SECTION 5.0
TABLE OF CONTENTS

SECTION 5.0
DYNAMIC SIGNAL ANALYZER (VNA)

Background

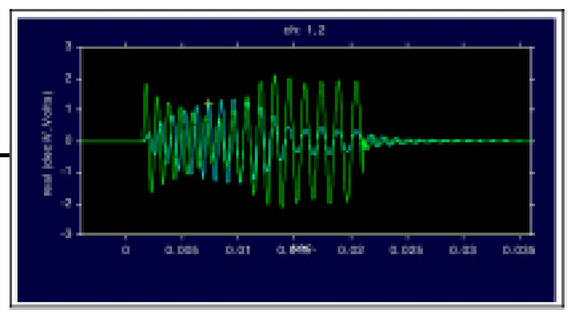
In SigLab software releases prior to v3.0, the typical software application for time histories, spectrum or transfer function and coherence measurements were ***vos***, ***vsa*** and ***vna*** respectively. With the release of SigLab v3.0, ***vna*** has evolved to include all of the measurement applications one would expect in a dynamic signal analyzer. In combination with virtual function generator, ***vfg***, this dynamic signal analyzer (still referred to as ***vna***) is optimized for the following multi-channel measurements on up to 16 channels:

- time histories
- power spectrum
- transfer function
- coherence
- cross spectrum
- cross and auto correlation
- impulse response
- FFT

vna Measurements

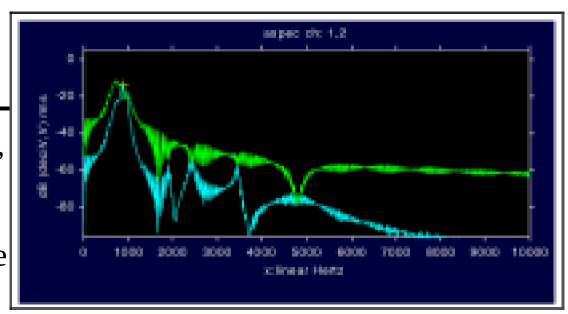
vna performs the following measurements:

Time Histories



his is the most basic measurement made by SigLab. The ***vna*** makes an excellent "digital" oscilloscope, with features such as pre- and post-triggering, a variety of display options, including Lissajous (Orbit) plots, and long record capture (***vcap***).

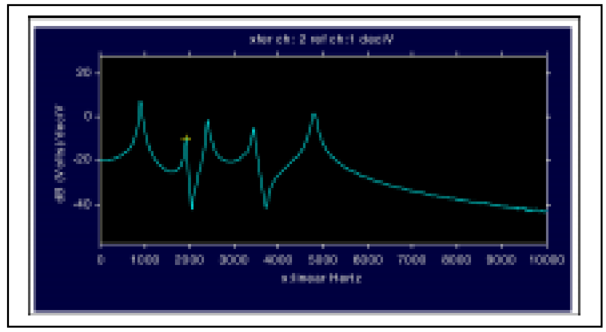
Power Spectrum



With 90 dB guaranteed spurious free dynamic range, SigLab excels at spectrum analysis. The ***vna*** software supports 12 different y-axis unit selections (dB, rms, rms/Hz etc.) plus optional single or double integration and differentiation.

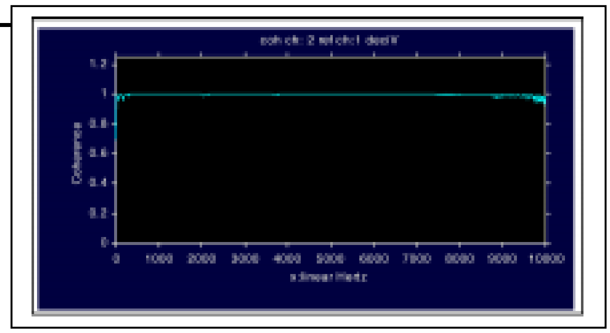
Transfer Function

vna is optimized to make transfer function measurements quickly and accurately. You can choose to have the excitation to the device under test (DUT) be automatically optimized. Features such as automatic rejection of overloaded data, and manual data reject are useful for modal impact testing.



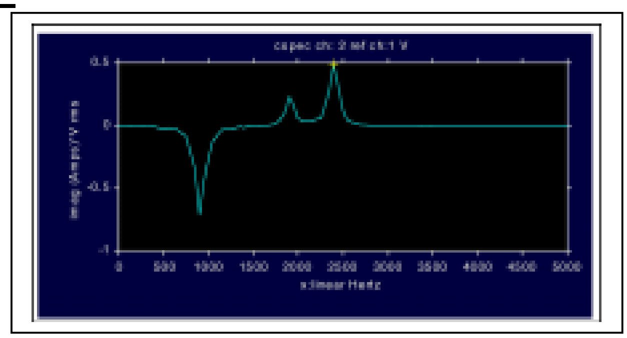
Coherence

Coherence is displayed as a guide to the quality of the transfer function measurement. Coherence measures the power from the DUT channel that is caused by the power in the excitation to the DUT. Both coherence and transfer function are displayed simultaneously and in “real-time”.



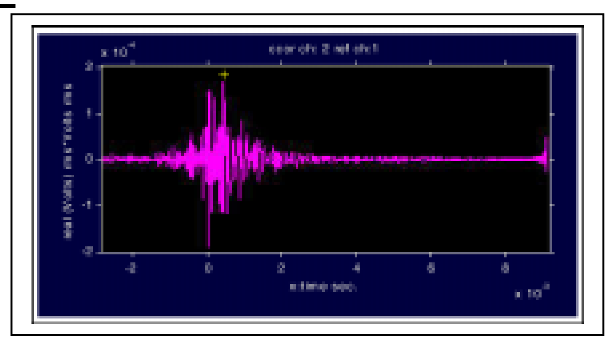
Cross Spectrum

The cross spectrum is used to measure complex power (e.g. volts*amps). When a noise intensity probe is connected to SigLab, the cross spectrum can be used to make acoustic intensity measurements.

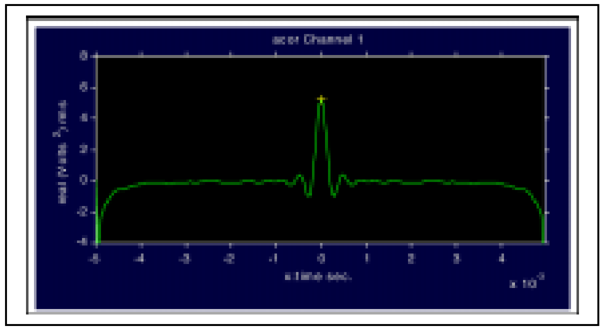


Cross Correlation

Cross correlation is an indicator of the amount of similarity between two signals shifted in time. It is often used to estimate transmission path time delays. It is sometimes used for acoustic measurements to determine the relationship between vibration of a structure (accelerometer) and sound that is produced by the vibration (microphone).

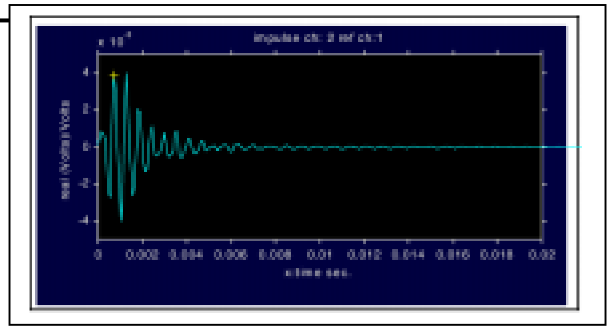


Auto Correlation



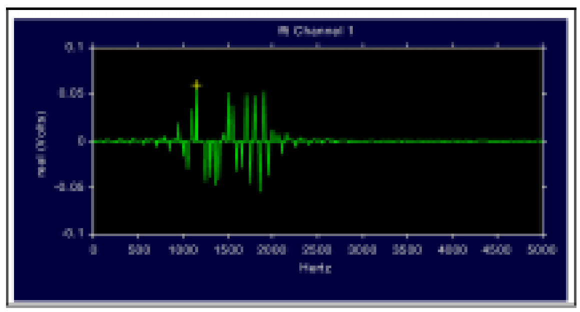
Auto correlation is a measure of the similarity between a signal and a shifted version of itself. It is very good at extracting periodic waveforms from a noisy environment. It is often used in acoustics for echo measurements and in the development of active noise suppression systems.

Impulse Response



The impulse response is computed by taking the inverse Fourier transform of the transfer function.

FFT



The FFT of a signal may be viewed in real-time. Since it is a complex quantity, the real, imaginary, or magnitude may be selected for display. The imaginary vs. the real may also be plotted.

vna Overview

Measurement Setup Window

The ***vna*** application consists of two main windows, the “setup” window and the “plot” window. The setup window controls all of the data acquisition parameters and many of the processing parameters. The setup window contains four control groups and a menu bar. The control group in the lower right quadrant will either control Triggering or Excitation parameters, depending on the selected Excitation mode.

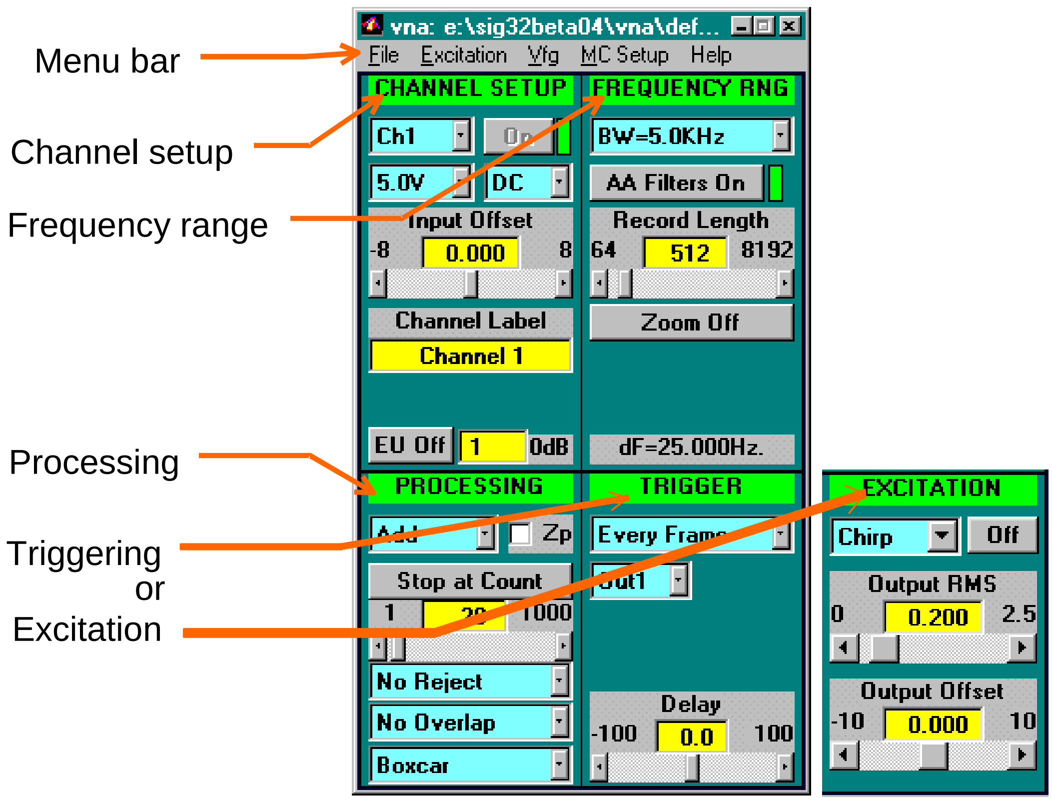


Figure 5-1 Measurement Setup Window

Plot Window

The plot window is resizable and contains:

- one or two axis in which measurement results are plotted
- cursor controls and readouts
- controls to define the type of measurement
- controls to define the measurement units
- menus for setup, display format, storage, and cross channel function selection

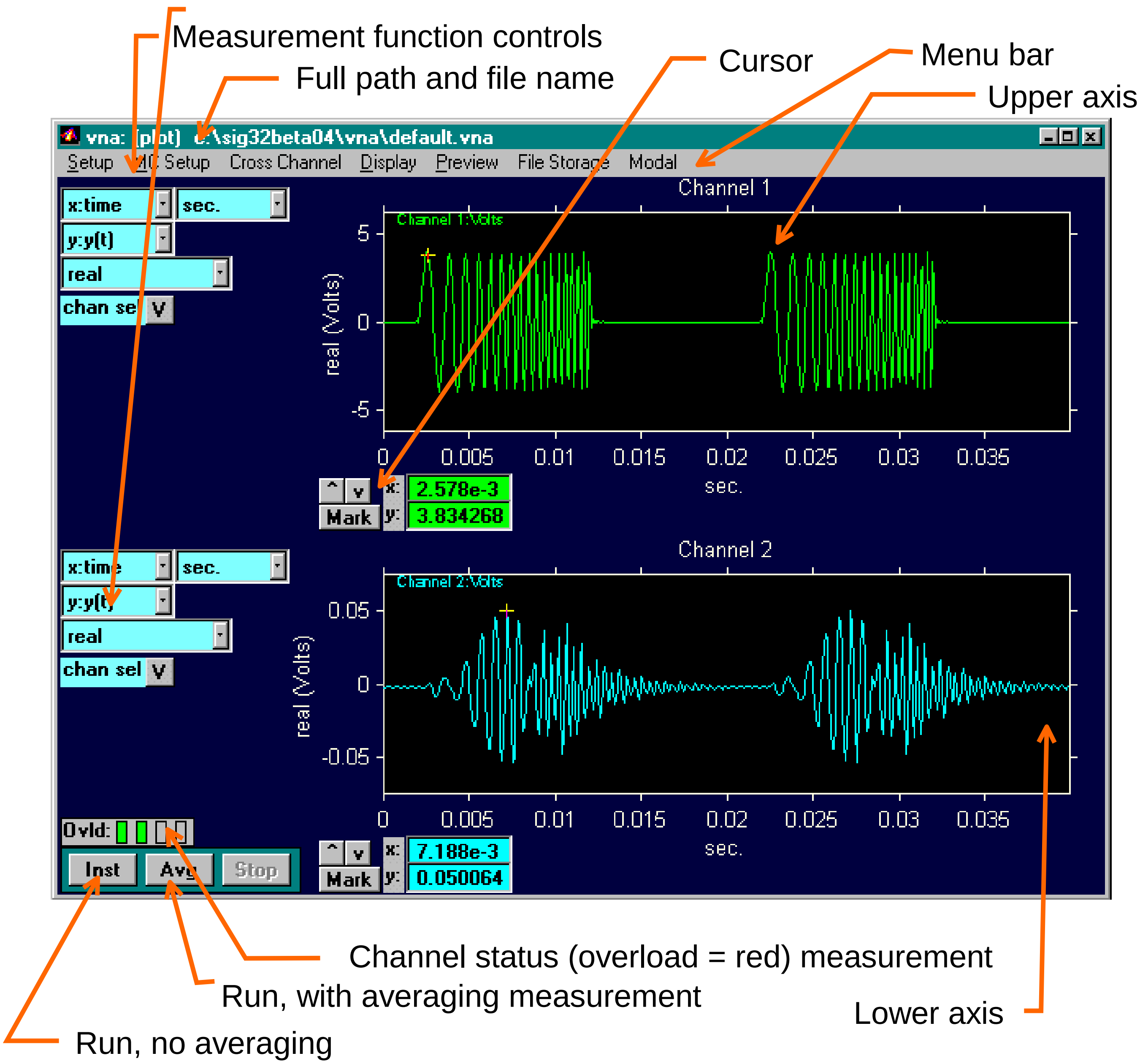


Figure 5-2 Plot Window

DUT Connection

This guide will include numerous measurement examples using the acoustic tube supplied with SigLab. Unless otherwise stated, it will be connected as described in the Device Under Test (Section 4.0).

Starting vna

You may start ***vna*** with either ***SigDemo*** (See Section 4***)*** by typing ***vna*** in the MATLAB® command window. When ***vna*** is typed at the MATLAB prompt, the application loads with the setup conditions stored in the *default.vna* file. Windows similar to the previous Measurement Setup and Measurement Plotting windows will appear.

🕮NOTE

The ***vna*** application uses both the input and output subsystems of SigLab and will not start if any other application (with the exception of the ***vfg***) is active. Other applications must be terminated for ***vna*** to run. See Common Ground, Application Conflicts.

Setup Window Details

First, the Menu Bar Selections are described below. Then the operation of each of the control groups is explained.

Excitation Menu

The SigLab hardware contains a flexible, high quality, output subsystem that may be used as excitation of the DUT for a variety of measurements. When the ***vna*** application is invoked, it takes control of this output subsystem.

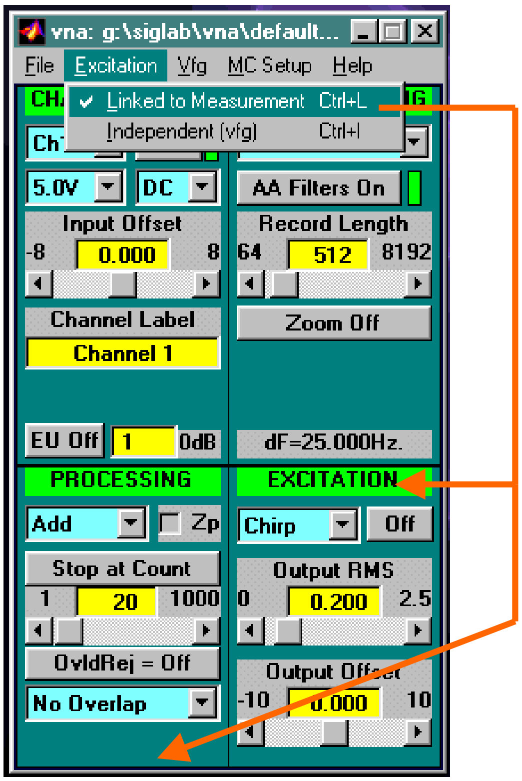
vna can make many measurements including transfer function and coherence. When the ***vna*** is used to make transfer function measurements, the output subsystem is often used to provide the excitation energy to the DUT either directly, or by driving a power amplifier and shaker. It can be a challenge, especially to new users, to properly set up the excitation signal parameters with the analysis window and triggering parameters. To simplify this setup process, ***vna*** provides the “Linked” mode where the excitation signal parameters are linked to the measurement parameters.

🕮 NOTE

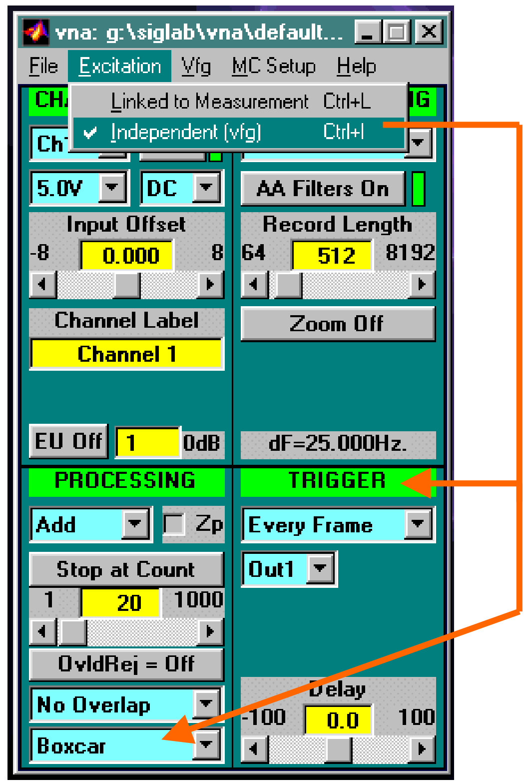
While in this linked mode, the SigLab output channel one cannot be controlled by the function generator (**vfg**) application.

When in linked mode, output channel one is controlled with the controls shown in the EXCITATION control group. Note also that the FFT analysis window popup is missing in the PROCESSING controls group because the analysis window is selected automatically when in linked mode.

