ARTISAN®
TECHNOLOGY GROUP

Your **definitive** source for quality pre-owned equipment.

Artisan Technology Group
(217) 352-9330 | sales@artisantg.com | artisantg.com

Full-service, independent repair center
with experienced engineers and technicians on staff.

We buy your excess, underutilized, and idle equipment
along with credit for buybacks and trade-ins.

Custom engineering
so your equipment works exactly as you specify.

- Critical and expedited services
- In stock / Ready-to-ship
- Leasing / Rentals / Demos
- ITAR-certified secure asset solutions

Expert team | Trust guarantee | 100% satisfaction

Find the *Vibration Research VR-8500* at our website: *Click HERE*

Information in this document is subject to change without notice.

This document last revised 9-Jan-2003.

1 *VibrationVIEW* Help Index

How to ...

Call Vibration Research Corporation at (616) 669-3028 with additional questions, or contact us at support@vibrationresearch.com.

Menu commands

Graph menu
Cursor menu
View menu
Window menu
Help menu

Control Centers	**Test Definition**	**Stop Codes**
System Check		System Check
Sine	Sine	Sine
Random	Random	Random
Shock	Shock	Shock
Field Data Replicator	Field Data Replicator	Field Data Replicator
Data Recorder		

2 How to ...

- Set up the system
- Set the shaker system limits
- Register Demonstration mode
- Register File Viewer mode
- Configure the inputs
- Set the accelerometer calibration
- Verify shaker system operation
- Calibrate the system
- Enter test definitions
- Run tests
- Display and print graphs
- Store and retrieve data
- Use memorized drives to instantly start tests
- Tune the controller parameters
- Import and export Field Data Replicator waveforms
- Integrate VibrationVIEW into remote applications
- Create customized reports

2.1 Quick Setup Instructions

1. If you are connecting the hardware to your own computer, read the "Computer requirements" section.
2. Install the hardware as described in the "How to install the hardware" section.
3. Install the software as described in the "How to install the software" section.
4. Run *VibrationVIEW* by double-clicking the *VibrationVIEW* icon on the desktop.
5. Set the accelerometer mV/g levels as described in the "How to set the accelerometer calibration" section.
6. Follow the steps described in the "How to verify shaker system operation" section. This step verifies that the shaker system (amplifier, shaker, and accelerometer) is operating properly. If you are running the system "looped on itself" to see how things are working (recommended before starting a real test), you may omit this step.
7. Select a Sine test by selecting the Test..Sine menu command. (If you have not purchased the Sine Test package, proceed to the "How to run tests" for the test package that you have purchased.)
8. Click the "Run" button in the Sine Control Center. The "Run" button will change from black to yellow, and the test will begin to run. When the shaker output reaches the desired level, the "Run" button will change from yellow to green and the frequency will begin sweeping. If the test shuts down due to an error, a message will be displayed in the "Stop Code" line of the Sine Control Center. Click the "Info" button to get more

information about the displayed stop code.

9. If you have trouble running the Sine test, you can make a new test with all the default values by selecting the File..Define New Test menu command. Click the "Next >" button to advance through the new test wizard dialog boxes, accepting the default values for each. Repeat step 8. If the system is "looped on itself", this default test should run.
10. Click the "Stop" button in the Sine Control Center to stop the test. Open a different test by selecting the Test..Open Test menu command and selecting one of the tests listed. The name of the current test appears in the Sine Control Center, in the status bar at the bottom of the screen and also at the bottom of graphs when the "Data at Bottom" check box is selected in the Sine Graph Settings dialog box.
11. Click the "Run" button in the Sine Control Center to start the new test.
12. Read the rest of the manual (or online help) for details on how to change parameters, change tests and change test types. Note: Pressing the <F1> key at any time while running *VibrationVIEW* will open up a window with help information describing the active window or dialog box.

2.1.1 Computer requirements

- 133 MHz or higher Pentium-compatible CPU
- 128 megabytes (MB) of RAM or higher recommended (64 MB minimum supported)
- Network adapter (required for VR8500 connection)
- Second network adapter (optional for network installation)
- VGA-compatible or higher display adapter. (Screen resolution of 1024 x 768 or higher recommended)
- Keyboard, mouse or other pointing device
- CD-ROM or DVD-ROM drive (required for CD installations)
- Hard disk drive with 50 MB of free space
- Microsoft Windows NT 4.0,Windows 2000, or Windows XP operating system.
- Windows-compatible printer(optional).
- Vibration Research Corp. supplies the VR8500 I/O Box and required cables.

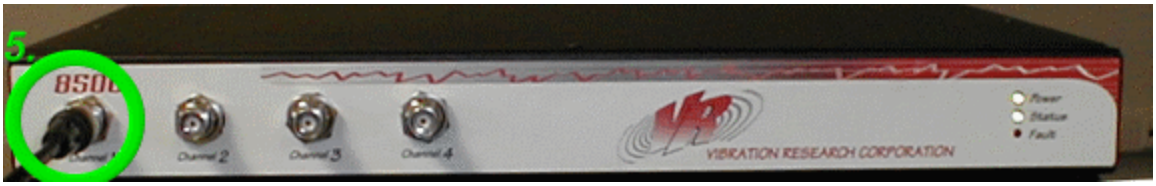
2.1.2 How to install the hardware

Connecting the *VibrationVIEW* I/O hardware:

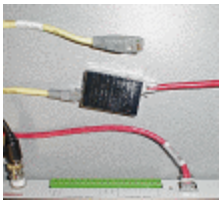
1. Set up the computer in the conventional configuration, with power cables, mouse, keyboard, and monitor.
2. Connect the VR8500 network port to the dedicated network card in the computer using the included yellow crossover cable. *If your VR8500 is labeled "Adapter Cable Required", the included red adapter cable and metallic CAT5 inline coupler MUST be connected between the yellow crossover cable and the network port of the VR8500.*

3. Connect the line cord to the VR8500. The power input will automatically switch for voltage 90-250VAC and 50/60 Hertz.

4. Connect the shaker amplifier's input to the Drive output connector on the rear of the VR8500.
5. Connect an accelerometer to Channel 1 of the VR8500. Other accelerometers can be connected to Channels 2,3 and 4.



Note:
The red cable and coupler is required only if your VR8500 is labeled "Adapter Cable Required"



Connecting two or more VR8500 units

6. Two or more VR8500 units require a network switch.
7. Connect the first VR8500 to port 1 on the switch. Additional VR8500's can be connected to port 2,3, and 4 on the switch. Connect the computer to highest numbered port with the supplied cable.

8. Connect 120 VAC power cords to each of the VR8500's.
9. Connect the shaker amplifier's input to the Drive output connector on the *first* VR8500. For systems controlling two shakers simultaneously, connect the second shaker amplifier's input to the Drive output connector on the *second* VR8500.
10. Connect accelerometer channels 1-4 to channels 1-4 of the first VR8500, accelerometer channels 5-8 to channels 5-8 on the second VR8500, accelerometer channels 9-12 to channels 9-12 on the third VR8500 accelerometer and channels 13-16 to channels 13-16 on the fourth VR8500.

2.1.3 How to install the software

We recommend *Microsoft Windows NT* 4.0, *Windows 2000*, or *Windows XP* operating system.

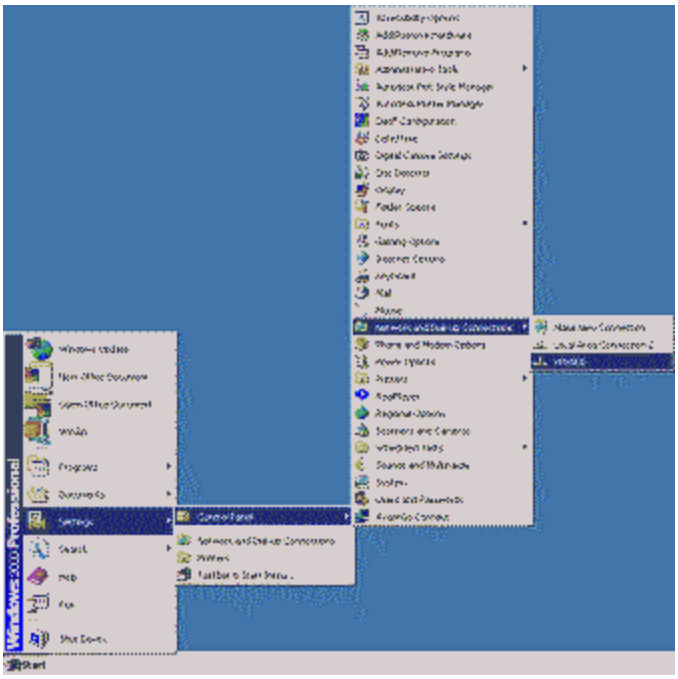
Installation in *Windows NT* 4.0, *Windows 2000,* and *Windows XP*.

Windows NT, *Windows 2000* and *Windows XP* require Administrator privileges to install the programs. In the following steps it is assumed that the *VibrationVIEW* CD supplied with the system is in the CD-ROM drive R:\ and that the *Windows NT* system directory is C:\WINNT. Adjust these to match your system configuration. When you are logged in as Administrator, perform the following steps:

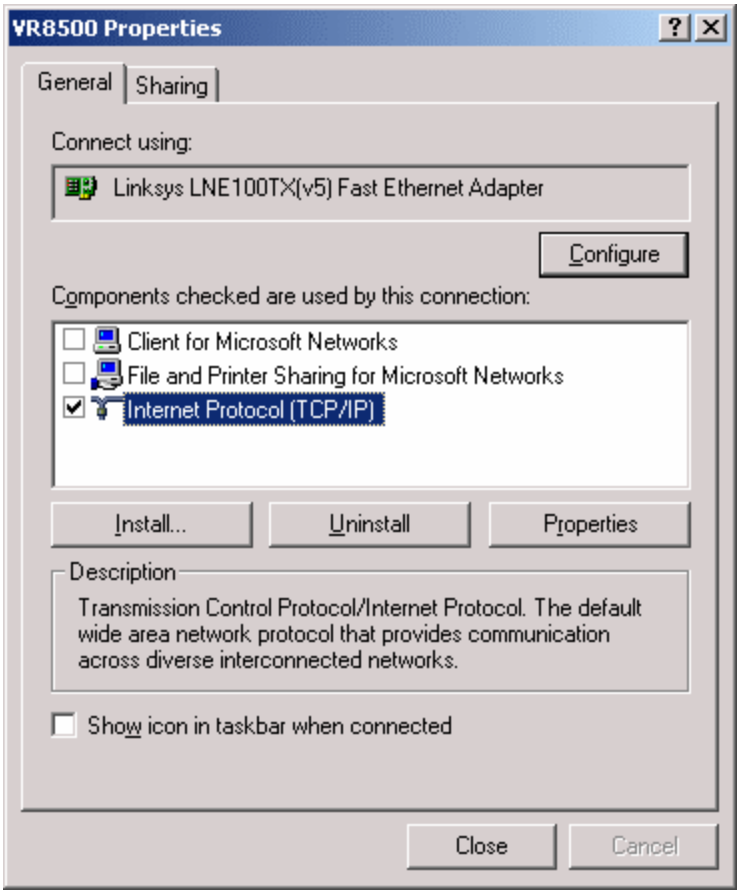
1. Run the program "R:\install\setup.exe" on the CD-ROM (where R: is the drive letter for your CD-ROM drive).
2. The install program (setup.exe) will prompt for the location to install the program. Click the "Next >" button in each dialog box to accept the default values. When the installation

is complete, click the "Finish" button.

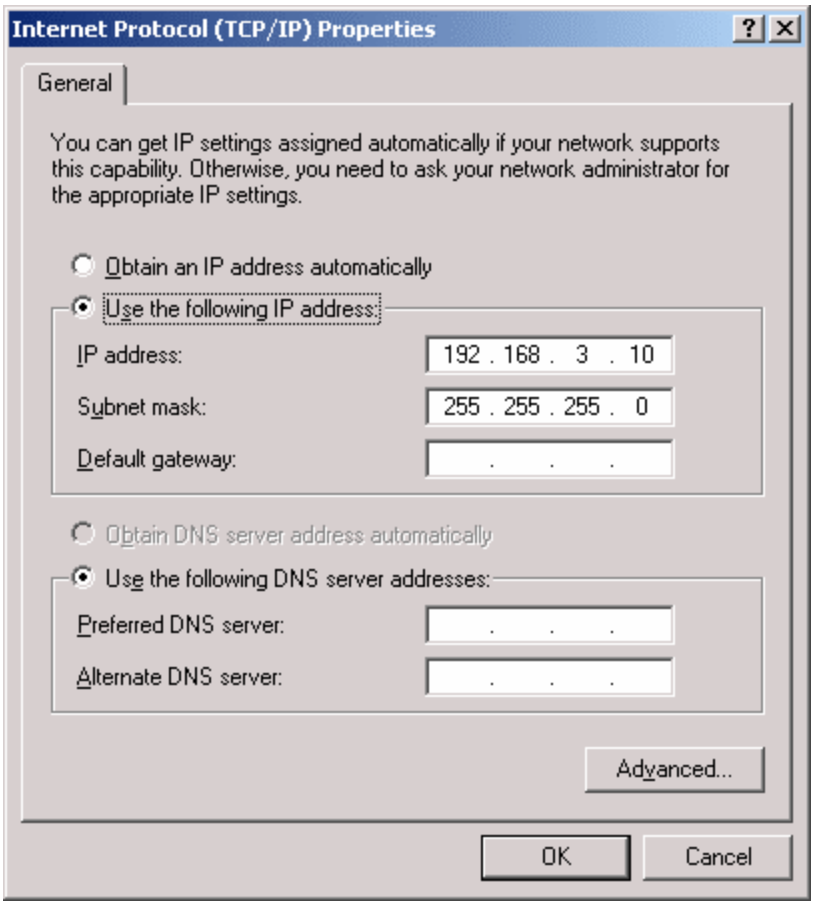
3. Setup the dedicated network card for *VibrationVIEW*. Select the network card dedicated to the VR8500 from the Network and Dialup Connections applet in Control Panel. The example screen shows a network connection named "VR8500". Your network connection will display the name assigned when the network card was installed, typically "Local Area Connection 2".



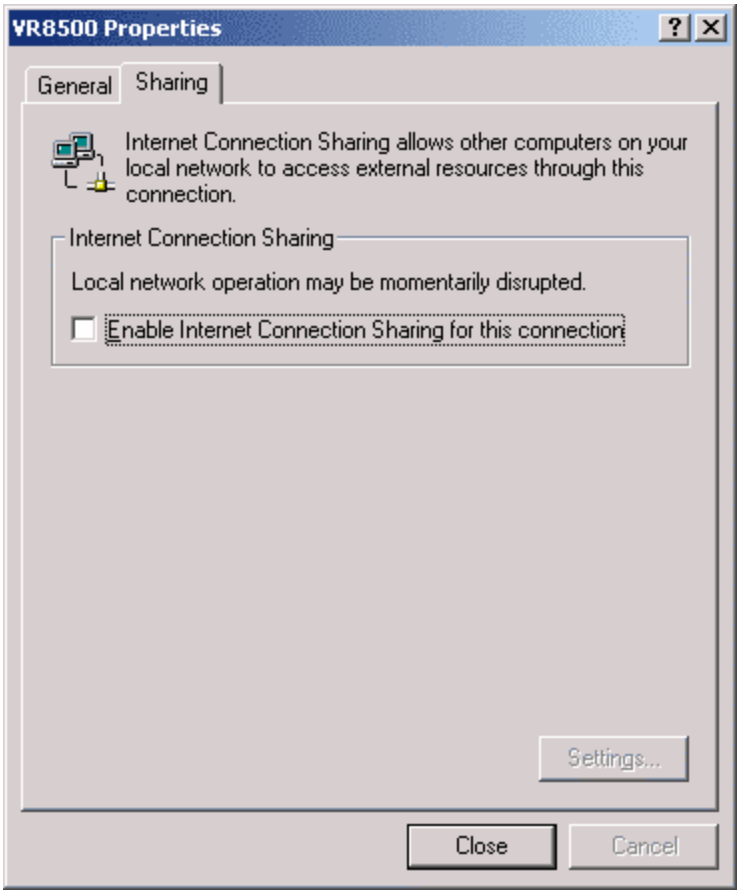
4. Remove the check marks from "Client for *Microsoft* Networks", and "File and Printer Sharing for *Microsoft* Networks." Select Internet Protocol (TCP/IP) and click the "Properties" button to continue.



