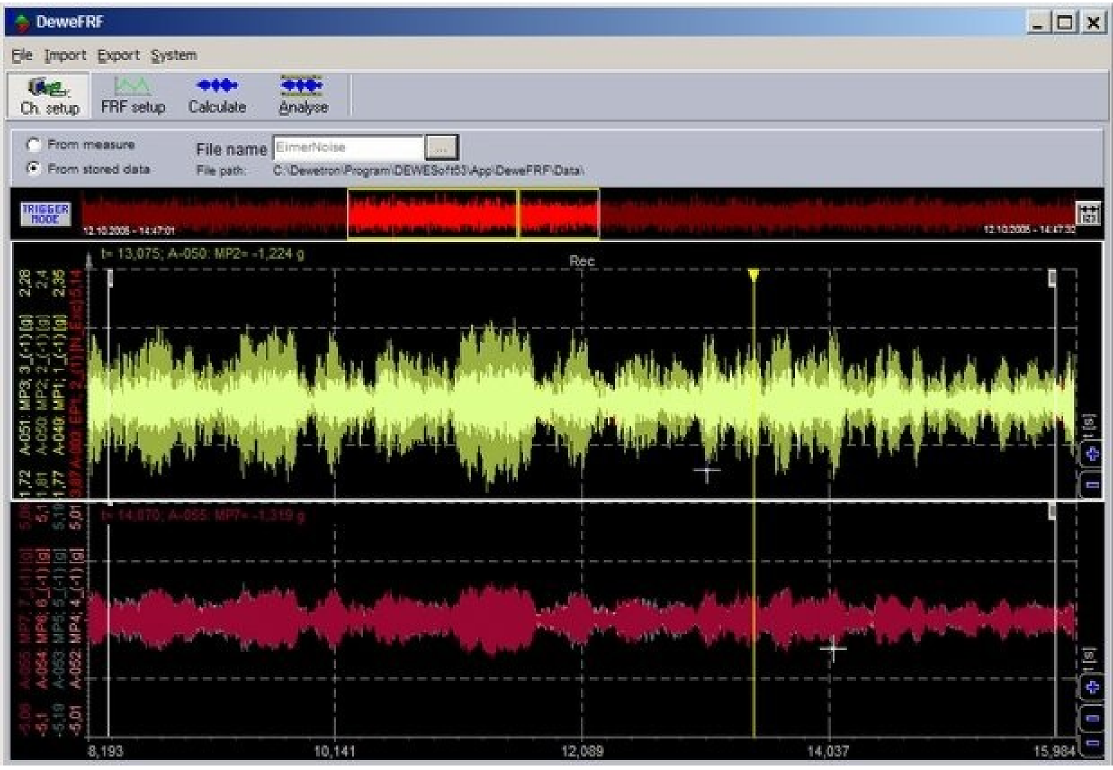
3. Window zoom

Additionally to the described method you can also **zoom in both axis** 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:



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 axis** (but *only* signals in zoom window, signals of other graphs - recorders is zoomed in only at time axis because there is *only* one time base).

To undo the last zoom simply right click once - as already done 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.

7.2 FRF setup

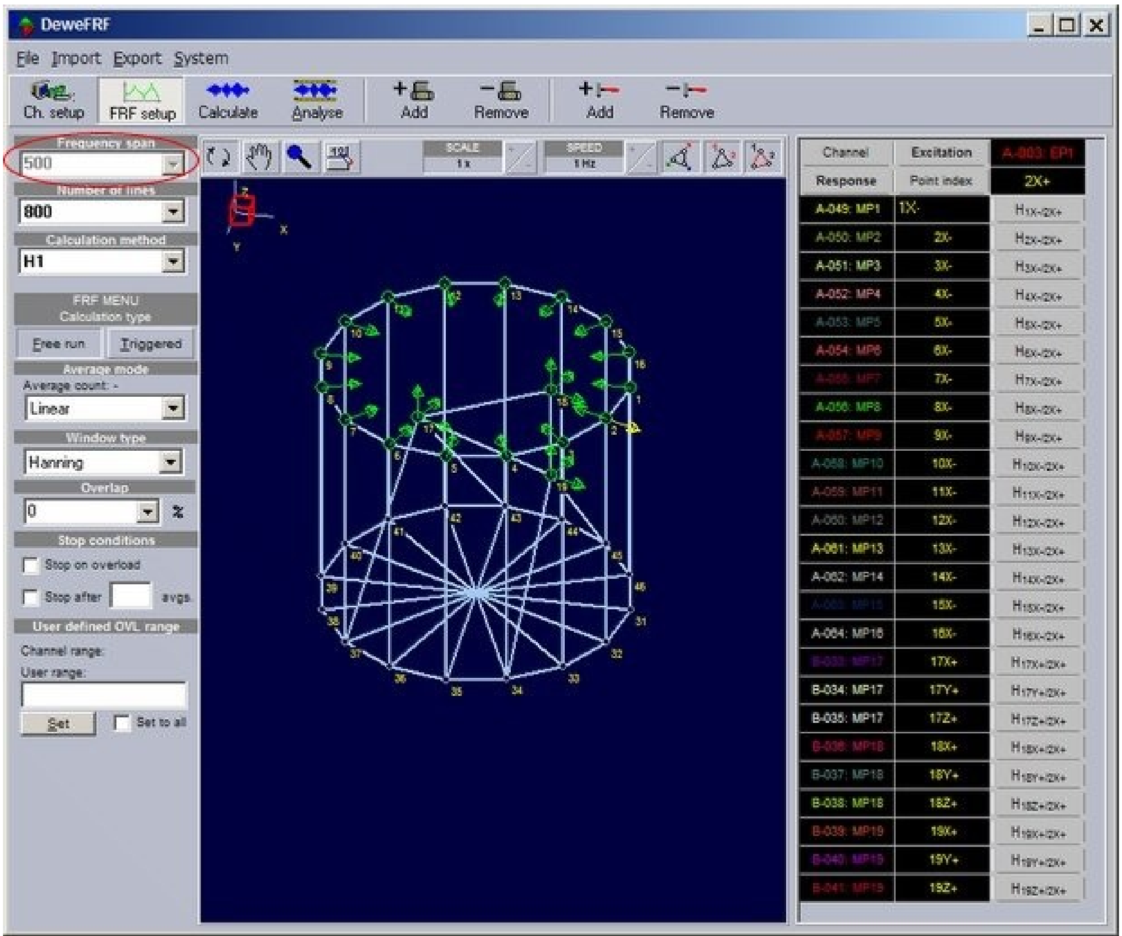
In FRF Point & Geometry setup - FRF setup for **offline calculation** we can define DeweFRF modal analysis software.

The FRF setup can be selected by pressing  tool button.

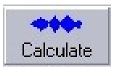
In **FRF setup** we can define:
Geometry of mechanical structure with nodes, tracelines as well as local coordinate systems
Channels assignment excitation and responses channel, adding/removing excitations or responses and changing point index
FRF property: number of lines, calculation method and FRF Calculation type with all of its property
About Import/Export FRF setup data as Project data file see → Data management
When the Channel and FRF setup are <u>done</u>, go to the *next step* → **Calculate**

First it is recommended to *load* or *define* the geometry - *see* → Import/Export Geometry.

FRF setup for **offline calculation** is the same as for **directly** measurement, except *Frequency span* can be <u>not set</u> because it is defined by the sampling rate of time domain data which is loaded - *see* → **FRF Setup**.



7.3 Calculate

The **Calculate** can be started by clicking the 

FRF tool icon.

Calculate is procedure for **offline calculation** - in Ch. setup *selected* range of the stored data will be immediately recalculated with parameters which is selected in FRF setup

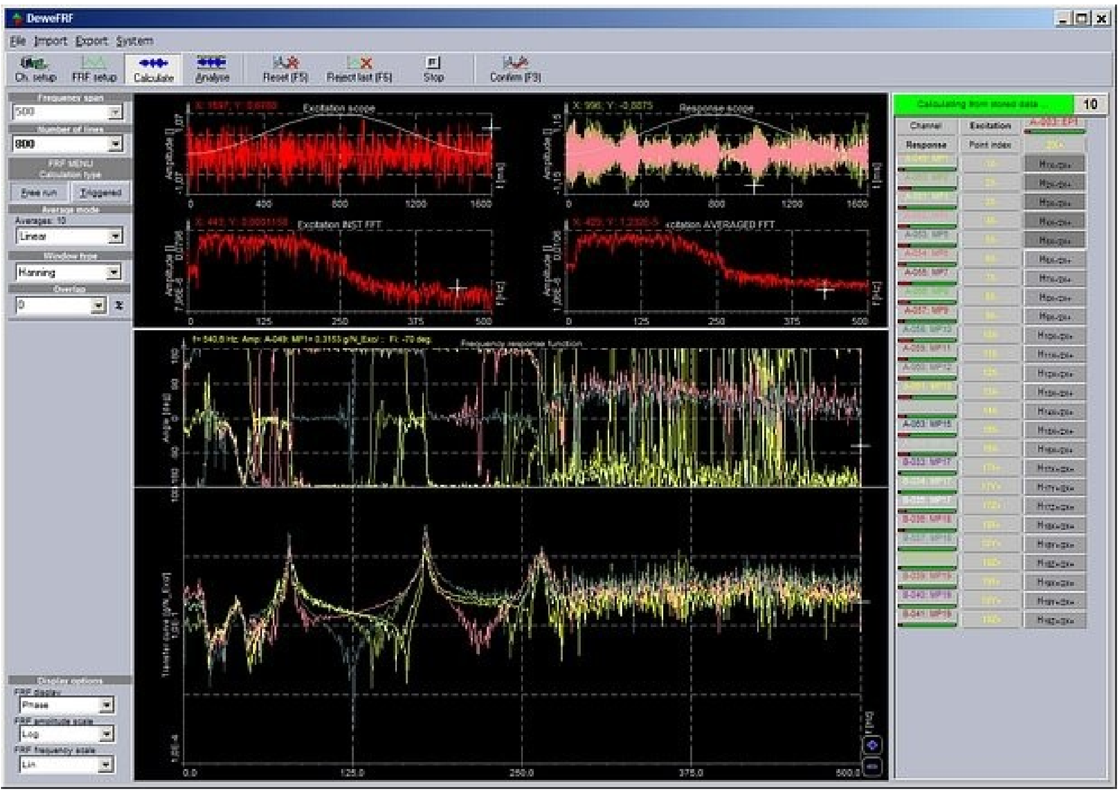
Property panel number of lines, calculation method and FRF Calculation type with all of its properties; same as and in fact part of FRF setup; *see* → **FRF property panel setting**

Graph display set Excitation scope, excitation FFT (current and averaged), the response scope and the frequency response function display; *see* → **Measure - Graph display**

Transfer curve selector transfer function list (selector what is shown on selected graph), information about the amplitudes, number of taken averages, information about last trigger; *see* → **Measure - Transfer curve selector**

Calculate tools help to manage and control calculate

When the Channel, FRF setup and Calculate are **done**, go to the *next step* → **Analyse**



FRF parameters can be furthermore adjusted (except *Frequency span* can be not set) and *FRFs* can be *recalculated* by pressing *again* **Calculate.**

Calculate tools

The *Calculate tool bar* in upper part of screen of the **offline calculation** has only one applied action button for basic operation:

Calculate tools

- **Confirm (F9)** — When the data is *correctly* calculated, the currently measured FRFs are stored in calculation results and we get automatically **Analyse** procedure.
- Reset (F5), Reject last (F6), Stop — Not applied in Calculate.