Excitation "Linked"



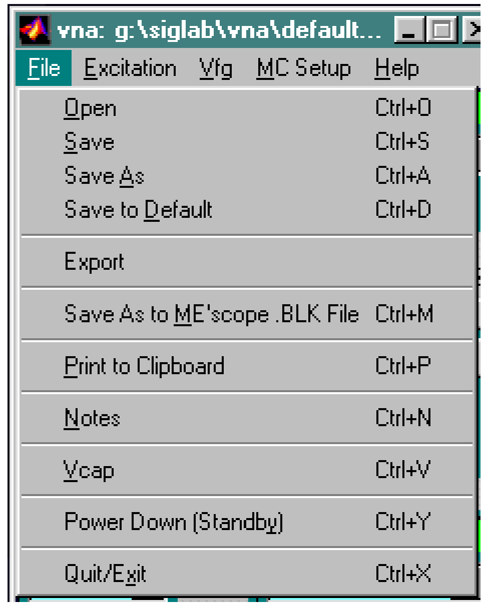
Excitation "Independent"



The "independent" mode is used for virtually all other measurement conditions. When you select this mode, the function generator (***vfg***) application has control of output channel one, and the TRIGGERING control group appears. You also now have control over the FFT analysis window as seen in the PROCESSING control group (above right).

vfg Menu

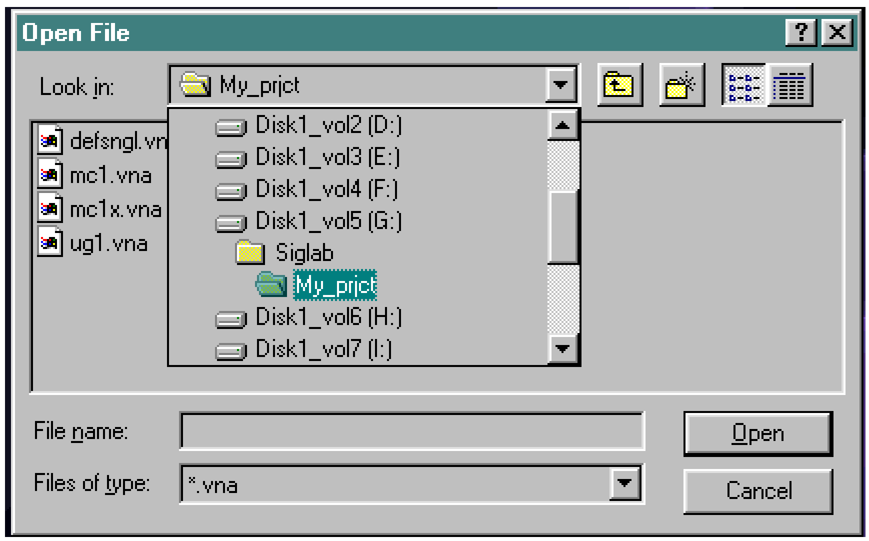
When you select this menu pick, the ***vfg*** will open or, if ***vfg*** is already open, the focus will be transferred to the ***vfg*** window.



File Menu

The file menu contains the submenus shown to the right. The first four submenus pertain to common file operations.

All ***vna*** files end with *vna.* as the extension. These files contain both measurement setup state and measurement results.



Selecting the *Open* submenu opens the Open File dialog as shown. Selecting a file from the list causes the following four actions:

1. Reposition the setup and plot display windows to coordinates stored in the file.
2. Set the measurement and plot controls to those parameters stored in the file.
3. Load and plot the measurement data stored in the file.
4. Update the path and filename in the setup and plot window title bars

The full path and file name of the file that is currently open is shown in the plot-window title bar. Selecting the *Save* submenu causes the control state and measurement results to be saved to the file that is currently open.

NOTE

The measurement functions to be stored are determined by the setting in the plot window File Storage dialog. (See File Storage Dialog)

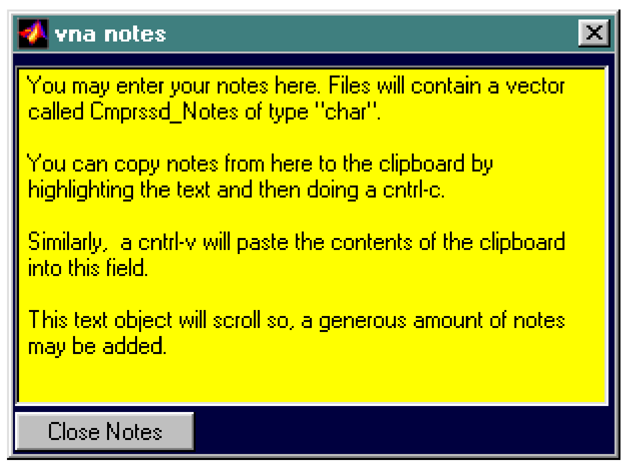
The *SaveAs* choice opens a modal dialog that is similar to the *File Open* dialog previously described. You can enter a new file name and/or select a new path. If a file is picked from the list, it will be overwritten with the new control state and measurement information. It is highly recommended that you save ***vna*** files with the *vna* extension. If you specify only the file name, the *vna* extension will automatically be assigned.

When ***vna*** is invoked with no input arguments, it loads the setup stored in the *default.vna* file from the *\siglab\vna* directory. When you choose the *Save to Default* menu pick, this file will be overwritten with the current measurement setup state and measurement data.

The *Export* selection is covered in the Link to Excel section of the manual (Section 4). This allows SigLab measurements to be exported to Excel for post-processing or presentation in that environment.

The *SaveAs to ME'scope* selection invokes the ME'scope file conversion Utility which facilitates moving SigLab measurements to the ME'scope analysis package (Section 7.0).

The *Print to Clipboard* menu makes a copy of the Setup Window and places it in the clipboard. An alternative method to do this is to use the alt-print screen keys.



It is often desirable to attach a note about test conditions to a measurement file. The *Notes* submenu activates a modal dialog box for note entry.

The notes box is shown to the right. If you saved your file before adding any notes, simply add your notes and re-save the file.

Highlighting, copying, pasting, and scrolling are supported within the notes text box.

The ***vcap*** menu invokes the long record capture software. With this auxiliary application, long data records can be captured to SigLab memory or to hard disk for subsequent analysis. See the ***vcap*** tutorial in Section 2.0 for details.

The *Power Down (Standby)* menu selection is used to manually reduce the power consumption when measurements are not being made. The power is reapplied to the analog and signal processing hardware when this menu is selected again (the check mark will disappear) ***or*** when a measurement is started using the *Inst* or *Avg* buttons.

🕮Note

It is prudent to reapply power at least10 seconds before a measurement is to be made to allow the analog circuitry to stabilize. This is especially important if you are using accelerometers powered by the internal accelerometer bias source since long time constants are always involved.

The *Quit/Exit* menu simply closes the ***vna*** application. If the setup file was not the *default.vna* file and measurement parameters were changed, the user will be prompted to save the changes.

Channel Setup Controls

The channel setup group of controls allows the user to set parameters that are unique to each channel in the system. For instance, the full-scale voltage and input offset may be set to a different value for each channel.

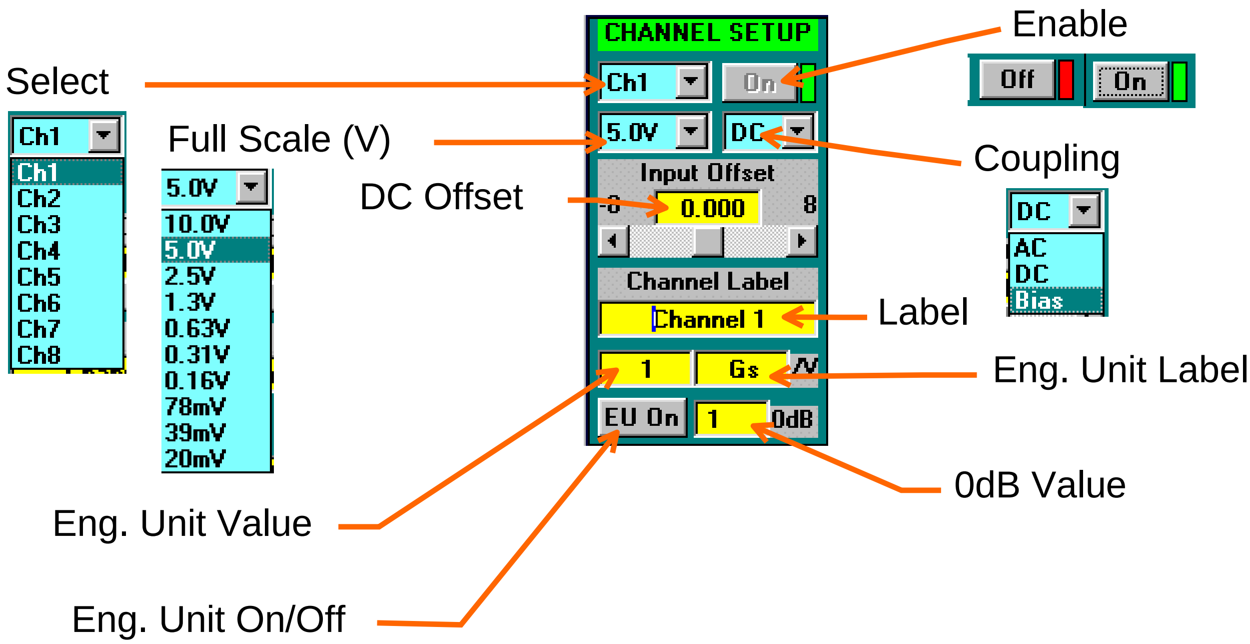


Figure 5-3 Channel Setup Control

Select

The *Select* popup contains the list of input channels present in the SigLab system. When you select a channel, the remaining objects display the setup states for the selected channel.

Enable

You activate or deactivate the selected channel with the *Enable* button. When you activate a channel, a green “LED” indicates that the channel is enabled and measurement data will be acquired by this channel when the *Inst* or *Avg* button is pushed in the Plot Window. When you deactivate the selected channel, the LED turns red. Typically, you should enable only the channels on which measurements will be made. In **vna,** channel one is always enabled and cannot be disabled.

Full Scale

The channel *Full Scale* voltage popup allows you to select from 20 mV full scale to 10 volts full scale for each channel. Given the measurement dynamic range of the SigLab hardware, you need not try to use a significant fraction of the full-scale range. If the input signal does exceed the full-scale range, overload detectors will report this condition to the operator in two ways. First, the red Input Overload indicator on the SigLab hardware will illuminate. Secondly, if the overloaded channel is enabled, the Channel Status indicator in the plot window will change from green to red.

Coupling

You use the *Coupling* popup to select between *AC* or *DC* input coupling. If the bias source option has been installed, the *Bias* option is also visible. When you select *Bias*, an ICP® compatible 4mA source is activated to power ICP-style transducers. See the ICP Bias Option section for more information.

DC Offset

The *DC Offset* control is only relevant and only visible when DC coupling has been selected. Current SigLab hardware allows a ± 8 volt offset range for the 10 and 5-volt full-scale input ranges, and a ± 2.5 volt offset range for the full-scale ranges equal to and less than 2.5 volts.

Labels

You can enter a 14-character custom label for each input channel in the *Labels* field. This information is used in the **vna** plot window for graph labeling.

Engineering Unit On/Off

The *Engineering Unit On/Off* button controls whether or not engineering units are factored into the measurements. The natural unit for the system is volts. However, more often than not transducers are used to convert a physical quantity (acceleration, force, velocity, etc.) into a voltage for measurement by SigLab. Engineering units are used to account for the transducer unit conversion and scaling properties such that the measured quantities will be interpreted as acceleration, velocity etc.

Engineering Unit Value

The *Engineering Unit Value* field is visible when you select *Engineering Units On*. You enter the specific conversion factor here. The entry is in *units per volt.* Transducers may be calibrated in units per volt or volts per unit. For example, an accelerometer might have a conversion factor of 25mV/g . In this case the entry in the field would be 1/(25mV/g) yielding 40 g/volt. Simply type 1/0.025 (Enter) in the field to invert the conversion factor.

The Engineering Unit Label field displays five characters, but more can be used if desired. Generally, a one or two character label will suffice.

Zero dB Reference

The decibel display format plots the relationship between two quantities. Typically, the quantities are related to power.

It is calculated by: $dB = 10 * \log(\frac{P_1}{P_0})$. Since power is proportional to the square of voltage, decibels can also be expressed as $dB = 10 * \log(\frac{V_1^2}{V_0^2}) = 20 * \log(\frac{V_1}{V_0})$ where the voltages are expressed as root-mean-square values. It is clear from both of these equations that $dB = 0$ when $P_1 = P_0$ or $V_1 = V_0$.

Therefore P_0, or V_0, may be considered as ***reference levels***. For instance if we are talking in terms of dBm (dB milliwatts) we are saying that $P_0 = 0.001$ watt. If we are talking of dBVrms or dBV for short (decibels with respect to 1 volt rms), we are saying that $V_0 = 1$Vrms.

The point is that *the reference level must be specified* before the dB measurement is meaningful. The normal default reference is 1.0 and if engineering units are not being used, this means that 0 dB is 1 Vrms.

The *0 dB Reference* field allows *you* to specify the zero dB autospectrum reference level in Vrms (or EUrms) for each channel to be different from the default of 1. So, if you want to have your 0 dB level be 10 Vrms, you just enter 10 in the 0dB field. If you want it to be 0.1Vrms, you would enter 0.1. Then, if you measure a 0.1Vrms signal (e.g. a sine wave, with amplitude correction on!) it would read out at 0 dB. If you measure 1Vrms signal, it would read +20 dB since the reference is 0.1Vrms.

Sometimes you would like to take a measurement that is being displayed in dB and force a specific point on the measurement curve to displayed as 0 dB. For example, assume that you measure a spectral peak with the cursor and it is at -13.9788 dB Vrms at 1000 Hz. Then, assume that you want to have this peak be at “0 dB”. Making this point 0 dB allows you to calibrate the graph to easily make measurements of other *peaks with respect to this one*.

To make this peak read out 0dB, the entry to the 0 dB field would be:

$dBref = 10^{-13.9788/20} = 0.2$ or in MATLAB notation: dBref=10^((-13.9788)/20) = 0.200.

Therefore, for this example, 0.2Vrms would be the 0 dB value.

A shortcut function called *sz* (for “***s***et ***z***ero”) is provided to make this easy. Simply type:

sz(-13.9788)

into the zero dB field, and the sz function will return the proper value($V_0 = 0.2$ Vrms).

You could also just switch to the Vrms display mode, cursor the peak at 1000 Hz, and enter this number (0.2 for this example) in the 0 dB field. Once the 0dB reference has been set to 0.2, the 1000 Hz peak will read 0dB and other measurement points will read out with their dB values with respect to the 1000 Hz spectral component.

Taking this all one step further, microphones are often used with SigLab. Typically, the 0 dB reference point is set at the threshold of hearing which is a sound pressure level of 2.0e-4 µbar rms. Therefore, the 0dB field would be set to 2e-4 since the 0 dB ref works with either Volts or Engineering Units.

Microphones are calibrated to produce a specified amount of voltage per µbar. Here is where the Engineering Units (EUs) come in. Assume, for this example, the microphone produces 5 mV/µbar. To enter the appropriate Engineering Units the numeric entry would be 1/0.005 (= 200) with an Engineering Unit Label of µbar.

The system will now read the sound pressure level in dB with 0 dB being 2e-4 µbar.

Next, assume a microphone calibrator is to be used to check the microphone calibration. Assume the calibrator generates a Sound Pressure Level (SPL) of 114 dB. The sound pressure level (L) is L = 20*log10(p/2e-4) therefore

p = 2e-4*(10^(L/20)) = 100.2374 µbar = pstd.

Assume that when the calibrator is measured the cursor indicates an SPL of 113.25 dB. This represents 0.75 dB error. To correct for this microphone calibration error, the engineering unit must be recalculated. Note that we are putting our complete faith in the microphone calibrator!

The measured sound pressure is:

pmeas= 2e-4*10^(113.25/20) = 91.9454 µbar and therefore the engineering unit must be corrected by the ratio of pmeas to pstd.

The new engineering unit will then be 1/(5e-3*pmeas/pstd) = 218.0. Entering this number will then provide the desired reading of 114 dB at the calibrator frequency.

To simplify this task, we have included a MATLAB routine called *calmic.m* that returns an adjusted engineering unit. Typing *help calmic* at the MATLAB prompt returns:

```
» help calmic

 function calmic(nominal_cal_factor,dBmeasured,dBSPLref)

   Inputs to calmic
   nominal_cal_factor: An initial guess of the actual calibration
                       for the microphone, e.g. 200 micro-bars/volt.
                       This value comes with the microphone but it
                       doesn't have to be a close guess. This number
                       was used as the initial engineering unit factor.
```

```
dBmeasured:        The measured level in dB using the
                   nominal_cal_factor. This number will probably
                   not be exactly what you are aiming for. You will
                   get this number by cursoring the waveform and
                   then reading out in dB.
dBSPLref:          The level in dBSPL that is being
                   input to the microphone from a "known" source.

Returns:
    adjusted_cal_factor: Use this number as your new engineering
                         unit factor.
```

For this example *calmic* is used as follows:

```
» calmic(200,113.25,114)
ans =

  218.0369
```

MC Setup Menu

When you select the MC Setup (Multi-Channel Setup) menu from the Setup or Plot Windows, a new window opens. This window provides a view of the channel-related information for all channels in a tabular view. The screen capture below shows the MC Setup window, as it would appear for an 8-channel system. This method of entering and viewing channel specific parameters is more convenient for dealing with many channels.

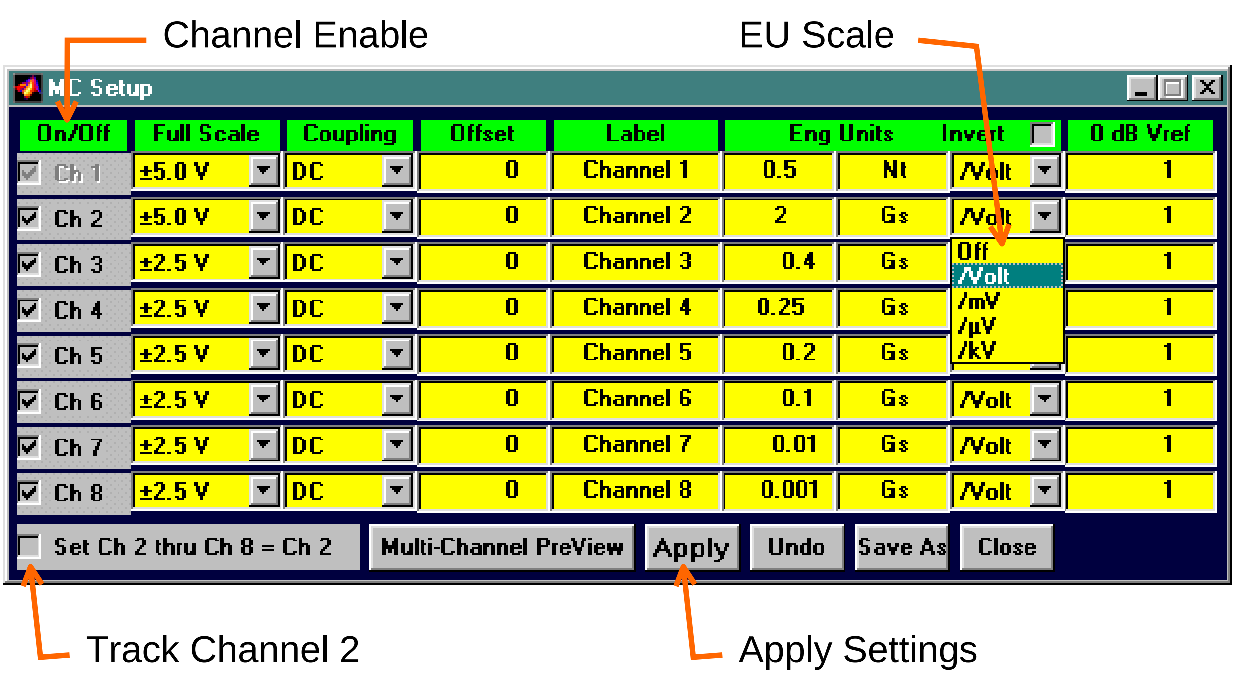
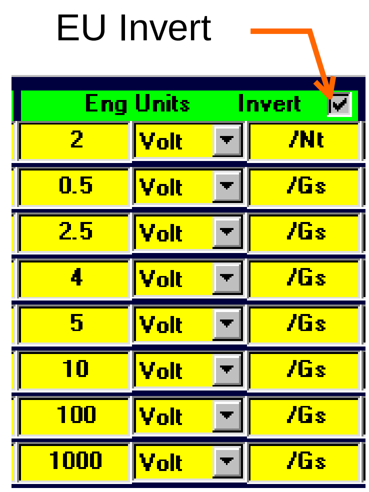


Figure 5-4 MC Channel Setup Control

The purpose and operation of each control is the same as those previously discussed. The one exception, Engineering Units, has a selectable scale factor of Volt, mV, etc. This scale control is also used to turn Engineering Units on/off.

The “Invert” check box allows the entry of the Engineering Unit conversion factors to be in terms of Volts/EU as shown to the right.



By checking the *Set Ch 2 thru …* check box, you can automatically cause any changes made to channel 2 to be made to all channels beyond 2.

Until the *Apply* pushbutton is pushed, the changes in the various entries will have no affect. The *Undo* button can be used to restore the initial state of the controls *until* the *Apply* pushbutton is pushed.

The *Multi-Channel PreView* button launches the multi-channel preview window.

The *Save As* button opens the Save As dialog box.

