OROS -Series3/NVGate®

User's Manual

Table of Contents

Chapter 1: Hardware 3
Hardware 3
Chapter 2: Environment 11
Hardware / PC connection 11
NVGate Installation and Update 13
IP address rules 16
Chapter 3: Software environment 17
Modes 17
On-Line acquisition 20
Post analysis 22
Stand-alone 25
Start'n Go 27
Interface description 30
Analyzer Setting Browser (ASB) 31
Project Manager 35
Control panel 38
Log window 40
Toolbar 41
Display 47
Chapter 4: Tutorials 51
First lesson: spectrum analysis 51
Acquisition and recording 65
Waterfall lesson 72
Reporting 81
Post analysis 88
Synchronous order analysis 93
Order tracking with FFT 97
Structural analysis 104
Acoustic analysis 111
Chapter 5: How do I 117
How do I Display Numerous results ? 117
How do I check the current value of my 3 major settings ? 119
How do I compare current analysis with saved results ? 124
How do I split my record in multiple files ? 126
How do I perform the same measurement as previous one ? 128
How do I create and use my units ? 130
How do I automatically start with my preferred setup ? 134
How do I get my results in the same format ? 136
How do I use trigger on a mask crossing ? 138
How do I post-analyze all my signals with the same setup? 140

Chapter 1: Hardware

Hardware

OROS-3 Series Instruments Range

This help applies to the whole OROS-3 Series analyzer range.

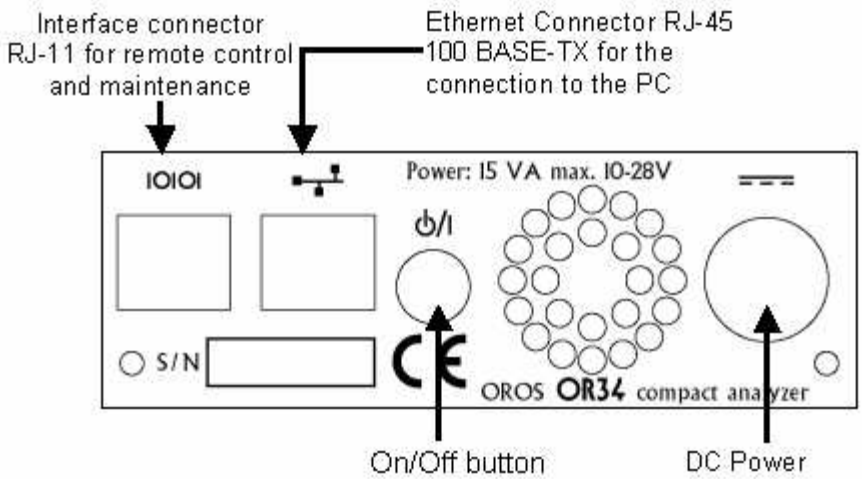
OR34-OR35

Front Panel

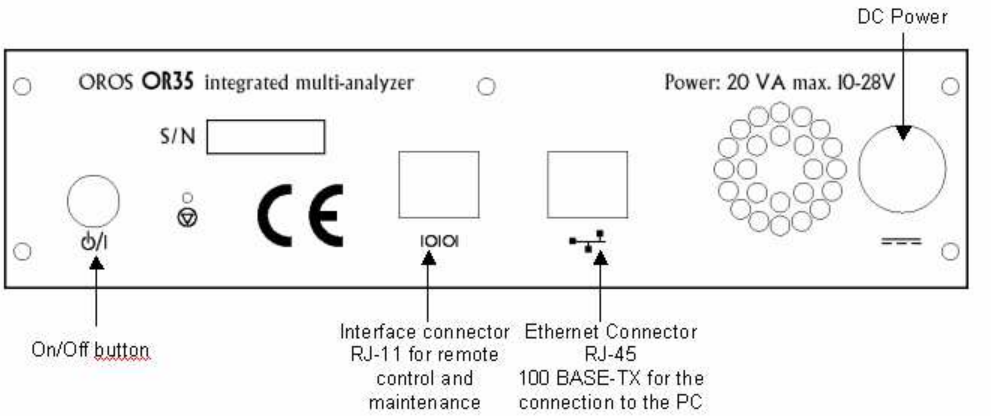
	OR35	OR34
Inputs	1 to 8 Dynamic Inputs	1 to 4 Dynamic Inputs
Out.	Generators 1 & 2	Generator 1
Ext.	External Sync.1 & 2	External Sync.1 & 2
Overview		

Back Panel

OR34



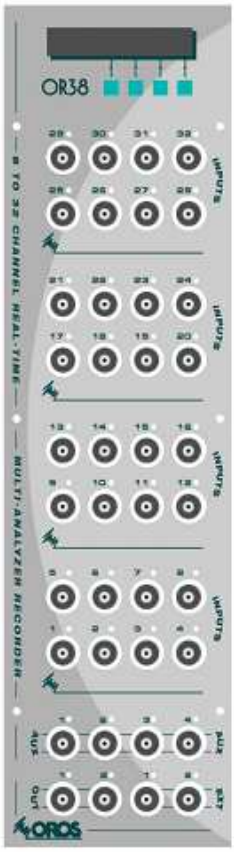
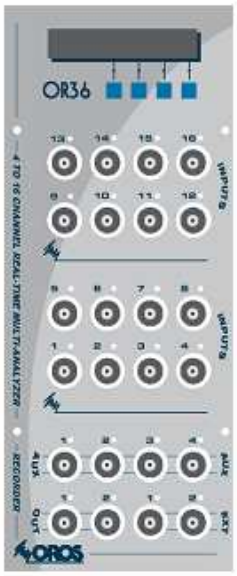
OR35



1 to 4 DC Inputs are available on the back panel.

OR36-OR38

Front Panel

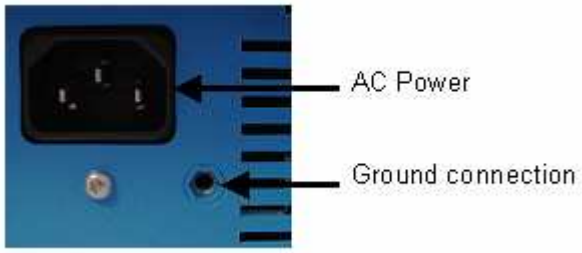
	OR38	OR36
Inputs	1 to 32 Dynamic Inputs DC Inputs[1]	1 to 16 Dynamic Inputs DC Inputs[1]
Out.	Generators 1 & 2	Generators 1 & 2
Ext.	External Sync.1 & 2	External Sync 1 & 2
Aux.	Auxiliary connectors[1]1 to 4	Auxiliary connectors[1]1 to 4
Overview		

[1] *Depending on the purchased options*

Back Panel

AC Power supply

The OR38 hardware can be AC power supplied using the following plug



Battery

OROS-3 Series Hardware contains an internal battery block particularly useful for autonomous operations (except for OR34) or in case of temporary power failure (all OROS-3 Series analyzers).
The batteries of OR3X Hardware are designed to provide maximum trouble-free life.
To obtain the longest battery life, please follow the following advice:

- Temperature above normal room temperature will shorten battery life. If OROS-3 Series Hardware is stored or shipped at more than 40°C (104°F), recharge it before running the system.
- Charge the battery in cool area (The battery is in charge when OROS-3 Series Hardware is plugged to a DC or AC power supply and powered on. Charge stops when the Front Panel indicates 100%).
- Never store OROS-3 Series Hardware with a discharging battery. Recharge it immediately after being discharged.
- All batteries gradually lose their charge (the higher the temperature is, the

quicker the batteries lose their charge). If you store your system for a long time without using it, recharge the batteries every two or three months. This practice will prolong battery life.

Fan

The cooling fan is used to reduce the temperature inside OROS-3 Series Hardware. From the Front Panel or NVGate Software, the user can manually switch it on or switch off (for very sensitive acoustic measurements).
The fan operation will be automatically forced when the temperature inside the OROS-3 Series Hardware reaches 50°C and stops at 43°C.

Do not cover the OR3X Hardware in order to let the ventilation operate properly

General advices

Review the following safety precautions to avoid injury and prevent damage to this product or any products connected to it. To avoid potential hazards, use this product only as specified.

Only qualified personnel should perform service procedures.

To avoid fire and personal injury:

- Use proper power cord. Use only the power cord specified in this product and certified for the country of use.
- Connect OROS-3 Series Hardware to the ground. This product is grounded through the grounding conductor of the power cord. To avoid electric shock, the grounding conductor must be connected to earth ground.

To avoid electric hazards, always connect OROS-3 Series Hardware to the ground during battery operation.

- Respect all terminal ratings. To avoid fire or shock hazard, observe all ratings and marking on the product.
- Do not operate without covers. Do not operate this product with covers or panels removed.
- Avoid exposed circuitry. Do not touch exposed connections and components when AC outlet is connected.
- Do not operate with suspected failures. If you suspect there is any damage to this product, make it inspected by qualified service personnel.
- Provide proper ventilation.
- Do not operate in wet/damp conditions.

- Do not operate in explosive atmosphere.
- Keep product surfaces clean and dry.

To know more see the Installation guide provided with the software

Powering On/Off OROS-3 Series analyzer

OR36-OR38

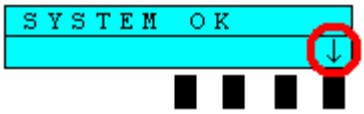
Only OR36 and OR38 hardware have a LCD screen on the front panel.

To start OR36/OR38 Hardware, press the right-hand button of the Front Panel. " OROS-3 Series Powering up" is displayed on the LCD screen.

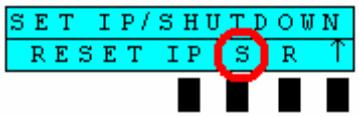
Hardware auto-tests are performed until "System OK" appears on the screen.

If auto-tests return an error message, please contact your OROS Support.

Power off OR36/OR38 Hardware by pressing on the right-hand button of the front panel, then on the second button for "S" (Shutdown) and confirm by pressing "Y" (Yes).
From the LCD screen:



Pressing the right-hand button: . The following menu is displayed:



This menu allows to:

- Shutdown the analyzer (second button for "S")
- Reset the analyzer (third button for "R")
- Go back to the first menu (fourth button for "").

Answer ”Y” for Yes

If OR36/OR38 Hardware does not respond, press the right-hand button until it turns off. Then, power it on again after a few minutes.
Note: This type of shutdown may reduce disk life.

OR34-OR35

A beep is produced when the hardware is connected to the mains.

Power on OR34/OR35 Hardware by pressing the power On/Off button of the back panel and wait until a beep is produced (2sec) then release the button. The LED is fast flashing green (system is loading) and then slow flashing green (system OK).

Power off OR34/OR35 Hardware by pressing the power On/Off button of the back panel and wait until a beep is produced (2sec). The LED is flashing red.

I**f OR34 Hardware does not respond**, unplug the main power supply.

I**f OR35 Hardware does not respond**, unplug the main power supply and press the reset button of the back panel.

OR35 - 4, 6, 8 ch

OR36 - 4, 8, 12, 16 ch

OR38 - 8, 16, 24, 32 ch.

OR34 - 2, 4 ch.

OROS 3-Series analyzers feature an advanced and powerful **remote controller** as an option.
This device will allow remote controlling a lot of features:

This remote controller is ideally suited for field measurement thanks to its robustness, secured wired connection, bright display and flexibility.

Chapter 2: Environment

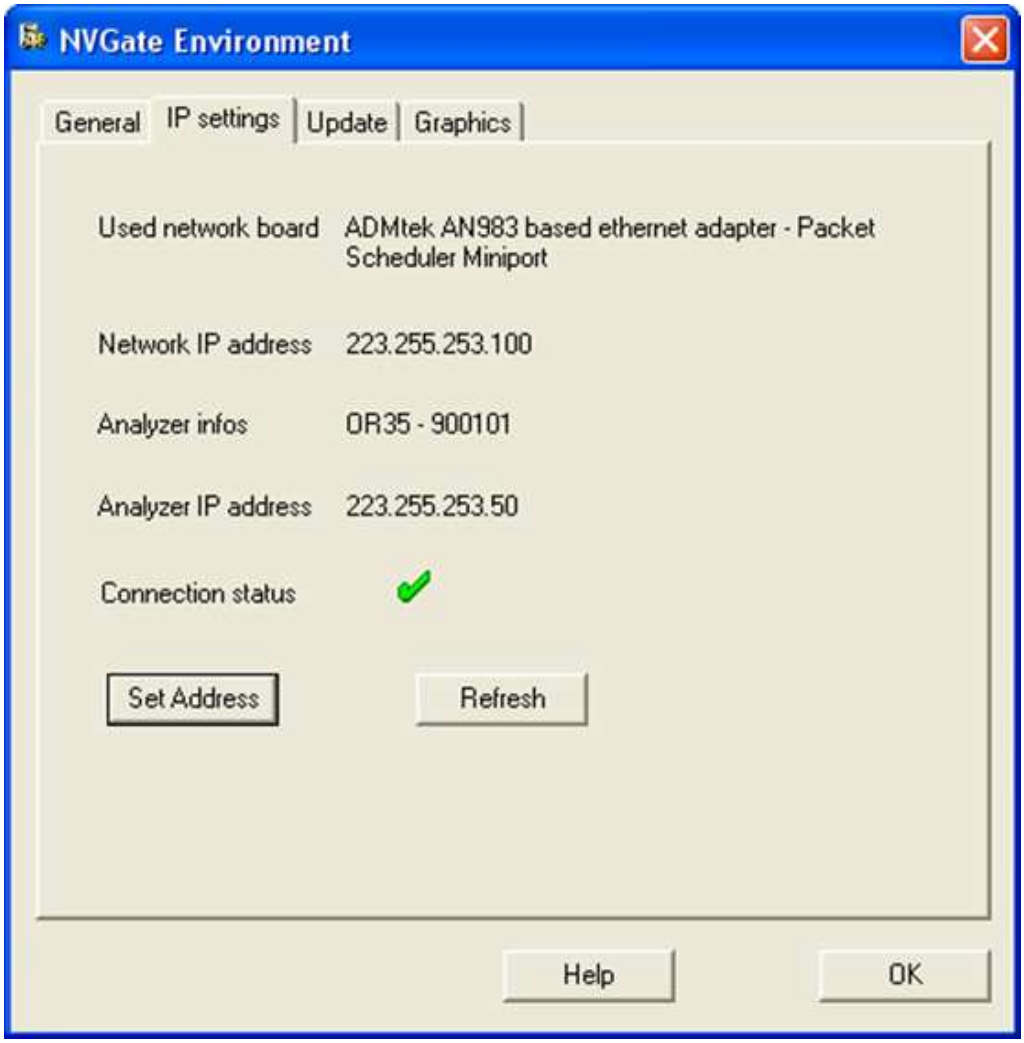
Hardware / PC connection

How to connect and validate the connection between PC and analyzer?

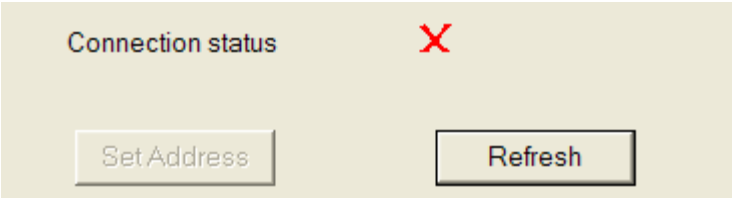
Connect the analyzer to the PC with ethernet link

Note: Use exclusively the OROS ethernet link for the connection

Go to 'NVGate Environment' in the Installation setup. 'IP settings' window allows to check that the hardware is connected to the PC.



If the analyzer is not recognized like below, press Refresh to activate the connection.



If the red cross is still displayed, see the '[IP address rules]' to respect for good connection.

Note: to know more details about the hardware and PC connection see the **'Installation guide'**.

NVGate Installation and Update

NVGate Installation

Equipment required for the installation

For the installation in the "Office Mode" (without OROS-3 Series Hardware):

- Parallel port dongle or USB dongle
- Software CD containing NVGate Installation Setup

For the installation in the "Connected Mode" (with OROS-3 Series Hardware connected):

- OROS-3 Series Instrument (OR34, OR35, OR36 or OR38)
- AC Power cord and/or DC power supply
- 1 crossed Ethernet connection cable
- Software CD containing NVGate Installation Setup
  Windows 2000, XP, Microsoft Excel and Word are registered trademarks of Microsoft company.

Note: before installing NVGate® check that the configuration of your PC is Windows XP Pro Service pack 2 or Windows 2000 Service Pack 4. For Windows 2000 and Windows NT.
User must be **power user or administrator** in order to use NVGate Software.

Start the installation

Insert the CDRom in the appropriate driver and wait for the Autorun program (or run the "setup" application from the NVGate Roots.
The following window is displayed:

Then the following is displayed:

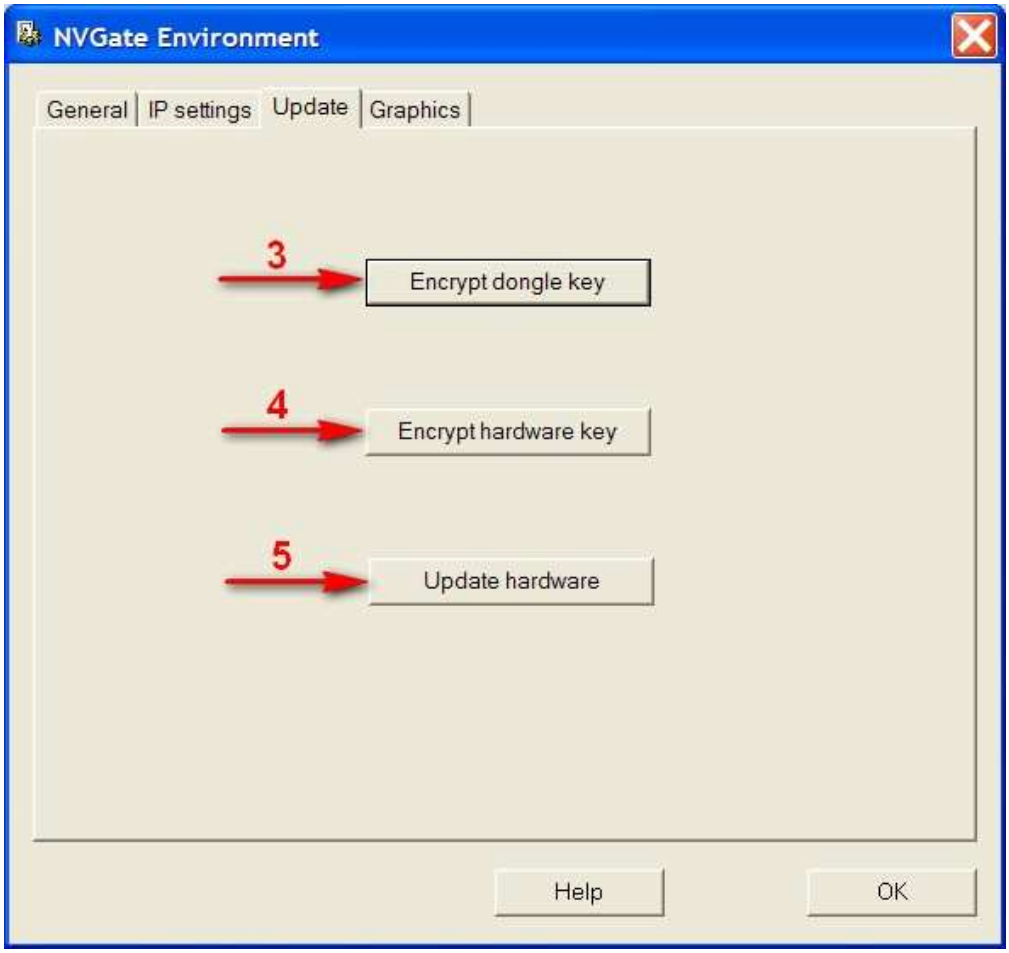
Click on 'Next >'
Then follow the different steps through the wizard.

Note: to know more details about the software installation see the **'Installation guide'**.

NVGate Update

The following procedure is available for update to NVGate V5.0.

1) Unzip key files sent by your OROS local dealer and copy it to a local directory (e.g. c:\temp\OROSkeys.)
2) Log to: www.myoros.com with your personal account and password.
3) Go to the following address :
http://www.myoros.com/modules.php?name=Contenu&pa=showpage&pid=34
4) Download the standard version, unzip the files
5) Click on setup.exe and follow the instructions.
6) When you are asked for the key files : click on browse and locate your keys (e.g. c:\temp\OROSkeys) then click on Ok
7) At the end of the installation, you must update the analyzer middleware and encrypt your dongle and hardware:
1. Connect your analyzer, and plug your dongle.
2. Launch NVGate environment (automatically launched at the end of NVGate installation)
3. In the Update tab click on encrypt dongle key
4. In the Update tab click on encrypt hardware key
5. In the Update tab click on update hardware key



NVGate 5.0 is now installed and the license keys have been updated. Contact your local dealer in case of any trouble.

IP address rules

Few rules have to be respected for a good connection.

To set the IP, your firewall must be deactivated, also only one hardware can be connected.

During the installation, the PC communicate with all networks working on it, then analyzer in version V4.10 and higher responds by giving its IP address, serial number and type of hardware. The creation of the IP address of the hardware is automatic. There is a communication between hardware and PC, the IP address is created on the PC network card where the hardware is connected.

- Used network board: address created on the network card of the PC
- Network IP address: address of the network where the Hardware is connected (this address may be automatically created)
- Analyzer infos: information concerning the type (OR34,35,&ldots;) and the serial number of the connected hardware
- Analyzer IP address: Analyzer IP address
- Connection status: the red cross indicates:
  - Analyzer is stopped
  - Connection problem or cable problem (not a cross cable)
  - Firewall activated (this function can only go through the Windows firewall)

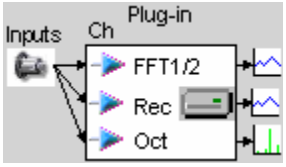
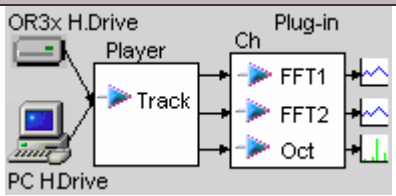
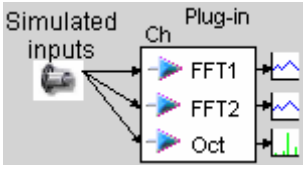
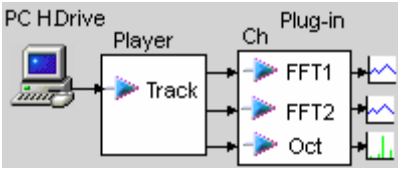
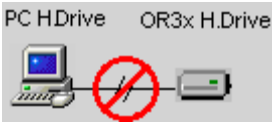
It is possible to change the IP address of the hardware, but it is necessary to use a compatible pair of IP address. There is no verification or protection for compatibility when the user is entering a new address IP. In that case the Hardware will not be detected and the status will be a red cross.

Chapter 3: Software environment

Operation modes

Modes

Here are the different Operating and Analysis modes you can use with OR3X/NVGate:

	Analysis mode	
Operating mode	**On-line**	**Post analysis**
Connected		
Office		
Stand-alone		N.A

You can see the current analysis mode and the operating mode in the Status bar on the bottom of the desktop:

Show or hide the Status bar from the **View** menu.

On-line

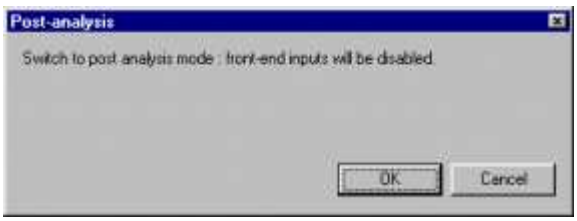
This mode is available with the Office, the Connected or Stand-alone Operating mode.

- The Connected operating mode is used to record the signal provided by the inputs and perform real-time analysis (FFT, 1/n Oct...).
- The Office operating mode means no Hardware is connected. It is most frequently used to set up and prepare projects and measurements.
- The Stand-alone operating mode is used to leave the OR3X hardware running on the measurement site and take away your laptop. See more details about the Stand-alone mode.

See more details...

Post analysis

This mode is available with the Office or the Connected Operating mode. It allows to compute the saved signals (compute FFT analysis on a recorded signal for example).
To use it, from the **Channels** menu select **Post-analysis**. The following window is displayed:



You should have at least one recorded signal file on your OR3X or PC Hard Drive before using the Post analysis mode.
The OR3X or the PC Hard Drive (according to your selection) sends the recorded signal through the Player module in order to display the signal on the screen.
The inputs are not accessible anymore, they are replaced by Tracks (find all the information about the recorded channels in the Player module).

Note: In the post analysis mode, the Front-end is not available anymore.

Tip: It is also possible to re-record a recorded signal file that either the OR3X or the PC Hard Drive sent through the Player module.

See more details...

Connected

This mode corresponds to the use of NVGate connected to an OR3X hardware through the Ethernet link. This configuration uses the hardware resources (DSPs and local hard disk) for On-Line analysis as well as Post-analysis.
The available options in connected mode are set up in the hardware license key. The dongle (USB or parallel) license key is not requested to operate this mode.

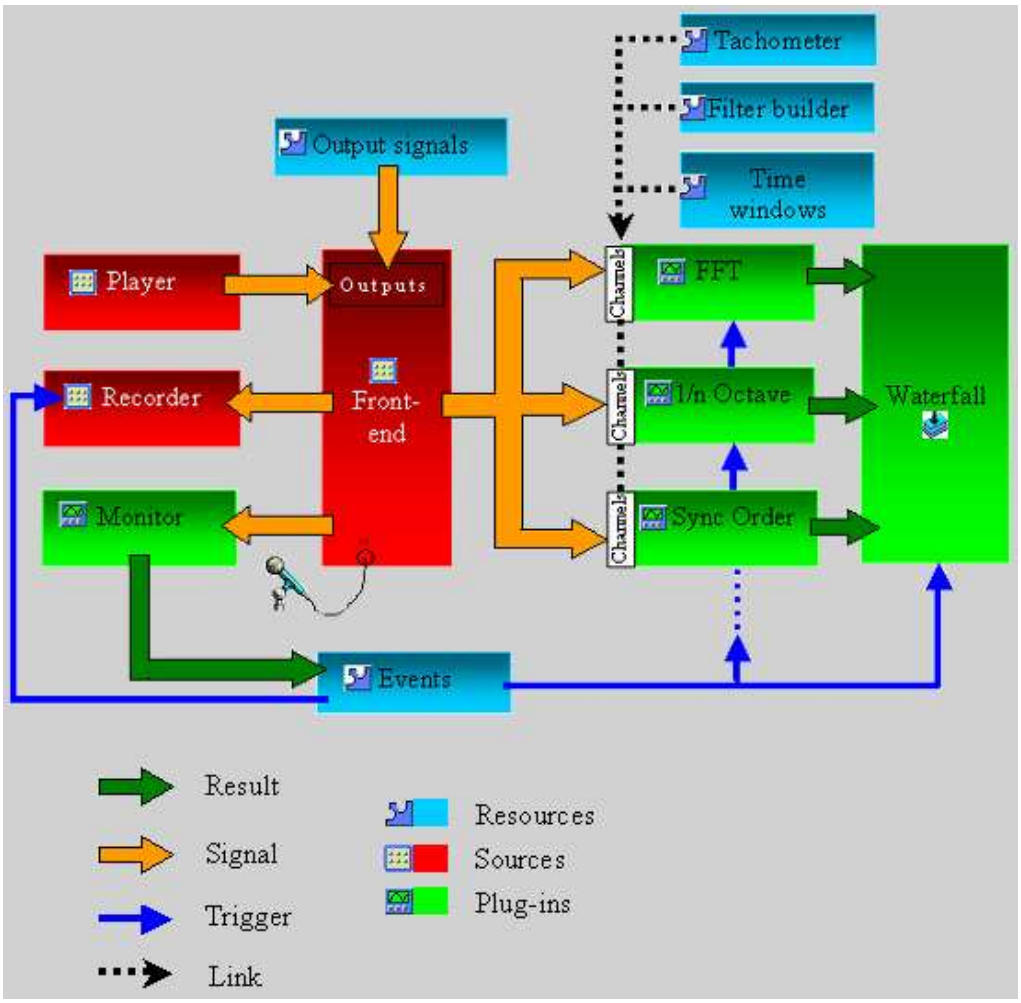
Office

The Office mode uses exclusively the PC resources (CPU, hard disk) for post-analysis and result exploitation, with no connection to OR3X hardware.
The available options in Office mode are set up in the dongle (USB or parallel). The dongle (USB or parallel) license key is requested to operate this mode.

Both modes (Connected and Office) can be run simultaneously on different PCs allowing independent use of each license key (for example: On-Line acquisition and Office post-analysis).

On-Line acquisition

The following synopsis presents the basic structure of the NVGate multi analyzer architecture for real-time acquisition and analysis. In this mode, OR3X processes parallel analysis with all plug-in analyzers running at the same time (FFT, 1/n Octave, SOA, Recorder&ldots;).
The Front-end inputs may be connected to any plug-in analyzer through the channels. There are up to 32 (8 for OR34/OR35 analyzer) channels on each plug-in analyzer. Any input plug-in analyzer channel combination is allowed.



The plug-in analyzers share general resources such as Filters, Events, Time windows, and Tachometers. These resources can be associated with any channel or plug-in analyzer with a centralized set of parameters: for example, by applying filter 1 to channel 1 to 8 of each plug-in analyzer and filter 2 to channels 9 to 16 you can modify filtering characteristics directly from the resource settings.

The Filter builder provides general purpose filters for low pass, high pass, pass band, stop band and integration functions that apply to the time domain signals. For example: distributing Input I to channel 1 to 3, will provide simultaneous analysis of acceleration, velocity and displacement of an accelerometer by applying Integrator filter to channel 2 and double Integrator

filter to channel 3.
The Time windows resources allow the modification of force and response weighting windows. For example: associating the force window with channel 1 and the Response window to the rest of channels allows the modification of response characteristics from a single set of parameters in the resource.

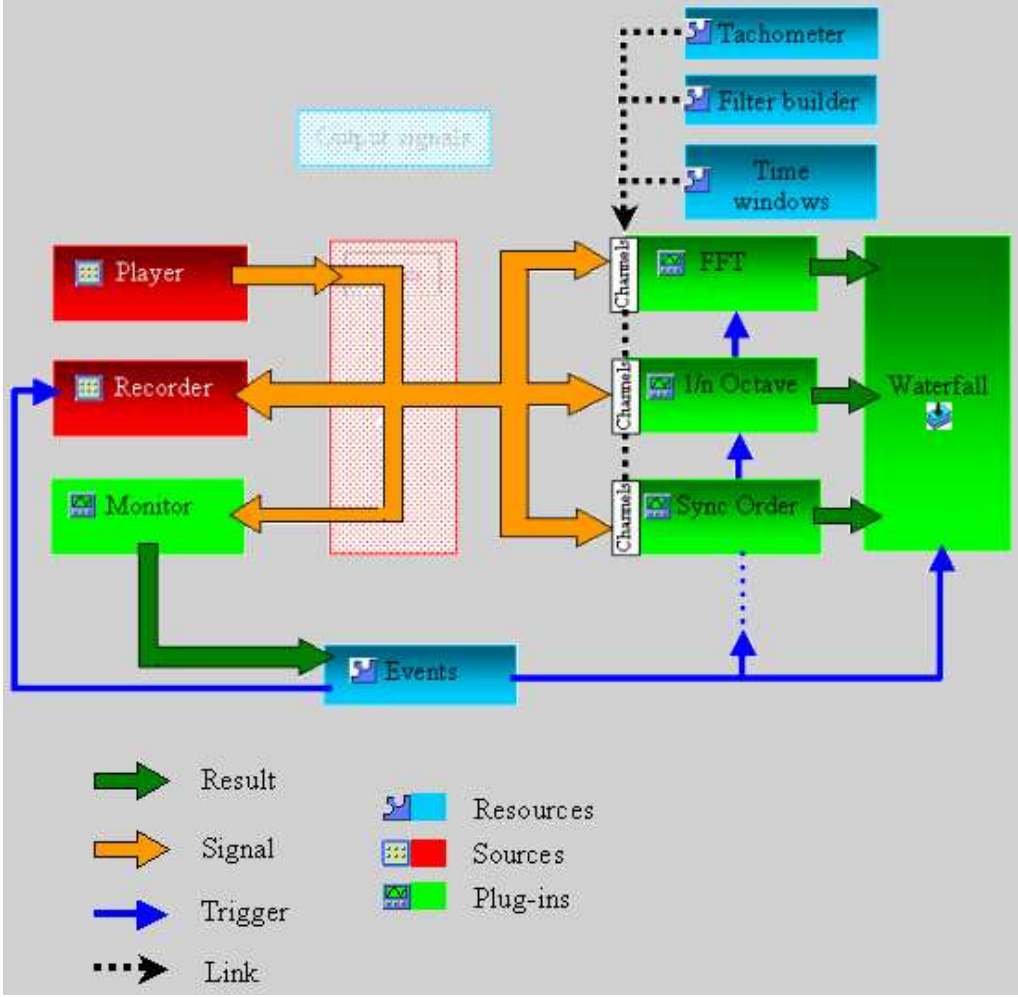
The Event Definition resource is used to create an event that can start, stop and trigger plug-in analyzers acquisition. These events can be based on different criteria (DC level, RPM, time&ldots;) as well as on results computed by the Monitor (Kurtosis, RMS, DC&ldots;). For example: an event based on an input RMS level can be used to start and/or stop the FFT averaging and the Recorder simultaneously by connecting the event to the plug-in analyzer trigger sub-module.

The Waterfall module collects and synchronizes the results computed and provided by the different plug-in analyzers. These results are displayed in segments and can be sorted in relation to selectable references (Time, RPM, DC&ldots;). For example: while computing orders, it is possible to track these values by adding them and the Tachometer (the reference) to the Waterfall. You will then be able to display order profiles and switch the reference between time or RPM. It is then possible to add any new result (value or spectrum) or any reference (DC input, 2nd tach) to the Waterfall.

Post analysis

In the Post analysis mode, the Front-end and Output signals are no longer available. The only source able to provide recorded time signals is the player.
Post analysis features exactly the same functions as the On-Line mode replacing the inputs by the Player tracks. The Player tracks are then connectable to the plug-in analyzer channels and can be used by the resources.
Post analysis processing can be performed on the DSP (Connected mode, only for OR36/OR38) as well as on the PC CPU (Office mode).

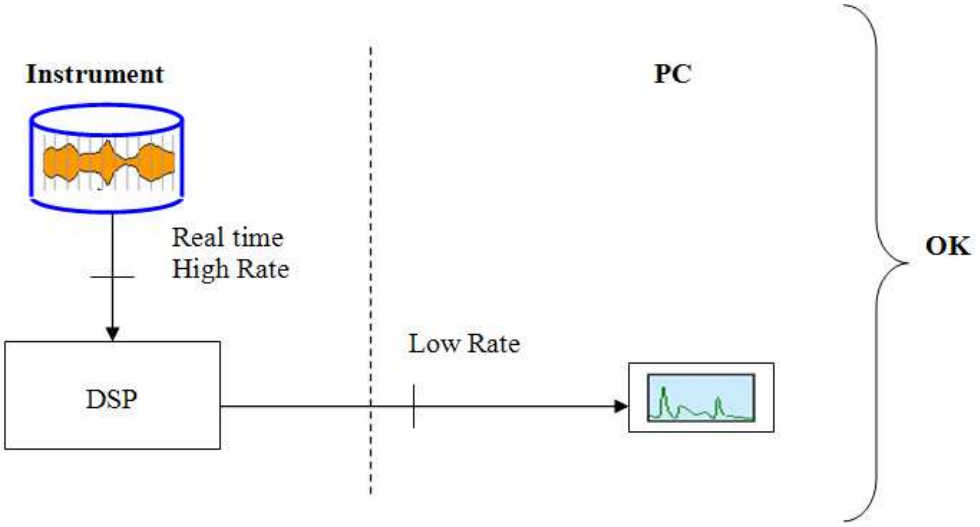
In addition to the Post analysis, it is possible to split the recorded signal files (channels and duration) by re-recording selected channels and time segments in a new signal file.
Post analysis processing time depends directly on the computation load. It may be faster or slower than the real-time analysis.
Using the Post analysis accommodates larger requests in terms of computation power. This cannot be achieved in real-time analysis.



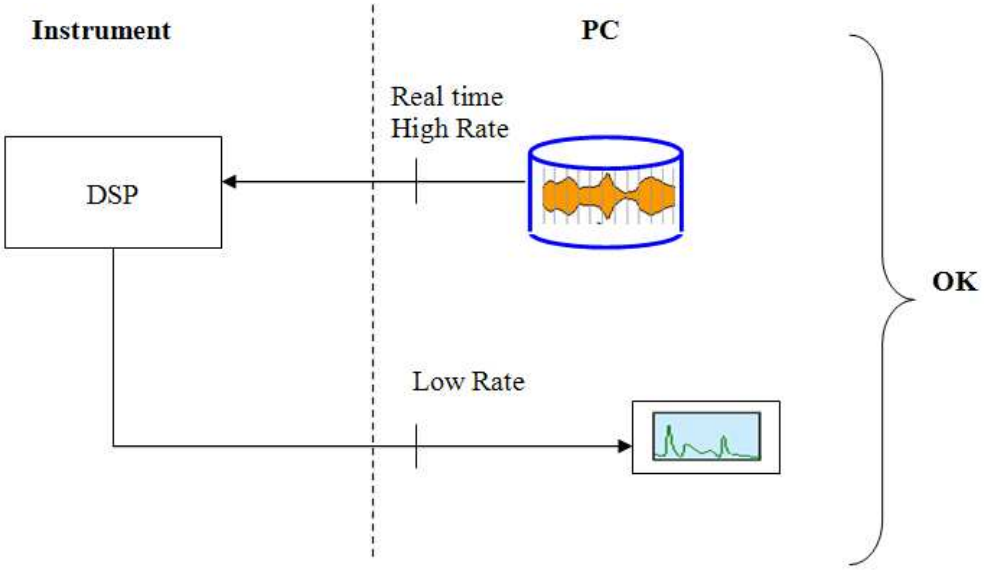
If the time signal is recorded on the PC or on the Mobi-disc™, it is highly recommended to be in office mode (hardware not connected) in order to avoid low performances and malfunctions.

Example

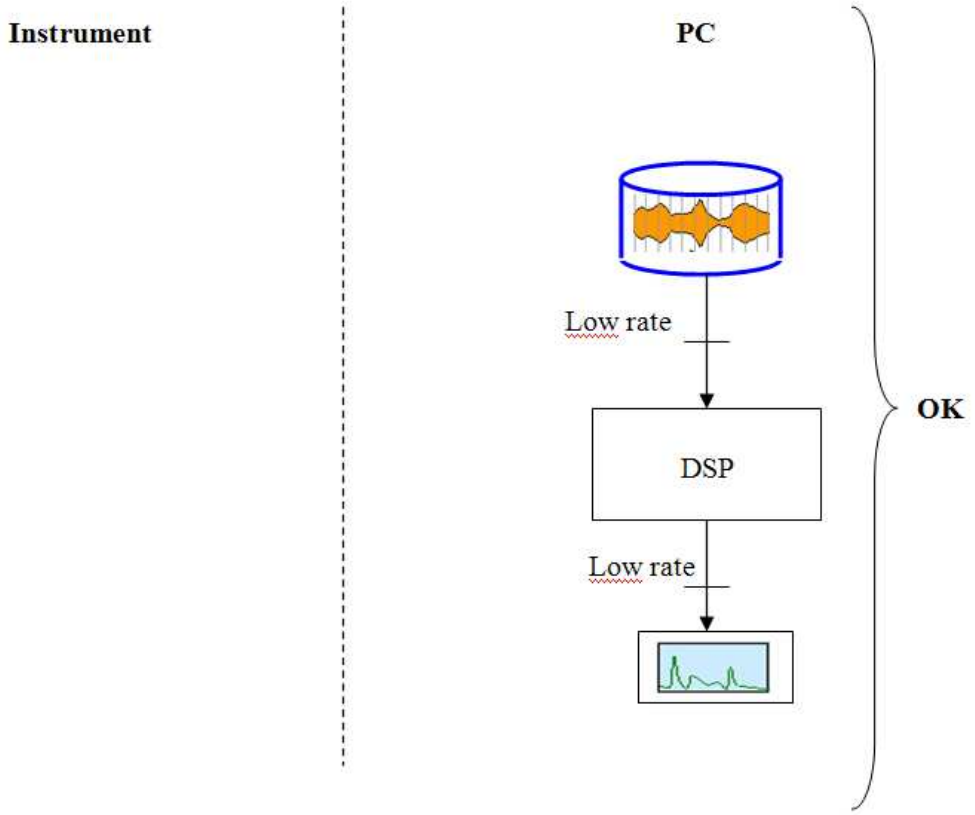
Connected mode, signal on the analyzer



Connected mode, signal file on the PC



Office mode, signal file on the PC



Stand-alone

The Stand-alone mode allows your OR35/OR36/OR38 analyzer to collect data on its own without any PC connected to it. It is useful for long term monitoring and survey. You can then leave the OR3X hardware on the measurement site and take away your laptop.
You can also set up several OR3X hardware from the same PC.

Running the Stand alone mode

1. First, set up your analyzer with NVGate® in the connected mode:

2. Then put the analyzer in the stand-alone mode using the file menu. Your PC is now free for other uses, including NVGate® for post-analysis:

When you click **Stand alone**, NVGate® shuts down. You can now disconnect the OR3X hardware from the PC and put it on the measurement site.

Note: When you will start again NVGate® (with no hardware connected or with another hardware), this icon appears in the Project Manager to show you that the corresponding project was set-up for a Stand-alone session. So it is recommended that you don't modify this project to avoid any errors during the re-connection. You can modify the project when the corresponding hardware is re-connected.

3. Your OR3x is now running in stand-alone mode. You can control it through the Front panel for start/stop and manual trigger. During those operations the analyzer is able to record on its local hard disk, perform linear averaging, or even compute profiles (DC, RPM, Order):

Note: If you are working with an OR35 hardware, you may click Run from NVGate® before you select the Stand alone mode.

Real-time signal

4. When the acquisition is finished, you have just to re-connect the hardware to the PC and start NVGate®. The project that corresponds to the hardware you have connected will automatically open. All the predefined records and analysis results are available for saving:

Retrieving Saved results

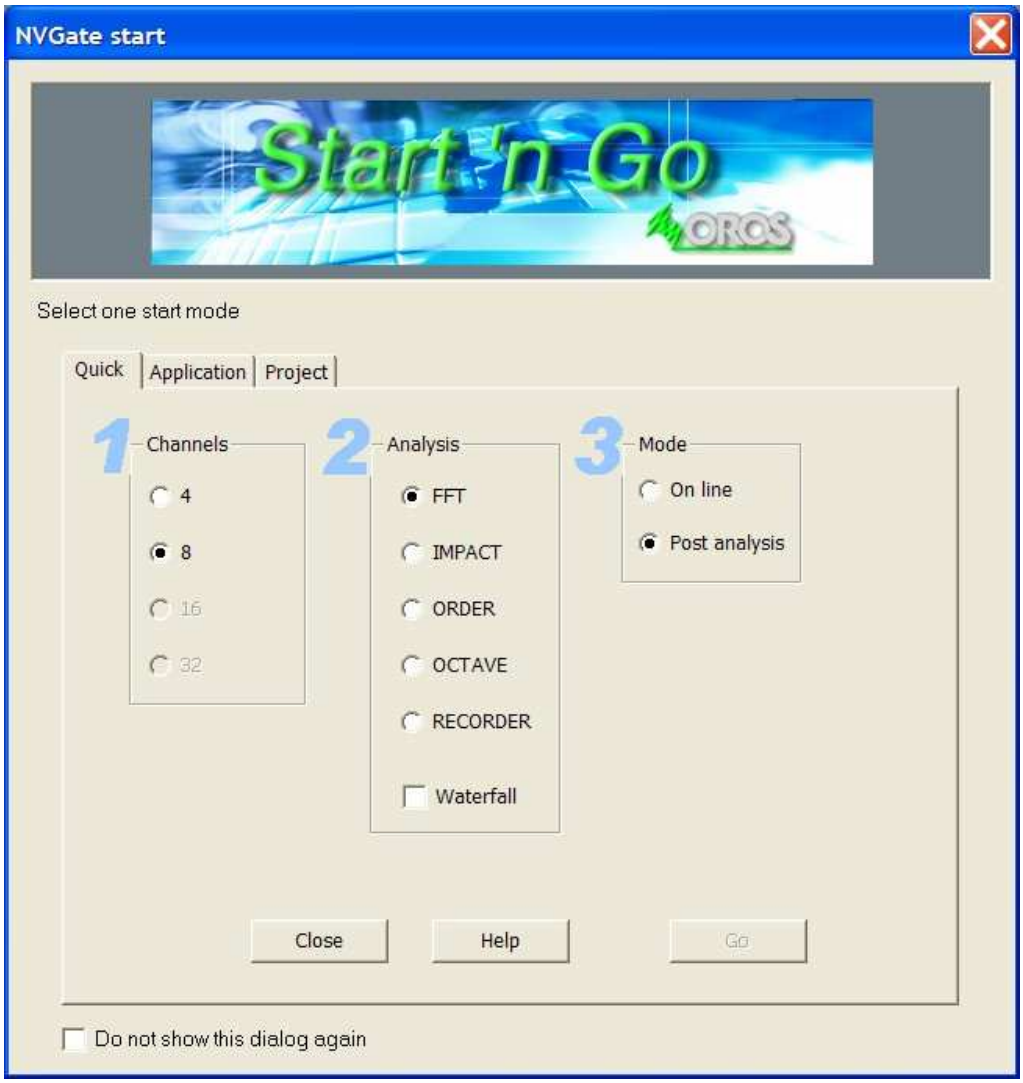
Stand alone features

- You can run several Stand alone sessions at the same time with several OR3X hardware.
- You cannot use the macro for a Stand alone session.
- It is not possible to use waterfall in a Stand alone session.
- It is not possible to record time signals with OR35 analyzer.
- When you want to run an OR34/OR35 hardware in the Stand alone mode, you must click Run from NVGate® before. OR34 and OR35 hardware have no front panel control buttons.

