
Brüel & Kjær Pulse Primer



University of New South Wales
School of Mechanical and Manufacturing Engineering

Prepared by Michael Skeen and Geoff Lucas
Revised by Gareth Forbes



Rev-0: September 2005

Rev-1: October 2008

Brüel & Kjær Pulse Primer

Brüel & Kjær Pulse Primer	1
Communication between the Pulse Front-end and the PC.....	3
Configuring Pulse IP Addresses	3
If it wasn't quite that easy.....	4
Basic measurements.....	5
Time Signal and Spectrum Analysis.....	5
Equipment Setup.....	5
Setting up the template in Labshop.....	6
Frequency Response Functions.....	23
Equipment Set-Up.....	23
Template Set-Up	23
Setting up a new transducer	35
Exporting measurements.....	37
Export to matlab.....	38
Exporting to matlab on the same computer as Pulse is running	42
Where to find more information	43

Communication between the Pulse Front-end and the PC

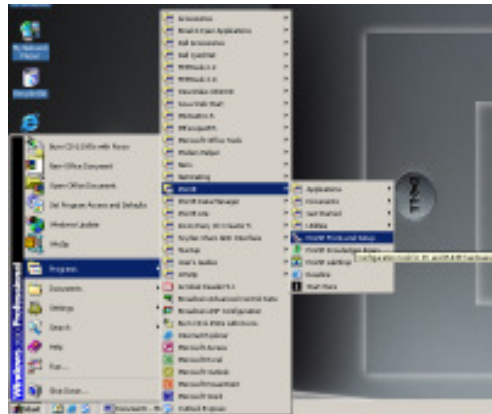
Configuring Pulse IP Addresses

The Pulse Front-end communicates with the PC using an Ethernet connection. The communication protocol used is the same as that used to connect your PC to a network. In fact, it is possible to connect multiple Pulse units to a single PC using an Ethernet hub or to set up communication with the Pulse Front-end over a standard network, though this will not be discussed here.

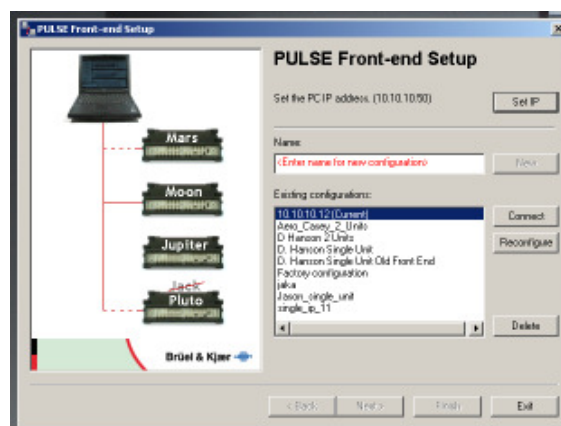
Before the PC can communicate with the Front-end, the PC that the system is to be connected to must know what IP address that the Pulse Front-end is using. It may turn out that the PC was last connected to the Pulse Front-end that you wish to use. In this case this process can be skipped.

The following steps describe the process of setting up the PC to communicate with the Pulse Front-end using the 'Pulse Front-end Setup' Program.

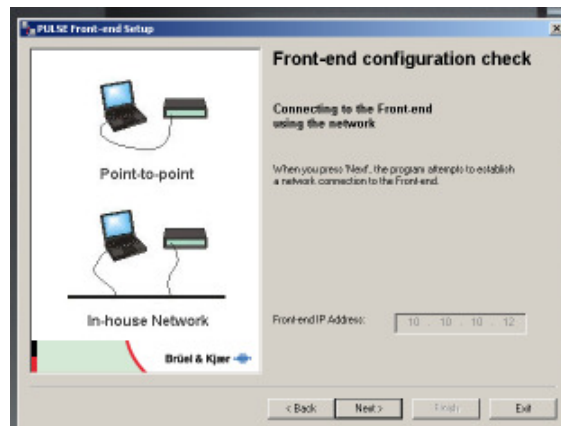
1. Load 'Pulse Front-end Setup' Program.



2. Each Pulse system has a label specifying the IP Address or a name for the Pulse Front-end that should be used with this system. Either look for this IP Address or this name in the 'Existing configurations' box and select it. In this case the IP address for this Pulse Front-end is 10.10.10.12, this has been selected. When this has been selected, press the 'Connect' button.



3. The next screen lets you start a Front-end configuration check. Run this check by pressing the 'Next' button.



4. The next screen tells you that you have been successful in setting up communication between the Pulse Front-end and the PC. Click finish and exit the program. The equipment is now ready to start taking measurements.



If it wasn't quite that easy...

There are a number of things that can go wrong setting up communication between the Pulse Front-end and the PC. The following gives a quick guide to solving these problems.

- The PC may not be configured with a valid IP Address to communicate with the Front-end successfully. Try setting the IP address on the PC as following using the control panel and network connection. (Details are not given here on how to do this as it varies with Operating System.)
 - IP Address : 10.10.10.50
 - Subnet Mask : 255.255.0.0
 - Preferred DNS: 172.36.0.250
 - Other values are not important.
- The IP address or name of the Pulse Front-end may not match the IP address stored in the Front-end. It is possible to change this IP address using the 'Pulse Front-end Setup' Program and an RS-232 connection. Detail of how to do this are not given here as this should not have to be performed regularly, nor by the in-experienced user.
- A cross-over Ethernet cable must be used, unless a network hub is in use.

Basic measurements

Although, Pulse comes with numerous built-in setups for measuring various situations, for example Modal Test Consultant is a powerful template performing modal analysis, it is important to understand how to setup these templates, so that the user can understand how to modify these pre-set configuration if necessary, or how to setup templates that are not provided.

Time Signal and Spectrum Analysis

The most basic Sound and Vibration measurements that can be made are Time Signal and Spectrum analysis. The following section gives instructions on how to setup a simple measurement of the vibrations associated with tapping a beam.

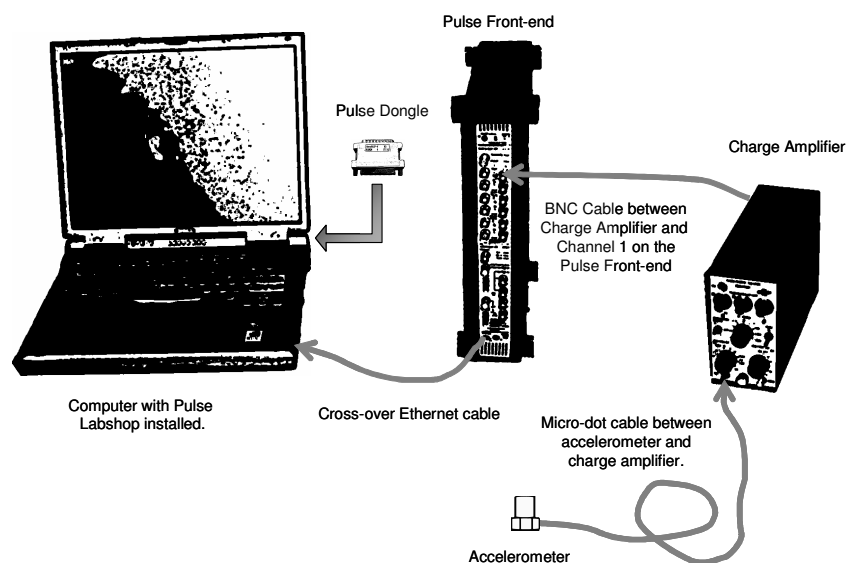
Equipment Setup

The following equipment is required for this demonstration.

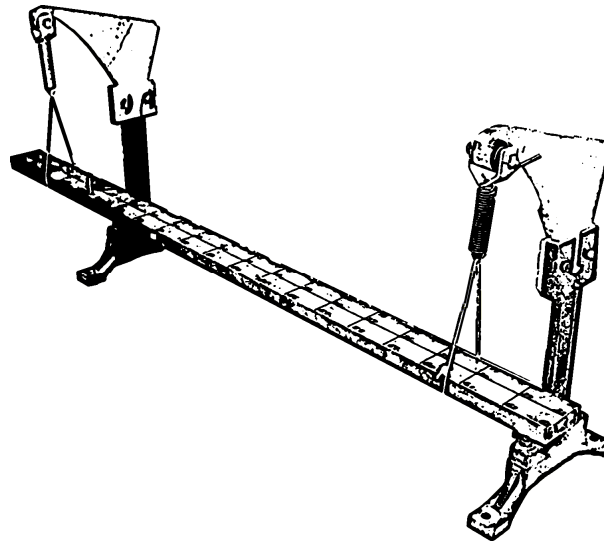
- 1x Pulse Front-end.
- 1x Pulse Front-end power supply.
- 1x PC with Pulse Labshop installed.
- 1x Ethernet crossover
- 1x Pulse Dongle.
- 1x B&K 2635 Charge Amplifier.
- 1x B&K 4393 Accelerometer.

It is possible to substitute another type of Charge Amplifier and/or accelerometer if these are not available.

This equipment should be connected as shown below.



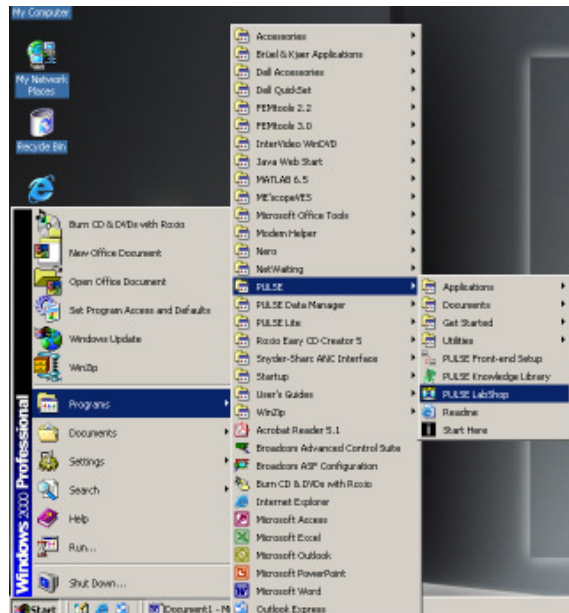
The accelerometer can be attached to anything that the vibration spectrum or time response can be measured. A simple setup may involve mounting an accelerometer to a table and the response of the tapping the table can be measured. The results given in this document have been measured using a suspended beam.



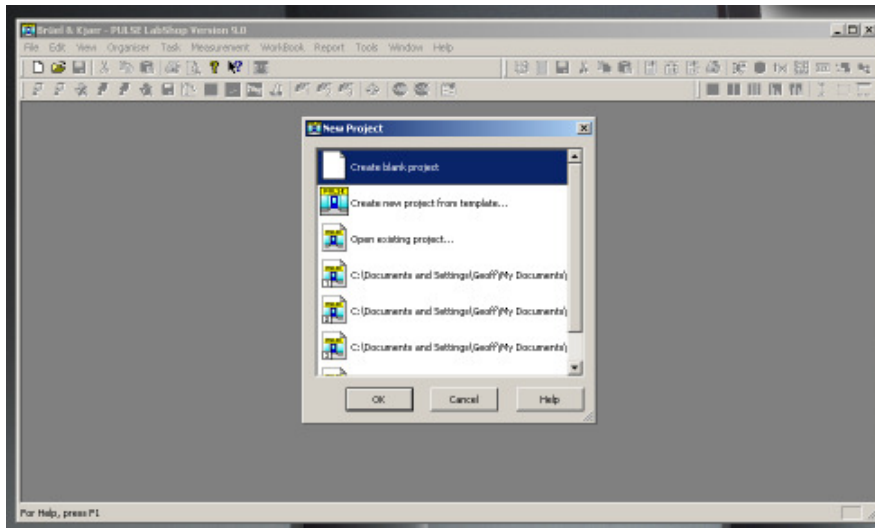
Setting up the template in Labshop

The following describes how to setup a Pulse Labshop template to measure the time signal and spectrum response of an accelerometer mounted on a beam that is tapped with a finger.

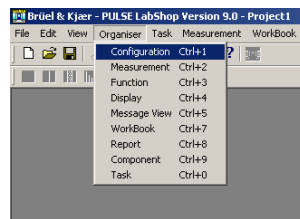
1. Load the 'Pulse Labshop' program.



2. Create a blank project by selecting 'Create blank project' and then clicking 'Ok'.

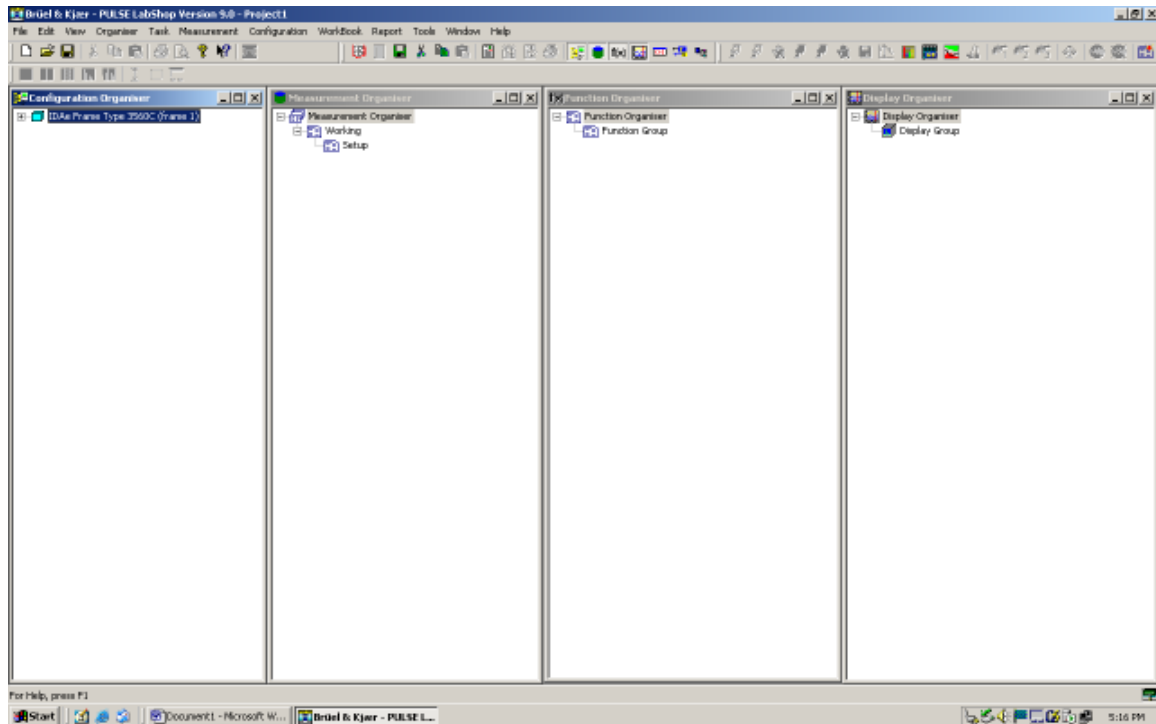


3. In order to setup a template, there are four important windows. These are opened by selecting 'Organizer' from the menu and then clicking on each of the four window titles listed below.