Frequency Controls

The Frequency Range Controls group sets the sampling parameters common to all channels.

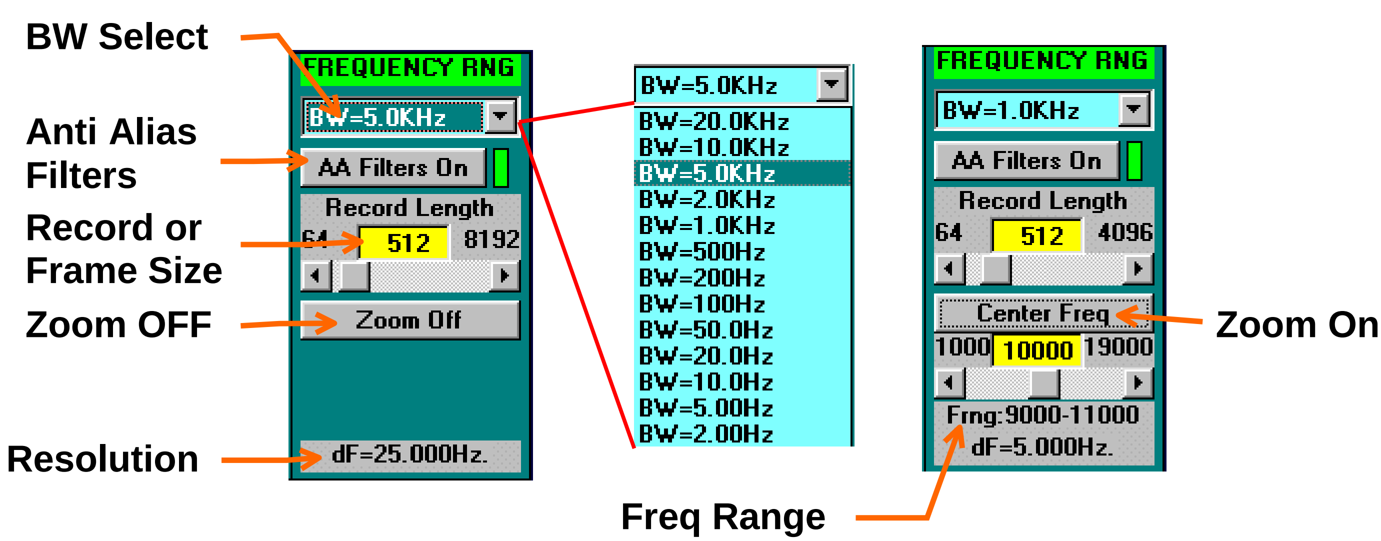


Figure 5-5 Frequency Range Selection Control

Analysis Bandwidth Selection

Since most of the measurements made with the Dynamic Signal Analyzer will be in the frequency domain, the *BW Selection* popup menu shows a list of available analysis bandwidth (BW) selections. The example above applies to the 20 kHz systems. The 50-21 system offers 13 selections between 5 Hz and 50 kHz.

🕮NOTE

SigLab's sampling frequency is always given by: Fs = 2.56*BW.

Anti Alias Filters

The "*AA Filters*" pushbutton allows you to turn the anti alias filters on/off for all but the highest available bandwidth.

Bandwidth & Record Length

The selection of analysis BW *and* Record Length (or "Frame Size") determines the ultimate frequency resolution of the analysis:

dF = 2.56*BW/(Record_Length)= frequency resolution

The actual frequency resolution will be greater than this value when an FFT analysis window is used (e.g., Hanning).

In *Base Band* (Zoom is off) analysis the maximum record length of 8192 points yields 3201 frequency domain analysis "lines". The number of analysis lines N is related to the Record Length by:

N = (Record_Length/2.56)+1

Zoom

Zoom or band translation allows you to focus the SigLab analysis power onto a specific frequency range. This is very useful for studying closely spaced resonances or spectral phenomenon with closely spaced side bands. With *Zoom on* (upper right graphic), the maximum record length is 4096 points. The *Center Freq* slider allows you to set the Center Frequency and the BW popup allows you to pick the analysis bandwidth. The frequency range of the analysis will be the Center Frequency ± the BW and is displayed in the *Frng: lower – upper* readout.

Excitation Controls

The Excitation Controls group is only visible when the analyzer is operating in "linked mode." This "linked mode" is usually only used for estimating transfer function. When in this mode, the **vna** application takes control of output channel one and sets the output excitation according to the selected input-channel parameters.

Excitation Select

For optimum transfer function measurements, the excitation should excite the DUT over the desired analysis bandwidth but not beyond this frequency range. The chirp and band-limited random are the most popular choices for accomplishing this task. You select either the Chirp or Random excitation from the *Excitation Select* menu.

Output On/Off

Until a measurement is started by pushing the *Inst* or *Avg* buttons, excitation is Off as shown by the *Output On/Off* pushbutton. This button may be toggled to the *On* state to check the excitation level to the DUT.

As the Frequency Controls (described in the previous section) are changed, the excitation signal is automatically adjusted to track these changes. This considerably simplifies transfer function measurements since you do not need to be concerned about the detailed parameters of the excitation.

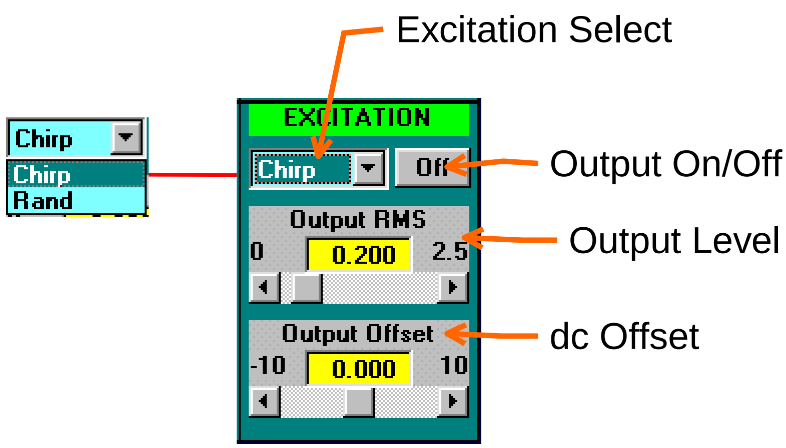


Figure 5-6 Excitation Control

When you want more control over the DUT excitation parameters, (e.g., to measure distortion, or time domain responses), you use the ***vfg*** control the excitation as described in the *Excitation Menu* section.

For more information on the measurement of transfer functions please refer to the application note: *Estimating Transfer Functions with SigLab* (included in the appendix).

Output Level

The *Output Level* slider sets the output drive level in volts rms. If the proper drive level to the DUT is not known, it is prudent to start with a small level. The peak level for Random can easily be 4 times the rms setting. It is only approximately 1.414 times the rms level for the Chirp selection.

Output Offset

A dc offset can be added to the output signal with the *Output Offset* slider. This offset is present regardless of whether the output is on or off.

Triggering Controls

SigLab offers full control over triggering parameters. These triggering controls only appear when the excitation is set to *Independent*.

Trigger Mode

The trigger *Mode* popup allows you to selection one of five triggering modes:

- *Off (Free Run)*: no triggering. Acquisition and processing repeat on a record by record basis. This mode is often used in spectrum analysis or while performing initial setup.
- *Every Frame*: wait for a trigger event for each input record. This mode is used for time-domain analysis as well as estimating transfer functions for modal analysis by impact testing.
- *1st Frame*: waits for a single trigger event, then subsequent acquisition is free-running.
- *Manual Arm*: Will not detect a trigger event until the *Arm Button* is pushed and the *Arm LED* turns green (see graphic below). The trigger must be re-armed after each input record.
- *1st-Manual Arm*: Will not detect a trigger event until the *Arm Button* is pushed and the *Arm LED* turns green (see graphic below). The trigger need only be armed once since, after the first trigger event, the acquisition is free-running.

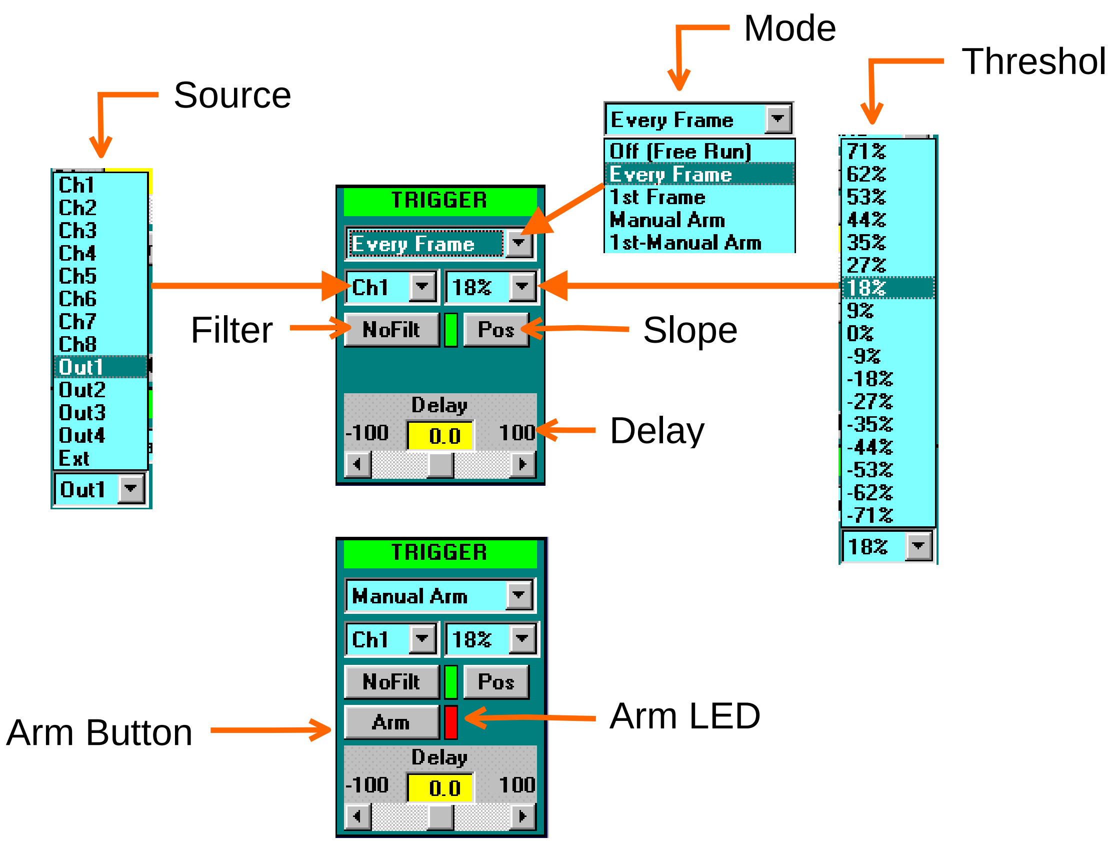


Figure 5-7 Trigger Setup Control

Trigger Source

The trigger *Source* popup selects the signal to be monitored for a trigger event. There are three source categories to choose from:

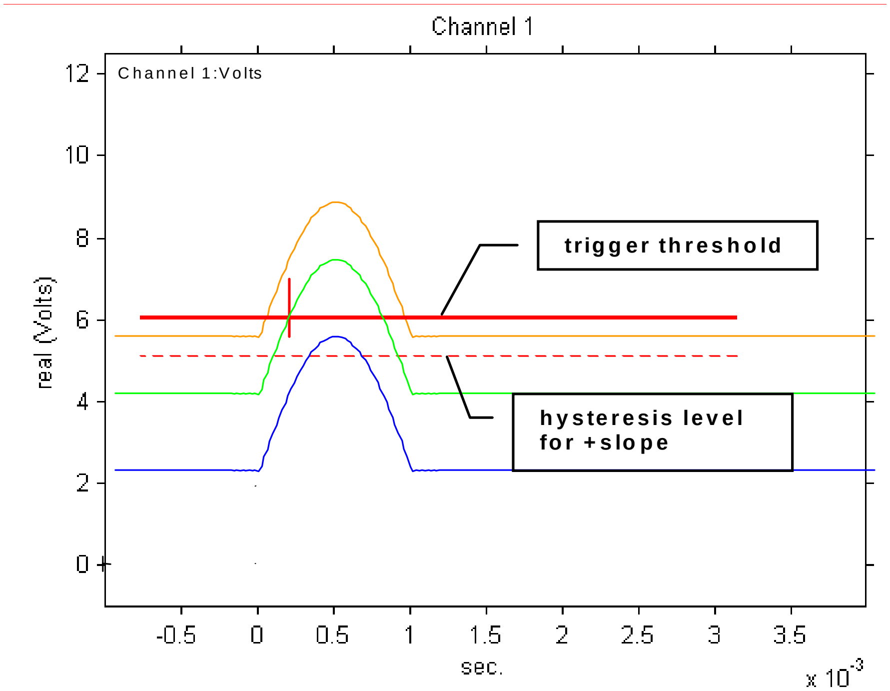
1. input channels (Ch1…ChN, analog)
2. output channels (Out1 … OutM, digital)
3. External (Ext, digital).

Input Channel as Trigger Source

When an *input channel* is selected as the trigger source, the *Threshold*, *Slope*, and *Filter* controls become relevant. Triggering from an analog signal can often be a frustrating task for many users. One of the major sources of confusion is the selection of the proper trigger threshold, particularly for uni-polar pulses like those from a modal impact hammer.

The trigger circuitry in SigLab has built in hysteresis to prevent false triggering on noisy signals. The amount of this hysteresis is fixed at 9% of full scale, which also happens to be exactly equal to the trigger threshold resolution.

Consider the example to the right. Assume the trigger level is set to 62% and the full-scale range is 10 volts. This puts the trigger level at 6.2 volts as indicated by the top bold horizontal line. For a *positive slope* selection, the hysteresis level is always *9% of full scale less* than the threshold. This would be at approximately 5.3 volts as shown by the broken line.



With a positive slope selection, for a trigger event to occur, the signal must pass from below the hysteresis level up to or beyond the trigger threshold. Consider these three examples:

1. Although the top (orange) pulse passes through the threshold with a positive slope, it was not below the hysteresis level, therefore no trigger will be generated.
2. The middle (green) pulse passes from below the hysteresis level through the threshold, and will generate a trigger event.

3. The bottom (blue) curve passes through the hysteresis level, but fails to equal or exceed the trigger threshold, therefore, no trigger event.

Note that the signal must return to below the hysteresis level if a subsequent trigger event is to ever take place.

For a *negative slope* selection, the hysteresis level is *above* the trigger threshold by 9%. The signal must come down through the hysteresis level and then pass below (or equal) the trigger threshold for a trigger event to be generated. Again, note that until the signal passes back up through the hysteresis level, another trigger cannot occur.

Output Channel as Trigger Source
An important feature of the SigLab output source is that it generates a digital trigger signal for use by the input subsystem as the trigger source. This eliminates the *slope, threshold, and filter* controls required to set up triggering on an analog signal. Whenever you use ***vfg*** to generate the DUT excitation, *Output Channel* is, without question, the trigger source to choose. A trigger pulse is generated for each repetition of the output waveform. Triggering is therefore reliable and independent of all input channel and acquisition parameters.

External Trigger
Pin 2 of the 15-pin Digital I/O connector on the SigLab back panel is labeled as *–mb_trigger*. This pin is actually both input and an output. When *external* triggering is selected, Pin 2 is an input. Note that in a multi-box configuration the master box (the one containing channel 1) should be the only box with a connection to *–mb_trigger*. A trigger event is generated whenever this line goes from a TTL “high” level to a TTL “low” level. Pin 15 of this connector is system ground. The pin-out drawing of the connector is shown on the back panel label. (See Section 3.0)

When *external* triggering is not selected, the *–mb_trigger* pin is a trigger output and therefore *should not be electrically driven or loaded* by any external circuitry.

Filter Button

The Filter button has two states:

1. *HfRej* : the aa-filters are applied to the data before the trigger detector
2. *NoFilt* : the maximum hardware analysis bandwidth will be used for the trigger source

The *NoFilt* selection uses the measurement data stream at the maximum analysis bandwidth as the trigger source. This is the most common choice since the small duration transients that are often the trigger events of interest, will not be filtered out by a low bandwidth setting. At reduced bandwidth settings, the signal is more delayed from the trigger event because the trigger does not pass through the anti-alias filtering chain. This is usually of no consequence since you can usually adjust the pre/post trigger slider to compensate for this. The green LED is shown for the *NoFilt* selection since this is the foolproof choice.

The *HfRej* selection uses the filtered and decimated data stream as the trigger source. For baseband analysis, this means the trigger signal has gone through all the anti-alias filtering. If zoom has been selected, the trigger circuit only looks at the real part of the trigger signal. The correlation between the t=0 point on the time history graph and the trigger point on the waveform will be the best when the filtered mode is selected. However, frequently this is not an important consideration and the inability of the trigger to detect small transients (like pulses from a modal impact) often results in inadequate triggering. The yellow LED next to the button is provided as an operator warning.

Trigger Delay

The *trigger delay* slider allows one to observe up to one record length before the trigger event (negative values) to one record length after the trigger event. The control is calibrated in percent of record length. Typically a value of –5% to –15% is used to capture transients such as modal force impacts.

Processing Controls

You use the Processing Control to set the *averaging mode* and *averaging parameters*, select the *analysis window*, and specify *record overlap* and *zero-padding*, which is used for correlation measurements.

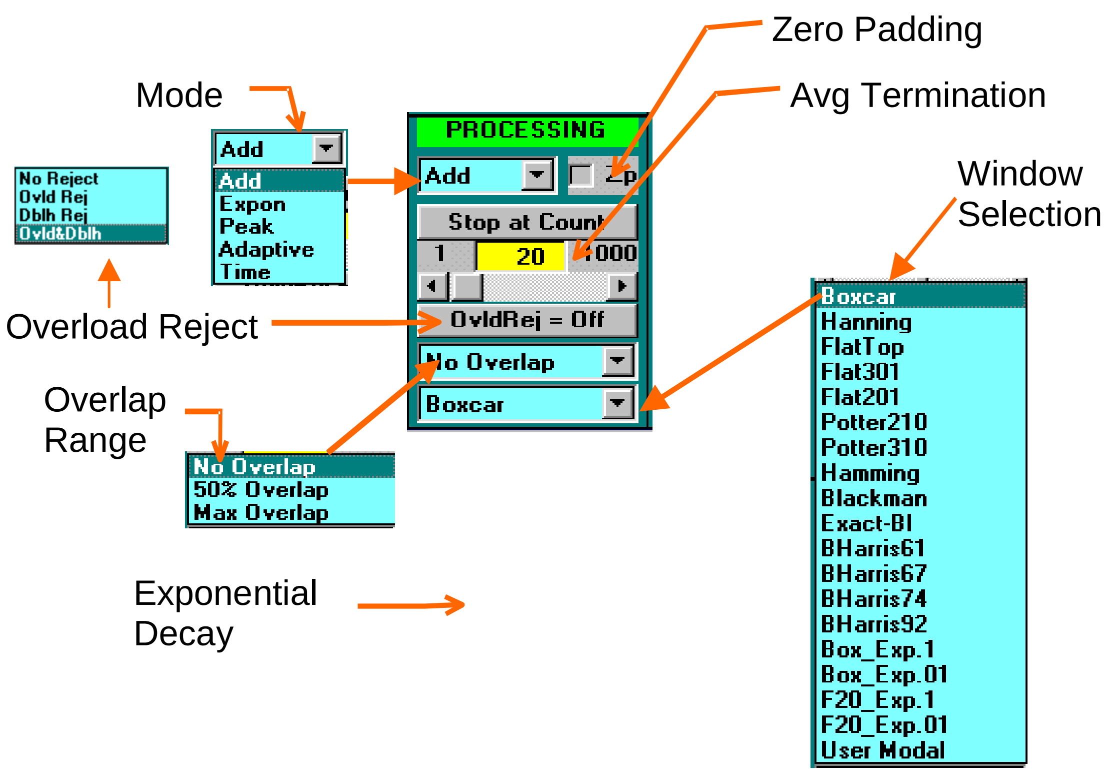


Figure 5-8 Processing Control

Averaging Modes

The ***vna*** application supports five averaging modes. The first four average frequency domain data. This frequency domain averaging affects the auto and cross spectrum calculations. This impacts all measurements except for time histories. The last averaging mode averages successive time histories additively before transformation to the frequency domain.

The *Add(itive)* mode weights all measurement data equally. Averaging is automatically terminated when the desired average count is met. The average Y_n is actually calculated recursively as shown below. It is equivalent to a simple summation of the inputs X_n to the

$$Y_n = (X_n + (n-1) \cdot Y_{(n-1)}) / n = \frac{1}{n}\sum_{i=1}^{n} X_i$$

averaging process where the *n* represents the nth input to averaging process.