7.4 Analyse

Analyse is procedure to analyzing measured data which are acquiring from **offline calculation** (*see* → Measure methods). Analyse for **offline calculation** is the same as for **directly** measurement - *see* → **Analyse**. In the Analyse window you see all the *measured data, FRF result graph, analysis tools* and all the *points* which are measured (*geometry*):

FRF graph: displays measured FRFs with different Display options

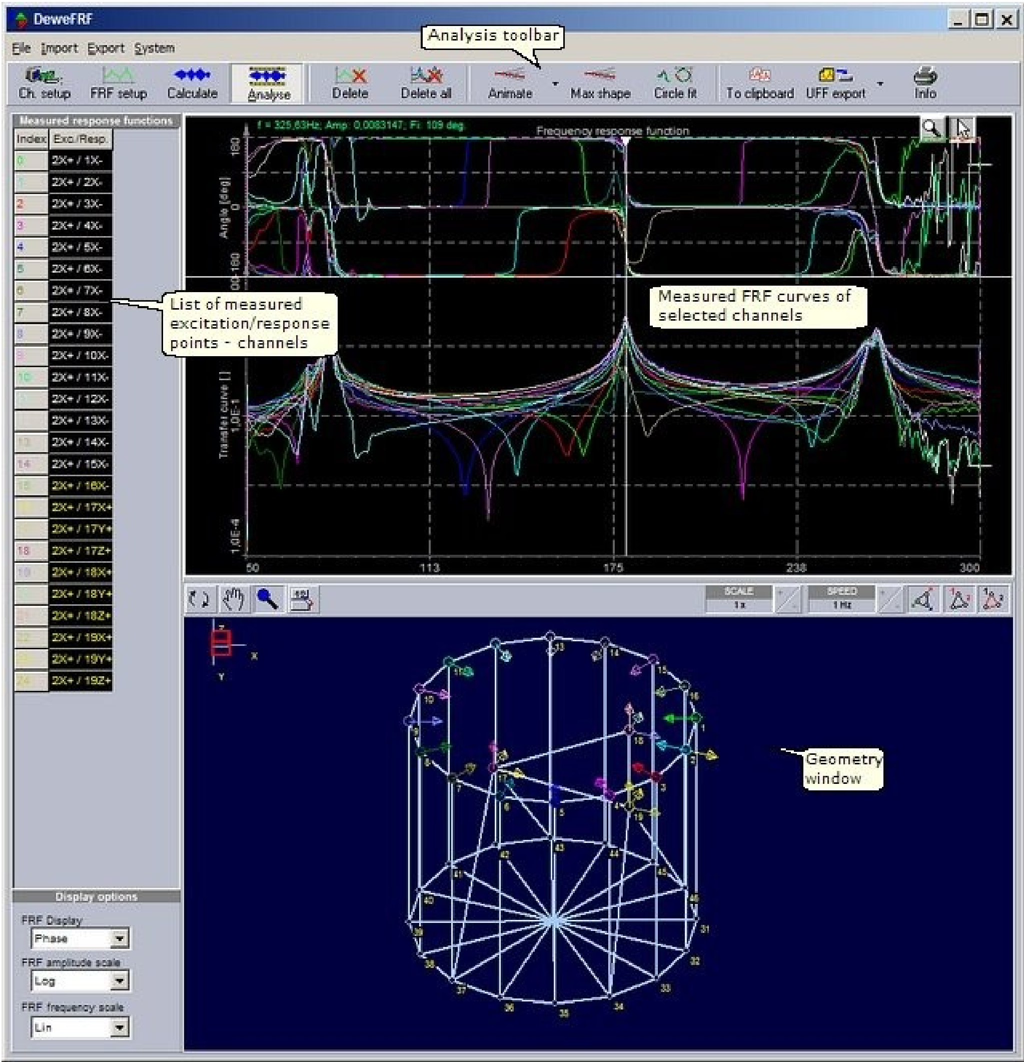
List of measured FRFs: the list of measured FRFs is the selector for the display these on the graph

Geometry of mechanical structure whit nodes, tracelines as well as local coordinate system

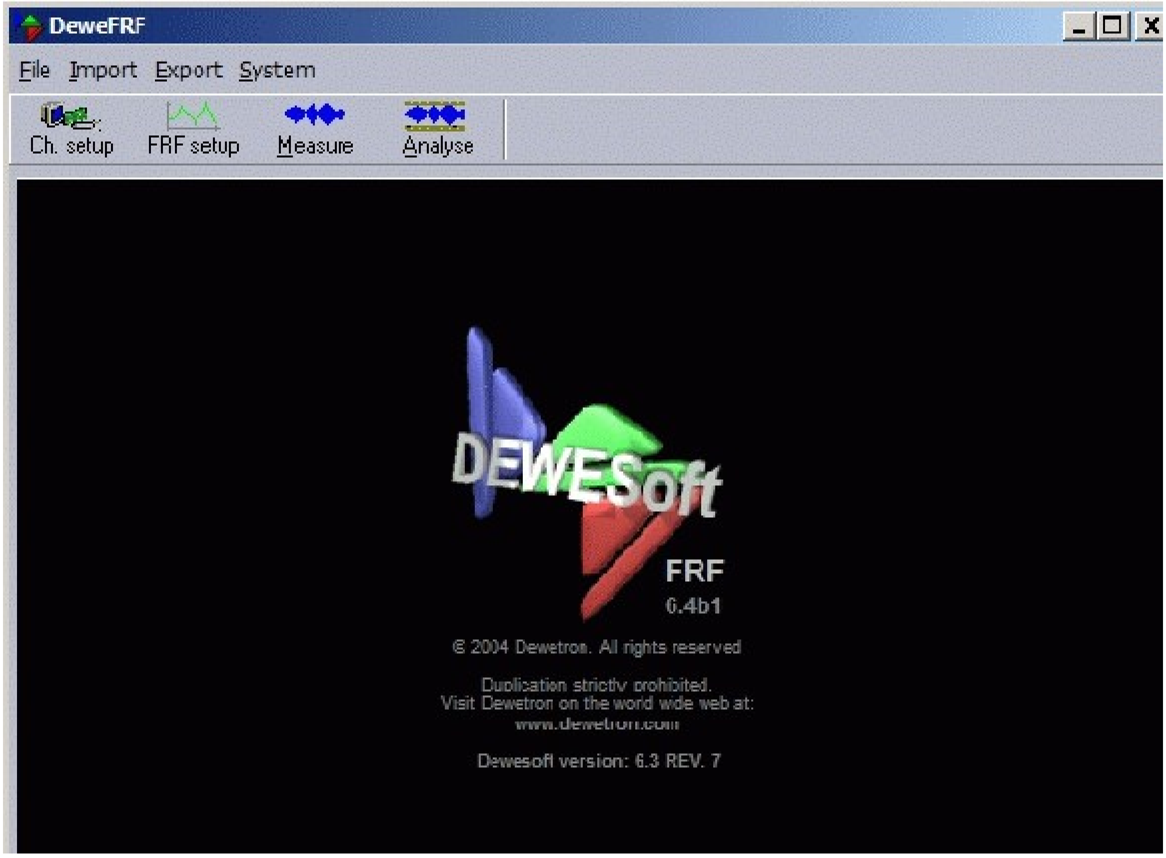
Analysis tools: for observation of the deflection of shapes with animating the points on the structure or to show the maximum deflection shapes (**Animation**, **Max shape** or **Circle fit**), tools for **export FRFs** and **information** about the settings of the project used to take FRFs

When FRFs are stored, the **Analyse** window can be started by clicking on 

FRF tools icon or starts automatically when **Confirm (F9)** button is pressed in **Calculate** procedure:

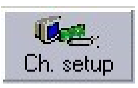


8 DeweFRF software interface



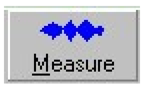
After start DeweFRF *basic* screen is displayed:

- In first row upper parts of screen are different **menus**: **File** **Import** **Export** **System**
- In second row upper parts are icons with DeweFRFs basic **tools**:

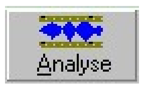


for **Channel setup**

for **FRF setup**



for **Measure** FRF data

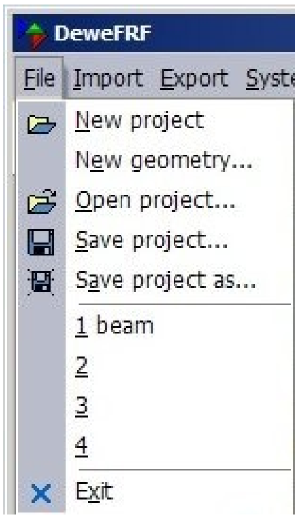


for **Analyse** FRF data

- In main part are data of: **central softwares producer**, **version** and **copyright**.

8.1 Menus

File menu



- open new project
- define new geometry
- open (load) saved project
- save project
- save project under different name
- open (load) last saved project

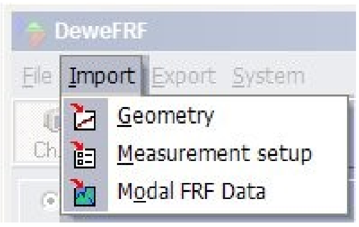
- exit Dewe FRF software

All choices displays standard Windows dialogs except:

New project open **Analog tab** for set all channels on Ch. setup tool screen.

New geometry ... open **FRF setup** tool with separate window **Point Marker** to define new measured structure

Import menu



load *existing* (previous defined and saved) geometry of mechanical structure

import Measurement setup data from *Dewesoft setup file* or *Dewesoft text setup*

load *existing* (previous saved) Modal FRF Data - measured data from *universal data file*

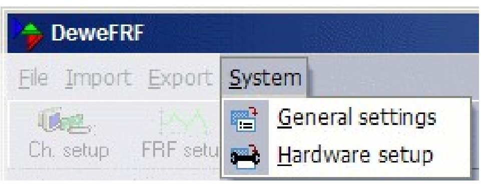
Export menu

save *new* or *modified* existent geometry of mechanical structure

export *existent* (modified) or *new* Measurement setup data

save Modal FRF Data - measured data

System menu



General and Hardware setting for work with DeweFRF software.

Part IV

Reference Guide

DEWE software can be used with recommended system hardware and software - **System requirements**.

Before installing DEWE software for working in **real** mode, you have to install appropriate *Hardware*, **Hardware driver**(s) and licence key - **Licensing**.

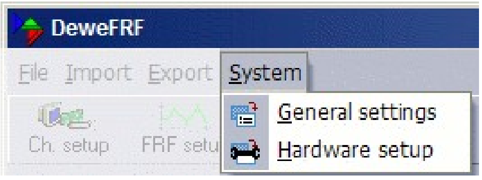
Before use DEWE software first of all choose is appropriate **DeweFRF System setting** in System menu.

In this guide is also information about **Theory of frequency analysis** and **H1, H2 and Hv calculation algorithms**.