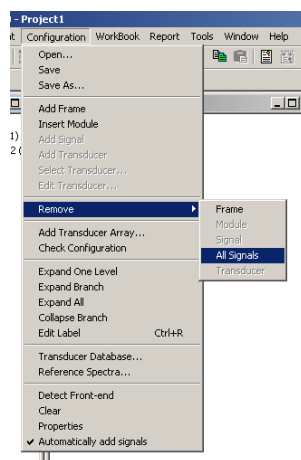



- **Configuration**
This window contains details of the inputs and outputs on the Pulse Front-end and to specify what instruments are connected to this Front-end.
- **Measurement**
This window is used to setup signal grouping and to specify which analysers are to be used.
- **Function**
This window is used to setup the outputs from the analysers and how they are to be used.
- **Display**
This window shows which measurements are currently displayed and allows modifications to be made to the way the measurements are displayed.


4. The four windows mentioned previously are very important for setting up the Pulse Labshop template. It is important that they are easily accessible. It is therefore recommended that these windows be setup as shown below. This configuration follows the flow that the setup process uses.
i.e. Configuration → Measurement → Function → Display

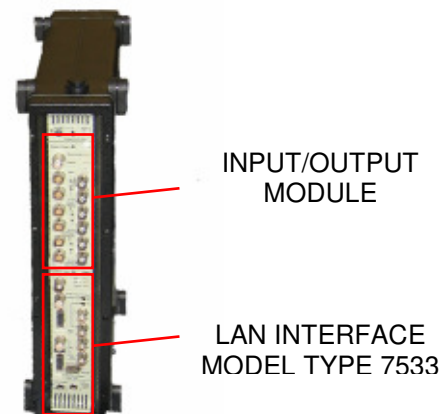
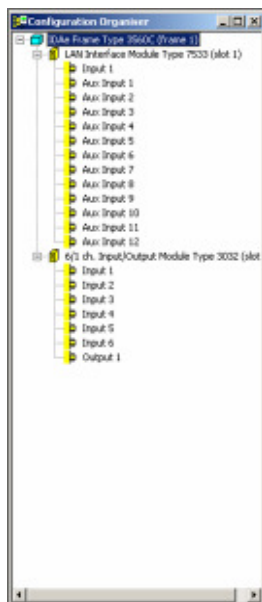


5. When creating a new blank template, Pulse Labshop automatically sets up signals and attaches them to each of the Front-end inputs. In this case, only one input is going to be used. Therefore, all the signals that have setup by Pulse need to be removed. This is done by selecting Configuration → Remove → All Signals. This step can be thought of as resetting the Front-end ready for a new program.

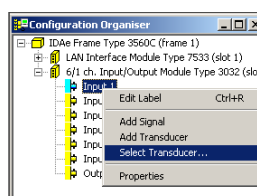


- The next step is to expand the information in the configuration organizer. This is done by clicking on the  icons. This window shows all the all the input and outputs that are available on the Pulse Front-end. The Pulse Front-end consists of two basic components. Firstly, there is the Pulse Frame, this is the black box that contains the electronics, and these frames come in various different sizes. Different electronic modules are connected into the Pulse Frame during manufacture. These modules contain the various acquisition hardware modules.

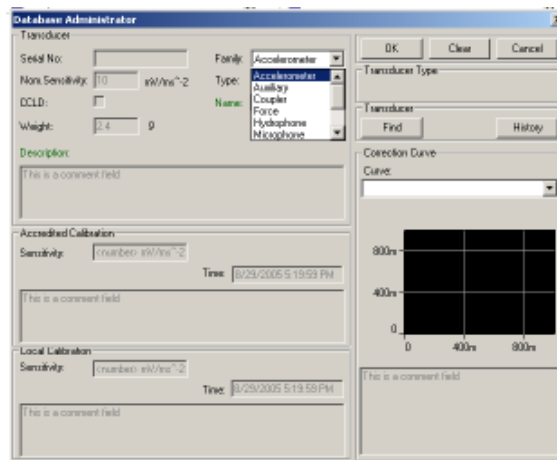
The LAN Interface Module Type 7533 is common to most Pulse Front-ends. This is the unit that allows communication between the PC and the Pulse Front-end. This module also contains a number of low speed inputs that sample at 10 Hz. Generally, these inputs are not used. It is also possible to get LAN Interface Modules that include input and output hardware. The other input module/s can be used for data acquisition and signal generation. These modules come in various different configurations. As the inputs on the LAN Interface module are not going to be used close them by selecting clicking on the  icon.



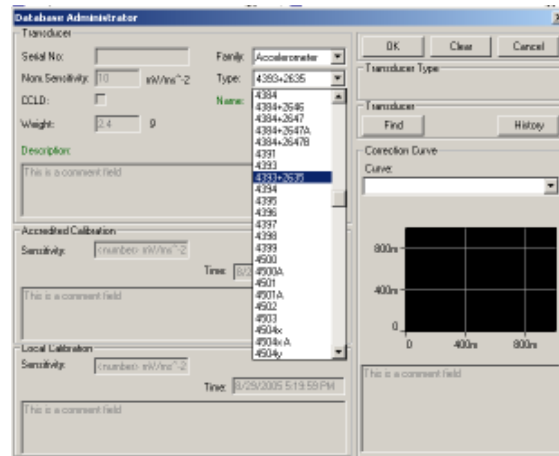
- In order for measurements to be made with Pulse, a transducer has to be connected to one of the inputs shown in the configuration organiser. This is done by right clicking on the input that the transducer will be connected to and then selecting 'Select Transducer'.



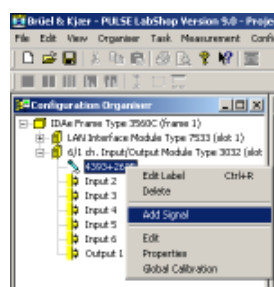
- This brings up the 'Database Administrator' window. Using this window, it is possible to attach many different types of transducers to the input. In this case an accelerometer is going to be used for the measurement. This is selected under family.



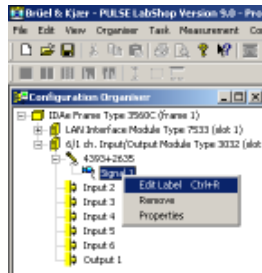
- The next step is to select the name of the transducer that is to be used. In this case a Type-4393 Accelerometer is to be used, with this connected to a Type-2635 Charge Amplifier. The 4393+2635 configuration should be available under type, select this. If it is not available, you will have to either setup this configuration using the section in this guide, 'Setting up Transducers', or modify the setting of an existing transducer for this setup.



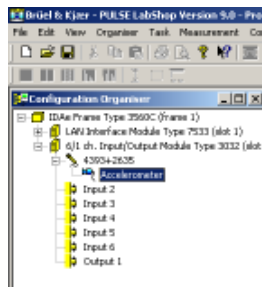
- Now that the transducer is connected in the software, a signal needs to be allocated to this transducer. This is done by right clicking on the transducer and selecting 'Add Signal'.



11. This signal can be given a name by right clicking on it and selecting 'Edit Label'. The name in this case should be 'Accelerometer', but any name can be used.

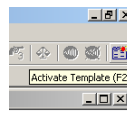



12. The 'Configuration Organizer' window should now look as shown below.

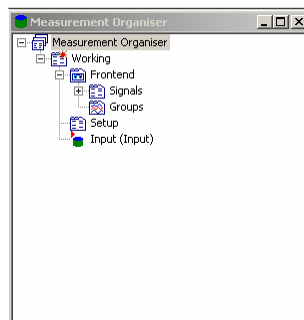


13. To make the changes that have been made to the 'Configuration Organizer' applicable to other windows it is necessary to click the 'Activate Template' button or press F2.

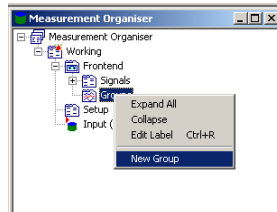
This button is used to apply many changes that can be made to this and other windows. If changes are made to the template, but they don't appear to have taken effect, use this button to update all of the information in the template.



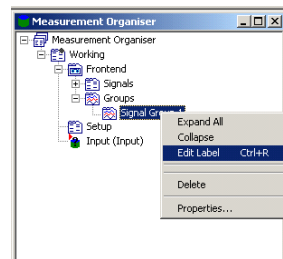
14. For the moment, work in the 'Configuration Organizer' window is complete. The next steps involve using the 'Measurement Organizer' window. The list in the 'Measurement Organizer' should be expanded by clicking on the  or by right clicking the text 'Measurement Organizer' and selecting 'Expand all'.



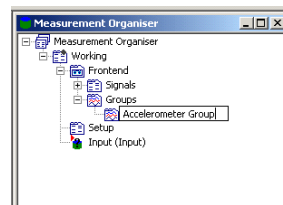
15. It is now required to setup signal groups. Signal groups are required because they are the analysers that will be used shortly to accept multiple signals. It is only possible to feed a signal into a Pulse Analyser by grouping it first. This step may seem a bit pointless when there is only one signal, but the need for this step will become clear when multiple signals are being used. In this case, right click 'Groups' and select 'New Group'.



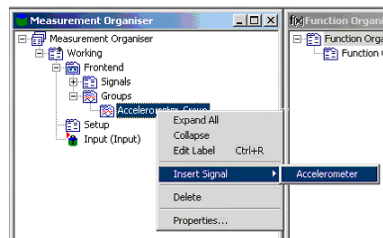
16. It is possible to rename this signal group by right clicking the group and selecting 'Edit Label'.



17. Any name can be used for this group, but in this case the name 'Accelerometer Group' is suggested.

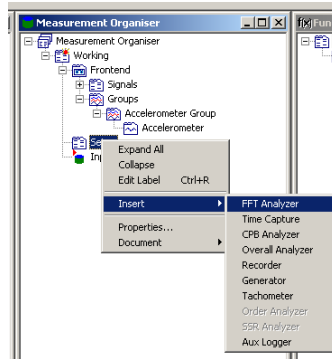


18. Now a signal must be added to this signal group. This is done by right clicking the signal group label and selecting 'Insert Signal' → 'Accelerometer'.

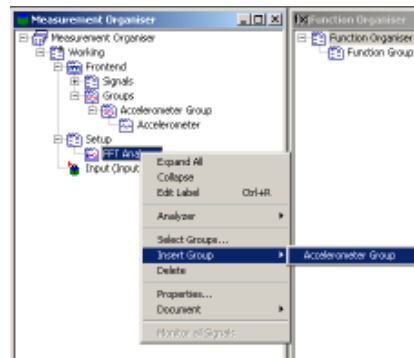


19. The type of analyser that will be used needs to be set. In this case, the time domain and frequency spectrum of the signal is required. An 'FFT Analyser' is appropriate for this type of analysis.

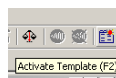
To insert this type of analyser into this template, right click on 'Setup', then select 'Insert' → 'FFT Analyser'.



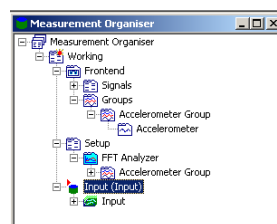
20. This analyser now needs to be told which signals to analyse. This is done by connecting this analyser to 'Accelerometer Group'. To do this right click 'FFT Analyser' and select 'Insert Group' → 'Accelerometer Group'.



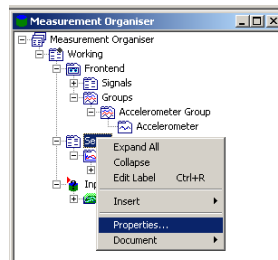
21. Quite a few changes have been made since the template has been updated. Click the 'Activate Template' button now, or press 'F2' to carry the changes that have been made over to the 'Input' section of the 'Measurement Organiser'.



22. The details of the 'FFT Analyser' and input signal are now available in the 'Input' section. Remember to click the 'Activate Template' button regularly so that changes are updated throughout the template.



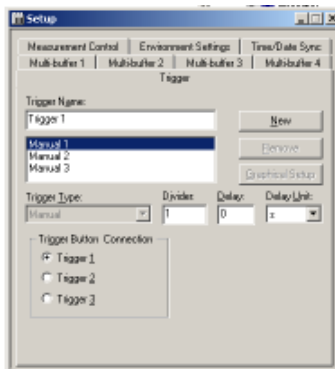
23. Now the 'FFT Analyser' needs to be set up. This process is started by right clicking 'Setup' and selecting properties.