The *Expon(ential)* mode weights old measurement data less than new measurement data. It is a useful averaging mode when the statistics of the signal are slowly varying. The calculation is:

$$Y_n = ((1-\lambda) \cdot X_n + \lambda \cdot Y_{(n-1)})$$

where lambda is the averaging parameter. When lambda is zero, there is no averaging. As it approaches one, old data is weighted heavily and not easily “forgotten.” The time constant in frames (or records) is given by:

$$\mathrm{T}_{frames} = 1 / (1-\lambda)$$

This represents the number of frames it would take for the averaged signal to decay to 1/e (about 1/3) of its initial value after the signal is removed.

The *Peak (Hold)* mode captures the maximum value and preserves it. It is normally used only with spectrum analysis.

$$Y_n = max(X_n, Y_{n-1})$$

The *Adaptive* mode is a mixture of exponential and additive averaging. When the lambda factor in exponential averaging is close to one, the startup convergence to the mean is very slow because each new sample is multiplied by such a small number (one minus lambda).

Adaptive averaging addresses this by having good startup properties, like additive, while maintaining the ability to track slowly varying signal statistics like exponential.

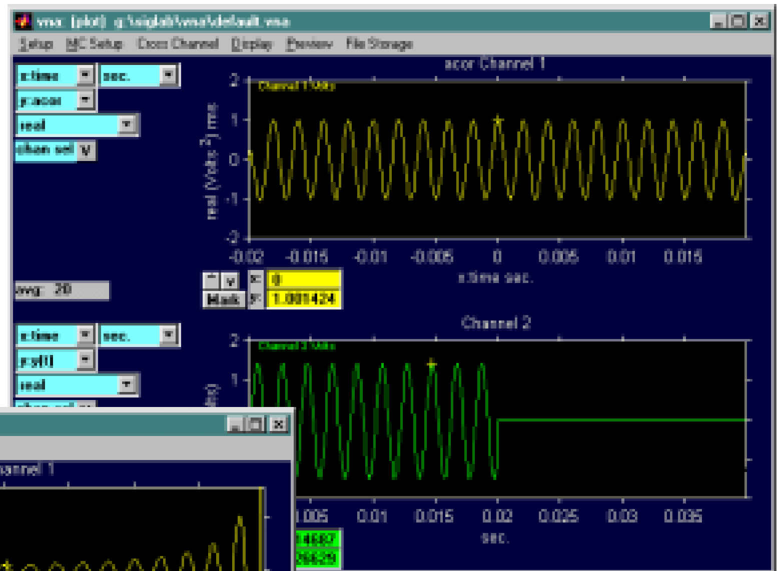
$$Y_n = R_n \cdot X_n + \lambda \cdot \left(\frac{R_n}{R_{n-1}} - 1\right) \cdot Y_{n-1}$$

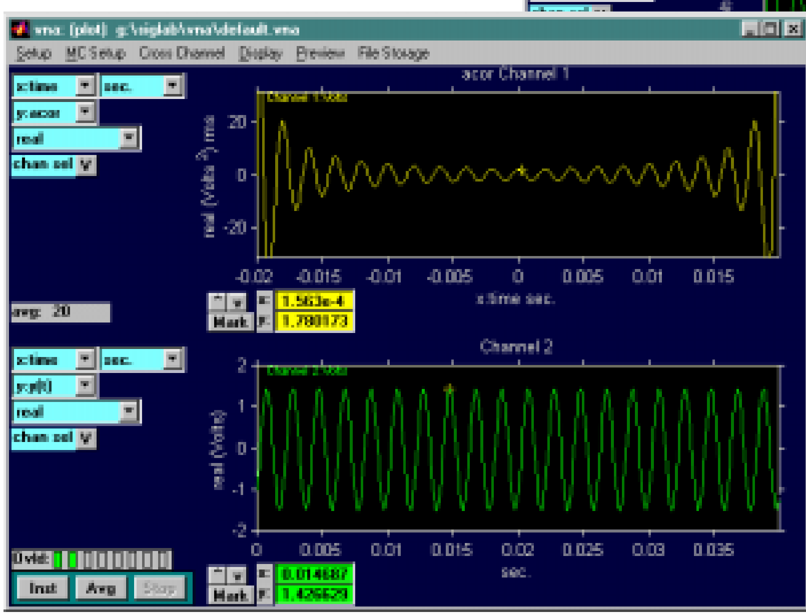
$$R_n = \frac{1-\lambda}{1-\lambda^{n-1}}$$

Time domain averaging uses the same averaging algorithm as additive, but performed on the time histories. A triggering mode is normally used with time averaging. Note that transfer function coherence will be unity when time averaging is used.

Zero Padding

Zero padding is only required for accurate measurement of correlation. For example, the auto-correlation of a sine wave of 1.0 Vrms is a cosine function with 1 V^2 rms peaks. This is shown on the right in the upper axis. A rectangular window was used. Note the affect of *zero padding* is to wipe out one half of the time history with zeros.





The graph to the left shows the same auto-correlation with the exception that *zero padding* is not on. Note the severe error in the auto correlation function where it exceeds 30 V^2 rms. Thus, *zero padding* is required for correlation measurements. To avoid errors in all other measurements, it must be disabled.

Overload and Double Hit Reject

The *Overload and Double Hit Reject* popup turns off and on any combination of the two forms of Automatic Rejection available. Overload Rejection causes any data that is corrupted by input acquisition overloads (signal level exceeds full-scale range) to be excluded from the averaged measurement data automatically. Double Hit Rejection will be explained further below under the Plot window Modal Menu item. These features are most often used when performing modal tests with impact hammers. They should be enabled only when required. Automatic rejection of data can be confusing, particularly Overload as frames will be rejected whenever an overload is detected on any channel whether or not that channel is enabled.

Overlapped Processing

The *overlap processing* selection menu provides three choices: *none*, *50%*, and *maximum* possible. At low analysis bandwidths, display updates can become very slow. Overlapped processing addresses this by combining measurement data from previous frames with new measurement data (time domain) to create a full frame. The 50% selection will use up to 50% of the previous time history in the new frame, but no more. This can improve the time required for a given level of measurement variance by almost 2:1. Overlap factors beyond 50% do not make a further improvement in the averaged results, but they do provide more rapid display updates. Overlapped processing does not provide any benefit when triggering is on or when bandwidths exceed approximately 1000 Hz.

Analysis Window Selection

SigLab provides a good selection of FFT analysis windows. These are useful in spectrum analysis as well as for transfer function estimation. When you set the Excitation mode to *Independent*, you have control over the analysis windows. When you set the Excitation mode to *Linked*, the window selection is made automatically: Boxcar for a chirp excitation, Hanning for random. Although an exhaustive treatment of which window to select is beyond the scope of this manual, a few common situations will be covered.

The window shape can be viewed by selecting a time history for display in the upper axis and then clicking on Display - Analysis Windows as shown on the right. The window (Hanning in this case) is plotted using the red markers. Changing the window selection will not update the plot of the window until a new measurement is made.

A detailed tutorial on windows can be found online at: www.dspt.com/sig/frsttov.html

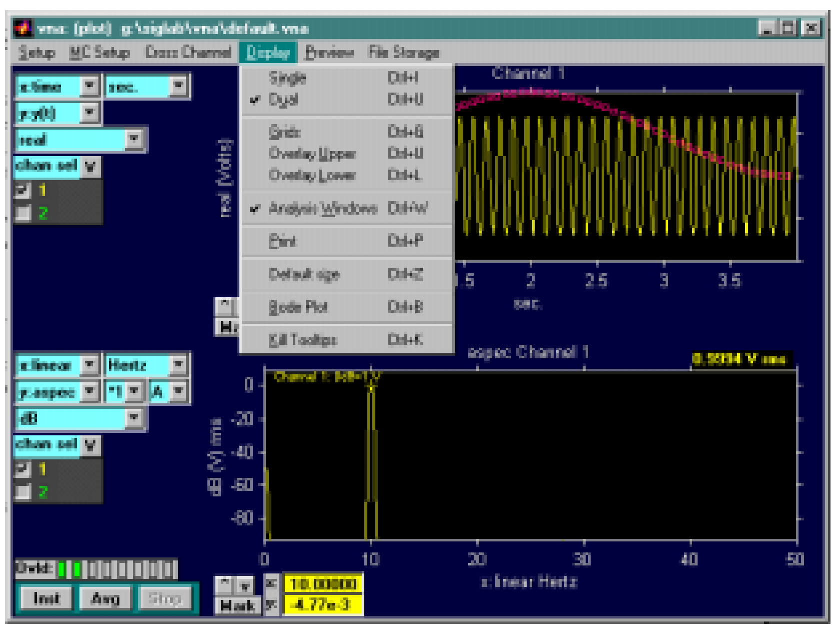


Figure 5-9 FFT analysis window display

The three first windows in the list (Boxcar, Hanning and Flattop) will satisfy the bulk of a typical user's needs. The last four windows in the list are intended for modal analysis.

Boxcar
The "boxcar" window is actually no window at all and is sometimes called the "rectangular" window. It should be used when:

1. Analyzing transients that are entirely within the time record.
2. Signals components are exactly on spectral lines, leakage is not a problem
3. Any correlation measurements are being done.

Hanning
Hanning is probably the most general purpose of any of the analysis windows. It has good leakage suppression and does not cost too much in the way of lost frequency resolution. For analyzing signals with either random or periodic components, it is hard to beat. Use Hanning when:

1. Periodic signal components not on spectral lines are present
2. Leakage is a problem
3. Random or band-limited random signals are analyzed.

When more frequency resolution is required you should use either Zoom processing, a longer time record, or both.

Flattop
Windows, or lack thereof, will introduce an amplitude error if a sinusoid is not on a spectral line. The Flattop window is expressly designed to minimize this amplitude error. Use the FlatTop window when:

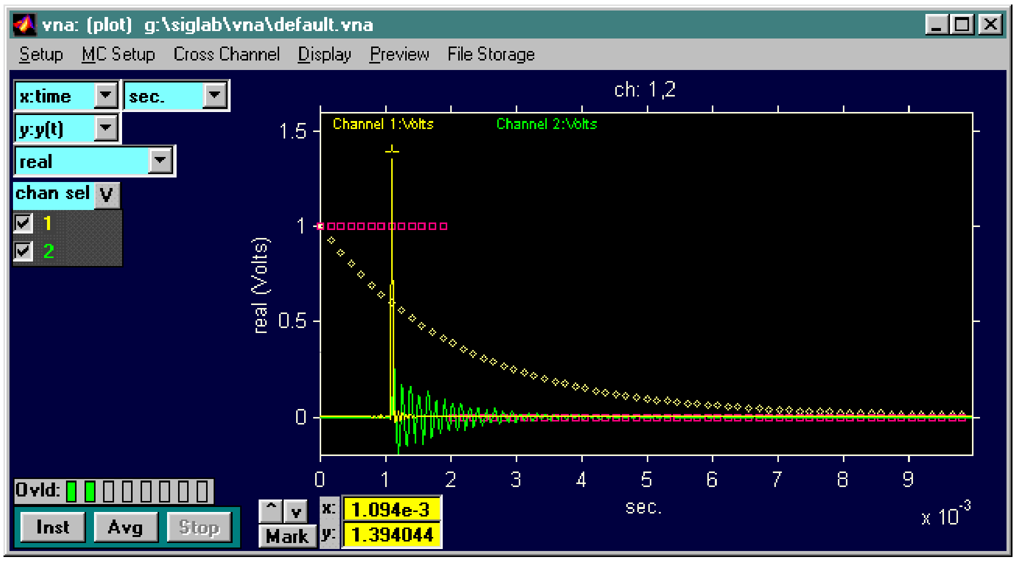
1. Maximum sinusoid amplitude accuracy is required
2. Frequency resolution is not an issue

Modal Windows
The Modal windows are:

1. Boxcar, Exponential 0.1
2. Boxcar, Exponential 0.01
3. Force 20%, Exponential 0.1
4. Force 20%, Exponential 0.01
5. User Modal

The first entry (Boxcar or Force) indicates the window that will be used on channel one, which, is the reference channel. The Force 20% has a weight of unity for the first 20% of the measurement record, and zero thereafter. It is important to setup the pre-trigger value such that the force excitation lies within the force window. The exponential window is applied to all the response channels (2 and up). The 0.1 indicates that the exponential has decayed to 0.1 at the end of the frame while the 0.01 has decayed to 0.01 at the end of the frame.
The User Modal again uses a Force window on channel 1 and an Exponential Window on all response channels, but here the user sets the force percentage and exponential decay factor, as described under the Plot window Modal menu item below.

The plot below shows:
1. Force 20% window (red rectangle markers)
2. Impulse excitation (yellow trace with cursor cross on peak)
3. Exponential 0.01 response window (yellow diamond markers)
4. DUT response (green trace)

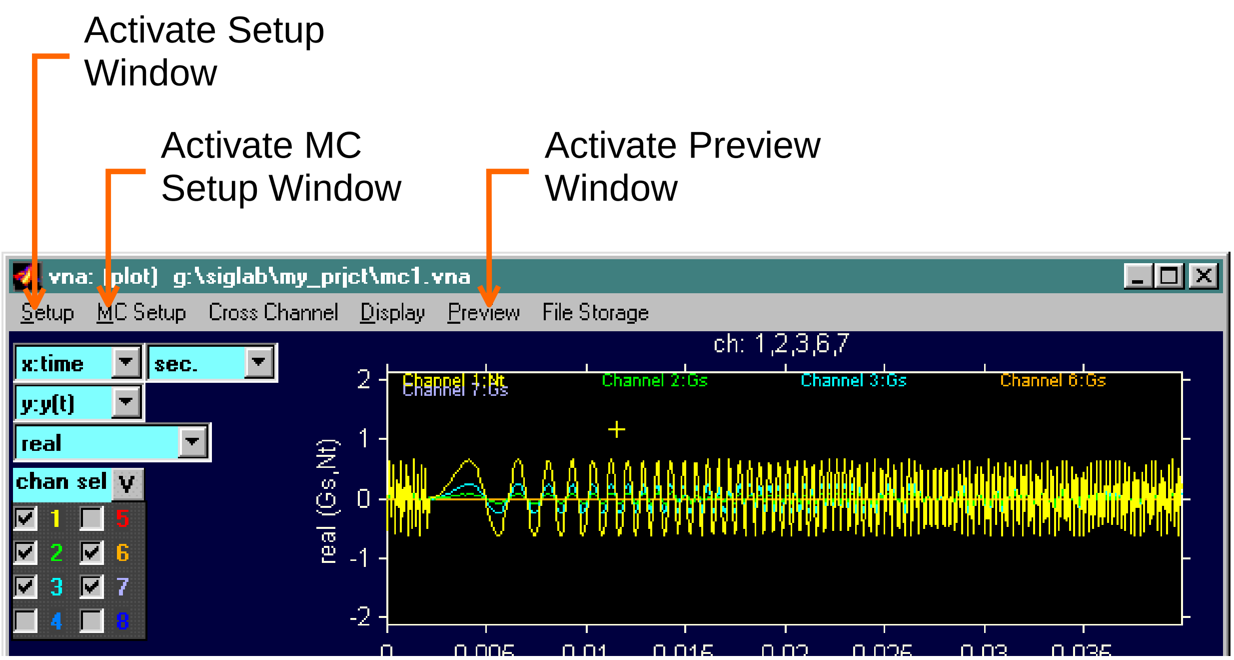


Plot Window Details

The plot window controls the measurement function selection, formatting, and display. It may be resized and positioned anywhere on the desktop. The size and position are saved to the measurement file. Normally the plot window is not minimized, but if it is, you can restore it by selecting it from the operating system task bar. You identify it by the word "plot" as shown below.

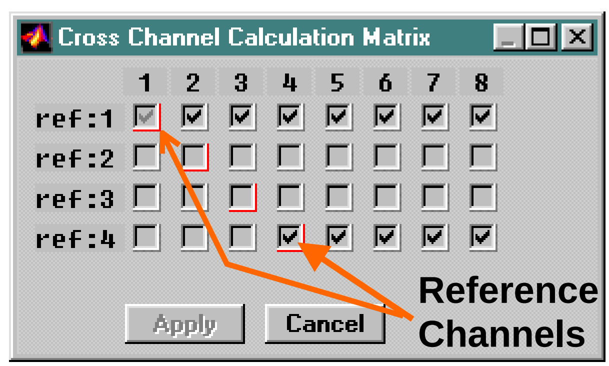
Plot Window Menu Bar

Use the *Setup* or *MC Setup* selection to open or transfer focus to these windows, which were described in the previous section. The *Cross Channel*, *Preview* and *File Storage* selections open or transfer focus to windows that are described below. The Display selection opens a dropdown menu that allows you to control the size and appearance of the plot window itself.



Cross Channel Menu

SigLab's multi-channel measurements support up to four reference channels for cross channel functions such as transfer function and cross spectrum. Clicking on the *Cross Channel* menu opens the *Cross Channel Calculation* dialog shown to the right. The size and shape of this dialog is determined by the number of input channels present in any given hardware configuration. The number of rows is equal to the number of channels in one SigLab box (2 for 50-21 & 20-22, 4 for 20-42). The number of columns is equal to the number of channels up to a maximum of 16.



You use the *Cross Channel Calculation* dialog to accomplishes three tasks:

1. Enabling of specific channels. The Cross Channel Calculation dialog is normally only used only when multi-reference measurements are being made. Typically, you will use the *Setup* window or the *MC Setup* window for enabling and disabling channels.

2. Selection of Channels 1 to 4 as reference channels for cross channel measurements. The red borders around the check boxes on the main diagonal indicate that checking these defines the channel corresponding to that row as the reference channel on that row.

3. Selection of which channels are response channels for each reference channel.

The *Cross Channel Calculation* dialog box operates as follows:

- Channel 1 is the primary (and most often used) reference channel. It is always enabled and cannot be disabled.

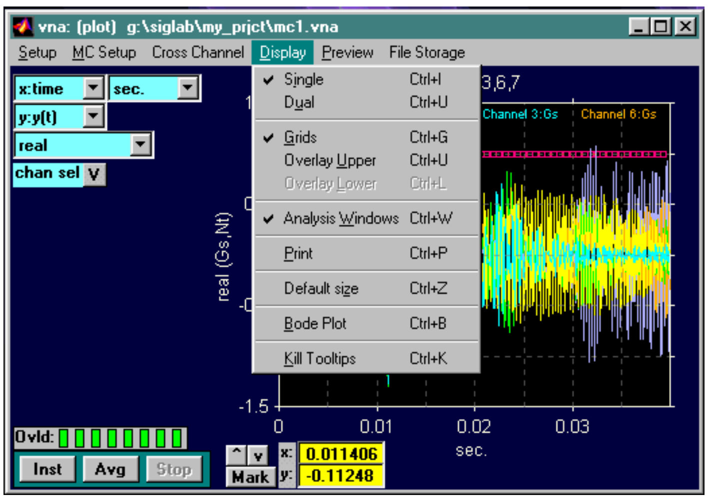
- There can be up to four reference channels. For multi-channel systems, the reference channels must be one or all of the channels in the first SigLab module. In the above example, channels 1 and 4 are designated as reference channels while 2 and 3 are not.

- When a check box is checked, the channel corresponding to that column is enabled. The above example shows channels 1-8 as enabled. Cross functions between responses 2 through 7 and reference channel 1 will be computed. Cross functions between responses 5 through 8 and reference channel 4 will be computed.

- Until the Apply button is pressed, no changes will be made. Since this is a modal dialog box, either the *Apply* or the *Cancel* button must be pressed to continue.

Display Menu

The *Display Menu* is shown below. You use this menu to control the size and appearance of the Plot Window as well as to select some other options.



Single/Dual

The *Single/Dual* submenu specifies whether you want one or two axes to be visible in the plot window. If *Single* is selected, the lower axis is hidden and the upper axis is expanded. If Dual is selected, both the upper and lower axes are visible.

Grids

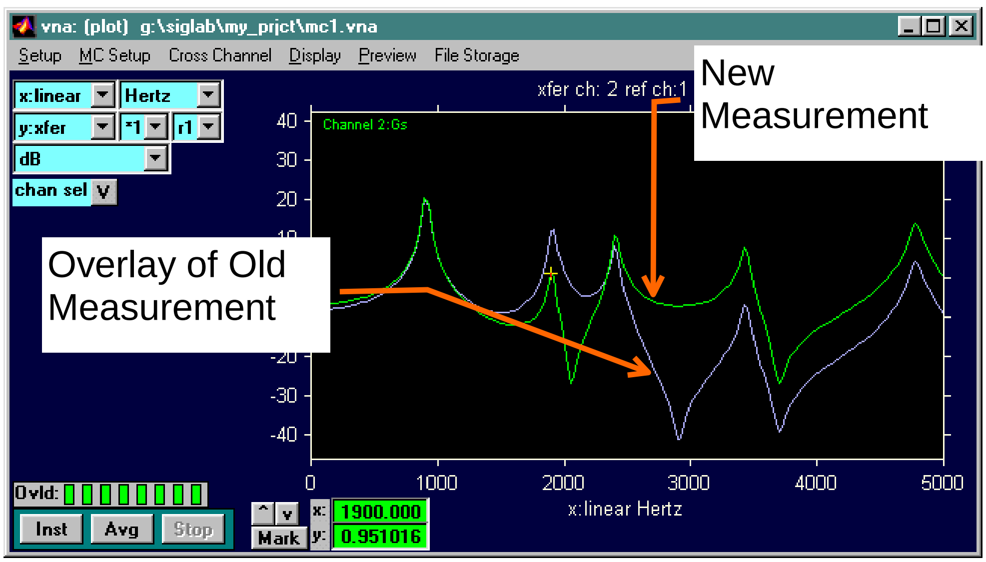
The grids menu pick turns axis gridlines on or off. The color and intensity of the gridlines can be set using the *vicolor* (Section 4.0) application.