Start'n Go

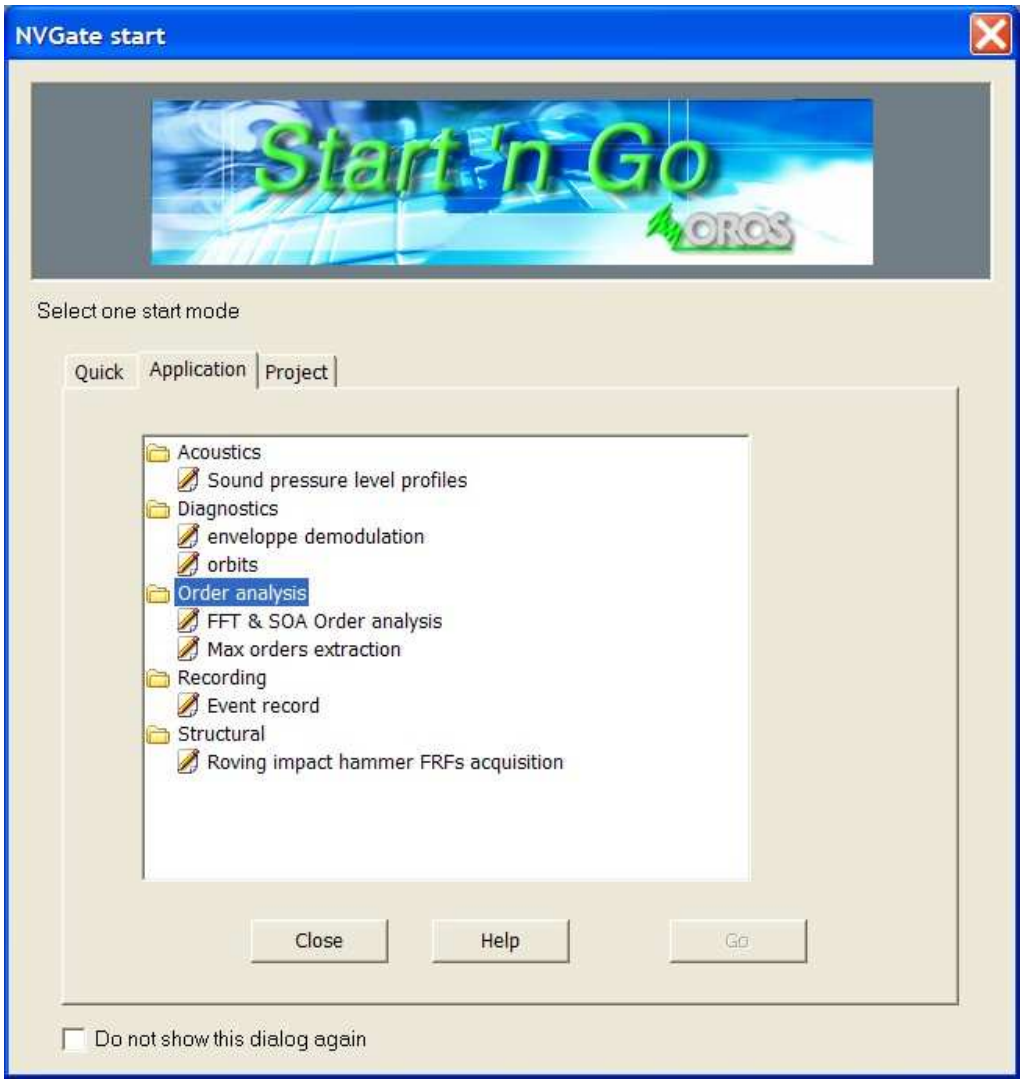
When you launch NVGate, the Start'n Go window is automatically opened. Start'n Go function will help you to start a project.
There are 3 different ways to start:

Quick



Select only the number of channels, the plug-in used and the analyzer mode.
A typical project is automatically loaded depending on this selection.
This function simplifies the project building operation while allowing modifications to personalize the project.

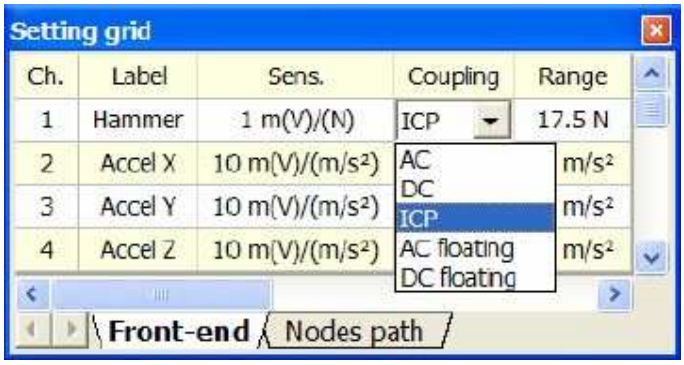
Application



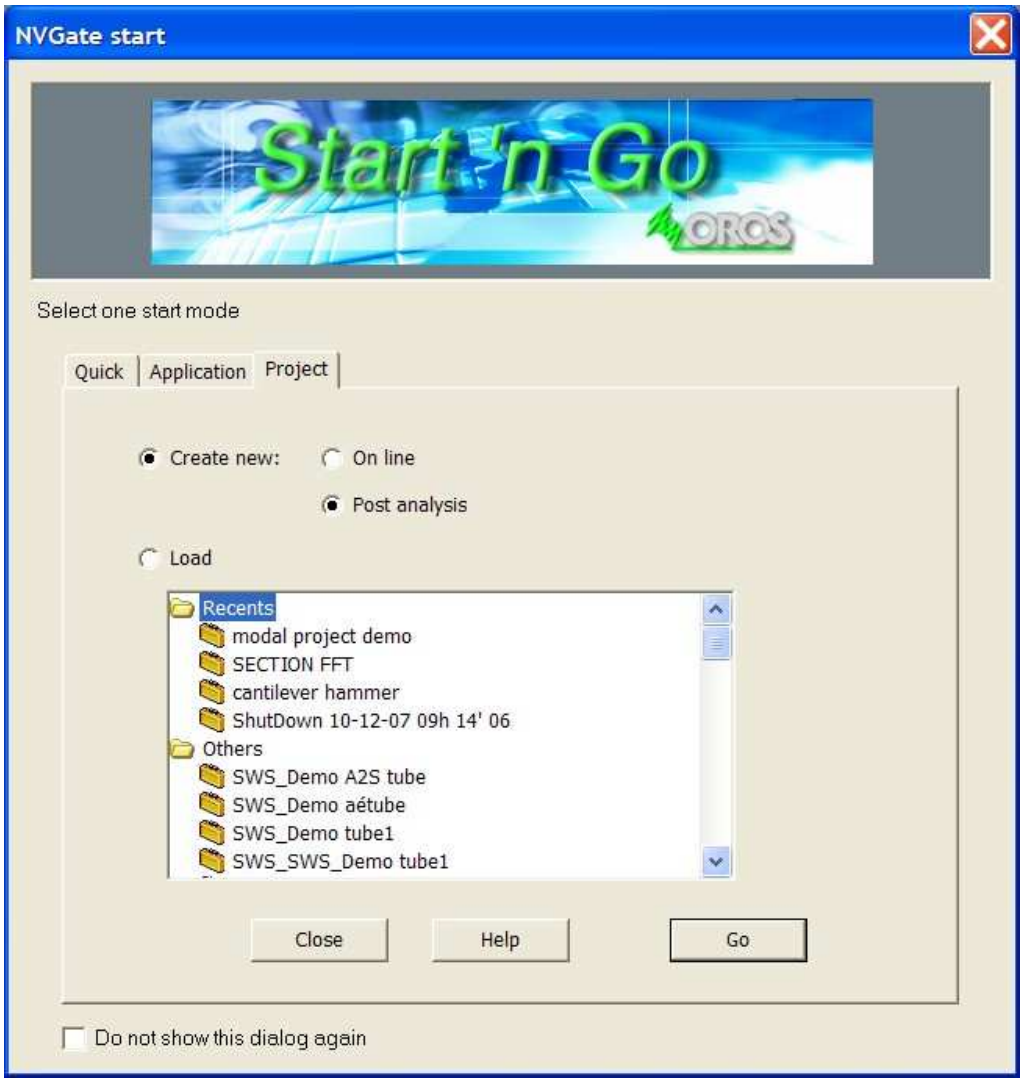
In this case, you can choose a kind of application: Diagnostics, order analysis, recording...
A project will then be created depending on your choice. Modifications are still possible.

Note: Every projects automatically built from 'Quick' or 'Application' are a base for a kind of tests and can be modified to be completely adapted to your application.

With each predefined setup (quick application), a grid of settings is available. This grid allows direct modifications of presented settings.



Project

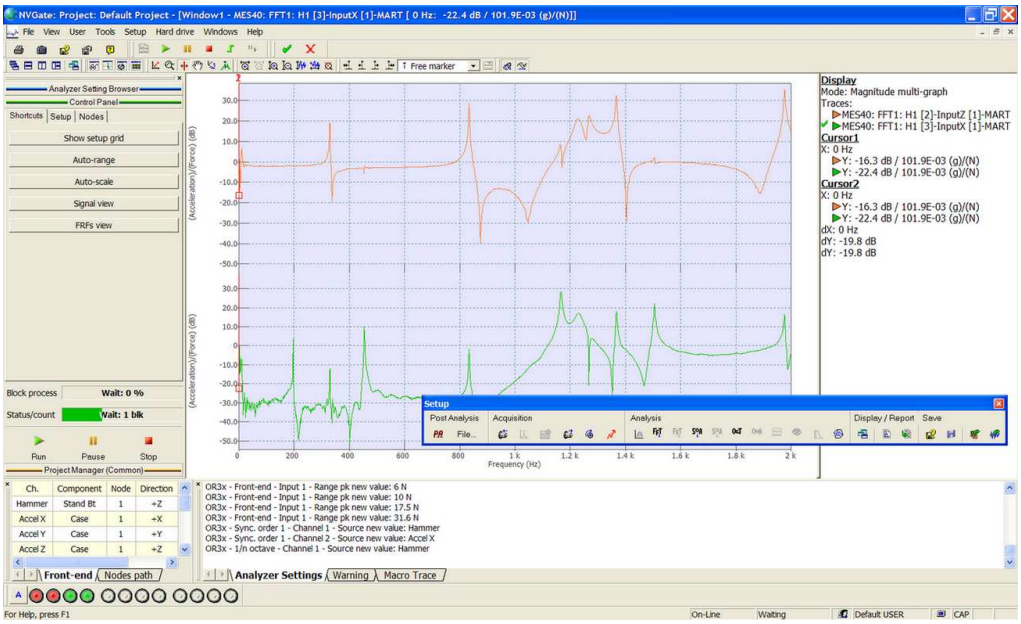


The last option allows to select directly an existing project. All the projects are listed and the recent ones are pointed up.

Interface description

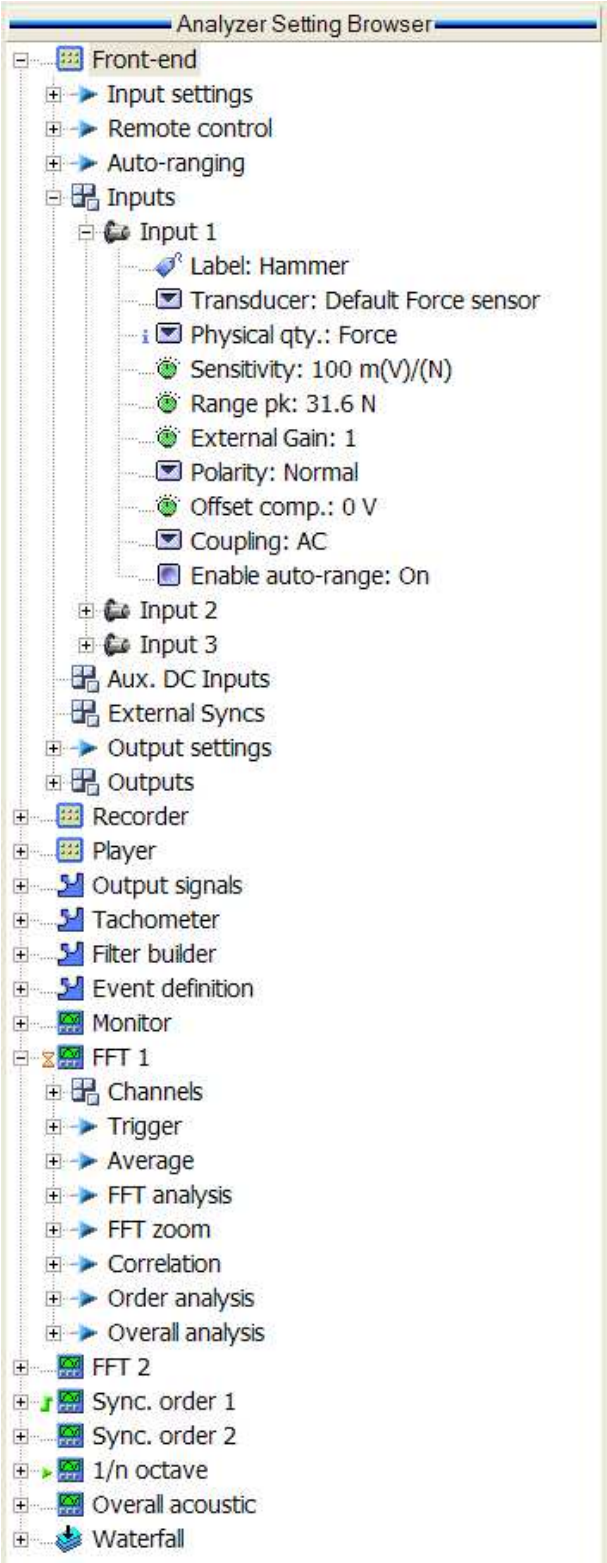
See below the main interface of NVGate.
Click on the link below or directly on the picture to have more information on the different parts:

- Analyzer Setting Browser
- Control panel
- Project manager
- Log window
- Active window
- Toolbar
- Settings grid



Analyzer Setting Browser (ASB)

The analyzer is controlled through a tree of settings called the Analyzer Setting Browser (ASB). This tree consists of a set of modules corresponding to the main functions:

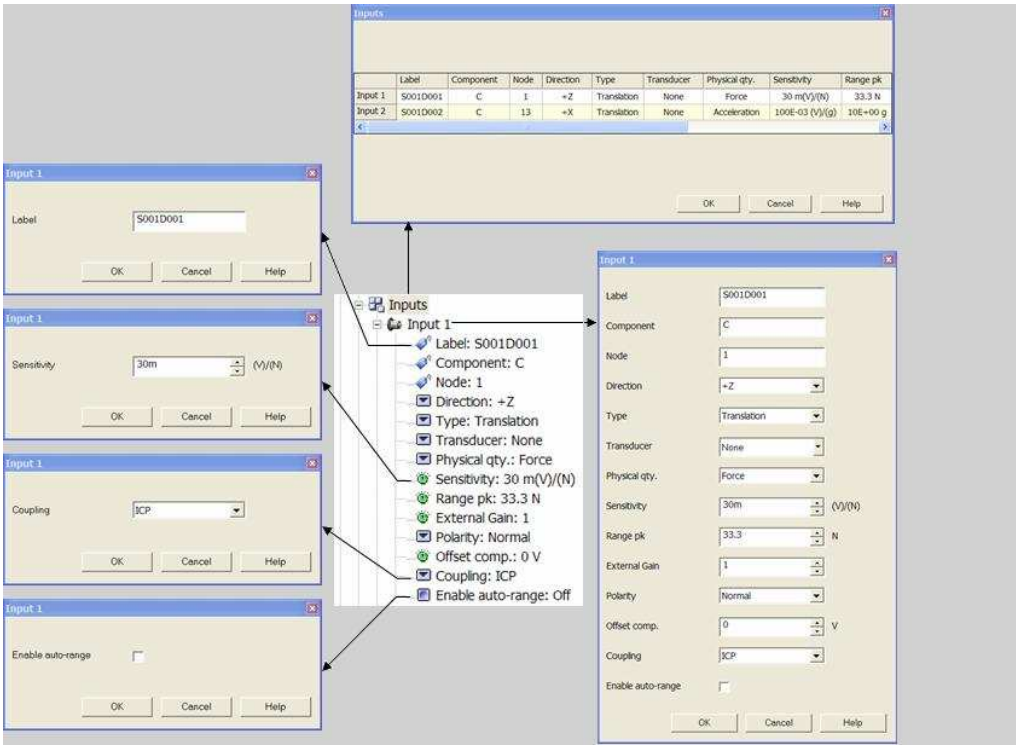


Note: The "information" icon may be found next to a setting. It means you cannot change the setting value.

Tip: When using the right click on any item from the ASB, you can copy the value(s) contained in the selection (it can be a module, collection, sub-module or just a setting) and paste it (them) into an other item which contains compatible value(s). It is also possible to drag and drop a scalar type setting ( ) from the ASB to a displayed window (for example the center frequency of the FFT Zoom).

Ways of editing a setting

Right click on an item and select **Properties** (see the following examples). You can check your setup and also change the settings in real time by double clicking on the cell you want to change. The drop-down menu or the mouse wheel can also be used.
A setting can be set to its default value by right clicking on it and by selecting **Set to default**.



Analyzer states

In the Analyzer Setting Browser, you can find the analyzer states icons. They was made to show you visually the state of each plug-in analyzer at the same time, follow the icon descriptions:

The analyzer plug-in is stopped

The analyzer plug-in is running

The analyzer plug-in is paused

The analyzer plug-in is recording

The analyzer plug-in is waiting for a trigger: defined Event or Manual ( )

The analyzer plug-in is waiting for the user answer: Accept ( ) or Reject ( ) the signal

The plug-in analyzer is failed (the analyzer is not able to do the data acquisition anymore, a warning message will appear)

There are three types of **modules**:

- The **sources** that acquire, store and play back signals
- The **resources** that provide events, filters, weighting windows, and tachometers to the plug-in analyzers
- The **plug-in** analyzers that process independent analysis modes
- The **Waterfall** that synchronizes the results computed by the plug-ins

The modules contain different functions located on **sub-modules**.
A **collection** is a box that contains a group of sub-modules of the same type (inputs, filters, channels, etc.). The sub-modules of a collection can be active (visible) or inactive (hidden). Right click on it to add/remove channels.

The **settings** are inside the sub-modules. The ASB has a maximum depth of 3 levels: module, sub-module, and settings. In the case of multiple similar sub-modules (inputs for example),
The user sets his analyzer simply by changing the settings. The settings are grouped in families:

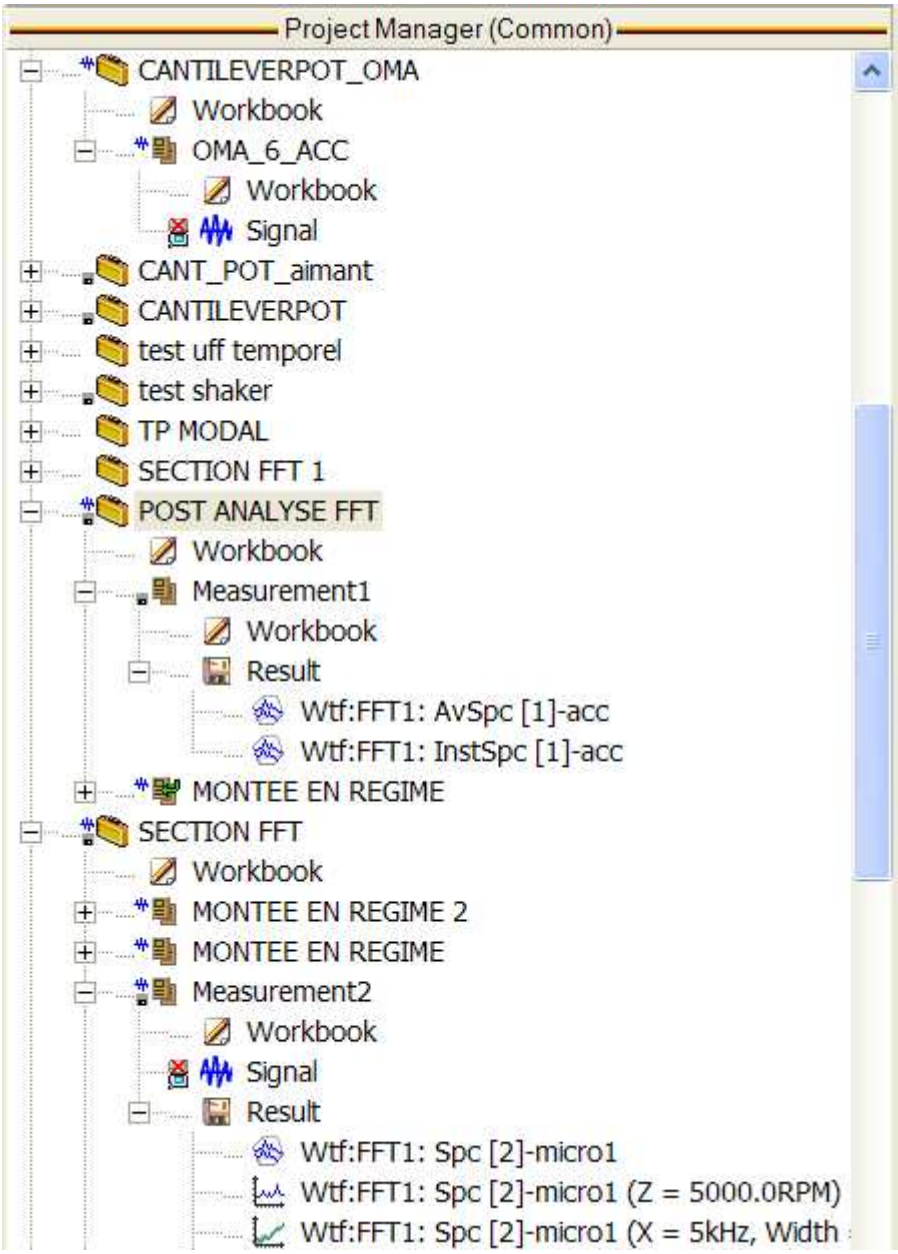
- The scalar settings are every numerical value which can be entered from the keyboard.
- The label settings are every character string which can be entered from the keyboard.
- The enumerated settings are a selection of an item from a list (items are

character strings or numbers).

- Table is a group of values.
- Matrix are selection of a matrix intersection.
- Boolean allows to activate or deactivate functions (inputs, auto calibration, zoom...).

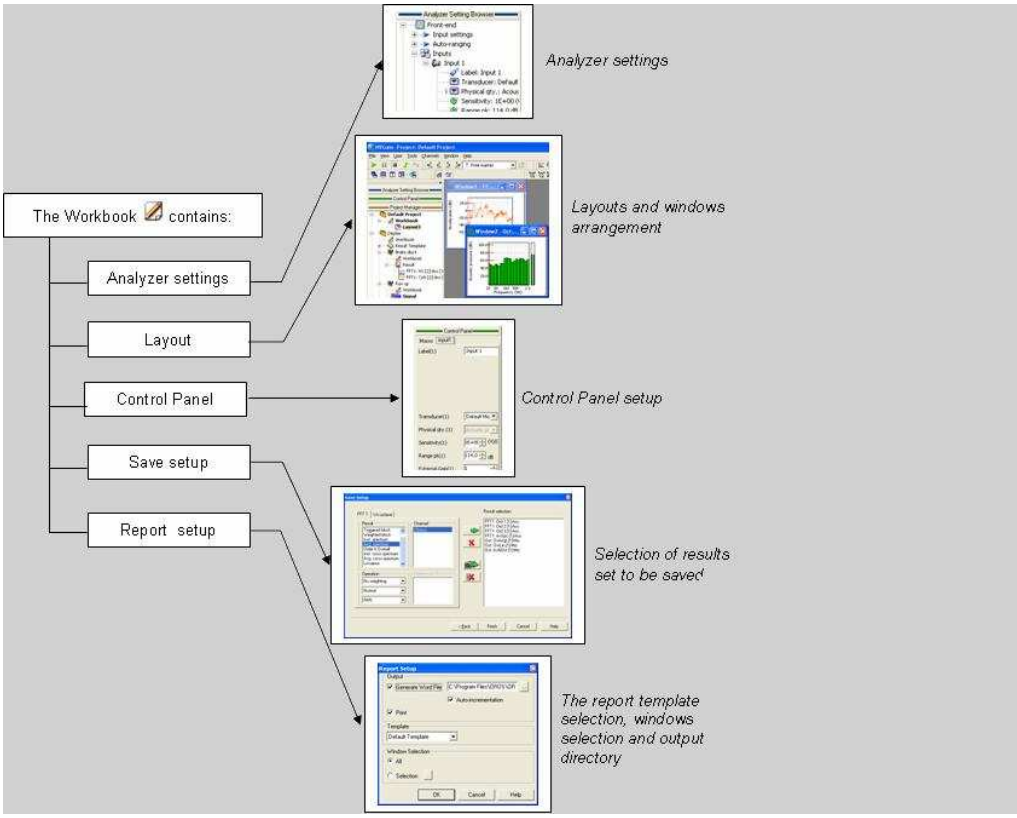
Project Manager

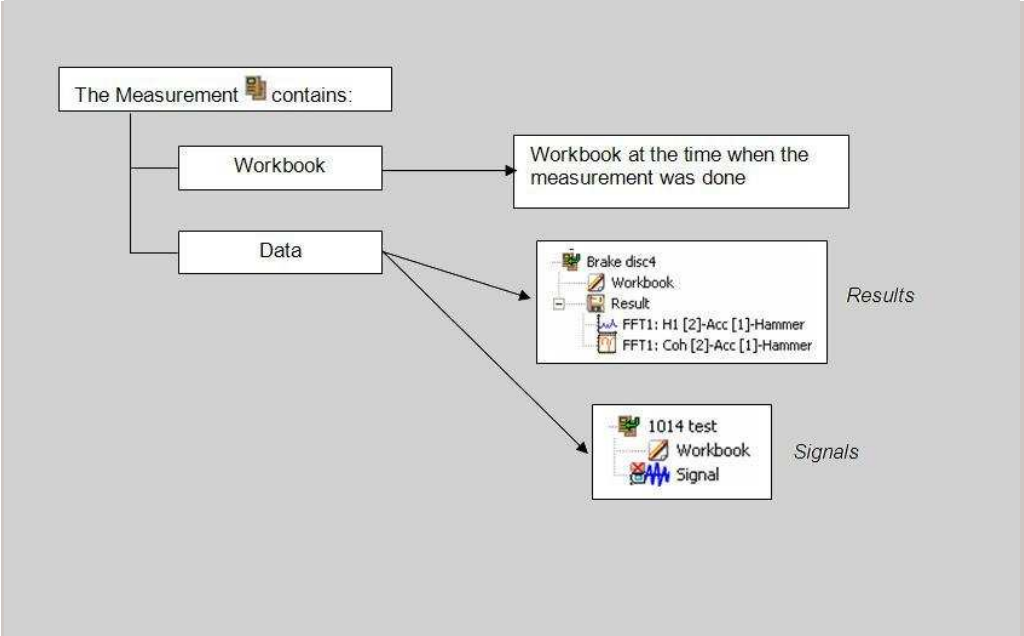
The Project Manager is used to store all your projects, results, and measurements... There are several display options that enable you to show or hide certain items.

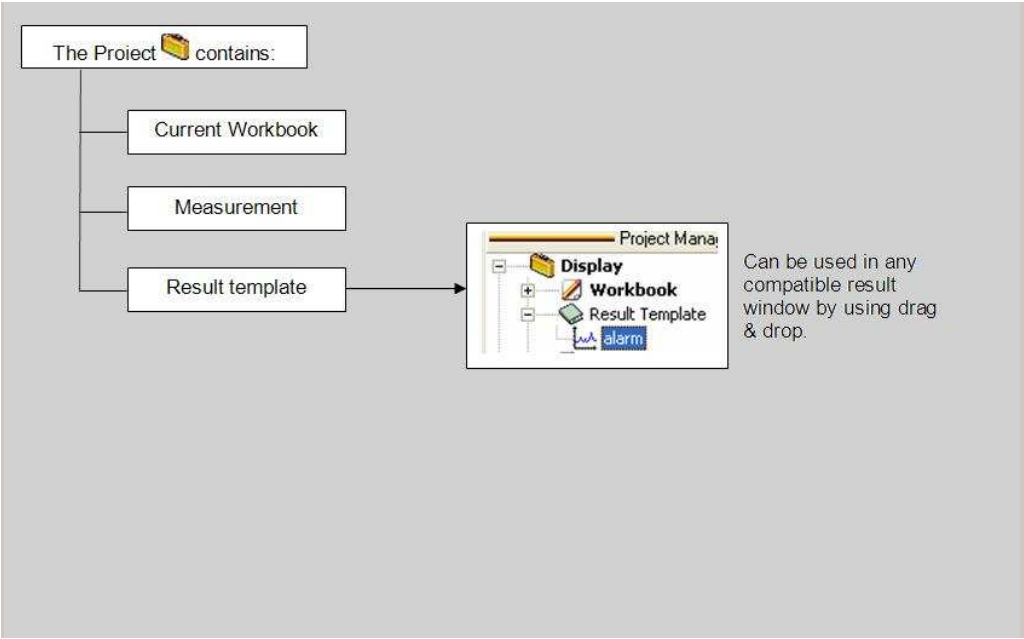


Tip: you can drag and drop any type of result from the Project Manager to the windows top to display the result window.

The Project Manager is made up of 3 main parts; the **Workbook**, the **Project** and the **Measurement**.


The Workbook contains:
Analyzer settings
Layout
Control Panel
Save setup
Report setup
Analyzer settings
Layouts and windows arrangement
Control Panel setup
Selection of results set to be saved
The report template selection, windows selection and output directory


The Measurement contains:
Workbook
Data
Workbook at the time when the measurement was done
Brake disc4
Workbook
Result
FFT1: H1 [2]-Acc [1]-Hammer
FFT1: Coh [2]-Acc [1]-Hammer
Results
1014 test
Workbook
Signal
Signals



In front of the project name POST ANALYSE FFT

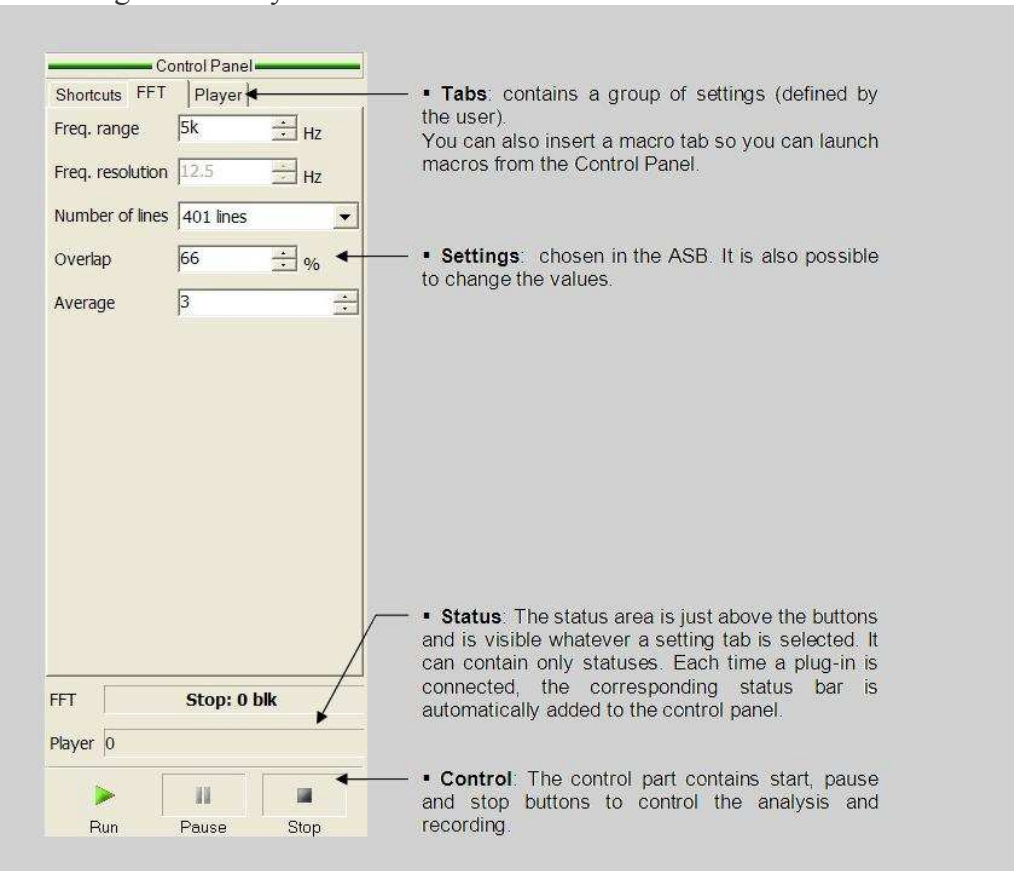
The item indicates if a project/measurement contains a signal file.

The item indicates if a project/measurement contains results.

Control panel

Control panel overview

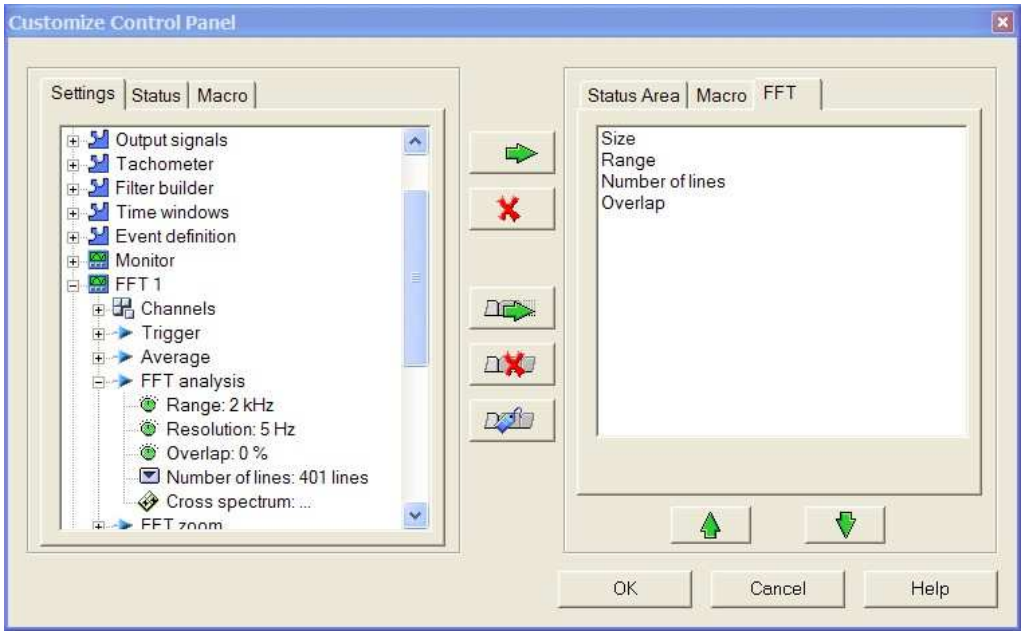
The configurable Workspace area provides quick access to settings and/or ASB status. It looks similar to the ”Favorites" in Internet Explorer. The settings can be classified into customizable tabs. The Control Panel also contains general analyzer command buttons.



Tip: a setting from the ASB can be added to the Control Panel by right clicking the setting in the ASB and select Add to Control Panel.

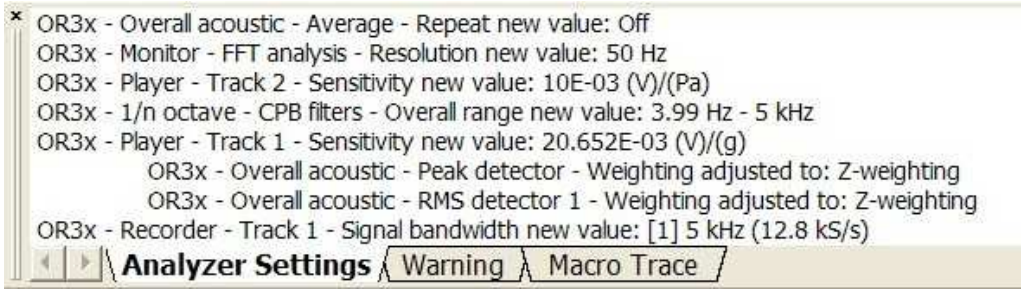
Control panel customization

The control panel is a part of the Workspace. It is used to control the measurement process. It is user-customizable and saved in the current project. Customization can be reached by right clicking on the Control Panel or from the **Tools** menu and **Customize Control Panel**...
The customize Control Panel is very useful when you create a restricted user. A restricted user has no access to the ASB and Project Manager, he can only access the Control Panel. The administrator may use "Customize Control Panel" function to set up a restricted user project.



Tip: Use drag and drop to add a setting from a place to another. The destination tab must be compatible with the setting you try to add to..

Log window



Analyzer Settings

The Log Window shows all the changes made to the modules (Front-end, Filter builder, Monitor, FFT x, etc.) in the ASB.
Some setting values need to be adjusted when another setting value is modified. So when you change a setting value in the ASB (new value: ...) which may affect another setting value, the Log window marks the changes made by the analyzer (adjusted to: ...).
Right click on this area:

- Select Copy (to paste a setup in a text document for example)
- Clear the display in the Analyzer Settings tab.

Warning

The Warning tab shows any unauthorized operations you have carried out or errors reported by NVGate.
Right click on this area:

- Select Copy (to paste a setup in a text document for example)
- Clear the display in the Warning tab.

Macro Trace

The Macro Trace tab shows the operations (the start and stop macro execution, errors) made in the macro mode.
Warnings do not stop macro execution but major errors will.
Right click on this area:

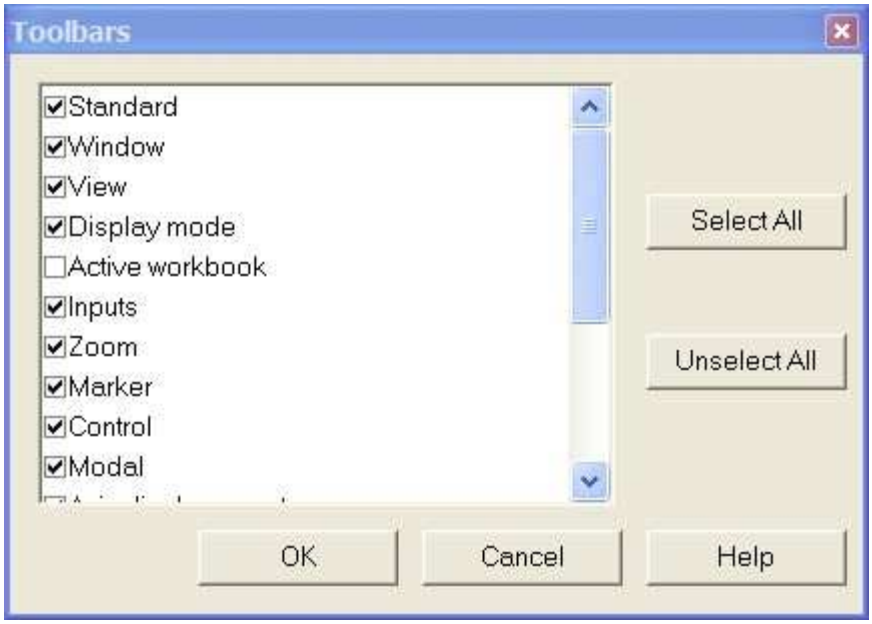
- Clear the display in the Macro Trace tab.

Tip: double clicking on the error icon in the Macro Trace tab will open the Macro Editor on the failed command.

From the toolbar click on to show/hide the Log window (Analyzer settings, Warning and Macro Trace information).
For example, if one microphone is connected to 2 inputs, the Log Window will then open if it was not displayed.
The Log Window is a dockable item.

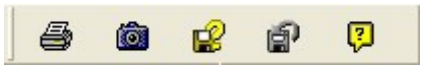
Toolbar

To display or mask the toolbars, from the **View** menu, select **Toolbars**. The following window appears:



Main toolbar description

Standard



- **Print a report using the user layout**, this function prints a report you designed from the Template editor. The report to be printed is the one selected in the Report setup.
- **Print the active graphic window**, this function prints the graph of the active window with information such as the project name, user name and the trace type you are about to print.
- **Save Setup**, this function opens the Save setup window so you can select which result you want to save.
- **Save all available results**, this function allows to save results after you stop the analysis (in case you forgot to specify which result could have been saved). Before to press this button, right click on the window containing the result you want to save and select "Add to result selection".
- **Help topics** , this function displays the main On-line Help topics.

Window

- **Cascade**, windows are automatically arranged in cascade.
- **Tile horizontally**, windows are automatically tiled in a horizontal arrangement.
- **Tile vertically**, windows are automatically tiled in a vertical arrangement.
- **Automatic tile**, windows are automatically arranged according to the type of windows to be tiled (viewmeter, waterfall, profile...).
- **Add/Remove window**, this is a shortcut to the "Add/Remove..." function available from the Window menu.

View

- **Log window**, this function shows/hides the Log window (Analyzer settings, Warning and Macro trace information).
- **Analyzer status viewer**, this function shows/hides the Analyzer states (Analyzer name, Item name and its value).
- **Workspace**, this function shows/hides the Workspace (Analyzer Setting Browser, Control Panel and Project Manager).
- **Setting grid,** this function shows/hides the setting grid.

Zoom

- **Zoom in X,** horizontal zoom in.
- **Zoom out X,** horizontal zoom out.
- **Zoom in Y,** vertical zoom in.
- **Zoom out Y,** vertical zoom out.
- **Auto-scale on all Y axis,** auto-scale on all active traces.
- **Auto-scale X/Z on all,** X and Z axis auto-scale on all active traces.
- **Zoom reset**, Zoom 100%.

Control

- **Run**, is used when the connections and settings have been created. Press the play button to display traces or to start recording.
- **Pause**, suspends analysis and recording.
- **Stop**, stops analysis and recording.
- **Manual trigger**, use it as an event to start or stop a measurement.
- **Play by steps**, *available only in post-analysis, when the file is played by step*. The command starts a new step or pauses the current step. It works independently of the Run and Pause commands.
- **Recorder arming**, arms the recorder.

Modal

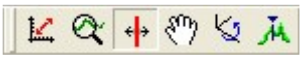
This toolbar is designed for modal analysis. It is used to accept or reject results or to make a manual trigger.

- **Accept**, accepts the measurement.
- **Reject**, rejects the measurement.