1 DeweFRF System setting

Before use **DeweFRF** software:

- for working in **REAL** mode, you have to install appropriate Hardware and **Hardware driver**(s). After installing software you have to install licence key - **Licensing**
- first of all choose is appropriate **hardware** and general **setting**. In DeweFRF **System** menu you can choose:



General setting and most important **Hardware setup**.

1.1 General setting

We turn on and off sound signal (beep) when acquiring trigger by checking/unchecking "Beep on trigger".

1.2 Hardware setup

The hardware setup screen is same in all DEWE software.

When you first run DeweFRF, it does not know which hardware you intend to use it with, so **Analog HW Device** determine what we must set first.

Depending on hardware you can running Dewe FRF software on a computer:

- that does **not have** an A/D card in **DEMO** mode or
- with DEWE-ORION-1624, most National Instruments cards, most Data Translation cards, most Spectrum and Microstar's DAP boards in **PROF (REAL)** mode (for DEWETRON modules you can **activate** this modules),

which are determined in System menu → **Hardware setup** → 

tab. After installing software you can for easy license handling of software and options install licence key - **Licensing**.

On this screen is also available Channel Grouping for high-channel count applications. The grouping can be done completely independent: each group can have different number of channels and start at any channel number required.

In **Hardware setup** is also available:

Math for activated mathematic functions,

Analog out an analog output for National Instruments M series boards,

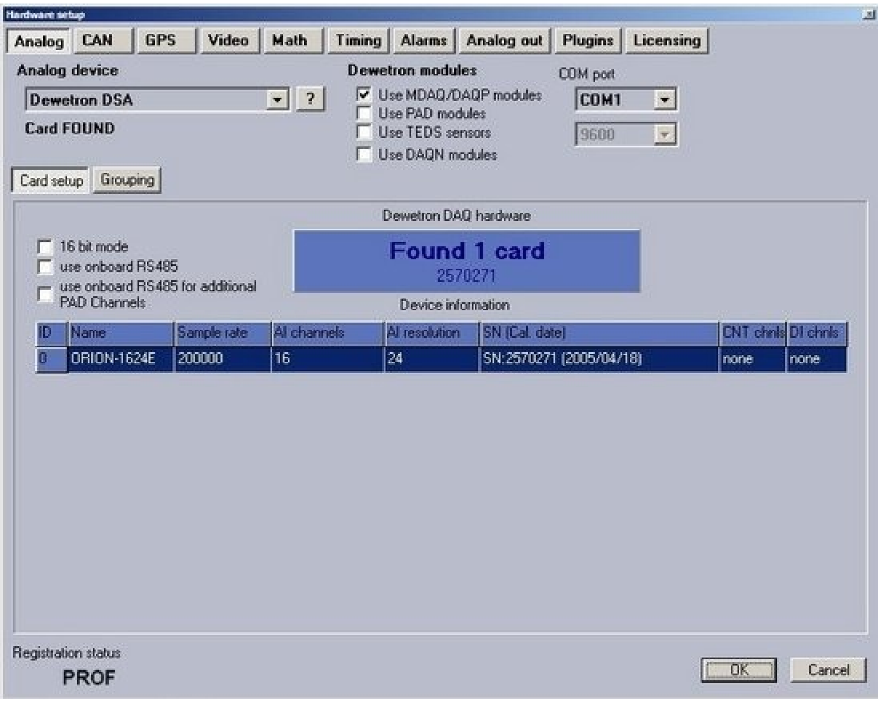
Licensing for easy license handling of software and options.

1.2.1 Analog HW Device

When you choose **Hardware setup** in System menu, initial screen with selected Analog - Card setup is displayed, depending on hardware you can running Dewe FRF software on a computer:

- that does **not have** an A/D card in **DEMO** mode or
- with DEWE-ORION-1624, most National Instruments cards, most Data Translation cards, most Spectrum and Microstar's DAP boards in **PROF (REAL)** mode. For DEWETRON modules you can **activate** this modules. Before use DEWE software in real mode, you have to install appropriate **Hardware driver**(s).

for explanation click on area of interest



- For high-channel count applications is also available Channel Grouping. The grouping can be done completely independent: each group can have different number of channels and start at any channel number required.

1.2.1.1 Hardware driver installation

Before installing DEWE software for working in **real** mode, you have to install the A/D board drivers. Be aware that the proper drivers are installed according to the following hardware driver list:

Manufacturer	Devices	Driver	Driver version
DEWETRON	DAQ- and PAD series	integrated	-
	DEWE-ORION-1624	RTdaq	2.0 or higher
National Instruments	E series A/D boards	NI-DAQ	6.9.3 or higher
	NI4472 and NI4551 series boards	NI-DAQ	6.9.3 or higher
	NI6602 series boards	NI-DAQ	6.9.3 or higher
	M series boards	NI-DAQmx	7.5 or higher
Data Translation	ISA, PCI and USB A/D boards	Omni CD	latest recommended
Microstar	DAP series boards	Accel 32	latest recommended
Spectrum	MI.30xx, MI.31xx, MI.45xx, MI.70xx series boards		latest recommended
Further hardware		under development	

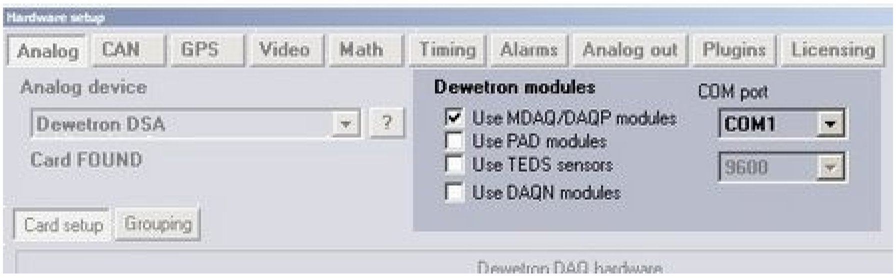
Detailed information about the installation procedure is available in the drivers documentation.

1.2.1.2 Activate DEWETRON modules

In System → Hardware setup → Analog tab - **Dewetron modules**

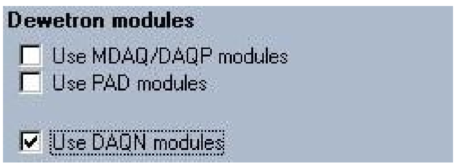
you can **activate** DEWETRON: **DAQN, DAQP/MDAQ, TEDS, PAD/EPAD** modules.

for explanation click on appropriate part of screen



Activate Dewetron DAQN modules

To activate DAQN series modules support simply *click* on the **Use DAQN modules** checkbox. This will enable a selection list of modules in the input setup.



Activate Dewetron DAQP/MDAQ modules

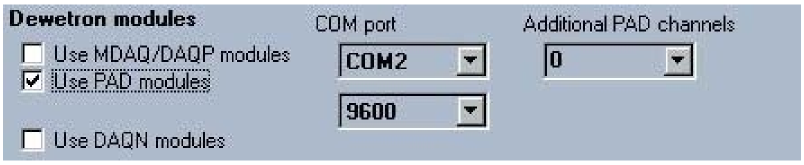
To activate DAQP series modules support in DEWE software *click* on the **Use MDAQ/DAQP modules** checkbox and *select* the **serial interface** which is used for the communication in COM port.

If you are using any DEWETRON hardware, like a DEWE-5000, DEWE-3020, DEWE-2010, DEWE-4010, DEWE-RACK, or DEWE-BOOK series product, then you should tell DEWESoft *which* COM PORT these modules are communicating on.

WARNING
Please check this to ensure that you have the correct com port, as incorrect settings can cause your system to hang up, or disable other devices that are already using the same com port.
Please notice that the communication speed is fixed to 9600 for DAQP/MDAQ series modules!

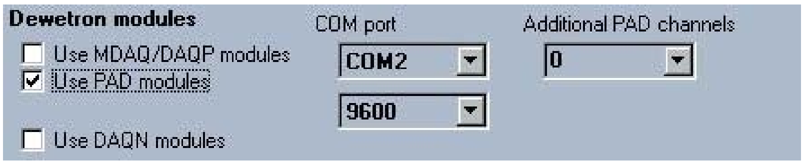
Activate TEDS support

TEDS sensors are supported by some DAQP/MDAQ series modules. When **DAQP/MDAQ** modules is *active*, the TEDS support can also be **activated** for these modules by *check* **Use TEDS sensors**.



Activate Dewetron PAD/EPAD modules

To activate PAD series modules support click on the **Use PAD modules** checkbox and select in COM port the serial interface which is used for the communication.

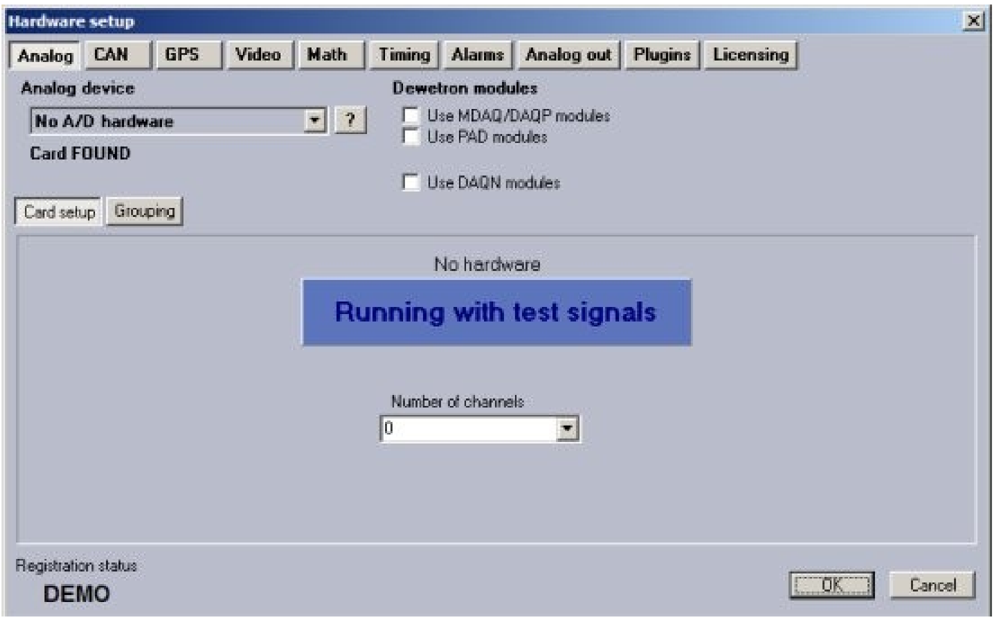


If you are using an expansion rack with additional modules or EPAD series modules, you can select **Additional PAD channels** to make them available.

1.2.1.3 DEMO mode

If you are running this software on a computer that does not have an A/D card or other required hardware, you can press the **CANCEL** button on Hardware setup screen, which will cause the software to generate waveform data for you, to allow the software behave realistically.

After starting Dewe software, if the program does not automatically show Hardware setup v dialog box (it should the first time you run it), please use the System menu and select **Hardware setup**.