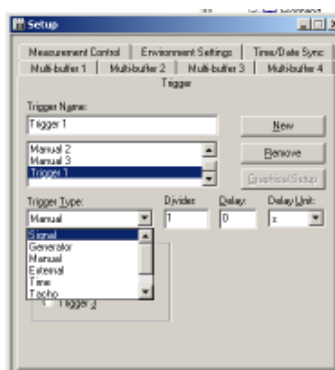
24. A number of different controls are available here controlling how the measurement are taken and stored. For this demonstration on the trigger will be used.

A trigger is some condition that has to be satisfied before a measurement will be taken. In this case the trigger will be when the signal or acceleration exceeds a certain level. This way the measurement will only be taken when after the beam has been tapped.

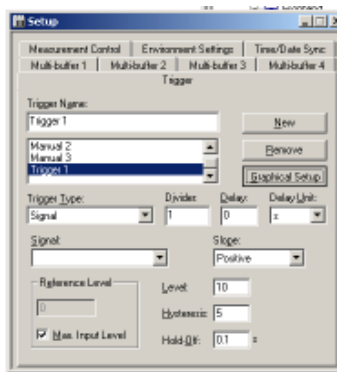
To do this, enter a new trigger name, in this case 'Trigger 1' and click the 'New' button.



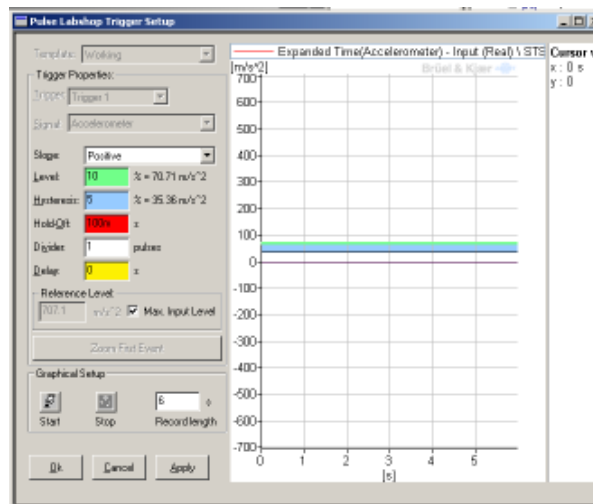
25. Now change the 'Trigger Type' to 'Signal'.



26. The easiest way to set the trigger level is to click the 'Graphical Setup' button.



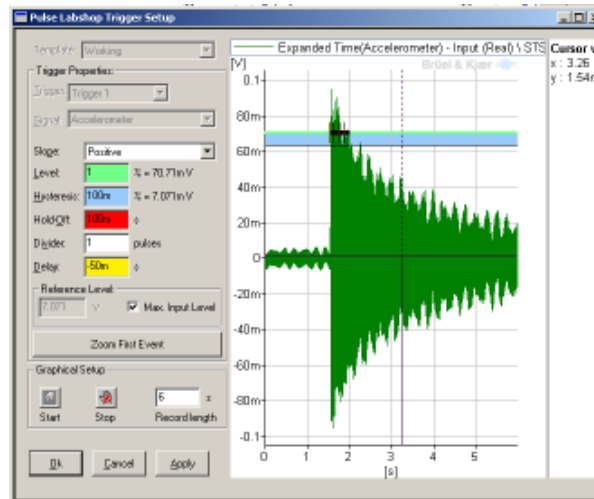
27. This program takes a measurement and then allows the trigger levels to be set based on this measurement. Click the 'Start' button and then tap the beam with a small amount of force.



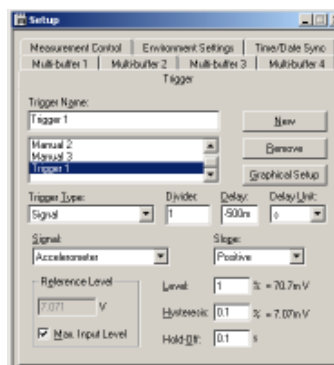
28. The response of this beam to this tap is shown on the graph. Now set the trigger level such that the initial response exceeds this level. The trigger level is shown on the graph as the green line. The trigger level can be adjusted by either entering a value in the green box, 'Level', or graphically moving it with the mouse.

The blue line or 'Hysteresis' level indicates the amount that the signal level must change by after the trigger is activated for that trigger to still be valid. A very large 'Hysteresis' level may mean that it is very difficult to activate the trigger.

When the trigger level is correctly set, press 'Ok'.

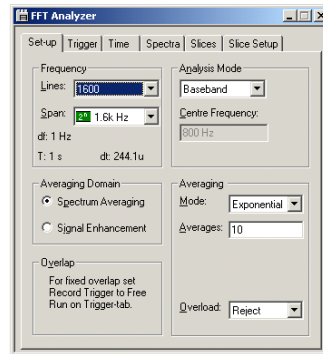


29. From the previous graph in the trigger window, it can be seen that by the time that the trigger is activated, some of the initial parts of the signal have been lost. It is therefore necessary to take the measurement on some of the data before the trigger is activated. This is done by putting a negative value into the delay field. In this case a delay of $-\frac{1}{2}$ a second is used. Note that this value is automatically changes to 500 milliseconds by Pulse Labshop. Click the button in the top left hand corner of the 'Setup' window to close it.



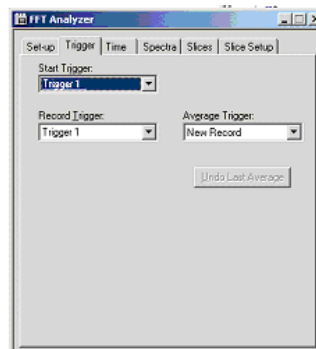
30. Now that the trigger has been configured, it is now necessary to configure the FFT analyser. Right click 'FFT Analyser' and select properties. This brings up the configuration for the FFT Analyser. Using this window it is possible to control a number of aspects of the FFT Analyser, including the resolution or number of lines used by the analysis, the frequency span of the analysis, type of averaging used and triggers can be applied.

The analysis that will be used here will only look at frequencies up to 1.6 kHz. Set the frequency span to 1.6 kHz and also select 1600 lines so that the measurement resolution will be 1 Hz.

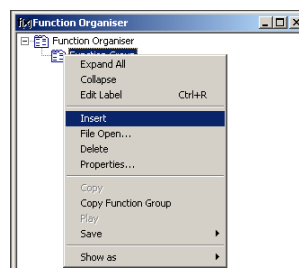


31. Click on the 'Trigger' tab to bring up the trigger tab for the 'FFT Analyser'. The trigger defined previously was called 'Trigger 1'. Set both 'Start Trigger' and 'Record Trigger' to 'Trigger 1'.

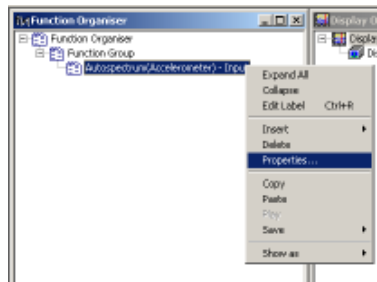
Close this window by clicking on the button in the top right hand corner. The FFT Analyser is now set up. Click 'Activate Template' or press F2 to make sure the changes are applied to the template.



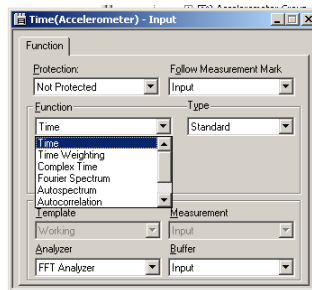
32. It is now time to move across to the 'Function Organiser' window. This window is used to control which measurements are to be recorder by the analysers. Expand 'Function Organiser' and right click 'Function Group'. Select 'Insert'. This will add a new measurement.



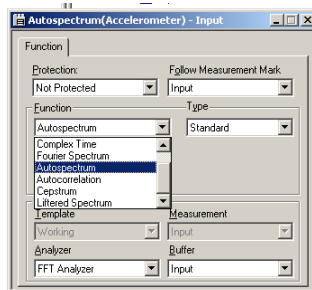
33. Pulse Labshop inserts a default measurement based on previous setting for the template. In this case, a time response is required, so right click the created function and select properties, so that the function can be changed to a time response.



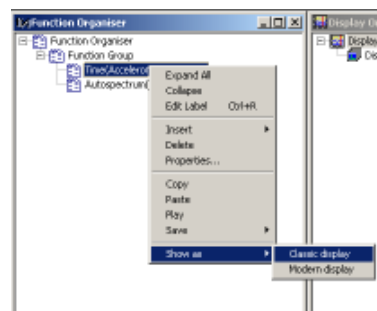
34. This brings up the functions property window. Change the function from 'Auto spectrum' to 'Time'.



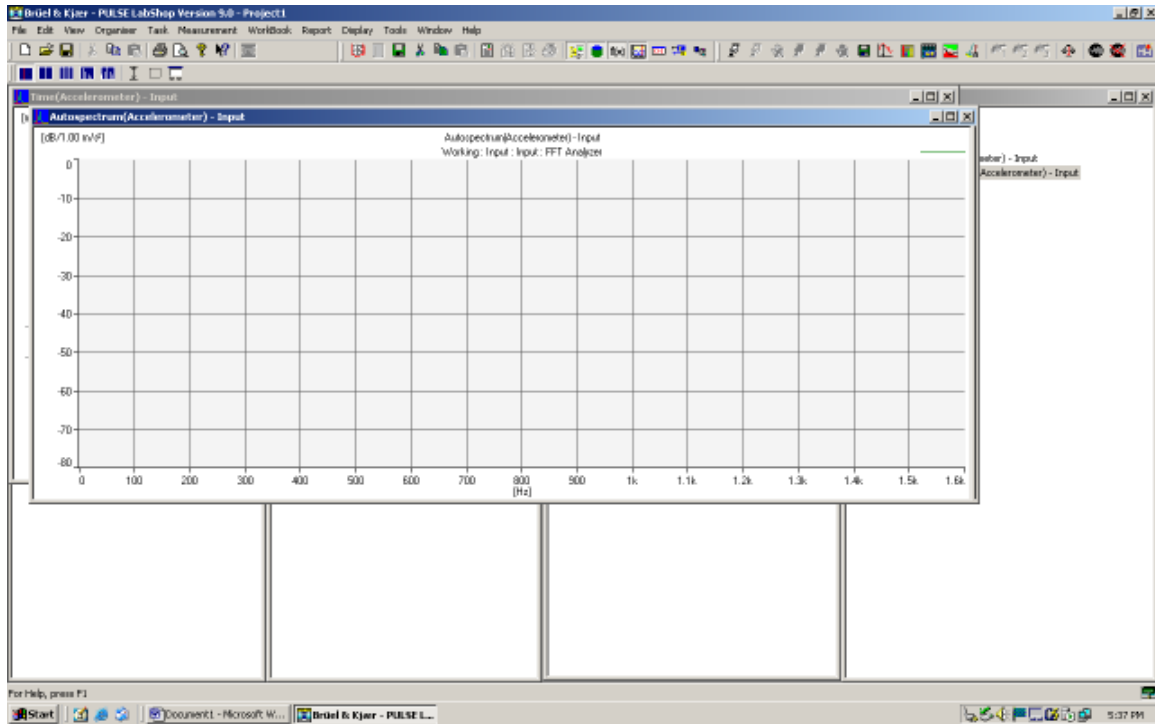
35. Insert another function, as per the previous instructions. Again right click the function and change the Function type to 'Auto spectrum'.



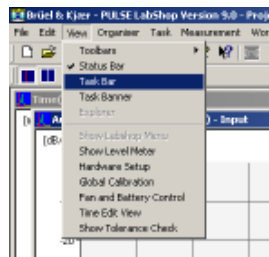
36. A plot of the resulting measurement is easily taken by again clicking on each of the created functions and selecting 'Show as' → 'Classic Display'. Do this for both the 'Time' and 'Auto spectrum' functions.



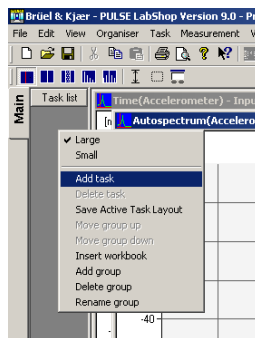
37. There should now be two windows that will display the results of the time and Autospectrum measurements. This means that there are now six windows open and switching between these windows could be confusing. It is possible to set up task displays so that it is easy to switch between different parts of the template.



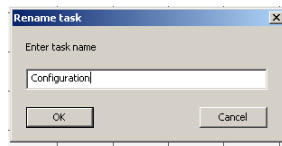
38. Turn on the task bar by selecting 'View' → 'Task Bar'.



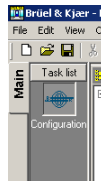
39. This will open the task bar on the left hand side of the screen. Right click on this task bar and select 'Add Task'.