Specialized modes

- Rotating mode, mode designed for any rotating machinery. The tach plug-in and the order tracking settings are available.
- Structural mode, mode designed for modal analysis. Used for FRF calculation and various settings, as well as the modal analysis toolbar.

Display mode



- **Scale mode,** a new pointer appears in the window area. With the mouse wheel you can scroll the graph along the X or Y axis. Keep the SHIFT button

pressed and use the mouse wheel to zoom. You can also use the CRTL button to make a smaller move. It is possible to combine both keys.

When the arrow mode is active, double click on the axis to see their properties.

- **Zoom mode**, a magnifying glass appears in the graph area. Keep the left button of the mouse pressed and select the area you want to zoom in.
- **Cursor mode,** is used to move the cursor. By default, the cursor is always displayed on the 0 of the X axis. Select it and use the mouse wheel to move it or drag and drop it. It is possible to link several different window cursors in order to move them simultaneously (to do that, right click on all the cursors you want to link and select link).
- **Move mode** is used to drag and move the entire graph by pressing the left button of the mouse.
- **Rotate the reference** is used for rotations (only available in the 3D mode) by clicking on the 3D graph.
- **Marker mode** is used to set markers. Once you have clicked on this icon, double click on the graph where you want to add the marker.

Setup toolbar

The setup steps of the analyzer are reachable directly with this toolbars in order to guide step by step through main setting of the measurement.
This toolbar is a setup wizard, that is to say that steps are scanned using the Next/Previous buttons available in each dialog.
All different steps are grayed until they become available. At any time during a measurement, a dialog box can be called by clicking the relevant button in the setup toolbar.

This toolbar is divided in 5 sections

Post Analysis

- **Post Analysis**, selection of the analysis mode (post-analysis or on-line).
- **Select record,** to load directly a file in the player.

Acquisition

- **Channels connection,**
- **Calibration,**
- **Player track connection,**
- **Output connection,**
- **Tachometer connection,**
- **Event connection**

Analysis

- **Filters connections,** filters affectation and settings.
- **FFT plug-in,** FFT analysis settings.
- **SOA plug-in,** SOA analysis settings.
- **Octave plug-in,** 1/N octave settings.
- **Overall acoustic,** Overall acoustic level settings.
- **Recorder,** Recorder settings.
- **Monitor,** Monitor settings.
- **Weighted window,** Weighted window settings.
- **Waterfall,** waterfall connections and settings.

Display / Report

- **Create a new window,** add/Remove layouts windows, traces.
- **Report manager,** report template management and edition.
- **Report setup.**

Save

- **Save setup,** results selection to save.
- **Save project,** save all current settings in a project.
- **Export results,** export result to TXT, UFF, Matlab, SDF.
- **Export file,** export signal to TXT, UFF, Matlab, Wav.

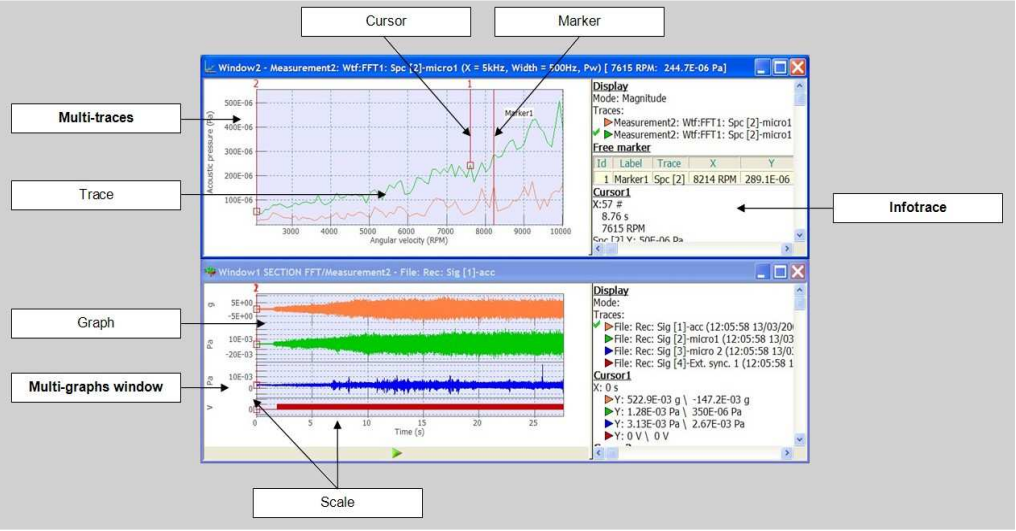
Inputs

- **Auto range**, performs an auto range on the active inputs. It is a shortcut to the action button in the Analyzer Setting Browser Front-end / Auto-ranging / Auto range.
- The inouts status is indicated by colors. The colors meaning is defined in the table below:

	The input is not active.
	The input is under loaded. Signal is weak.
	Normal.
	An overload has been detected, the signal on the input is correct.
	The input is overloaded.

Display

Different windows which contain one or several graphs or traces can be created. You can also add or remove some information in the Infotrace using the Infotrace properties. Manage all the display settings using the right click on almost any area. For each result type there are different available options using the right click.



Multi-graphs window

A graph is the area where traces are displayed.
A multi-graph window contains several graphs which display signals from the same source or plug-in analyzer. The graphs can be displayed with different physical quantities.

Multi-traces

You can display several traces in the same graph. You can also display in multi-trace 3D mode.
Multi-traces display is not available for Waterfall displays.

The **trace** represents a signal or a result.

Cursor

Displayed when cursor mode is selected: Used to move the cursor by left clicking and dragging.

Click on from the display mode toolbar in order to activate the cursor mode.

Marker

Select a type of marker from the marker toolbar and click on in order to activate the marker mode.
To add a marker on a graph, double click on the graph. Obtain information about a trace by using the marker table.

Scale

Displays the magnitude and the grid of the graph

Infotrace

Displays information about the graph:

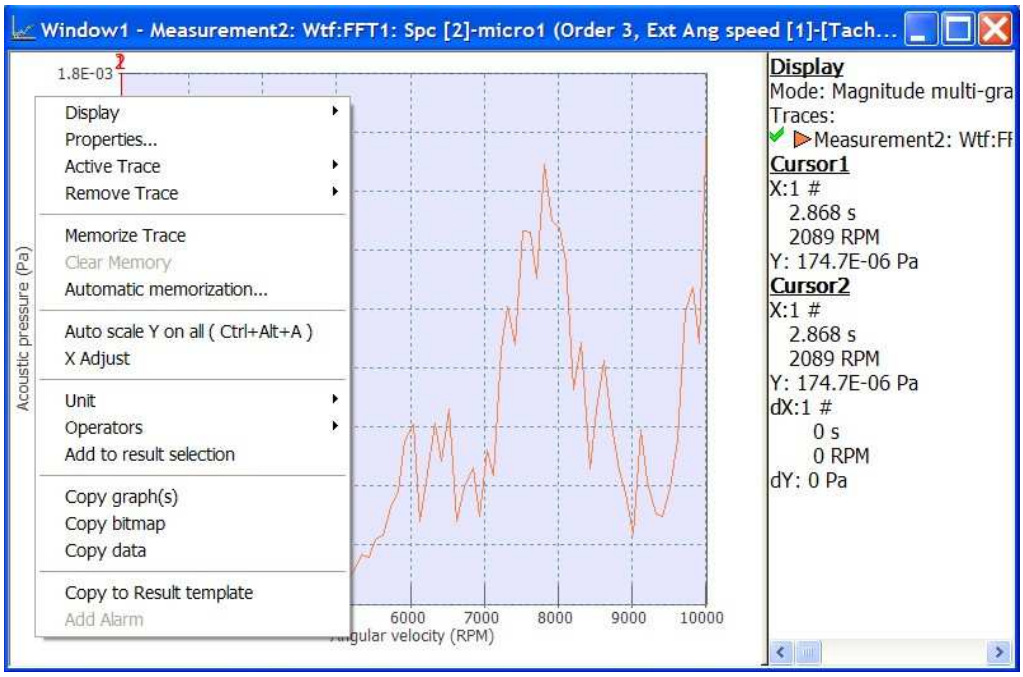
- **Graph mode**: magnitude, real...
- **List of traces with their states**: available or not / displayed in all areas or not / not displayed. You can move into the infotrace to select a specified trace and to follow cursor's value of this trace. Memorized traces are also displayed in the list (Memx)
- **Overall levels**: Displays the RMS level in 2D windows containing spectra or time signals.
- **Cursor information**: x, y, order, dx, dy if available
- **Marker arrays**

Using arrays of you keyboard you can select the active trace in the infotrace

Contextual menus

- Right click on the graph:

- Right click on the window:



- Right click on the infotrace:

Window1 - Measurement2: Wtf:FFT1: Spc [2]-micro1 (Order 3, Ext Ang speed [1]-[Tach...
Acoustic pressure (Pa)
1.8E-03
1.6E-03
1.4E-03
1.2E-03
1E-03
800E-06
600E-06
400E-06
200E-06
3000
4000
5000
6000
7000
8000
9000
10000
Angular velocity (RPM)
Display
Mode: Magnitude multi-gra
Traces:
Measurement2: Wtf:FF
Cursor1
X:1 #
Open
Customize...
Auto width
Copy Marker table
Close
Y: 174.7E-06 Pa
dX:1 #
0 s
0 RPM
dY: 0 Pa

Chapter 4: Tutorials

First lesson: spectrum analysis

Introduction

Objectives:

build a typical project in NVGate. The different steps of the procedure will be described in this page.
To make your first contact with the software easy, we will work on the basic operations of a spectrum analysis project.

A project ca be created quickly thanks to the setup toolbar. this wizard introduces the used functions through dedicated windows. We can find windows for:

- Input connection to the plug-in analyzers
- Setup of Front-end and plug-in analyzers
- Selection of results to be displayed and/or saved

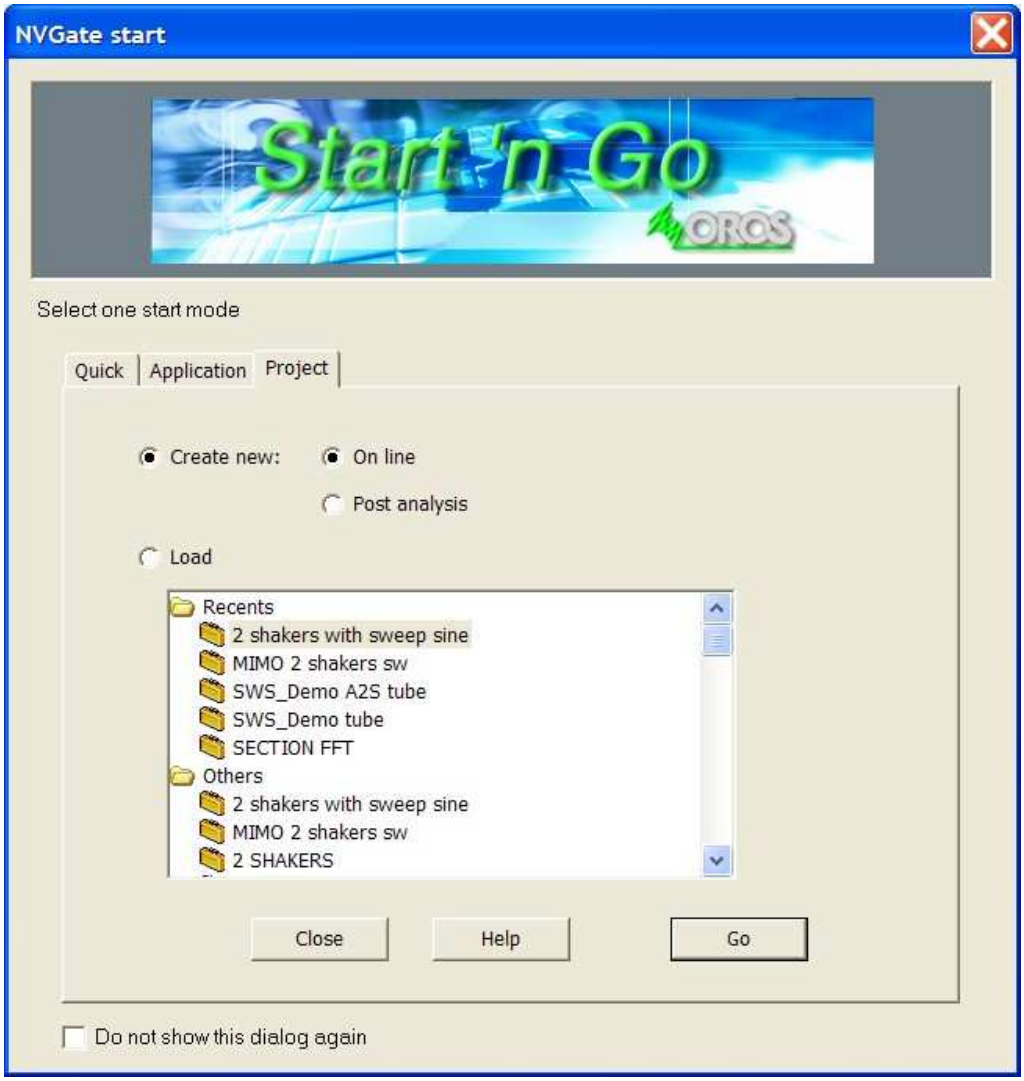
Equipment:

- Analyzer
- PC with NVGate
- Ethernet link
- 2 transducers. Here one accelerometer and one microphone will be used.

Procedure

Project building

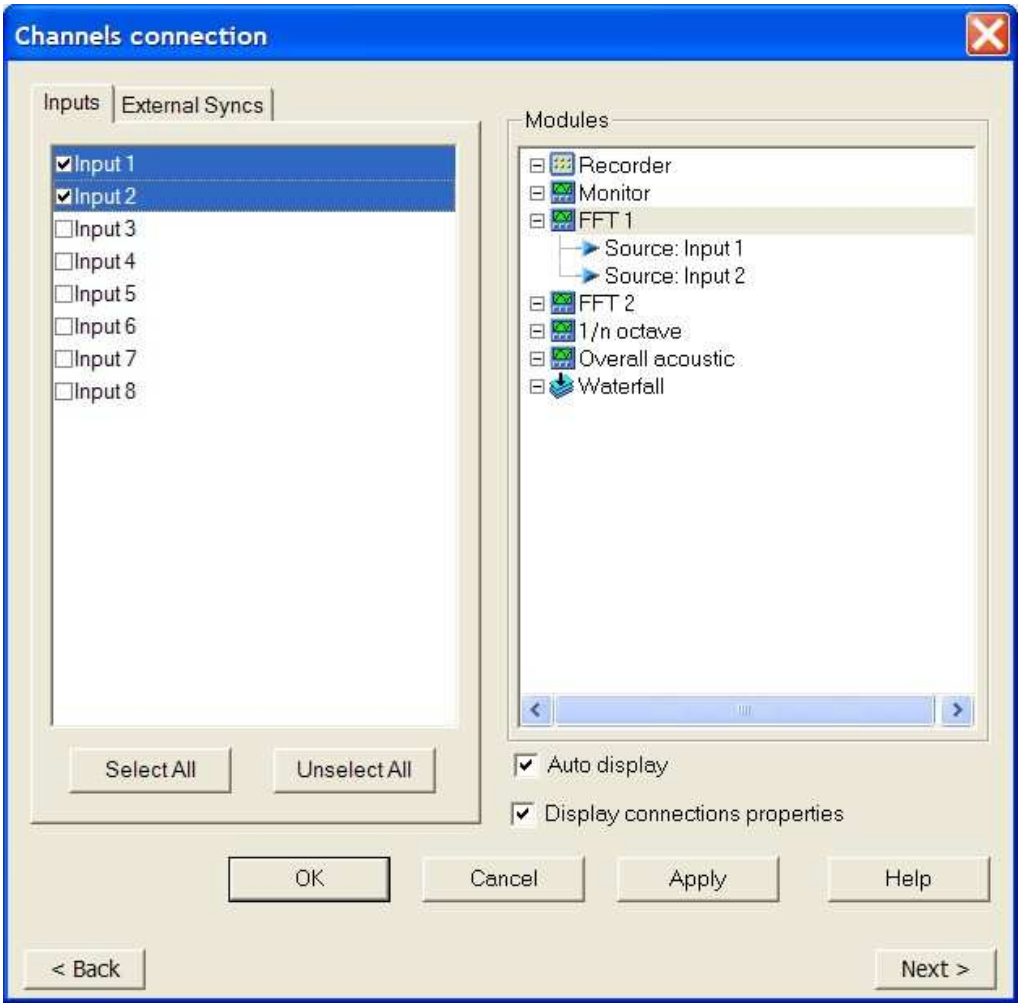
By launching NVGate, the Start'n Go window is automatically displayed.
Create a new On line project.



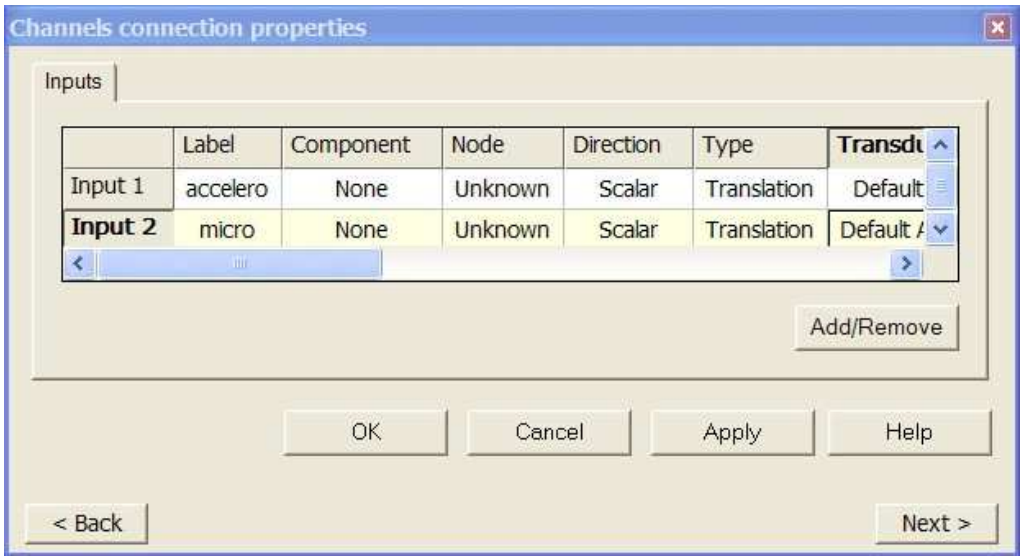
The setup wizard is automatically launched. It will guide through the different steps with dedicated windows.

Channels connection and properties

Connect Input1 and Input 2 on FFT plug-in by using Drag and Drop.



Define your transducer properties: label, transducer, coupling, range...

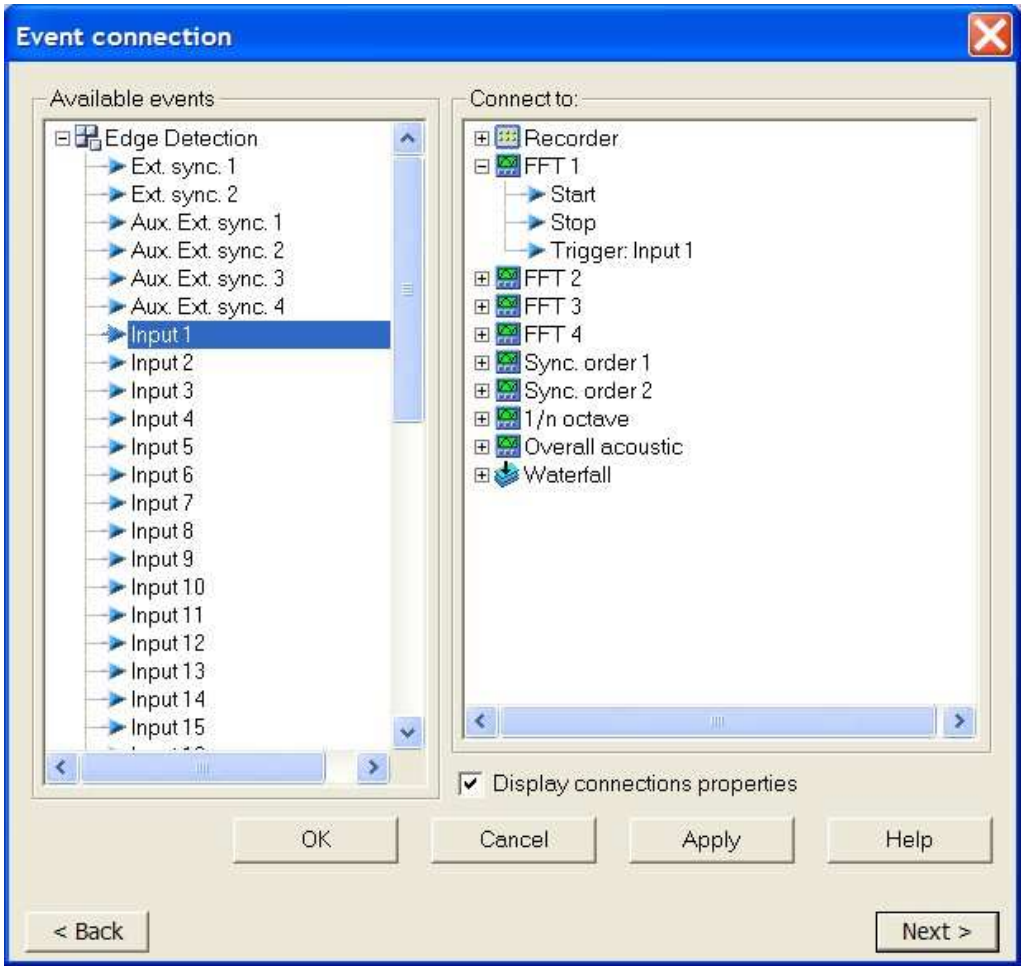


Tip: It is possible to assign an entire column entering information in the column header.

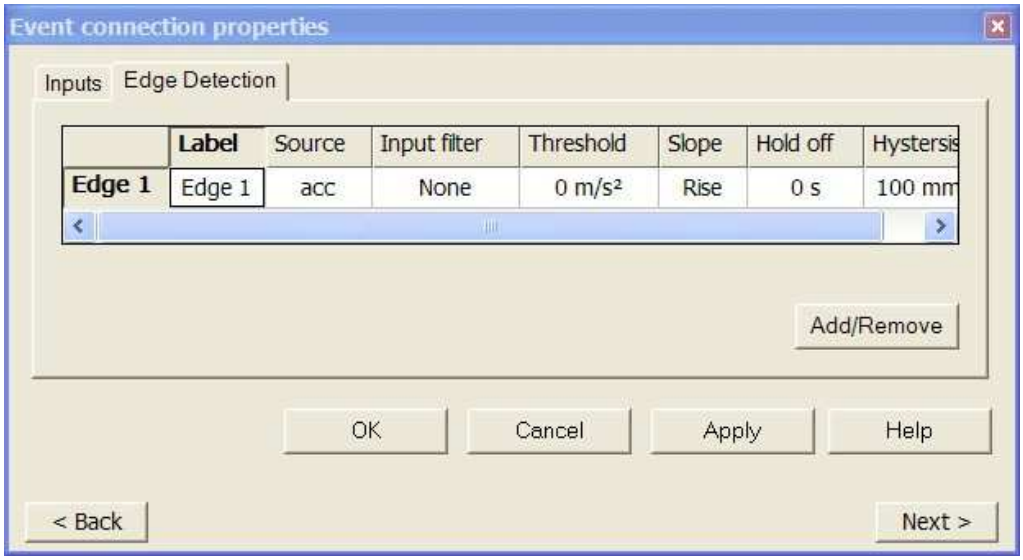
Tip: You can copy and paste any corresponding area (line(s), part of line(s), group of column from Excel).

Event connection and properties

We want to trig when the microphone level reaches a defined value.
So connect input 2 on FFT plug-in trigger and setup this event properties.

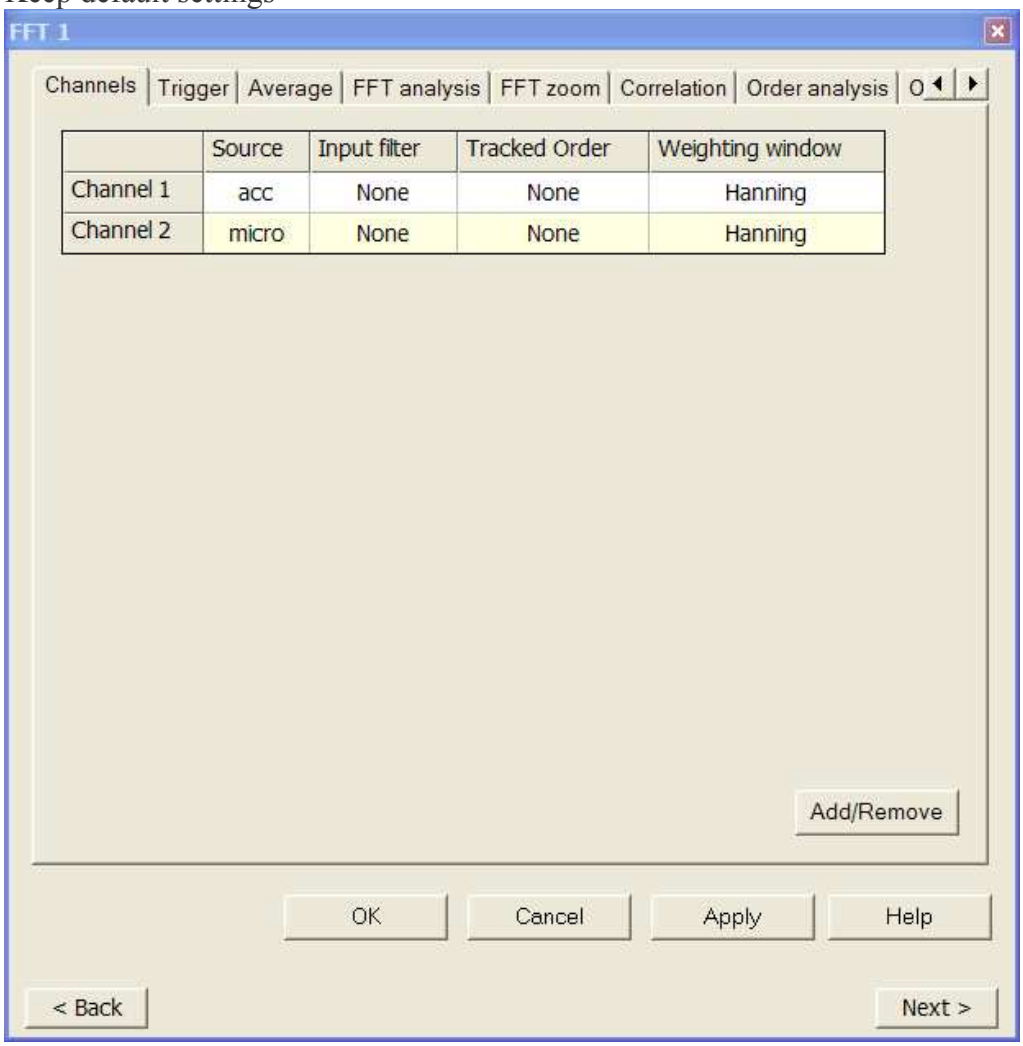


The value used for trigger is represented by the 'threshold'. This setting will be fixed later.

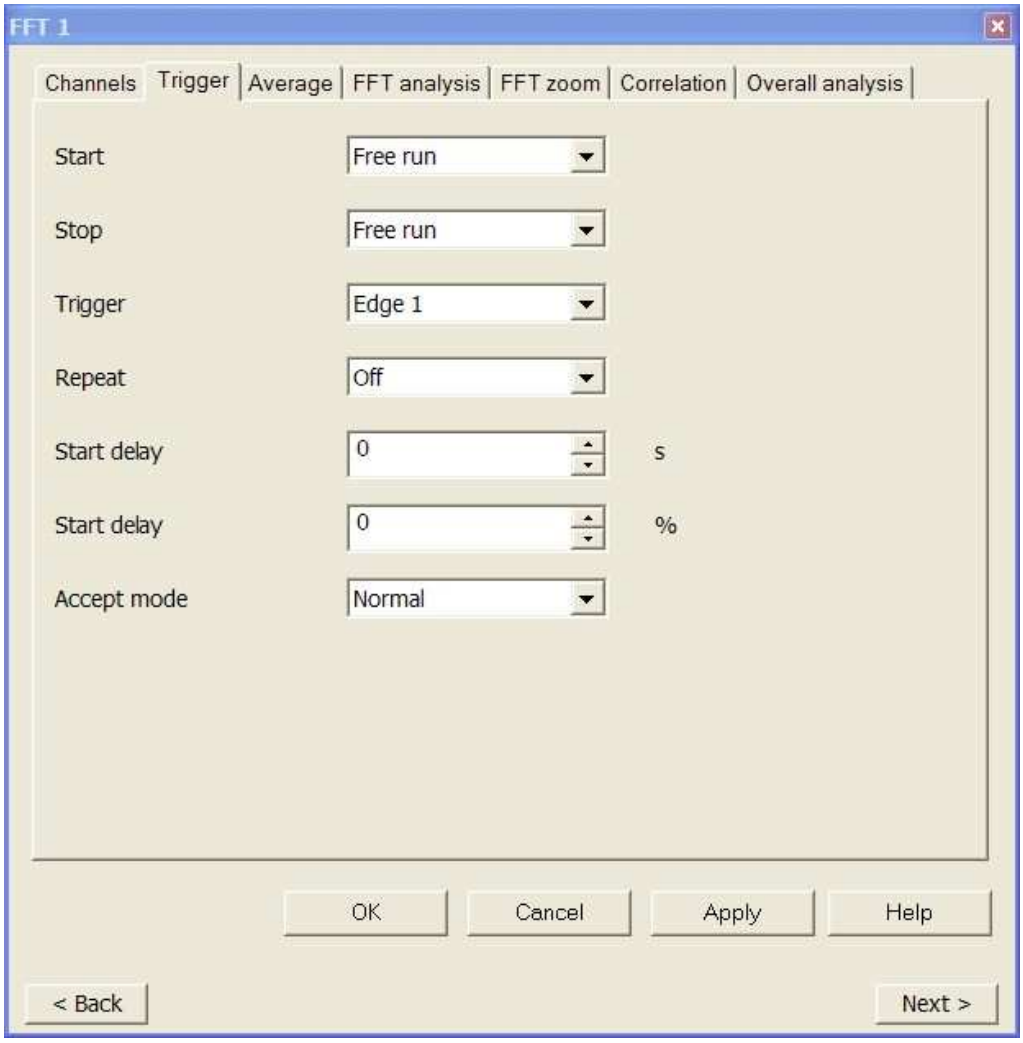


FFT settings

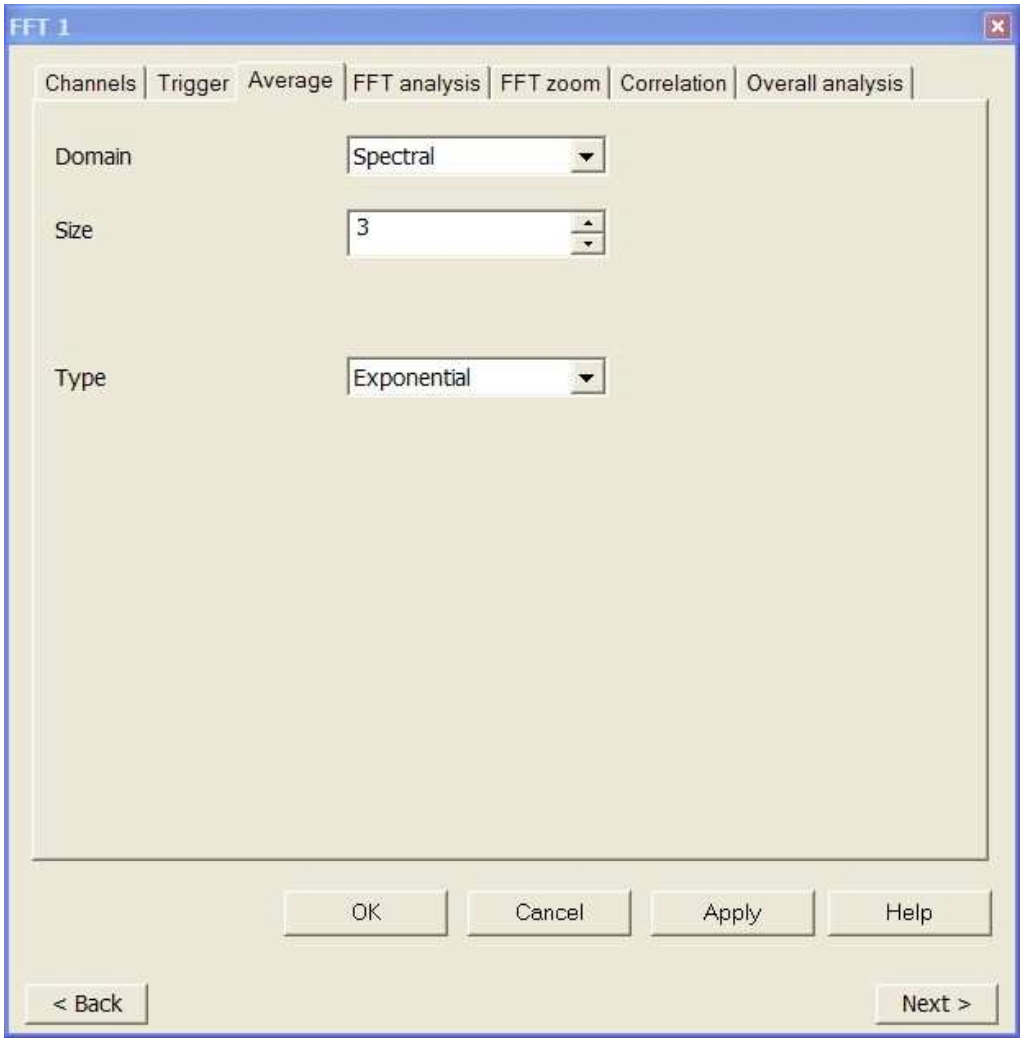
Channels: indicates if filters or weighting windows are applied on channels.
Keep default settings



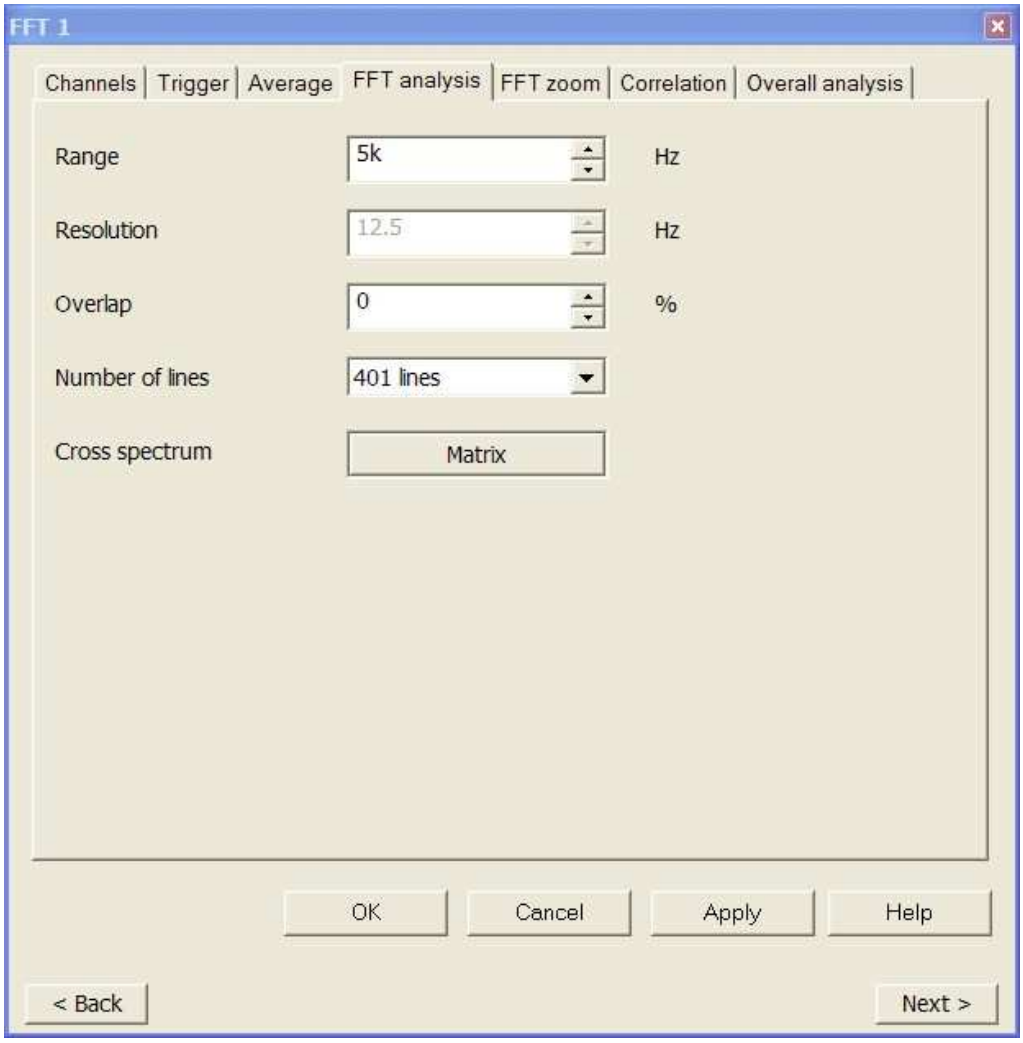
Trigger: parameters used to setup the trigger
Notice that the event 'Edge 1' is automatically linked to the trigger.



Average: keep default settings



FFT analysis: modify the frequency range to 5 kHz

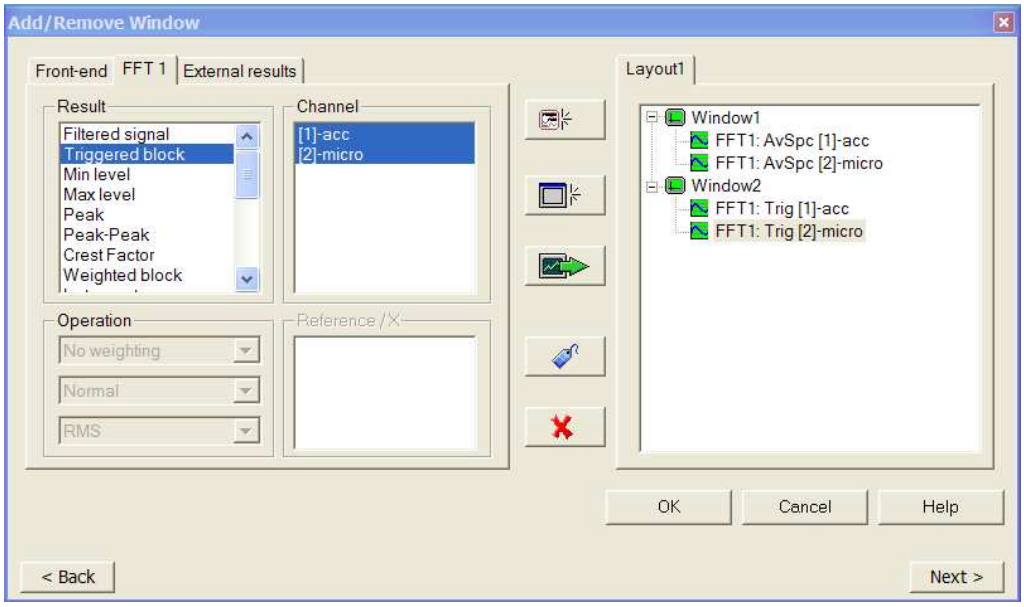


At this stage, Inputs are connected to the FFT plug-in and Front End and plug-in setups are done.

Now we have to select the results we want to display and save.

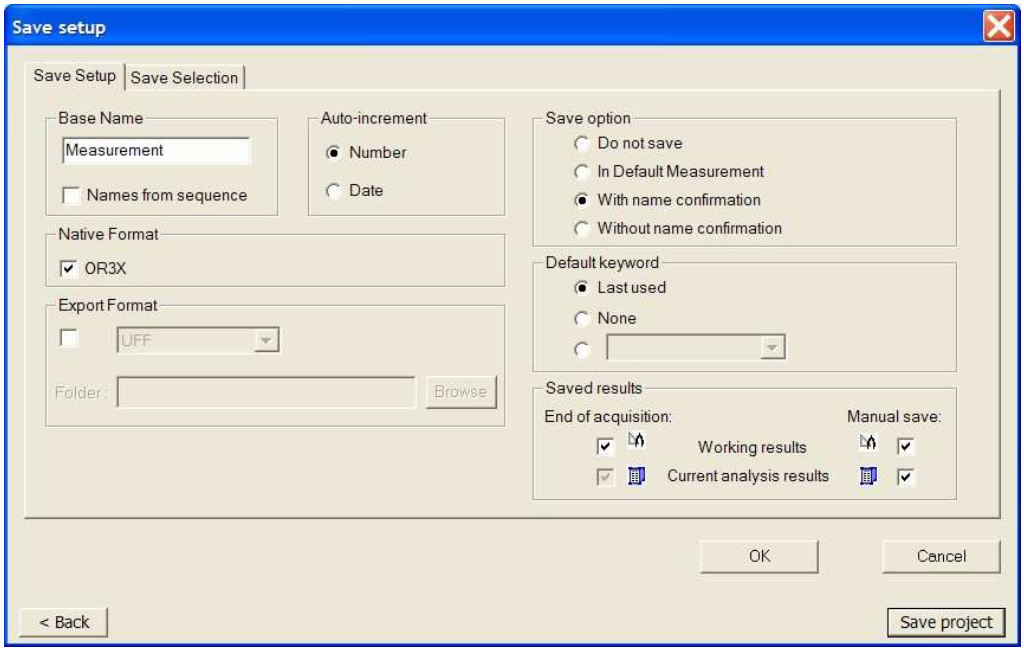
Display

Select the result from Front-end, selected Plug-in or external results.
In this project we are interested in Trigger block and average spectrum of both inputs.
Create 2 windows for these two kind of results.



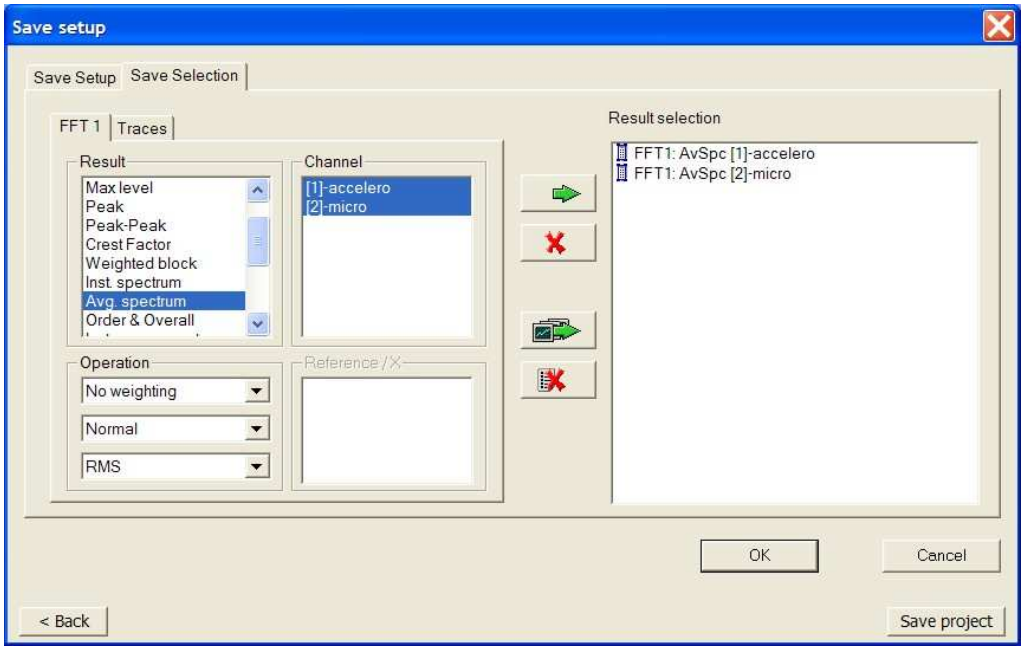
Save

Save setup: This window allows to chose the way of saving results.
By selecting 'with name confirmation', a window will be displayed after each measure for confirmation.