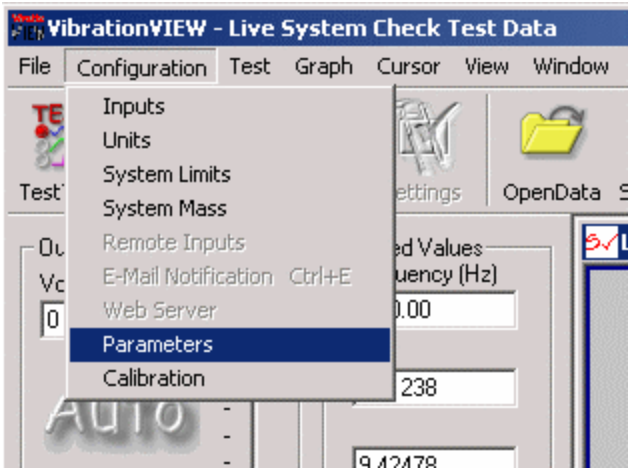
5. Assign address 192.168.3.10 and subnet mask 255.255.255.0 as shown in the Internet Protocol (TCP/IP) Properties dialog box. If multiple network cards are installed in the computer the IP address can be adjusted. Other recommended values are 192.168.2.10 and 192.168.4.10, although any Class C or Class D subnet will work. Click the OK button to return to the previous dialog.



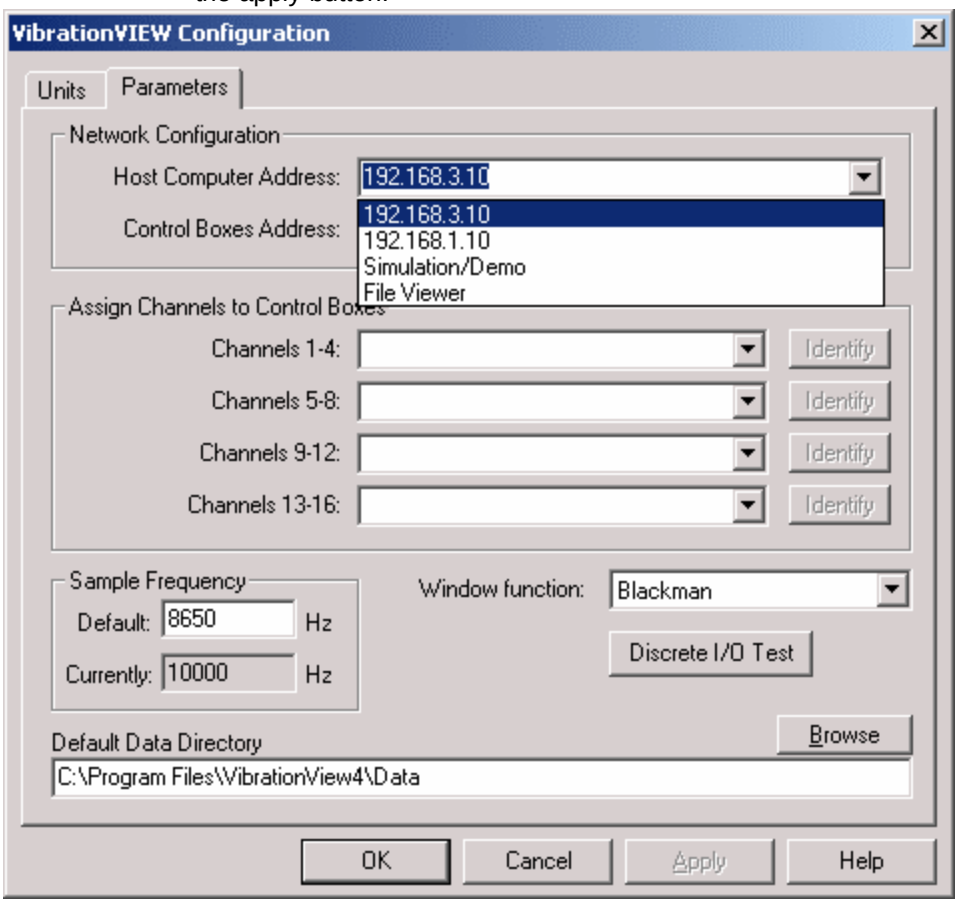
6. Click the Sharing tab in the Properties dialog box. Make sure Internet Connection Sharing for this connection is unchecked (NOT enabled).



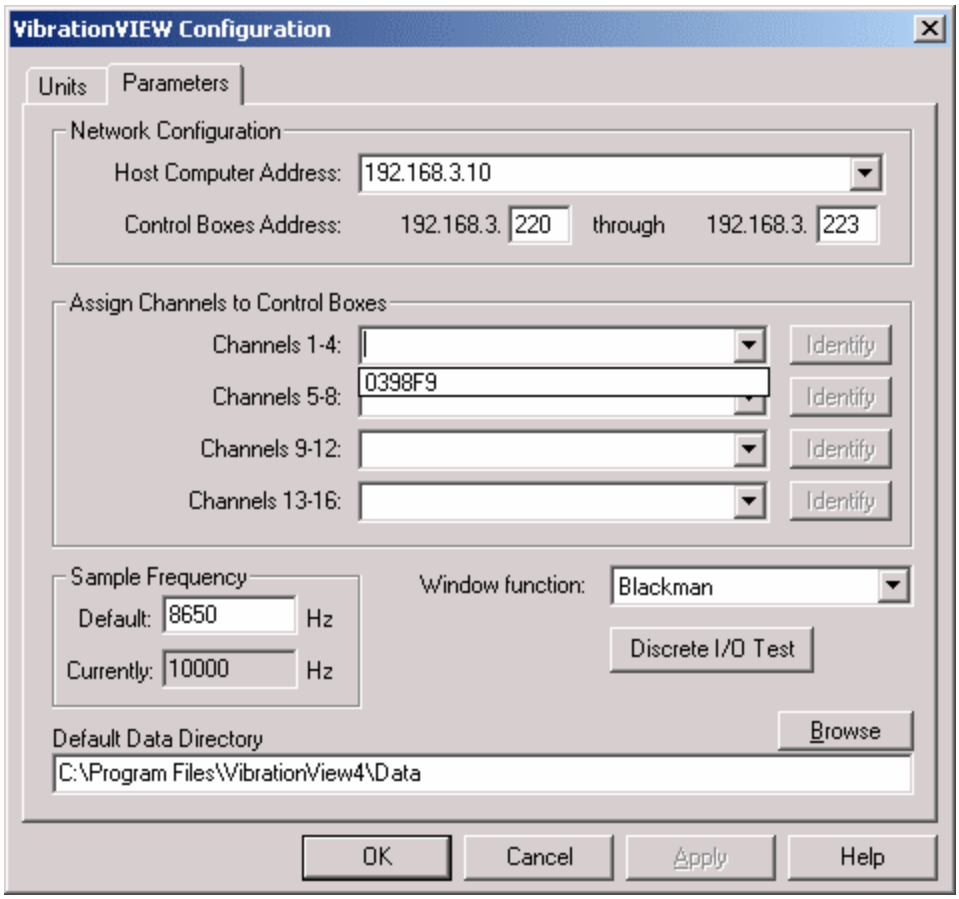
7. *VibrationVIEW* must be configured to use the appropriate network connection after the network is setup. Start *VibrationVIEW* by double-clicking the *VibrationVIEW* icon on the desktop. Select the Configuration..Parameters menu command.



8. Select the network card previously configured. Typically this will be 192.168.3.10. Click the apply button.



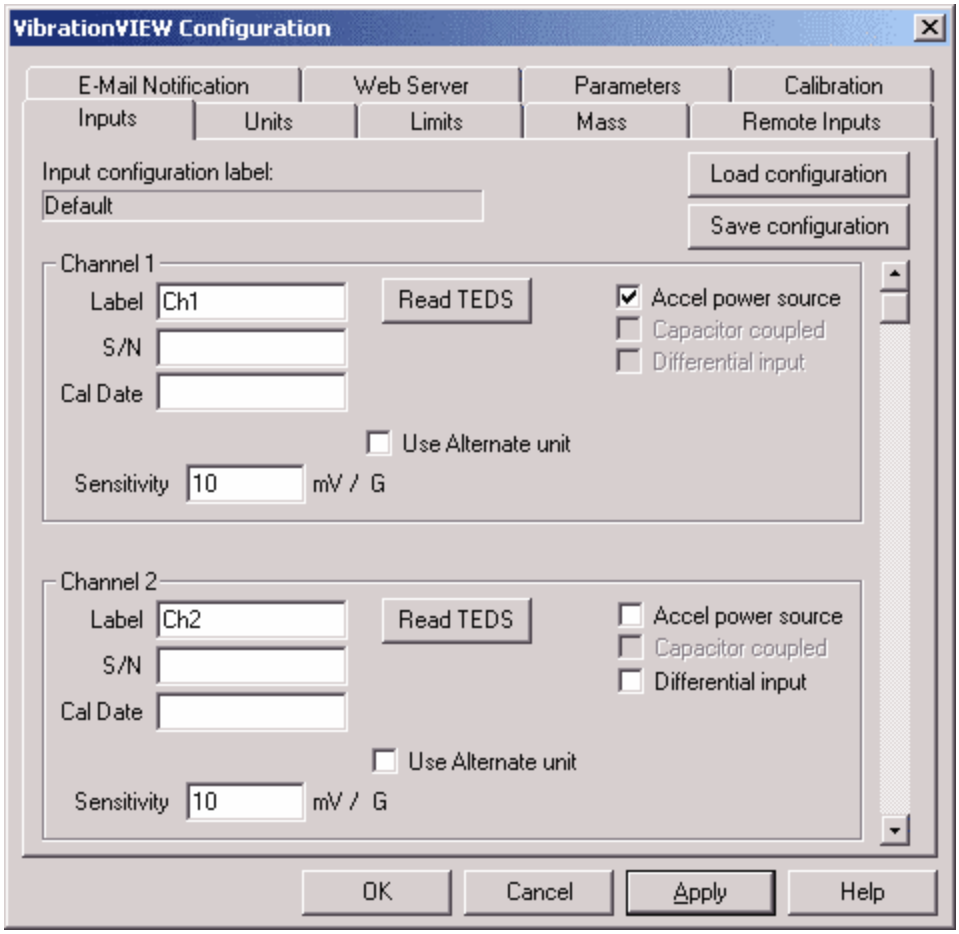
9. Your VR8500 serial number will be added to the drop down menu for Channels 1-4 when it initially connects to the computer. Select the appropriate VR8500 serial number to configure channels 1-4 and, optionally, channels 5-8, channels 9-12 and channels 13-16. The VR8500 control boxes are assigned addresses from the range listed in the Control Boxes Address parameters. This does not need to be changed.



2.1.4 How to setup Inputs

The inputs must be configured for the each connected accelerometer. Each input can be configured as single ended, differential and optionally capacitor coupled. Single ended inputs can also provide a constant current source to power your accelerometers. Other types of accelerometers such as charge coupled accelerometers require external conditioning.

When the appropriate values are entered, click the "OK" button. The new accelerometer sensitivity factors will take effect immediately.



2.1.5 How to Register Simulation/Demonstration Mode

To enable *VibrationVIEW* in Simulation/Demo mode, do the following:

1. Select the Help..Enter Registration Code menu command.
2. Click the "Request Demo Code" button.
3. Fill in all the fields in the "Demo Registration" form and click the "Email Request Form" button to request activation.
4. After you receive response e-mail with an activation code, copy the entire message to the clipboard.
5. Select the Help..Enter Registration Code menu command.
6. Click the "Paste code from clipboard" button.
7. Enter "Demo" into the "Licensed to" field of the Registration Code Dialog Box
8. Click the OK button.

If *VibrationVIEW* is not already configured for Simulation mode:

1. Select the Configuration..Parameters menu command.
2. Set "Host Computer Address" to "Simulation/Demo".
3. Click the OK button.

2.1.6 How to **register File Viewer mode**

To enable *VibrationVIEW* in File Viewer mode, do the following:

1. Select the Help..Enter Registration Code menu command.
2. Enter your Vibration Research supplied User Name into the "Licensed to" field of the Registration Code dialog box
3. Enter your Vibration Research supplied Registration code into the "Registration code" field of the Registration Code dialog box
4. Click the OK button.

If *VibrationVIEW* is not already configured for FileViewer mode:

1. Select the Configuration..Parameters menu command.
2. Set "Host Computer Address" to "File Viewer".
3. Click the OK button.

2.1.7 How to Read the Front Panel Lights

Power The Power Light will be lit whenever the power is turned on (switch on rear of VR8500). If the power light fails to illuminate, verify that the power cord is plugged into a live outlet and the power switch is turned on.

If the Power light flashes or flickers the unit may require service. Contact Vibration Research for assistance.

Status Flashing with one flash every 2 seconds (one second on and one second off) indicates that the box has not been initialized yet.

Flashing with one flash every 1 second (mostly on, with just a brief flash off) indicates the box is connected to *VibrationVIEW* and operating normally.

Steady on indicates the box is initialized, but is not connected to *VibrationVIEW*.

Steady off indicates the box is booting.

Fault The fault light illuminates briefly when the box is initially power cycled for a lamp check.

If the fault light remains on the unit may require service. Contact Vibration Research for assistance.

2.2 How to set the shaker system limits

The controller checks the system specifications to determine if the shaker is able to perform the current test. The shaker force, velocity and displacement limit values only need to be entered once when the system is first configured. The fixture and product masses that are required to properly compute the acceleration limits of the shaker must be entered every time the fixture or product mass is changed.

To set the system limits, select the Configuration..System Limits menu command. In the dialog box, select either the system matching your setup or enter the specifications of your system. These specifications should be listed in the manual for your shaker system.

To enter the system mass, select the Configuration..System Mass menu command. Enter the masses for all the shaker parts on your system. If your system does not have one of the listed parts, enter zero for that item's mass. The sum of these mass values is used to calculate the maximum operating acceleration for your system using Newton's law:

Force = mass * acceleration

2.3 How to set the accelerometer calibration

The calibrated mV/g settings for your accelerometers are entered by selecting the Configuration..Inputs menu command. For each channel, enter the calibration factor (in mV/g) for the accelerometer in use on that channel. This calibration factor is normally stamped on the accel or supplied on an accelerometer calibration sheet. You may also enter a serial number and a calibration date for each accelerometer to aid in tracing the current calibration settings.