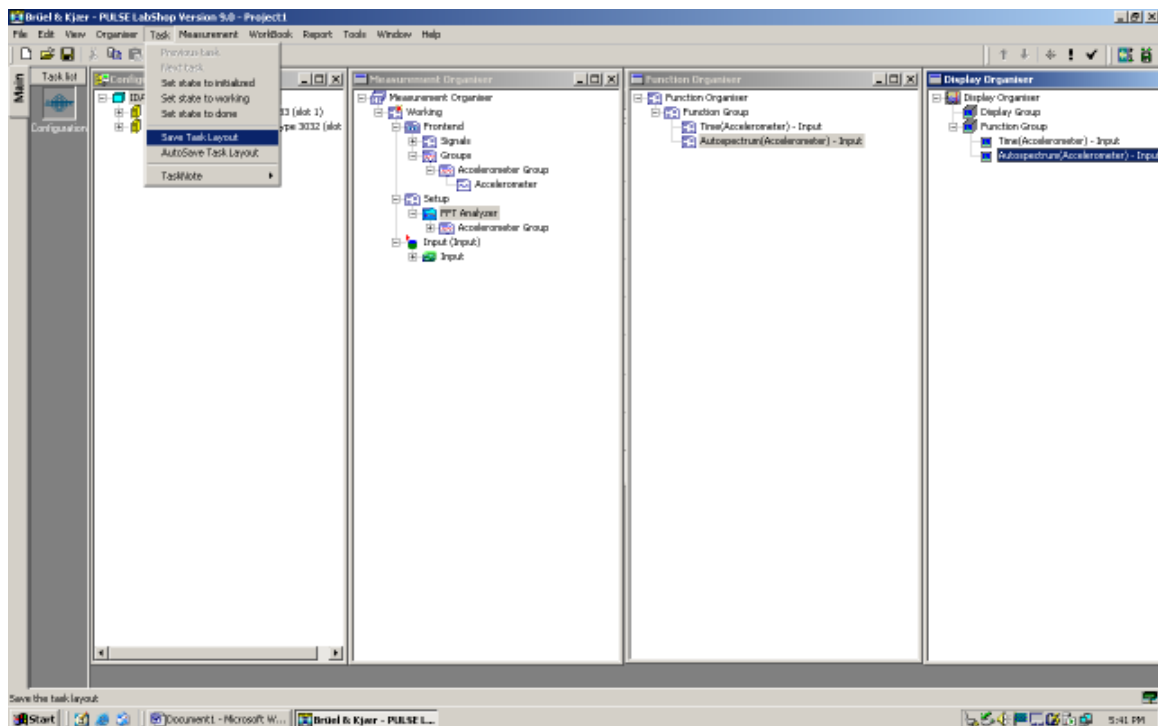
40. Name the task display that you are creating. The first one that will be created here will be to display the configuration windows. Type 'Configuration' and press the 'Ok' button.



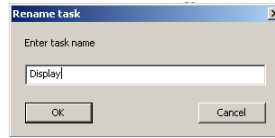
41. This will create a 'Configuration' button. The icon for this button can be changed by right clicking the button and selecting 'Change Icon'. Click the configuration button so that it is highlighted as shown below.



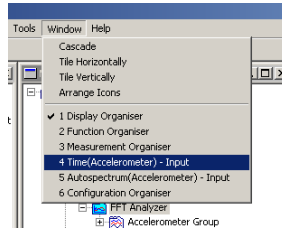
42. Organise the windows so that you can see all of the four configuration windows and then select 'Task' → 'Save Task Layout'. This saves this display to the configuration button, now when ever this button is pressed these four windows will come up. Before trying this out, firstly set up another task button that will display the time and Autospectrum plots.



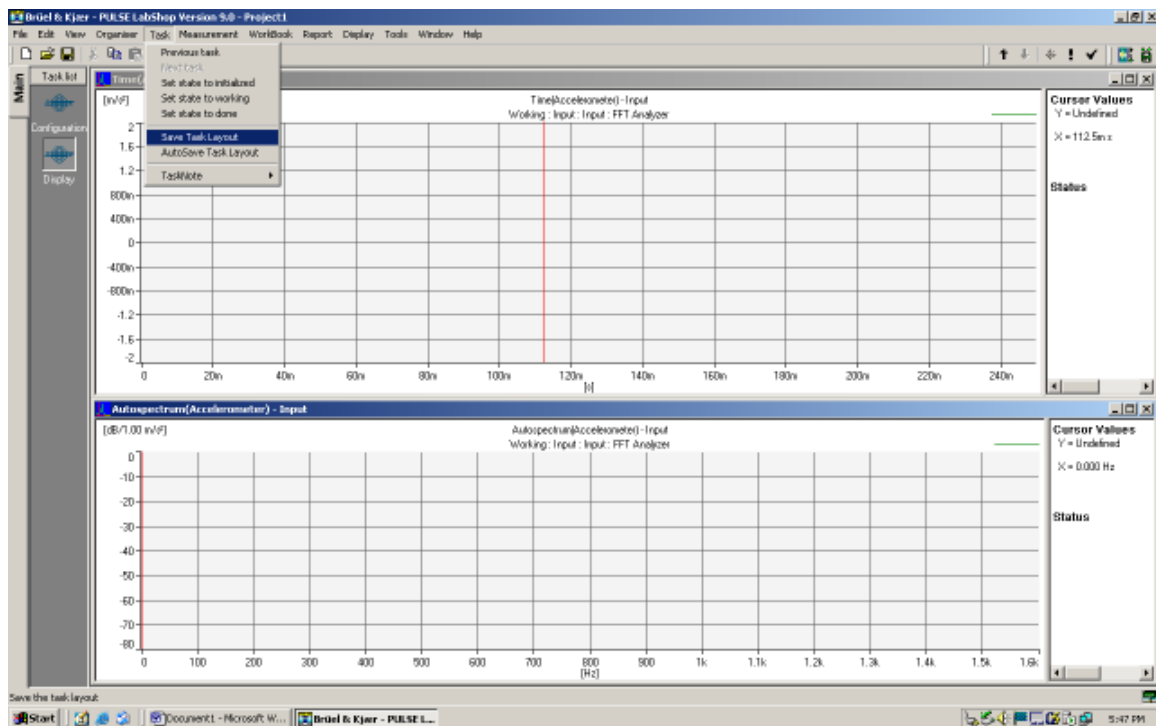
43. Right click the Task Bar again and select 'Add Task'. Name this new task 'Display'.



44. Now bring up the two results windows. Use the 'Window' menu to bring 'Time' and 'Autospectrum' to the front windows.

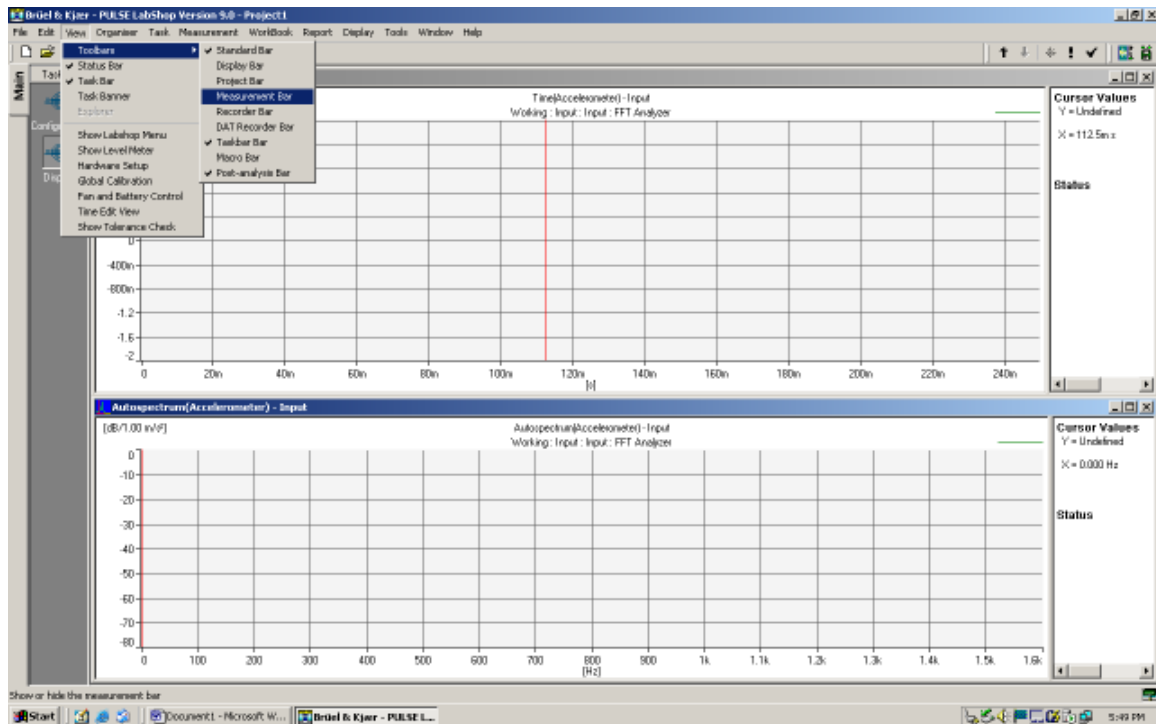


45. Rearrange these windows so that both results will be clearly visible. Select the 'Display' task button and then select 'Task' → 'Save Task Layout' from the menu.

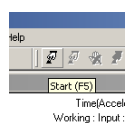


46. Try switching between the windows by clicking on the task buttons.

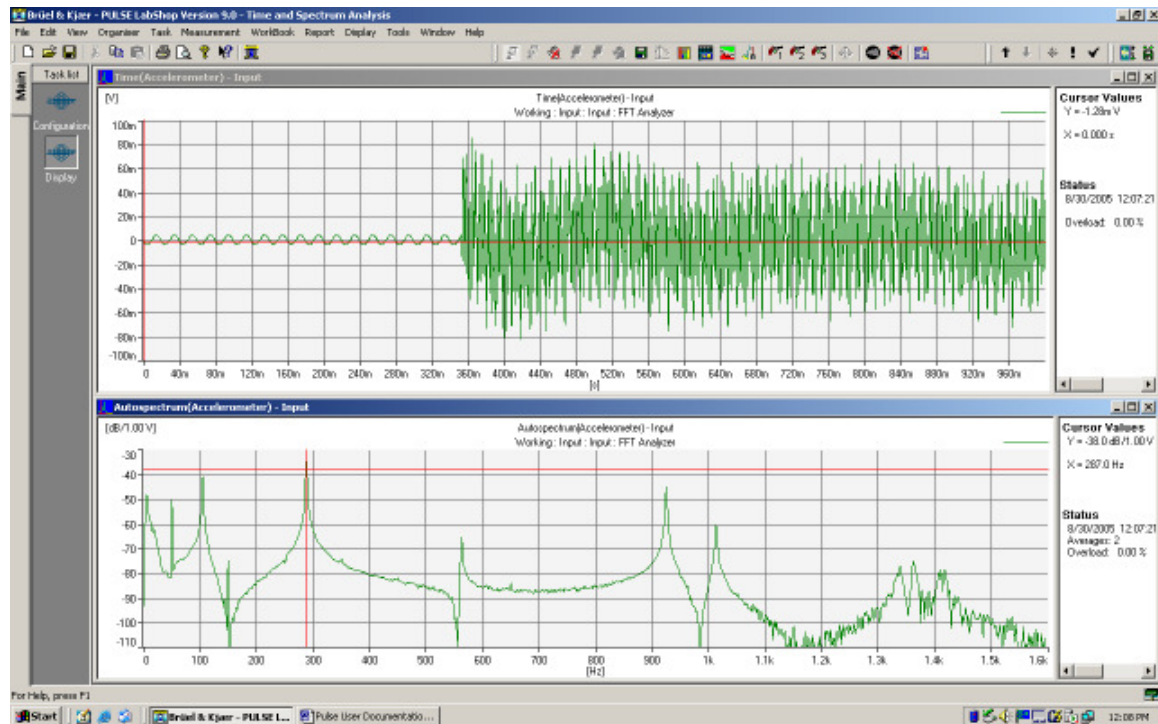
47. The 'Measurement Bar' toolbar has been disabled for the 'Display' task. Click the 'Display' task to activate this window. The 'Measurement Bar' toolbar contains the 'Activate Template' button and also the buttons that are used to start and stop the measurements. Reactivate this toolbar by selecting 'View' → 'Toolbars' → 'Measurement Bar'. Then select 'Task' → 'Save Task Layout' to ensure that the toolbar is displayed each time the 'Display' window is opened.



48. It is now possible to start taking measurements. Click the 'Start' button on the 'Measurement Bar' toolbar. The measurement process will not start until the trigger conditions have been met. Therefore, tap the beam to start the measurement process.



49. The display will take a couple of seconds to display the response. The time and Autospectrum response should then be displayed in their respective windows. The graphs may need to be scaled, if so right click on the graph and select properties. Under the Y Axis tab click Autoscale. Alternately the use of the shortcut keys F10, Ctrl+F10 or Alt+F10 for the different autoscaling options.



50. It is possible to save the template by selecting 'File' → 'Save' from the menu. Similarly, in the future this template can be opened using 'File' → 'Open'.

Frequency Response Functions

Equipment Set-Up

The following equipment is required for this demonstration.

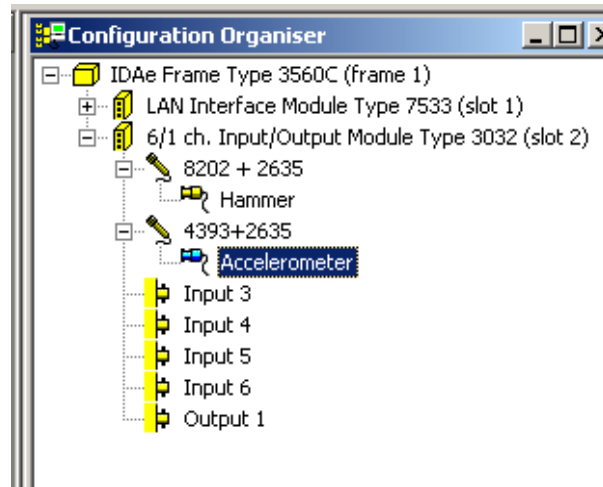
- 1x Pulse Front-end.
- 1x Pulse Front-end power supply.
- 1x PC with Pulse Labshop installed.
- 1x Ethernet crossover
- 1x Pulse Dongle.
- 2x B&K 2635 Charge Amplifier.
- 1x B&K 4393 Accelerometer.
- 1x B&K 8202 Hammer

It is possible to substitute another type of Charge Amplifier and/or accelerometer if these are not available.