Note: if the options 'with name confirmation' or 'without name confirmation', the selected results are automatically saved at the end of the acquisition.

Save selection: this window allows to select the results you want to save.
Here, select accelerometer and microphone average spectrum.



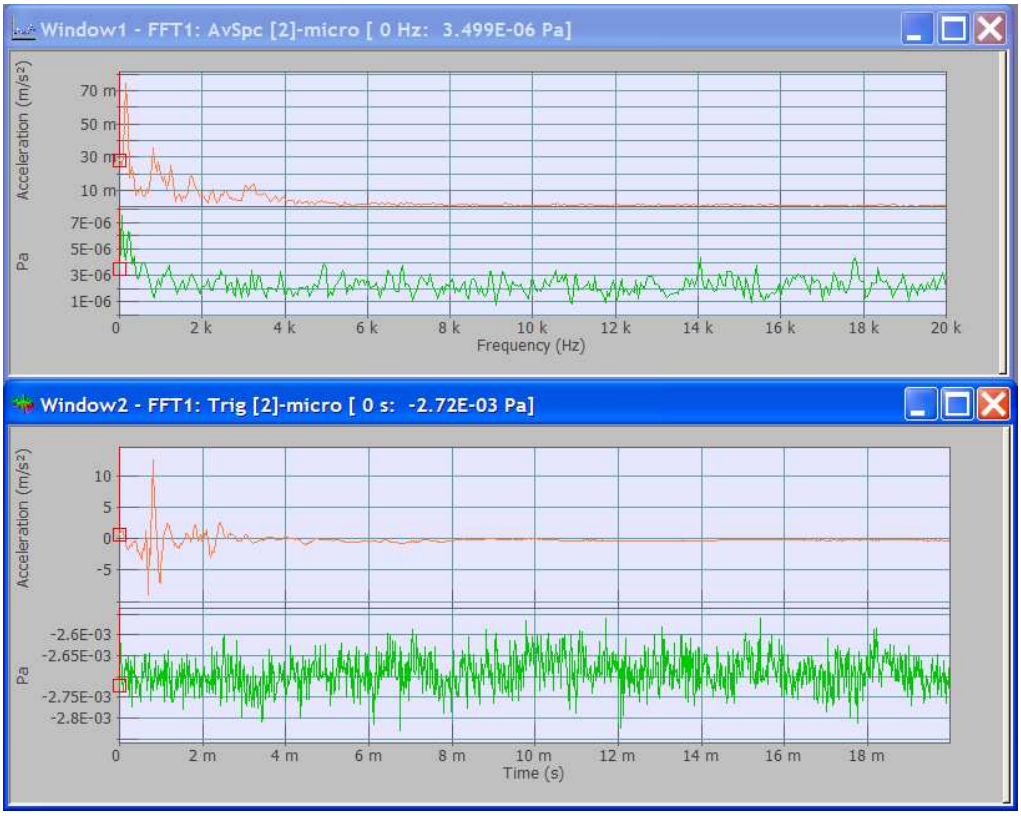
And save the project

The corresponding interface is automatically display.
The content of the interface and the Analyzer Setting Browser (ASB) depends on the settings.
You can access and modify parameters every time: in the ASB or by clicking on dedicated button from the setup toolbar.

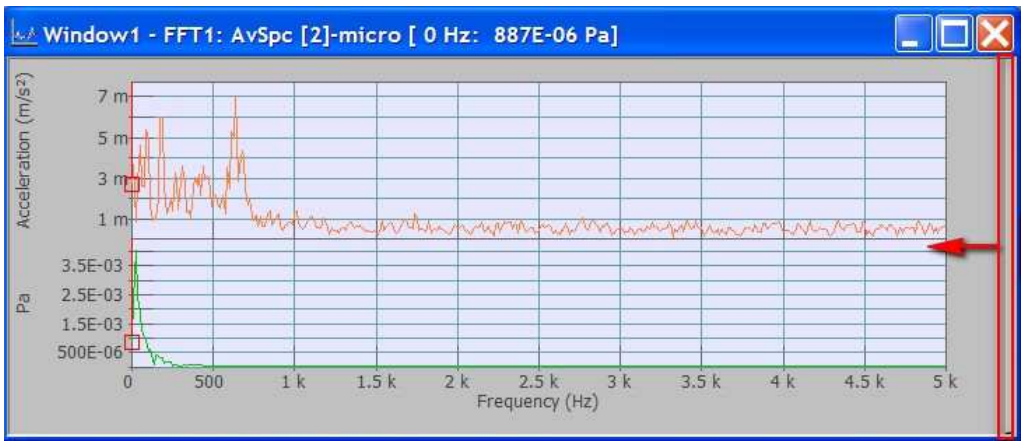
functions and information useful during the acquisition are displayed in the Control Panel

The status bar i the control panel indicates the global analyzer status (Run, Pause and Stop)
as well as the plug-in status (Number of block, waiting for trigger...)

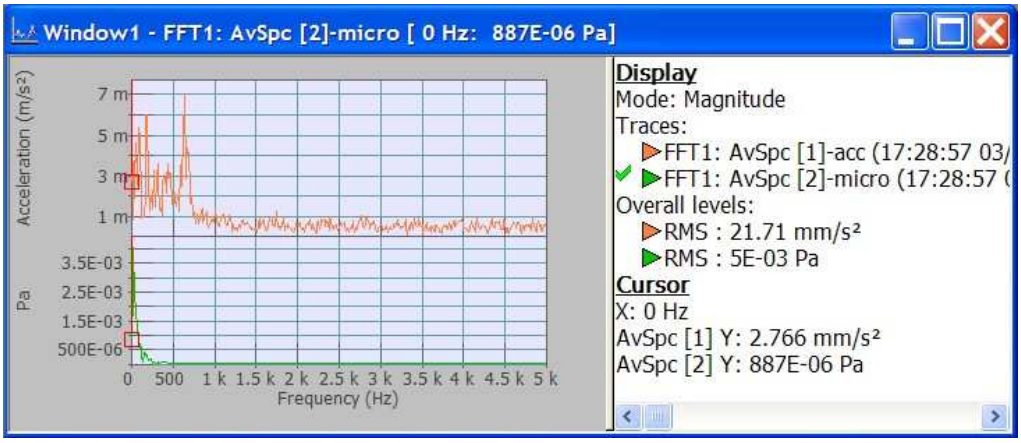
Click on run to start a first acquisition and impact the structure where the accelerometer is fixed for the trigger.



Note: You can automatically adjust Y axis to signal amplitude by using CRTL+A (Auto-scale) shortcut. This function is also available from the Zoom toolbar



Click on the bar on the right and drag it to the left. A window called Infotrace will be displayed.



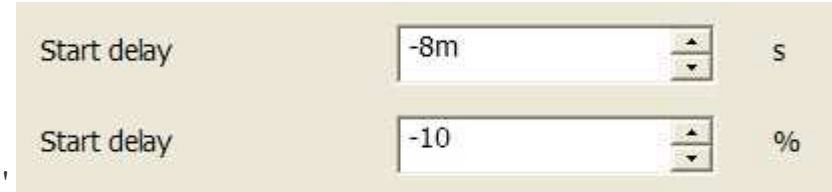
The Infotrace displays information about graphs:

- Graph mode (Magntude, real...)
- List of traces with their status
- Overall levels
- Cursor Information
- Marker arrays

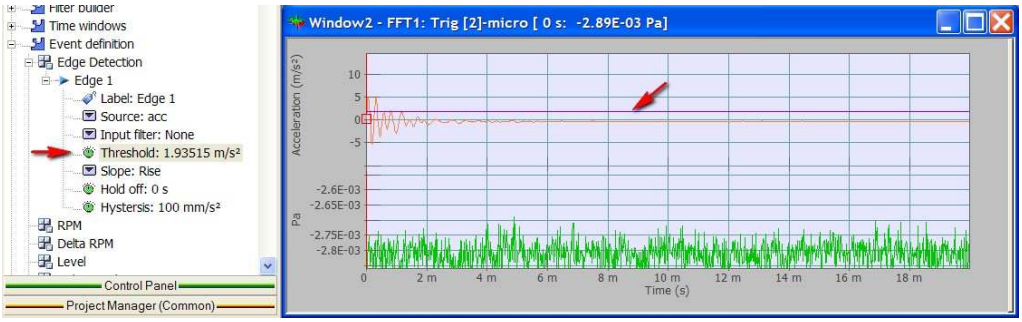
Project personalisation

After the first Run, you can adjust some parameters depending on the obtained results.
Here, we can typically set up the event threshold and add a start delay.

- **Start delay:** Click on and add a negative value for the 'start delay

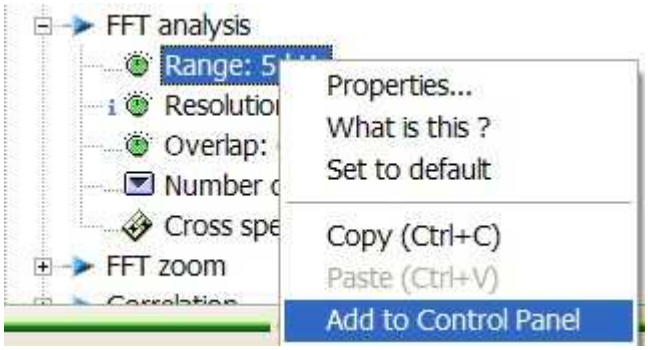


- **Threshold**: Drag and drop the threshold from the ASB to the micro trigger block. By this way , you can set up the threshold value directly on the graph.

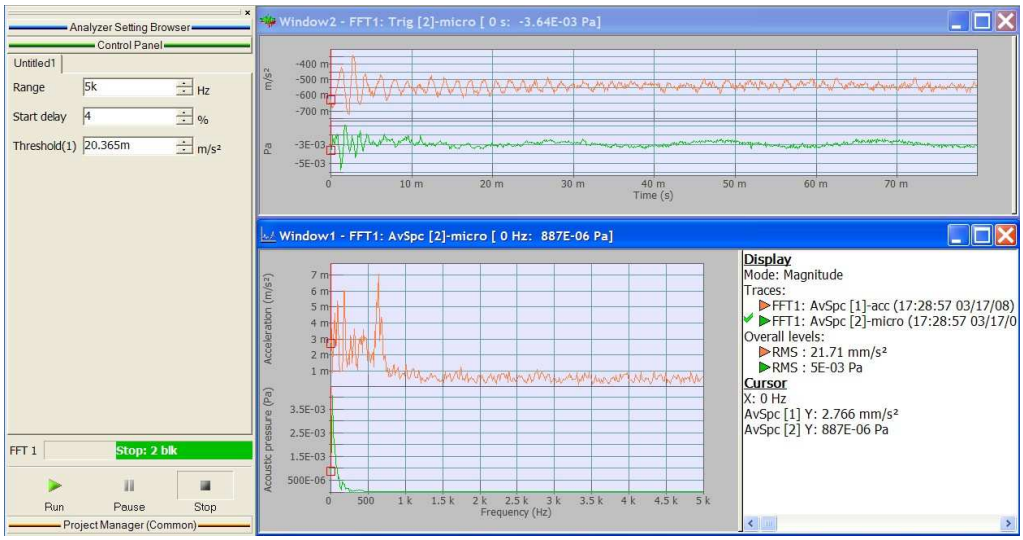


- **Control Panel customization:**

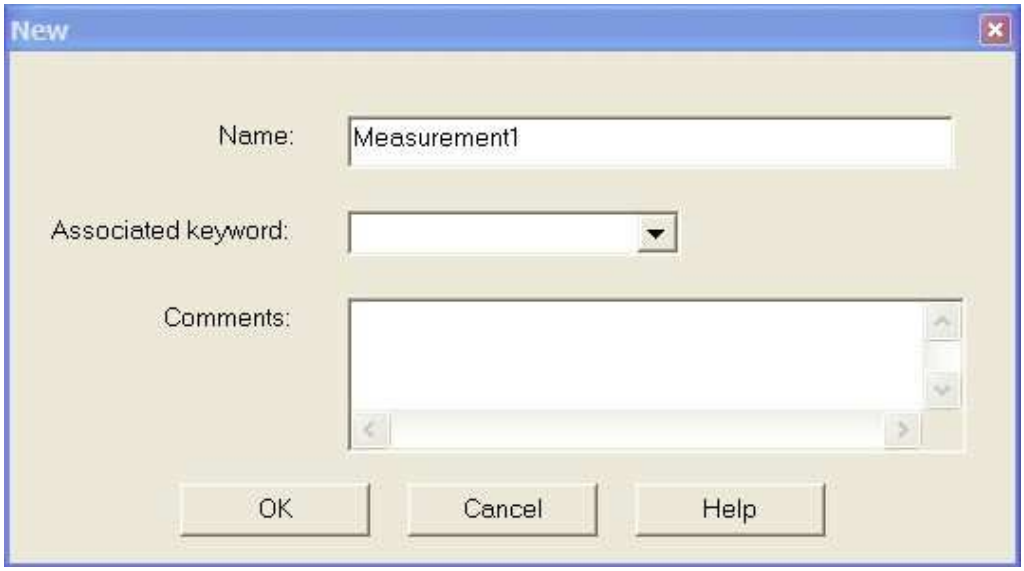
  If in your project, some parameters need to be easily accessible, you can add them into the control panel.
  In the ASB, right click on the selected parameter and 'Add to control panel'.



For the second run, the interface will look like the picture below:



After a Run, the selected results are automatically saved and a window is displayed asking the name of the measurement.
The results are saved in a measurement added to the project .


New
Name:
Measurement1
Associated keyword:
Comments:
OK
Cancel
Help

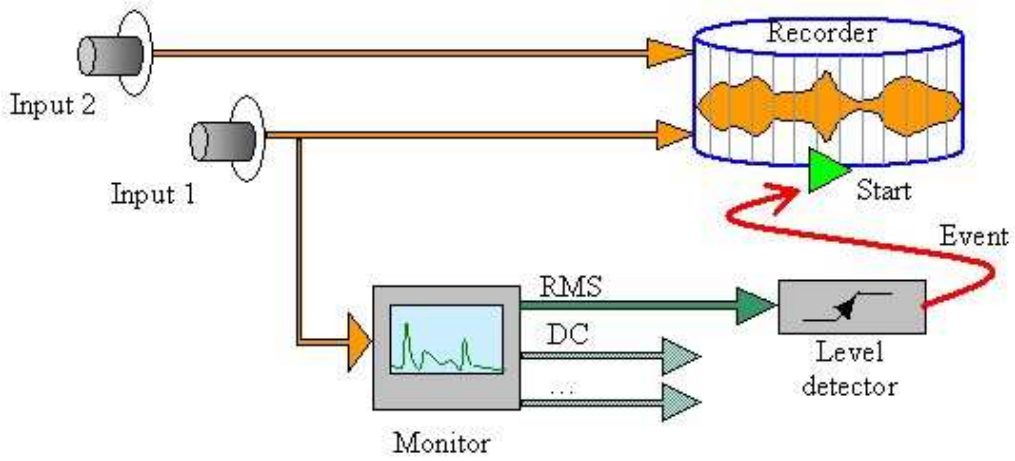
spectrum analysis
Workbook
Measurement1
Workbook
Result
FFT1: AvSpc [1]-accelero
FFT1: AvSpc [2]-micro

Acquisition and recording

Introduction

Objectives:

This lesson explains how to record time domain signal using the Recorder module. In this session, we will use the RMS level of one of the recorded signal to trigger the recording.



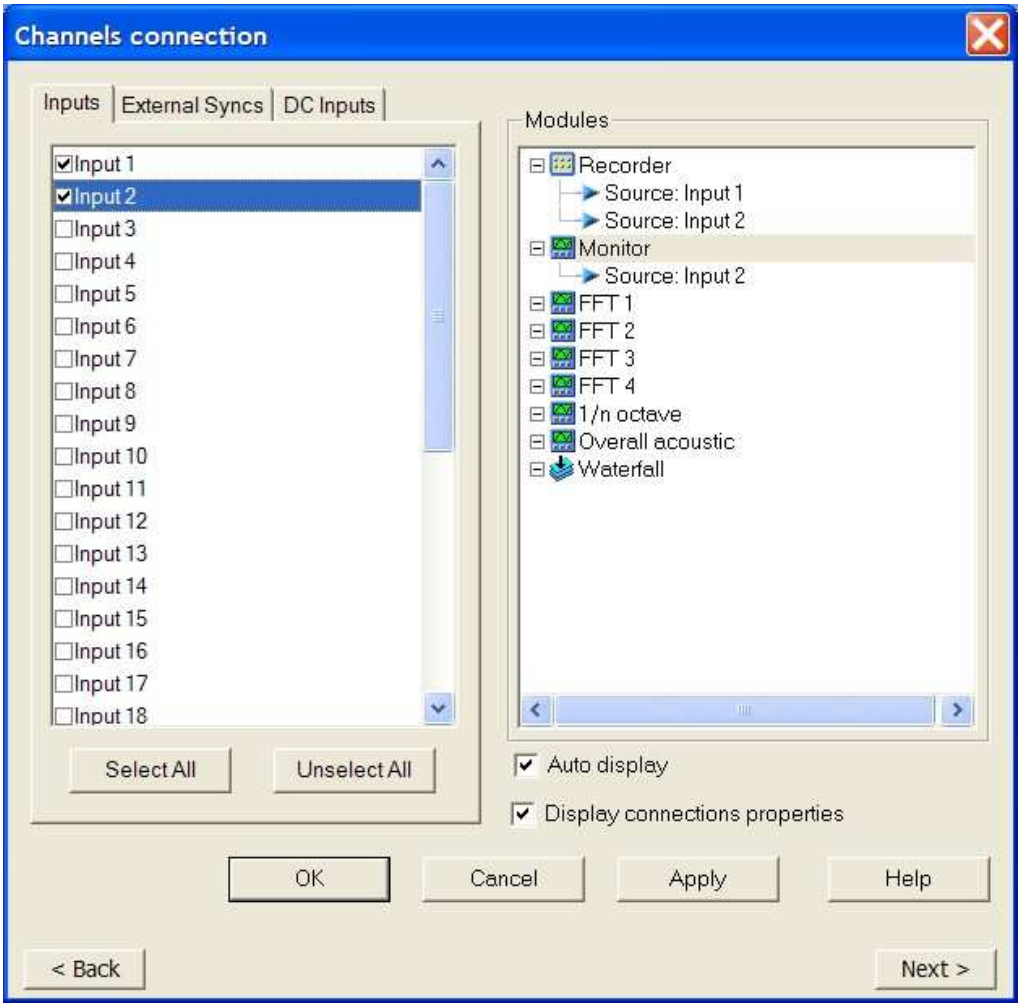
Equipment

- Analyzer
- PC with NVGate
- Ethernet link
- 2 transducers. Here one accelerometer and one microphone will be used.

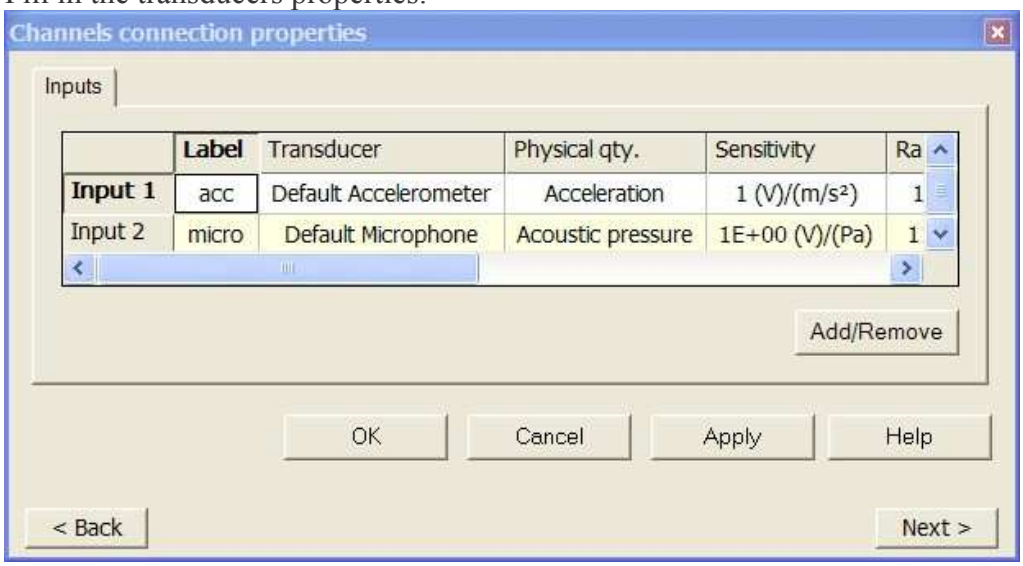
Procedure

Channels connection and properties

Connect the accelerometer to Input1 and microphone to Input2.
Drag & drop Input1 to the recorder and Input2 to the recorder and the monitor.



Fill in the transducers properties.

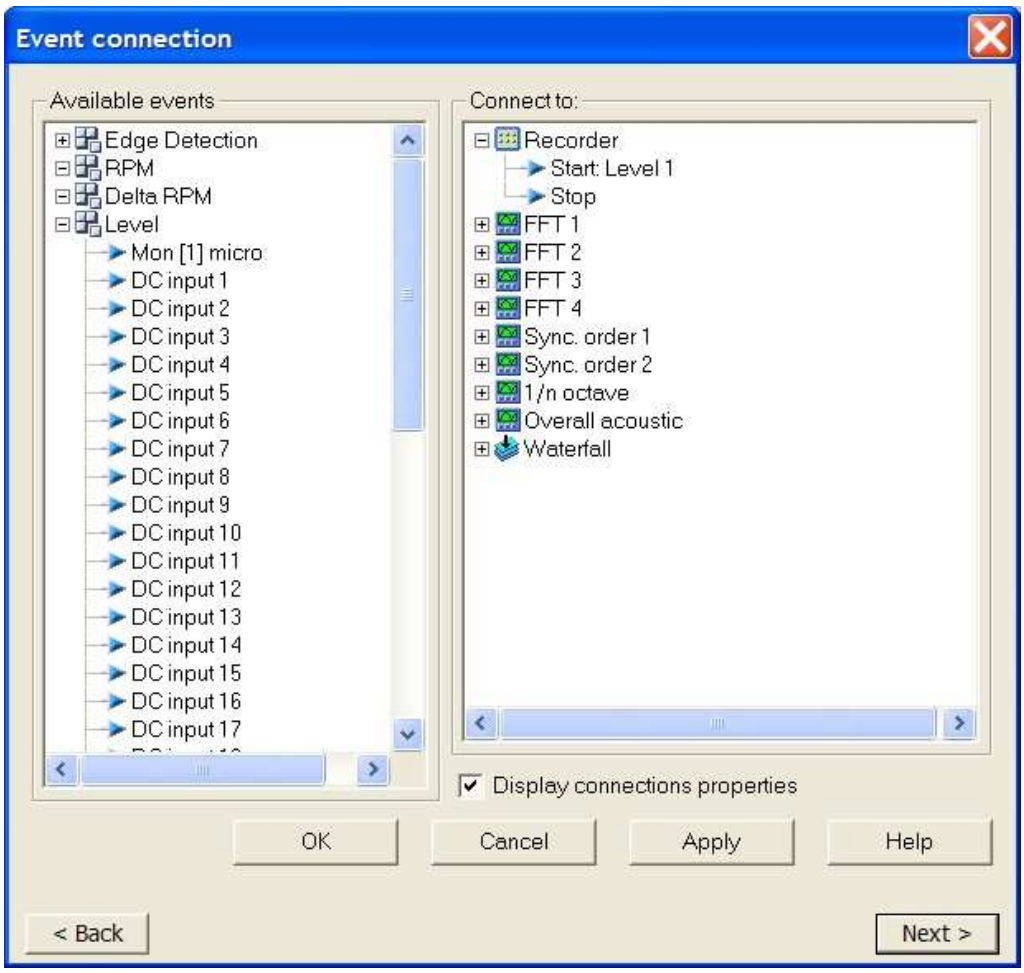


Event connection and properties

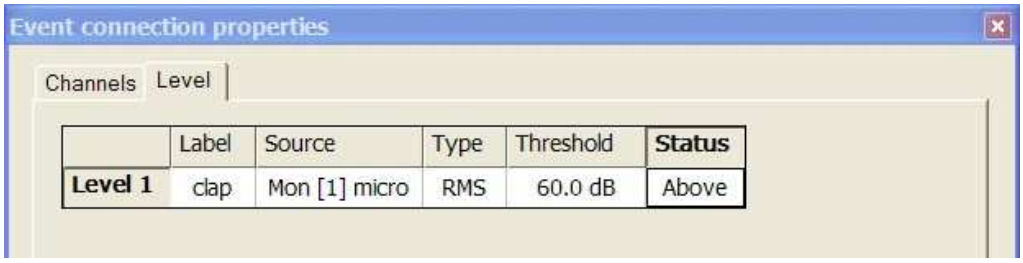
We will create an event corresponding to a Level detection in order to use it to start the Recorder. The Monitor window will be use to visualize current Input2

RMS level and so setup the threshold.
We choose to clap in the hands to start recording.

Drag & drop the microphone in 'Level' to the recorder and click on next.

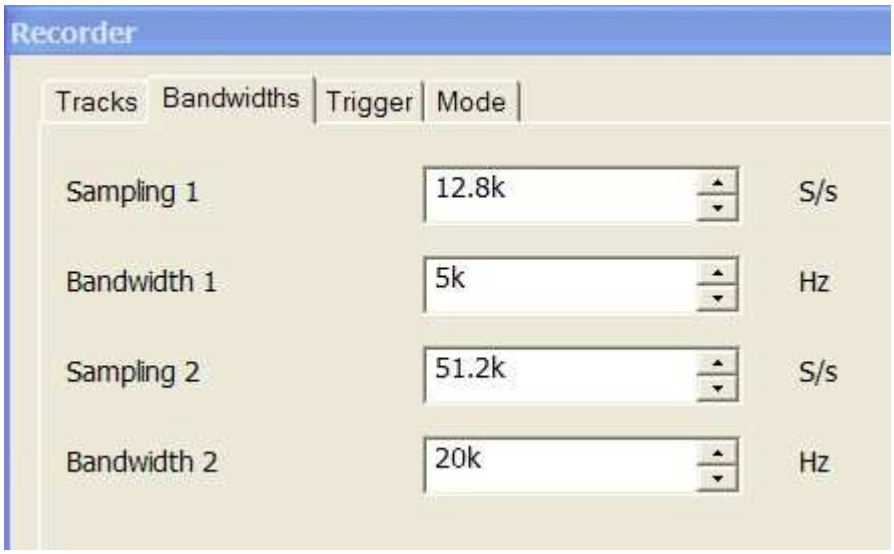


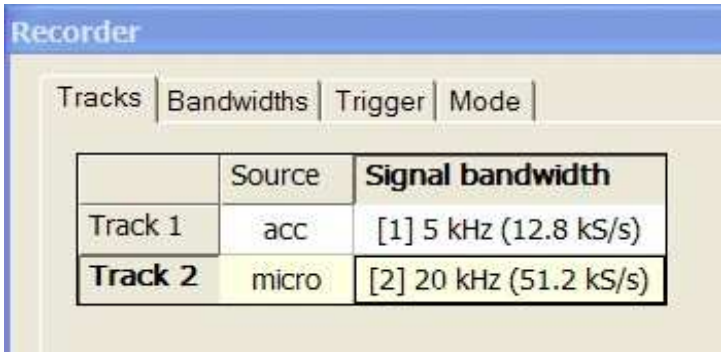
Fill in the event properties: select the RMS type and setup the threshold to 60dB.



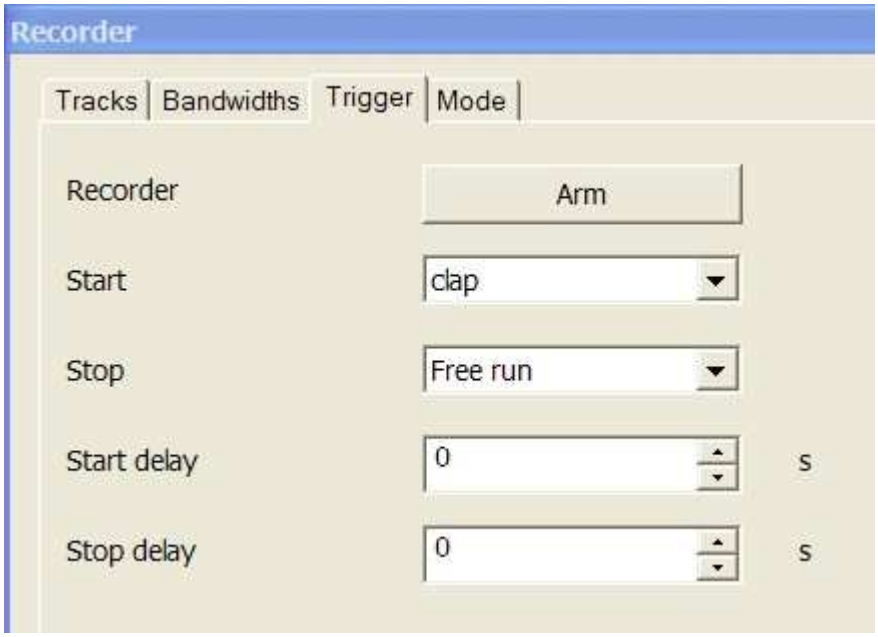
Recorder settings

Bandwidths: possibility to select 2 different bandwidths with 2 associated sampling. You can associate one or the other to the channels.

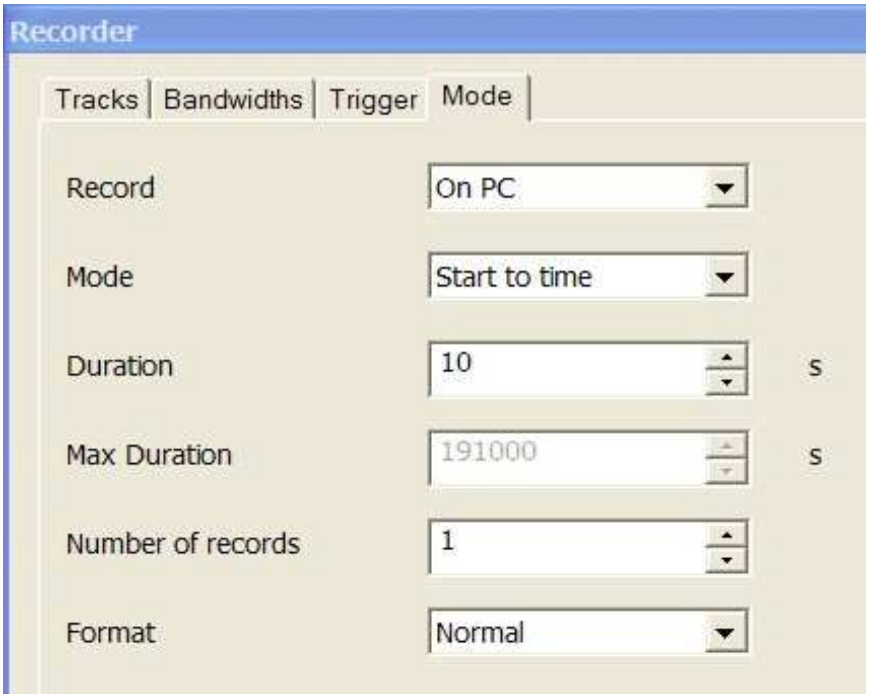




Trigger: use the event 'clap to start the record.



Mode: select the mode start to time and set the duration to 10s.



Click on OK.

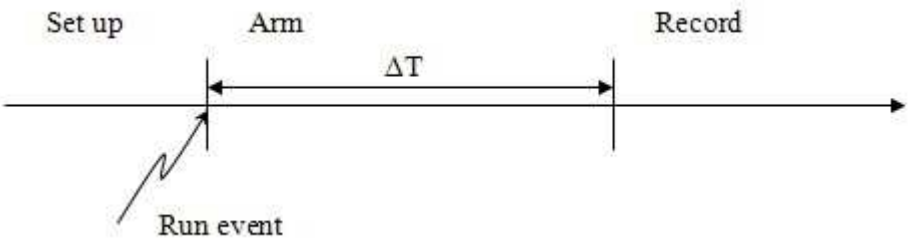
The corresponding interface with record signal window and microphone RMS level window is automatically display.

Click on run to start a first acquisition and clap your hands to start the record.

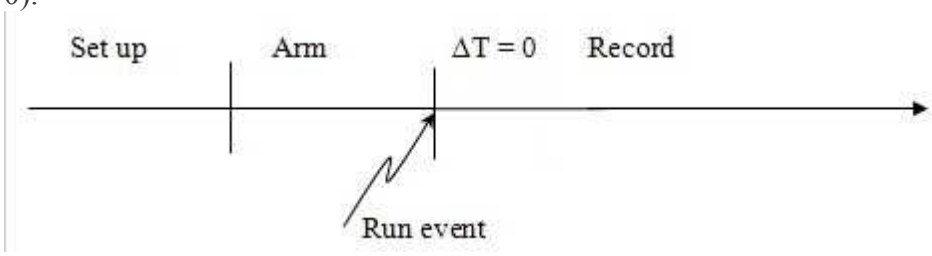
At this stage, you can note the function 'arming' implemented in the recorder.

Arming of the recorder is a backup of your record settings allowing an interesting gain of time:

- Unarmed record: the recorder is armed automatically by the start event. Then between the beginning of the start event and recording, there is a time ΔT corresponding to the arming (i.e. the backup of the record settings).

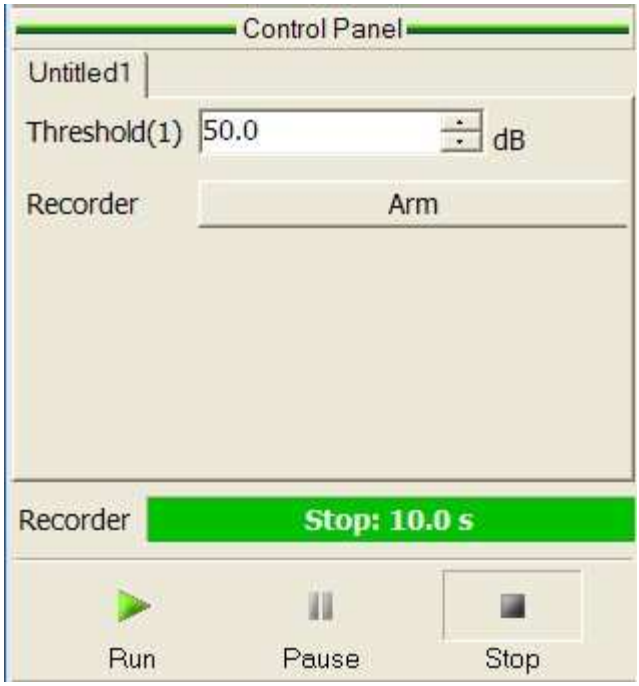


- Armed record: after the setup the recorder is armed (i.e. backup of the record settings allowing more security for you record). Then at any moment, when the recorder is armed, a ’Run event' starts the record instantaneously (ΔT = 0).

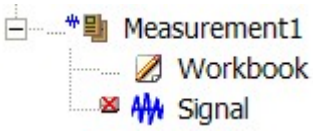


Project personalisation

Customize the control panel:
You can add the 'arm' function to the control panel as well as the threshold linked to the event by right clicking on it in the ASB



After a Run, the signals are automatically saved.
The results are saved in a measurement added to the project



.

Waterfall lesson

Introduction

Objectives:

This project shows the basics operations with a waterfall. Typically to extract order by analyzing rotating machine.
To go further in the order analysis, see 'order tracking' tutorial.

Equipment

- Analyzer
- PC with NVGate
- Ethernet link
- an accelerometer
- a tachometer
- a rotating machine

Position the accelerometer on the rotating machine and connect it to input 2.
Connect the tachometer to Ext.Sync1.

Procedure

Use the setup toolbar to define parameters.

Channels connection and properties

The accelerometer is placed on the structure of the rotating machine and connected to Input 1
Tachometer is connected to Ext Sync 1
Connect these 2 channels in NVGate and fill in the channels connection properties.
Accelerometer label will be 'accelero' and tachometer label will be 'tacho'.

Note: Rotating Mode must be selected

Tachometers connection

Tachometer information will be use to full in the waterfall. We decide to display average spectrum in the waterfall every 50 rpm.
So in this 'tachometer connection' window, drag and drop Ext. Sync.1 on the waterfall as well as on Delta RPM 1.
Fill in the 'Delta RPM properties' especially 'Delta velocity' fixed at 50 rpm.

Event connection

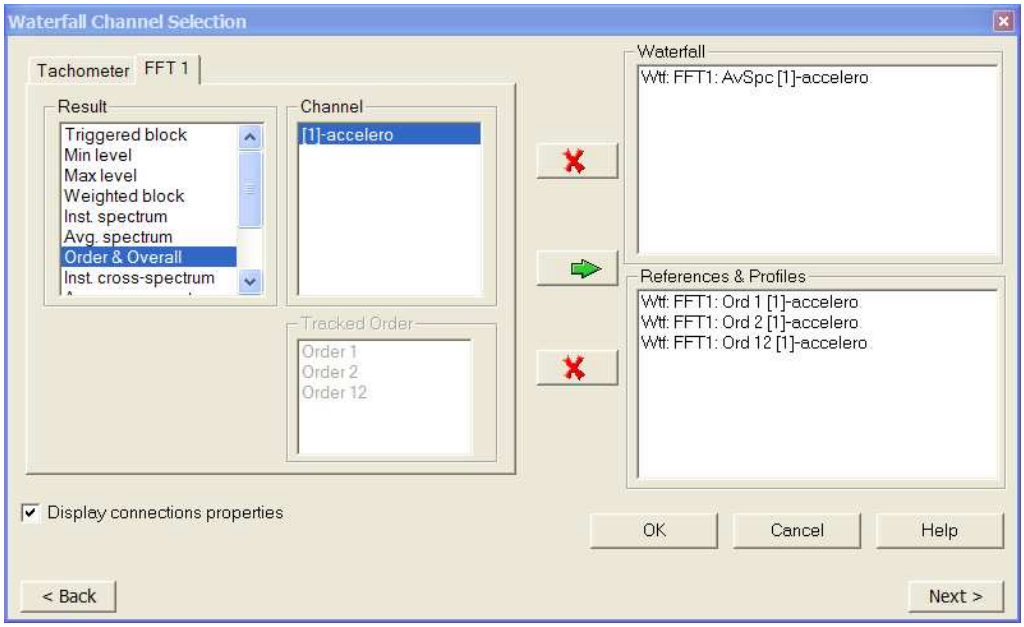
The 'Delta RPM' defined previously will be use to trig the waterfall so, in this window, drag and drop 'Delta rpm 1' on the waterfall.

FFT properties

Define FFT parameters, especially the frequency range can be set to 2kHz.

Waterfall definition

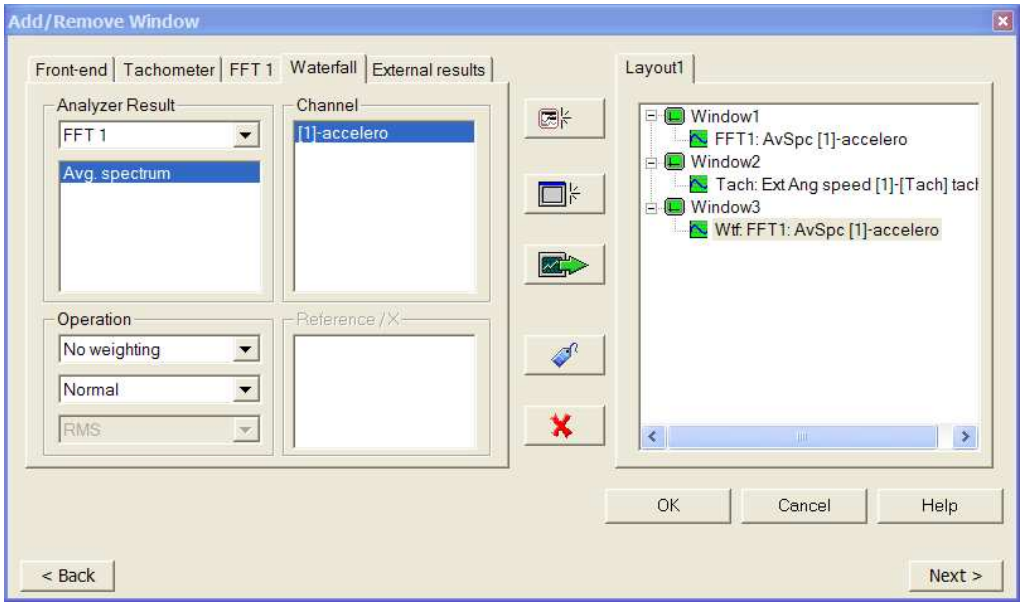
Drag and drop Ext.Tach Angular Speed on References&Profiles and display accelerometers average spectra in the waterfall.



Windows

Display:
- Ext. Tach. Angular speed
- Accelerometer average spectrum
  ➡ To check if speed and accelerometer signal are correctly displayed.

- Waterfall containing average spectra



Save the project

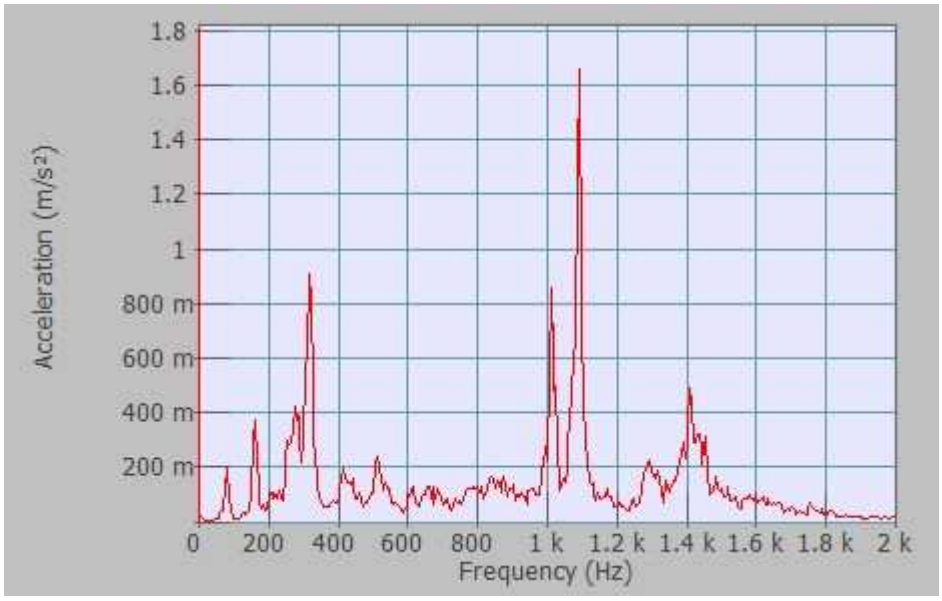
Acquisition

Last settings

Press Run.
Check that the speed is exact and that the accelerometer gives correct information.
Regarding the results, adjust parameters if necessary.

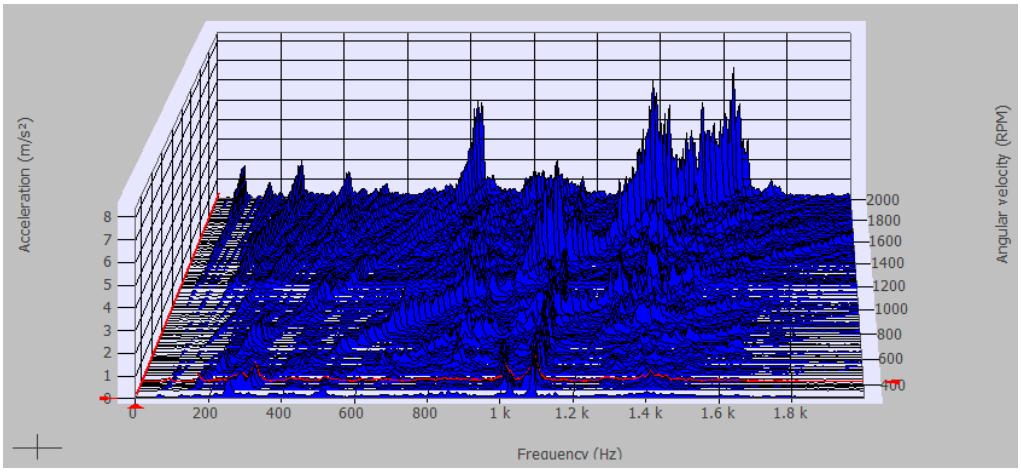
Note: You can adjust properties of this window by right clicking on it.