If you want to run the software in *purely **demo*** mode (i.e., if you do not have either a National Instruments, Data Translation, Microstar iDSC, or Microstar DAP A/D card in computer), select in

System → Hardware setup → Analog tab - Analog device from the drop-down menu **No A/D hardware**

and then choose in **Number of channels** how *many* signals the system should generate for you. 4, 8, 16 are popular numbers.

To indicate the demo mode, Dewe software displays **No hardware**, **Running with test signals** and the Registration status displays **DEMO**.

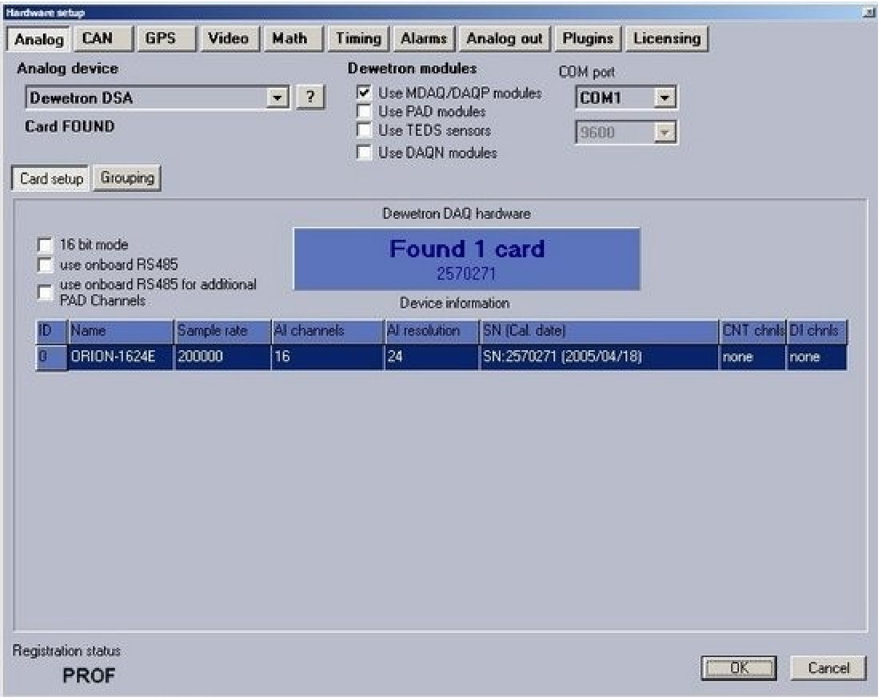
1.2.1.4 REAL - PROF mode

You can running Dewe FRF software on a computer with DEWE-ORION-1624, most National Instruments cards, most Data Translation cards, most Spectrum and Microstar's DAP boards in **PROF (real)** mode.

Before use DEWE software in real mode, you have to install appropriate **Hardware driver**.

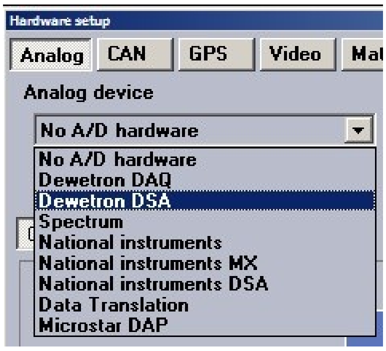
Dewe software will automatically identify the A/D card that you have installed and set it up for you.

for explanation click on appropriate part of screen



There will be nothing special that you need to do expect of selecting the right manufacturer from the drop-down menu in:

System → Hardware setup → Analog tab - **Analog device**



and activate DEWETRON modules in:

System → Hardware setup → Analog tab - **Dewetron modules**

Depending on the manufacturer, the installed hardware offers different settings.

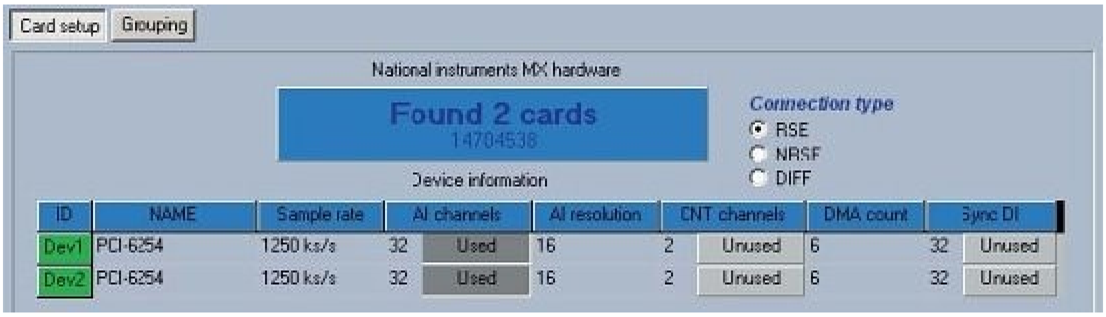
As an example, you can set the National Instruments hardware to RSE (referenced single ended - standard with DEWETRON systems), NRSE (non referenced single ended) or DIFF (differential).

Card specification is displayed in button part of screen **Device information**.

1.2.1.5 Channel grouping

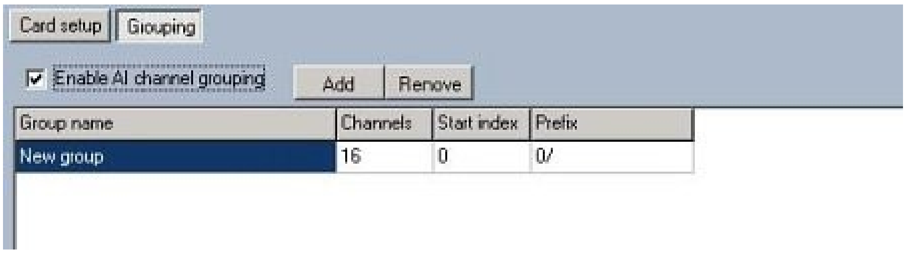
A very nice (and important) feature for high-channel count applications is the ability to group channels. The grouping can be done completely independent: each group can have different number of channels and start at any channel number required. Normally the systems will be set up with hardware-like groups, that means 16 channels per group.

Here is an example where grouping can be used: the system includes two 32-channel A/D boards (64 channels total). Let's see how to set up 4 groups each with 16 channels.

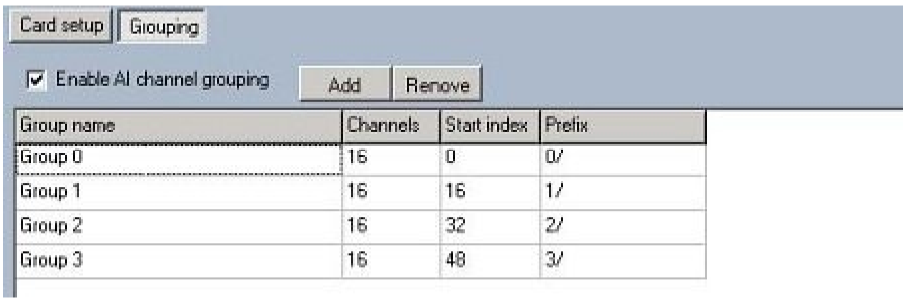


To use the grouping for this system you have to click the 

icon and activate the **Enable AI channel grouping** checkbox. The **Add** / **Remove** buttons will become active:



Now press the **Add** button to enlarge the table to the required number of groups. In our example we would like to define 4 groups, therefore we need 4 rows.



The following columns have to be filled out:

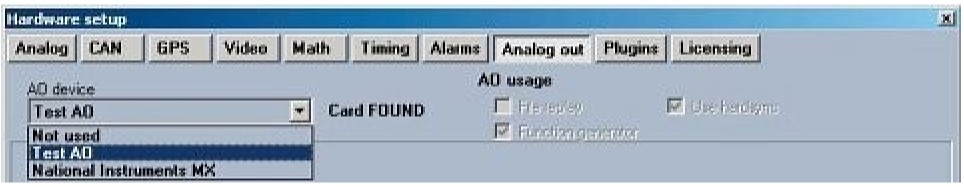
- **Group name** Can be any name, just helps you to identify the group.
- **Channels** Number of channels within this group (in our example always 16).
- **Start index** The number of the first channel within the selected number of channels.
- **Prefix** The channels are named automatically from 0 to the selected number of channels; as this will cause multiple entries with same channel names, a prefix will be added. Can be any text - try to keep it short.

After you have done this settings, the groups are already available in the **Setup**.

1.2.2 Analog out setup

Since DEWE Software 6.2, an *analog output* is available for National Instruments M series boards. This function is only available with **Function Generator** option in DEWE software.

For **test** issues set the hardware to **Test AO**.



Please find more information about analog out in User Guide → **Channel Setup** and **Measure**.

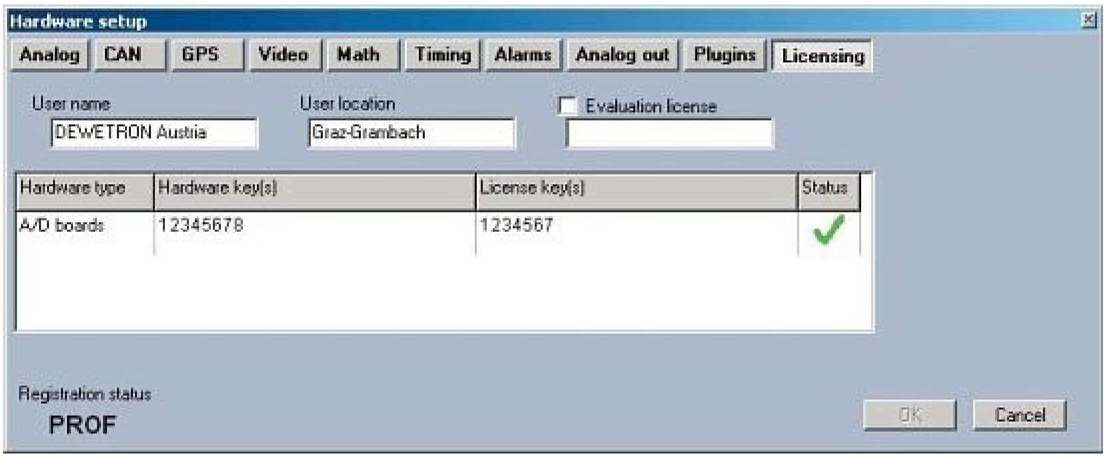
1.2.3 Mathematic functions

The mathematic functions are activated as a standard. Please read more about DEWE softwares mathematics in **DEWESoft Manual - Online mathematics**.

Mathematic functions are activated by checking appropriate checkbox.

1.2.4 Licensing

A new licensing tab offers the possibility for easy license handling of software and options. You have to enter your Username and User location first, then click into the License key(s) field and enter your license key(s).



If the license key is correct, the red cross in Status will be replaced by a green check mark and the registration status displays the version: LT, SE (Standard Edition), PROF (Professional Edition), DSA (Dynamic Signal Analyzer).

Save hardware key list

For systems with several A/D boards or other sources it is possible to store a hardware key(s) list into file (`*.dwkey`). To do that simply right-click in the licensing table and select Save hardware key(s) to file. This file is usefull for DEWETRON to make the installation easier.