Template Set-Up

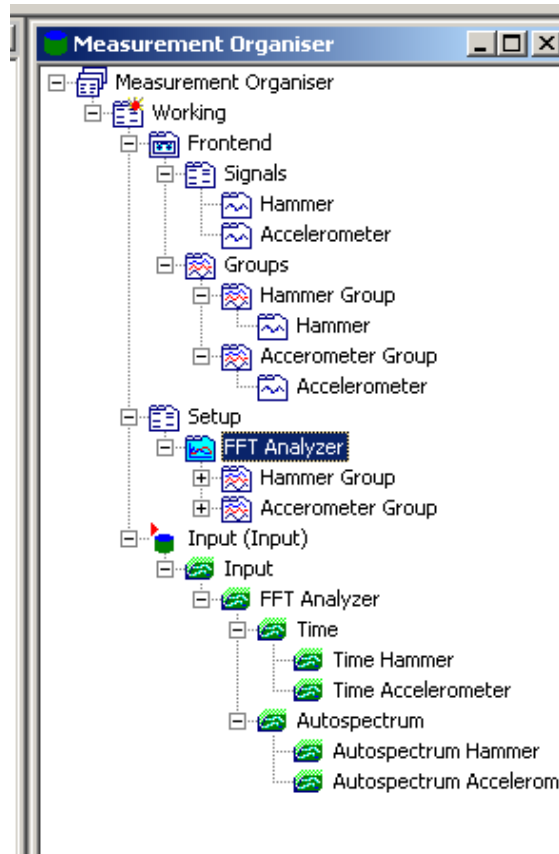
1. Follow the steps 1-11 in “Time Signal and Spectrum Analysis” to set up two signals:

- 8202 Hammer and 2635 Pre Amp in channel 1.
- 4393 Accelerometer and 2635 Preamp in channel 2.

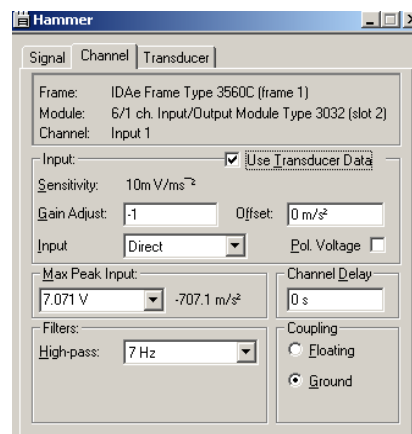


2. Activate the Template
3. Set up two signal groups in the Measurement Organiser and insert the signals as shown in steps 14 – 18 in “Time Signal and Spectrum Analysis”
4. Insert the FFT into Setup in the Measurement Organiser tree and then insert both groups into this, see steps 19 & 20 of the previous section.
5. Activate the Template

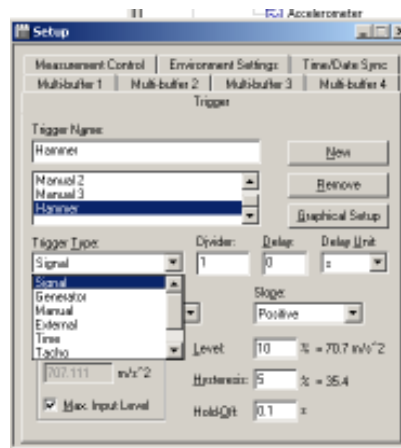
The measurement organiser should now look like this.



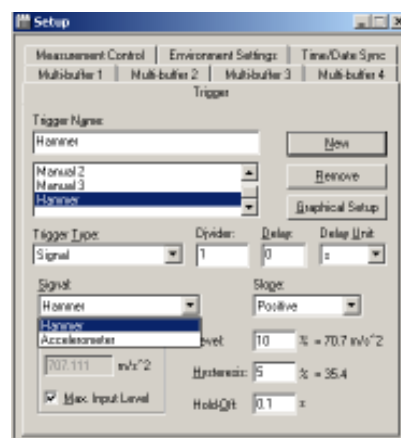
6. Right click on the hammer signal in the measurement organiser and select properties. Choose the Channel tab and change the Gain Adjust to -1 . This allows for the negative signal from the hammer.



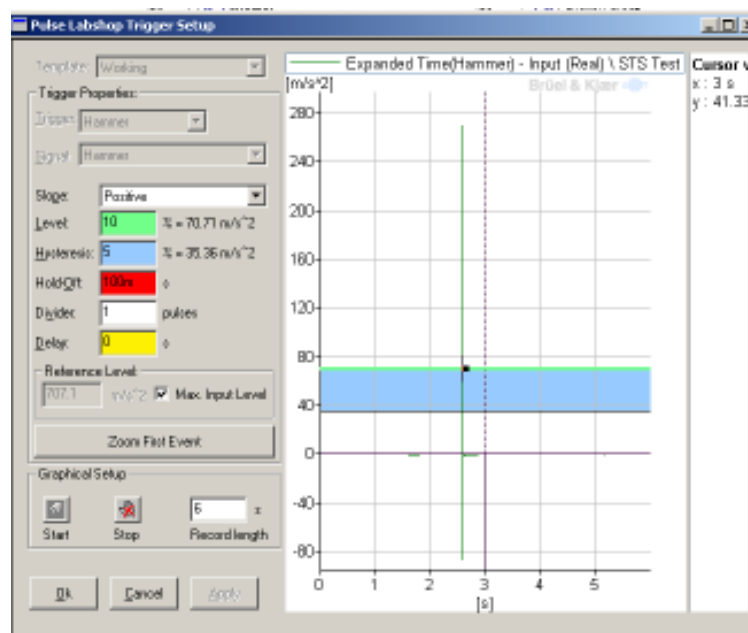
7. To set up the trigger right click Setup and left click on properties. Choose the trigger tab if it isn't automatically chosen. Click the new trigger button and insert a name, in this case hammer has been used. Change the trigger type to signal.



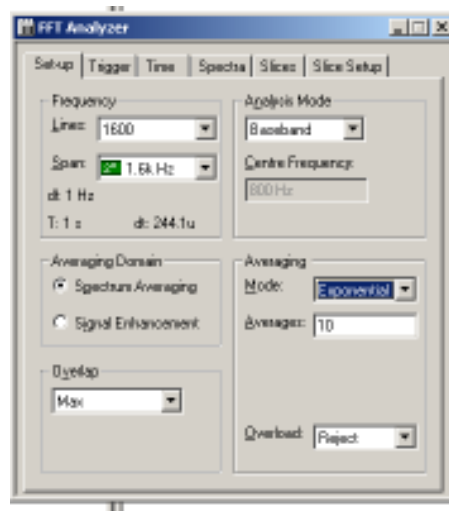
8. Chose the type of signal to trigger from, in this case the hammer.



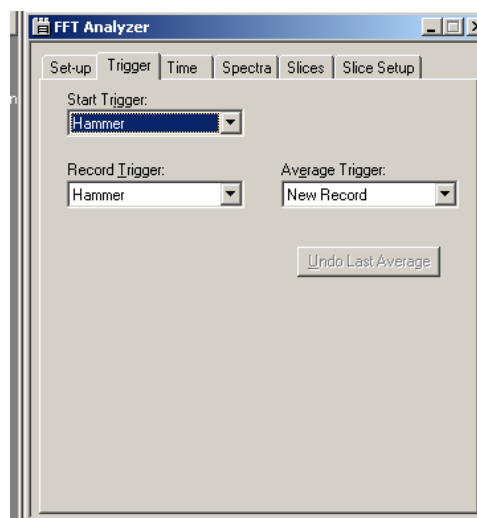
9. Click on graphical setup. Click on start and tap the beam with the hammer as if you were performing a measurement.



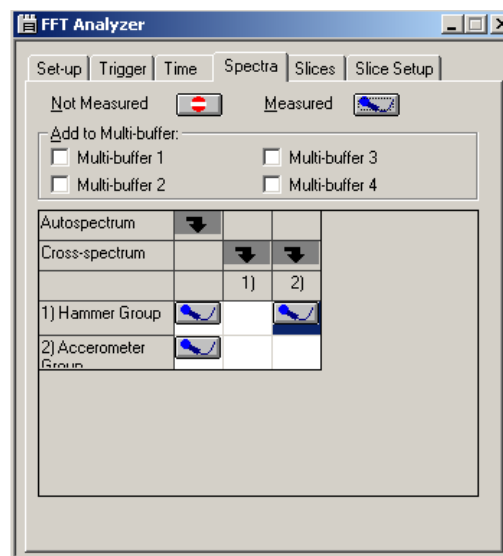
10. The impact of the hammer should be seen clearly. Set the Level and the Hysteresis lines to an appropriate amount (usually 10 - 20% for the Level, and 5% for the Hysteresis). This can always be changed later if the hammer is triggering too early or not at all. Also set the delay to -0.5s. Refer to sections 28 – 29 of “Setting up the Template in Labshop” for more information.
11. Click ok and close the Setup box.
12. Right click FFT Analyser and select properties. The analysis that will be used here will only look at frequencies up to 1.6 kHz. Set the frequency span to 1.6 kHz and also select 1600 lines so that the measurement resolution will be 1 Hz.



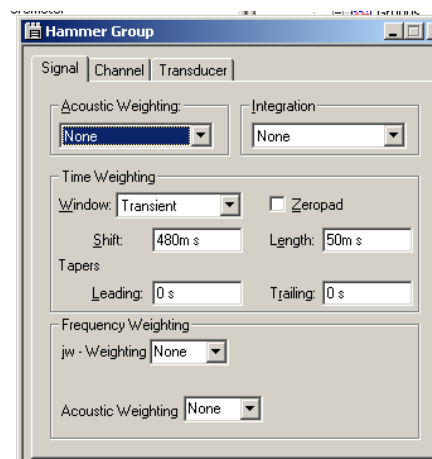
13. For this example three measurements will be taken and the results all will be averaged, to do this insert 3 for the number of averages. Change the averaging mode to linear.
14. Click the trigger tab. Set the Start Trigger to Hammer, Record Trigger to Hammer and Average Trigger to New Record.



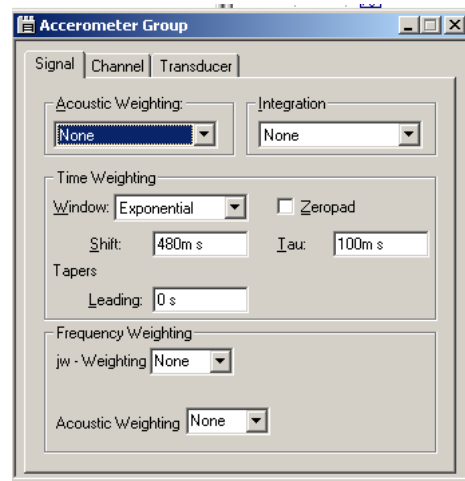
- Click on the Spectra tab and chose the hammer group under cross spectrum. Click the measured button. This allows the frequency response function to be measured and displayed. The window should look like the picture below.



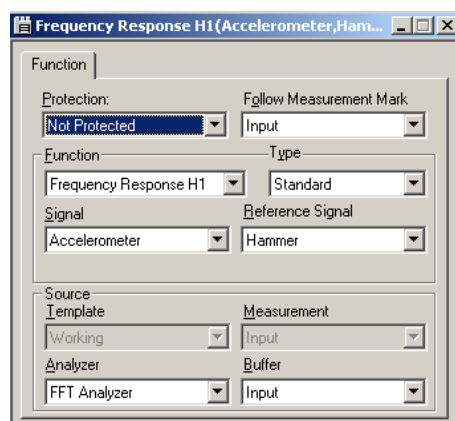
- Under Setup and FFT Analyser in the measurement organiser tree, right click the Hammer Group and chose properties. Change the Time Weighting Window to Transient, the shift time to 480ms and the length to 50ms. The shift time describes the time until the window starts, the length sets how long the window will be.



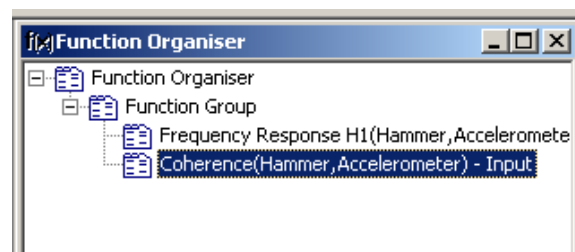
- Now open the properties of the Accelerometer Group. Change the time Weighting Window to Exponential, the Shift to 480ms and Tau to 100ms. Tau describes the rate at which the exponential curve decays (the smaller it is the faster it decays).



18. Activate the template and move to the function organiser.
19. Right click function group and select insert. This will add a function called Autospectrum (Hammer). Right click it and go to properties.
20. Change function to Frequency Response H1. Set the Signal as Accelerometer and the Reference Signal as Hammer.

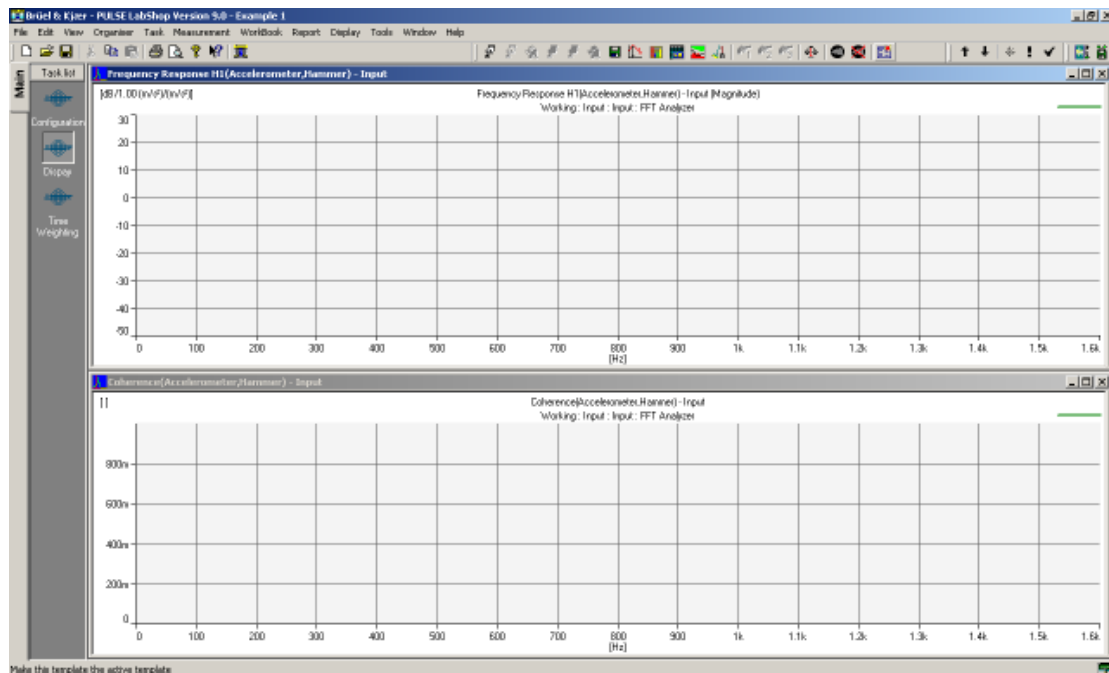


21. Similarly add a new function and change the function to Coherence.



22. Right click on both functions and chose Show As, Classic Display.
23. Follow steps 38 - 46 in “Time Signal and Spectrum Analysis” to create two tasks in the task bar, one to show the configuration and the other to show the two displays that have just been created.