Accelerometer spectrum



Acquisition with waterfall

After these last settings, press Run and make a Run up with your rotating machine.
The slices representing accelerometer average spectra will be added to the waterfall every 50 RPM.



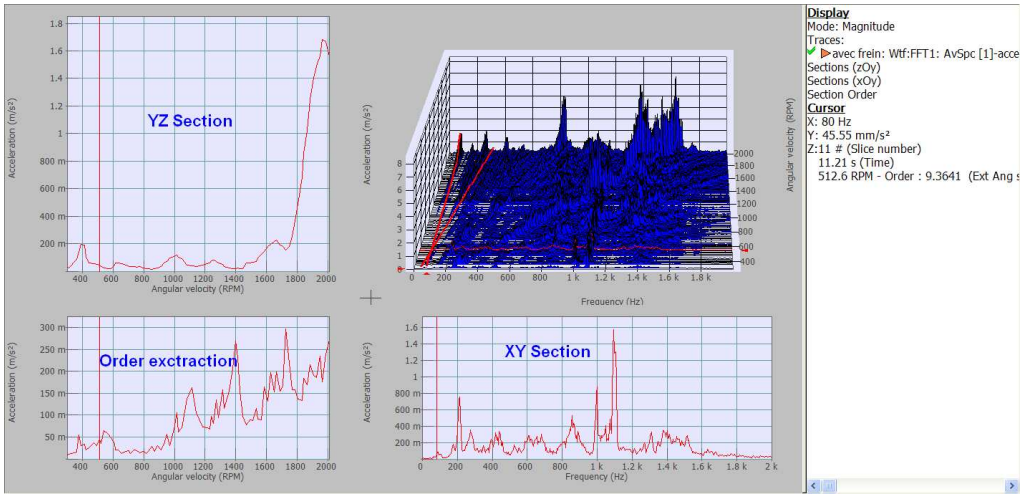
Waterfall Operations

The Waterfall window offers a wide range of operation in order to go further in your measurement. It is possible to display XY and YZ sections as well as the extraction section.

All graphs

Right click in the waterfall window and select graphs/all graphs. The waterfall window is composed of 4 mains area:

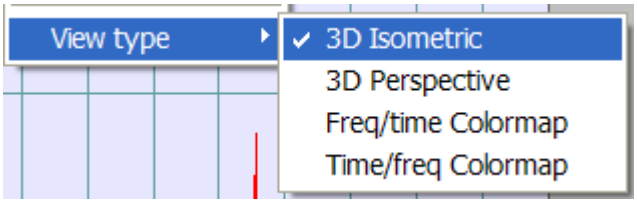
- **3D** waterfall graph or **ColorSpectrum** view
- **XY** section
- **YZ** section
- Extraction section (allows to extract orders from a spectrum with a tach or extract a frequency from order spectrum with a tach).



Cursors give information about the displayed Waterfall trace (current X, Y and Z coordinates are available from the Infotrace).
Move the cursors either by using the drag & drop in the 3D view or by using the arrow pad of the keyboard (the cursor mode must be selected).

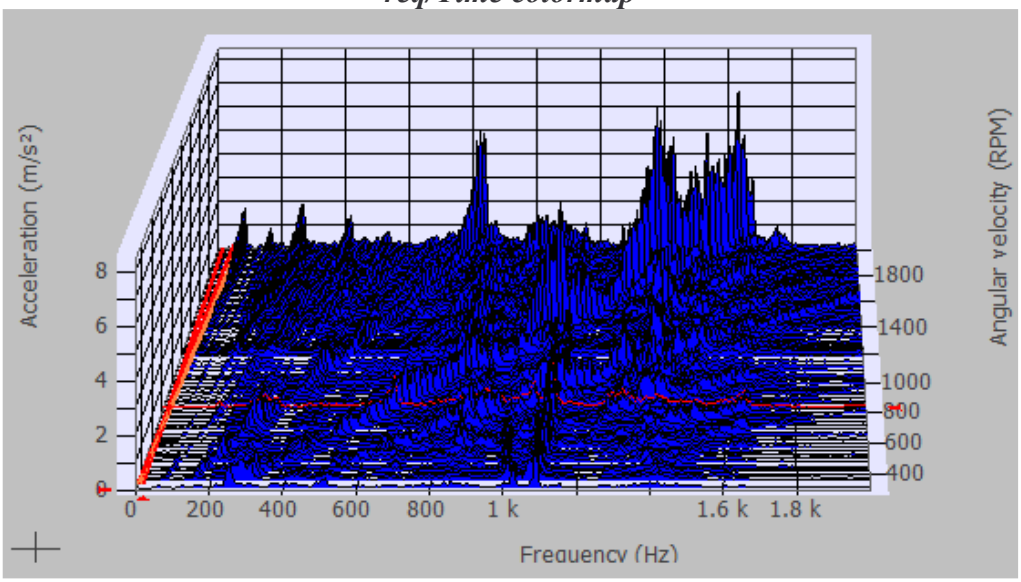
Tip: Move the mouse over the cross in the middle of the window and click in order to resize manually the graph window.

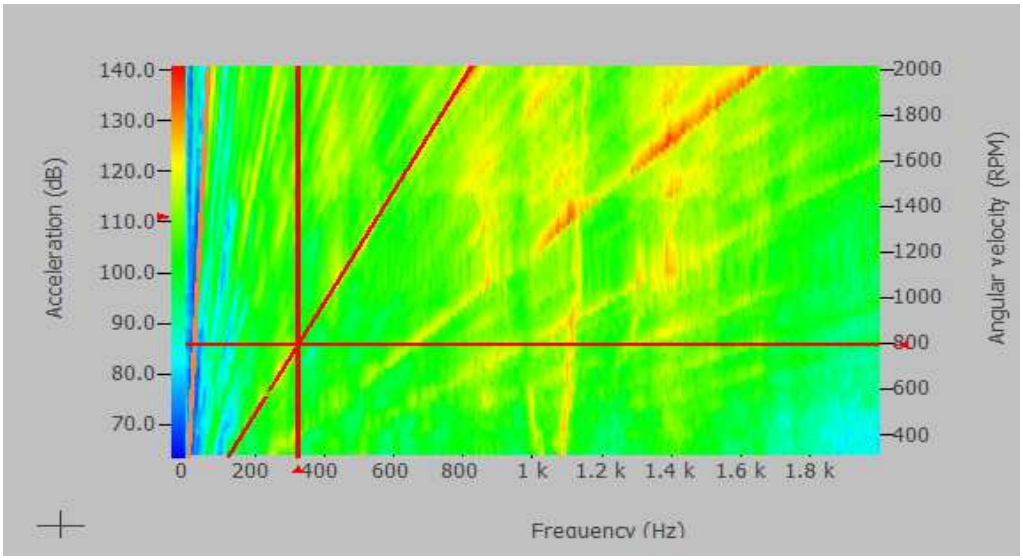
3D waterfall graph or ColorSpectrum view



3D Isometric

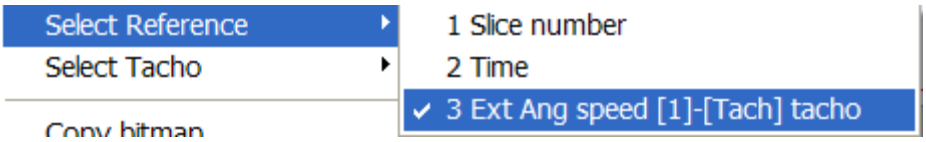
Freq/Time colormap

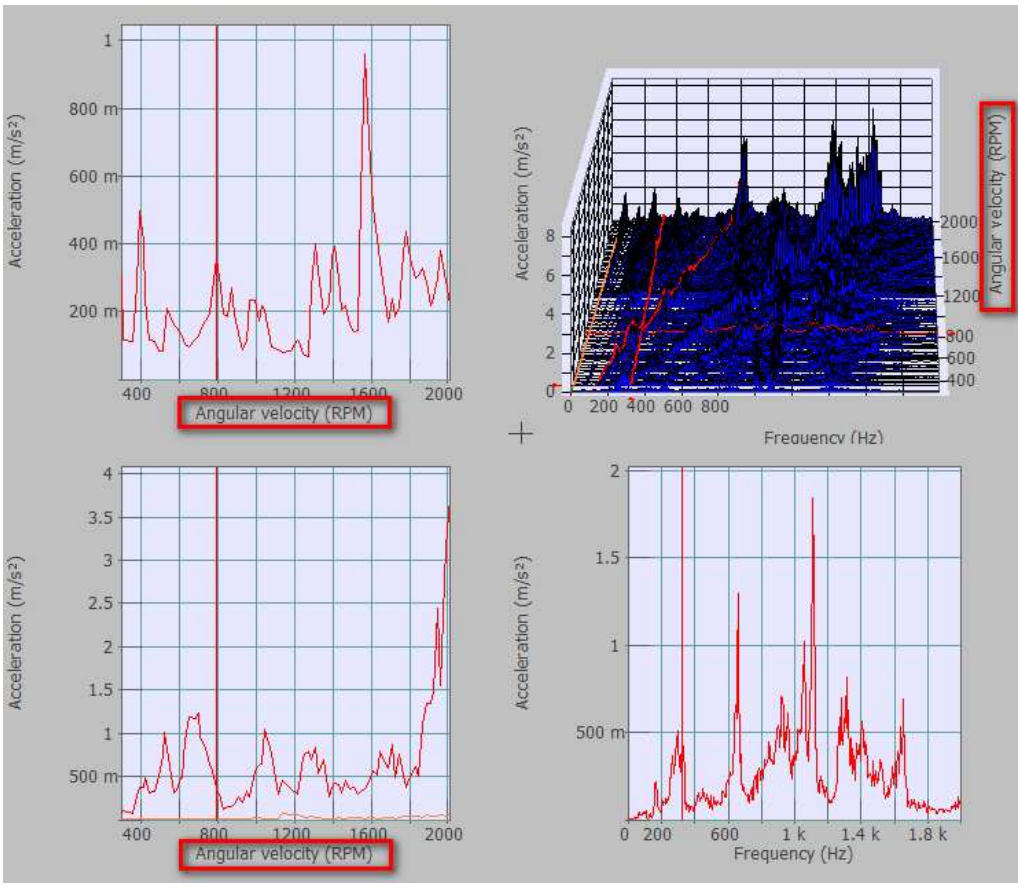




Select reference

You can also change the Waterfall reference.

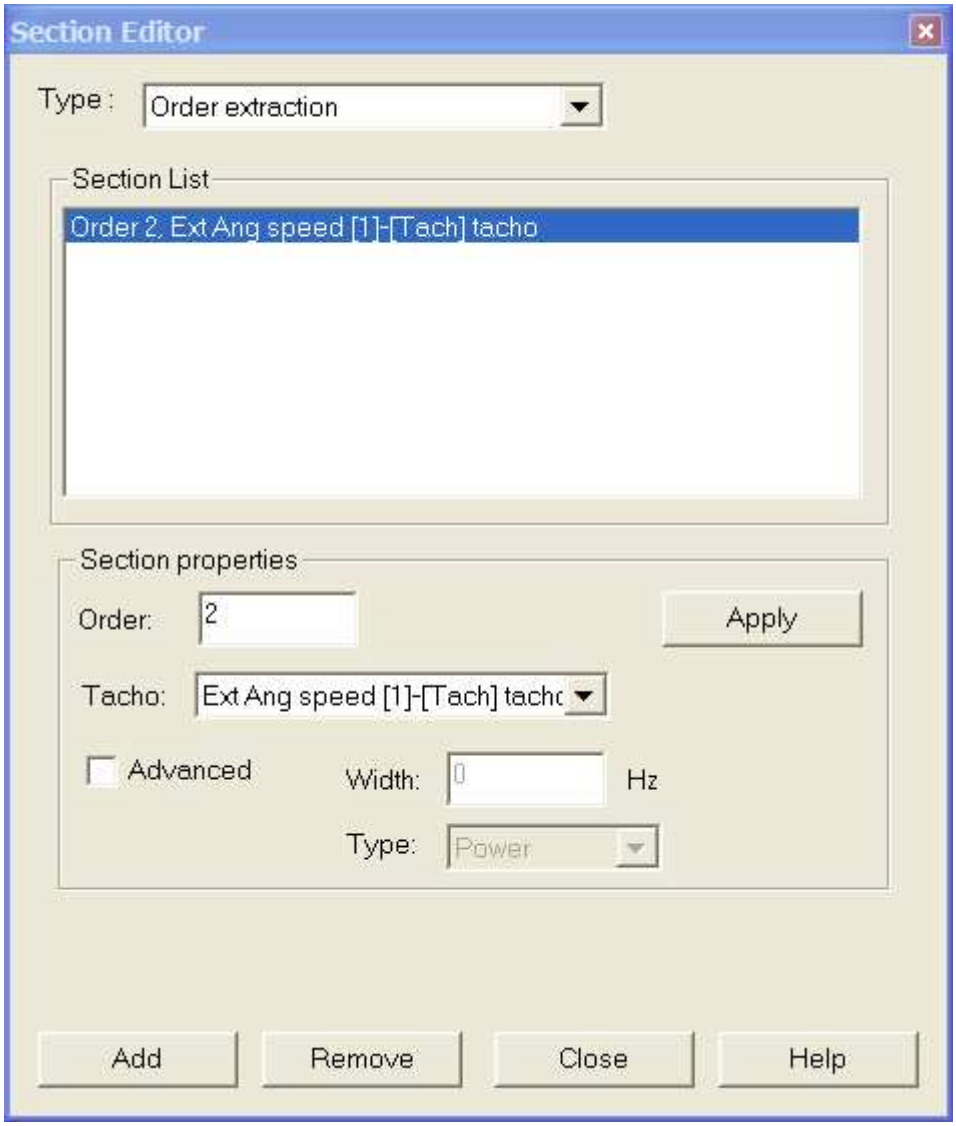


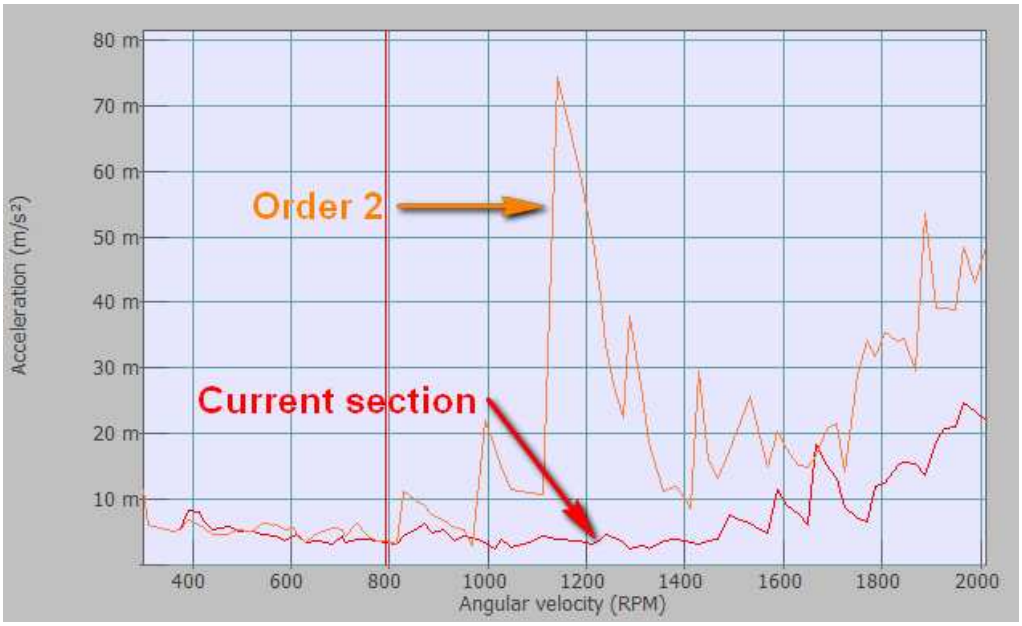


Section manager

You can fine tune the section parameters by using the section manager available by right clicking on the waterfall window.

For example, you want to visualize the second order.

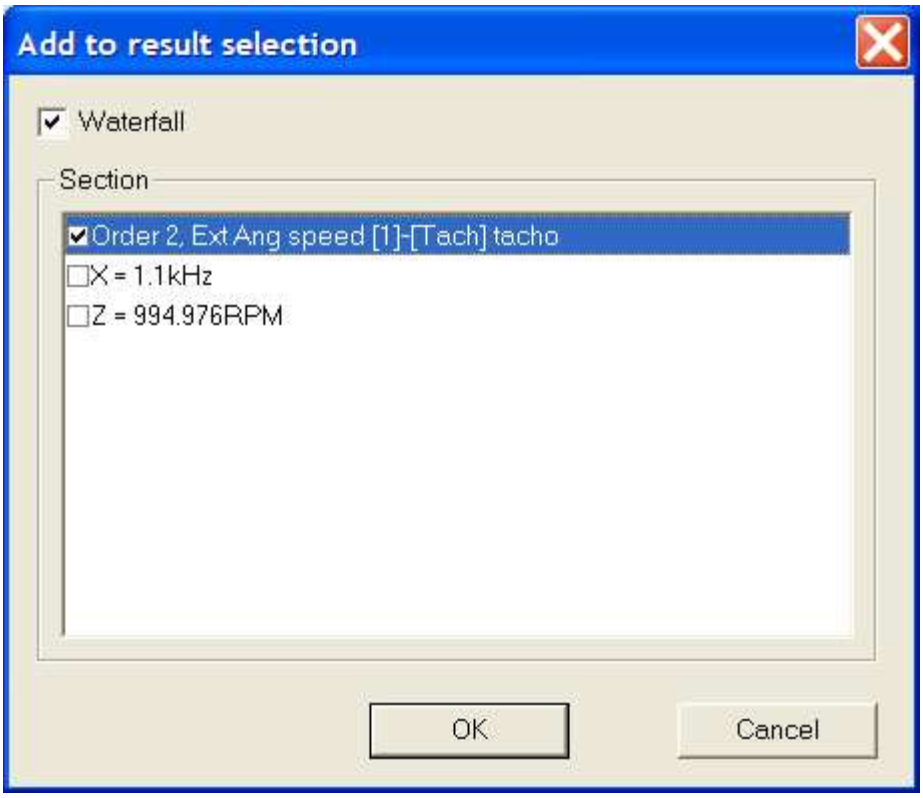




Save results

This section gets you to select and save the waterfall results.

Right click on the waterfall window and select **Add to result selection**. A window that allows you to select the results to be saved is displayed:



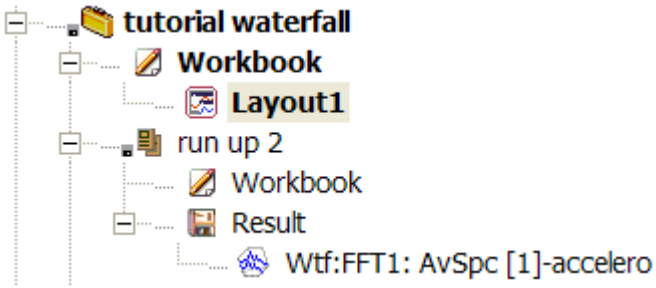
You can select to save:

- The entire Waterfall result

- The extraction section (in this case it is the order 1 section) independently
- The XY section independently
- The YZ section independently

Click on **OK**.

Click on 'Save results' , your results selection will be automatically saved:



Reporting

Introduction

Objectives:

Create your own report templates with the different options implemented in NVGate.
A Report Template is a sheet where you can organize the graphs, marker tables, plug-in analyzer settings and images to be printed out.

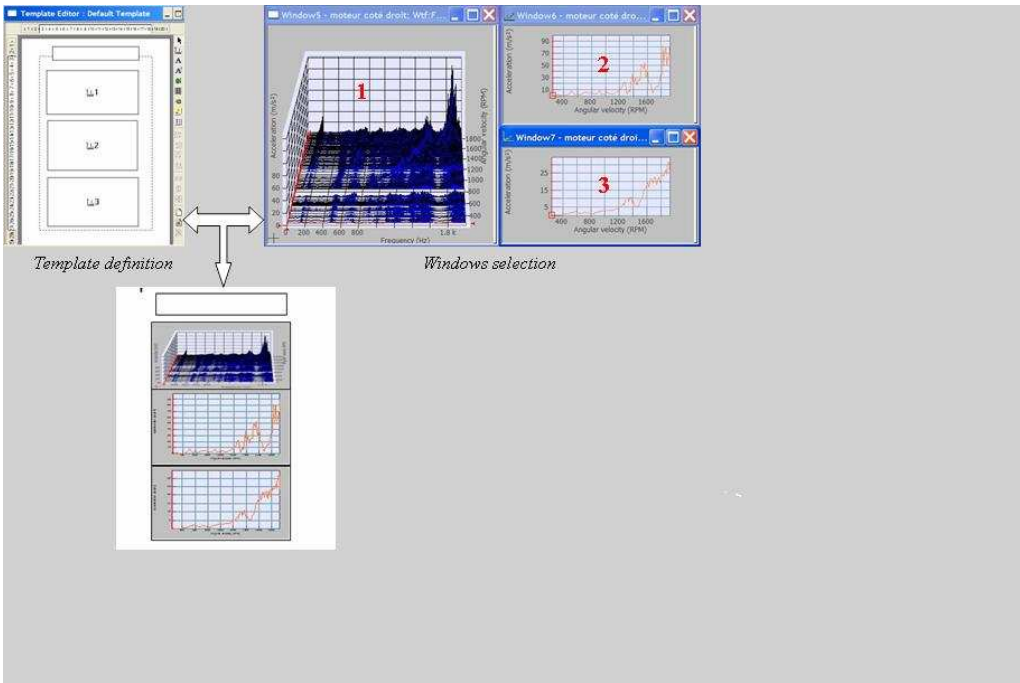
Material

- PC with NVGate

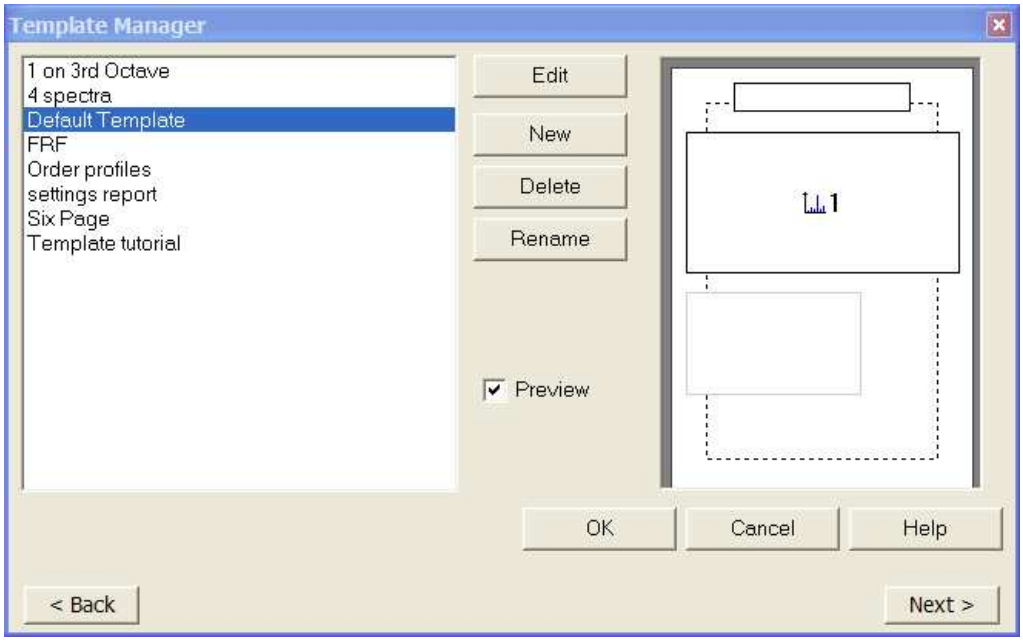
Procedure

Principle

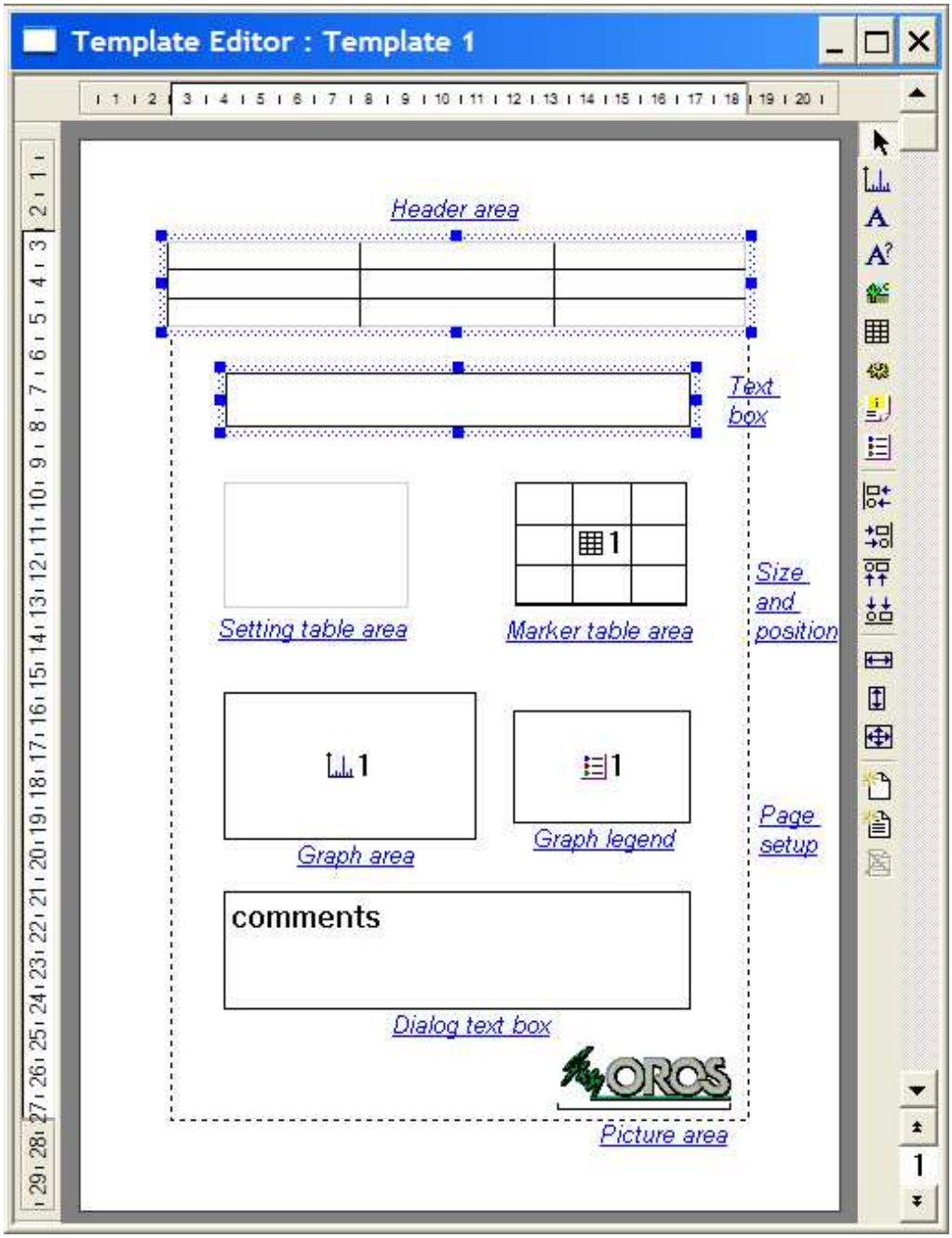
A report can be build in 2 steps: a template definition and a windows selection. In the first step, define the items you want to see in you report. And in a second step, associate the window to
the defined items.



Create a report template



You can edit an existing template in order to modify it (change the text into the text window, the number of graphic windows, etc.) or you can create, delete one or more report templates
The report can be personalized by using different area.



Using the text area

From the Template Editor, click on the **Text area**. A blue frame appears.
Right click on the **Text area**.

Select **Properties** to set the Size/Position of the area and Text.
Click on the **Text** tab (to keep in focus while working outside the "Properties" dialog box, you can click on the thumbtack on the top left corner). You can decide whether or not to display a frame with the "Draw a frame" check box.

When you have typed the text, click outside the window to return to the Template Editor.

Using the dialog box text area

From the Template Editor, click on the **Dialog box text area**. A blue frame appears.
Right click on the **dialog box text area**.
Select **Properties** to set the Size/Position of the area and Text.

- From the **Text** tab, enter the text you want to display in the dialog box as a message to the user when he will print the report (for example, enter "`Comment`").
- From the **Query** tab, enter the default text to be modified by the user (for example, enter "`No problem encountered`").

Note: If you want to write several lines, press **CTRL** + **Return** keys at each end of line.

Inserting a graph in the window area

From the Template Editor, click on the **Window area**. A blue frame appears.
Right click on the **Window area**.
Select **Properties** to set the Size/Position of the area and Graph.

- Click on the **Graph** tab (to keep in focus while working outside the "Properties" dialog box, you can click on the top left corner icon). With the index box, select the number of window you want to insert in the area. You can decide whether to display a frame or not with the "Draw a frame" check box.

Using the marker table area

From the Template Editor, click on the **Marker table area**. A blue frame appears.
Right click on the **Marker table area**.
Select **Properties** to set the Size/Position of the area and Marker Array.

- Click on the **Marker Array** tab (to keep in focus while working outside the "Properties" dialog box, you can click on the top left corner icon). With the

index box, select the number of window you want to insert in the area. You can decide whether to display a frame or not with the "Draw a frame check" box.

Using the setting table area

From the Template Editor, click on the **Setting table area**. A blue frame appears.
Right click on the **Setting table area**.
Select **Properties** to set the Size/Position of the area and settings.

1. Click on the **Settings** tab and click on the thumbtack icon on the top left corner of the window (this is to keep the focus on it). You can decide whether to display a frame or not with the "Draw a frame" check box.
2. From the ASB, select a setting and add it into the setting table area by using drag & drop (the setting will appear in the setting table area properties window).

Inserting images (logo, pictures, etc.) in the picture area

From the Template Editor, click on the picture area icon and draw a window with the pointer.
Click on the **Picture area**. A blue frame appears.
Right click on the **Picture area**.
Select **Properties** to set the Size/Position of the area and Bitmap.

- Click on the **Bitmap** tab (to keep in focus while working outside the "Properties" dialog box, you can click on the top left corner icon) and type the name of the bitmap file you want to insert or browse folders by clicking on ...

Then click outside the window to return to the Template Editor.

Using the header area

From the Template Editor, click on the **Header area**. A blue frame appears.
Right click on the **Header area**.
Select **Properties** to set the Size/Position of the area and Header Info.

- Click on the **Header Info** tab (to keep in focus while working outside the "Properties" dialog box, you can click on the thumbtack on the top left corner). You can decide whether to display a frame or not with the "Draw a frame" check box.

Using the graph legend area

From the Template Editor, click on the **Graph legend area**. A blue frame appears.
Right click on the **Legend area**.
Select **Properties** to set the Size/Position of the area and Legend contents.

- Click on the **Legend** tab (to keep in focus while working outside the

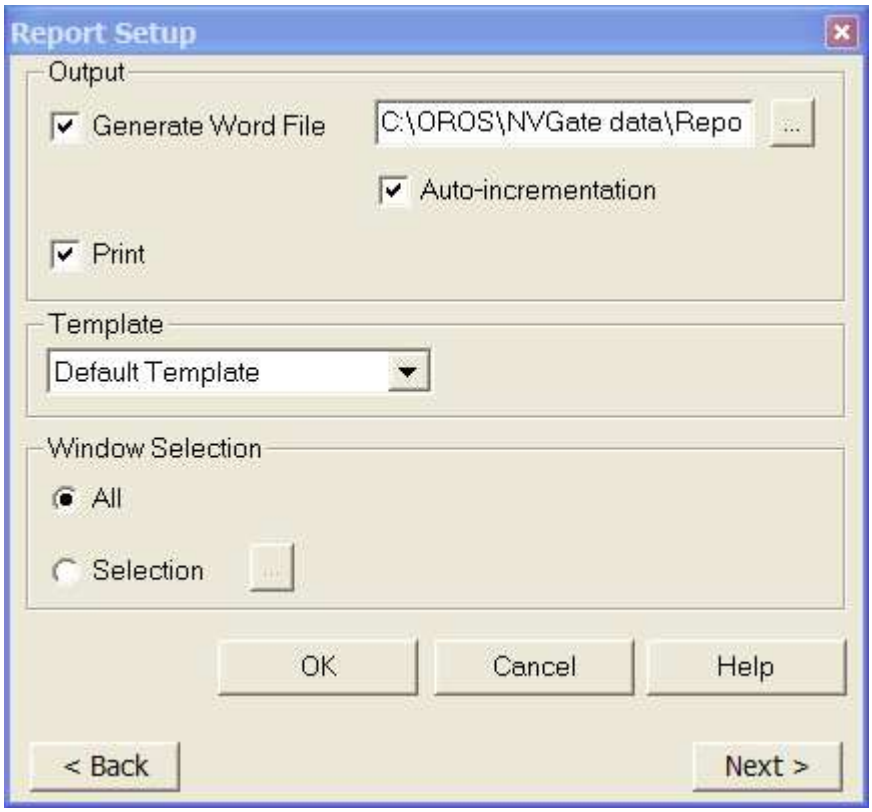
"Properties" dialog box, you can click on the thumbtack on the top left corner). The index box is used to assign the window number to the legend area. You can decide whether or not to display a frame with the "Draw a frame" check box.

Adjusting the Size and Position of the areas

From the Template Editor, right click on a drawn area and click on **Properties**. Select the **Size/Position** tab (the following window is displayed).
Enter the values in the corresponding boxes.

Report setup

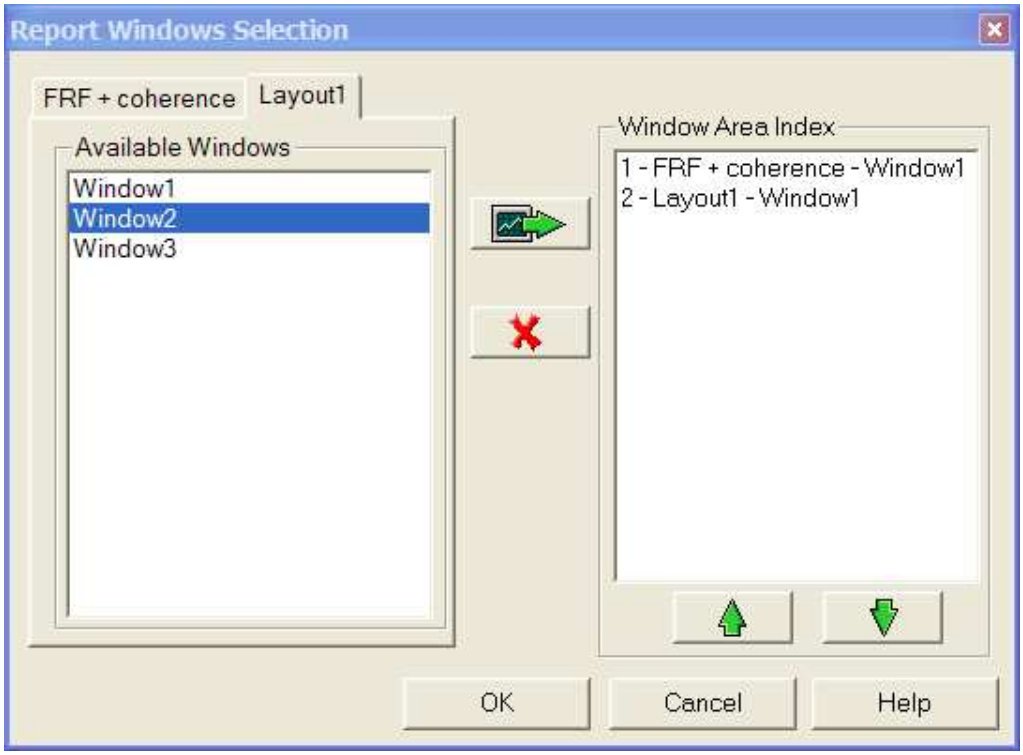
This function allows to print directly the considered template or generate a Word File. There is the possibility to export a template into a Word file so that you can see and manage what has been inserted in the different template areas.



Check **Generate Word File**. The template is then created in a default directory. You can change the destination by clicking on the browse button. If you want to generate several Word files without creating each time a new name, check the Auto-incrementation box.

Select the template to be generated in the Word File with the drop-down menu in the Template part.

Manage the Window Selection: check **All** to put window1 in area 1 of the template, window2 in area 2, etc. Check **Selection** and click on the browse button to choose another arrangement. The following window is displayed:



Select the layout from which you want to add the windows (in the Available Windows column, the left one).
The "Window area index" column contains the windows you want to be displayed in the report.
Select the windows in the "Available windows" column. You can arrange the windows using the 2 green arrows at the bottom of the "Position in Template" column.

Click on OK

From the **File** menu, select **Print Report**., a Word File is generated.

Post analysis

Introduction

Objectives:

his lesson explains how to post-process recorded signal(s) files, (FFT analysis, 1/n Octave, re-recording) using the Player module and the Post-analysis mode.
It is recommended that you go through the Acquisition and recording before starting this part.
In this session, we will obtain the spectrum of a selected signal file segment.

Equipment

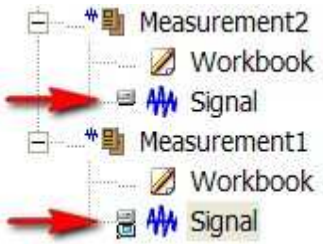
- Analyzer
- PC with NVGate
- Ethernet link
- 2 transducers. Here one accelerometer and one microphone will be used.

Procedure

The Post analysis mode operates as the On-line acquisition but the signals are provided by the Player. The Front-end and Output signals are not available anymore. The only source is the Player that is able to provide recorded time signals. The Player tracks are then connectable to the plug-in analyzer channels.

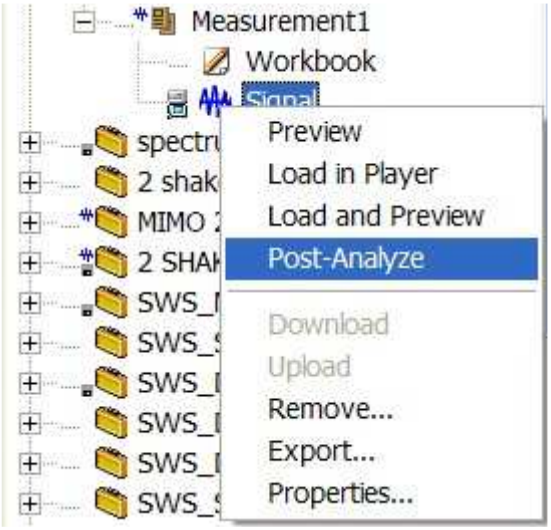
From the **Channels** menu select **Post-analysis** . The Post-analysis warning appears, click on **OK**.

Project Manager information



Note: If you start NVGate in office mode, the signal you want to use in post-analysis have to be saved on the PC. If you start NVGate in connected mode, the signal can be saved either on the analyzer or on the PC.

Player



By right clicking on the signal you want to analyze, then you can **preview**, **load**, **load & preview** or **Post-Analyze** it.

- **Preview** let you visualize the recorded signal envelope. Contents of signal files can be visualized in a high-speed preview even if you are manipulating huge file. This new technology allows multiple files preview at the same time. Very large domain record (several Gigabytes) can be previewed in a click, even if the hardware is not connected. Means that the files can be located on the hardware OR3x disk and visualized (not post-analyzed) on the PC without request of downloading, or Hardware connection.
- **Load in player** makes your signal available for post-analysis without viewing it.
- **Load & Preview** is a combination that let you see the envelope of the recorded signal you are post-analyzing.
- **Post analyze** is a combination that let you see the envelope of the recorded signal you are post-analyzed, switch to post-analysis and start the setup menu for post-analysis.

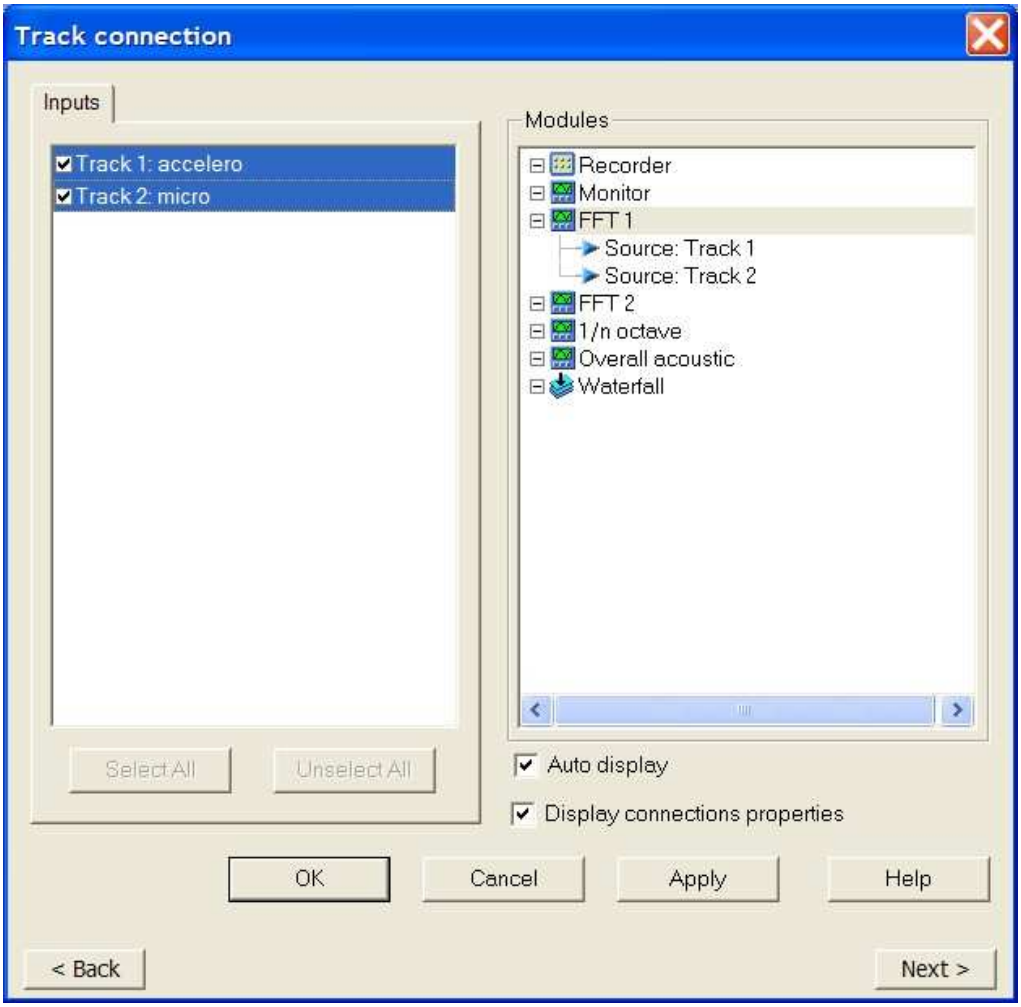
First step: playback the signal

Click on preview.
At this stage you can replay the recorded signal in the player by clicking on the playback button below the window.

The cursor follows the played back signal.

Second step: Post-Analysis

Click on Post-Analysis
added to the replay function, the 'tracks connection' window opened.



This window allows to associate the different tracks of your signal to any plug-ins.

By clicking on next, you will find **the same functions than in on-line mode**. You can obtain the same results than ones calculated in the **First lesson** by using the same process.
The unique difference is the use of tracks instead of channels.

Tip: It is possible to load successively several signal files in the Player and post process them in the same plug-in analyzer keeping the settings.

After setup, Click on the **Run** button to start post processing. Wait until the entire segment is processed.

While setting the **Start** and **Stop offset** the cursors move automatically on the signal file window. You can modify Start and Stop offset directly from the graph by dragging them to the selected time with the cursor mode

Tip: You can use the bottom slider to move the start and stop offset simultaneously.