TEDS accelerometers are supported through the "Read TEDS" button. TEDS is a standard for smart plug and play transducers. To use TEDS you must have a TEDS accelerometer. Do not attempt to use the TEDS interface on conventional accelerometers. To read calibration information directly from the accelerometers click the "Read TEDS" button.

When the appropriate values are entered, click the "OK" button. The new accelerometer sensitivity factors will take effect immediately.

2.4 How to verify shaker system operation

To verify that the input and output levels and frequencies are correct, perform the following procedure.

1. Select the Test..System Check menu command to switch the system into System Check mode.
2. Select the Configuration..Inputs menu command. Verify that your accelerometer sensitivity and power source settings are configured appropriately. The constant current source is enabled in this dialog box. If you are using constant current accelerometers or a conditioner that requires a constant current source make sure the "Accel power source" is checked. Click the "OK" button.
3. Enter a frequency of 30 Hz and a displacement of 0.1 inches (peak-to-peak)in the System Check Control Center. These are the default values.
4. Verify that the input levels are near zero.
5. Click the "Auto" button. The output level should slowly ramp up until the shaker approaches a 0.1-inch peak-to-peak displacement.
6. If the shaker does not move, click the "Stop" button to turn the output off. Verify that the Drive output connector from the VR8500 is connected to the shaker amplifier's input, that the amplifier is turned on and that the amplifier output is connected to the shaker. Return to step 4.
7. If the shaker still does not move, connect the Drive output connector of the VR8500 to an oscilloscope or true RMS voltmeter, set the output voltage level to 0.5 volts (0-to-peak) and verify that you have a 0.5 volt (0-to-peak) amplitude reading (0.35 volts RMS). If the Drive output connector is not measuring 0.35 volts RMS, your system may require calibration.
8. If the shaker vibrates, but the input waveform remains flat, click the "Auto" button a second time so that the text on the "Auto" button is dark and the output stops ramping up. Check the accelerometer cables to verify that they have good connections and that the accelerometer conditioning equipment is turned on. If you are using the internal current

source to power the accelerometer, verify that "Accel power source" on Configuration..Inputs dialog box is checked. If there still is no input signal, connect the conditioned accelerometer signal to an oscilloscope or voltmeter and verify that you see a voltage reading. If there is no reading, click the "Stop" button to turn the output off, replace the accelerometer cable, accelerometer and/or the accelerometer conditioning equipment with devices known to be working and return to step 4. Note: most problems are due to bad cable connections.

9. Once you get both an output and an input signal, use a displacement meter to verify that the shaker peak-to-peak displacement matches the value shown in the System Check Control Center. If it does not match, select the Configuration..Inputs menu command to verify that the accelerometer sensitivity settings match the calibrated values for the accelerometers you are using.

2.5 How to calibrate the system

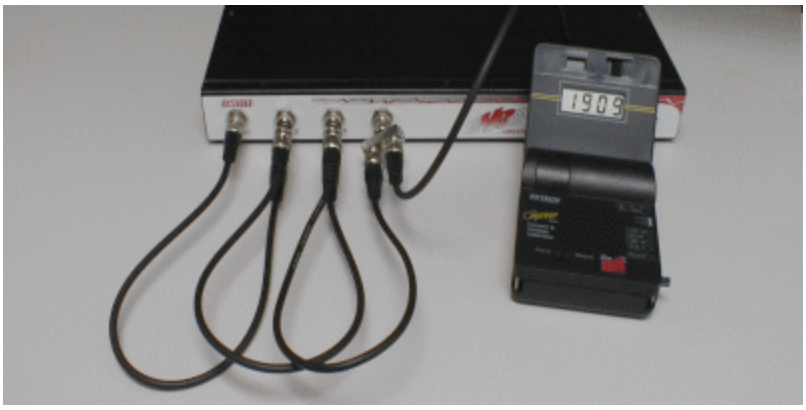
This calibration procedure should be performed annually. ***Note: If you just received your system from Vibration Research this calibration does not need to be done until the due date on the Calibration Certificate in the front of the manual.***

Required for Calibration

(4) BNC cables
(3) BNC T-connectors
(1) Stable calibrated DC voltage source

Procedure requires approximately 30 minutes.

1. Connect a power cord to the VR8500.
2. Turn the unit on.
3. Run *VibrationVIEW*, and allow 1 hour for the temperature to stabilize before performing calibration..
4. Select the Test..System Check menu command.
5. Select the Configuration Calibration menu command
6. Click the "Next >" button on the calibration wizard.
7. Connect four coaxial cables together with the three T-connectors.
8. Connect one end of the cable to a stable DC reference voltage source (1.000-2.000 Volts DC).
9. Connect the T-connectors and the end of the cable to channels 1-4 of the VR8500.



10. Enter the Calibrated DC reference voltage from the power source into the dialog box and click the "Next >" button.
11. The VR8500 will cycle through all the sample frequencies and calibrate all four channels

to the reference source.

12. Connect one BNC Cable between the Drive output connector and channel 1 and a second cable between the COLA/AUX output connector and Channel 2. Click the "Next >" button.

13. The VR8500 will cycle through all the sample frequencies and calibrate both outputs to the inputs.
14. When the calibration is complete click the "Next >" button one more time to make calibration permanent. ***Warning!! This will overwrite the previous calibration information in the VR8500, so be sure that you have properly calibrated the unit before making these changes permanent.*** The EEPROM password to make these changes permanent is BurnEE (case-sensitive). Click the "OK" button in the password dialog box after entering the password.
15. Click the "Report" button to save the gain factors and offsets to a text file. This report documents that each sample rate was calibrated to the reference and serves as a diagnostic tool in case of calibration issues.
16. Click the "OK" button in the System Calibration Parameters dialog box to apply the new settings.

Calibration is now complete.

2.6 How to enter tests

- How to enter a Sine test
- How to enter a Random test
- How to enter a Shock test
- How to enter a Field Data Replicator test

2.6.1 How to enter a Sine test

Select the Test..Sine menu command to switch into Sine Test mode.

Select the File..Define New Test menu command to begin defining a new test. This will guide you through the following series of configuration dialog boxes. Default values will be supplied for all parameters. If you are unsure about a parameter, use the default value. After all the values for each dialog box are entered, click the "Next >" button to advance to the next dialog box. To return to a previous dialog box, click the "< Back" button.

1. *Sine Profile* The amplitude and frequency breakpoints and the desired control parameters (acceleration, velocity or displacement) for the test are entered here. Use the scrollbar to scroll through the defined segments and the Insert/Delete buttons to add or

remove segments. The small arrow next to the numbers on the left of the window indicates the current insertion/deletion point. A test may be composed of up to 200 segments.
The frequency and direction at which the sweep begins, along with the sweep rate, are entered at the bottom of this dialog box.

2. *Sine Schedule* The duration of the test is entered here. Durations may be entered in time duration, number of sweeps or number of cycles. The test schedule is also used to schedule sweeps scaled to different amplitude levels and to select one or more fixed frequency tones.
3. *Sine Sweep* Your sweep rate is set next. Enter the sweep rate in the "Speed" box, and click the drop box to select desired units.
4. *Sine Parameters* The feedback control parameters for the test are entered here. For most tests, the parameters may be left at their default values. In some cases (such as when the control accelerometer sees a large resonance) these values will need to be tuned.
5. *Sine Limits* The tolerance and abort limits for the test are entered here. The control tolerance and abort limits apply to the control signal. If monitoring channels are being used, enable the appropriate channels with the check boxes and enter the plus and minus abort limits for the monitor channel. All limits are measured in dB relative to the demand signal at the active frequency.
   The tolerance lines are reference lines shown on the graphs (brown dashed lines) and are used to determine when to switch to "Run" mode when starting a test.
   The Plus and Minus aborts are limits that, when exceeded, cause the test to abort. The control abort lines are shown on the graphs as solid red lines.
   The maximum output voltage limits, both absolute (Max Output parameter) and relative to the measured acceleration level (Max System Gain parameter) are also entered here. These parameters are safety limits that will cause the test to abort when something goes wrong (for example, when an accelerometer cable comes loose.) Refer to the "How to tune Sine controller parameters" section for more information on how to determine the appropriate Max Output and Max System Gain parameters for your test.
6. *Sine Channels* The control channels for the test are selected here, as well as a way to combine the selected control channels and whether or not to use tracking filters for either the control channel or the monitor channel inputs.
7. *Sine Data Storage* The directory into which the data generated by this test will be stored is entered here. Data can be automatically stored at a regular time interval, sweep interval, at the end of each level (levels are defined using the Schedule dialog box) and at the end of the test. Select the check boxes for your desired data storage times and enter the desired interval times/cycles.

At this point the test definition is complete. Click the "Finish" button to close the dialog box. You will be asked if you want to save the test definition to the hard drive. Click the "Yes" button, enter the desired name and click the "Save" button to save the test.

To access parameters for specialized tests, select the Test..Edit Test menu command and click the Resonance, Extremal or Step Test tabs.

2.6.2 How to enter a Random test

Select the Test..Random menu command to switch into Random Test mode.

Select the File..Define New Test menu command to begin defining a new test. This will guide you through the following series of configuration dialog boxes. Default values will be supplied for all parameters. If you are unsure about a parameter, use the default value. After all the values for each dialog box are entered, click the "Next >" button to advance to the next dialog box. To return to a previous dialog box, click the "< Back" button.

1. *Random Table* The amplitude and frequency breakpoints for the test are entered here. Use the "Insert" and "Delete" buttons to add or remove segments. The small arrow next to the numbers on the left of the window indicates the current insertion/deletion point.
2. *Random Schedule* The duration and output amplitude for the test are entered here. Durations are entered in time, using the format hours:minutes:seconds.
3. *Random Parameters* The feedback control parameters for the test are entered here. For most tests the parameters may be left at their default values. In some cases (such as when the control accelerometer sees a large resonance) these values will need to be tuned.
4. *Random Limits* The tolerance and abort limits for the test are entered here. All limits are measured in dB relative to the demand signal. The tolerance lines are reference lines shown on the graphs (brown dashed lines) and are used to determine when to switch to "Run" mode when starting a test. When the number of lines outside the abort limits exceeds the"Max Outlier %", the test will be aborted. The abort lines are shown on the graphs as solid red lines.
   The Max System Gain and Output Threshold parameters are safety limits that depend on your particular shaker and amplifier. Refer to the "How to tune Random controller parameters" section for more information on how to tune these parameters for your system.
   The maximum output voltage limit is also entered here. This parameter limits the maximum output voltage produced by the controller.
5. *Random Channels* The control channels are selected here, as well as averaging or extremal channel combination methods.
6. *Random Data Storage* The directory into which the data from this test will be stored is selected here. Data can be automatically stored on a regular time interval, at the end of each level (levels are defined using the Schedule dialog box) and at the end of the test. Select the check boxes for your desired data storage times and enter the desired interval times.

At this point the test definition is complete. Click the "Finish" button to close the dialog box. You will be asked if you want to save the test definition to the hard drive. Click the "Yes" button, enter the desired name and click the "Save" button to save the test.

2.6.3 How to enter a Classical Shock test

Select the Test..Shock menu command to switch into Shock Test mode.

Select the File..Define New Test menu command to begin defining a new test. This will guide you through the following series of configuration dialogs. Default values will be supplied for all parameters. If you are unsure about a parameter, use the default value. After the all values for each dialog box are entered, click the "Next >" button to advance to the next dialog box. To return to a previous dialog box, click the "< Back" button.

1. *Pulse* The desired pulse width, shape and amplitude for the test are entered here. Also, the allowable pre-pulse and post-pulse acceleration levels are entered here as a percentage of the pulse peak acceleration level.
2. *Schedule* The duration of the test, in number of pulses, is entered here. The schedule also may be used to enter levels scaled to different amplitudes.
3. *Parameters* The parameters that control the behavior of the control loop are entered here. The parameters generally may be left at the default settings. Refer to the "How to tune Shock controller parameters" section for more information on how to fine-tune these parameters for your system.
4. *Limits* The tolerance and abort limits for the test, set as a percentage of the peak output and measured relative to the demand time waveform, are entered here. The tolerance lines are reference lines shown on the graphs (brown dashed lines) and are used to determine when to switch to "Run" mode when starting a test. The Plus and Minus

aborts are limits that, when exceeded, cause the test to abort. The limits on the allowable drive voltage are also entered here.

5. *Channels* Select which channel or channels you want to use as the control signal here. When selecting multiple channels, the time waveforms of the selected channels are averaged together.
6. *Data Storage* The directory into which the data from the test will be stored is selected here. Data can be automatically stored at a regular pulse interval, at the end of each level (levels are defined using the Schedule dialog box) and at the end of the test. All pulses that lie outside of the defined tolerance lines can also be stored.

At this point the test definition is complete. Click the "Finish" button to close the dialog box. You will be asked if you want to save the test definition to the hard drive. Click the "Yes" button, enter the desired name and click the "Save" button to save the test.

2.6.4 How to enter a Field Data Replicator test

Select the Test..Field Data Replicator menu command to switch into Field Data Replicator Test mode.

Select the File..Define New Test menu command to begin defining a new test. This will guide you through the following series of configuration dialog boxes. Default values will be supplied for all parameters. If you are unsure about a parameter, use the default value. After the values for each dialog box are entered, click the "Next >" button to advance to the next dialog box. To return to a previous dialog box, click the "< Back" button.

1. *Playback* The name of the playback file, the record level and the sample rate for that file are entered here. There are also buttons for exporting and importing waveform data to/from Windows .WAV files, where the file can be filtered, cropped or otherwise manipulated using a standard Windows WAV file editor. For details on importing and exporting waveform data, refer to the "How to import and export Field Data Replicator waveforms" section. The playback file can be optionally recorded at this time by pressing the "Record Playback File" pushbutton.
2. *Record* The playback file can be optionally recorded at this time. The record option is accessable by pressing the "Record Playback File" pushbutton on the "Playback" tab. Entered desirered sample rate in the sample rate dialog, select connected channel, and press the "Next >" button to start data-recording. Press "Next >" again to stop record, and enter the storage data file name. The wizard will return to the playback tab, where the newly recorded file ready for playback.
3. *Schedule* The time duration of the test is entered here. The test schedule also is used to schedule durations where the output signal is scaled to different amplitudes.
4. *Parameters* The feedback control parameters are entered here. In most cases the default values will be sufficient. Refer to the "How to tune Field Data Replicator controller parameters" section for more information on how to tune the parameters specifically for your system.
5. *Limits* The abort limits for the test are entered here. The control (Ch1) abort limit applies to the control signal, measured using the accelerometer connected to channel 1. The reference (Ch2) abort limit applies to the reference signal that is either read from a file stored on your hard drive or read from input channel 2 on the front of the *VibrationVIEW* I/O unit. These limits are measured as RMS G's (or whatever the selected acceleration unit is). The Drive limit (the maximum RMS output voltage) is also entered here.
6. *Filter* The frequency range over which the controller will operate is entered here. Typically one would select control from 0 Hz up to 40% of the sampling rate. If you wish to filter out low frequencies to limit the displacement requirements or filter out high frequencies to avoid shaker resonances, a smaller frequency range can be specified.
7. *Data Storage* The directory into which the data from this test will be stored is selected here. Data can be automatically stored at a regular time interval, at the end of each level (levels are defined using the Schedule dialog box) and at the end of the test. Select the

check boxes for your desired data storage times and enter the desired interval times.