Overlay Upper and Overlay Lower

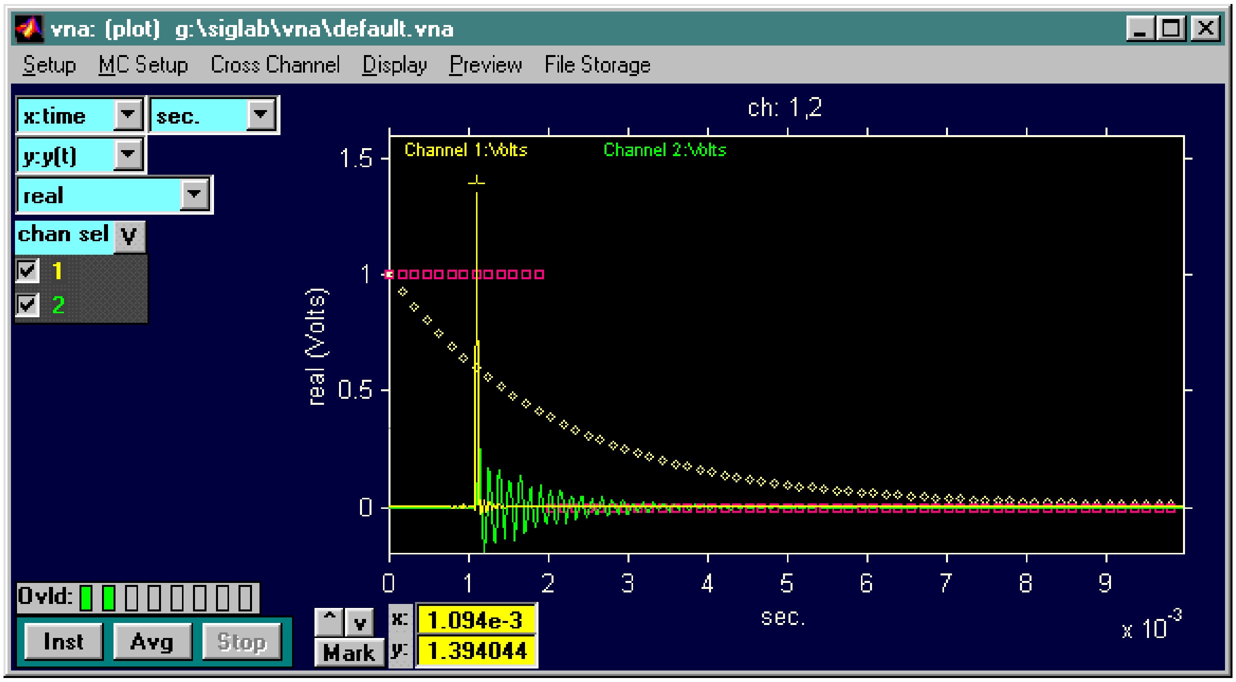
The Overlay menu picks are analogous to software "grease pencils." This feature allows comparison of measurement results. When more than one measurement result is plotted on an axis, the measurement to be overlaid must be selected by using the mouse to select the respective measurement plot line with the cursor before doing the *Overlay* menu pick. For example, measurement data stored in files can be compared to a new measurement by opening the file, selecting the desired trace, selecting Overlay Upper or Overlay Lower from the Display menu, and then performing the new measurement. Reselecting the *Overlay* menu toggles the overlay line visibility.

Analysis Windows

Use this selection to view a plot the selected *Analysis Window* in the upper axis. Simply click on the *Analysis Windows* selection while there is a time history displaying in the upper axis. The plot of the shape of the analysis windows being used will overlay the time histories.

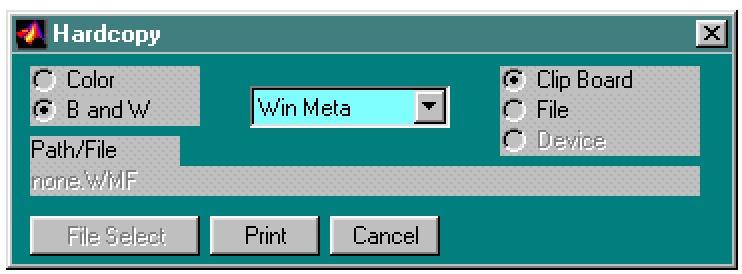


The plot below shows:
1. Force 20% window (red rectangle markers)
2. Impulse excitation (yellow trace with cursor cross on peak)
3. Exponential 0.01 response window (yellow diamond markers)
4. DUT response (green trace)



Print

You select the *Print* menu to open the Hardcopy dialog shown right. You use this dialog box to control the printing of the Plot Window. For information on this dialog, see Section 4.0.

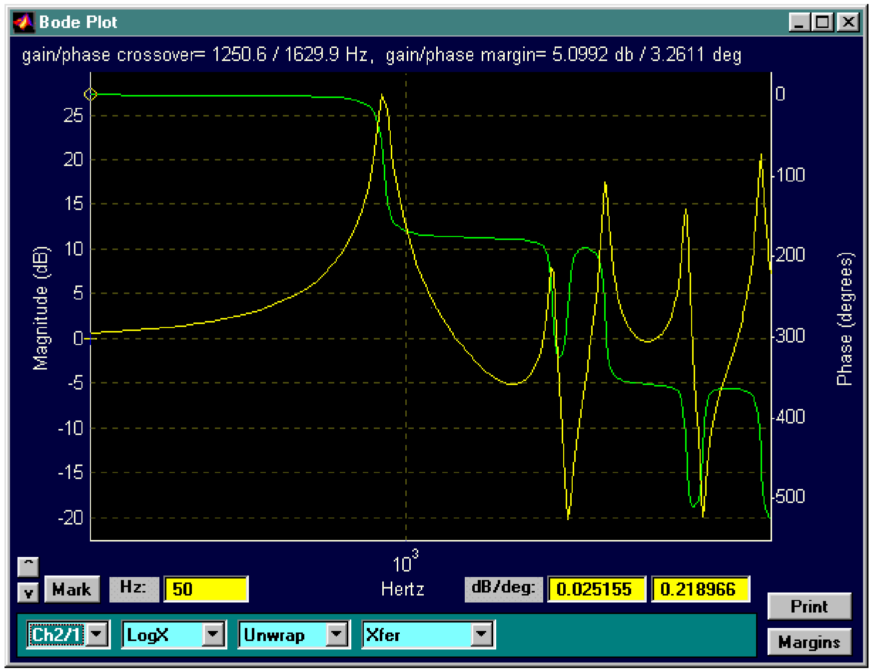


Default Size

You use this menu pick to restore the Plot Window to the default size. This can become necessary because the Plot Window is resizable.

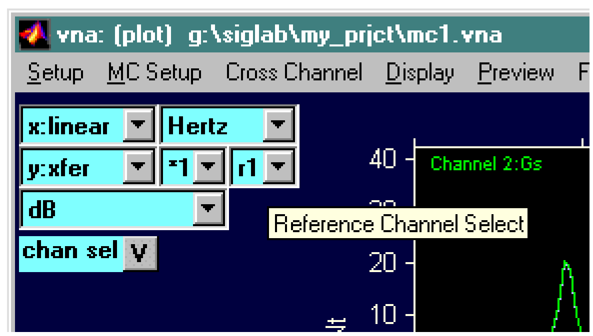
Bode Plot

This menu pick invokes the Bode plot utility shown right. For more information, see the manual section on the Bode utility.



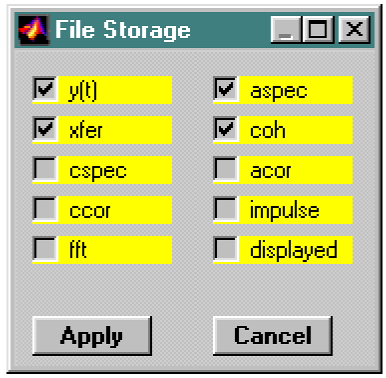
Kill Tooltips

Tooltips are used to aid the first time user in navigating the GUI. For example, when you place the mouse cursor near the popup labeled r1, the light box appears with the text “Reference Channel Select.” If you wish to turn off this behavior, click on the *Kill Tooltips* menu. They will, however, return the next time ***vna*** is invoked.



File Storage Menu

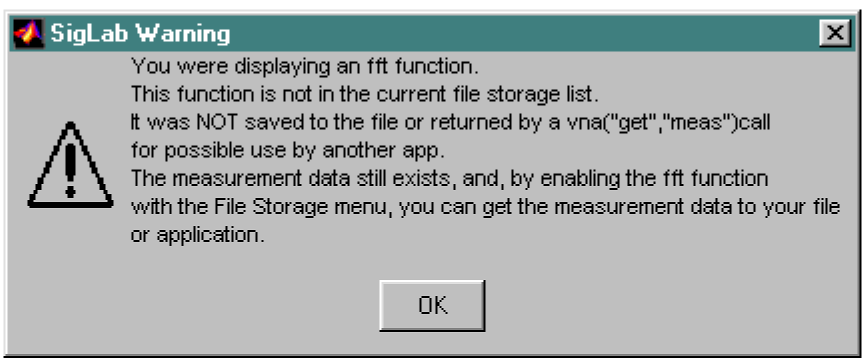
The ***vna*** application can measure and compute up to nine different measurement functions. In any one measurement, it is unlikely that you will need all nine functions. By clicking on the *File Storage* menu, you will open the File Storage modal dialog shown to the right. You may then select which of the nine possible functions are to be stored to the measurement file.



If you check the10th selection on the lower right ("displayed"), ***vna*** will store any functions that are currently being displayed on any axis even if they have not been explicitly chosen for storage.

You must then click on *Apply* to mark the data for storage. The data will not actually be saved until you perform a *File/Save* operation.

When you perform a *File/Save* operation with measurements being displayed that have not been marked on the File Storage list and the "displayed" check box is not checked the following warning will be issued:

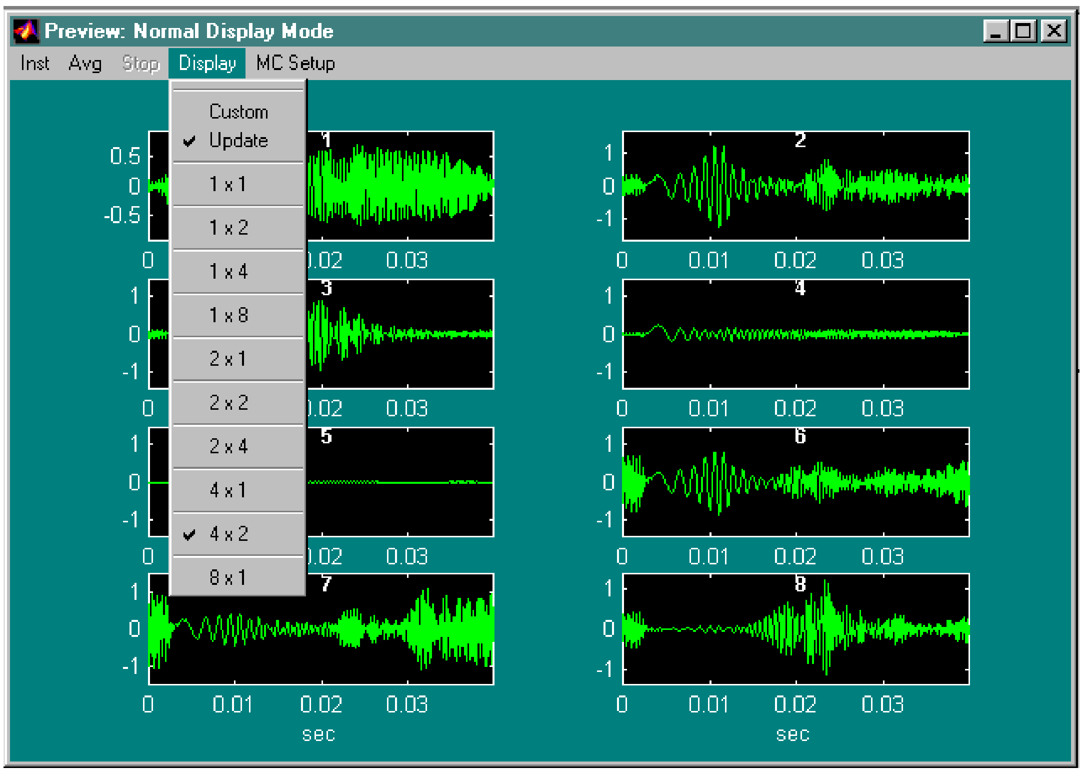


Preview Menu

You click on the *Preview* menu to open the *Multi-Channel Preview Window* as shown below. This window gives you a simultaneous real-time view of up to 16 time histories, each on a separate axis. This view is intended to provide a quick look at the signal integrity of each channel. The y range of each axis is set to the corresponding full-scale setting of each channel. Thus, you have a quick comparison of the signal level from each channel with respect to its full-scale value. This window is resizable.

Inst, Avg and Stop menus

The *Inst* , *Avg* and *Stop* menus work like the corresponding buttons in the ***vna*** Plot Window.



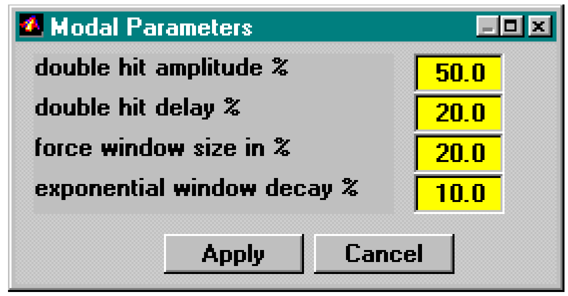
Display Menu

The ***Display Menu*** allows the user to select from a variety of axis organizations. The graphic above shows the 4x2 format.

MC Setup Menu

Clicking on the *MC Setup* menu transfers focus to the MC Setup window.

Modal Menu



The Modal menu calls up the dialog box depicted at right. This dialog allows you to set up the parameters that define Double Hit Rejection and the "User Modal" window option from the Processing Control Panel.

Double Hit rejection is a form of Automatic rejection used only in impact hammer tests. This feature is for the hammer tester who wants to be certain he or she does not corrupt their data with an unintended second tap of the hammer on the structure mid-frame. When this feature is enabled (from the Overload and Double Hit Rejection popup on the Processing Control Panel), data frames will be discarded and not included in the average when a "double hit" is detected on the Reference Channel (i.e., channel 1), based on the values in the first two fields on the Modal Parameters Dialog. A double hit is defined as a second peak in the channel 1 data, which has an amplitude equal to or greater than the amplitude of the first peak times the **double hit amplitude %**. Furthermore, this second peak must occur after the first peak by at least the **double hit delay %** times the length of the frame. For example, suppose we have a frame that is 100msec in length, with a first peak of 2 Volts occurring 5msec into the frame and the Modal parameters is set as above. in order to qualify as a double hit, the second peak must have an amplitude greater than or equal to 1 Volt and must occur at least 25msec into the frame.

The third and fourth fields in the Modal Parameters Dialog determine the values used to define the custom windows used with the User Modal selection from the Window popup on the Processing Control Panel. When the User Modal window is selected, a force window will be used on channel 1 which has a cutoff point equal to the **force window size in %**. For example, if this value is set to 30%, then the first 30% of the frame on channel 1 will be treated as if it had a boxcar window on it, while the last 70% will be fully attenuated down to 0. If a boxcar window is desired for the reference, set this value to 100%. Furthermore, when the window selection is User Modal, an exponential window will be applied to all the response channels, such that the window starts at t=0 at 100% and then exponentially decays down to the **exponential window decay %** at the end of the frame. Similarly, here, if a boxcar is desired on the response channels, set this value to 100%.

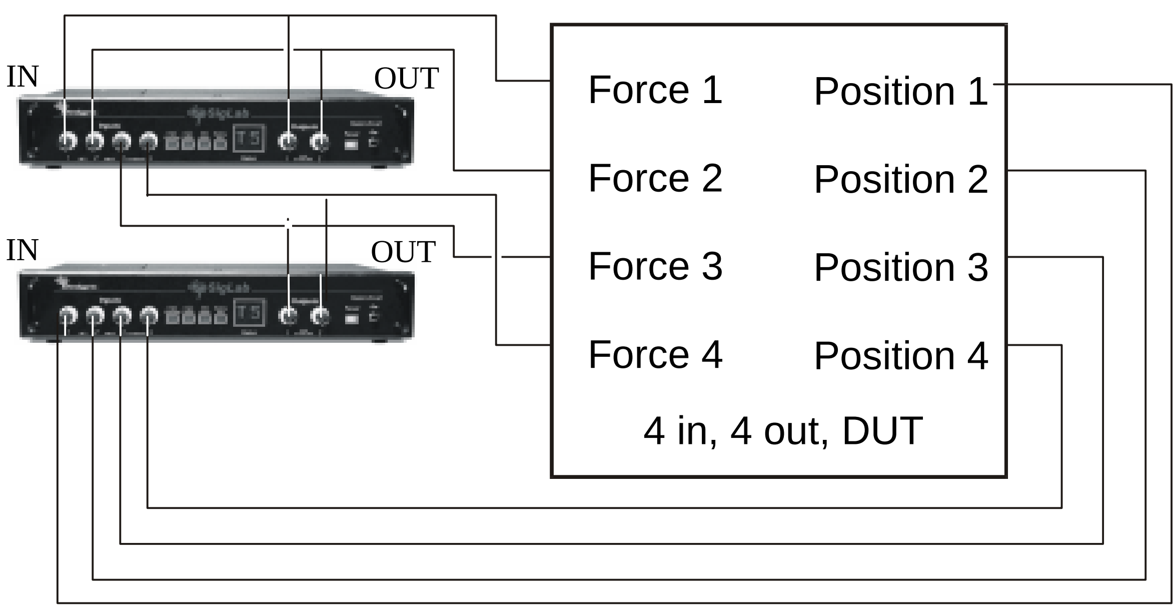
MIMO Menu

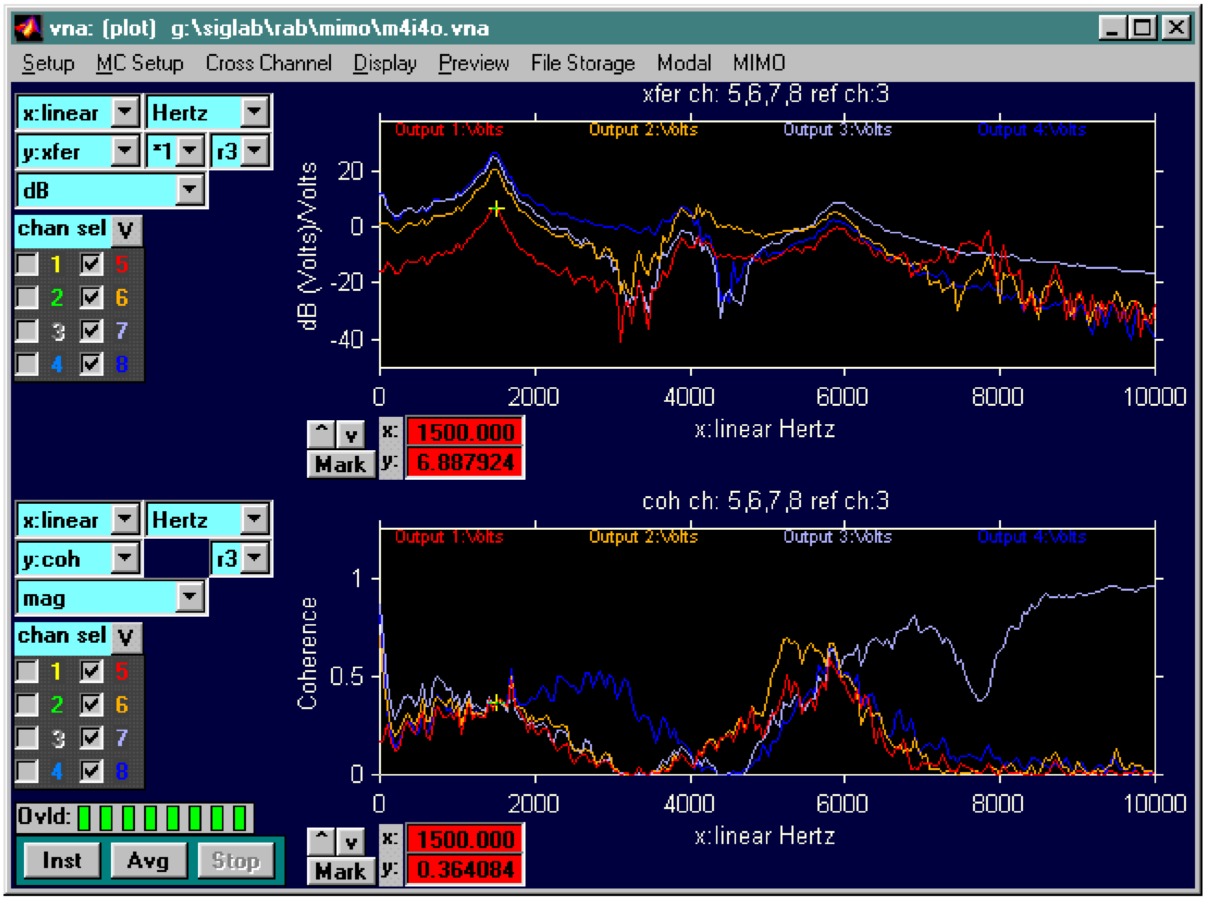
MIMO is the commonly accepted acronym for multi-input, multi-output systems. The problem with doing transfer function analysis on a multiple input system, however, is how does one determine which part of a given response is the result of input 1 and which part is the result of input 2, and so forth. In the general case, this is extremely difficult, if not impossible. However, there exists a class of MIMO system, where the multiple inputs are being excited by uncorrelated sources, whereby it is possible to "unscrew" the different components from the respective inputs. This situation does not often arise in pure electrical work, but does occur in structural applications and some control systems applications.