Signal is on the analyzer

Signal is on the analyzer and on the PC

Synchronous order analysis

Introduction

Objectives:

In this tutorial, we will show the basics of Noise & Vibration analysis on rotating machines with NVGate software

Order analysis allows to synchronize the results with tachometer pulses so the analysis is free of random interference.
In this tutorial, the plug-in **Synchronous Order Analysis (SOA)** with Waterfall order profile will be used.

Equipment

- OR3X Analyzer
- PC with NVGate
- Ethernet link
- 1 accelerometer
- rotating machine with integrated tachometer

Note: Rotating Mode must be selected

Procedure for synchronous Order Analysis

Channels connection and properties

The accelerometer is placed on the structure of the rotating machine and connected to Input 1
Tachometer is connected to Ext Sync 1.

Select Ext Sync 1 and connect Input1 to the SOA plug-in and fill in the properties

Tachometers settings

Drag&drop Ext Sync 1 to the SOA plug-in. Click on next anf fill in the tachometer settings especially pulse per revolution, min speed and max speed

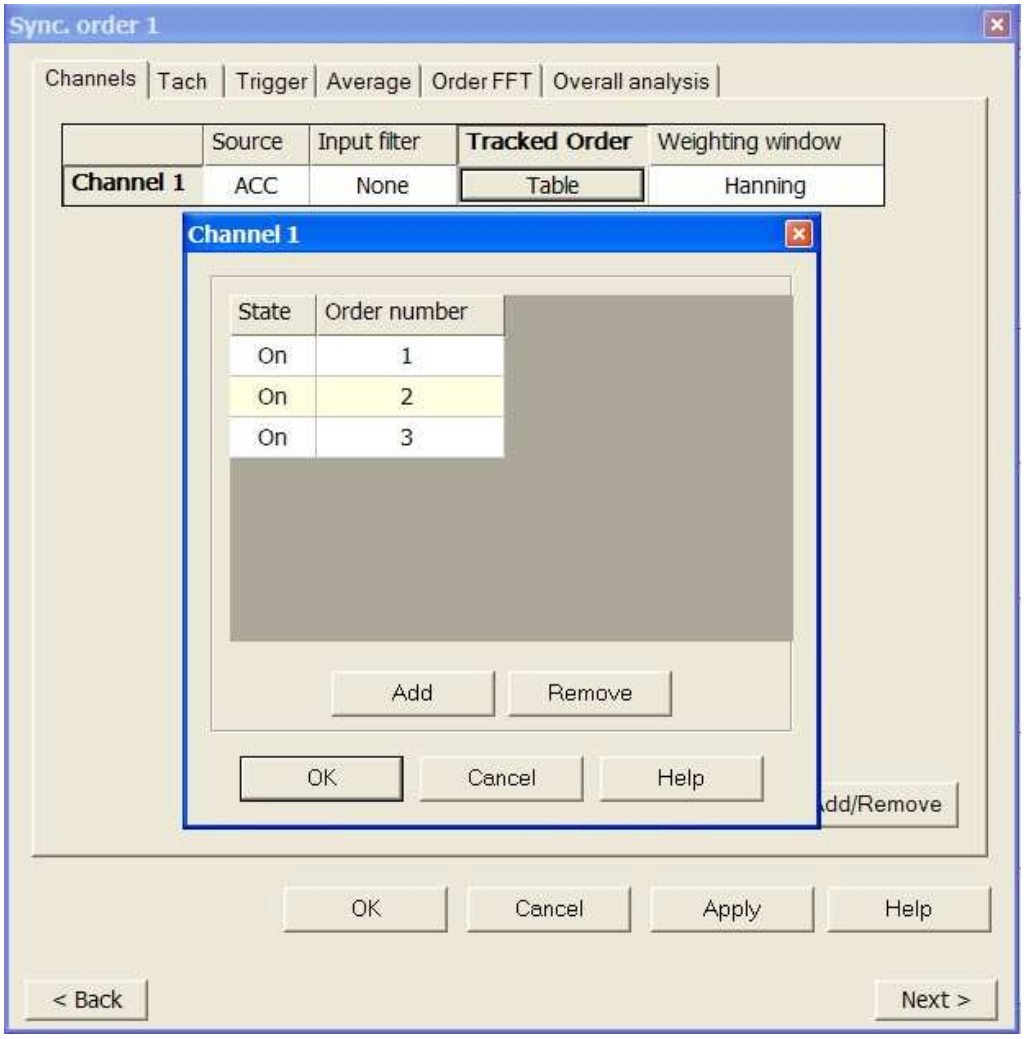
Event connection

We need to create an event to fill up the waterfall;
In the part delta RPM, drag&drop Ext Sync to the waterfall and fill in the

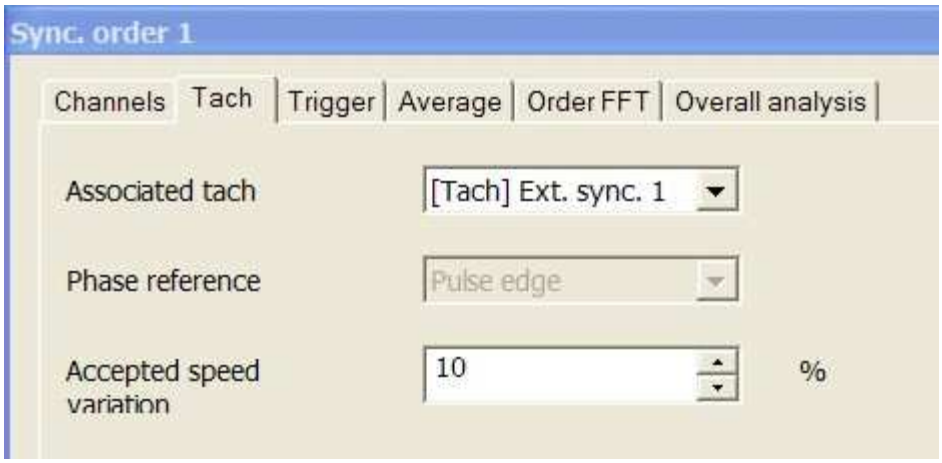
event properties: delta velocity and slope.
If you define delta rpm=50rpm, the waterfall will be fed with one new SOA spectrum.

SOA

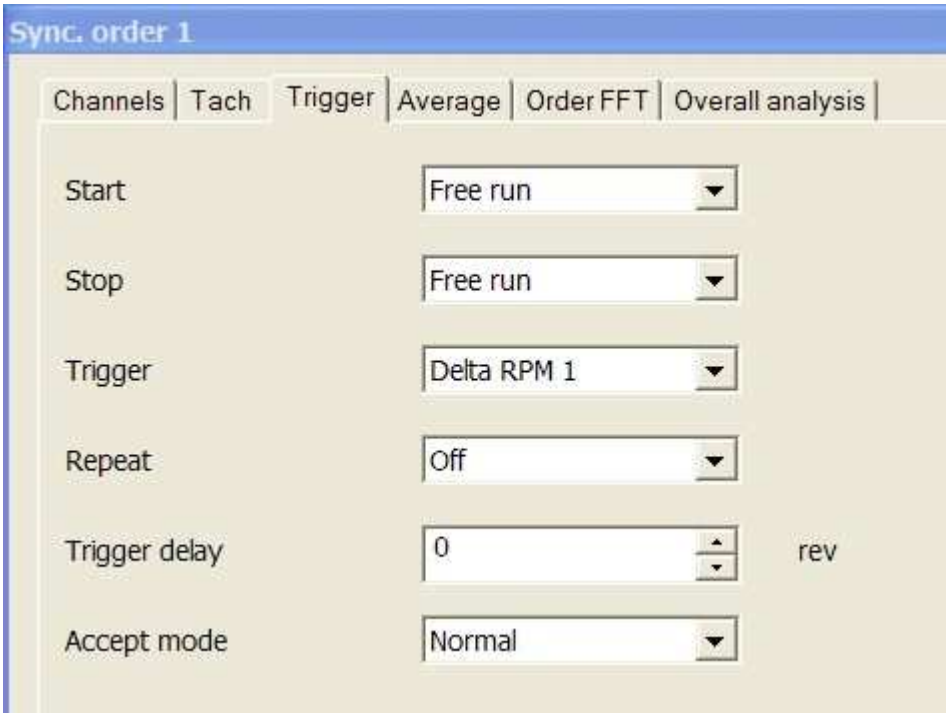
Channels: in this window select the orders you want to track.



Tach: select Ext Sync 1 as associated tach

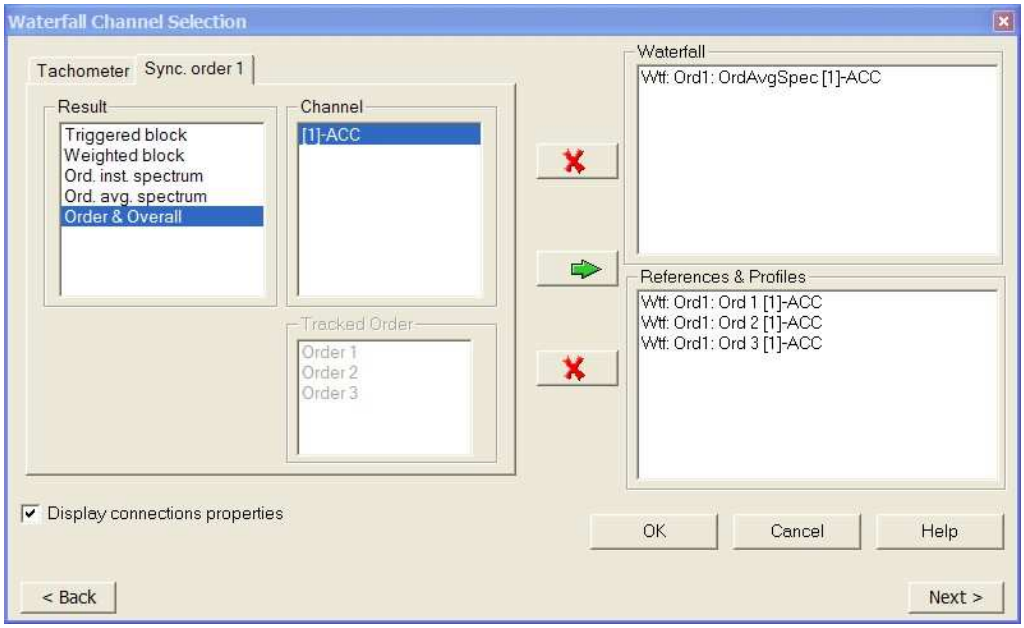


Trigger: check that the event 'Delta RPM 1' is defined as the trigger.



Waterfall

Add Order&overall to the references&profiles and order average spectrum to the waterfall

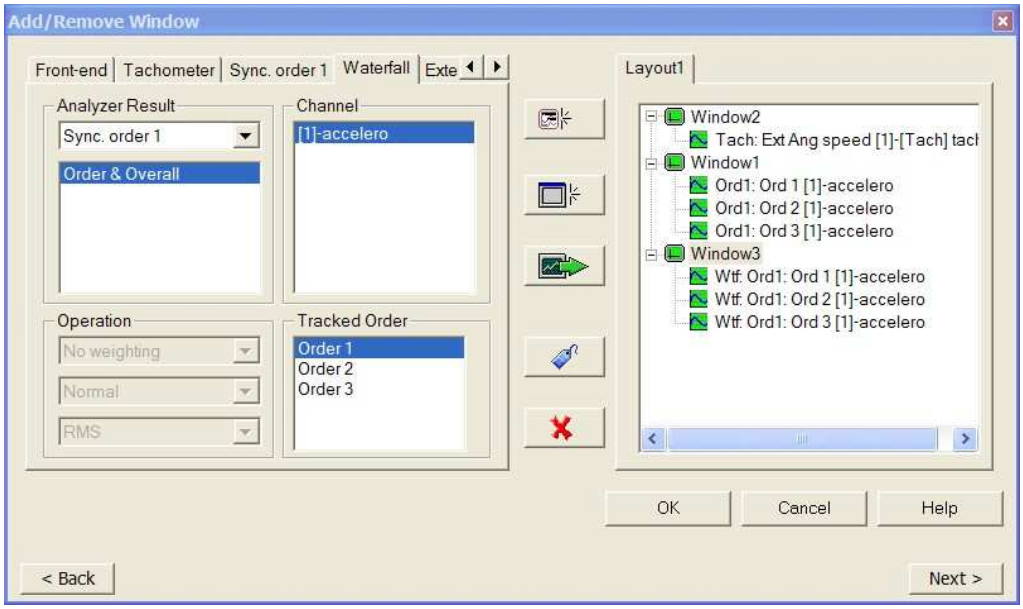


Create window

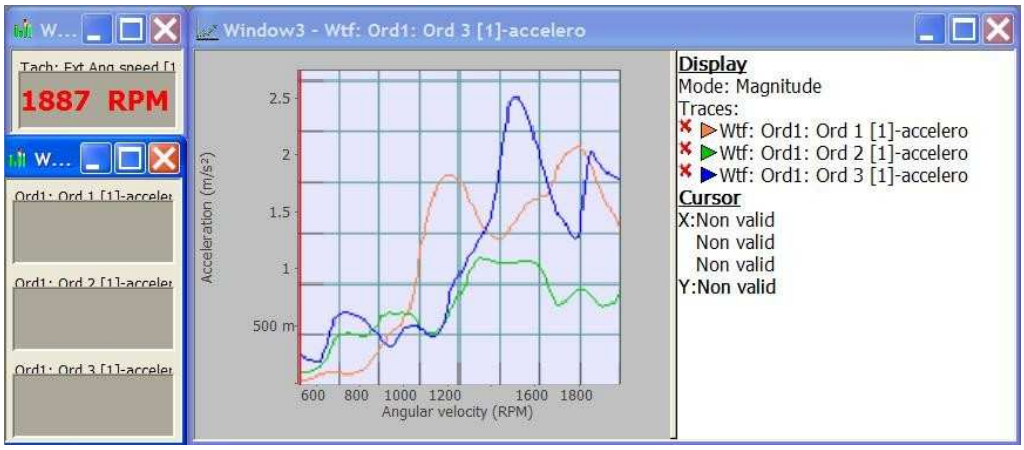
Select the Waterfall tab. In the Analyzer Result area, select Sync. order1. Select Order & Overall result then the channel where the accelerometer is

connected (in this case: Input 1). Select the orders in the Tracked order area. Click on "add a new trace" button.

By selecting the Sync. order1 tab and Order & Overall, the level of the tracked order will be displayed.



Click on the run button and run up the rotating machine speed in order to acquire slices.
The following results will be obtained.



Order tracking with FFT

Introduction

Objectives:

In this tutorial, we will show the basics of Noise & Vibration analysis on rotating machines with NVGate software

Order analysis allows to synchronize the results with tachometer pulses so the analysis is free of random interference.
There are three different methods to process order analysis:

- Extract order from FFT Waterfall
- Using the **Constant Band Tracking (CBT)** from the FFT plug-in analyzer with Waterfall order profile
- Using the **Synchronous Order Analysis (SOA)** plug-in analyzer with Waterfall order profile

Material

- OR3X Analyzer
- PC with NVGate
- Ethernet link
- 1 accelerometer
- rotating machine with integrated tachometer

Procedure using FFT waterfall and Constant Band Tracking

Channels connection and properties

The accelerometer is placed on the structure of the rotating machine and connected to Input 1
Tachometer is connected to Ext Sync 1.
Connect these 2 channels in NVGate and fill in the channels connection properties.
Accelerometer label will be 'accelero' and tachometer label will be 'tacho'.

Tachometers settings

After selecting Ext Sync 1 fill in the channels connection properties,

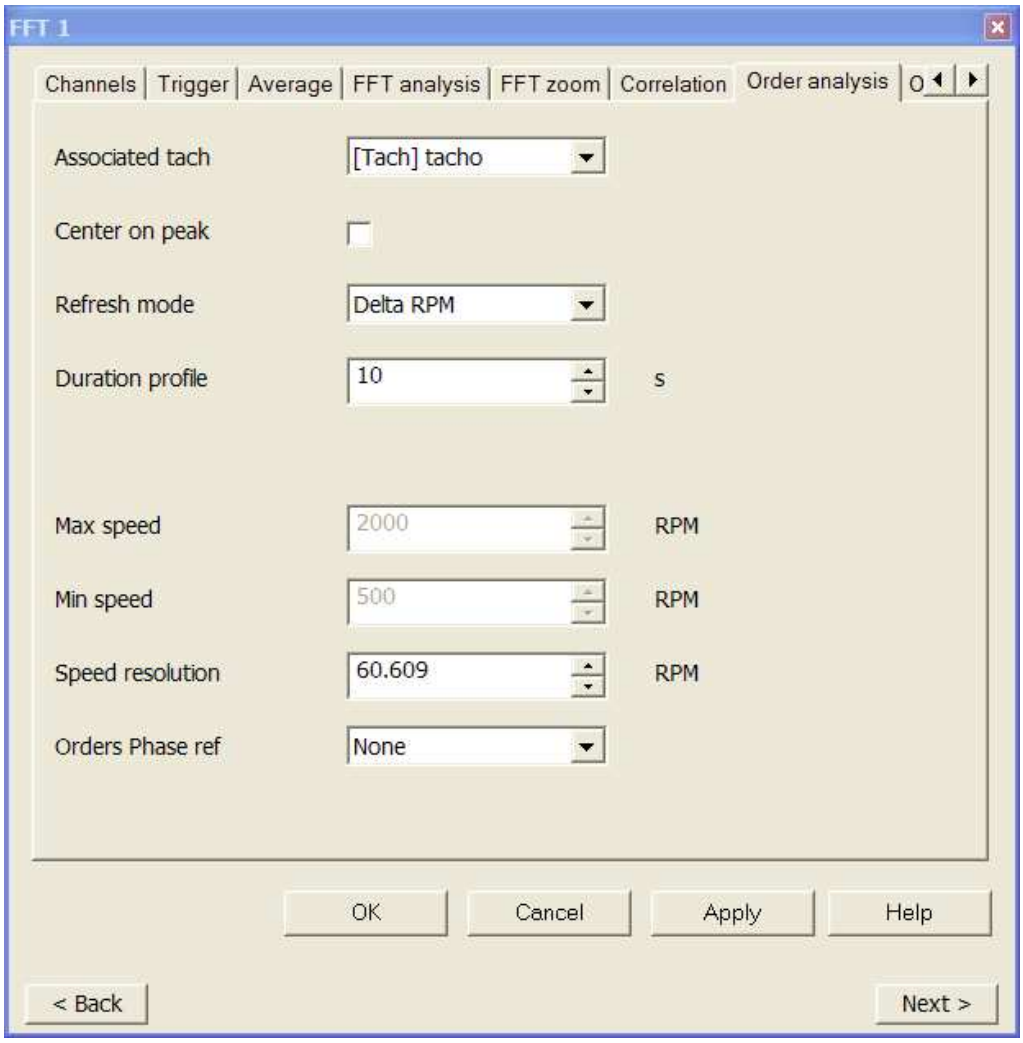
Note: 'tach' must be set to 'on'

Event connection

We need to create an event to fill up the waterfall;
In the part delta RPM, drag&drop Ext Sync to the waterfall and fill in the event properties: delta velocity and slope.
If you define delta rpm=50rpm, the waterfall will be fed with one new FFT spectrum.

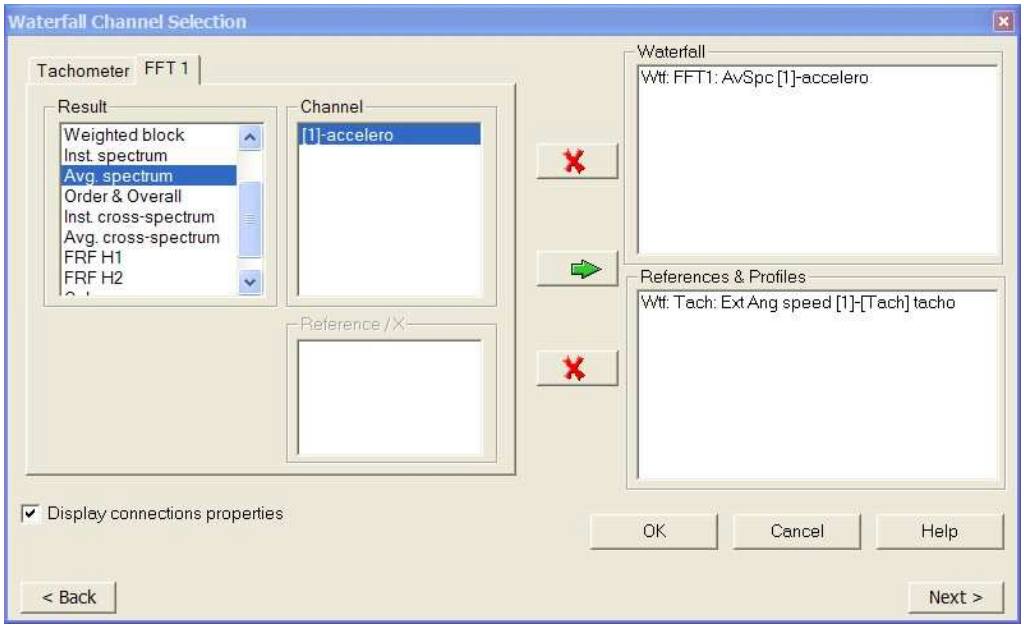
FFT properties

Define FFT parameters, especially the frequency range can be set to 2kHz.
In the window 'Order analysis', connect 'tacho' to associated tach. and click OK.



Waterfall definition

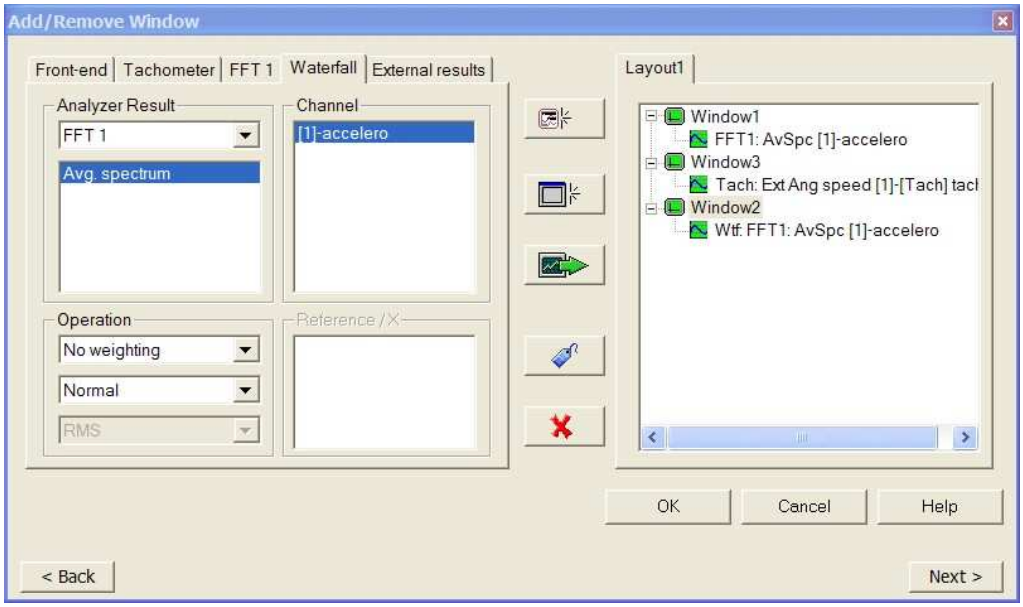
Drag and drop Ext.Tach Angular Speed on References&Profiles and display accelerometers average spectra in the waterfall.



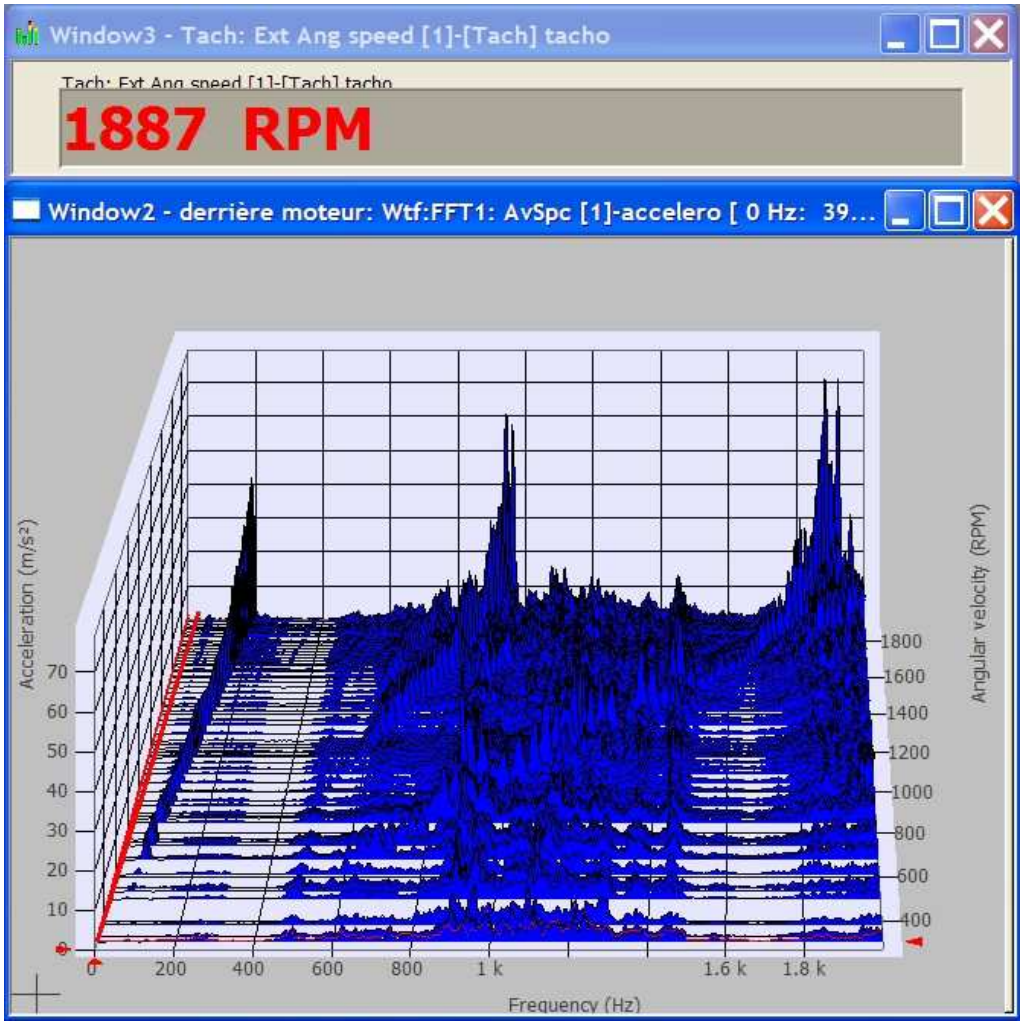
Check the waterfall properties, especially
- Trigger= =delta RPM 1,
- Mode: one shot
- Number of slices= (Max speed-Min speed)/delta rpm

Create windows

Display the External Angular Speed and the Waterfall FFT average spectrum of the accelerometer.

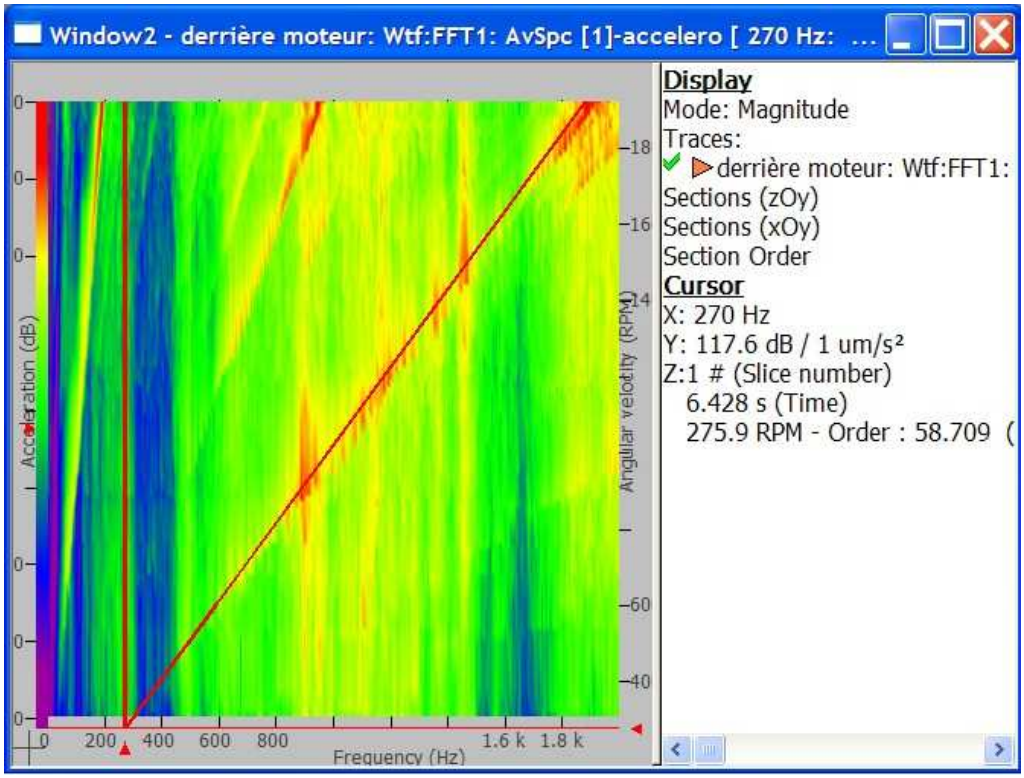


Press Run and Run up the machine from min speed to max speed. The waterfall will be filled in with average spectra calculated every delta RPM.



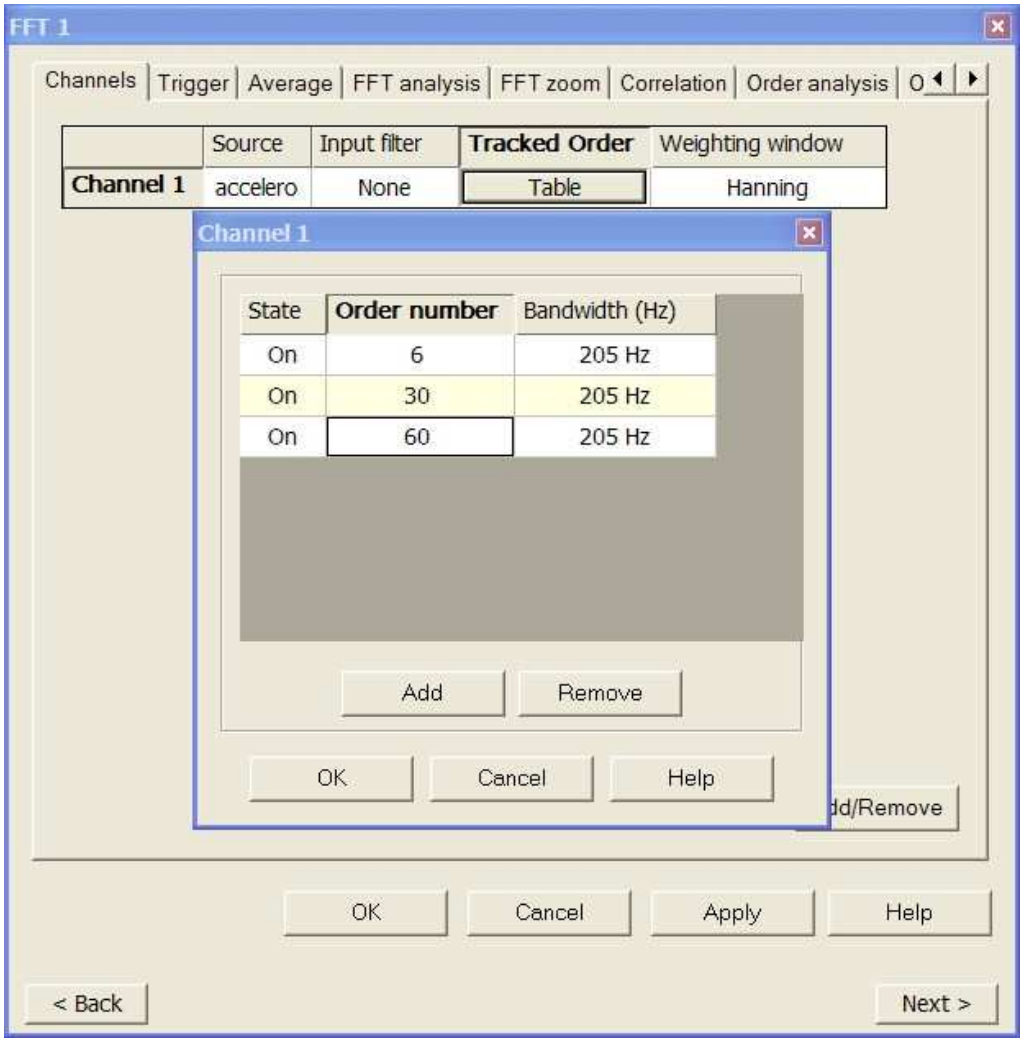
Right click on the graph and select view type/ FreqTime colormap.
Right click on the graph and select Y Scale/dB.

The waterfall will appear as below:

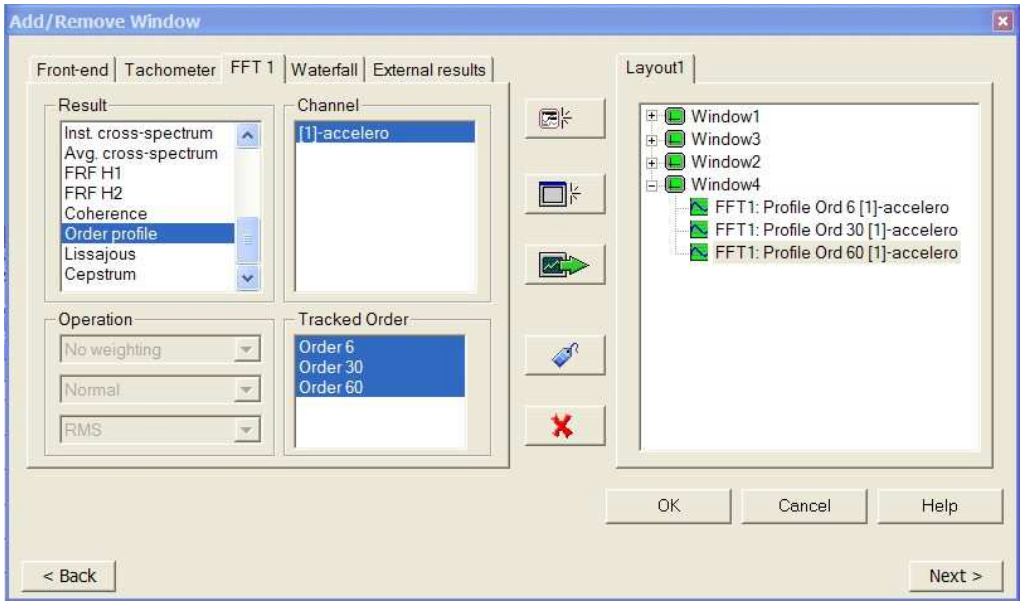


All the warerfall functions are available. To know more about these functions see the Waterfall lesson.
Different orders can be observed. Note the orders value with the cursors

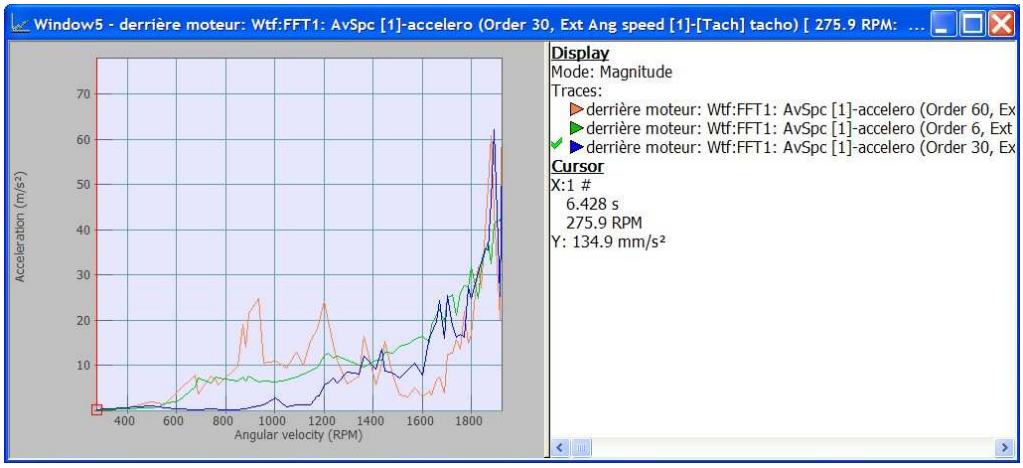
Click on the FFT icon in the toolbar and add the different observed orders.



Click on add window and add order profiles



Press Run and run up the machine. The following results will be displayed.


Window5 - derrière moteur: Wtf:FFT1: AvSpc [1]-accelero (Order 30, Ext Ang speed [1]-[Tach] tacho) [275.9 RPM: ...
Acceleration (m/s²)
70
60
50
40
30
20
10
400
600
800
1000
1200
1400
1600
1800
Angular velocity (RPM)
Display
Mode: Magnitude
Traces:
derrière moteur: Wtf:FFT1: AvSpc [1]-accelero (Order 60, Ex
derrière moteur: Wtf:FFT1: AvSpc [1]-accelero (Order 6, Ext
derrière moteur: Wtf:FFT1: AvSpc [1]-accelero (Order 30, Ex
Cursor
X:1 #
6.428 s
275.9 RPM
Y: 134.9 mm/s²

Structural analysis

Introduction

Objectives:

In this tutorial, we will show the basic operations for structural tests, to do, for example, modal analysis.

Through the different steps of a typical procedure, the useful functions for these kind of tests will be described.

Material

- OR3X Analyzer
- PC with NVGate
- Ethernet link
- 1 accelerometer
- 1 impact hammer

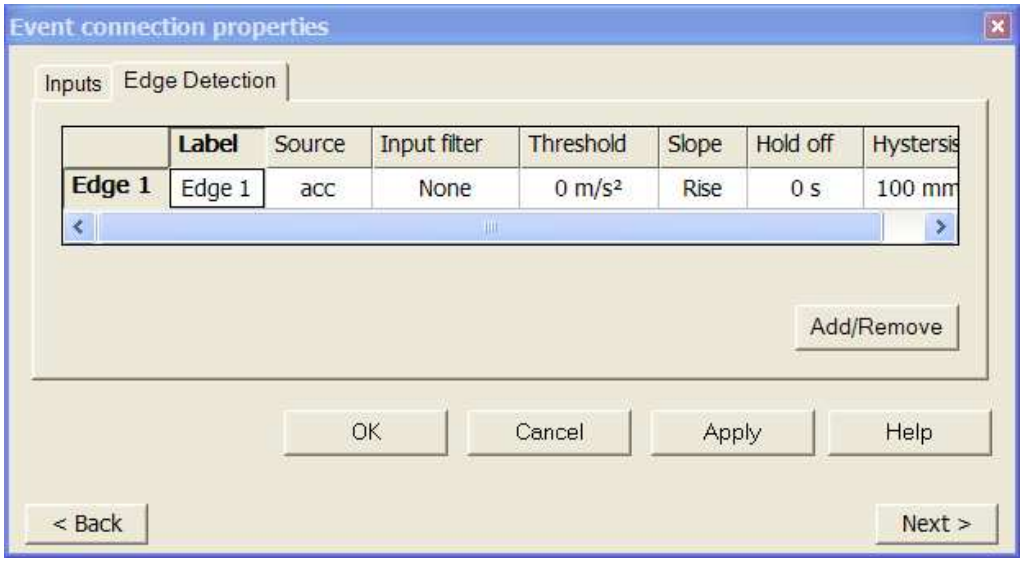
Procedure

Channels connections and properties

Connect the Impact hammer to input1 and the accelerometer to input2.
Open NVGate software and create a new project.
Connect Input 1 and Input 2 to the FFT1.
Fill in both channels properties.

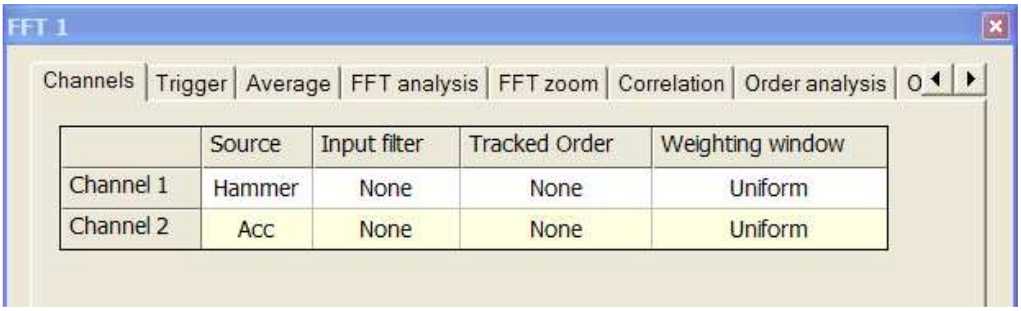
Event connection and properties

We will use the hammer signal to trig the FFT;
So create an event 'edge detection' on Input1 and connect it to FFT1.
Fill in the event properties:

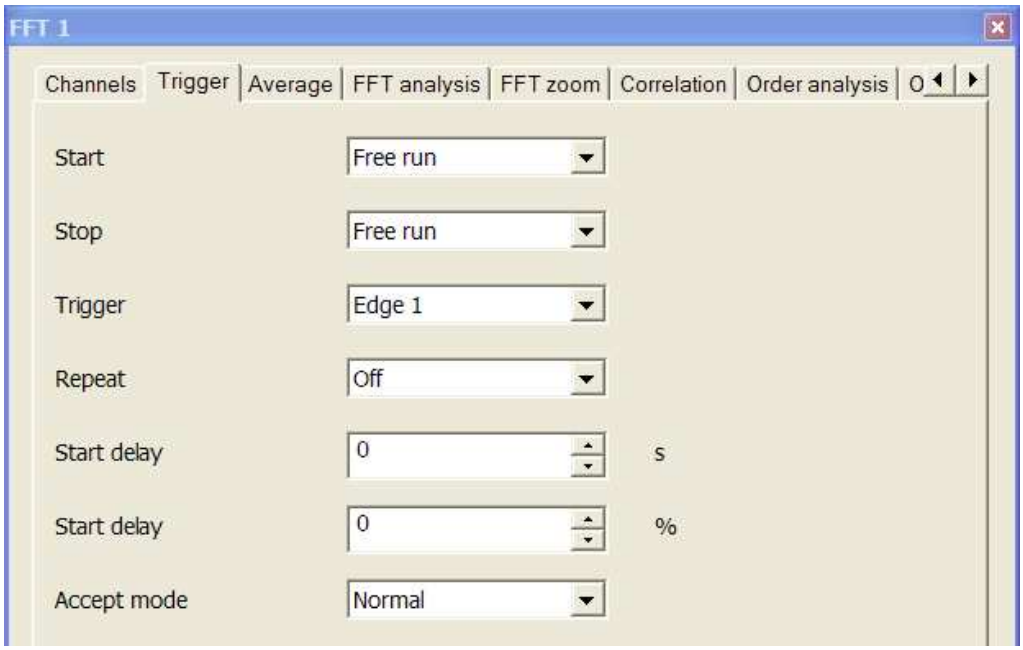


FFT Parameters

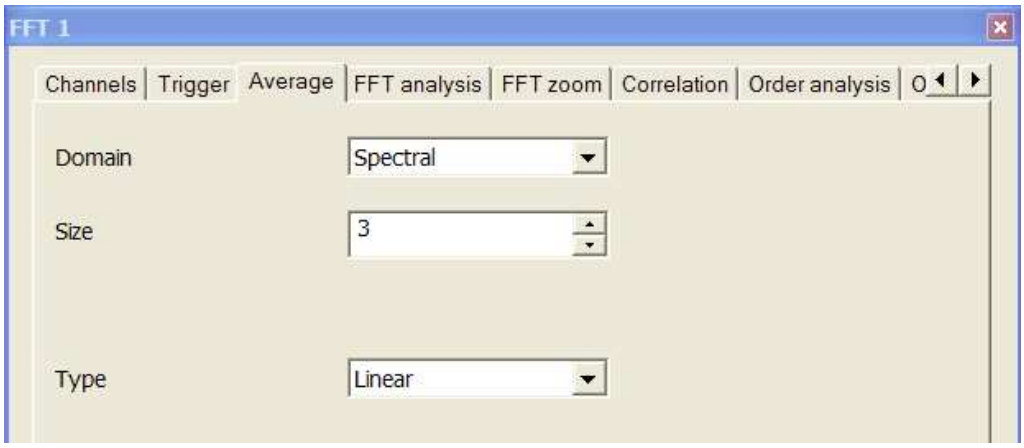
Channels:



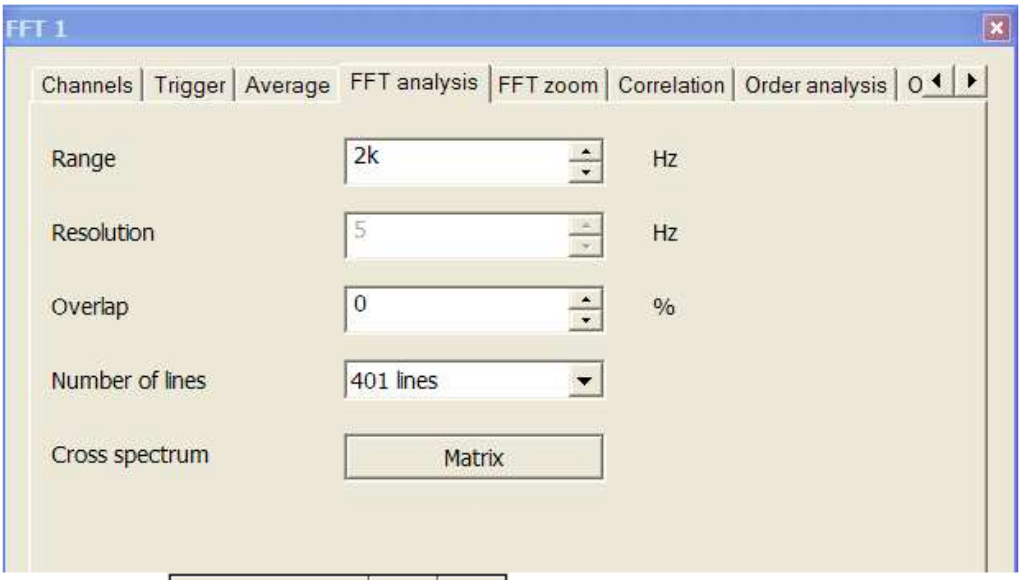
Trigger :
Select 'Edge1' for the trigger

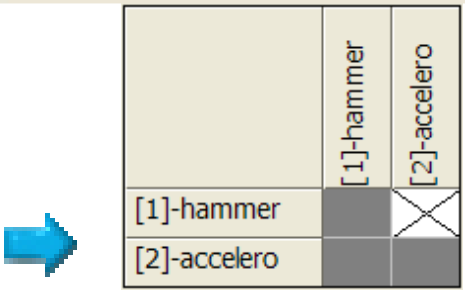


Average:
Fix the number of average to 3 and the average type to linear.