At this point the test definition is complete. Click the "Finish" button to close the dialog box. You will be asked if you want to save the test definition to the hard drive. Click the "Yes" button, enter the desired name and click the "Save" button to save the test.

2.7 How to run tests

- How to run the System Check test
- How to run a Sine test
- How to run a Random test
- How to run a Classical Shock test
- How to run a Field Data Replicator test

2.7.1 How to run the System Check test

Use this command to check the system operation. We recommend that you perform this function after making any changes to the system, such as changing accelerometers or cable connections. This is a good way to verify that the accelerometer, amplifier, shaker and control system are all functioning properly.

The principle behind system check mode is simple: A sinusoid of the selected frequency is output with a slowly increasing peak voltage until the desired acceleration/velocity/displacement setting is reached. The default settings of 30 Hz and 0.1 inches work well with most shakers. However, any setting may be used.

1. Place a displacement wedge on your shaker table.
2. Select the Test..System Check menu command to switch to System Check Test mode.
3. Open up a Time graph, displaying channel 1.
4. Turn on the shaker amplifier and any associated electronics.
5. Click the "Auto" button in the System Check Control. Watch the shaker table and input waveform carefully. You should see a sine wave in the graph window and the shaker should begin to oscillate at a visible level. With the default settings of 30 Hz and 0.1 inches you should see a 0.1 inch peak-to-peak displacement on the shaker and a 4.6 G peak sine wave on Channel 1.
6. Once you are convinced the system is operating properly or if you notice something wrong, click the "Stop" button to turn the output off.

2.7.2 How to run a Sine test

To run a sine test, select the Test..Open Test menu command or click the "Open test" toolbar button. A list of test names is displayed. Select Files of type "*VibrationVIEW* Sine Profiles (*.vsp)" and select the test you wish to run. Click the "Open" button to load that test.

To run the test, click the "Run" button in the Sine Control Center. To stop the test click the "Stop" button. Notice the other buttons displayed in this control box. The "SweepUp," "SweepDown," and "SweepHold" buttons control the sweep direction. The radio buttons labeled Accel, Vel and Disp control the parameter displayed for the demand and control readings for each of the eight input channels.

You can also manually adjust the frequency by clicking the "Frequency" button. This will automatically place the output into SweepHold mode and prompt you for the desired output frequency.

2.7.3 How to run a Random test

To run a random test, select the Test..Open Test menu command or click the "Open test" toolbar button.

A list of test names is displayed. Select Files of type "*VibrationVIEW* Random Profiles (*.vrp)" and select the test you wish to run. Click the "Open" button to load that test.

To run the test, click the "Run" button in the Random Control Center. Once the test begins, the "Stop" button will turn red and the "Run" button will turn yellow indicating that the test is starting. Once the desired output level has been reached the "Run" button will turn green.

To stop the test click the "Stop" button. When the test is stopped, both the "Stop" and "Run" buttons will be gray.

2.7.4 How to run a Classical Shock test

To run a classical shock test select the Test..Open Test menu command or click the "Open test" toolbar button. A list of test names is displayed. Select Files of type "*VibrationVIEW* Shock Profiles (*.vkp)" and select the test you wish to run. Click the "Open" button to load that test.

To run the test, click the "Run" button in the Shock Control Center. Once the test begins, the "Stop" button will turn red and the "Run" button will turn yellow indicating that the test is starting. Once the desired output level has been reached the "Run" button will turn green.

To stop the test click the "Stop" button. When the test is stopped, both the "Stop" and "Run" buttons will be gray.

Clicking the "Hold pulse" button will stop the pulse output until you click the button a second time. Clicking the "Open Loop" button will continue outputting the same drive pulse shape repeatedly without making control updates to the waveform.

2.7.5 How to run a Field Data Replicator test

To run a Field Data Replicator test select the Test..Open Test menu command or click the "Open test" toolbar button. A list of test names is displayed. Select Files of type "*VibrationVIEW* Data Replay Profiles (*.vfp)" and select the test you wish to run. Click the "Open" button to load that test.

To run the test, click the "Run" button in the Field Data Replicator Control Center. Once the test begins, the "Stop" button will turn red and the "Run" button will turn yellow indicating that the test is starting. Once the desired output level has been reached the "Run" button will turn green.

To stop the test click the "Stop" button. When the test is stopped, both the "Stop" and "Run" buttons will be gray.

2.8 How to display and print graphs

The graphs for *VibrationVIEW* are where the majority of the test actions are observed. To display a graph, select the File..New Graph menu command or click the "New Graph toolbar" button. A dialog box appropriate for the current test will appear. See the appropriate section for specifics on the available graph types:

- System Check Graph Settings dialog box
- Sine Graph Settings dialog box
- Random Graph Settings dialog box
- Shock Graph Settings dialog box
- Field Data Replicator Graph Settings dialog box

Several keyboard and mouse shortcuts are available to manipulate the graph display:

- Ctrl-G to edit the graph settings

- Ctrl-A to autoscale the vertical (Y) axis
- Ctrl-F to autoscale both the vertical and horizontal axes
- Ctrl-Q to autoscale all open graphs
- Ctrl-D to toggle the cursor display
- F8 to insert an annotation
- F7 to remove an annotation (click on the annotation text with the left mouse and then press F7)
- F4 to move an annotation (click on the annotation text with the left mouse, press F4, move the annotation to the new location, and click the left mouse button to select the new position)

- Click and hold down the right mouse button, move the mouse to draw a rectangle, and release the mouse button to zoom in on an area of the graph.
- Double-click the right mouse button to zoom out to show all of the data

Using the Clipboard
While a graph is displayed, it can be copied to the Windows Clipboard by clicking on the graph with the mouse and selecting the File..Copy Graph menu command, clicking the "Copy Graph" toolbar button or by pressing Ctrl+C. After copying the graph to the clipboard, switch to the application into which you want to paste the graph image and use that application's Edit..Paste menu command. You can then resize the graph to meet your needs. The Edit..Paste command will only work in applications that are able to use metafile graphics (e.g. *Microsoft Word*).

Windows also allows you to copy the active dialog box or the entire screen to the clipboard:

- To copy just the active window or dialog box, hold down the "Alt" key and press the "Print Scrn" key (usually found above the numeric keypad).
- To copy the entire screen to the clipboard, press the "Print Scrn" key. Then go to the desired application, and use that application's Edit..Paste menu command to insert the copied image.

Using the Printer
To print a graph, select the graph by clicking on it with your mouse, and then by selecting the File..Print Active Graph menu command or clicking the "Print Graph" toolbar button. A dialog box will appear prompting you for the printer to which to print the graph. Select an appropriate printer, and click the "OK" button.

You can also select the File..Printer Setup menu command or the File..Print Options menu command to modify your printer parameters.

Monochrome laser printers will produce the better graph output if you select the BLACK AND WHITE mode rather than the COLOR mode when creating a new graph. To change a graph from color to black-and-white, edit the graph settings by selecting the Graph..Edit Graph Settings menu command or clicking the "Edit Graph" toolbar button, change the style from "Color" to "Black and White" and click the "OK" button.

2.9 How to store and retrieve data

2.9.1 How to store Data

To store the data for the current test, select the File..Save As *VibrationVIEW* Data File menu command or click the  toolbar button. A dialog box will appear prompting you for a file name into which to store the data. When the "Save" button is clicked all parameters for the current test will be saved to the hard drive.

The default directory for saving data is set in the test specification. Select the Test..Edit Test menu

command and click the Data Storage tab in the Edit Test dialog box to define the default data directory for the current test.

The default file name has the form "Aug25-1617-0001" where "Aug25"indicates the date and "1617" indicates the time the test was started (i.e. August 25 at 4:17 pm). 0001 is an index value that is incremented every time a file is saved while the test is running. Therefore, for a test begun at 4:17 pm on August 25, the default names for the data files will be "Aug25-1617-0001", "Aug25-1617-0002", "Aug25-1617-0003", etc. Automatic data storage will always use this sequence of file names. When you manually store the data (using the File..Save Data menu command) you may change the file name to anything you desire before clicking on the "Save" button.

2.9.2 How to View Stored Data

To view stored data, select the File..Open Data menu command or click the "Open Stored Data" toolbar button. A dialog box will appear prompting you for the name of the file you wish to view. Select the desired file, and click the "OK" button. A graph with the stored data will be displayed. Select the Graph..Edit Graph Settings menu command (or press Ctrl-G) to change the traces and/or type of the graph.

Hint: While a stored graph is displayed, you can use the *left and right arrow keys* to scan backwards and forwards in time through all of the data files stored in the same directory as the currently displayed file. You can quickly scan through many stored data sets with this feature.

To create a formatted report of the data displayed in the current graph, select the File..Create Report menu command.

2.9.3 How to export Data to *Excel*

To store the data for the current test, select the File..Save As Text File menu command. A dialog box will appear prompting you for a file name into which to store the data. When you click the "Save" button all parameters for the current test are saved to the hard drive. This data can be directly imported into *Excel*. From *Excel* select the File..Open menu command. Change the Files of type drop-down box to display "Text Files (*.prn, *.txt, *.csv). Select a previously saved data file and click "Open".

2.10 How to use memorized drives to quickly start tests

The default behavior of the controller is to slowly ramp up the output level until the desired acceleration profile is reached. If the same test is to be run repeatedly under the same test conditions, the required output signal can be memorized and stored with the test to that the output will quickly come up to the desired level when the "Run" button is clicked.

2.10.1 Enabling memorized drive

To enable the memorized drive, start the test and wait until the output reaches the desired level. Then select the Test..Save Current Drive menu command You will be asked if you want the memorized drive to be saved to the hard drive with the current test. Click the "Yes" button (clicking the "No" button will enable memorized drive temporarily, until you exit and restart the program.)

2.10.2 Using memorized drive

Once a memorized drive is stored with a test, you will be given the option of using this drive when clicking the "Run" button to start the test. When prompted, click the "Yes" button to use the memorized drive signal, or click the "No" button to use the standard slow ramp startup procedure.

2.10.3 Disabling memorized drive

If you no longer wish to use the memorized drive stored with the current test, edit the test by selecting the

Test..Edit Test menu command and clicking the Schedule tab to edit the test schedule. The "Use memorized drive" setting for the first level controls whether or not the test will start up with the memorized drive. Clear this check box for the first schedule level and click the "OK" button to close the edit test dialog boxes and to save the test.

Note: The "Use memorized drive" option in test schedule levels 2 and above always use the output signal at the end of the previous level to determine the memorized drive signal, so these may be enabled without first selecting the Save Current Drive menu command. Save Current Drive applies *only* to the first level in the test schedule.

2.11 How to tune the controller parameters

- How to tune Sine controller parameters
- How to tune Random controller parameters
- How to tune Sine-on-Random controller parameters
- How to tune Classical Shock controller parameters
- How to tune Field Data Replicator controller parameters

2.11.1 How to tune Sine controller parameters

Click  on the toolbar.

Open the Sine Parameters dialog box.

Start with the settings:

Startup Time	10 seconds
Response Time	50 mS
Min Response Time	2 Cycles
Slew Rate	20 dB/sec
Abort Rate	40 dB/sec
Fractional Bandwidth	10%
Maximum Bandwidth	50Hz
Sample rate	Auto

The ***Startup Time*** parameter controls the approximate amount of time the controller will take to go from no output to the desired level. Having a long ***Startup Time*** (e.g. 10 to 20 seconds) is a good safety precaution because it allows the operator more time to abort the test if he sees that the shaker is moving but the input signals are not registering any acceleration (for example, when an accelerometer cable is not connected.)

The **Response Time** *and* **Min Response Time** *parameters are the primary loop tuning parameters.* The ***Response Time*** determines high frequency loop response. Shorter response times provide faster control response. Longer response times provide loop stability.

The ***Min Response Time*** parameter determines low frequency loop response. If the number of cycles specified in ***Min Response Time***, at a given frequency, is greater than the specified response time, then this parameter determines loop response.

The ***Slew Rate*** parameter determines the maximum slew rate during normal operation.

The ***Fractional Bandwidth*** parameter determines the bandwidth of the tracking filters at low frequencies. It is specified as a percentage of the test frequency.

The ***Maximum Bandwidth*** parameter determines the bandwidth of the tracking filters at high frequencies. The tracking filter bandwidth will be the smaller of the ***Fractional Bandwidth*** and the ***Maximum Bandwidth*** parameters. The transition will occur at the frequency where the ***Fractional Bandwidth*** equals the ***Maximum Bandwidth***, which will be Freq = (MaxBW) * (100%) / (FracBW)

The ***Sample Rate*** parameter generally should be set to automatic to allow the controller to select the optimum sample rate for the test. If the ***sample rate*** is set manually, it *must* be greater than 2.5 times the maximum frequency in the test specification.

To adjust the safety limits, open the Sine Limits dialog box.

Start with the settings:

Startup Max System Gain	5 V/G
Startup Output Threshold	0.005V
Startup Max Output	1V
Running Max System Gain	5 V/G
Running Max Output	1V

The ***Startup Max System Gain*** and ***Startup Max Output*** parameters are safety limits that are used to shut down the system when something goes wrong (e.g. an accelerometer cable falls off). Appropriate values will depend on your shaker amplifier gain and shaker table. To determine the proper values, perform the following steps:

1. Run through a complete sweep of the frequency range. If necessary, increase the ***Starting*** and ***Running Max Output*** and ***Starting*** and ***Running Max System Gain*** parameters to allow the sweep to complete.
2. Click the New Graph button to create a new graph of Output Drive vs. Frequency. Click the Autoscale button to scale the graph so all data is visible. Find the maximum value on the graph and set the ***Starting Max Output*** and ***Running Max Output*** parameters to 1.5 times this value.
3. Click the Edit Graph button to change the graph to System Gain vs. Frequency. Click the Autoscale button to scale the graph so all data is visible. Find the maximum value on the graph and set the ***Startup Max System Gain*** and the ***Running Max System Gain*** to 2 times this value.

The ***Startup Output Threshold*** parameter is the base output voltage when a test is starting.

The ***Running Max System Gain*** is a safety limit that is used to shut down the system when something goes wrong (e.g. an accelerometer cable falls off). This value typically is the same as the ***Startup Max System Gain*** parameter but may need to be increased if your shaker requires widely varying drive levels at different frequencies. If a test is incorrectly aborted due to a Max Run System Gain error, increase this value. The ***Running Max System Gain*** parameter can be determined by running a sweep through the desired frequency range. After completing one sweep, find the highest output voltage on a System Gain vs. Frequency graph. Enter 2 times the maximum value for the ***Running Max System Gain*** parameter.

The ***Running Max System Output*** parameter is a safety limit that is also used to shut down the system when something goes wrong. This parameter typically is double the ***Startup Max System Output*** but may need to be increased if your shaker requires widely varying drive levels at different frequencies. If a test is incorrectly aborted due to a Max Run Drive error, increase this value. The maximum output voltage required for a test can be determined by running a single sweep of the test. After completing one sweep, find the highest output voltage on the Output Drive graph (the cursor display function may be helpful in finding this value.) Enter 1.5 times the maximum value for the ***Running Max System Output*** parameter.

2.11.2 How to tune Random controller parameters

Click  on the toolbar.

Click the Parameters tab.