SigLab v3.1 added *multi-reference* cross function calculations, which could be post-processed by the savvy user to break out the contributions from multiple inputs, but we did not provide these calculations for the user. Beginning with SigLab v3.2, these calculations are now available to the user as a post-processing application (these calculations are far too computationally expensive to be performed as part of a real-time data acquisition loop). This capability is best demonstrated via an example of its use:

Consider a 4 mass, spring, damper mechanical “slinky” as a device under test (DUT). Assume that the slinky is instrumented such that a force input can be applied to each mass, and the position of each mass can be measured. This is a 4 input, 4 output system. There are 16 different transfer functions relating the outputs to the inputs. One can excite each input (one at a time) and measure the transfer function between the 4 outputs and each input. This requires the 4 channel transfer function measurement to be repeated 4 times … once per input. This method requires 5 analyzer input channels and one output channel.

An alternative approach is to excite all four inputs *simultaneously* . This requires 4 uncorrelated sources. The 4 responses from the DUT are measured along with the 4 excitations, therefore consuming 8 analyzer input channels and 4 analyzer output channels. The configuration is shown below. The upper SigLab provides excitation to Force 1&2 inputs while measuring all four DUT inputs. The lower SigLab provides the excitation to Force 3&4 while measuring all 4 DUT position outputs.

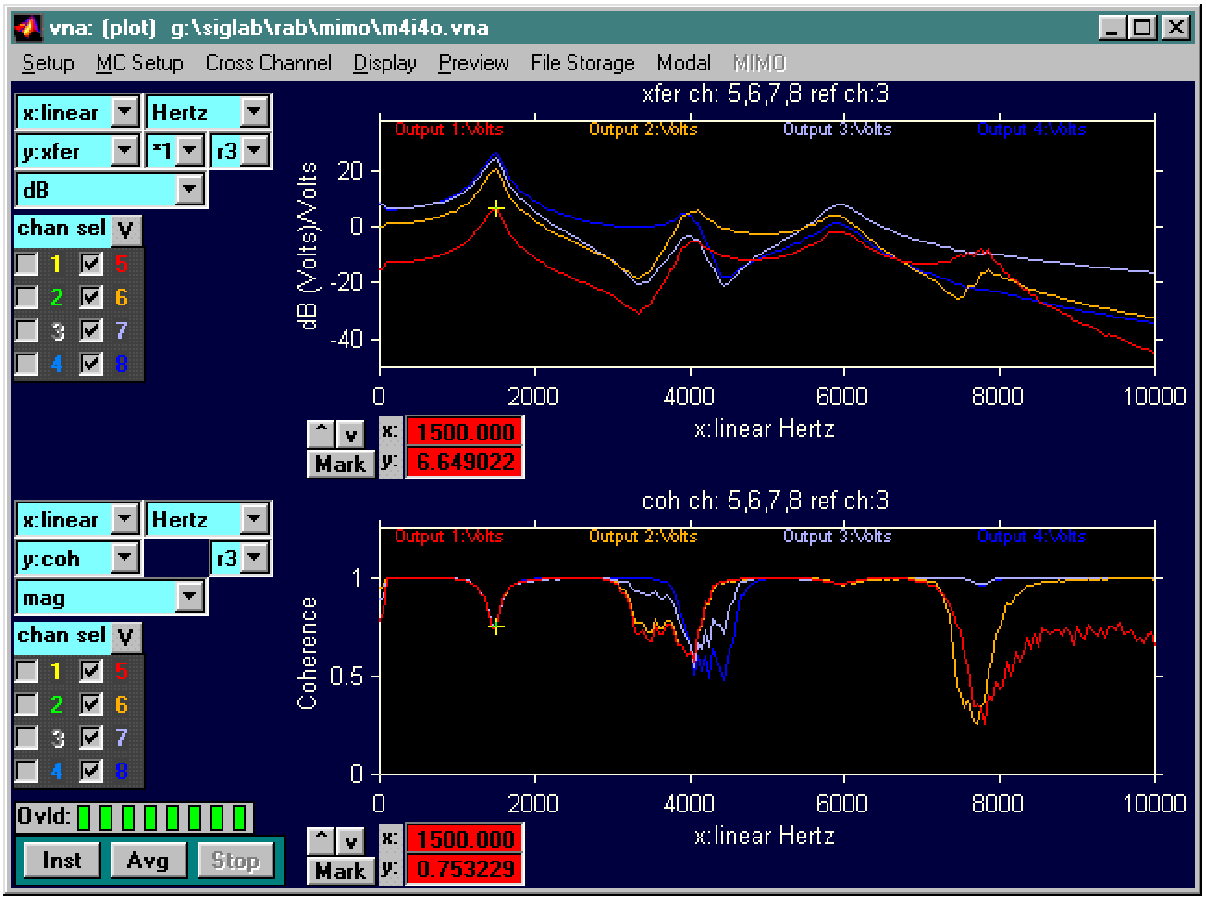




The graphic above shows 4 (out of the 16) transfer function and coherence measurements relating the 4 position outputs to input number 3. Notice that the transfer function estimates are not “clean” and the coherence is poor. This is a result of not accounting for the excitation signals present on other 3 inputs of the DUT. The other inputs (1,2 & 4) are adding significant (albeit colored) noise to the measurement. One hundred averages were used to compute this estimate.

However, if the measurement setup conditions are in a MIMO format (see below), the MIMO menu item (last Menu on the Menu bar in Figure 2) will appear when the averaging process is complete (this menu item is normally not visible). If the user clicks on this Menu item, the computation will commence and the following bar indicator will appear to provide the user feedback on the progress of the computation:

If the excitations and analysis parameters have been set up correctly (random noise matching the analysis frequency range, Hanning window) the user’s patience will be rewarded with the greatly improved transfer function estimates shown on the next page.

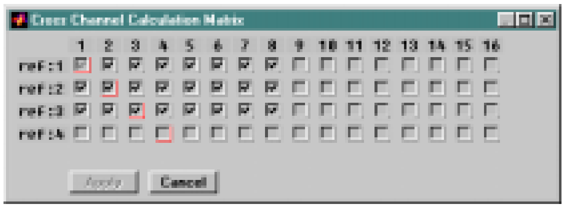


Setting Up a MIMO Measurement

First, you must connect the DUT to SigLab(s). Each input to the DUT should be connected to a SigLab output channel and to a SigLab input channel . These connections should be made so that the lowest channel numbers are used first. These DUT input channels are the *reference* channels of the measurement. Remember, only the channels in the first SigLab box can be used as reference channels. Next, each DUT output should be connected to a SigLab input channel. These *response* channels must start with the first free SigLab input channel.

Since multiple uncorrelated input excitations are required, the *independent* excitation mode must be selected from the excitation menu in vna. The function generator (vfg) can be used to provide the control over the excitation signals. A continuous random or band-translated random (if zoom is used) excitation should be generated for each DUT input (reference) channel. The settings of bandwidth and center frequency (if zoom is used) in vfg must match those used in vna if the MIMO calculations are to produce the desired result.

The settings in the vna cross channel computation matrix determine if a MIMO measurement is to be made. For instance, the graphic on the left below shows the cross channel computation setup for a 3 input, 5 output measurement. Note that rows ref:1 through ref:3 are checked indicating there are three inputs. The response channels 4…8 are checked (5 DUT outputs) and all cross functions are computed for the reference channels. In short, a full array of checks must exist for the measurement to be MIMO. If even 1 of the response/reference pairs is left unchecked, the MIMO menu will not appear.



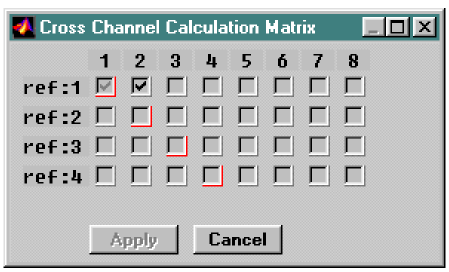

Measurement Examples

The remainder of the controls in the ***vna*** Plot Window is covered in the following typical measurement examples. These examples will use the Acoustic Tube provided with SigLab as the Device Under Test. You should follow along and duplicate these measurements to gain familiarity with operating SigLab. Because these Acoustic Tubes exhibit considerable variation from unit to unit, you should expect only coarse matches with the cross-channel measurement results. For the examples of measurements involving more than 2 channels, a multi-input multi-output DUT that is *not supplied* with SigLab is used.

To follow along with the examples:

1. Start MATLAB
2. Use ***vip*** to set the preferred path to siglab\vna\vna_demo
3. Start ***vna***
4. Connect the Acoustic Tube per the diagram in Section 4.0

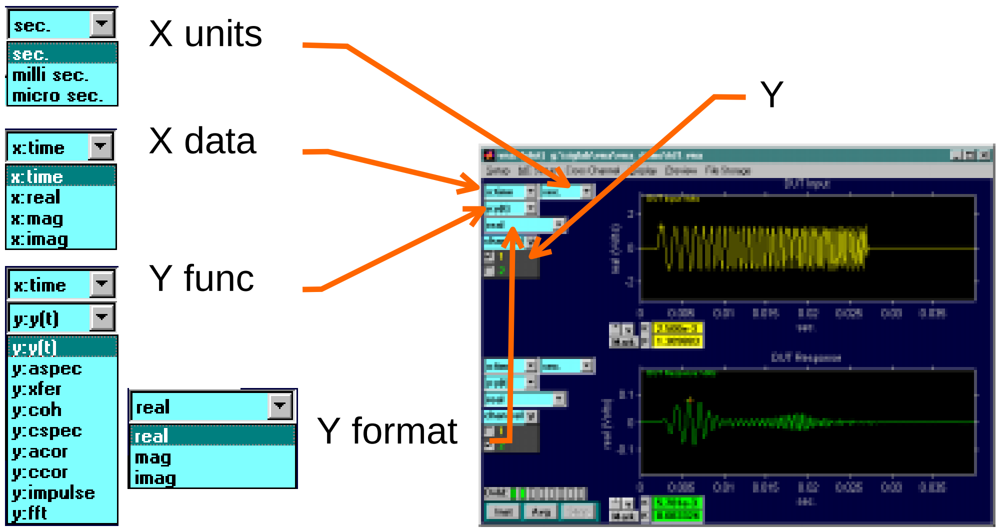
Unless otherwise noted, the Cross Channel Calculations will be set as follows:



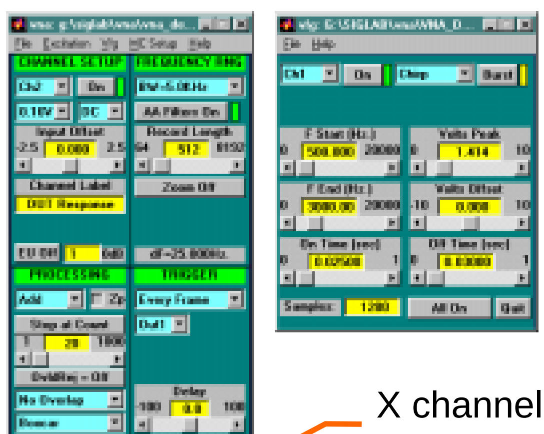
Note that the measurement setup files and screen captures were done on a host PC with a display resolution of 1024x768 pixels.

Time Domain

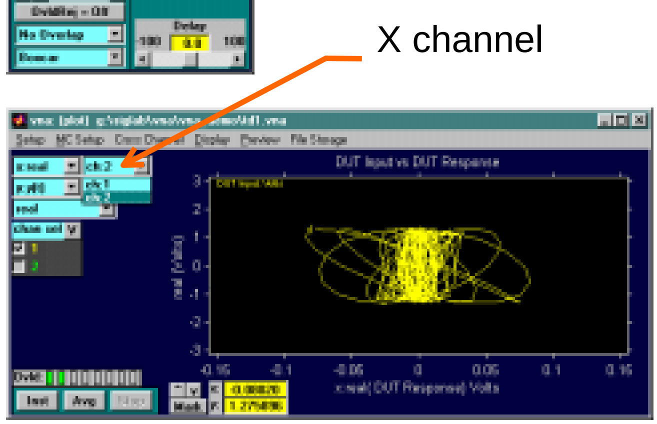
Time domain measurements are the simplest type of measurement that SigLab makes. The setup files *td1.vna* and *td1.vfg* were used to create a burst chirp excitation to the DUT (upper trace) and the response is shown on the lower trace. The acquisition and function generator settings are shown in the setup windows below. You set the ***vfg*** outputs to *All On* and push the *Inst* button in the plot window to make a new measurement. When you are done viewing the measurement, push the *Stop* button.



You use the *Y func* popup to determine what measurement function is to be displayed. In this case, you select *y:y(t)* to display the input time history. You normally set the *Y format* to the *real* part, since, there is no imaginary part unless zoom is being used. When you select the time history function, the *X data* indicates *time*. In this case, you can select seconds, milliseconds, or microseconds from the *X units menu*.



When you wish to see a Lissajous or Orbit plot format, choose the *X axis data* to be *real* (or *imag/mag* for zoomed data). When you do this, the *X axis units* popup is replaced with an *X axis channel* selector allowing you to select the channel whose time history is to become the x axis data. The adjacent plot window shows that channel 1 is the Y axis data and *ch 2* has replaced *time* as the X *data.*

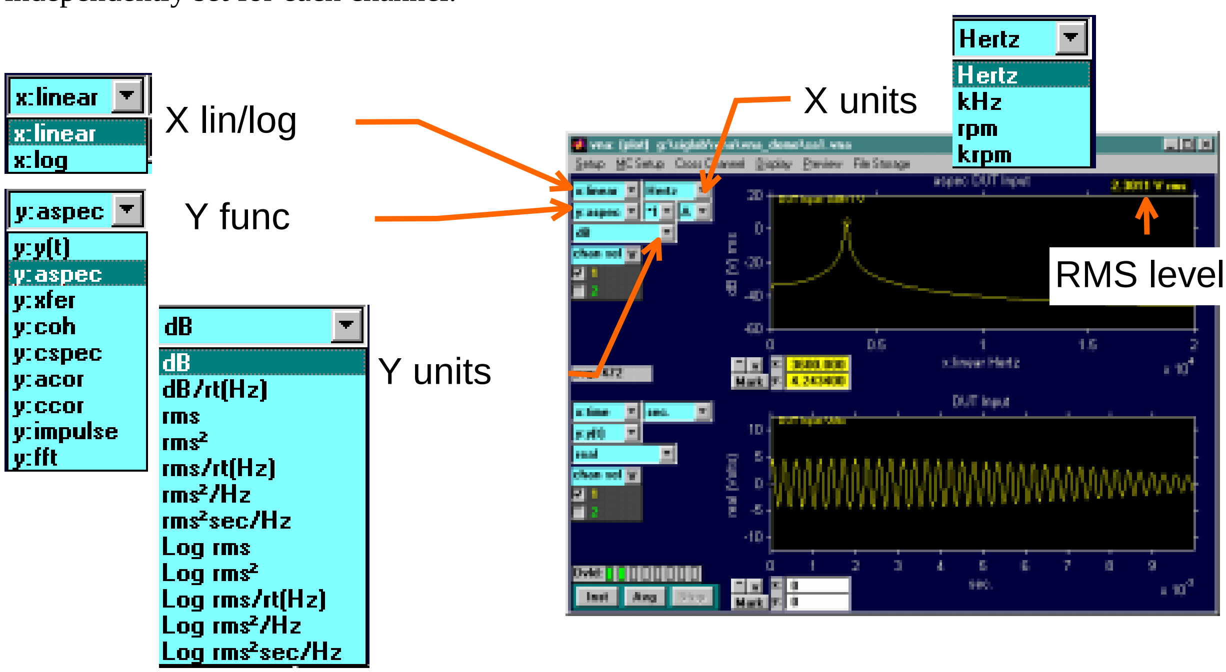


Time histories are displayed as un-averaged measurements independent of whether you push the *Inst* (Instantaneous) or *Avg* (Average) button to begin the measurement unless you have specified otherwise. This is because *freqency domain averaging* is is the default choice. Unless you explicitly choose *time domain averaging* (link to processing/averageing) un-averaged time histories will be displayed.

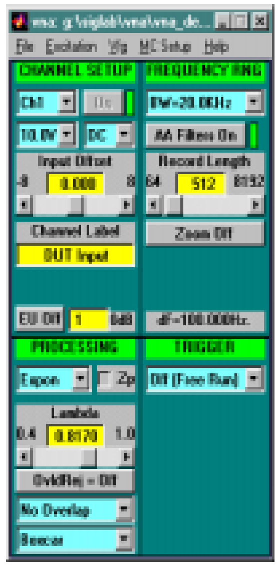
Spectrum Analysis

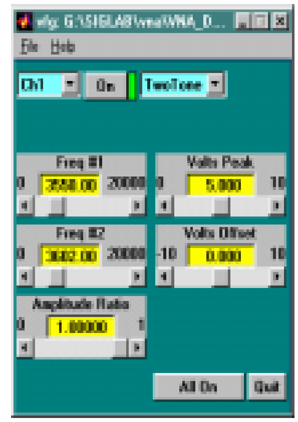
Spectrum analysis is a primary application of the Dynamic Signal Analyzer. The SigLab hardware is optimized in terms of noise and distortion. It has excellent anti-alias filters which are mandatory for high quality spectrum analysis. The screen capture below shows a typical spectrum analysis. The setup files used are *sa1.vna* and *sa1.vfg*. You set the upper axis *Y func* selector to *y:aspec* (*aspec* indicating auto-spectrum). You may then choose either a *linear* or *log* X axis format with the *X lin/log* control. The *X units* control allows a selection of Hertz or rpm (revolutions per minute) for rotating machinery measurements. For this example, you should select *Hertz*.

You use the Y *units* menu to select the desired display format. For this example, you should select *dB* (decibels). The zero-dB reference is set to 1-volt rms. Note that the zero dB value is included in the channel annotation inside the axis (yellow text) since the value can be independently set for each channel.



The real-time *RMS level* of the signal is computed and displayed over the *displayed X axis* range. If you change, the X axis scaling this readout will reflect the signal RMS level over the new range you have selected.