FFT analysis:
Reduce the frequency Range to 2kHz and activate cross spectrum between the hammer and the accelerometer.





Windows

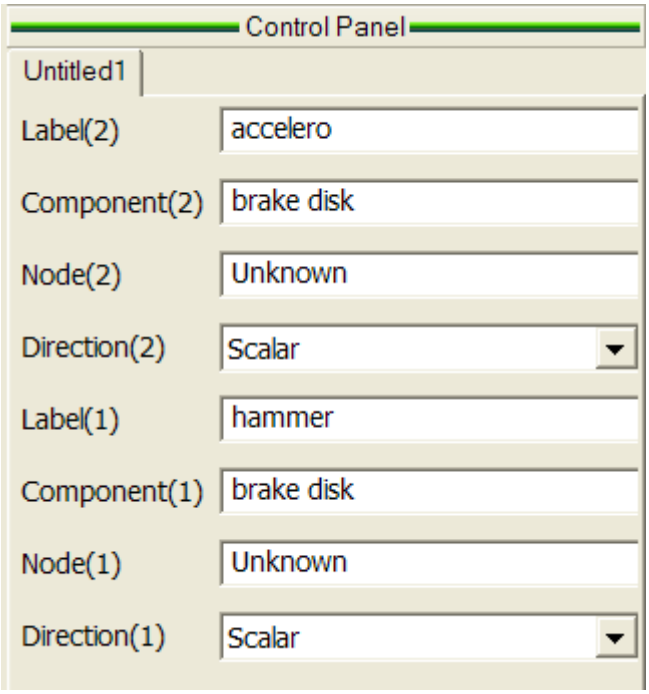
Create 2 layouts with:
- Hammer and accelerometer trigger blocks+ hammer and accelerometer average spectra.
- FRF H1 between accelerometer and hammer + coherence.

Save

Save : FRF H1 and coherence.

Project personalisation

For a modal acquisition, it's interesting to have information on the system's degrees of freedom.
So, it's practical to add this info to the control panel.



By this way you can modify easily the node and direction during the test. Typically, the accelerometer is fixed and the hammer is roving.

Adjust parameters

Press Run and impact the structure with the hammer.

By drag and Drop the threshold on the hammer trigger block , you can position it regarding the impact shape.

Before calculating FRF, it's important to adjust some parameters in order to obtain good data.
Indeed, to avoid leakage problem, the trigger block value at the beginning and at thTe end of the block must be 0.

- To obtain 0 at the beginning:
  Add a start delay in the window FFT1/Trigger
- To obtain 0 at the end:
  Increase the trigger block size by reducing the frequency range or increasing

the number of lines.
If it's not sufficient, add a weighting window 'response' to force the signal to 0.

At this stage, all the parameters are correct, the obtained data are good and can be use for a comprehensive analysis.

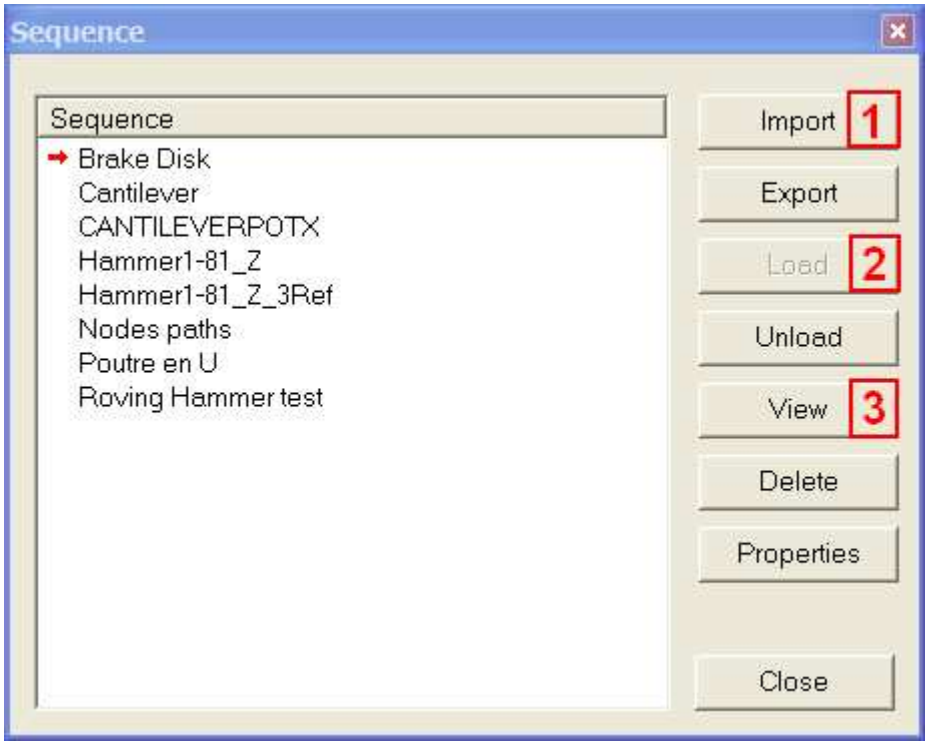
Acquisition optimization

The sequencer allows to realize quickly an acquisition by incrementing automatically the degrees of freedom.
Use a sequence implies:

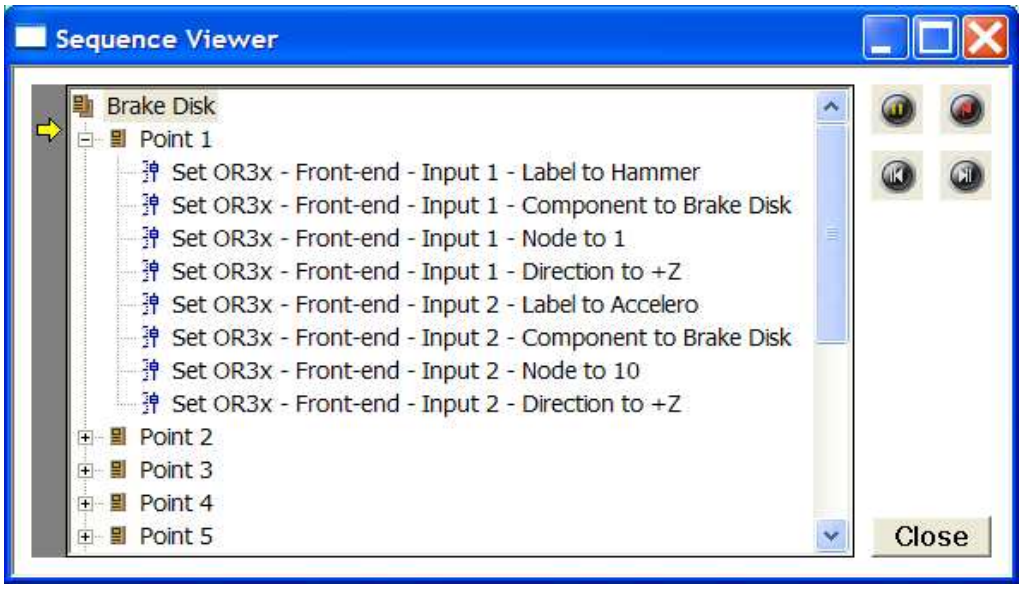
- create the sequence in an excel file
  These kind of files are stored in NVGate data/Sequences.
  The picture below represent a typical sequence. It describes the different steps of a measurement.

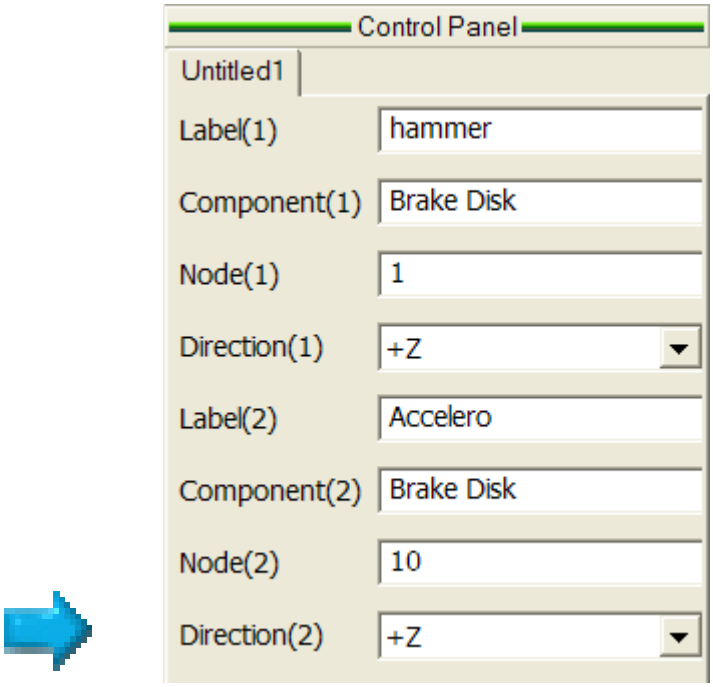
Brake Disk					
		Label	Component	Node	Direction
Point 1	Input1	Hammer	Disk	1	+Z
	Input2	Accel Z	Disk	10	+Z
Point 2	Input1	Hammer	Disk	2	+Z
Point 3	Input1	Hammer	Disk	3	+Z
Point 4	Input1	Hammer	Disk	4	+Z
Point 5	Input1	Hammer	Disk	5	+Z
Point 6	Input1	Hammer	Disk	6	+Z
Point 7	Input1	Hammer	Disk	7	+Z

- Import the sequence in the project
  By clicking on 'sequence' in the 'Tools' menu, the window 'sequence' allow to select the considered file.



- Sequence visualization and use





The sequence viewer allows to visualize the different steps of the sequence. Start the sequence and run the measurement. The information contained in the sequence are automatically updated. See the control panel.

Note: the yellow arrow indicates the position in the measurement process. You can jump manually from one point to another by right clinking on the point you want to jump.

Tip: In 'Save setup', select 'names from sequence' to name automatically each measurement.

Acoustic analysis

Introduction

Objectives:

In this tutorial, we will show the basic operations for structural tests, to do, for example, modal analysis.

Through the different steps of a typical procedure, the useful functions for these kind of tests will be described.

Equipment

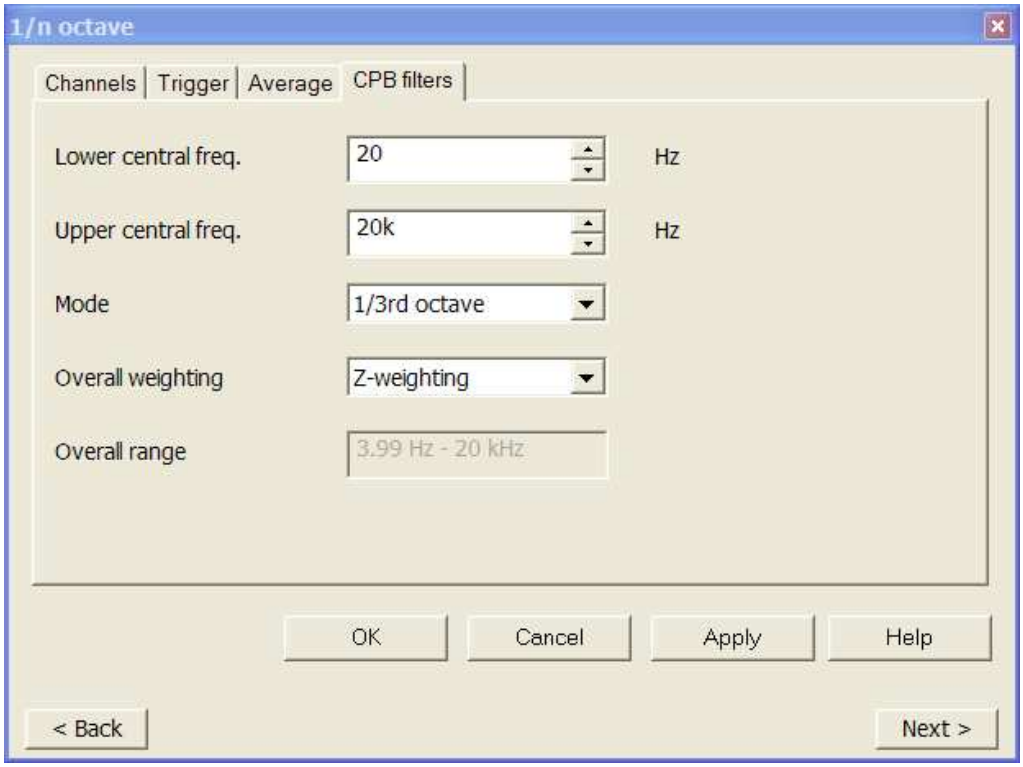
- OR3X Analyzer
- PC with NVGate
- Ethernet link
- 1 microphone

Procedure

Channels connections and properties

A microphone is connected to input1.
In the 'channels connections' window, drag and drop Input 1 on 1/n octave and overall acoustic.
Click on next and fill in the channel properties with the microphone characteristics.

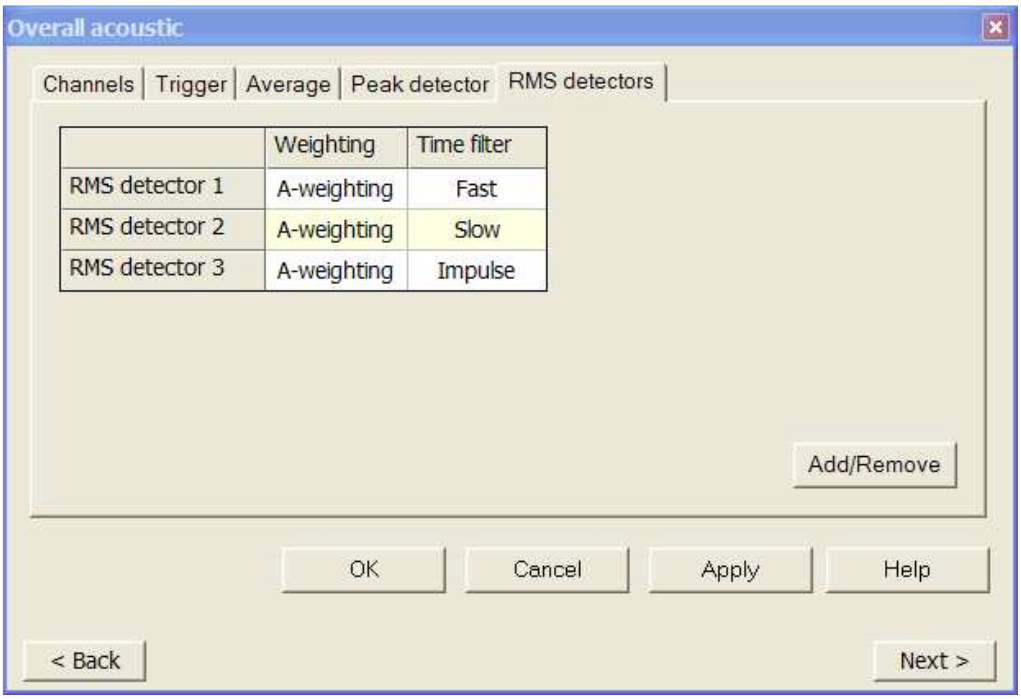
Octave



From the CPB filters allowing to define frequency range and resolution, select 1/3rd octave for mode setting.

Set the Average Type on Impulse.

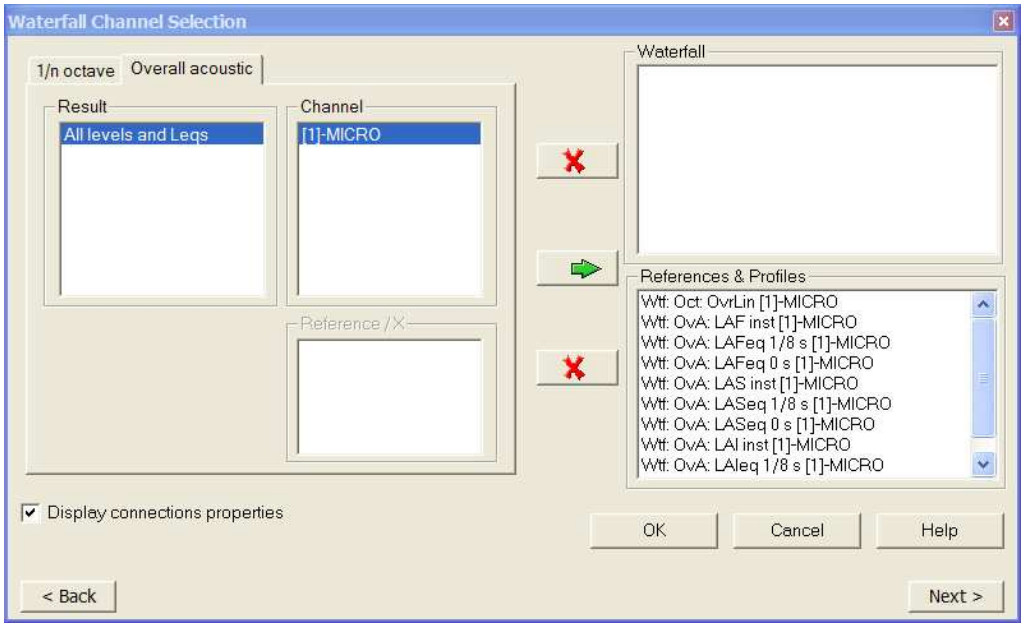
Overall acoustic



Waterfall

Select the 1/n Octave tab then overall level. Click on the add result button. The selected result will be displayed in the References & Profiles.

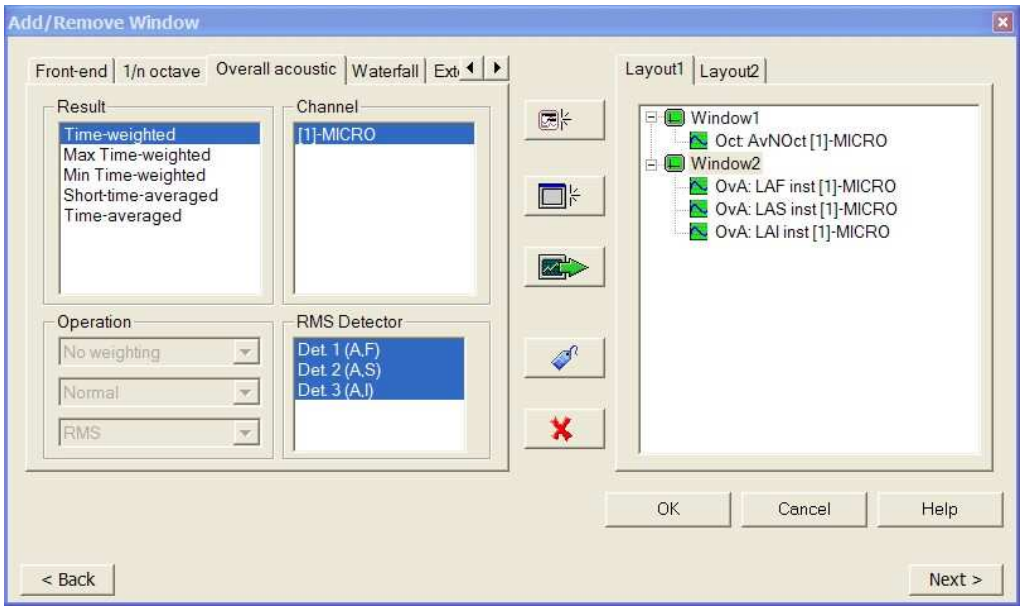
Select the Overall acoustic tab then All levels and Leqs. Click on the add result button. The selected result will be displayed in the References & Profiles.



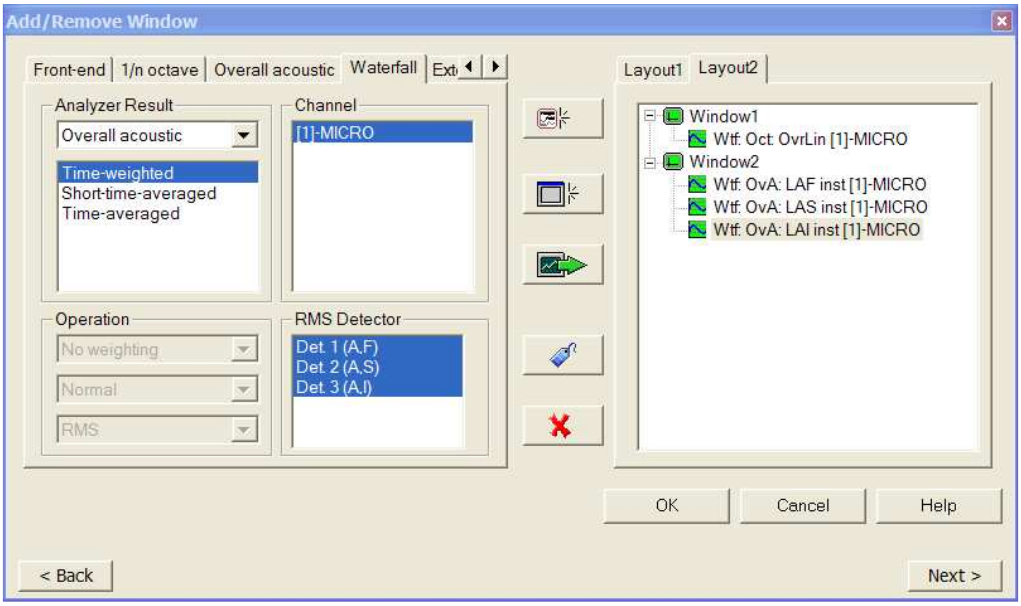
If you need more information, see the tutorial 'waterfall'

Create windows

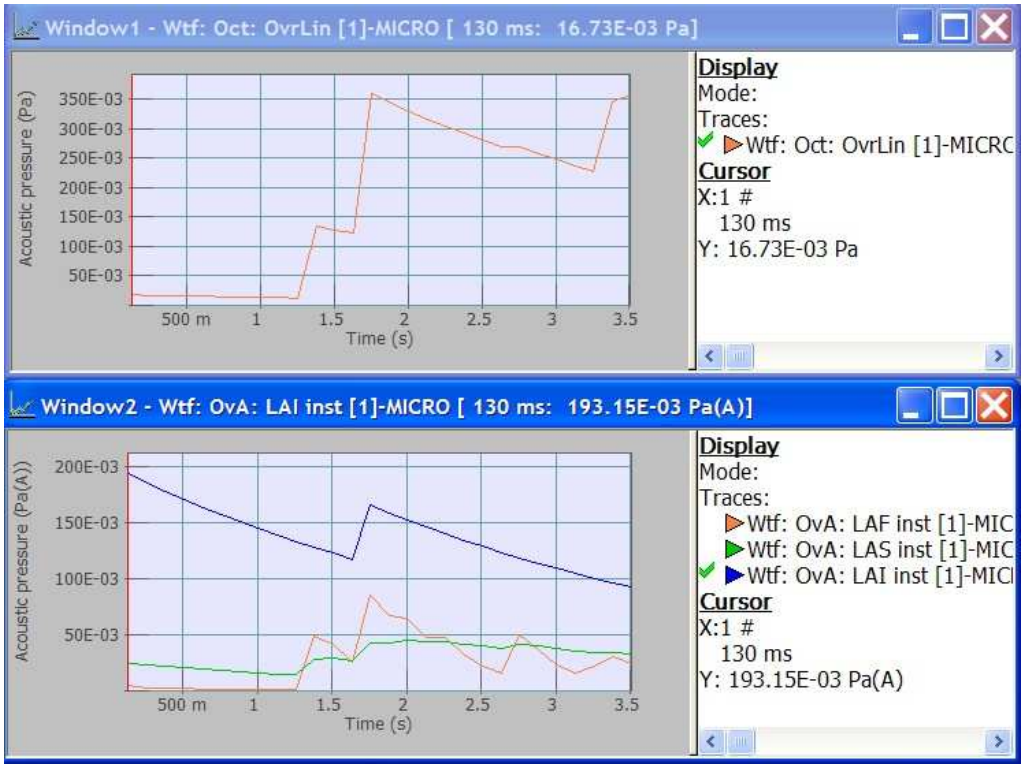
In a first layout:
The 1/n Octave windows is automatically displayed.
Select Time-weighted from the **Overall acoustic** tab, and the 3 RMS detectors in the RMS detector selection



Create a new layout Then add to this layout the overall level from the 1/n Octave and Time weighted from the Overall acoustic. Waterfall Profiles will be displayed in the layout Profile.



Press Run and make noise. The following results are obtained.

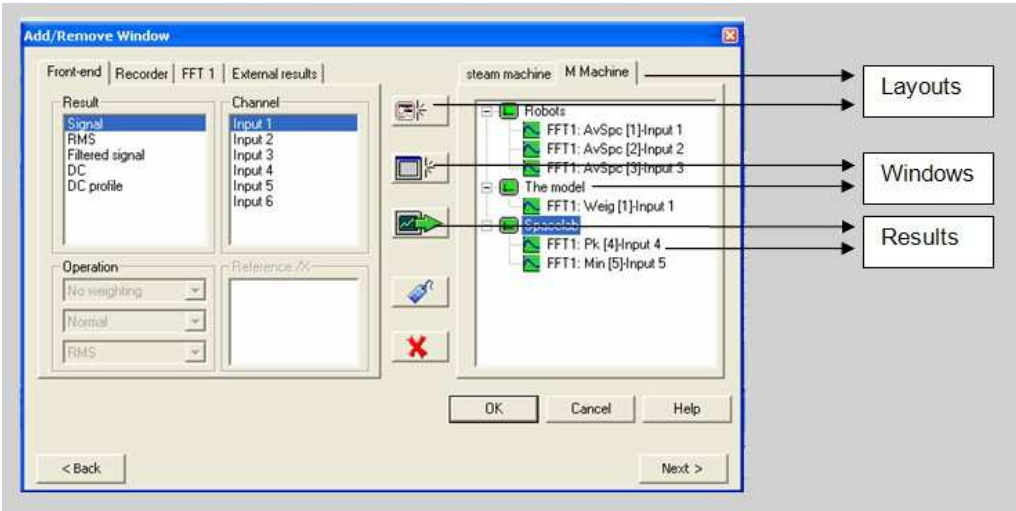

Window1 - Wtf: Oct: OvrLin [1]-MICRO [130 ms: 16.73E-03 Pa]
Acoustic pressure (Pa)
350E-03
300E-03
250E-03
200E-03
150E-03
100E-03
50E-03
500 m
1
1.5
2
2.5
3
3.5
Time (s)
Display
Mode:
Traces:
Wtf: Oct: OvrLin [1]-MICRC
Cursor
X:1 #
130 ms
Y: 16.73E-03 Pa
Window2 - Wtf: OvA: LAI inst [1]-MICRO [130 ms: 193.15E-03 Pa(A)]
Acoustic pressure (Pa(A))
200E-03
150E-03
100E-03
50E-03
500 m
1
1.5
2
2.5
3
3.5
Time (s)
Display
Mode:
Traces:
Wtf: OvA: LAF inst [1]-MIC
Wtf: OvA: LAS inst [1]-MIC
Wtf: OvA: LAI inst [1]-MICI
Cursor
X:1 #
130 ms
Y: 193.15E-03 Pa(A)

Chapter 5: How do I

How do I Display Numerous results ?

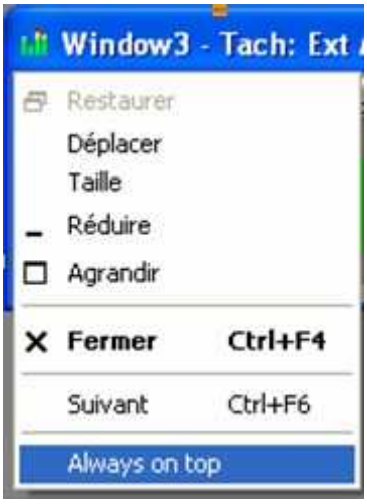
To display result you have to use this button:

The following window will appear.



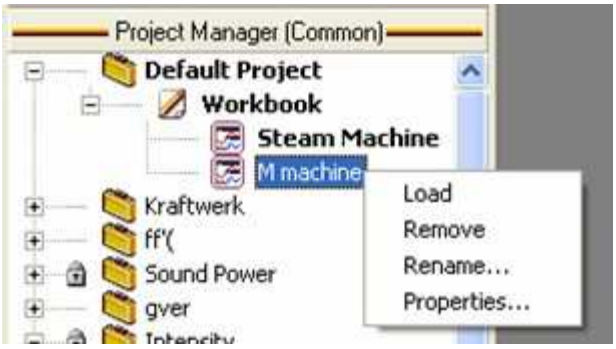
Remember that: the results are stored in windows, and the windows are stored in a layout. You can display results, windows and layouts by using the center button. You can see on the right how the results windows and layouts are stored.
If you click on OK, it will display the active Layout and automatically arrange the windows. Moreover, it's possible to rearrange the windows by this icon:

to have a window always on top by clicking on the upper left of the window



If you want to change to another layout you have to click here:

A layout is link to a workbook. If you want to cancel a layout you have to right click on the workbook in the project manager.

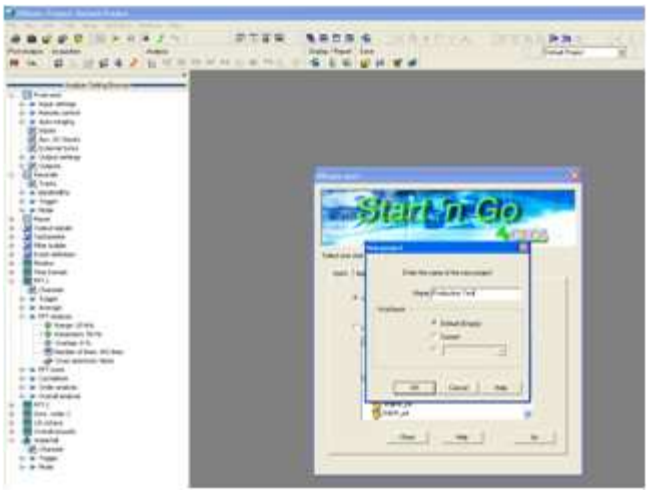


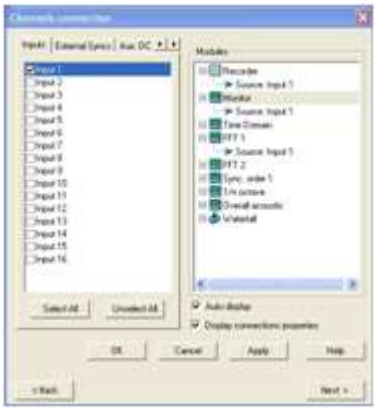
It's now possible for you to display numerous results by using NVGate®.

How do I check the current value of my 3 major settings ?

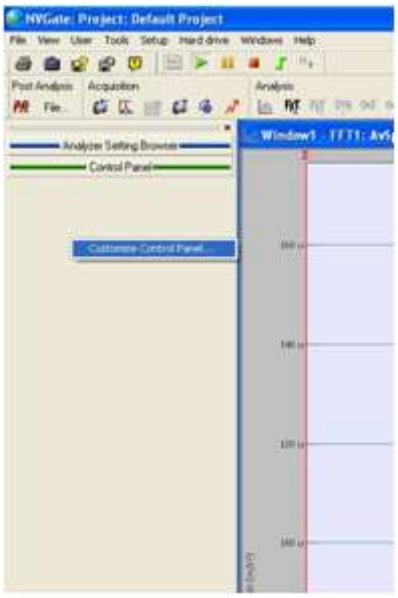
I need to control the status and to change some preferred settings

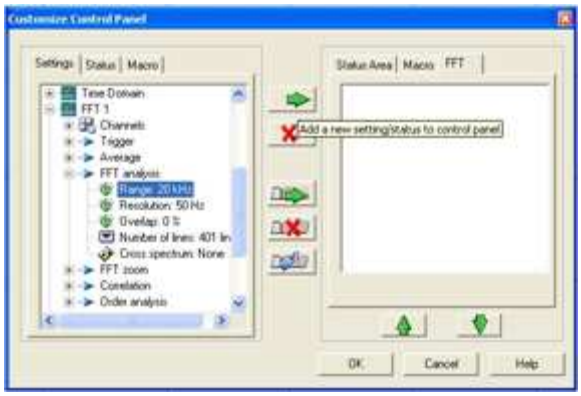
1- Built your project as usual

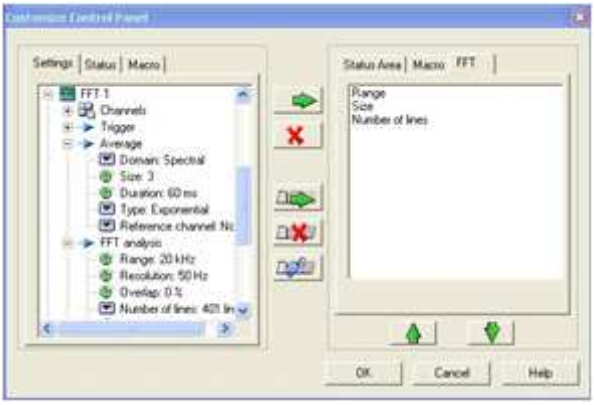


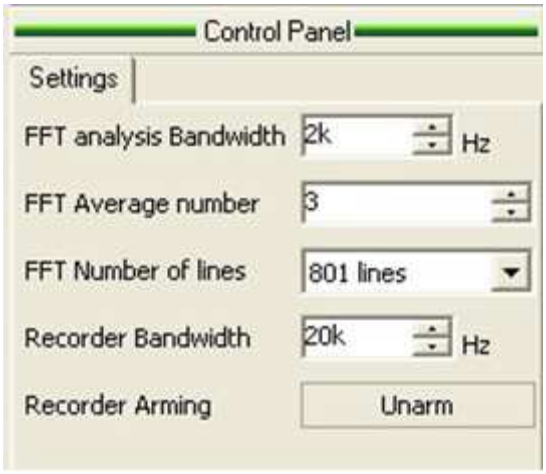


2- Add your preferred settings into the Control Panel



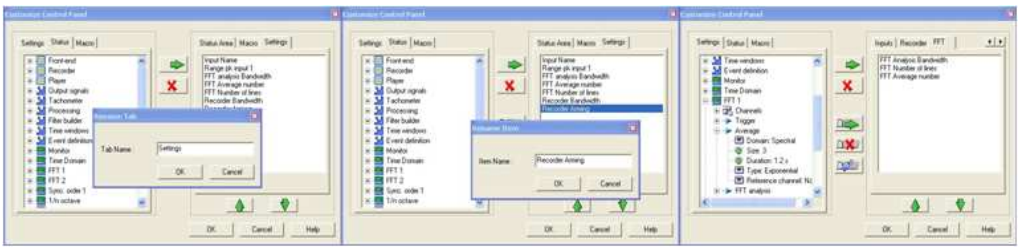


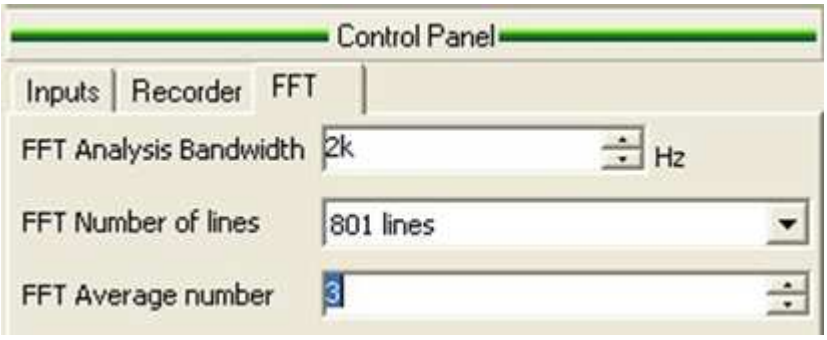




3- Add more status information into the Control Panel

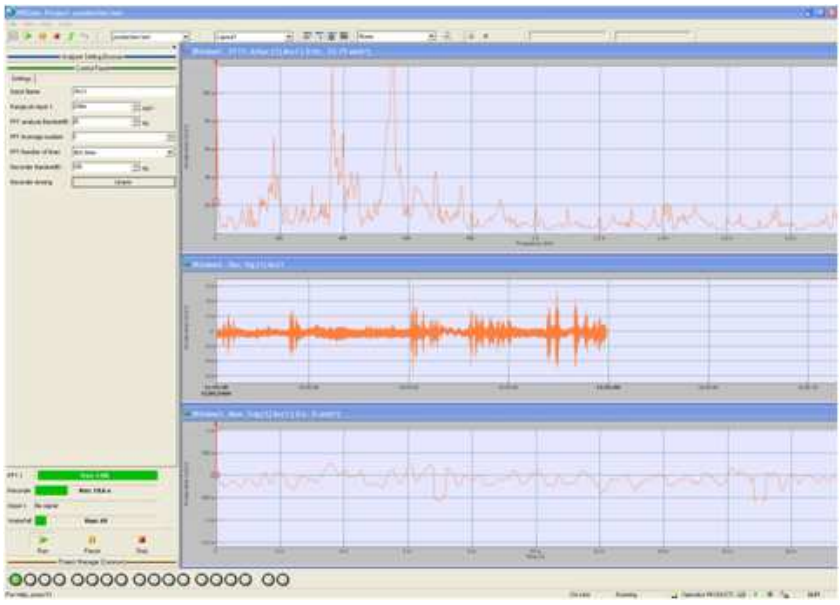
4- To go further: Add and Rename Tabs, Labels…

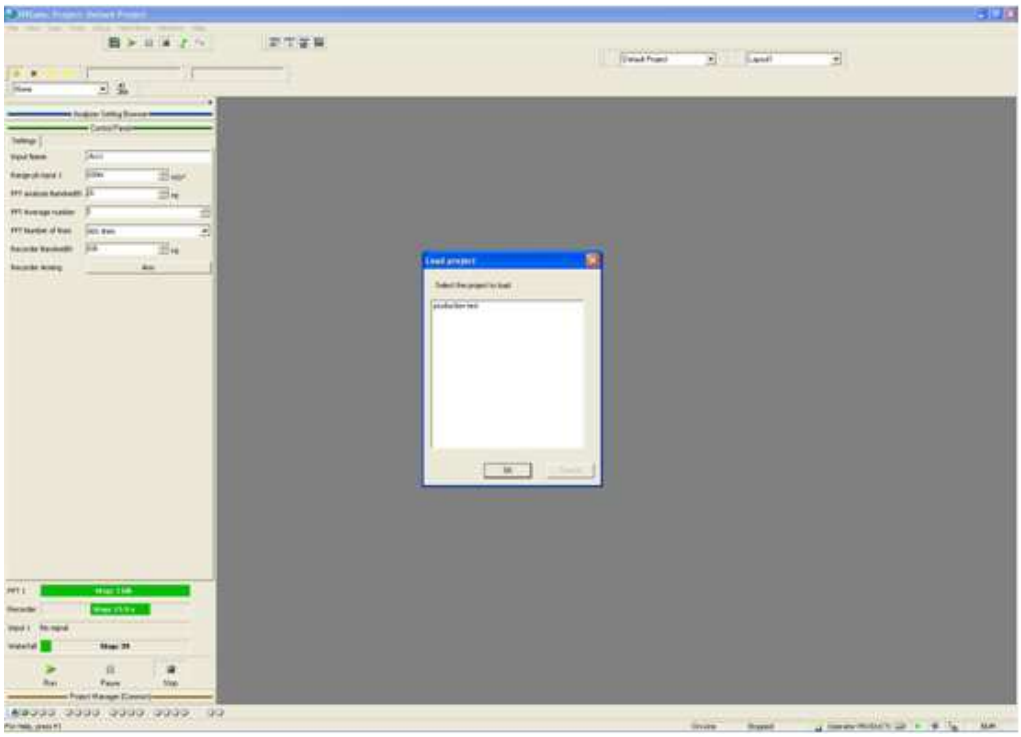




5- To go further: Use restricted user

If you ask to a non-specialist people to do some measurements and you want to be sure that he will not change the important settings, you can prepare a setup and put into the control panel only the settings allowed to be changed by the restricted user.