24. The display layout should look like this.

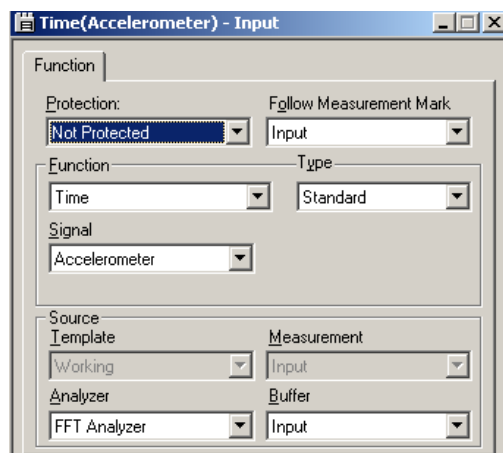


25. In the view tool bar select toolbars and measurement bar. This will provide you with buttons to start and stop the measurement. Make sure to save this in the display layout.

26. Now the time weighting functions need to be set. Move to the Configuration window. Under the Function Organiser create 4 new function groups:

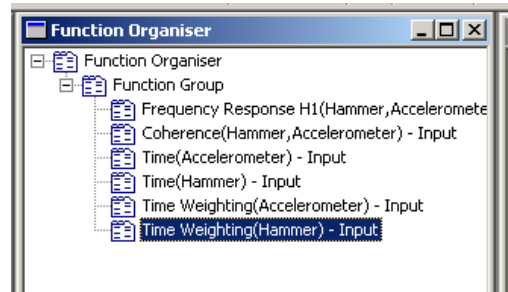
- Time (Accelerometer)
- Time (Hammer)
- Time Weighting (Accelerometer)
- Time Weighting (Hammer)

27. Add the 4 groups, right click on each and select properties.

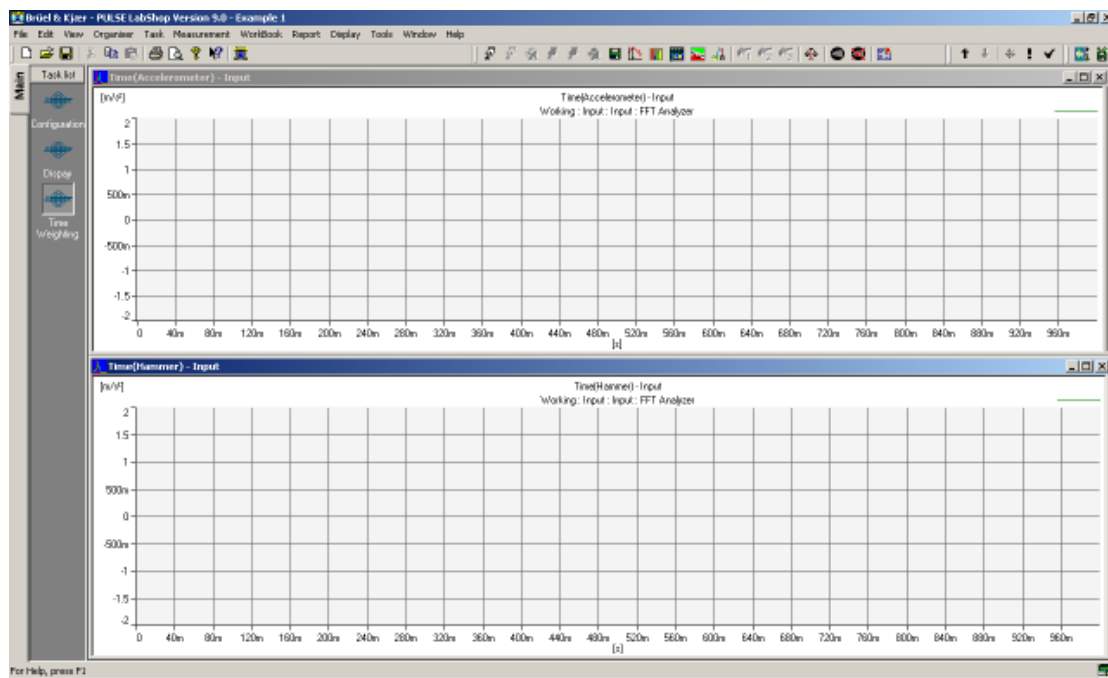


28. Select the appropriate function and signal for each.

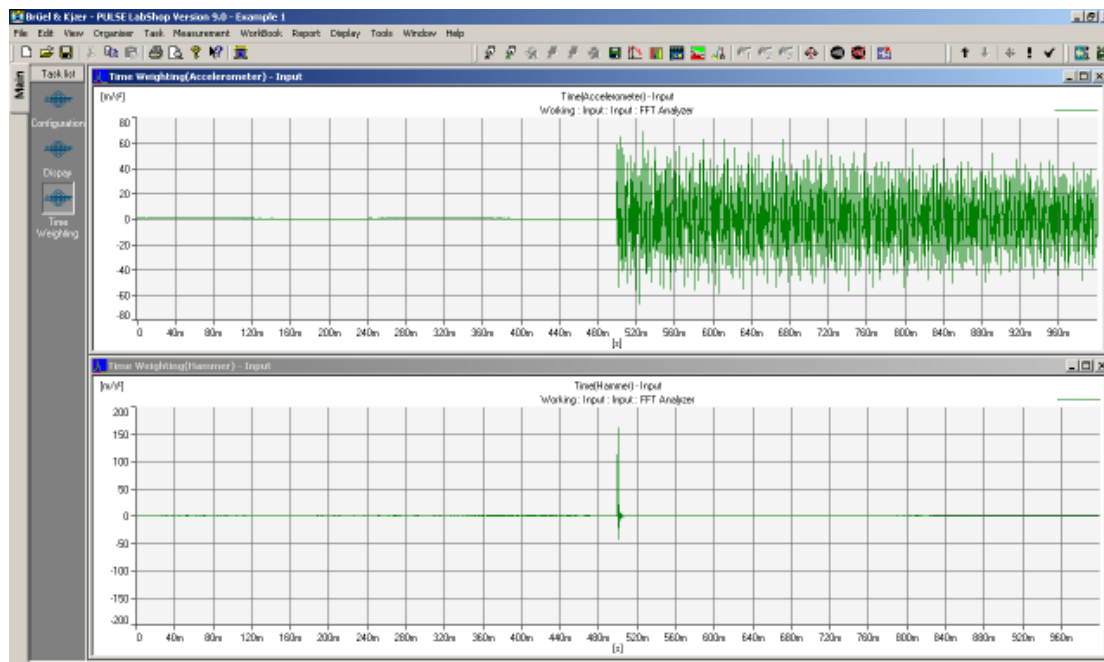
29. For the two Time Weighted functions select the time weight tab and chose Use Signal Weighting.
30. Close the box and activate the template.
31. Create classic displays for the first two Time Functions.



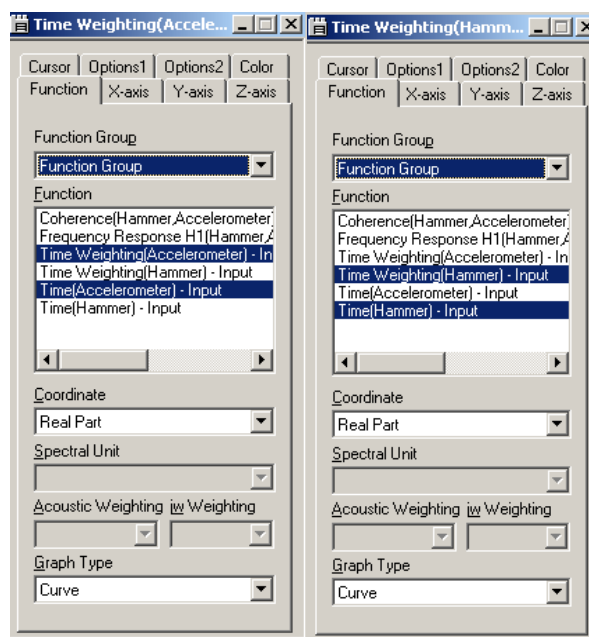
32. Create a new task in the task bar called Time Weighting and arrange similarly to the display task already created, showing the displays of Time (Accelerometer) and Time (Hammer).



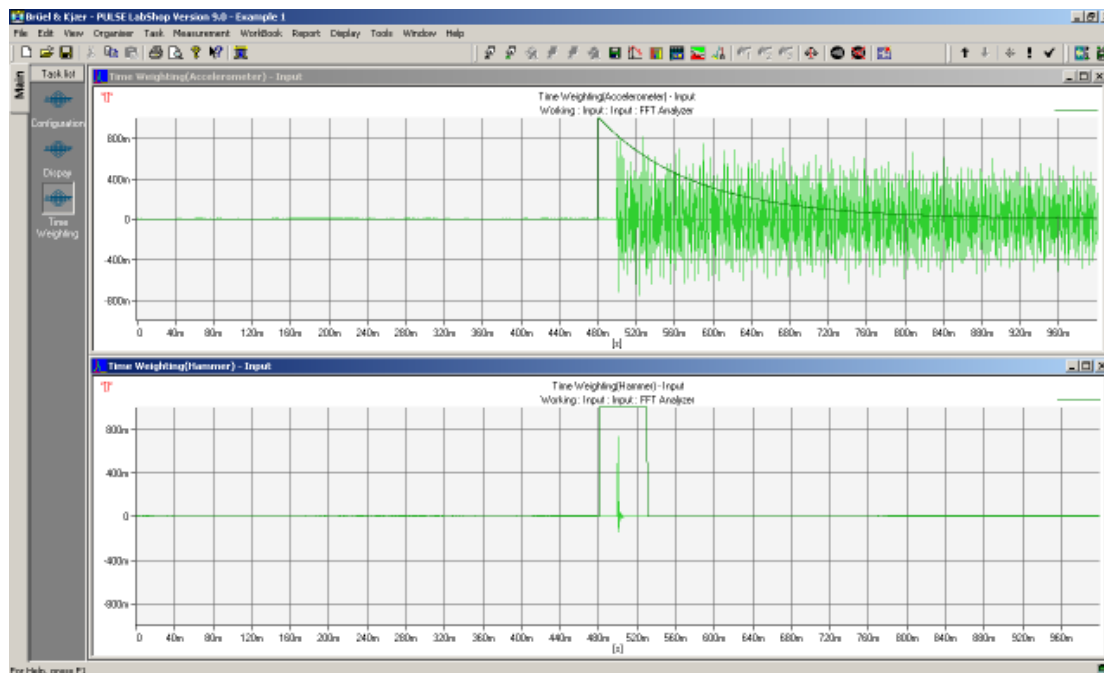
33. Activate the template.
34. Click the run button and tap the beam with the hammer. Click the Stop button. The time signals should show up like this. The graphs may need to be scaled, if so right click on the graph and select properties. Under the Y Axis tab click Autoscale. Alternately the use of the shortcut keys F10, Ctrl+F10 or Alt+F10 with different Autoscale options.



35. Right click on each graph and select properties. Under the function tab select Time Weighting (Accelerometer) for the first graph and Time Weighting (Hammer) for the second. The two properties boxes should look like this.

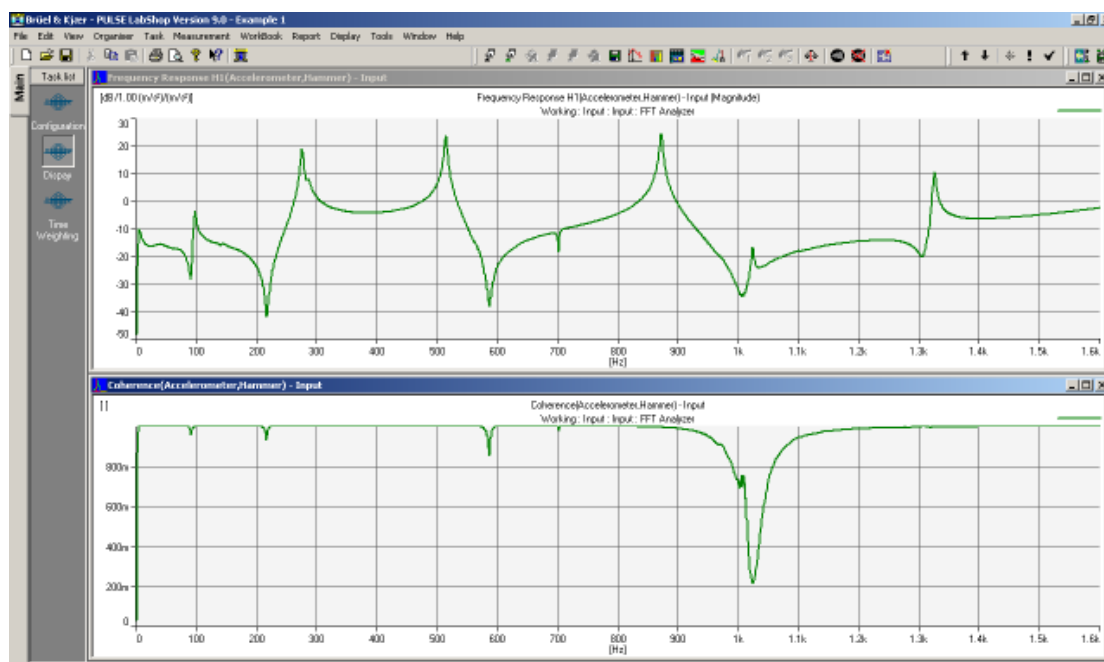


36. This will allow the time weighting function to be viewed over the time signal it will be weighting, allowing the weighting function to be adjusted to fit correctly. Notice that the zero value for each are not lined up. To change this click on the Y Axis tab and change the minimum value to -1 . Use the graphs to tune the Shift and Length values of the transient window and the Shift and Tau values for the exponential window. The transient window should cover the length of the force impulse. The exponential window should start just before the accelerometer signal and decay to zero before the end of the time record to avoid overlap error.