Start with the following values:

Starting Average	3
Starting Gain	0.1
Startup Time	15 sec
Running Average	20
Running Gain	0.05
Slew rate	20 dB/sec
Sample rate	Auto
Lines	800
Shutdown rate	60 dB/sec
Clipping	6x Sigma
Window type	Blackman

There are two sets of control parameters:

- The Starting parameters are used while the output is ramping up to the control level and generally have less averaging and a higher gain value because in this mode the system must respond quickly to equalize the response. A smaller Starting Gain will increase the time required to reach the desired output level.
- The Running parameters are used once the output has reached the desired level and generally have higher averaging and lower gain values than the corresponding Startup parameters because in this mode the system has already been equalized so only small adjustments need to be made to the control. Increasing the amount of averaging also results in a smoother spectrum.

The Gain parameters are multiplied by the error signal for each control loop. The larger the number, the faster the control will react. If the controller tends to overshoot on startup, you should reduce the ***Starting Gain*** and increase the ***Startup Time*** parameters. If the controller takes too long to get up to level, you can increase the ***Starting Gain*** and decrease the ***Startup time*** parameters. The Gain parameters should *never* be more than 1.0 and typically should be less than 0.3.

The ***Starting*** and ***Running Average*** parameters control the amount of averaging done on the frequency spectrum. The higher the number, the more averaging is done and the smoother the spectrum lines will be. However, high amounts of averaging also result in slower response, so high amounts of averaging will smooth over the time variations in the signal. The averaging values are equal to the number of frames of data averaged together. *Note:* The corresponding statistical degrees-of-freedom (DOF) value for the spectral measurements is double the averaging values entered here.

The ***Slew rate*** parameter sets the maximum rate at which the output will change, in dB/second. A slew rate of 20dB/second will allow the signal to get 10x larger within 1 second.

The ***Sample rate*** parameter in general should be set to automatic to allow the controller to select the optimum sample rate for the test. If the sample rate is set manually, it *must* be greater than 2.5 times the maximum frequency in the test specification.

The ***Lines*** parameter controls the frequency resolution of the control spectrum. The more lines you select, the higher the frequency resolution and therefore the faster the roll-off at low frequencies. When

running a test close to the displacement limits of your shaker, you should use the highest number of lines possible to better filter out the frequencies below the lowest frequency in your test specification. However, increasing the number of lines means the controller must gather more data before computing the frequency spectrum, which slows down the responsiveness of the system. If the system is responding too slowly, then it may help to decrease the number of lines used, especially for low sampling rates. Typically the number of lines should be less than the sampling rate.

The ***Sigma Clipping*** parameter sets the amount of clipping allowed on the drive voltage as a multiple of the RMS level. Typical Gaussian random noise has peaks about 4 times the RMS level, so the default level of 6 sigma does not clip the drive signal. In some cases when the peaks in the drive signal may trip an overvoltage limit on the amplifier, the sigma clipping level might be reduced to 3 or 4 to remove the highest peak voltages from the drive signal.

To tune the output drive safety limits, click on the Limits tab.

Set the ***Starting Max System Gain*** and ***Running Max System Gain*** settings to a large number (1000 V/G) to disable these safety limits. Then run the test, and wait until the control accelerometer's spectrum reaches the demand level. Read the "Volts rms" value displayed in the Random Control Center, and divide this value by the "Demand G RMS" value to determine the Volts/G ratio required by your system. Set both the ***Starting Max System Gain*** and the ***Running Max System Gain*** values to 4 times the computed V/G level required by your system. Set the ***Max Output*** value to about 1.5 times the measured "Volts rms" value.

The ***Output Threshold*** parameter is the output voltage level at which the shaker first starts to move. This value is typically 0.005 Volts for an Electro-Dynamic shaker and 0.050 Volts for a Servo-Hydraulic shaker. To determine the value for this parameter for your shaker system, start the test and note the "Volts rms" value displayed when the input spectrum first begins to increase. Enter this voltage as the Output Threshold parameter.

2.11.3 How to tune Sine-on-Random controller parameters

Click  on the toolbar.

Select the Parameters tab in the random test settings dialog box.

Start with the following values:

Response Time	1000 ms
Min Response Time	10 cycles
Slew Rate	10 dB/sec
Plus Abort Limit	9 dB
Fractional Bandwidth	5%
Maximum Bandwidth	10Hz

The ***Response Time*** and ***Min Response Time*** parameters control the speed of the response to changes in the system. Faster response times make the sine tones respond faster when sweeping through a resonance frequency, but make the sine tone more susceptible to the random background noise. The higher the number, the slower the response and therefore the more stable the control loop.

The ***Slew Rate*** parameter limits the rate of change in the sine tone output and the rate at which the sine tone starts up when the test starts. Reduce this value if the sine tone overshoots the desired level at test startup. Typically 10dB/second works well.

The ***Fractional Bandwidth*** and ***Maximum Bandwidth*** parameters set the bandwidth of the tracking filters used to extract the sine tones from the random background. The bandwidth used is the smaller of the two values, so the fractional bandwidth applies to lower frequencies, and the maximum bandwidth applies to higher frequencies. The crossover between the two occurs where the fractional bandwidth is equal to the maximum bandwidth, which will be at Freq = (MaxBW) * (100%) / (FracBW). Narrower bandwidths filter out more of the random background, but also make the sine readings less responsive to rapid changes because narrower bandwidths require more measurement time.

2.11.4 How to tune Classical Shock controller parameters

Click  on the toolbar.

Click the Parameters tab and enter the following values:

Feedback Gain	0.3
Min Frequency	0
Max Frequency	500 Hz
Low Pass Filter	Disabled
Startup Pulse Interval	1
System Lag	0
Sample Rate	Automatic

The ***Min Frequency*** parameter usually should be set to 0 to control the entire low frequency bandwidth of the system. In certain special circumstances, advanced users may use this parameter to reduce the low frequency components of the signal (typically when using a user-defined pulse type).

The ***Max Frequency*** parameter should be set to the bandwidth of your shaker system, or to the frequency at which the demand spectrum drops below about 4 times the noise level. The controller will only output and control frequencies below this value. As a rule of thumb, a maximum frequency level of 5000/PulseWidth, with the PulseWidth measured in milliseconds, is needed to faithfully reproduce a pulse. For example, a 7 millisecond wide pulse will require a maximum frequency of at least (5000/*T*) = 714 Hz.

The ***Low Pass Filter*** parameter allows the application of a digital low pass filter to the input waveforms. In general this filter should be disabled, but in certain cases where there is significant high frequency noise in the measurements, this noise can be removed by enabling the low pass filter. When enabled, all frequencies above the set cutoff frequency will be filtered out of the displayed signal.

The ***Startup Pulse Interval*** parameter controls the time between pulses when the test is starting and the controller is equalizing the output. One second between pulses is generally a good value.

The ***System Lag*** parameter is used to compensate for the time delay characteristic of your shaker system. A value of 0 will always work, but if your shaker system has significant lag, you may want to adjust this setting. One way to determine the lag in the system is to click the "Run" button and then the "Stop" button. The controller will output a single pulse. Compare the time delay between the demand pulse and the input pulse to see the lag in your system.

The ***Sample Rate*** parameter generally should always be set to Automatic to have the controller automatically select a suitable sample rate. Advanced users may switch this to manual mode to force the system to use a specific sample rate.

Click the "OK" button to save the test and run the test. Once the pulse reaches the demand level, take note of the peak output voltage (the Output volts peak value in the Shock Control Center) and the peak

demand level. Edit the test again (click the edit test toolbar) and select the Limits tab. On that tab, enter the following values:

Starting drive limits	
Max output	1.25 * OutputVoltsPeak
Max system gain	4 * OutputVoltsPeak/DemandPeak

Running drive limits	
Max output	1.5 * OutputVoltsPeak
Max system gain	1.5 * OutputVoltsPeak/DemandPeak

The ***Starting Max Output*** parameter typically should be more conservative (smaller than) the ***Running Max Output*** parameter but the ***Starting Max System Gain*** parameter should typically be higher than the ***Running Max System Gain*** parameter. This is because, during startup, the output is increasing and the ***Starting Max System Gain*** parameter will limit the rate at which the output can increase. For the settings above the output can increase by a maximum of a factor of 4 for each equalization pulse.

To finish editing the test and save it to the hard drive, click the "OK" button.

Using a memorized drive signal

Note: After a satisfactory pulse has been achieved, the Test..Save Current Drive menu command will memorize the drive signal. After memorizing the drive signal, the test will begin with the memorized drive and will not need to equalize the drive signal. This will make the controller converge to the desired shape typically on the first pulse. Note however, that if the test conditions change (the shaker amplifier's gain changes, or the load on the shaker changes) the memorized drive signal will no longer be accurate.

2.11.5 How to tune Field Data Replicator controller parameters

Click  on the toolbar.

Open the Field Data Replicator Parameters dialog box.

Start with the following values:

Lines:	800
Window Type:	Blackman
Startup Rate:	10 dB/sec
Abort Rate:	40 dB/sec
Spectrum Response time:	5 sec
Spectrum Control SNR:	6 dB
Phase Response time:	20 sec
Phase Control SNR:	20 dB
Startup Averaging	3 frames
Running Averaging:	10 frames

The ***Lines*** parameter defines the frequency resolution of the control loop. Increasing the number of lines allows better control of low frequencies and of sharp resonances but makes the controller start and respond more slowly to changing system dynamics.

The ***Window Type*** parameter defines the type of window function to use when computing the FFT spectrum. Different window types give different tradeoffs between frequency resolution and spreading of energy from one frequency to another. The Blackman window type works well in most cases.

The ***Startup Rate*** parameter defines the rate, in dB/second, at which the output comes up when starting a test. Increasing this rate will make the test start faster but if the test starts too quickly you may get some overshoot.

The ***Abort Rate*** parameter defines the rate, in dB/second, at which the output turns off when the test is stopped. This gives a smooth transition between running and stopping. Typically 40dB/second is a good choice for the abort rate.

The ***Spectrum Response Time*** parameter defines the responsiveness of the spectrum control loop. The spectrum control loop measures the amplitude of the frequency response function of the shaker/amplifier combination, and uses this to adjust the frequency weighting of the drive signal. Decreasing this response time makes the control loop respond faster to changes in the system, but may make the control loop less stable. Increasing this time makes the control loop respond slower to changes, which will make the control loop more stable and less susceptible to noisy measurements.

The ***Spectrum Control SNR*** parameter defines the minimum signal-to-noise ratio required for controlling the spectrum. If the desired signal is lower than this ratio above the input noise level, the control loop is suspended until the desired signal comes back. This prevents control loop updates when the measurements are very noisy relative to the demand signal.

The ***Phase response time*** parameter defines the responsiveness of the phase (time) control loop. The phase control loop measures the time delays between the demand and control waveforms, and adjusts the time delay (phase) of the output signal such that the demand and control overlap in time. Decreasing this response time makes the control loop respond faster to changes in the system, but may make the control loop less stable. Increasing this time makes the phase control response slower to changes, which will make the control loop more stable and less susceptible to noisy measurements.

The ***Phase Control SNR*** parameter defines the minimum signal-to-noise ratio required for controlling the phases. If the desired signal is lower than this ratio above the input noise level, then the control loop is suspended until the desired signal comes back. Phase readings are quite susceptible to noise, so the phase control SNR should typically be 10dB or greater.

The ***Startup Averaging*** and ***Running Averaging*** parameters define the amount of averaging (smoothing) to use on the frequency spectrum display. This number is the number of spectrum frames to average together. To get the DOF parameter, multiply this value by 2.

If the output increases too slowly at startup, increase the ***Startup Rate*** and decrease the ***Spectrum Response Time*** values. If the control overshoots the desired level, then decrease the ***Startup Rate*** parameter.

2.12 How to import and export Field Data Replicator waveforms

Waveform data for Field Data Replicator tests can be imported and exported using the following methods:

- Record a waveform from the input of the VR8500
- Import a waveform from a text (ASCII) file
- Import a waveform from a Windows .WAV file
- Export a waveform to a Windows .WAV file

2.12.1 How to record a waveform from the input of the VR8500

Analog recording is currently supported from any channel with the VR8500 hardware. The record option must be enabled in the software key.

1. Change to the Field Data Replicator test mode by selecting the Test..Field Data Replicator menu command.

2. Use Test..Define new test menu command to create a new field data replicator test.
3. The new test wizard displays the Playback tab with a "Record Playback File" pushbutton. Press this pushbutton to activate the record dialog box.
4. Select your sample rate and channel to record.
5. Start recording by pressing the "Next >" wizard button.
6. Press the "Next >" wizard button again when your recording is complete. Select a descriptive filename for the saved recording.
7. The new Field Data Replicator test is now initialized to replay this newly recorded file.

2.12.2 How to create a Field Data Replicator waveform from a text file

To convert an ASCII format file to the *VibrationVIEW* internal format (VFW format) file, use the following procedure:

1. Change to the Field Data Replicator test mode by selecting the Test..Field Data Replicator menu command.
2. Load the desired test, or define a new test.
3. Click the "Edit Test" button on the Toolbar
4. Select the Import tab in the Field Data Replicator Define dialog box
5. Choose the file that you want to read in the Input file field
6. Choose the file into which you wish to store the VFW waveform in the Output file field. To keep your files organized, we recommend that you put the output in the C:\Program Files*VibrationVIEW*\Drive, but any directory on the hard disk may be used.
7. Set the number of Header Lines to the number of lines to discard at the beginning of the file. This is used, for example, if the first few lines in the file contain a description of the data. If the data begins on the first line of the file, use 0 for the number of header lines.
8. If more than one value is shown per line, separated by commas, semicolons, or spaces, set the column number to select desired column of data. For example, some files have time information in the first column, X accel data in the second, Y accel data in the third, and Z accel data in the fourth. In this case you would set the column value to 3 if you wanted to import the Y accel data.
9. Click the "Rescan File" button, and verify that the header and body are properly separated (i.e. that the body section shows only the data, and that all of the header information is in the header section.)
10. Set the Sampling rate parameter to the sampling rate used when reading the data from the input file.
11. Click the "Convert File" button to begin converting the file. The time required for conversion will depend on the size of the file. A dialog box will show the progress of the conversion process.
12. Click the "OK" button in the dialog box that tells you that the conversion was successfully completed.
13. Click the "Exit" button to close the ASCII -> VFW dialog box. The new playback file, record level and sample rate will be automatically entered in the Field Data Replicator Define dialog box.
14. Click the "OK" button to close the Define dialog box and save the test to the hard drive.

2.12.3 How to create a Field Data Replicator waveform from a Windows .WAV file

To convert from WAV to VFW use the following procedure:

1. Change to the Field Data Replicator test mode by selecting the Test..Field Data Replicator menu command.
2. Load the desired test, or define a new test.
3. Click the "Edit Test" button on the Toolbar
4. Click the "Edit Playback File" button in the Field Data Replicator Playback dialog box.

This will start CoolEdit.

5. Select File..Open.from Cool Edit menu. Set Files of Type to "ACM Waveform (*.wav). Select your file and press "Open"
6. Select File.. Save As from Cool Edit menu. Set Files of Type to "VibVIEW Waveform (*.vfw). Select your Output file (recommended directory is C:\Program Files*VibrationVIEW*4\Drive.
7. Select the new playback file in the Field Data Replicator Playback dialog box.
8. Click the "OK" button to close the Define dialog box and save the test to the hard drive.
9.