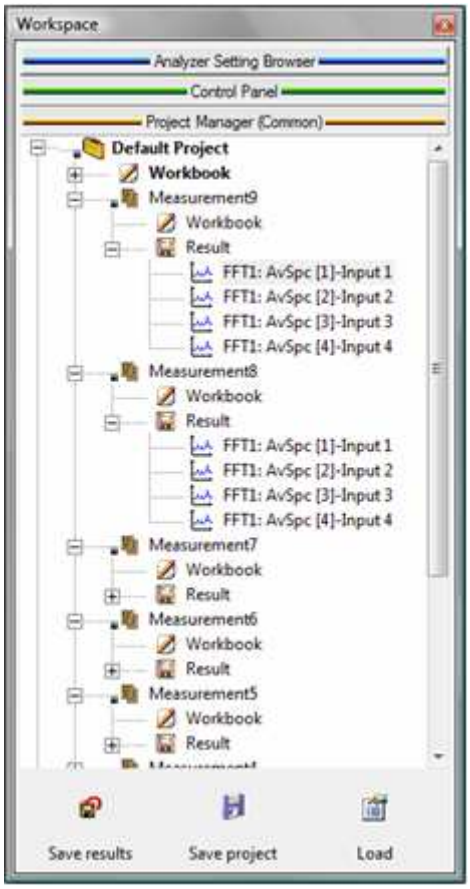


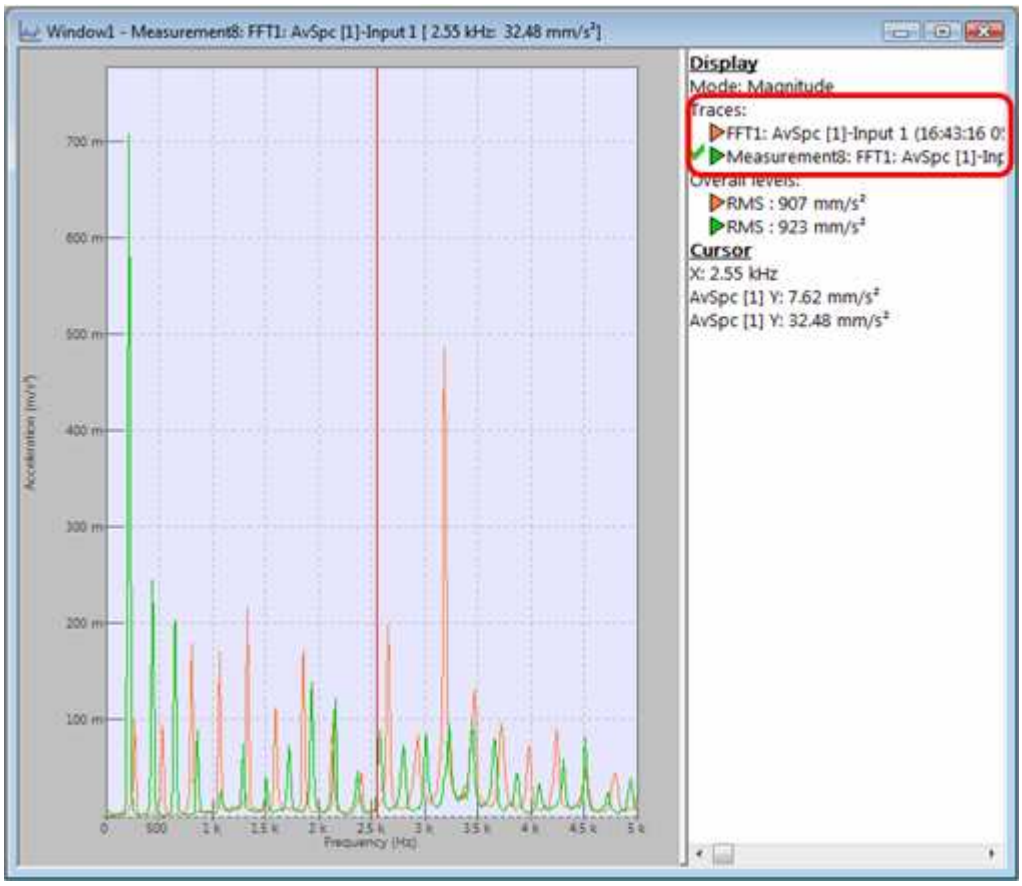


Note: The Analyzer Setting Browser is not available. The restricted user can only view states and change the settings available into the Control Panel.

How do I compare current analysis with saved results ?

While doing measurements on machinery or device, it is often interesting to compare current acquisition (or post- processing result) with previous measurement results. The goal is then to visualize the differences between the acquisitions in order to see trends or to identify typical defect signature.
This type of operation can be achieved easily with your NVGate software.
First you need to have saved results in your project manager. Then make an acquisition or post-processing of your unit under test noise or vibrations.





In order to compare your current analysis with one previous results;

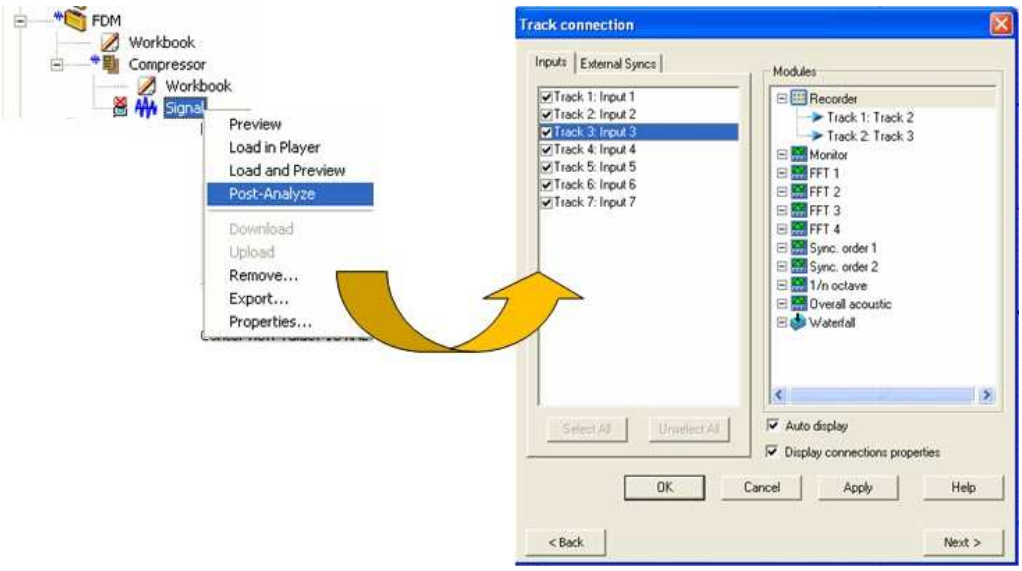
1. From the project manager, expand the project and the measurement where your results is saved
2. Click on the result icon and drag & drop it in to the current analysis windows.
3. If you want to remove the saved results from the acquisition windows select it as an active trace (use up/down arrows) and press CTRL + DEL in order to remove it. This is also available from the Properties contextual menu Remove trace&ldots; entry.

This feature is also useful for fast simple result comparison.

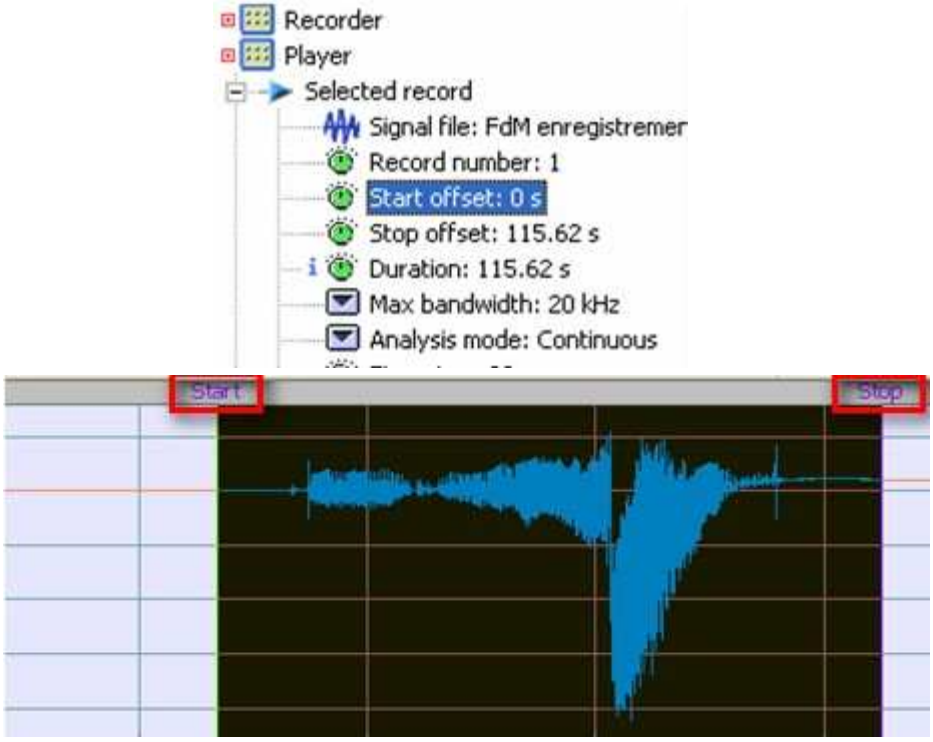
Note: it is possible to achieve the same function from the Add/Remove&ldots; entry of the Windows menu. When the dialog box is displayed, the saved results can be selected and displayed in the acquisition windows from the Externals results tab.

How do I split my record in multiple files ?

First, you have to post analyze the signal file. The shortest way is to go on the project manager and right click ”post analyse” on the signal file. Then you have to put your useful track in the recorder plug-in and click on OK.



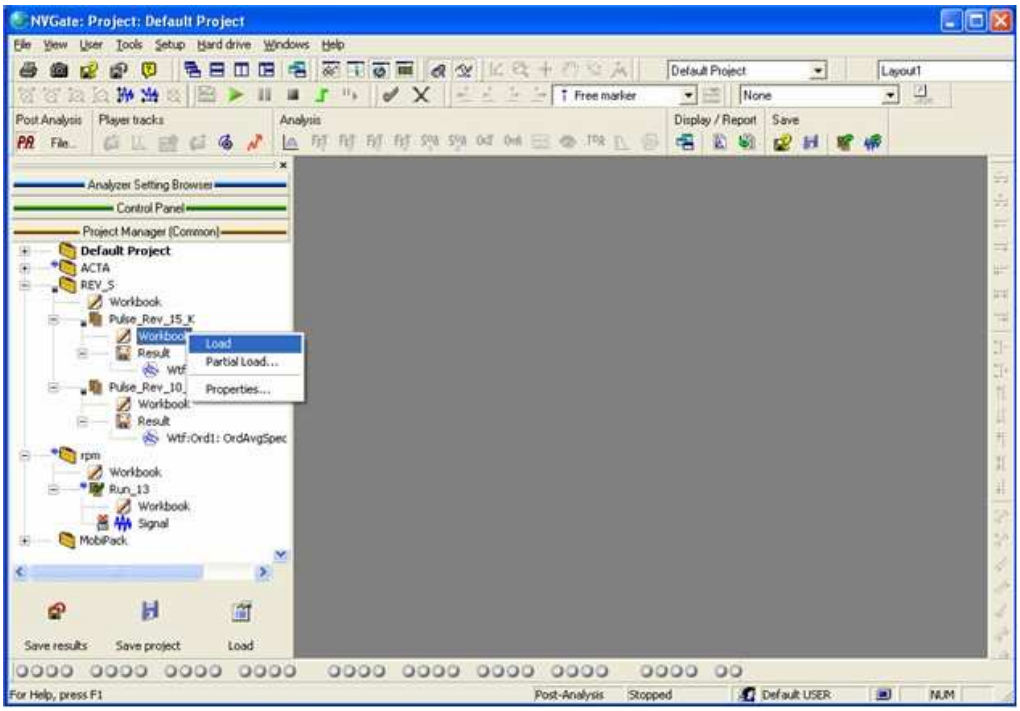
Now you have to define the period that you want to re-record by using start offset and stop offset on the player settings or on the graph.



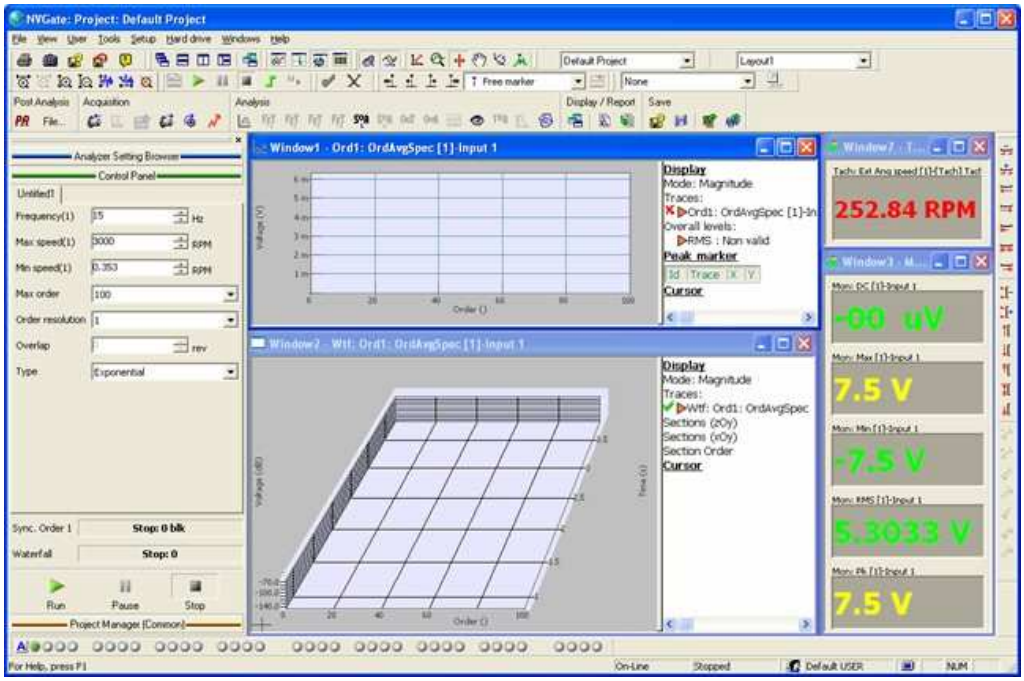
Finally, you just have to click on RUN

How do I perform the same measurement as previous one ?

NVGate manages measurement results into projects. For each measurement there are result (and or signal data) and a workbook. The workbook contains the settings used during the measurement.
You can easily setup again your analyzer to redo measurement with exact configuration as the one of measurement : just reload the workbook of this measurement. Right click on the ”workbook" and select ”load" :



In one click you get your analyzer ready to go for measurement:



This reloads the following :
- Analyzer setting
- Control Panel
- Layouts
- Save Setup
- Print Setup

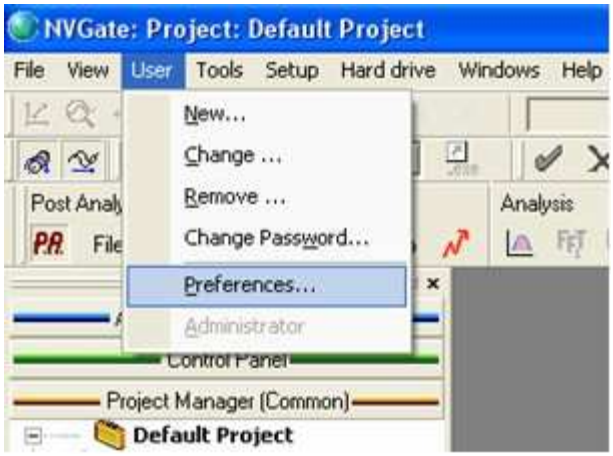
It is also possible to ”Load Partial" configuration and choose the setting to reload :

The main feature is that you can load workbook from any measurement of any project.
It ensures you to redo measurement with exact configuration of a previous one.

How do I create and use my units ?

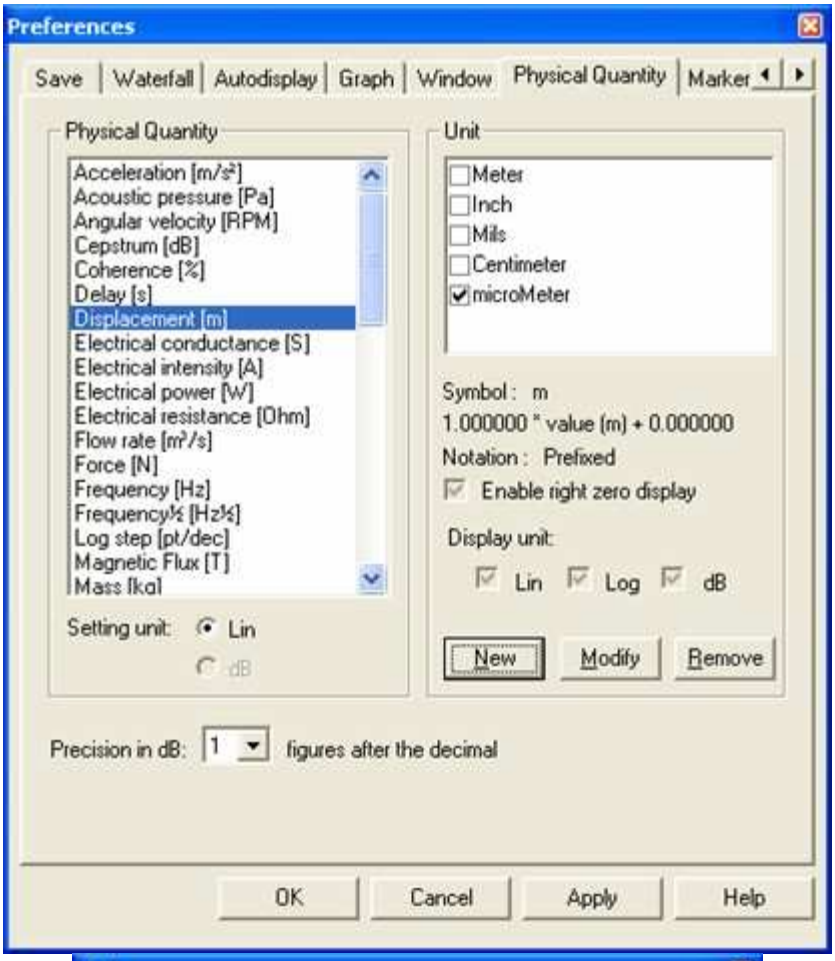
By default NVGate displays trace using SI unit.
For each of physical quantity you can select the associated unit. The most units are already set into NVGate. Despite of this, you can also create, customize and use your own unit.

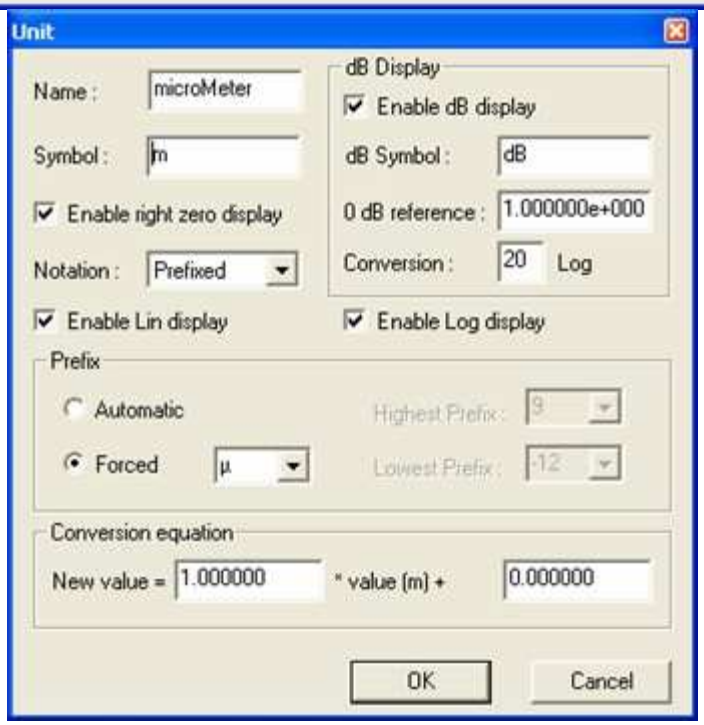
This is part of User preferences. This is available from ”User” Menu, option ”Preferences” :



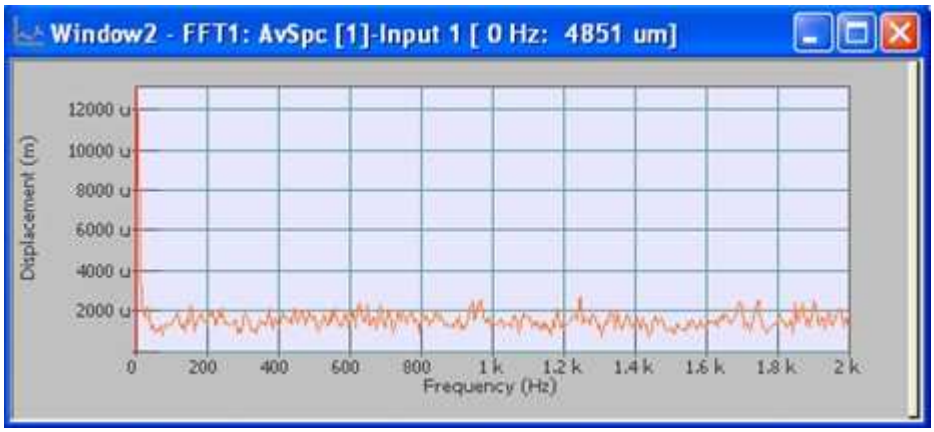
I want all data display with same pre-fix unit.

Select the ”Physical Quantity” Tab. Select the Physical Quantity and press ”New”. if you want displacement to be display in ”um” whatever is the result (by default NVGate automatically adjust the prefix of the unit, to select the most appropriate unit) :



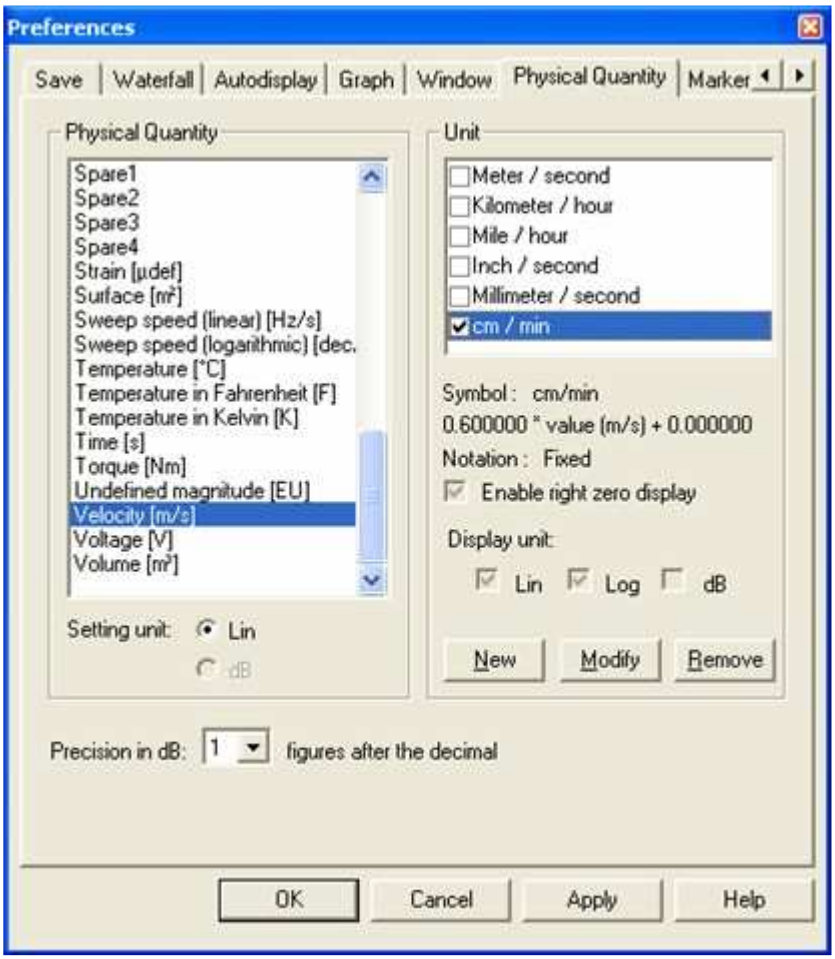


Define a new unit as ”micrometer” with a ”prefix” Forced with ”u”. Then each displacement result displayed by NVGate will use this unit :



I want to display data with a new unit.

You can build a new unit used by your application which is not yet defined into NVGate. For example displays the velocity in cm/min :



Just select the ratio conversion from this new unit to the SI unit : ” cm/min = 0.6 * m/s”

I measure data which the physical quantity is not yet defined.

There are 4 additional spare units available. These may be customize to define your own units.

Note : The user preference for unit affect display result only. Calculation is always done using the SI unit

How do I automatically start with my preferred setup ?

How do I make sure to get exact setting for each measurement?

Doing daily or weekly measurements, leads to used the same setup basis. As an example, the transducers and their channels affectation as well as the bandwidth are often predefined by your tests environment. In the same way the analysis setup stay the same for each analysis type (spectra, order FRF, acoustic profiles, etc.).

Projects load can achieve it, but in this case all the measurements are stored in one project. Using project will save all the setting modifications you've made for one specific measurement in your future setup. There is another way to manage predefined setup; the use of models.

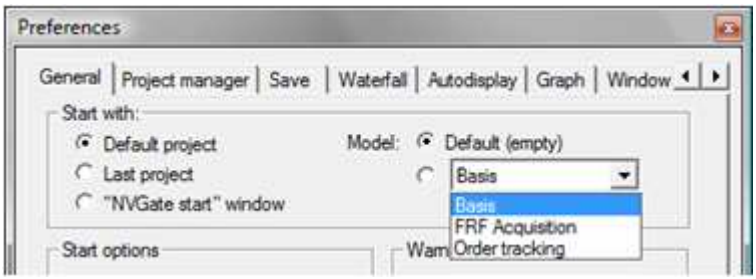
Models are copies of workbook, which content the analyzer and NVGate setup. The difference between workbook and models is that models cannot be changed by setup modification. Whatever the setting you modify, the model will remain the same at next recall.

The way to use the models is very simple.

1. Make your analysis setup

2. Save it in a model using the Workbook\Save As Model&ldots; entry of the File menu.

3. Then you may choose to start automatically on one of these setups by selecting it in your preferences. Select the User\Preferences menu
In this case you will start on a new project (default) with your preferred setup



4. During the test you may have to proceed to different analyses type. The models will be useful to store all your acquisition in a unique project. When you need to change the setup, choose the corresponding model from the Load Model&ldots; entry of the File menu.

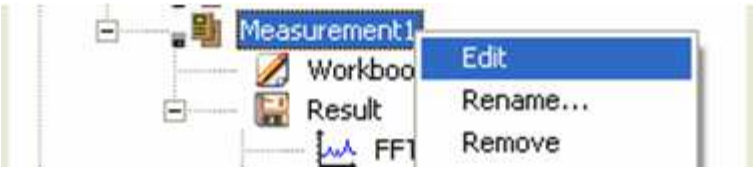
Note:The models are saved in the NVGate Data directory (default is C:\OROS\NVGate data\Workbook Library\User). You can exchange it between different NVGate installations.

Note: The model saves all the setup of the analyzer including the toolbar arrangement, associated signal files etc&ldots;

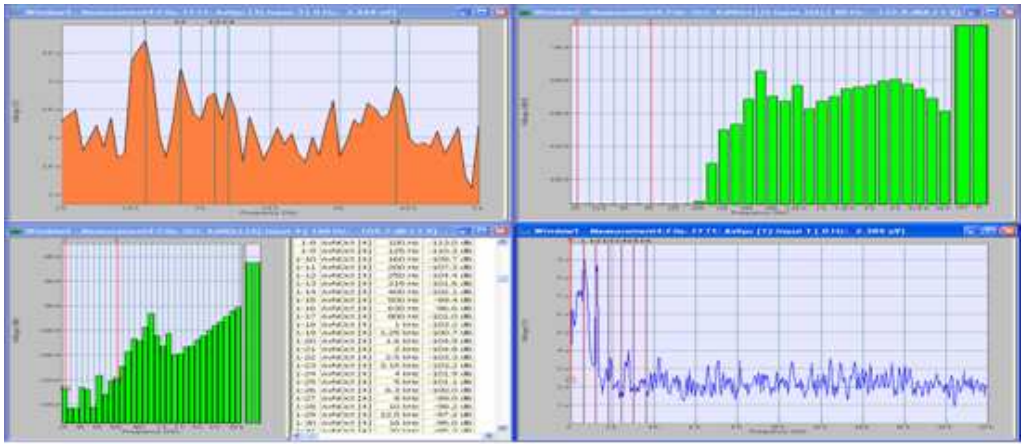
How do I get my results in the same format ?

When a user have saved several measurements with the same kind of data in it, it is possible to edit one measurement and apply the very same layout display to the rest of the measurements. It is basically electronic reporting. As an example, it can then be used to make easier an automatic generation of paper reports.
The display duplication from one measurement to another is a 3-steps procedure:

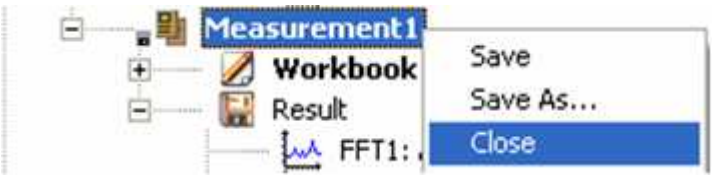
1- Edit a measurement by right clicking on it.



and display the saved results the way you want, with cursors, markers,&ldots;

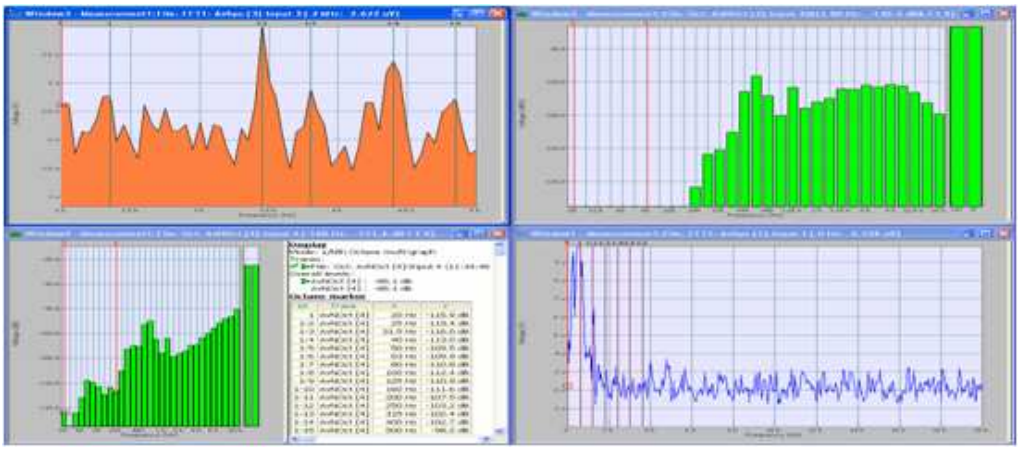


2- Close the measurement that was being edited and save it when NVGate asks to.



3- Edit a different measurement and then right click on the Workbook of the measurement you previously edited in order to do a Load Layouts.

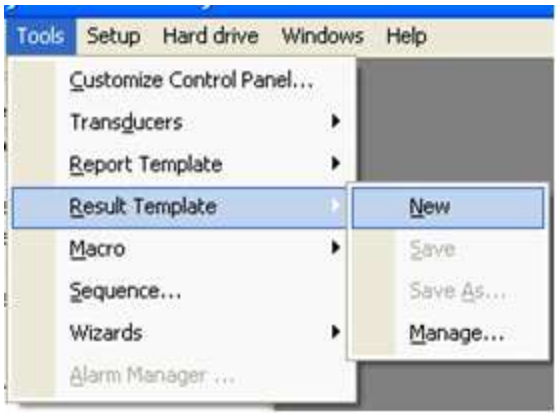
The measurement you are editing uses now the exact same display, cursors and markers as the previous one.



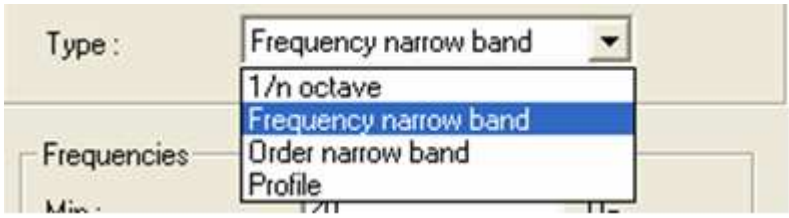
Finally, if this needs to be done for a very large number of inputs, it can be easily automated by a macro!

How do I use trigger on a mask crossing ?

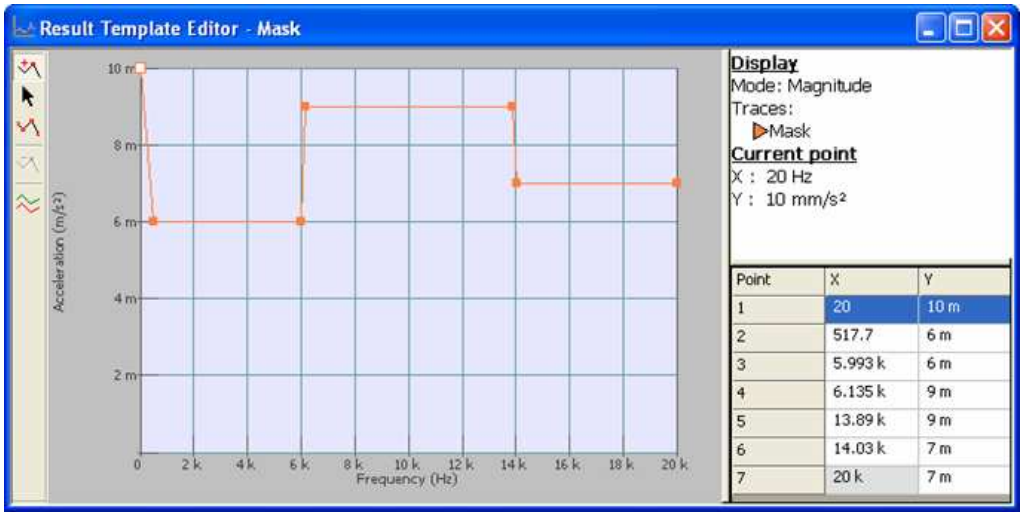
First of all, a template needs to be created from the tools drop/down menu.



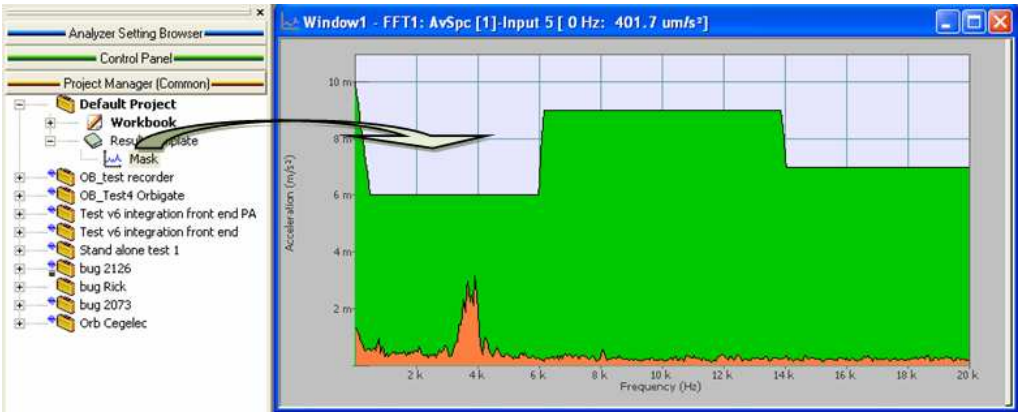
When creating the template, it is important to make sure the template will actually match the type of result next to be used for the triggering.



The template is then created points by points. The position of each point is defined either graphically or by keying in the exact coordinates in the table.



Then, the newly created mask has to be drag&drop into the proper window to create an alarm.



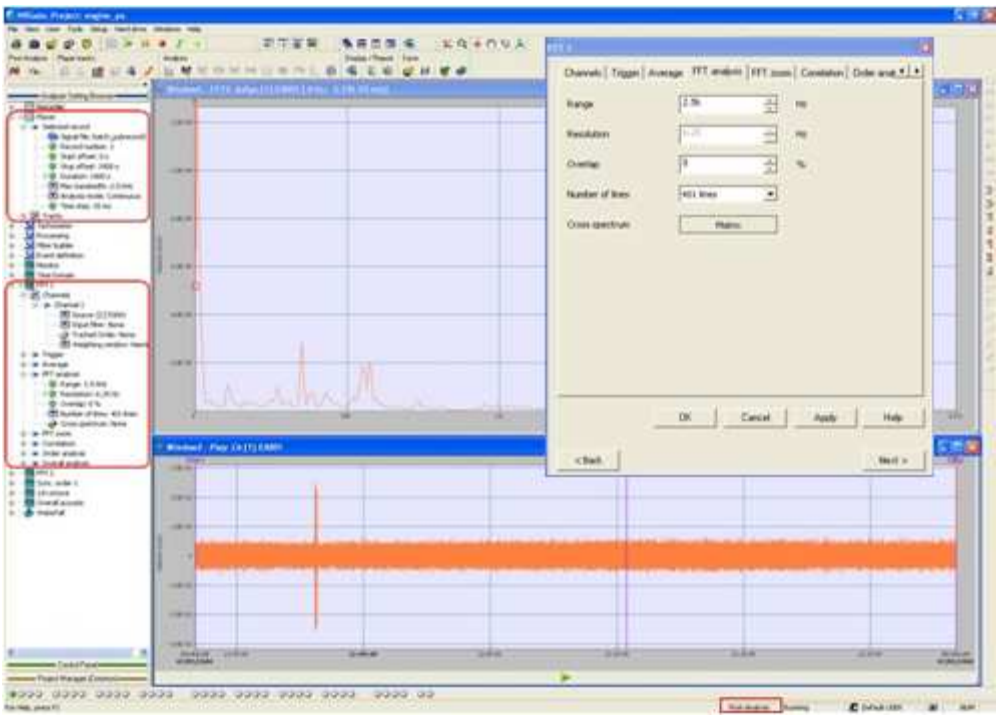
The alarm is activated by a right click in the grey area of the window to select, through Add Alarm, which trace will be compared to the mask. In case of multiple traces in the same window, multiple alarms can of course be created.

An alarm can be used to trigger any kind of events but ONLY from a macro where it is, there, considered as a ”type of condition". Therefore, it can be decided which condition the user will monitor (the trace stays under the mask, the trace crosses the mask,&ldots;)

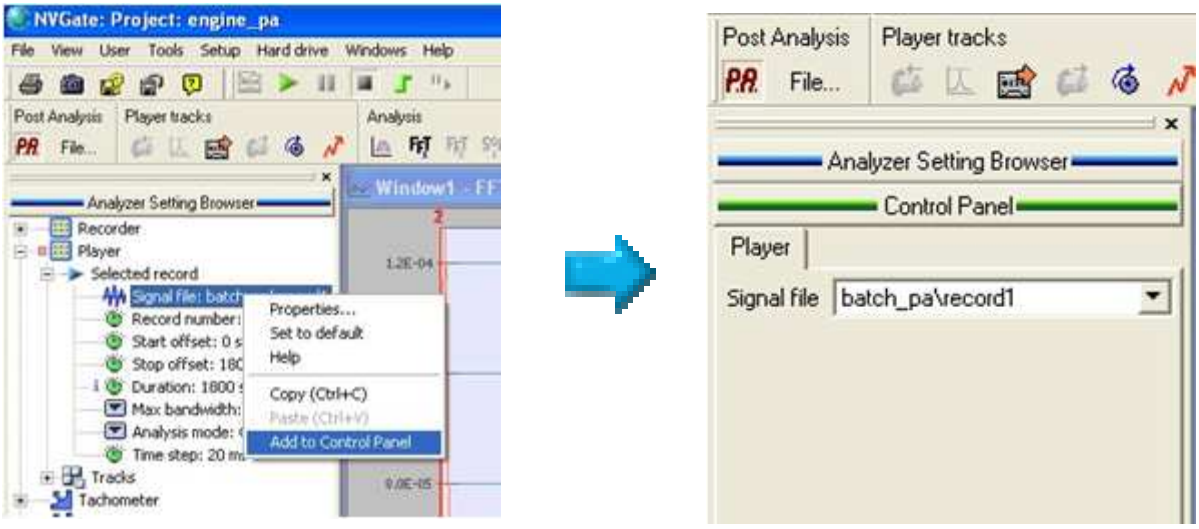
How do I post-analyze all my signals with the same setup?

I measured several systems with same parameters and I want to post-Analyze these signal files with the same setup.

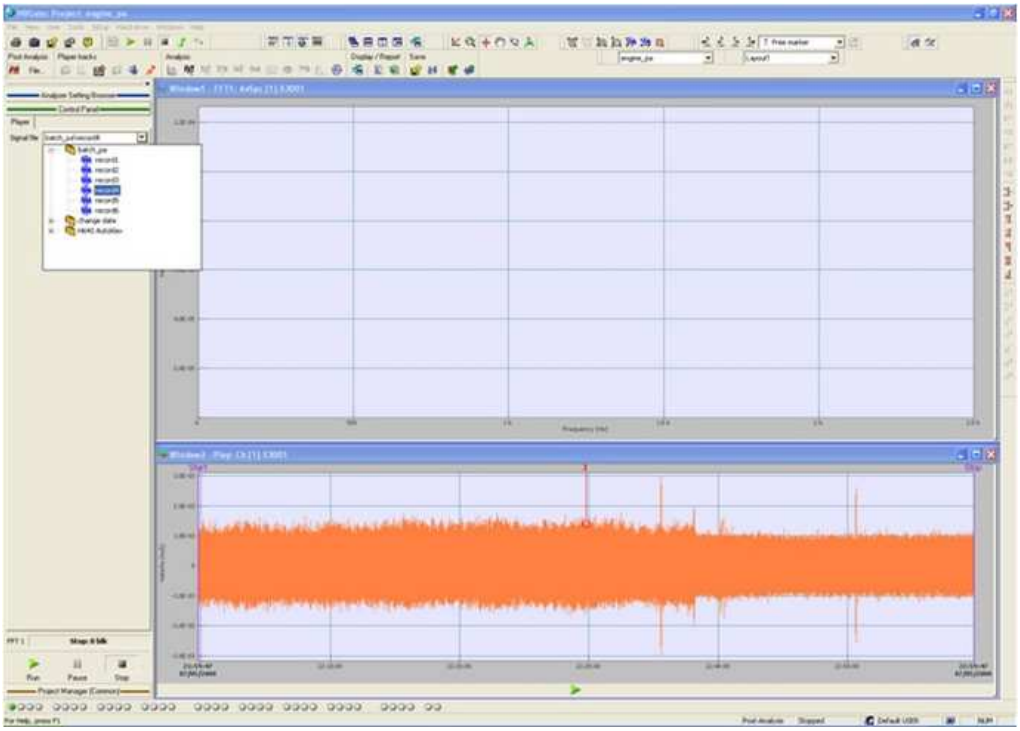
1- Create the Post-Analysis project and perform the analysis setup



2- Add the setting “selected record” into the Control Panel.



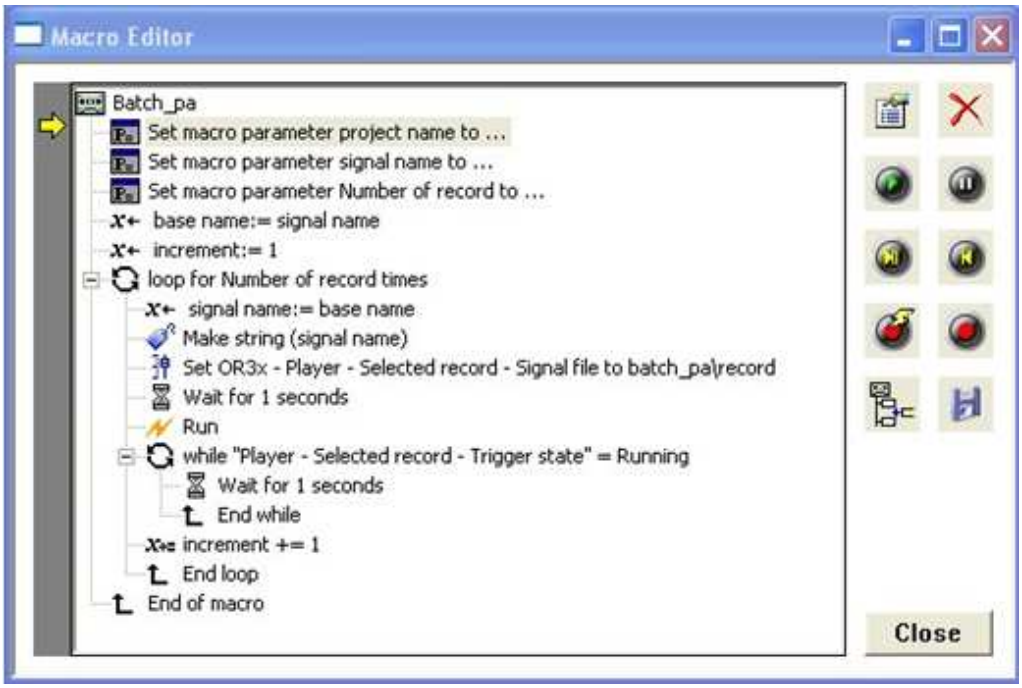
3- After each run, change the record and run again.



The same analysis settings are used for each signal file.

4- To go further: Use Macro functions

You could automate the switch between each signal file using Macro functions.



Your signal files should have the same base name and number increment. The macro “Batch_pa” is available on demand.