Load license code(s) from file

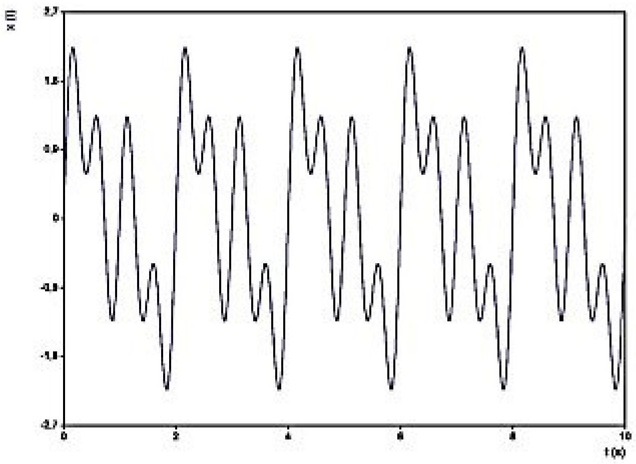
As mentioned above, systems may contain several A/D boards or other sources. In this case DEWETRON can create a License file (`*.dwcode`), which you can load to register your software. Right-click in the licensing tab and select Load license code(s) from file.

2 Theory of single channel frequency analysis

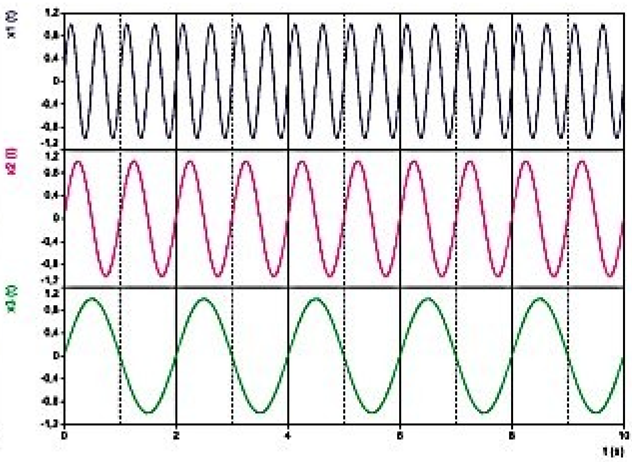
Frequency analysis is widely used tool because of its power to reveal the real nature of dynamic signals. Working for many years in practice doing measurements and lately also developing FFT analyzers, it became very obvious: knowing the theory of frequency analysis helps to reveal the problems we can anticipate and the ways to get around them. This article helps to understand the theory and answers some of the most asked questions regarding FFT:

2.1 What is frequency analysis

Frequency analysis is just another way of looking at the same data. Instead of observing the data in the time domain, with some not very difficult, yet inventive mathematics frequency analysis decomposes time data in the series of sinus waves. *Picture* 1 shows the signal, which consist of three sine waves with the frequencies of 0.5, 1 and 2 Hz, and then the decomposed signal (*Picture* 2).

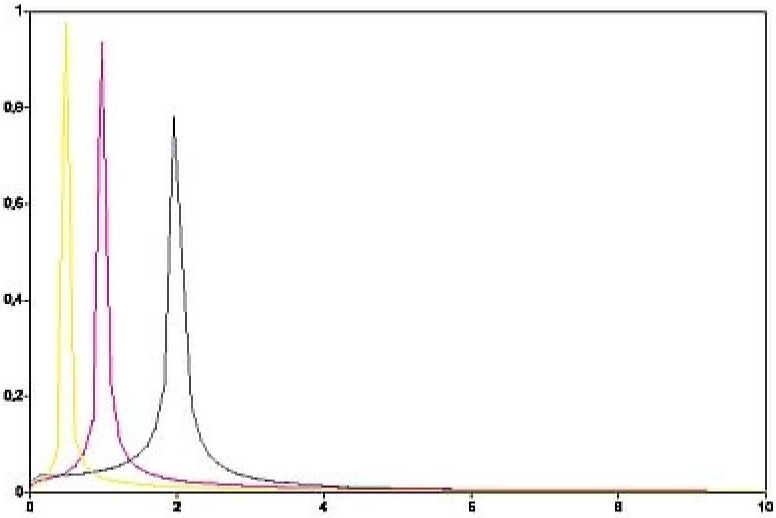


Picture 1: ORIGINAL SIGNAL



Picture 2: DECOMPOSED SIGNAL

Just to make those sine waves better visible, let‘s show them in a nicer way (*picture* 3). On x-axis there are frequencies and on the y-axis there are amplitudes of the sine waves.



Picture 3: DECOMPOSED FREQUENCY DOMAIN DATA

And this is really what the frequency analysis is all about: showing the signal as the sum of sinus signals. And the understanding how that works helps us to overcome problems that it brings with it.

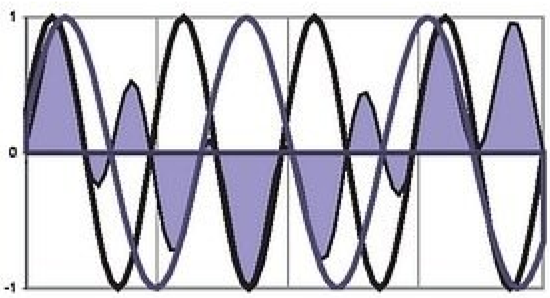
2.2 Theory of frequency analysis

Frequency analysis is converting time domain data to the set of sinus waves. So the theory says. But I like more if we say that frequency analysis checks the presence of certain fixed frequencies. Let's look at the equation of discrete Fourier's transform to figure out why:

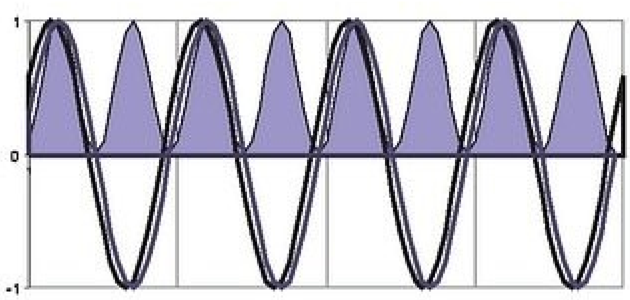
$$X(k\omega_0) = \sum_{0}^{N-1} x[n] \cdot (\cos(2 \cdot \pi \cdot k \cdot n / N) + j \cdot \sin(2 \cdot \pi \cdot k \cdot n / N)) \tag{1}$$

Let's think about how does it work. To check the presence of certain sine wave in a data sample, the equation does the following:

1) Multiplies the signal with sine wave of that frequency which we want to extract. Pictures 4 and 5 shows the signal with black line, which only consists 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

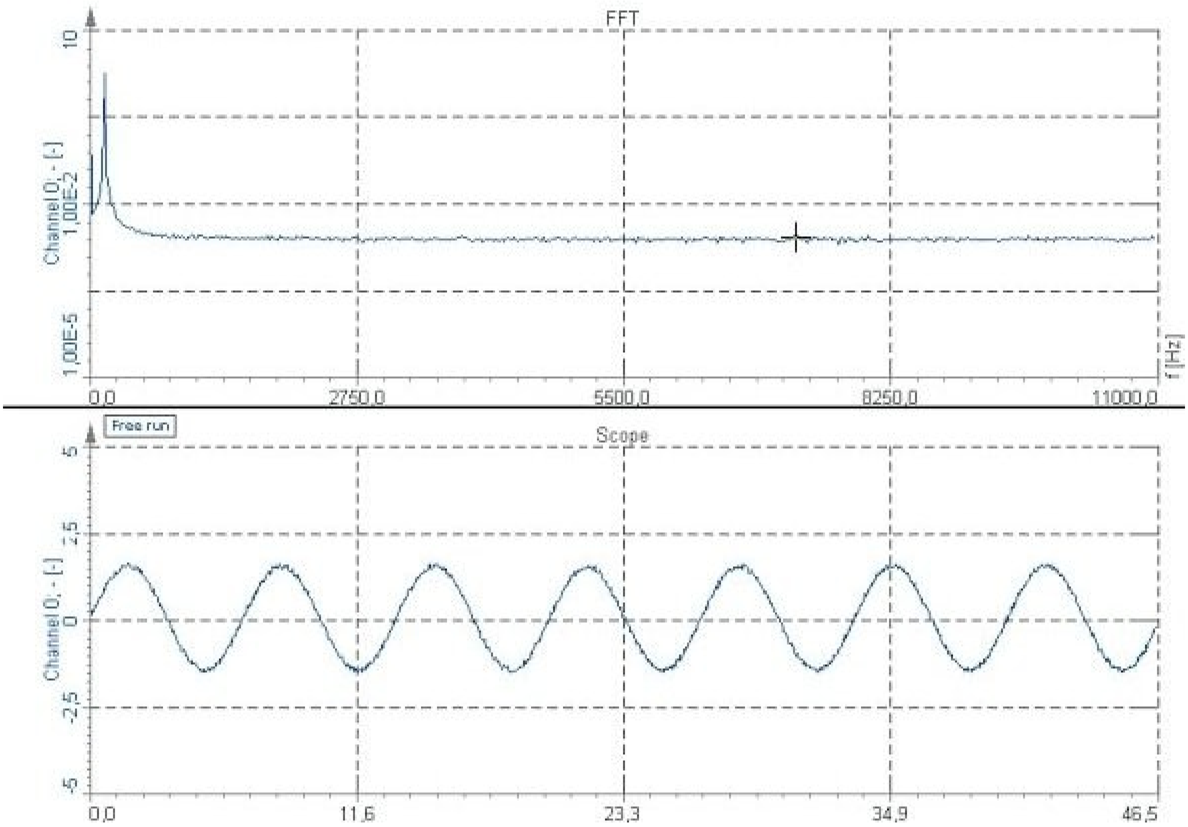
The principle shown above can extract basically any frequency from the sine wave, but it has one disadvantage. It is awfully slow. The next important step in the usage of DFT was the FFT algorithm. Because there is more than enough theory, let‘s just say that this analysis reduces amount of calculations 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) $$SampleRate = MaximumSignalFrequence \cdot 2 \cdot 1.25$$

1.25 is absolute minimum factor for getting right values also in the upper region of the FFT. This is the equation other way around of famous Nyquist criteria, which says that maximal signal frequency adequately presented in digitized wave is the half of sampling rate.



Picture 6: TYPICAL FFT SCREEN

Result of FFT is a set of amplitudes of certain frequencies. Number of lines in the set is user selectable, but they only change the resolution of the FFT. Line resolution is a change in frequency between two frequency lines, which are extracted from the signal and is calculated with equation:

(3) $$LineResolution = \frac{SampleRate / 2}{NumberOfLines}$$

So the question is: why not always use maximum number of available frequency lines, which gives more exact results? The answer is simple: because with larger frequency lines it takes more time to calculate FFT.

(4) $$TimeToCalculate = \frac{NumberOfLines \cdot 2}{SampleRate}$$

Just for fun we can also combine the equations above and we get

(5) $$LineResolution = \frac{1}{TimeToCalculate}$$

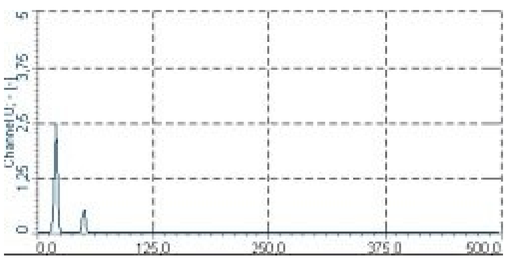
Let's look to the equations above and make a list for 22 kHz sample rate.