A measurement can now be taken.

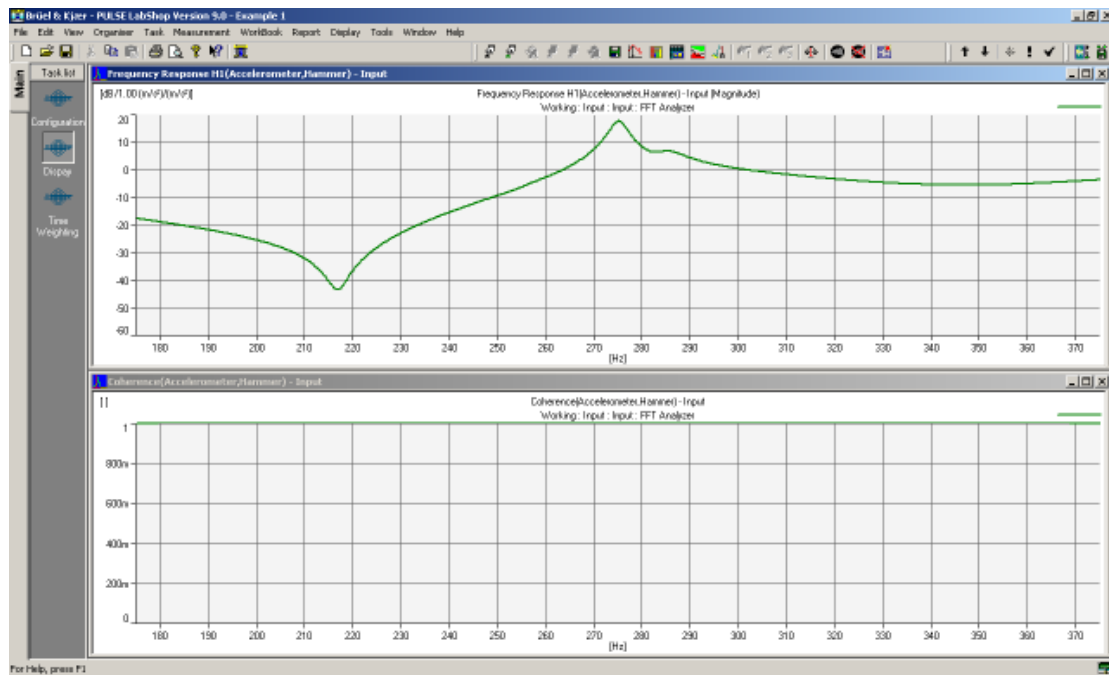
37. Click on the Display tab and activate the template. Click on the start button.
38. Take three successive measurements; making sure the beam has stopped vibrating before the next measurement is taken and that the hammer strikes the same spot each time. The FRF and Coherence displays should look something like this.



39. The coherence is a measure of how similar each of the averages are and it can be used as a check to see whether the measurement is good or bad. At each of

the resonances the coherence should be 1 as this is where the signal is the strongest. However at the anti-resonances it is expected that the coherence will drop as the signal is now not as strong and is being significantly influenced by noise in the system.

40. Now for example it may be important to zoom in on the resonance at 275 Hz and look at it with a greater resolution. To do this right click on the FFT Analyser in the Measurement Organiser and select properties. Change the analysis mode to Zoom and change the centre frequency to 275 Hz. Change the number of lines to 800 and change the span to 200 Hz. This has increased to resolution to 4 lines per Hz.



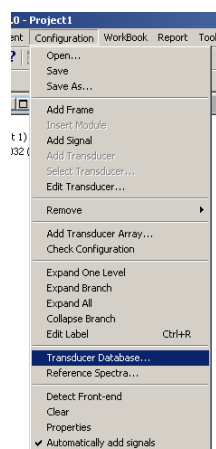
41. Notice how good the coherence has become; there isn't a drop now at the anti-resonance at 215 Hz.

Setting up a new transducer

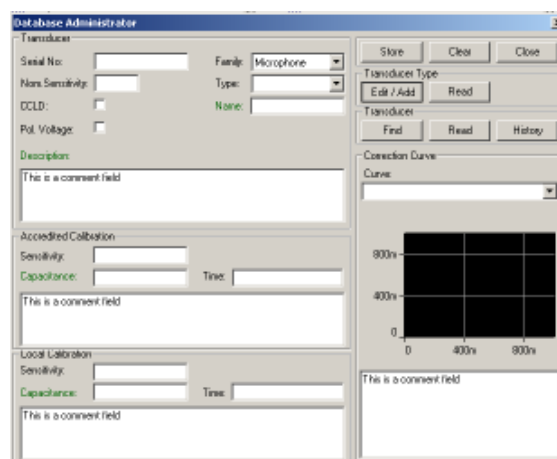
Setting up a new transducer under Pulse can be a useful task. While many B&K products are already in the Transducer database, many non-B&K products are not. The transducer database does not also readily support combining instruments, such as a 2635 Charge Amplifier and an accelerometer, which is still in use in many laboratories. By using adding these new combinations to the transducer database, it is easy to use any combination of transducers with Pulse.

In this example an entry will be made for a 4394 Accelerometer connected to Pulse via a 2635 Charge Amplifier.

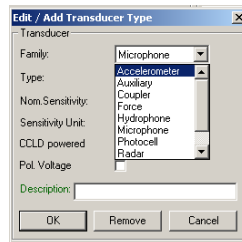
1. The first step in adding a new equipment setup to the Pulse Transducer database is to select 'Configuration' → 'Transducer Database'.



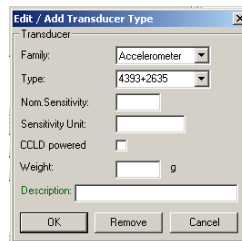
2. This will open the 'Database Administrator' window. Select 'Edit/Add' to add a new transducer setup.



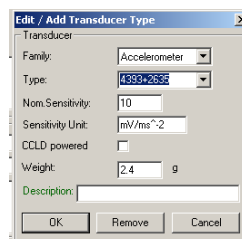
3. This will open the 'Edit / Add Transducer Type'. Select 'Accelerometer' from the 'Family' drop-down list.



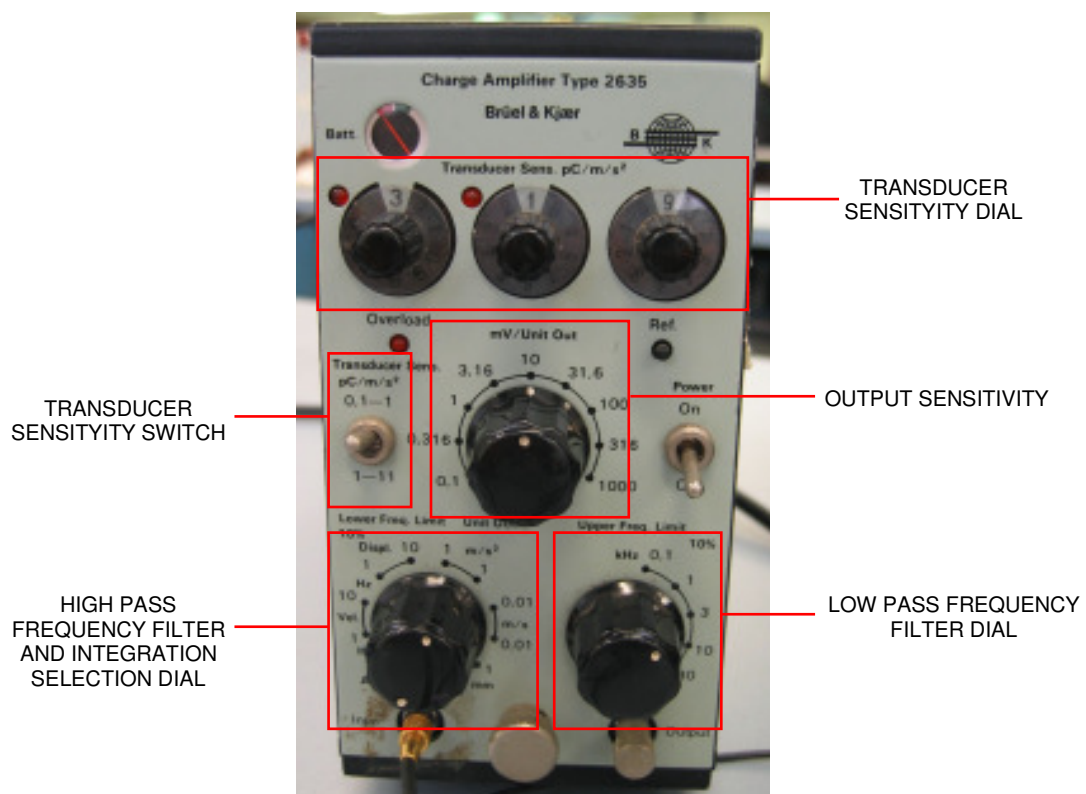
4. Type a name into the 'Type' edit box. In this case '4393+2635'.



5. Enter a 'Nom. Sensitivity' and 'Sensitivity Unit' for this combination. When a 2635 is used, the sensitivity can be set using the dials on the front of the unit. In this case select a 'Nom. Sensitivity' of 10 and a 'Sensitivity' of 'mV/ms⁻²'. The weight of the accelerometer can be found on the calibration sheet provided with all accelerometers. In this case a 4393 Accelerometer weighs 2.4g. A description can be added if desired.



Note: The sensitivity setting on the 2635 Charge Amplifier should be as shown in the picture below. The 'Transducer Sensitivity' is taken from the accelerometers calibration data sheet. The red LED indicates the decimal point and is adjusted with the 'Transducer Sensitivity' switch. The Charge Amplifier sensitivity or the sensitivity as seen by the analyser that the charge amplifier is connected to is changed by altering the 'mV/Unit Out' dial. The high and low pass filters are set using the lower two dials. The output can also be integrated with respect to time to provide a velocity or displacement signal, though it should be noted that some error can be introduced by integrating the signal as any noise in the signal is also integrated and can cause the signal to drift from a zero mean level.



- Click 'Ok' to apply the changes. Note that Remove can be used to remove a transducer from the Transducer Database. Also click the 'Close' button in the top right hand corner of the 'Database Administrator' window. The transducer is now setup and ready to use in a template.

Exporting measurements

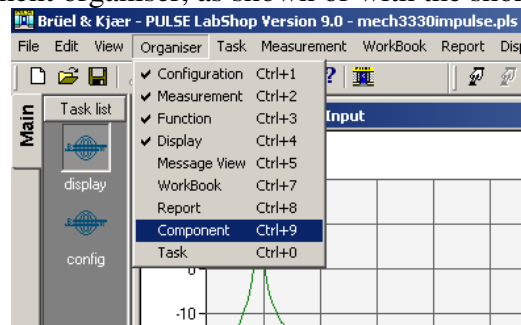
Once measurements are taken, post-processing or storing of the information outside of the Pulse labshop file is often needed. Pulse allows measurement data to be saved in various locations and with different file formats. How and what method should be used is obviously dependant on the final use. Two examples of note which may determine how you export the data are: modal analysis where another software system will be used for processing of the measurement data such as MEScope. In this instance it is advantageous to use the bulk save and file format given by Pulse in the configuration window. The other notable time is when you would like to output data

in the same units as displayed on the graphical output. Individual measurements can be saved like this through right clicking the graph and “save as”. However the most common post-processing of data occurs with the use of matlab. Exporting data in a manner able to be easily accessed in matlab will be explained in the following section.

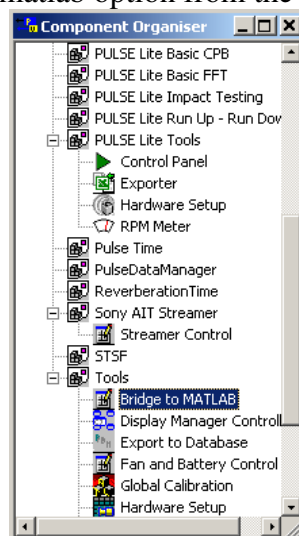
Export to matlab

Exporting data to matlab can be done in various ways. A method for exporting when the data is to be processed on a computer which you are not running the Pulse labshop software from is discussed in this section. See the next section for a slightly different and easier method when using matlab on the same computer as the Pulse labshop software is running.