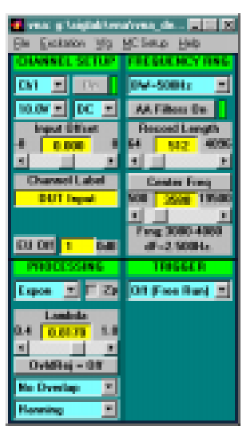


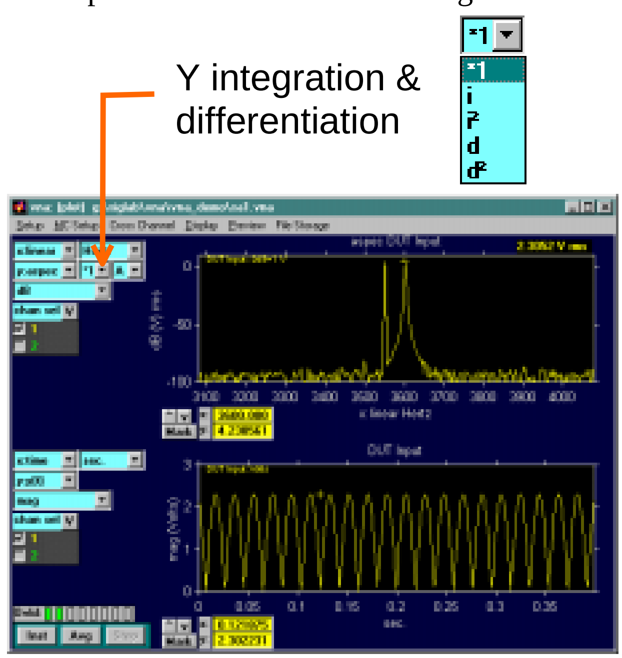
When multiple channels are being displayed on one axis, you can **switch the RMS readout** by clicking on the channel annotation text inside the axis. This action also brings the respective trace to the forefront and makes it much easier to cursor the selected channel.

For this measurement, the boxcar window was used, so there is potentially considerable spectral leakage. Notice that only one peak in the spectrum is apparent. However, you can see from the function generator control panel that a two-tone signal is being generated and the tones are spaced only 50 Hz apart. Here is where you can use SigLab's hardware zoom to obtain increased spectral resolution around a desired frequency.

By clicking on the *Zoom Off* button, which **toggles zoom on**, you will change the analysis resolution from 100 Hz to 2.5 Hz. This clearly shows the two signals (tones). Note that the zoom parameters (center frequency and bandwidth) were saved in the *sa1.vna* file. Now, when you change the Boxcar window to the Hanning window to reduce spectral leakage, you also decrease the frequency resolution. You now see that the signal at 3550 Hz, which was only one spectral line with the Boxcar window, is three spectral lines with the Hanning window.

You can change the upper *Y axis scale* to cover from 10 to –100 dB (link to common ground cursor) to demonstrate the superb linearity and noise qualities of SigLab's input and output subsystems.

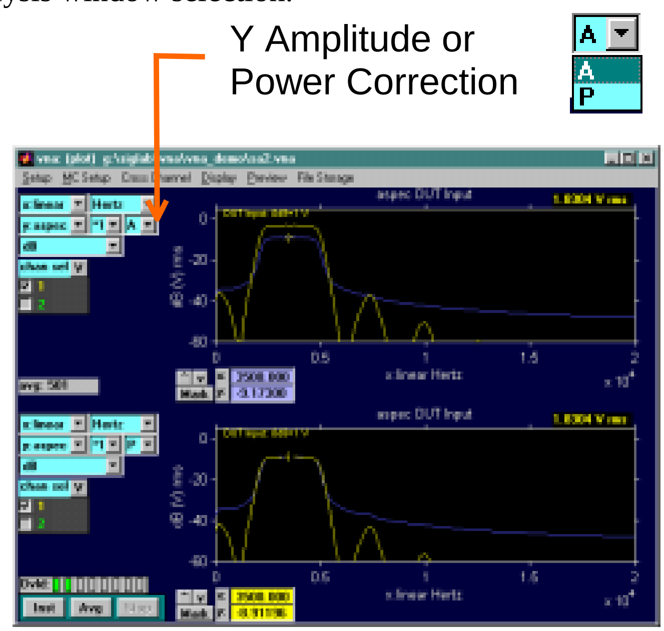




When you use accelerometers or velocity transducers with SigLab, Engineering Units become extremely important in order to display measurements in the units associated with the transducer.

Beyond Engineering Units, the Dynamic Signal Analyzer offers a selection of integration and differentiation for spectrum displays. The **1* selection indicates that no operation is being performed on the data. The *i* and i^2 represent single and double integration while the *d* and d^2 represent single and double differentiation. These operations are performed by dividing or multiplying the spectrum data by the radian frequency (omega). They do not affect time history results. With these options, you can make a measurement with an accelerometer and display velocity or displacement.

You use the *Y Amplitude or Power correction* control to correct for the measurement errors introduced by the selected FFT analysis window. When you need the correct amplitude readout for sinusoidal signals, select the amplitude correction "*A.*" When you are measuring power density, e.g., a non-periodic random signal, then use the power correction "*P*" to provide consistent results independent of analysis window selection.

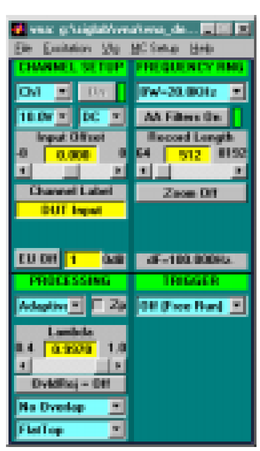


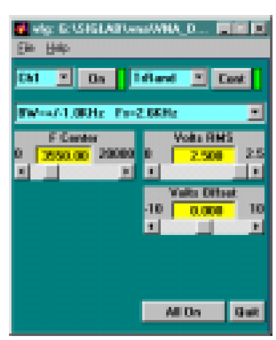
An example of the application of this factor is shown to the right. A bandlimited random signal is generated (file *sa2.vfg*) and initially analyzed (file *sa2.vna*) using the Boxcar window and approximately 500 averages. The resulting spectral displays were overlaid using the trace overlay feature and are shown in violet.

Note that both the Amplitude and Power selections provide the same measurement results using the Boxcar window since the Boxcar is actually no window.

You can change the analysis window to FlatTop and repeat the measurement. Notice that sidelobes on the noise are now visible since the spectral leakage is significantly reduced. However, there is a significant difference in the upper axis between the two traces in the mid-band frequencies of the noise.

When you select the Power correction factor for the lower axis you can see that there is virtually perfect agreement between the results.

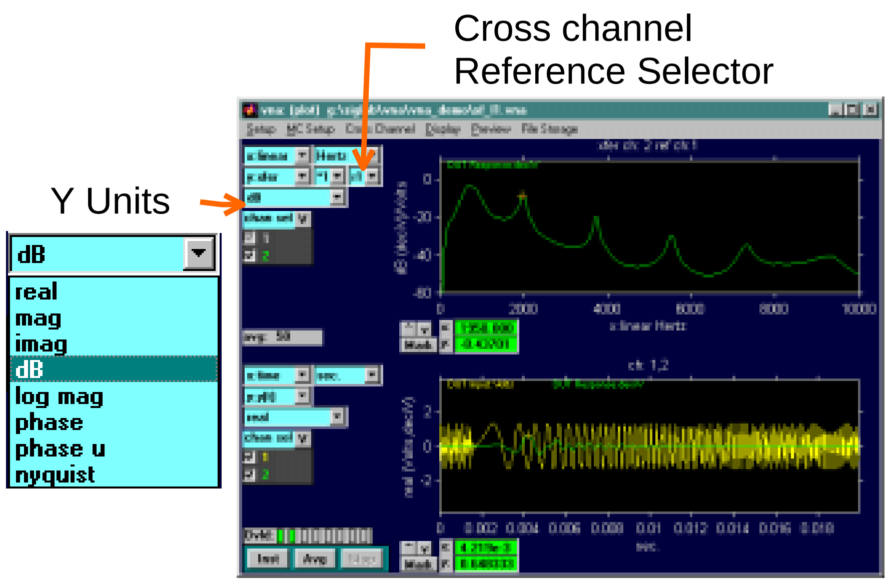




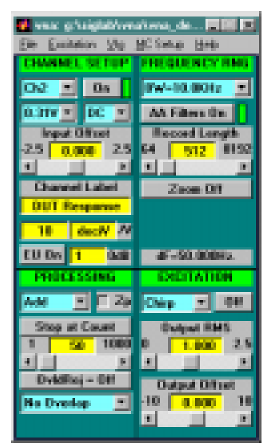
Transfer Function & Coherence (Linked Excitation)

Transfer function measurement is another SigLab strength. The easiest way to make this measurement is to use SigLab's internal source and leave the details of the setup to the software. When this is not possible, you use Independent Excitation and perform the set up of the excitation manually. This situation is covered in the following section -Transfer Function (Independent Excitation).

For this example, (file xf_l1.vna) the dB format is again chosen. The zero dB reference level for transfer functions is always one, and the measurement is not affected by the channel 0 dB settings. These are used exclusively for spectrum analysis.



You select the transfer function measurement by changing the *Y func* selector to *y:xfer*. When you do this, the *Y units* popup will contain the selections shown to the right. The transfer function measurement requires averaging. When you push the *Inst* button while displaying the transfer function, a warning dialog will appear. Exponential or Adaptive averaging may be selected with an appropriate forgetting factor when continuous display updates are desired. Setting the forgetting factor to zero (or a small number) is equivalent to no averaging.

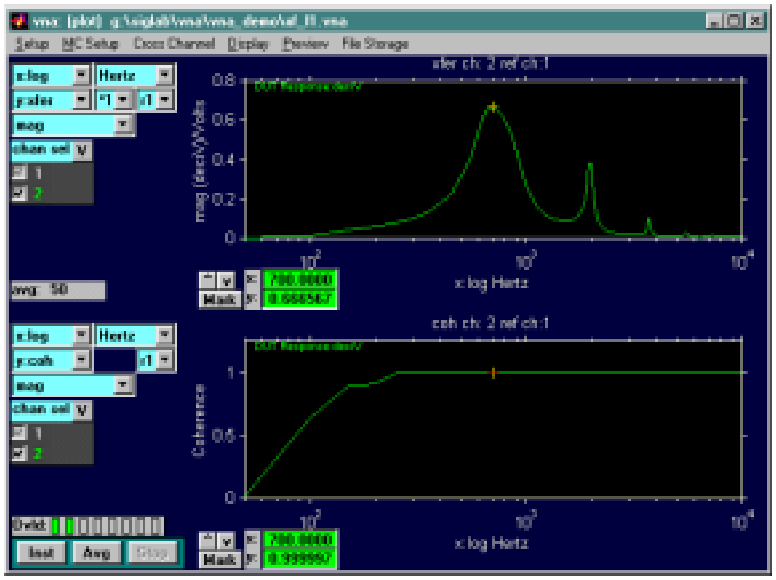


The transfer function is computed by taking the cross-spectrum between the reference and response channel and dividing it by the auto-spectrum of the reference channel.

This leads to an <u>unbiased</u> measurement if the reference channel is relatively noise free. For more information on transfer function measurements, see the application note entitled:

Estimating Transfer Functions with SigLab.

The upper axis shows the transfer function estimate of an Acoustic Tube. Be advised that this measurement will not agree in detail with that for any other Acoustic Tube because of variation between tubes. The lower axis shows a snap shot of the input and output time histories. The Cross Channel Reference Select popup indicates that channel 1 is the reference (*r1*) for this measurement. Since no other channels have been selected as references in the Cross Channel Selection matrix dialog, r1 is the only entry in this popup.



The screen capture to the right shows the same transfer function measurement, but with a different selection of display options. By clicking on the X axis *linear/log* popup, you can select a *log X* axis. Then you can use the *Y units* popup to select *mag*. The upper plot now displays the magnitude of the transfer function.

The lower plot displays the coherence function.. Coherence is an indication of that portion of the response power that is due to the input excitation. It has a range of zero to one. The coherence plot shows that the coherence is excellent at frequencies above about 300 Hz. At low frequencies, it drops to a very small number since the response of the DUT is very small. Whatever DUT output there is at low frequencies, it is not correlated with the DUT excitation. The coherence is calculated by dividing the magnitude of the cross-spectrum squared by the product of the reference and response autospectra. For the coherence measurement to be valid, more than one average must be taken, otherwise the coherence will be unity.

NOTE

The coherence, when time domain averaging is used, is meaningless and will always be unity.

The integration and differentiation options are also available with the transfer function measurement. In this case, they are computed by multiplying the transfer function

$$\frac{1}{j\omega}, \frac{-1}{\omega^2}, j\omega, -\omega^2$$

measurement by: for the *i* , *i*^{2} single / double integration and *d* , *d*^{2} differentiation operations.

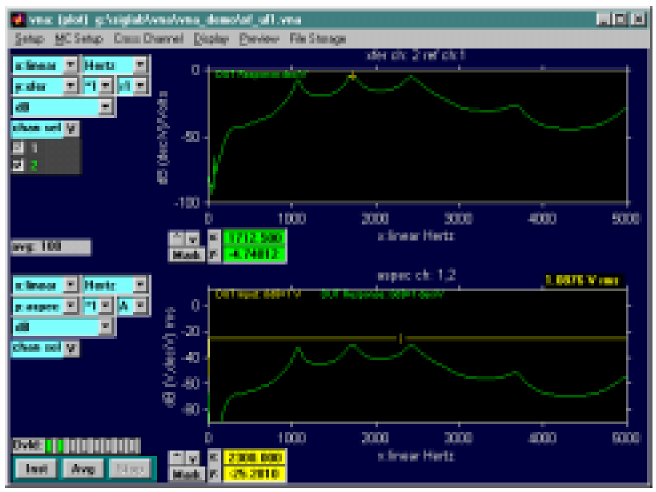
Transfer Function (Independent Excitation, Func. Gen.)

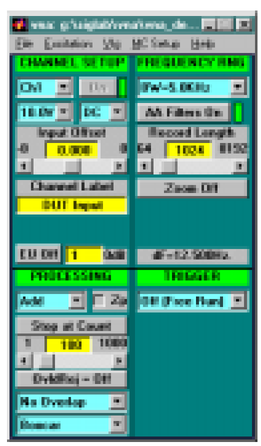
When you want full control over excitation parameters, select the *Independent* Excitation mode and use the Function Generator (***vfg***) application to set up the excitation parameters.

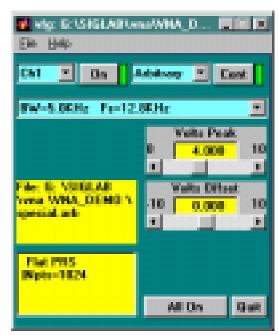
Here are some measurement setup suggestions:

1. Select the analysis bandwidth to cover only the region of interest, but no more.
2. Match the excitation bandwidth to the analysis bandwidth.
3. If a periodic signal is used (e.g., a user created arb, or chirp), match the spectral lines of the excitation to the analysis spectral lines and use the Boxcar window.
4. If a random signal is used, select an analysis window such as Hanning.

The example to the right illustrates points 1,2 and 3 above (files *xf_ul1.vna* and *xf_ul1.vfg*). The lower axis displays the power spectrum of the DUT excitation (yellow) and DUT response (green). The excitation is being provided by an arbitrary output (file *special.arb*, generated by the code in the file *my_arb.m*). The output bandwidth is set to agree with the input analysis bandwidth (5000 Hz) and the input frame size is set to agree with the output arbitrary record length (1024). This forces the excitation spectral lines to match the analysis spectral lines. The excitation spectrum is seen to be virtually flat, by design. The resulting transfer function is plotted on the upper axis.







Transfer Function & Coherence (Independent Excitation, Modal)

The most frequent use of the Independent Excitation mode will be acquiring transfer function measurements for Modal Analysis. The Dynamic Signal Analyzer application has numerous features that are very important for modal analysis measurements:

1. Multi channel setup
2. Multi channel previews
3. Automatic overload rejection
4. Manual frame rejection
5. Wide trigger processing bandwidth with "No Filt" setup option
6. Force and Response windows
7. Menu to Save data in ME'Scope format
8. Auto incrementing file names

Since measurements for Modal analysis often involve four or more channels, the multi-channel setup and preview windows are especially valuable. Their operation has been covered in previous sections of this manual.

Automatic overload rejection is another important feature for modal Impact testing. When this option is active, an overload on *any channel* will cause the frame to be rejected from the average. SigLab's extremely high dynamic range allows you to set the full scale inputs on all channels to be far greater than anticipated signal levels so that overloads should be a very infrequent event.

Impact hammers with force transducers typically produce a uni-polar pulse. The following points will insure reliable triggering and processing:

1. Set the force channel (usually channel 1) to a level that is about 2-3X the maximum desired trigger pulse level.
2. Set the trigger level to be either 9% or 18% of full scale. Note that for a uni-polar pulse, 0% is never an appropriate choice.
3. Choose "No Filt" under the Trigger setup, which provides a full bandwidth data path for the trigger circuitry.
4. Be certain that you know the polarity of the trigger pulse!
5. The Input Active light will be flashing when SigLab is waiting for a trigger event. When the light goes to a steady on condition (visible at low bandwidths), the trigger has occurred and the remainder of the frame is being acquired.
6. Set the pre-trigger between 0 and –10% so that the trigger event occurs well within the first 20% of the frame.

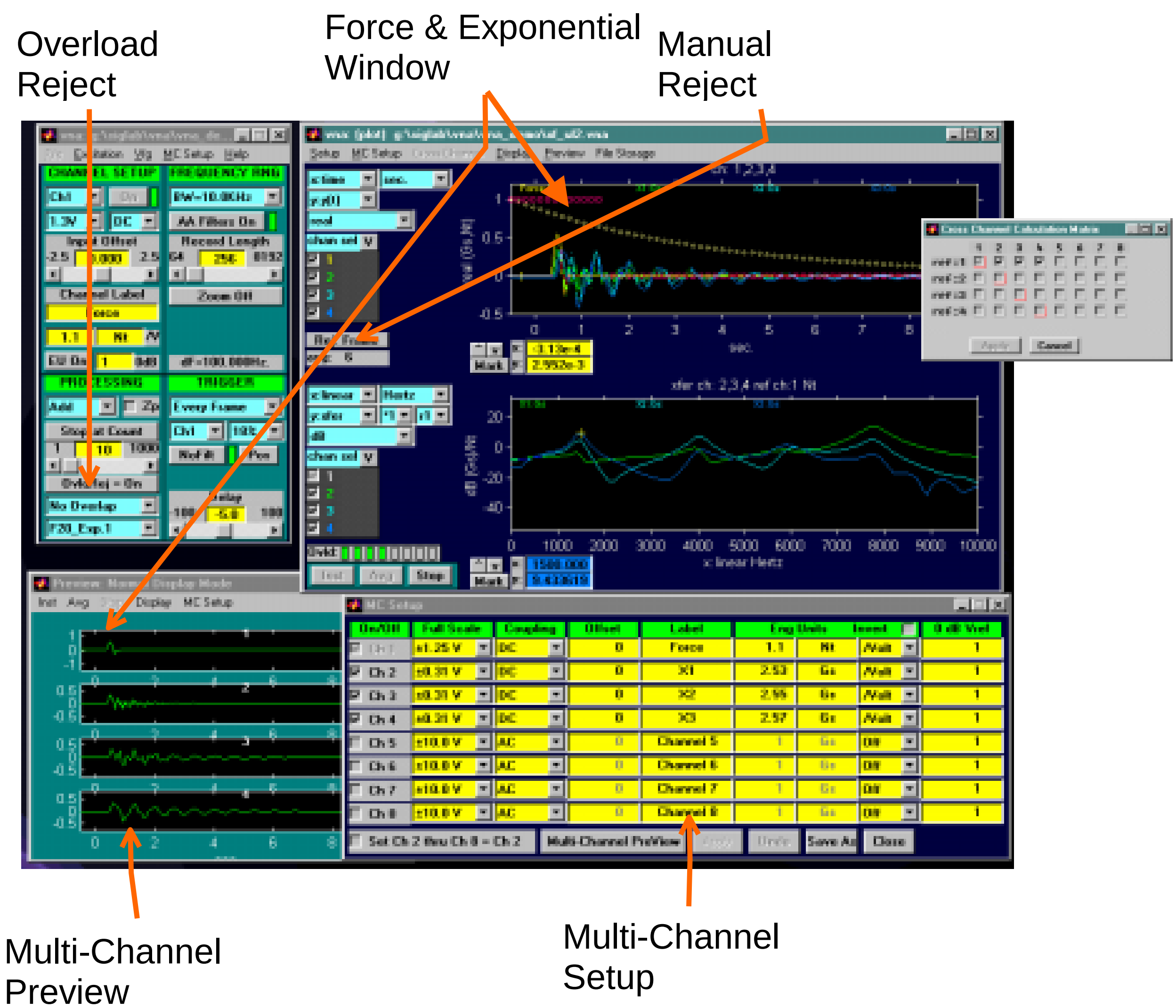
Manual Frame Reject

When a triggered acquisition is setup using an input channel as the trigger source, a *Rej. Frame* button will appear in the Plot window once the *Avg*(eraging) process has been started.

When this button is pushed, it rejects the current input frame from the averaged results. Some analyzers have two buttons, one for *accept* and the other for *reject*. These analyzers require you to click a button for each impact, which can be cumbersome.

With one *reject* button, *accept* is tacitly assumed and you only need hit the *Rej. Frame* when the occasional hammer blow goes awry. The downside of this operator simplification is that the last frame in an averaging sequence cannot be rejected. If you are troubled by this, simply choose the non-stop averaging mode and manually terminate the averaging when the desired average count is reached.