Number of lines	Line resolution	Calculation time
512	21,5 Hz	0,046 s
1024	10,75 Hz	0,093 s
4096	2,685 Hz	0,372 s
16384	0,67 Hz	1,49 s

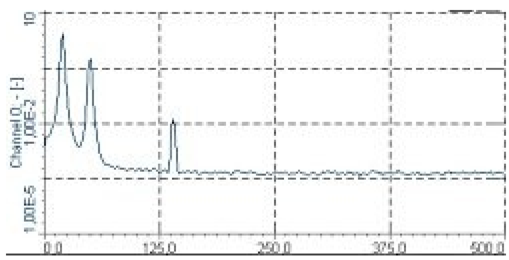
Table 1: EXAMPLE OF FFT SETTING

So the number of lines combined with sample rate also defines the speed of the FFT when non-stationary signals are applied. With more lines, FFT will appear slower and changes in signal will not be shown that rapidly.

Different amplitude scales of FFT can reveal more about the signal if used correctly. Linear amplitude scale gives the best view of maximum peaks in the signal, logarithmic can show more invisible peaks and signal noise but gives worse comparison of high and low peaks. Scale in dB gives the best estimation of signal noise if 0 dB is maximum measurable value and is also used in noise measurements, where the dB scaling is actually the result since the human ear has logarithmic sensitivity to noise.

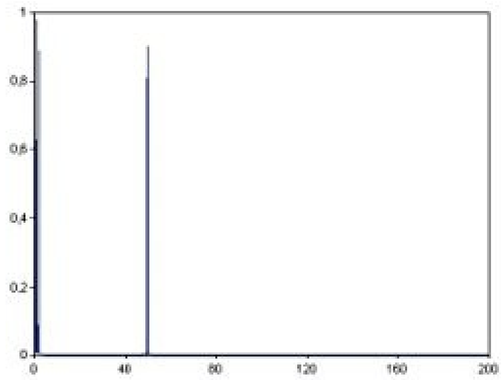


Picture 7: LINEAR AMPLITUDE SCALE

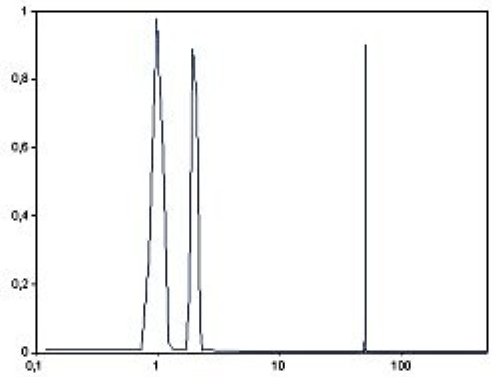


Picture 8: LOGARITMIC AMPLITUDE SCALE

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: LOGARITMIC 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, lets use the following one:

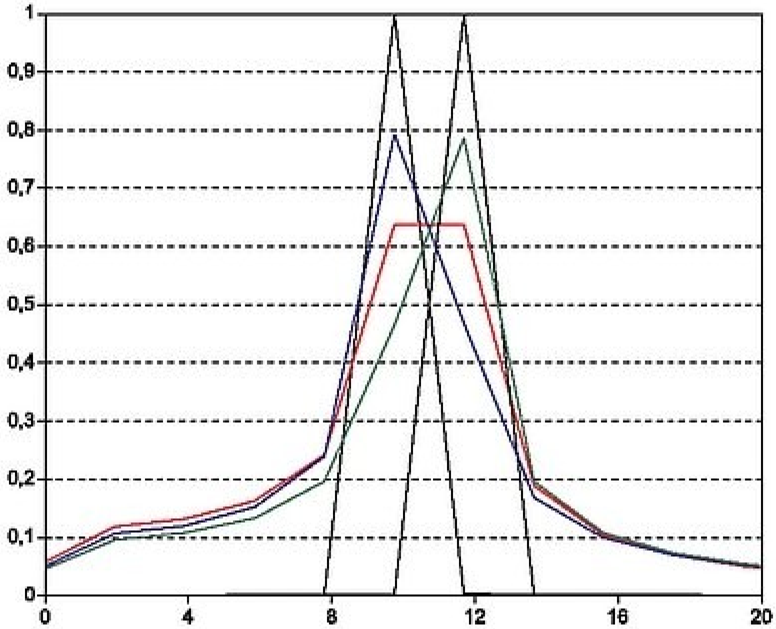
$$X(j\omega) = \sum_{n \to -\infty}^{n \to \infty} X[n] \cdot e^{-j\omega nT} \tag{6}$$

This is the theoretical discrete Fourier transformation and it has absolutely no error. The only thing, which could be bothering, is the sum goes from minus infinity to plus infinity. Because in the fast world we live in we don‘t have the time to wait that long, we run in problems.

1. Amplitude error (picket - fence effect)

The first problem is shown already in theory chapter. The sum can produce 'not null' results even if the signal does not correspond to the frequencies extracted from the signal. The pure frequencies are because of that somehow 'leaked' over neighbor frequencies. For the same reason if the frequency does not exactly fall on frequency line, amplitudes seem to be lower. This is called 'picket fence' effect.

Let‘s look at the *picture* 11 if a frequency is exactly on the spectrum line and some examples in between. On the picture below 10 and 12 Hz are the exact frequency lines. On the example there are 10 and 12 sine waves marked as black, which are transformed correctly, and there are frequencies in between which has lower amplitudes. Maximum amplitude error can go up to 35% of the correct value.



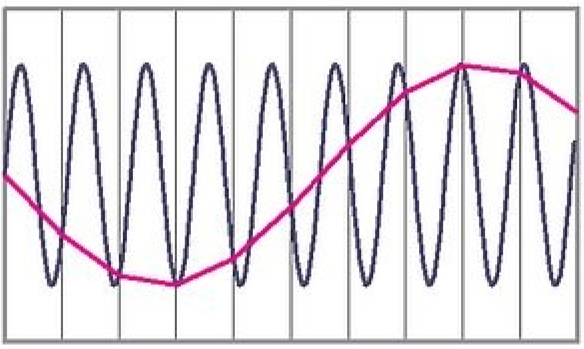
Picture 11: AMPLITUDE ERROR AND LEAKAGE OF THE FFT (NO WINDOW)

2. Leakage

Also we observe that especially if the sine wave is not on the frequency line, we get high amplitude values on both sides of the main band. The amplitudes are really high (with no window, it is about 10% of the original values for about 10 neighbor lines). If there is another sine wave in the signal in this region, which is lower than this 10%, it will be completely hidden by the leakage effect.

3. Aliasing

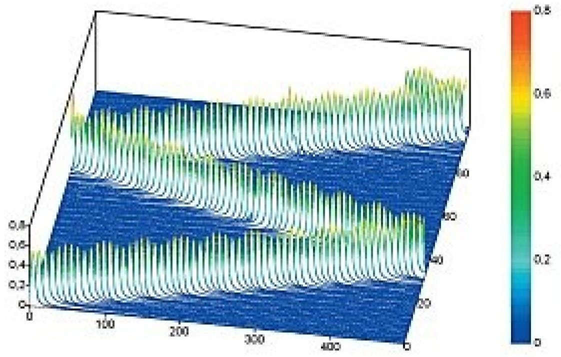
The other problem comes from the fact of the signal conditioning. If simple A/D converters are used, the sampling frequency must be at least twice higher than the maximum frequency of the signal. This effect is called aliasing effect. *Picture* 12 shows the reason for it. Vertical lines represents samples 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.



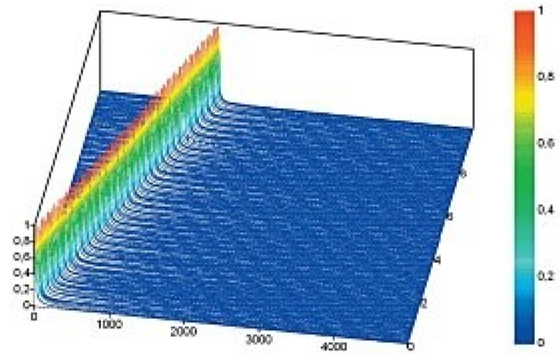
Picture 12: ALIANSING

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 *pictures* 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

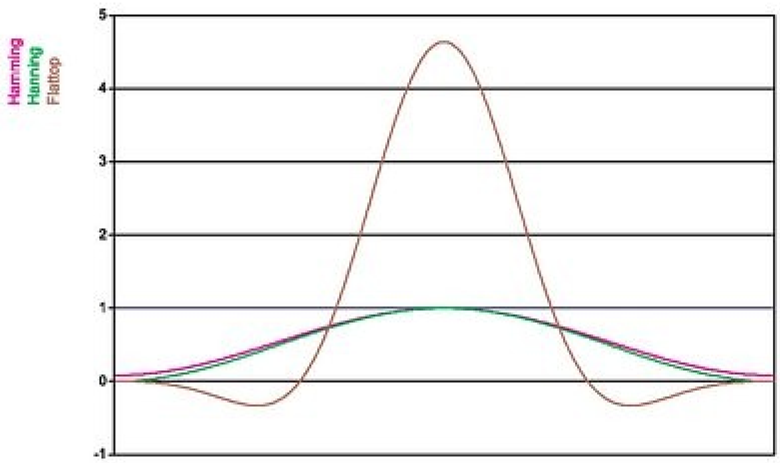


Picture 14: CORRECTLY SAMPLED SIGNAL

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 problems. 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

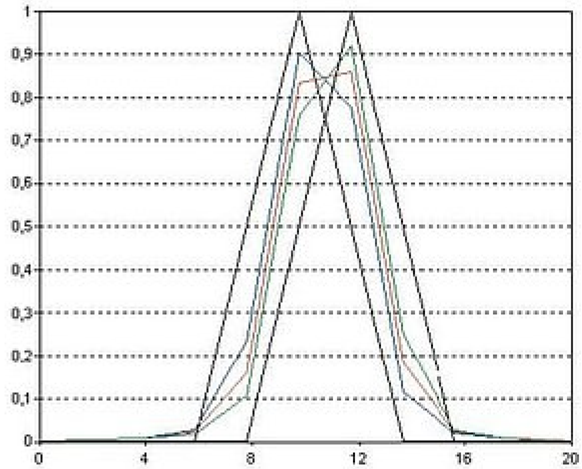
And here is the most common question to FFT: what are the differences between windows and when to use certain windows?

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).