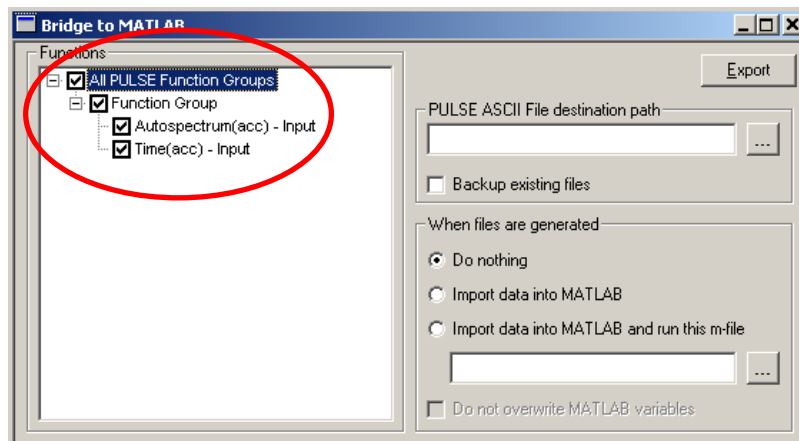
1. Open the component organiser, as shown or with the shortcut key Ctrl+9



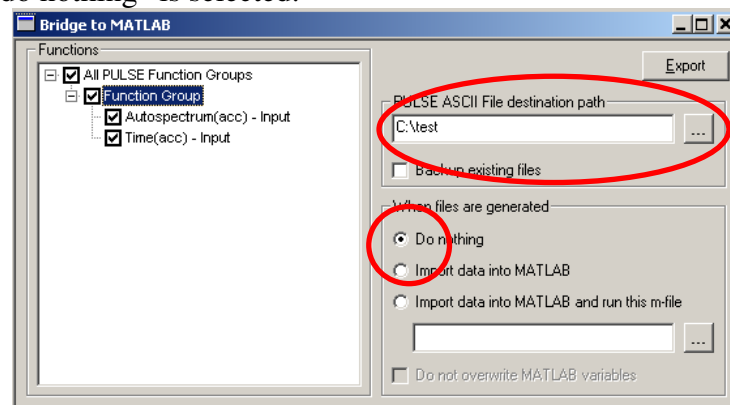
2. Now select the bridge to matlab option from the component organiser



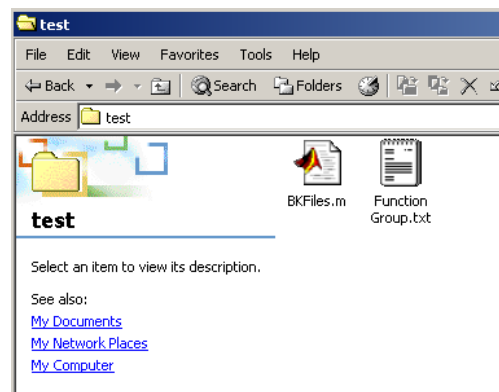
3. Upon selecting the bridge to matlab option a window should open as shown. The measured signals arranged into function groups are shown on the left hand side. Any combination of measured signals may be exported by selecting the check boxes, however here we will export all the signals being the autospectrum(acc) and time(acc).



4. After the desired signals for export are selected a folder to which they will be exported to needs to be given, in this case a folder called “test” was added to the C:\ directory to which we will export our measured signals, also the radio button “do nothing” is selected.

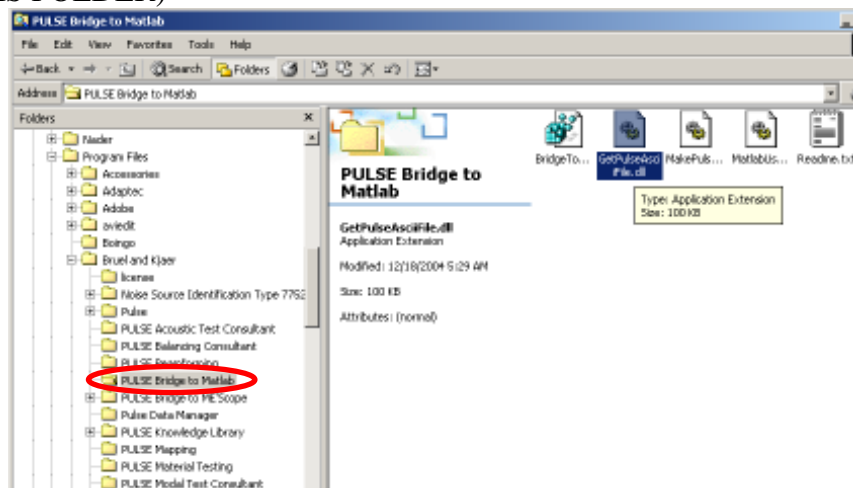


5. Once these selections have been made, click the export button.
6. Now it can be seen that our folder C:\test, contains two new files, one a text file containing our measured data, and the other a matlab m-file. There is only ever one m-file with the name “BKFiles.m”, but depending on the combination of signals being exported the number and name of the text files may change, but this will not be a problem as the same steps can be followed in either case.

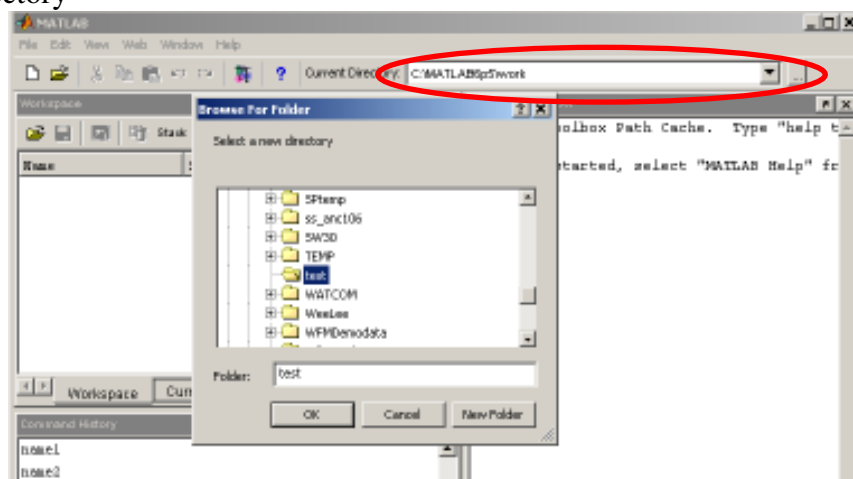


7. As we are assuming that the signals will be analysed on a different computer to the one which has the Pulse software installed, one more step needs to be followed before moving to the matlab environment. Browse to the folder called “Pulse Bridge to Matlab” and select and copy the file named

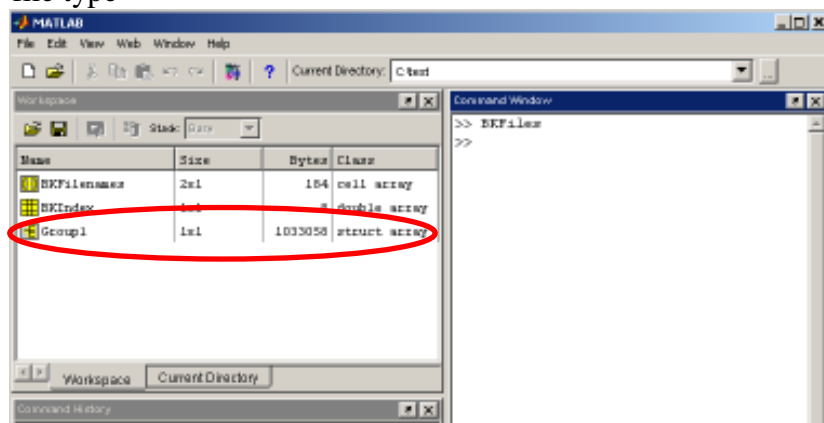
“GetPulseAscii.dll”. (NOTE, DO NOT CUT OR DELETE THIS FILE FROM THIS FOLDER)



8. Place a copy of the “GetPulseAscii.dll” file in the folder containing the data which was exported from Pulse, in our case C:\test
9. Now transfer the folder C:\test to the computer you wish to undertake the matlab post-processing in and open matlab
10. Once matlab is open, make the copied test folder as the matlab working directory



11. In the matlab command window type “BKFFiles”, the workspace will now contain three different files. The measurement data is located in the structural array file type

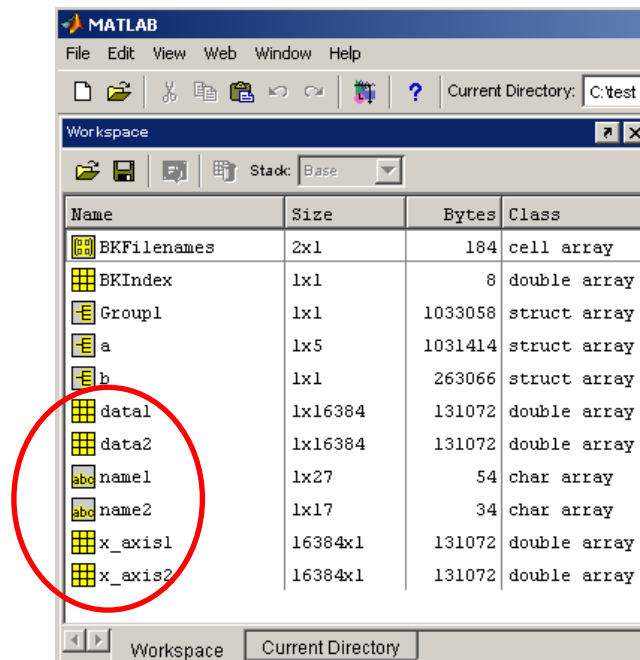


12. Now if you are familiar with structural array file types in matlab you will be able to interrogate and find all the measured data information within this file.

However if you are not familiar with structural arrays and wish to obtain the measurement data in a more familiar format containing only numerical values, copy and paste the following matlab script, between the % signs into the matlab command window and press enter.

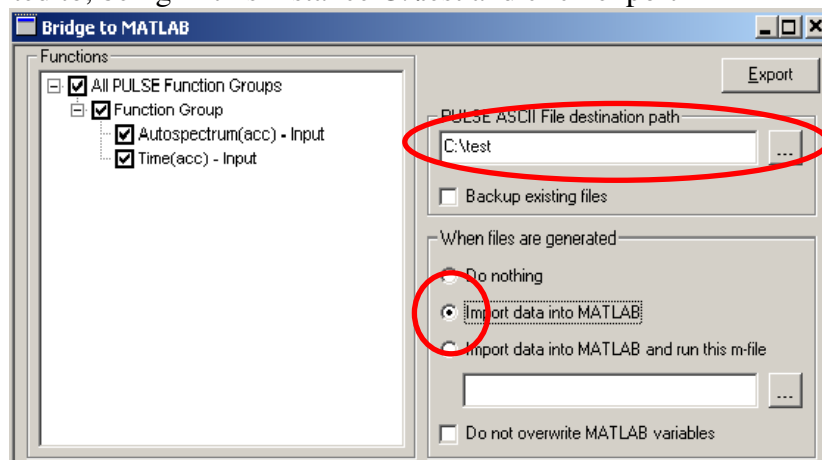
```
%  
for i = 1:length(BKFileNames)/2;  
    a = struct2array(eval(char(BKFileNames(i*2))));  
    if length(a)<2;  
        n = a('Name');  
        eval(['name' num2str(i) '=n;']);  
        b = a('DatasetSection');  
        c1 = b('Data')(1:end);  
        c2 = b('X_axis')(1:end);  
        eval(['data' num2str(i) '=c1;']);  
        eval(['x_axis' num2str(i) '=c2;']);  
    else  
        for j = 1:length(a);  
            n = a(j)('Name');  
            eval(['name' num2str(j) '=n;']);  
            b = a(j)('DatasetSection');  
            c1 = b('Data')(1:end);  
            c2 = b('X_axis')(1:end);  
            eval(['data' num2str(j) '=c1;']);  
            eval(['x_axis' num2str(j) '=c2;']);  
        end  
    end  
end  
%
```

13. Three new files will appear in the matlab workspace, being name\$, data\$ and x_axis\$ with the \$ ranging from 1 through to the number of measurement signals that were exported, in this case 2. The file named name\$ is the name of the measurement signal in Pulse, data\$ contains the measured data values and x_axis\$ contains the x-axis range of the corresponding data\$.

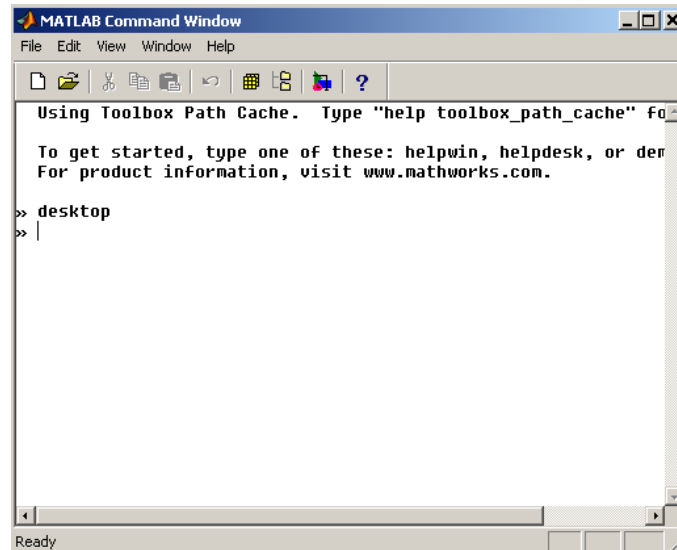


Exporting to matlab on the same computer as Pulse is running

1. Follow steps 1-3 from the following section
2. Select the “import data into matlab” radio button and the file folder to be exported to, being in this instance C:\test and click export



3. The matlab command window will now be open, although it looks like nothing has happened the data will now be in the matlab workspace. Type the command “desktop” into the matlab command window to view the matlab working environment in the default configuration, displaying command window and workspace etc.



4. Now follow steps 12 and 13 from the previous section.

Where to find more information

- Pulse demonstration videos, <http://www.bksv.com/Products/PULSEAnalyzerPlatform/PULSEDemonstrationVideos.aspx> (a nice feature shown in one video is the ability to have live measurements displayed in excel.)