2.12.4 How to export a Field Data Replicator waveform to a Windows .WAV file

To convert from FDR to WAV, use the following procedure:

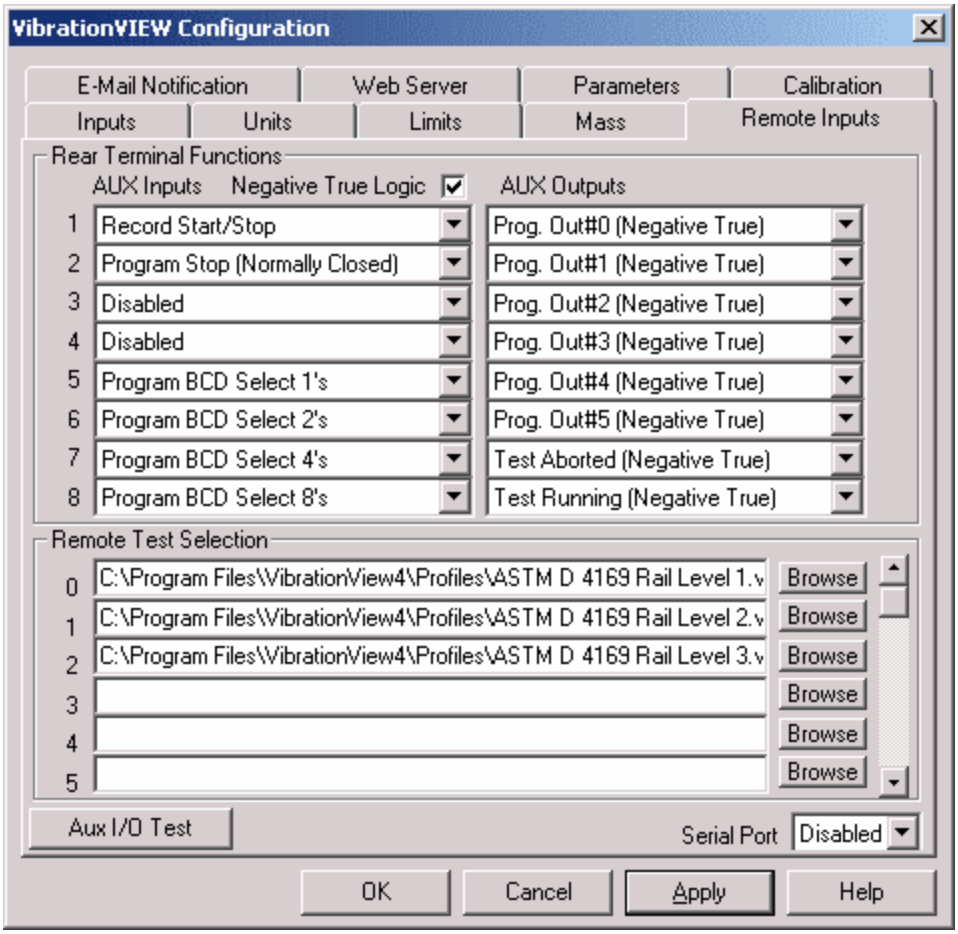
1. Change to the Field Data Replicator test mode by selecting the Test..Field Data Replicator menu command.
2. Load the desired test
3. Click the "Edit Test" button on the Toolbar
4. Click the "FDR -> WAV" button in the Field Data Replicator Playback dialog box
5. Click the "Edit Playback File" button. The will open Cool Edit and load current waveform.
6. Select File.. Save As from Cool Edit menu. Set Files of Type to "ACM Waveform (*.wav). Select your Output file, and click "Save" button.

2.13 How to integrate *VibrationVIEW* into remote applications

- Use the Remote Input Options
- Use ActiveX
- Use ASCII File

2.13.1 How to use Remote Inputs to control VibrationVIEW

This tab is accessed by selecting the Configuration..Remote Inputs menu command.



Use this tab to enable and configure the remote input option.

Note: Remote Inputs is an optional feature; the software key must be programmed to enable it If you haven't purchase the remote input option, you may ignore this configuration tab.

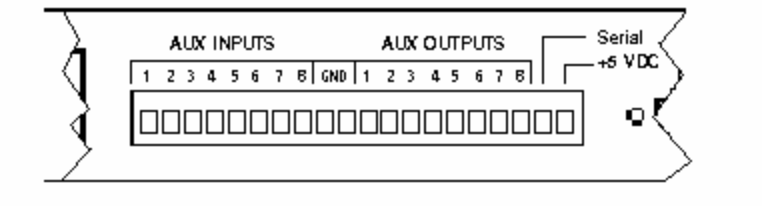


Figure: Input terminals on the rear of the VibrationVIEW IO box.
Each input can be assigned to any function. If multiple inputs are defined to the same function, all inputs must be true to trigger the function.
Available Input Functions

- Disabled No input funtions
- Program Start/Stop Start the program when input transitions true, Stop program when input transitions false.

- Program Stop Stop the program whenever the input is false.
- Program Start Start the program whenever the input is true.
- Resume Continue a program at step calling for operator acknowledge.
- Program BCD Select 1's
- Program BCD Select 2's
- Program BCD Select 4's
- Program BCD Select 8's BCD Select combine to form a Remote Test Selection.

Inputs can be forced to use Negative True logic. Negative true logic is inverted logic where the input is held LOW during the true state. Inputs are either all positive logic or all negative logic.

Available Output Functions

- Prog. Out#0
- Prog. Out#1
- Prog. Out#2
- Prog. Out#3
- Prog. Out#4
- Prog. Out#5 Program Out are selected from Test schedule tab.
- Test Aborted Indicates Test has aborted
- Test Running Indicates Test is running

Outputs can be individually selected for Negative True logic. Each individual output selected as Negative True will be held low to indicate a true state, and held high to indicate false state

Remote Test Selection (BCD Select)

When *any* input is assigned to a BCD Select, a test is automatically loaded when the remote input signal is used to start a test. This may be used to select different tests using remote electronics such as a PLC. Remote test selection is performed using discrete inputs assigned to Program BCD Select according to the following table, where a 0 and 1 indicate TTL voltage levels of 0 volts and 5 volts, respectively (threshold level is 2.2 volts). For each test number listed in the table below, there is a corresponding line in the Remote Input configuration dialog box configuring the name of the test to load.

BCD 8's	BCD 4's	BCD 2's	BCD 1's	Test number
0	0	0	0	0
0	0	0	1	1
0	0	1	0	2
0	0	1	1	3
0	1	0	0	4
0	1	0	1	5
0	1	1	0	6
0	1	1	1	7
1	0	0	0	8
1	0	0	1	9
1	0	1	0	10
1	0	1	1	11
1	1	0	0	12
1	1	0	1	13
1	1	1	0	14
1	1	1	1	15

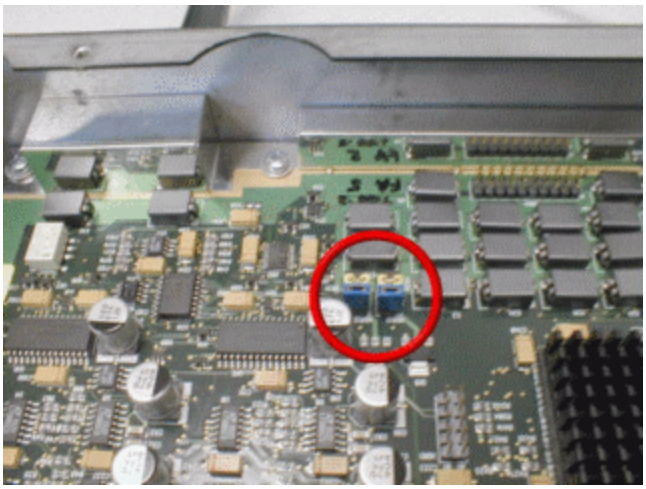
TTL Discrete Output Specifications

Discrete Output source (high) capability of 32mA
Discrete Output sink (low) capability of 64mA

Discrete Output HIGH voltage:
2.5V minimum at 3mA source current
2.0V minimum at 32mA source current

Output LOW voltage:
0.55V maximum at 64mA sink current

The outputs in TTL mode have enough current drive to sink a typical 5V SSR (connect +5V to SSR high input, SSR low input to VR8500 discrete output).



Jumpers HDR12 and HDR13 (shown in TTL output position)

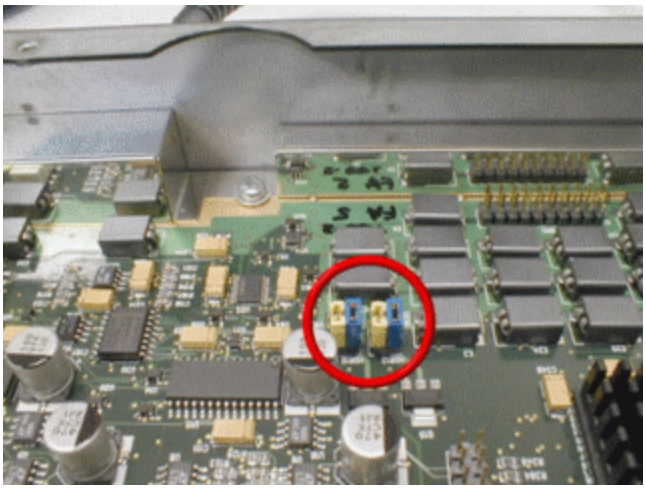
High Current Open Collector outputs

Discrete outputs 7 and 8 (7=test aborted, 8=test active) have jumpers on the circuit board which allow them to be changed to open collector outputs which can sink high currents (up to 1.5A). When configured in this way the outputs are inverted, so they sink the output to ground when active
.

Open Collector Output Specifications

Output "HIGH" (active):
Sink up to 1.5 amps to ground

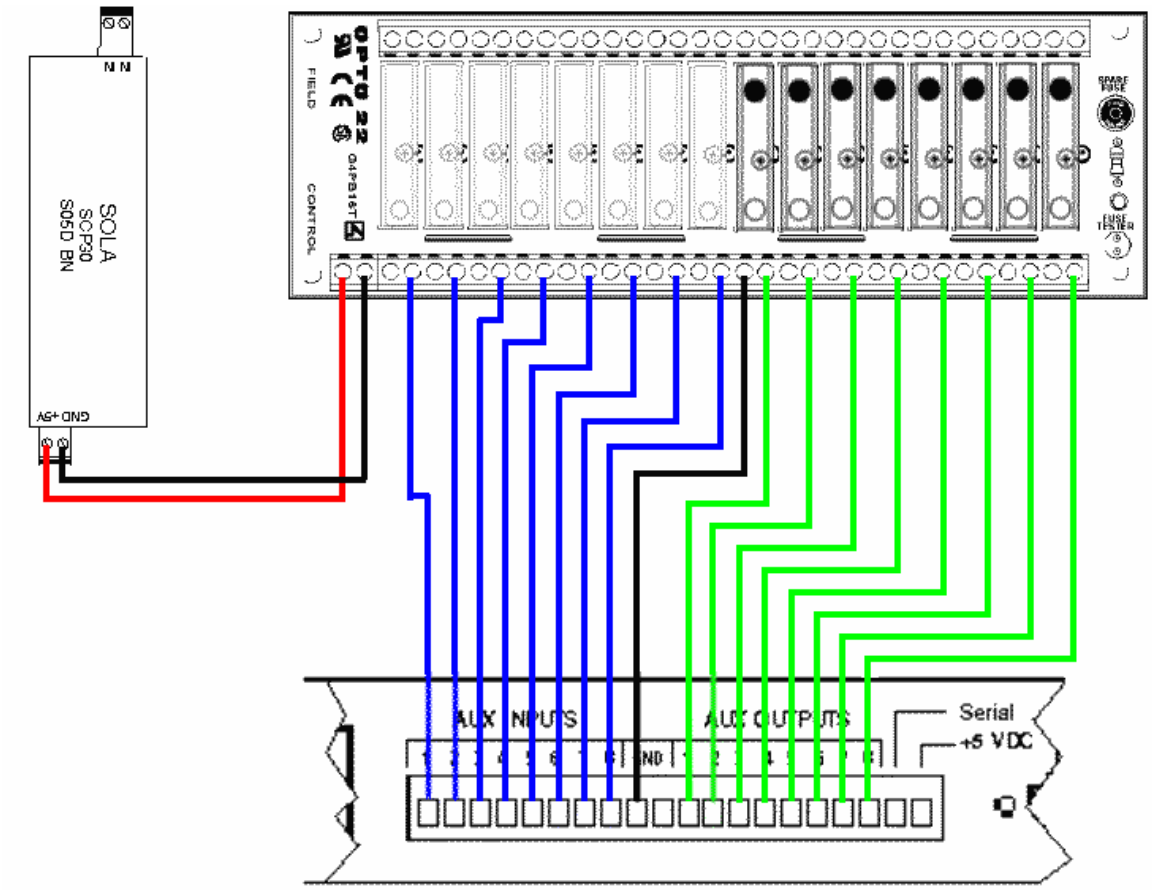
Output "LOW" (inactive):
Floating output, voltage must be between ground and 70 VDC



Jumpers HDR12 and HDR13 (shown in Open Collector output position)

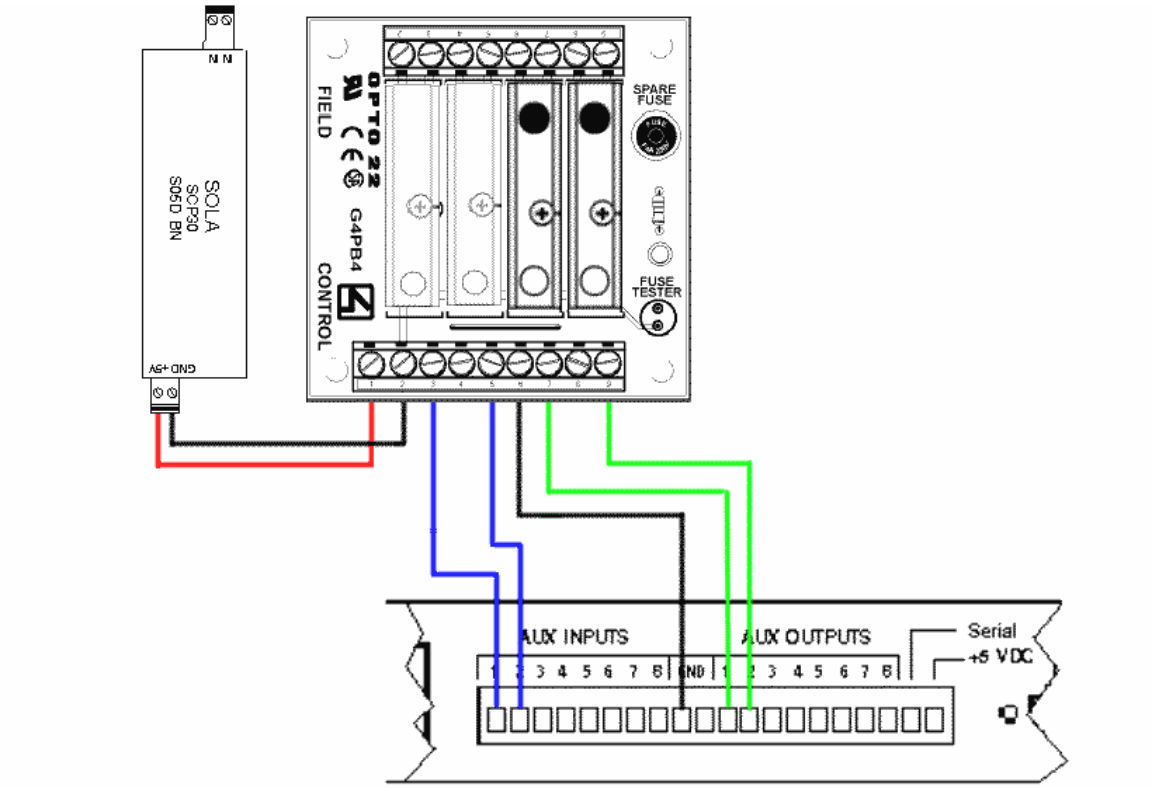
NOTE: ONLY on outputs 7 and 8 can be configured for Open Collector, and ONLY when the hardware is configured for Open Collector outputs. Failure to properly set the jumpers before using the outputs can result in damage to the hardware.