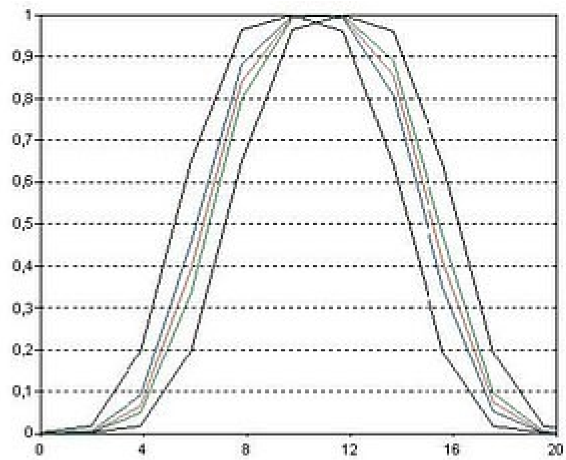
For general purpose, Hanning or Hamming are commonly used because they provide a good compromise between fall-off and amplitude error (maximum of 15 %). This comes from the fact that old frequency analyzers didn't have that many possibilities in terms of frequency lines and these two windows have narrow 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 sidebands 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.

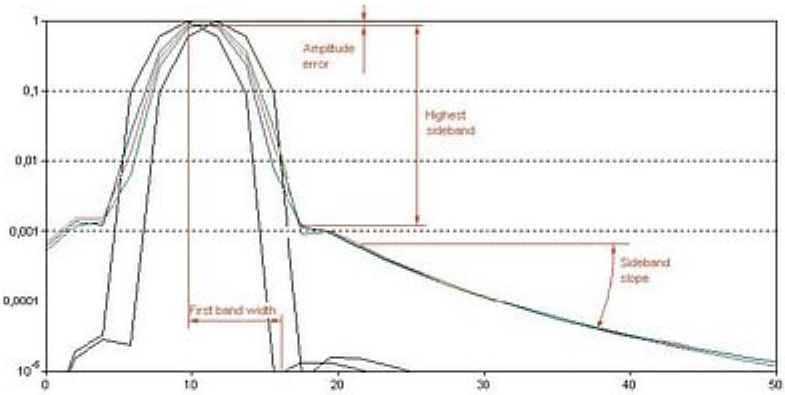


Picture 16*:* HANNING WINDOW



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.



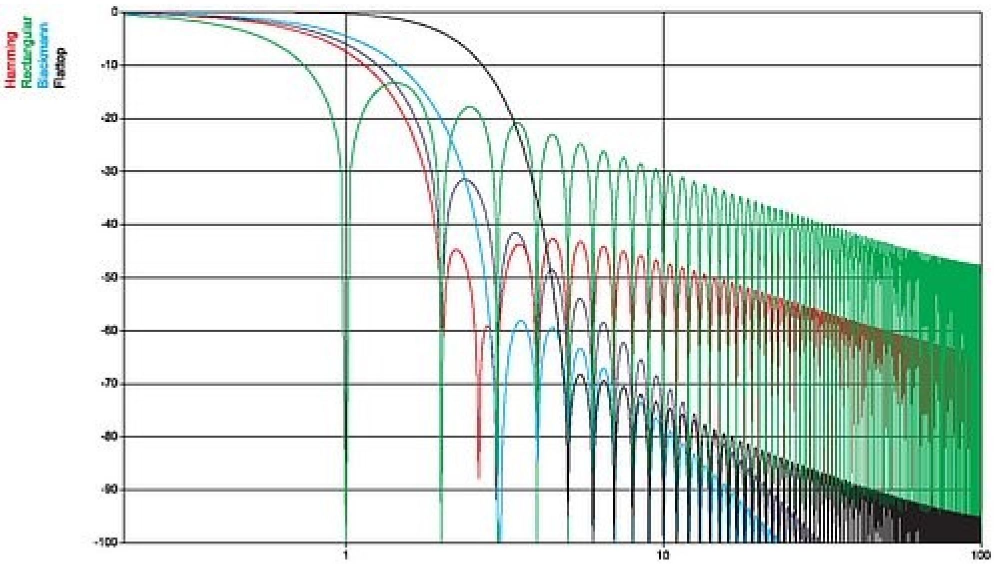
Picture 18: WINDOW PROPERTIES DESCRIPTION

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.

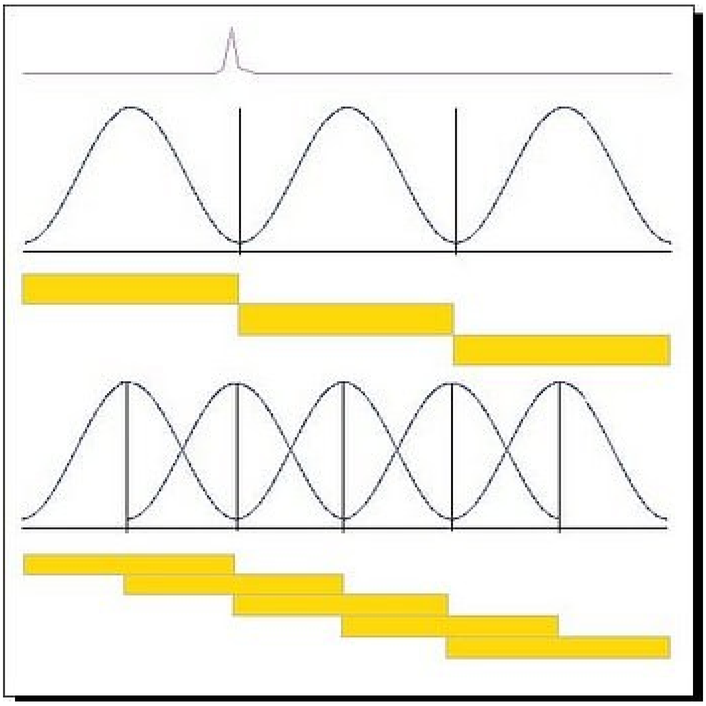
There is one more trick with windows: if we are sure that all the frequencies will fall on its frequency lines, rectangular window will give us the best result. For example to measure the harmonics of the power line (50 Hz in Europe or higher), choose 6400 or 9600 sample/sec sampling rate, that the line 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 average the frequency lines. We can do linear averaging (each FFT counts the same in the results), exponential (FFTs becomes less and less important with time) and the peak hold (only maximum results are stored and shown).

There is one more thing about the averaging: loss of information. When averaging is used with window functions, we could loose some data due to the window multiplication effects.



Signal

Windowing and averaging with 0% overlap

Windowing and averaging with 50% overlap

Picture 20: OVERLAPPING

In *picture* 20 there is one example where the signal only consists of one pulse. If we average the result, use window function and we are unlucky, signal will fall in region where the window sets the values to zero and in resulting FFT we will never see this pulse.

That‘s why there is a procedure called overlapping which overcomes this problem. It no longer calculates averages one after another, but takes some part of the time signal, which is already calculated and uses it again for calculation. There could be any number for overlap, but usually there is 25 %, 50%, 66,7% and 75% overlapping. 50% overlapping means that the calculation will take half of the old data. Now all data will be for sure shown in the resulting FFT.

With 66,7% and higher overlapping, every sample in the time domain will count exactly the same in the frequency domain, so if it‘s possible, we should use this value for overlapping to get mathematical correct results.

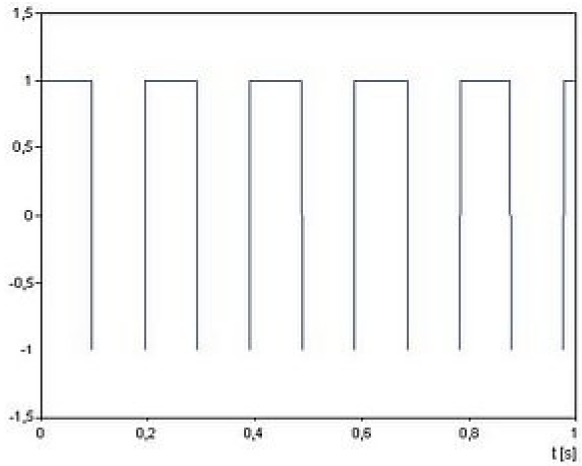
By the way, do you know what 'real-time' frequency analyzer means? It means that it is able to calculate and show data with 66,7% (or 50 % in some literature) overlapping and therefore has no data loss.

2.7 Representation of different signals in the FFT

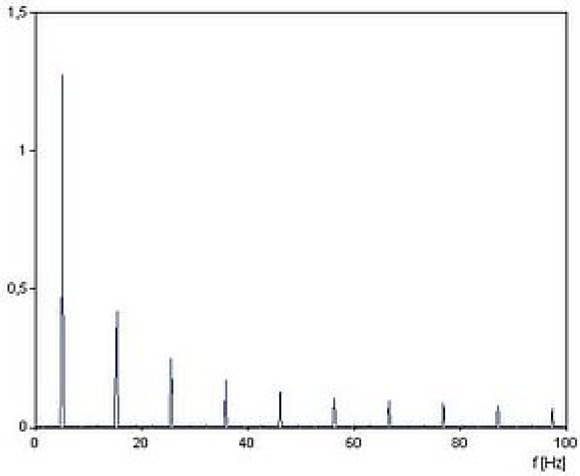
For now, we have only seen how the sine waves looks like in the FFT, but let‘s observe how other types of the signals look like:

1. Triangle, rectangular

All signals that are periodic with time but are not pure sine waves, produces base harmonic component as well as additional higher harmonics. More the signal is not like sinus, higher the harmonics are.



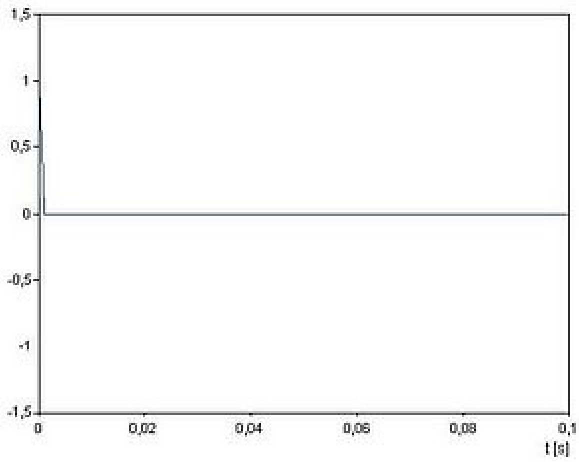
Picture 21: TIME DOMAIN SQUARE WAVE



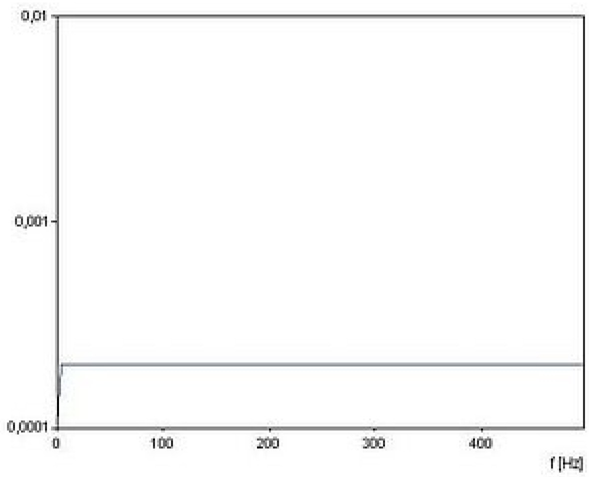
Picture 22: FREQUENCY DOMAIN SQUARE WAVE

2. Impulse

Impulse is quite interesting thing. It cannot be described with sum of sine waves. Or in different words: it is shown equally on all of the frequency lines. That‘s the reason why we use it as the basic excitation principle to get frequency responses of the system. The other ones are swept sine and noise, but this is already a part of another story - dual channel frequency analysis.