The following example cannot be carried out with the Acoustic Tube DUT since a single input, 3 output system is required.



In this example measurement, channel 1 is the (single) reference and channels 4,3,2 are the response channels. You can see this from the Cross Channel calculation matrix in the above screen capture. This example shows some of the measurement issues and features that we have discussed.

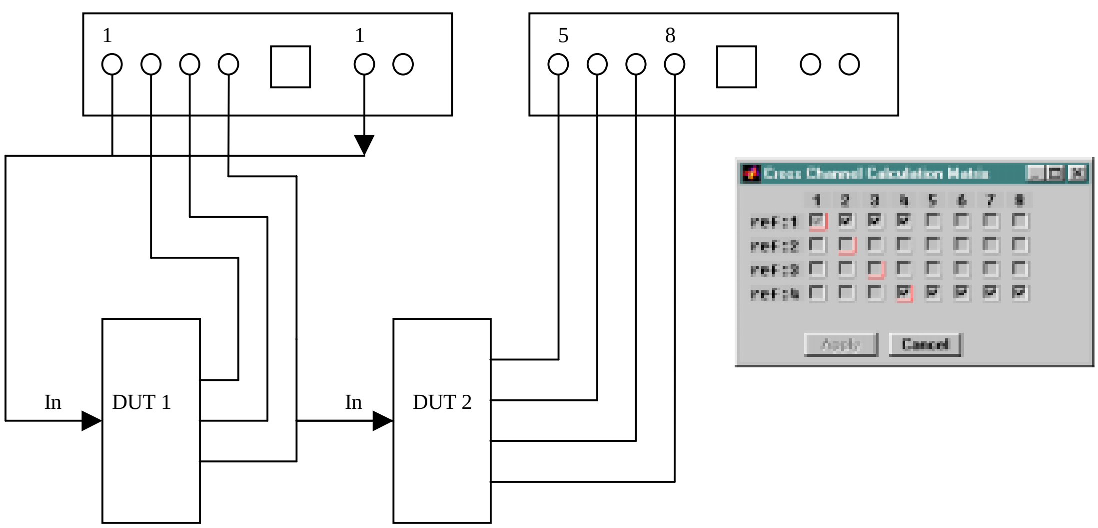
The force and exponential response windows are selected and are plotted along with the 4 channels of time histories on the upper axis. The trigger threshold is set to %18 and a pre-trigger delay of –5% puts the trigger event (channel 1 in yellow) in the center of the force window. The manual reject button is visible in the Plot figure window. Automatic overload rejection is enabled. The *Manual Reject Frame* button will appear when the following conditions are selected:

1. Trigger Every Frame
2. An input channel is used as the trigger source
3. Averaging has been started

For information on file name and channel label conventions relevant to modal analysis, review Section 7.

A Multi Reference Measurement

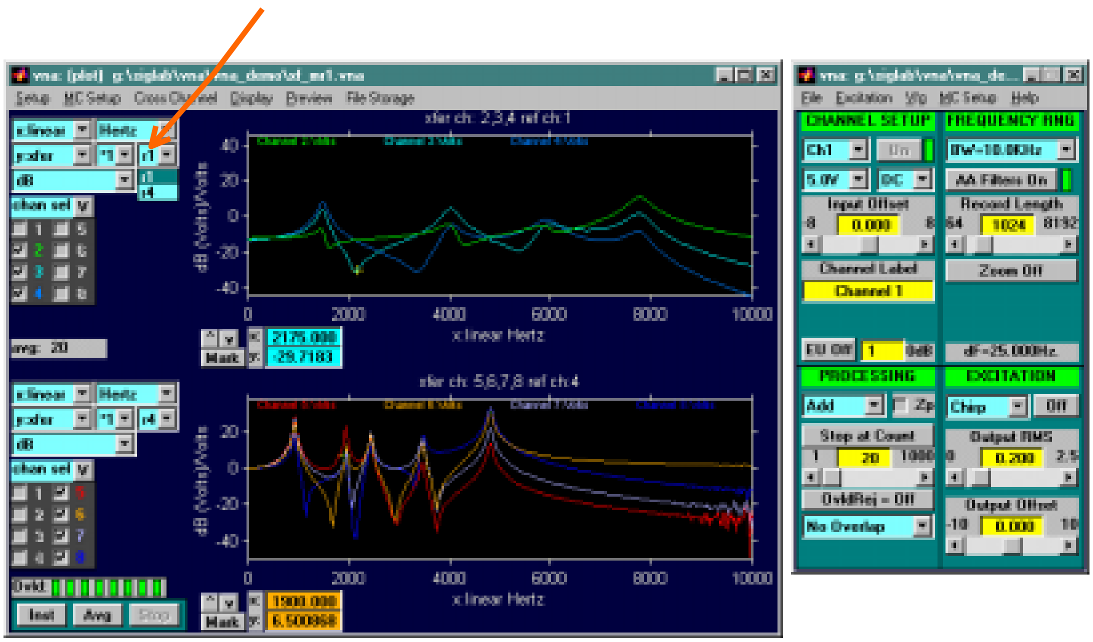
The Dynamic Signal Analysis software supports cross function calculations with up to 4 independent reference channels. **Channel 1 is always designated as a reference** and channels 2,3 and 4 **may** be designated as reference channels. To demonstrate a multi-reference measurement scenario, two multi-output DUTs were connected as shown in the highly simplified block diagram below. The excitation to DUT1 is from SigLab output channel 1 while the excitation for DUT2 is the third response from DUT1.



The Cross Channel calculation matrix (above right) is setup to enable measurements of the cross functions between channels 4, 3 and 2 with channel 1 as the reference, and, between channels 5,6,7 and 8 with channel 4 as a reference. Channel 4 therefore serves as both a response and a reference channel.

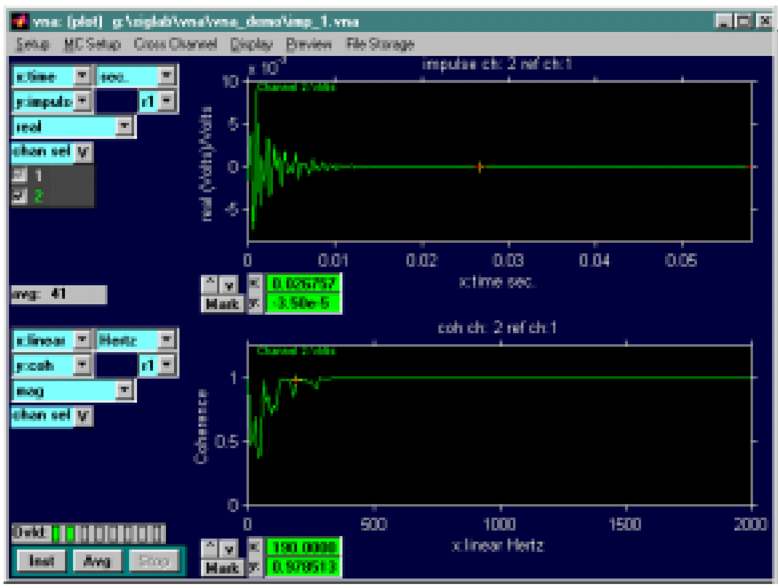
The graphic below shows the 7 transfer function measurements. You can choose a reference channel to be associated with each axis. In this example the upper axis has the measurements associated with reference channel 1 while the lower axis has those associated with reference channel 4. Since the excitation to DUT2 is a response from DUT1, it is anything but "flat". Its power spectrum will closely match the blue curve in the upper axis. Using channel 4 as a reference compensates for this sub-optimal excitation.

Ref. Chan. Select



Impulse Response

The impulse response is simply the inverse Fourier transform of the transfer function. Although this is simple in principal, it is really a deconvolution of the excitation from the DUT's response and, as such, is not very numerically robust. Small errors in the transfer function can lead to large errors towards the latter portion of the impulse response. You can measure the impulse response of the Acoustic Tube DUT by connecting it in the normal way for a transfer function measurement. The file *imp_1.vna* has the setup and measurement data shown in the following screen. The impulse response is shown in the upper axis along with the coherence in the lower axis. The impulse response is not valid for measurements that use zoom.



Auto-correlation

Auto-correlation is analysis of the similarity between a signal and a time-shifted version of itself. It can be used to extract a periodic signal from noise. To explore an example:

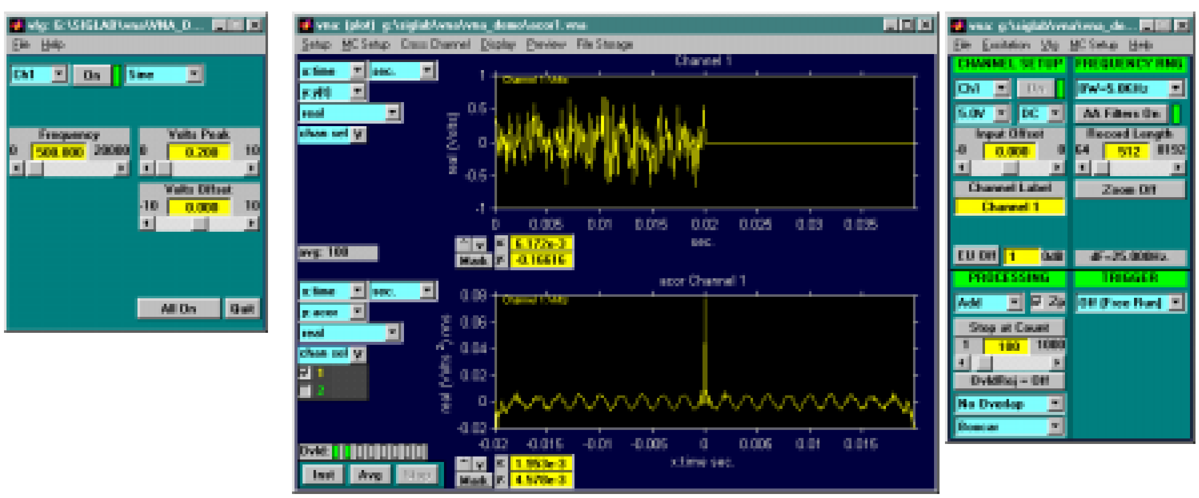
1. Connect Output 1 and Output 2 together using the RCA Y cable (or equivalent). Since SigLab's output impedance is fifty ohms, this connection sums the two outputs and divides the respective amplitudes by a factor of 2

2. Connect this summed signal to Input Channel 1.

3. Set Function generator channel 1 to create a 500 Hz sine wave at amplitude of 200 mV peak.

4. Set Output number 2 (this example cannot be done with a single 50-21) to produce 20 kHz bandwidth random noise with a 1 volt RMS level. This setup is in file *acor1.vfg*.

You now can use the setup file *acor1.vna* for the control settings to make the measurement shown below. Notice that when you push the Avg button the lower axis, which is displaying the auto-correlation, will converge to the measurement shown.

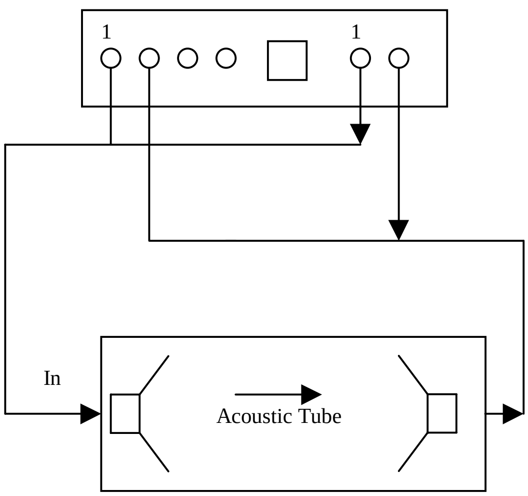


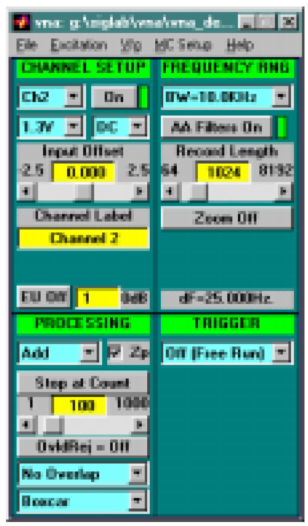
The upper trace (time history) shows that zero-padding has been enabled. You should use the Boxcar analysis window for the best correlation measurements. The auto-correlation function (lower axis) will always be a maximum when the time shift is equal to zero. The spike centered at t=0 is from the random noise. The lower level periodic waveform is due to the sine wave buried in the noise. Note that the sine wave is not visible in the time history plot but is clear in the auto-correlation plot.

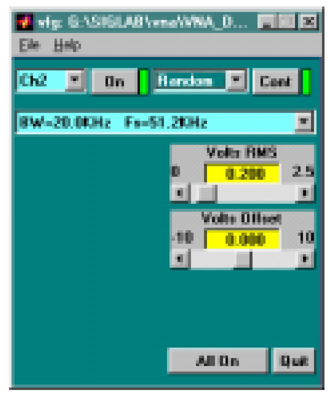
Cross-Correlation

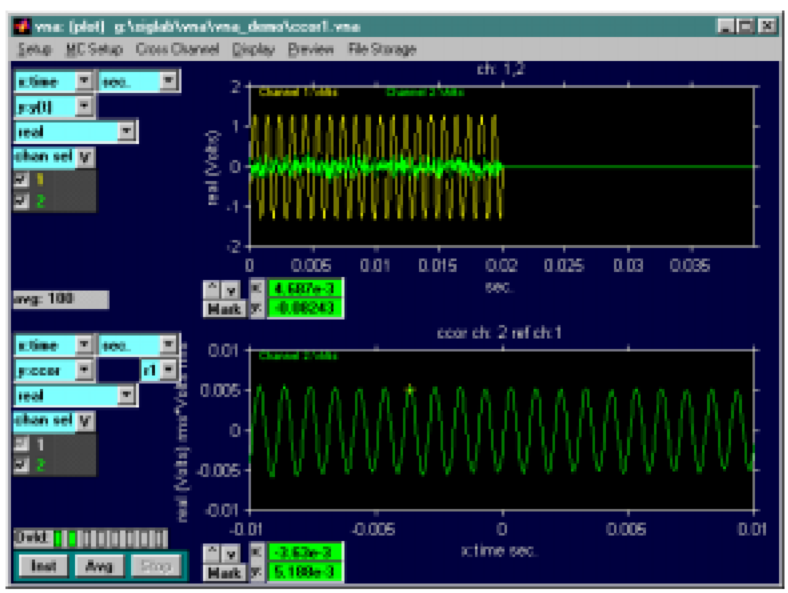
Cross-correlation measures the similarity of two different signals as a function of the time shift between them. If the same signal is present in two waveforms it will be reinforced in the cross correlation function while any un-correlated signals such as noise will be reduced.

To demonstrate cross-correlation, connect the Acoustic Tube as shown below in the simple block diagram. This is the “normal” connection scheme but output channel 2 will ***seriously*** corrupt the response of the tube with noise before the response is measured by input channel 2. You will need two Y cables (or equivalent) to duplicate this setup. The Acoustic Tube is analogous to a communication path where the signal is transmitted by the left hand transducer (internal to the tube), attenuated and corrupted by noise, and then received by the right hand transducer.





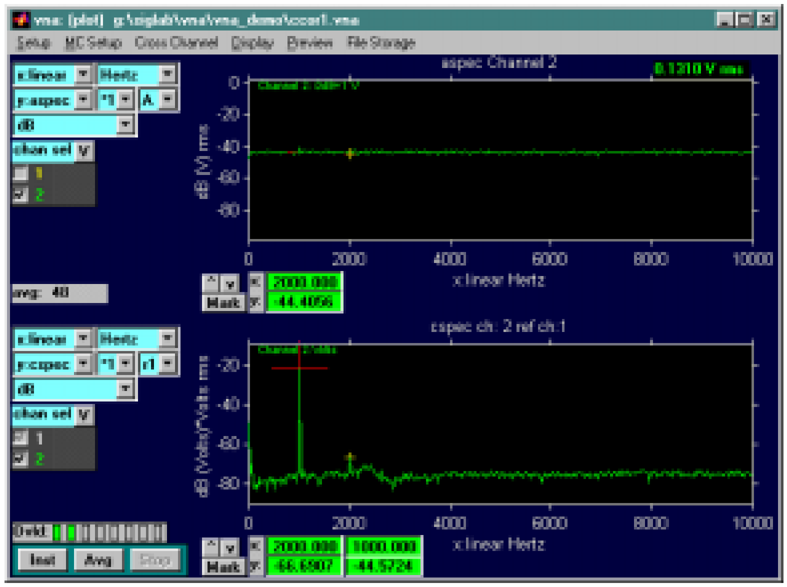




The setup files for *vna* and *vfg* are *ccor1.vna* and *ccor1.vfg* respectively. Turn on the function generator outputs (*All On*) and click on the Avg button to produce the plots shown to the right. The input to the tube is driven by a 1000 Hz sine wave and measured by channel 1 (upper axis yellow trace). The response of the tube is measured by channel 2, but is severely corrupted by the broad band noise from the function generator. Notice that the sine wave is not at all visible in the channel 2-time history (green trace). In fact, if you switch to the auto-spectrum display you will see that the 1000 Hz signal is not visible there as well. The cross-correlation (lower axis) does a rather remarkable job of extracting the signal from the noise because of the availability of the “reference” signal into channel 1.

Cross Spectrum

The previous setup can be used to demonstrate the Cross-Spectrum function. You should turn off the zero-padding, and select the auto-spectrum display only for channel 2 on the upper axis as shown below. You then select the *cross spectrum* between channel 2 and channel 1 on the lower axis. Next, you press the Avg button to generate a display similar to the one below. Notice that the 1000 Hz signal is barely visible in the auto-spectrum in the upper axis. The cross spectrum (lower axis) clearly shows the 1000 Hz component with a 40+ dB margin over the noise.



The cross power spectrum is used in acoustic intensity measurements. The acoustic intensity I is given by: $I = \frac{-1}{\rho \cdot \Delta r} \cdot \frac{Im(Cspec)}{\omega}$

Where ρ = density of air, ω =frequency in radians per second, Δr =spacing of the microphones, and $Im(Cspec)$ =imaginary part of the cross spectrum.

The Engineering Units may be used to enter the microphone calibration factors as well as the $\frac{-1}{\rho \cdot \Delta r}$ term. The integration option selection can be used to provide the $\frac{1}{\omega}$ term. Note that when the integration option is selected, the cross spectrum is multiplied by a factor of $\frac{1}{j\omega}$.
Since the j factor is included in the integration, the *real* part of the integrated cross spectrum should be chosen for the display. These features will allow the acoustic intensity to be displayed directly.

Example: (in MKS units)

Assuming a microphone cal factor $Kmic = \frac{1\mu bar}{5mV} = \frac{2 \times 10^{-2} bar}{V}$

And one $bar = 10^5 nt \,/\, m^2$

therefore, $Kmic = \frac{2 \times 10^{-2} \times 10^5 nt}{V \times m^2} = \frac{2 \times 10^3 nt}{V \times m^2}$

The density of air at STP is $\rho = 1.293 kg \,/\, m^3$ and assumes the microphone spacing of the intensity probe is $\Delta r = 1.2 \times 10^{-2} m$.

The $\frac{-1}{\rho \cdot \Delta r}$ term must be accounted for in the engineering units. Let the $\frac{1}{\Delta r}$ term be included in the channel 1 engineering unit, and the $\frac{-1}{\rho}$ term be included in the channel 2 EU as follows:

$$EU1 = \frac{2 \times 10^3 nt}{m^2} \times \frac{1}{1.2 \times 10^{-2} m} = 1.666 \times 10^5 \frac{nt}{m^3} \text{ per Volt}$$

$$EU2 = \frac{2 \times 10^3 nt}{m^2} \times \frac{-1}{1.293 kg \,/\, m^3} = -1.546 \times 10^3 \frac{nt \times m}{kg} \text{ per Volt}$$

Since the cross spectrum calculation effectivly multiplies the 2 measurement channels, and, this result is multiplied by the $\frac{1}{j\omega}$ factor, the dimensions of the net result will be as follows:

$$\frac{nt}{m^3} \times \frac{nt \times m}{kg} \times \frac{1}{\omega},$$

Representing the above units by the fundamantal units of **M**ass, **L**ength, and **T**ime gives:

$$\frac{M \times L^2 \times T^{-2}}{L^3} \, x \, \frac{M \times L^2 \times T^{-2}}{M} \times \frac{1}{T^{-1}} = \frac{M \times L^2 \times T^{-3}}{L^2} = \frac{Watts}{Area}$$

Therefore, selecting the real part of the single integration of the cross power spectrum with the proper engineering units, will display acoustic intensity in units of watts per square meter.