***Example* Remote IO Wiring**

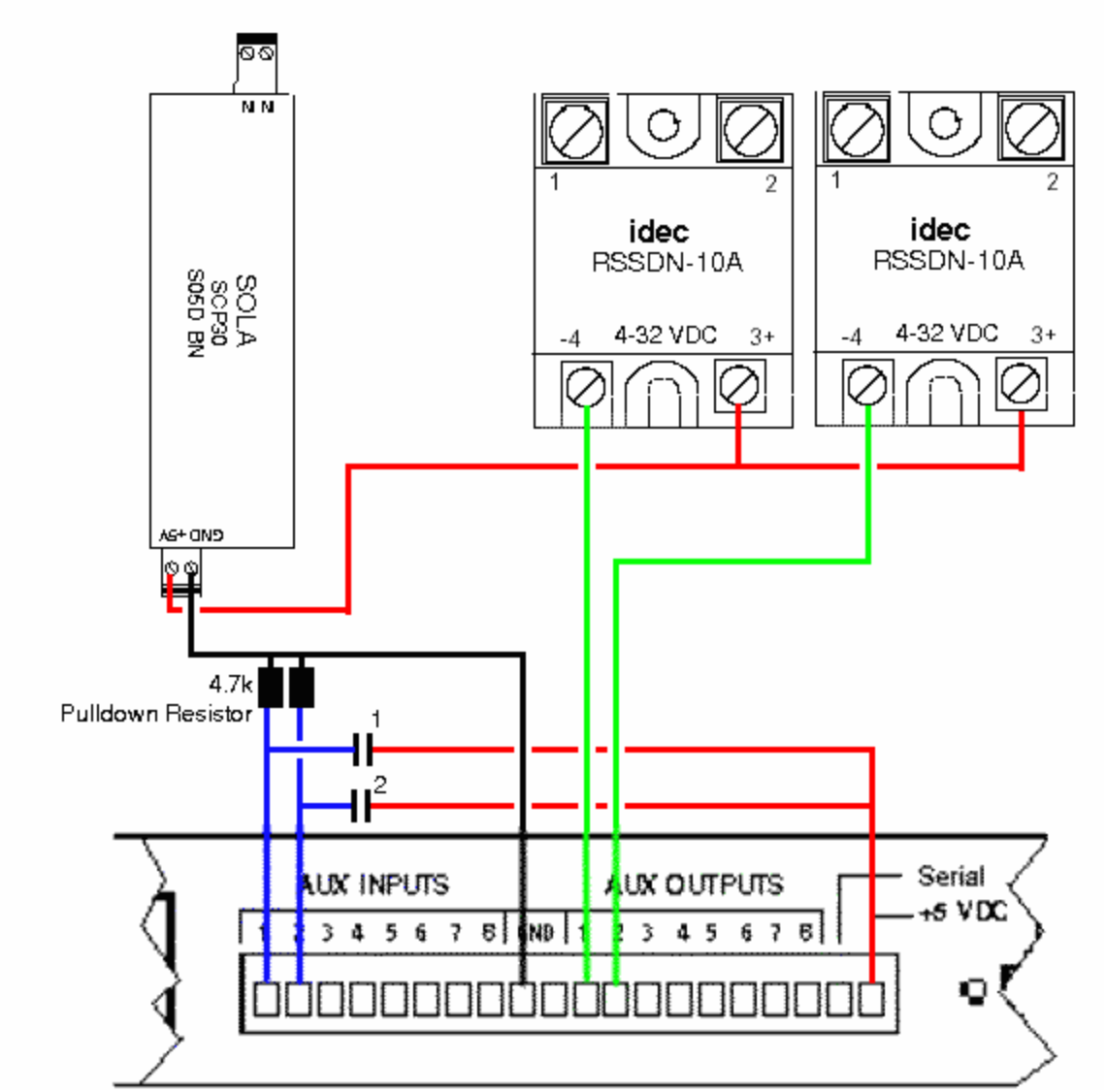


An *Opto 22* G4PB16T provides isolation for all 8 inputs and outputs. G4 modules can be plugged into the

rack to interface AC or DC field circuits.



An *Opto 22* G4PB4 provides isolation for any 4 inputs and/or outputs. G4 modules can be plugged into the rack to interface AC or DC field circuits.



Solid State Relays provide output isolation, and capability of switching 10Amps high voltage AC.

Inputs shown using internal 5 Volt supply,
Note: pulldown resistors are required.

2.13.2 How to use ActiveX to control VibrationVIEW

Remote control requires the optional ActiveX component of *VibrationVIEW*.

Manually registering the *VibrationVIEW* Type Library

The type libraries are contained in a binary file that contains all type information needed to utilize *VibrationVIEW ActiveX* control.
To register the type library, you can use **regtlib.exe** as follows:
Go to **Start » Run**.

Type the following in the dialog box that appears:
regtlib "C:\Program Files*VibrationVIEW*4*VibrationVIEW*.tlb"

Using the ActiveX interface to VibrationVIEW

There are two interfaces to *VibrationVIEW*. Each must be created separately, if both data retrieval and control functionality are required.

VibrationVIEWLib.IVibrationVIEWControl

AbortEdit	Abort edit previously started with Edit Test
Activate	Activate *VibrationVIEW*
EditTest <filename>	Edit specified test file
GetErrorString eString,ecode	Get error string related to errorcode (Not supported from VBA)
Maximize	Maximize *VibrationVIEW* window
Minimize	Minimize *VibrationVIEW* window
Restore	Restore *VibrationVIEW* window
Run Test <filename>	Load and run test <filename>
SaveData <filename>	Save *VibrationVIEW* data file <filename>
StartTest	Start currently loaded test
Status nStringSize,sWideString,nStopCode	Get the current stopcode
StopTest	Stop running test

VibrationVIEWLib.IVibrationVIEWData

AccelUnit sUnit,nConverions	Get accelerometer unit and conversion factor to m/s^2
Channel nCount, nArray(0)	Get the channel values
ChannelLabel sLabel,ChannelNum	Get the channel unit label associated with ChannelNum
ChannelUnit sUnit,ChannelNum	Get the channel unit associated with ChannelNum
Control nCount, nArray(0)	Get the control values for each loop
ControlLabel sLabel,LoopNum	Get the control unit label associated with sLabel,LoopNum
ControlUnit sLabel,LoopNum	Get the channel unit associated with LoopNum
Demand nCount, nArray(0)	Get the demand values for each loop
DispUnit sUnit,nConverions	Get accelerometer unit and conversion factor to m
GetErrorString	Get error string related to errorcode (Not supported from VBA)
Output nCount, nArray(0)	Get the output values for each loop
Vector	Not Supported
VectorLabel	Not Supported
VectorUnit	Not Supported
VelUnit	Not Supported

See also:
Excel example spreadsheet on distribution CDROM \Examples\ActiveX

2.13.3 How to use an ASCII file to control VibrationVIEW

Remote control requires the optional ActiveX component of *VibrationVIEW*.

To control *VibrationVIEW* with an ASCII file:

1. Edit the "C:\Program Files\VibrationVIEW4\shaker.ini" file
2. Locate the "[System Parameters]" section in the file.
3. Add this line: "Remote Control File=C:\Program Files\VibrationVIEW4\RemoteControl.txt".
4. Start *VibrationVIEW*.

VibrationVIEW will now monitor the file "C:\Program Files\VibrationVIEW4\RemoteControl.txt".
VibrationVIEW will report status in a new file "C:\Program Files*VibrationVIEW*4\RemoteControl.Status".
To remotely control *VibrationVIEW* you can write the following commands to this RemoteControl.txt.

load <FileName>	This command will load test profile specified with fully qualified path <FileName>
run	Run the currently loaded test
mat <FileName>	Save the data to file <FileName> as a *Matlab* file
csv <FileName>	Save the data to file <FileName> as a comma delimited file
save <FileName>	Save the data to file <FileName> as a *VibrationVIEW* file
stop	Stop the currently running test.
status	Write the current StopCode to RemoteControl.Status

See also:

Visual Basic (version 5) example code on distribution CDROM \Examples\VBTest

2.14 How to create customized reports

Reports are generated using mail merge type processing. The program reads in a template file, substitutes data values and graphs in place of keywords, and writes the resulting data to an output file. The template files can be either plain text or Rich Text Format files. Plain text files can have data values but no graphs. Rich Text Format files (using extension .rtf) can contain data values and graphs, as well as any text formatting and other graphics elements that can be inserted into an RTF file.

The default templates are stored in subdirectories of the directory c:\Program Files*VibrationVIEW*\Templates. Sine templates are stored in the Sine subdirectory, Random templates in the Random subdirectory, etc. To create custom reports, new templates can be created using a RTF editor file and stored in the appropriate directory. One suitable RTF file editor is WordPad, a part of Windows 95/98/NT that is usually installed under Start..Programs..Accessories..WordPad. If this program is not installed on your computer, it can be added by selecting it in the "Add/Remove Programs" control panel, Windows Setup tab, listed under "Accessories". Most Windows Word-Processor programs will also read and write RTF files.

Data Values

Data values can be placed in the file using a parameter field [PARAM:name] where the square-brackets indicate the beginning and ending of the field, and the first 6 characters of the field are PARAM: in all uppercase letters. The "name" selects which parameter to insert. The available parameters depend on the type of test. Refer to the following tables for lists of parameters that are valid for each type of test:

- Sine data values
- Random data values
- Shock data values
- Field Data Replicator data values

TEDS Data Values

TEDS data is available in the reports whenever TEDS accelerometers are used. Teds data can be placed in the file using a parameter field [TEDS:channel,name] where the where the square-brackets indicate the beginning and ending of the field, and the first 5 characters of the field are TEDS: in all uppercase letters. The "channel" selects the input channel to read. The "name" selects which parameter to insert; the available parameters depend on the model of accelerometer.

Graphs

Graphs can be placed in the file using a graph field [GRAPH:type,flag1,flag2] where the square brackets indicate the beginning and ending of the field, and the first 6 characters of the field are GRAPH: in all uppercase letters. The "type" selects which type of graph to insert, and the flag values select style of plot and which traces to display on the plot. Refer to the following tables for lists of graph types and flags that are valid for each type of test:

- Sine graph settings
- Random graph settings
- Shock graph settings
- Field Data Replicator graph settings

HTML Reports

HTML reports are used with the Optional WebServer module. Vibration Research has included a full set of HTML reports to provide remote data access and test control. HTML files are always stored in the C:\Program Files*VibrationVIEW*4\templates\<test type>\html directory (where <test type> is Data Replay, Random, Shock, Sine, or System Check.

HTML reports can be generated with any HTML editor. The data fields can be inserted at any point on the page by inserting [PARAM:name] where the square-brackets indicate the beginning and ending of the field, and the first 6 characters of the field are PARAM: in all uppercase letters. Graphs can also be placed in the file using a graph field [GRAPH:type,flag1,flag2] where the square brackets indicate the beginning and ending of the field, and the first 6 characters of the field are GRAPH: in all uppercase letters.

2.14.1 Sine report parameter names

Values defining the test profile and controller parameters:

Plus Abort:	[PARAM:+Abort]
Minus Abort:	[PARAM:-Abort]
Plus Tolerance:	[PARAM:+Tol]
Minus Tolerance:	[PARAM:-Tol]
Data file name	[PARAM:Filename]
Profile name:	[PARAM:Testname]
Sample rate:	[PARAM:SampleFrequency]
Startup time:	[PARAM:StartupTime]
Startup Threshold Voltage:	[PARAM:ThresholdVoltage]
Startup gain limit:	[PARAM:StartSystemGainLimit]
Startup max output:	[PARAM:StartMaxOutputVoltage]
Response Time:	[PARAM:ResponseTime]
Response Cycles:	[PARAM:ResponseCycles]
Running Slew Rate:	[PARAM:RunRate]
Run gain limit:	[PARAM:RunSystemGainLimit]
Run max output:	[PARAM:RunMaxOutputVoltage]
Tracking filter Fractional BW:	[PARAM:TrackBWFractional]
Tracking filter Fixed BW):	[PARAM:TrackBWFixed]
Ch1,2 Tracking filter:	[PARAM:TrackCh12]
Sweep type:	[PARAM:Sweep]
Step test setting:	[PARAM:StepTest]
Step frequency:	[PARAM:StepFrequency]
Step on time:	[PARAM:StepOnTime]
Step off time:	[PARAM:StepOffTime]
Resonance control gain:	[PARAM:ResGain]
Resonance max step:	[PARAM:ResMaxStep]
Resonance max drift:	[PARAM:ResRange]
Control channels:	[PARAM:Channels]
Channel combination method:	[PARAM:Combine]
Extremal mode enabled:	[PARAM:Extremal]
Extremal acceleration limits:	[PARAM:MaxA1] [PARAM:MaxA2]

	[PARAM:MaxA3]
	[PARAM:MaxA4]
	[PARAM:MaxA5]
	[PARAM:MaxA6]
	[PARAM:MaxA7]
	[PARAM:MaxA8]
	[PARAM:MaxA9]
	[PARAM:MaxA10]
	[PARAM:MaxA11]
	[PARAM:MaxA12]
	[PARAM:MaxA13]
	[PARAM:MaxA14]
	[PARAM:MaxA15]
	[PARAM:MaxA16]
Extremal velocity limits:	[PARAM:MaxV1]
	[PARAM:MaxV2]
	[PARAM:MaxV3]
	[PARAM:MaxV4]
	[PARAM:MaxV5]
	[PARAM:MaxV6]
	[PARAM:MaxV7]
	[PARAM:MaxV8]
	[PARAM:MaxV9]
	[PARAM:MaxV10]
	[PARAM:MaxV11]
	[PARAM:MaxV12]
	[PARAM:MaxV13]
	[PARAM:MaxV14]
	[PARAM:MaxV15]
	[PARAM:MaxV16]
Extremal displacement limits:	[PARAM:MaxD1]
	[PARAM:MaxD2]
	[PARAM:MaxD3]
	[PARAM:MaxD4]
	[PARAM:MaxD5]
	[PARAM:MaxD6]
	[PARAM:MaxD7]
	[PARAM:MaxD8]
	[PARAM:MaxD9]
	[PARAM:MaxD10]
	[PARAM:MaxD11]
	[PARAM:MaxD12]
	[PARAM:MaxD13]
	[PARAM:MaxD14]
	[PARAM:MaxD15]
	[PARAM:MaxD16]
Control averaging:	[PARAM:CtrlAveraging]
Channel aborts:	[PARAM:Ch1Abort]
	[PARAM:Ch2Abort]
	[PARAM:Ch3Abort]
	[PARAM:Ch4Abort]
	[PARAM:Ch5Abort]
	[PARAM:Ch6Abort]
	[PARAM:Ch7Abort]
	[PARAM:Ch8Abort]
	[PARAM:Ch9Abort]