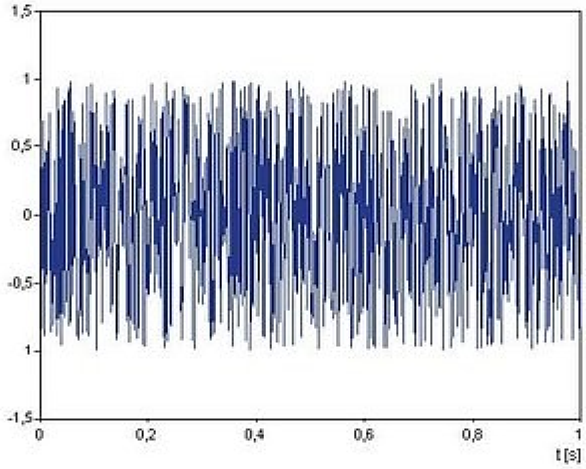
Picture 23: TIME DOMAIN IMPULSE



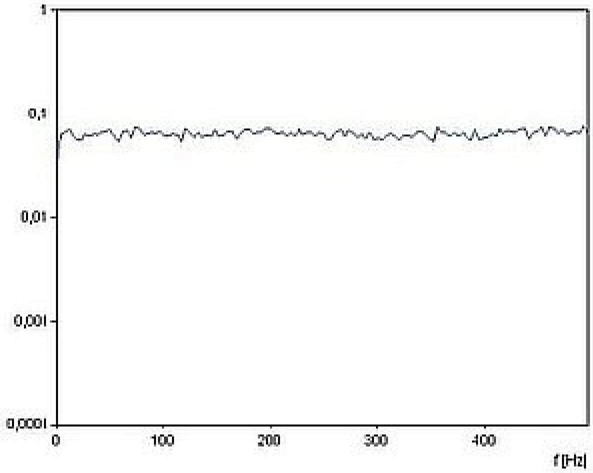
Picture 24: FREQUENCY DOMAIN IMPULSE

3. White noise

The theory says that white noise consists of all frequencies. That‘s why the infinite frequency spectrum of the white noise is the straight line. However the shorter the samples are, more different amplitudes for certain frequencies we get in the noise level. It is the best to use averaging to get fixed noise line. *Picture* 24 shows already averaged FFT of the white noise.



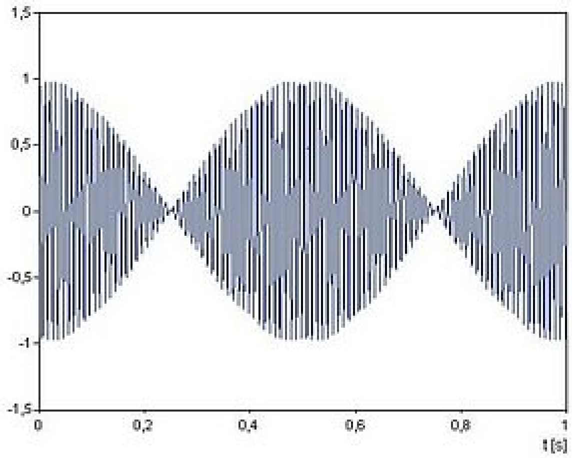
Picture 25: TIME DOMAIN NOISE



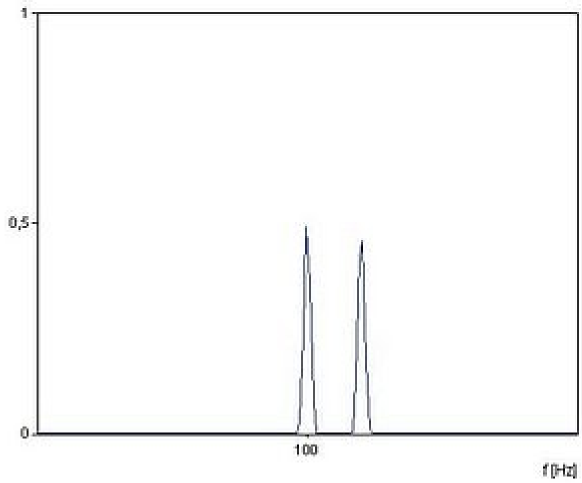
Picture 26: FREQUENCY DOMAIN NOISE

4. Beating (two closely spaced signals)

Beating in time domain is somehow hidden and looks like one frequency with changing amplitudes. Only FFT reveals two frequency lines, if we choose high enough line resolution. The difference between two frequencies is the modulation frequency shown in the time domain.



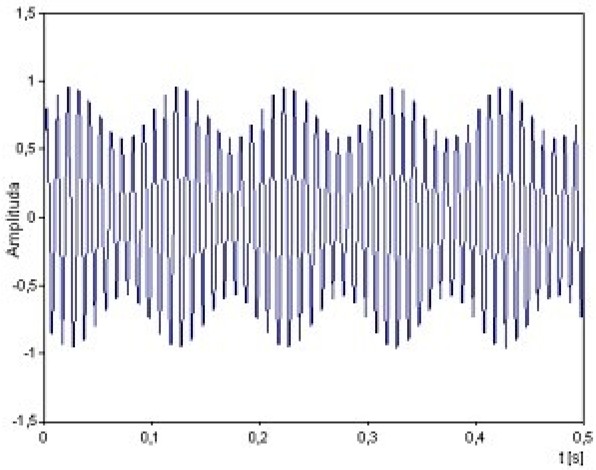
Picture 27: BEATING IN TIME DOMAIN



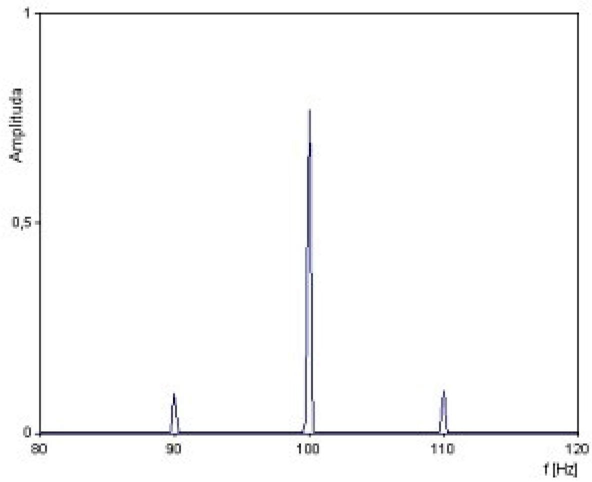
Picture 28: BEATING IN FREQUENCY DOMAIN

5. Amplitude modulated signal

Amplitude modulated (AM) signal is shown as two sideband frequencies. Difference between the base frequency and the sideband frequency is the modulated frequency (10 Hz in this case) also seen clearly in the time domain. The rule here is the same like with beating - to reveal the modulation, we should choose high enough line resolution. In fact, the time signal, which is the base for the FFT calculation, should show some modulation peaks. When windowing is used (we know that the base band could be even 4 lines wide) and the main band, which is always the highest, covers the modulation with low line resolution, time signal should show at least 16 or 32 modulation peaks that the modulation is shown in the FFT.



Picture 29: TIME DOMAIN AM SIGNAL



Picture 30: FREQUENCY DOMAIN AM SIGNAL

2.8 Conclusion

There's no doubt about it - FFT is a powerful analysis. But it is just like a car. If we know its limits, we can get so much more performance from it. That's why it is important to know how it is calculated and the tricks that might come handy. The next step in one channel FFT analysis is the real signal analysis, which comes from the knowledge of the measurement. If it's machine vibration, we have to know the structure and the properties of mechanical elements and structures. If it is an electronic circuit, we have to know the properties of electronic components to figure out the responses.

With this step the FFT analysis steps out of being a mathematical principle and becomes just a handy tool, like it really should be.

3 H1, H2 and HV algorithms for calculation

H1 algorithm

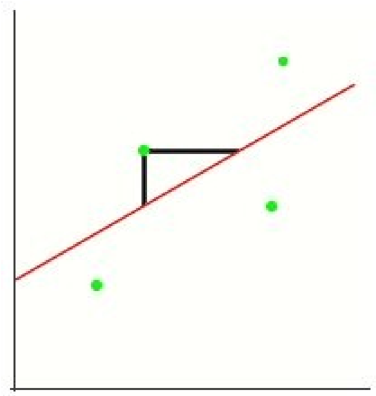
H1 algorithm expects **noise** on output.

H1 algorithm tends to minimize the noise on output. In the presence of noise it underestimates amplitude at resonances and anti-resonances.

H2 algorithm

H2 algorithm expects **noise** on input.

H2 algorithm tends to minimize the noise on input. In the presence of noise it underestimates amplitude at resonances and anti-resonances.

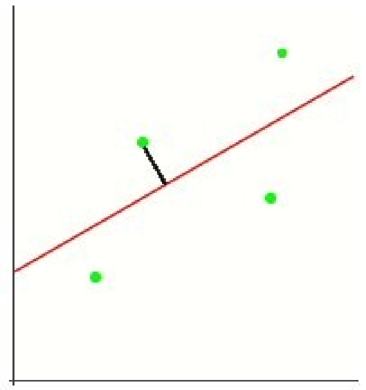


Vertical offset - leas squares (minimizes distance in x or y direction).

Hv algorithm

Hv algorithm expects **noise** on input and output.

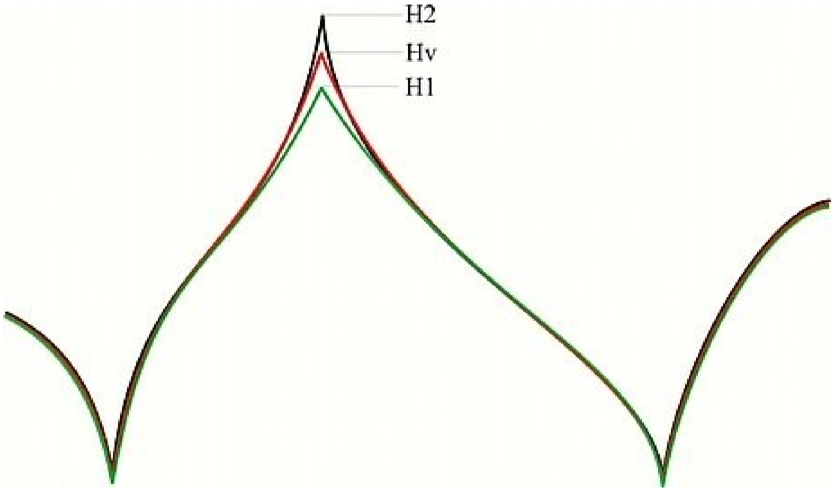
Hv algorithm tends to minimize the noise on input and output. It is always bounded by H1 and H2. The only cut down is that because of it's complexity it uses a lot of computing power.



Perpendicular offset - total leas squares (minimize distance in x *and* y direction).

Comparison of given method

If there is *no* noise present all three algorithms give the *same* results. That is why everything that is possible should be done in order to eliminate noise on input and output.



Index

- G -

- H -

- I -

- M -

- N -

- O -

- P -

- R -

- S -