	[PARAM:Ch10Abort]
	[PARAM:Ch11Abort]
	[PARAM:Ch12Abort]
	[PARAM:Ch13Abort]
	[PARAM:Ch14Abort]
	[PARAM:Ch15Abort]
	[PARAM:Ch16Abort]
Channel moment arms:	[PARAM:Ch1MomentArm]
	[PARAM:Ch2MomentArm]
	[PARAM:Ch3MomentArm]
	[PARAM:Ch4MomentArm]
	[PARAM:Ch5MomentArm]
	[PARAM:Ch6MomentArm]
	[PARAM:Ch7MomentArm]
	[PARAM:Ch8MomentArm]
	[PARAM:Ch9MomentArm]
	[PARAM:Ch10MomentArm]
	[PARAM:Ch11MomentArm]
	[PARAM:Ch12MomentArm]
	[PARAM:Ch13MomentArm]
	[PARAM:Ch14MomentArm]
	[PARAM:Ch15MomentArm]
	[PARAM:Ch16MomentArm]
Annotation line 1:	[PARAM:Note1]
Annotation line 2:	[PARAM:Note2]
Annotation line 3:	[PARAM:Note3]
Data directory:	[PARAM:DataDirectory]
Data storage settings:	[PARAM:DataStorage]
Segment table:	[PARAM:SegmentTable] (table format: Start Frequency, Start Amplitude, End Frequency, End Amplitude)
Schedule table:	[PARAM:ScheduleTable] (table format: Duration, Level)
Schedule table with dig outputs:	[PARAM:DigitalOutputTable] (table format: Duration, Level, DO0, DO1, DO2, DO3, DO4, DO5)

Values giving the test results:

Stop code:	[PARAM:StopCode]
Start time:	[PARAM:StartTime]
Current time:	[PARAM:Time]
Current level:	[PARAM:CurrentLevel]
Current modifier:	[PARAM:Modifier]
Time on current level:	[PARAM:LevelTime]
Time running this test:	[PARAM:RunTime]
Elapsed:	[PARAM:Elapsed]
Remaining:	[PARAM:Remaining]
Current Frequency:	[PARAM:Frequency]
Control acceleration:	[PARAM:CtrlAcceleration]
Control velocity:	[PARAM:CtrlVelocity]
Control displacement:	[PARAM:CtrlDisplacement]
Current demand level:	[PARAM:Demand]
Channel acceleration:	[PARAM:Ch1Accel]
	[PARAM:Ch2Accel]
	[PARAM:Ch3Accel]

	[PARAM:Ch4Accel]
	[PARAM:Ch5Accel]
	[PARAM:Ch6Accel]
	[PARAM:Ch7Accel]
	[PARAM:Ch8Accel]
	[PARAM:Ch9Accel]
	[PARAM:Ch10Accel]
	[PARAM:Ch11Accel]
	[PARAM:Ch12Accel]
	[PARAM:Ch13Accel]
	[PARAM:Ch14Accel]
	[PARAM:Ch15Accel]
	[PARAM:Ch16Accel]
Channel velocity:	[PARAM:Ch1Vel]
	[PARAM:Ch2Vel]
	[PARAM:Ch3Vel]
	[PARAM:Ch4Vel]
	[PARAM:Ch5Vel]
	[PARAM:Ch6Vel]
	[PARAM:Ch7Vel]
	[PARAM:Ch8Vel]
	[PARAM:Ch9Vel]
	[PARAM:Ch10Vel]
	[PARAM:Ch11Vel]
	[PARAM:Ch12Vel]
	[PARAM:Ch13Vel]
	[PARAM:Ch14Vel]
	[PARAM:Ch15Vel]
	[PARAM:Ch16Vel]
Channel displacement:	[PARAM:Ch1Disp]
	[PARAM:Ch2Disp]
	[PARAM:Ch3Disp]
	[PARAM:Ch4Disp]
	[PARAM:Ch5Disp]
	[PARAM:Ch6Disp]
	[PARAM:Ch7Disp]
	[PARAM:Ch8Disp]
	[PARAM:Ch9Disp]
	[PARAM:Ch10Disp]
	[PARAM:Ch11Disp]
	[PARAM:Ch12Disp]
	[PARAM:Ch13Disp]
	[PARAM:Ch14Disp]
	[PARAM:Ch15Disp]
	[PARAM:Ch16Disp]
Channel 1-to-2 phase:	[PARAM:Ch2Phase]
Schedule loop counter:	[PARAM:LoopCount]
Current drive output:	[PARAM:OutputVoltage]
Current system gain:	[PARAM:SystemGain]
Peak system gain:	[PARAM:PeakSystemGain]

Unit names:

Acceleration units:	[PARAM:UnitAcceleration]
Velocity units:	[PARAM:UnitVelocity]
Displacement units:	[PARAM:UnitDisplacement]

Alternate units:	[PARAM:Unit1] [PARAM:Unit2] [PARAM:Unit3] [PARAM:Unit4] [PARAM:Unit5] [PARAM:Unit6] [PARAM:Unit7] [PARAM:Unit8] [PARAM:Unit9] [PARAM:Unit10] [PARAM:Unit11] [PARAM:Unit12] [PARAM:Unit13] [PARAM:Unit14] [PARAM:Unit15] [PARAM:Unit16]

Accelerometer calibration details:

Channel name:	[PARAM:Ch1Name] [PARAM:Ch2Name] [PARAM:Ch3Name] [PARAM:Ch4Name] [PARAM:Ch5Name] [PARAM:Ch6Name] [PARAM:Ch7Name] [PARAM:Ch8Name] [PARAM:Ch9Name] [PARAM:Ch10Name] [PARAM:Ch11Name] [PARAM:Ch12Name] [PARAM:Ch13Name] [PARAM:Ch14Name] [PARAM:Ch15Name] [PARAM:Ch16Name]
mV/G sensitivity:	[PARAM:mVg1] [PARAM:mVg2] [PARAM:mVg3] [PARAM:mVg4] [PARAM:mVg5] [PARAM:mVg6] [PARAM:mVg7] [PARAM:mVg8] [PARAM:mVg9] [PARAM:mVg10] [PARAM:mVg11] [PARAM:mVg12] [PARAM:mVg13] [PARAM:mVg14] [PARAM:mVg15] [PARAM:mVg16]
Calibration data:	[PARAM:Cal1] [PARAM:Cal2] [PARAM:Cal3] [PARAM:Cal4] [PARAM:Cal5]

[PARAM:Cal6]
[PARAM:Cal7]
[PARAM:Cal8]
[PARAM:Cal9]
[PARAM:Cal10]
[PARAM:Cal11]
[PARAM:Cal12]
[PARAM:Cal13]
[PARAM:Cal14]
[PARAM:Cal15]
[PARAM:Cal16]

2.14.2 Random report parameter names

Values defining the test profile and controller parameters:

Plus Abort:	[PARAM:+Abort]
Minus Abort	[PARAM:-Abort]
Plus Tol:	[PARAM:+Tolerance]
Minus Tol:	[PARAM:-Tolerance]
Max Plus RMS:	[PARAM:Max+RMS]
Max Minus RMS:	[PARAM:Max-RMS]
Demand:	[PARAM:Demand]
Demand Displacement:	[PARAM:DmndDisp]
File name:	[PARAM:FileName]
Test profile name:	[PARAM:TestName]
Sample rate:	[PARAM:SampleFrequency]
Control channels:	[PARAM:Channels]
Channel combination method:	[PARAM:Combine]
Sigma clipping:	[PARAM:SigmaClipping]
Control lines:	[PARAM:Lines]
Max outlier percent:	[PARAM:MaxOutlierPercent]
Max drive voltage:	[PARAM:MaxDrive]
Start control gain:	[PARAM:StartControlGain]
Start averaging:	[PARAM:StartAveraging]
Start max system gain:	[PARAM:StartMaxSystemGain]
Start threshold volts:	[PARAM:ThresholdVoltage]
StartUp time:	[PARAM:StartupTime]
Run control gain:	[PARAM:RunControlGain]
Run averaging:	[PARAM:RunAveraging]
Run max system gain:	[PARAM:RunMaxSystemGain]
Run slew rate:	[PARAM:RunSlewRate]
Abort slew rate:	[PARAM:AbortRate]
Analysis Window Type:	[PARAM:Window]
Annotation line 1:	[PARAM:Note1]
Annotation line 2:	[PARAM:Note2]
Annotation line 3:	[PARAM:Note3]
Data directory:	[PARAM:DataDirectory]
Data storage:	[PARAM:DataStorage]
Breakpoint table	[PARAM:SegmentTable] (Table format: Frequency, Amplitude, Slope)
Schedule table	[PARAM:ScheduleTable] (Table format: Duration, Level)
Schedule table with dig outputs:	[PARAM:DigitalOutputTable] (table format: Duration, Level, DO0, DO1, DO2, DO3, DO4, DO5)

Values giving the test results:

Stop code:	[PARAM:StopCode]
Start time:	[PARAM:StartTime]
Current time:	[PARAM:Time]
Current level:	[PARAM:CurrentLevel]
Memorized drive:	[PARAM:MemorizedDrive]
Current modifier:	[PARAM:Modifier]
Time running:	[PARAM:RunTime]
Level duration:	[PARAM:LevelDuration]
Time on this level:	[PARAM:LevelTime]
Schedule looping count:	[PARAM:LoopCount]
Output drive voltage:	[PARAM:OutputRMS]
Control measurement:	[PARAM:Control]
Control displacement:	[PARAM:CtrlDisp]
Current system gain:	[PARAM:SystemGain]
Full-band RMS measurements:	[PARAM:Ch1]
	[PARAM:Ch2]
	[PARAM:Ch3]
	[PARAM:Ch4]
	[PARAM:Ch5]
	[PARAM:Ch6]
	[PARAM:Ch7]
	[PARAM:Ch8]
	[PARAM:Ch9]
	[PARAM:Ch10]
	[PARAM:Ch11]
	[PARAM:Ch12]
	[PARAM:Ch13]
	[PARAM:Ch14]
	[PARAM:Ch15]
	[PARAM:Ch16]
In-band RMS measurements:	[PARAM:ibRMS1]
	[PARAM:ibRMS2]
	[PARAM:ibRMS3]
	[PARAM:ibRMS4]
	[PARAM:ibRMS5]
	[PARAM:ibRMS6]
	[PARAM:ibRMS7]
	[PARAM:ibRMS8]
	[PARAM:ibRMS9]
	[PARAM:ibRMS10]
	[PARAM:ibRMS11]
	[PARAM:ibRMS12]
	[PARAM:ibRMS13]
	[PARAM:ibRMS14]
	[PARAM:ibRMS15]
	[PARAM:ibRMS16]

Unit names:

Acceleration units:	[PARAM:UnitAccel]
Displacement units:	[PARAM:UnitDisplacement]
Spectral density units:	[PARAM:UnitDensity]
Alternate units:	[PARAM:Unit1]
	[PARAM:Unit2]

[PARAM:Unit3]
[PARAM:Unit4]
[PARAM:Unit5]
[PARAM:Unit6]
[PARAM:Unit7]
[PARAM:Unit8]
[PARAM:Unit9]
[PARAM:Unit10]
[PARAM:Unit11]
[PARAM:Unit12]
[PARAM:Unit13]
[PARAM:Unit14]
[PARAM:Unit15]
[PARAM:Unit16]

Accelerometer calibration details:

Channel name:	[PARAM:Ch1Name]
	[PARAM:Ch2Name]
	[PARAM:Ch3Name]
	[PARAM:Ch4Name]
	[PARAM:Ch5Name]
	[PARAM:Ch6Name]
	[PARAM:Ch7Name]
	[PARAM:Ch8Name]
	[PARAM:Ch9Name]
	[PARAM:Ch10Name]
	[PARAM:Ch11Name]
	[PARAM:Ch12Name]
	[PARAM:Ch13Name]
	[PARAM:Ch14Name]
	[PARAM:Ch15Name]
	[PARAM:Ch16Name]
mV/G sensitivity:	[PARAM:mVg1]
	[PARAM:mVg2]
	[PARAM:mVg3]
	[PARAM:mVg4]
	[PARAM:mVg5]
	[PARAM:mVg6]
	[PARAM:mVg7]
	[PARAM:mVg8]
	[PARAM:mVg9]
	[PARAM:mVg10]
	[PARAM:mVg11]
	[PARAM:mVg12]
	[PARAM:mVg13]
	[PARAM:mVg14]
	[PARAM:mVg15]
	[PARAM:mVg16]
Calibration data:	[PARAM:Cal1]
	[PARAM:Cal2]
	[PARAM:Cal3]
	[PARAM:Cal4]
	[PARAM:Cal5]
	[PARAM:Cal6]
	[PARAM:Cal7]

[PARAM:Cal8]
[PARAM:Cal9]
[PARAM:Cal10]
[PARAM:Cal11]
[PARAM:Cal12]
[PARAM:Cal13]
[PARAM:Cal14]
[PARAM:Cal15]
[PARAM:Cal16]

2.14.3 Classical Shock report parameter names

Values defining the test profile and controller parameters:

Plus Abort:	[PARAM:+Abort]
Minus Abort	[PARAM:-Abort]
Plus Tol:	[PARAM:+Tolerance]
Minus Tol:	[PARAM:-Tolerance]
Max Start Voltage:	[PARAM:StartMaxV]
Max Run Voltage:	[PARAM:RunMaxV]
Control channels:	[PARAM:Channels]
Inverted channels:	[PARAM:Inverted]
Control Gain:	[PARAM:Gain]
System Lag:	[PARAM:Lag]
Max Frequency:	[PARAM:MaxFrequency]
Sample rate:	[PARAM:SampleFrequency]
Pulse Amplitude:	[PARAM:Demand]
Pre Pulse:	[PARAM:PrePulse]
Post Pulse:	[PARAM:PostPulse]
Pulse Width:	[PARAM:Width]
Pulse Polarity:	[PARAM:Polarity]
Pulse Type:	[PARAM:Type]
MilStd limits:	[PARAM:MilLimits]
File name:	[PARAM:FileName]
Test profile name:	[PARAM:TestName]
Annotation line 1:	[PARAM:Note1]
Annotation line 2:	[PARAM:Note2]
Annotation line 3:	[PARAM:Note3]
Data directory:	[PARAM:DataDirectory]
Data storage:	[PARAM:DataStorage]
Schedule table:	[PARAM:ScheduleTable]
Schedule table with dig. outputs:	[PARAM:DigitalOutputTable]
SRS synthesis method:	[PARAM:SRSSynthesis]
SRS analysis:	[PARAM:SRSAnalysis]
SRS damping value:	[PARAM:SRSDamping]
SRS pulse length:	[PARAM:SRSPeriod]
SRS breakpoint table:	[PARAM:SRSBreakpoints]

Values giving the test results:

Stop code:	[PARAM:StopCode]
Start Time:	[PARAM:StartTime]
Current Time:	[PARAM:Time]
Current Level:	[PARAM:CurrentLevel]
Memorized Drive:	[PARAM:MemorizedDrive]
Modifier:	[PARAM:Modifier]

Control peak level:	[PARAM:Control]
Output drive peak:	[PARAM:Output]
Pulses:	[PARAM:Pulses]

Unit names:

Acceleration units:	[PARAM:UnitAcceleration]
Velocity units:	[PARAM:UnitVelocity]
Displacement units:	[PARAM:UnitDisplacement]
Spectral density units:	[PARAM:UnitDensity]

Accelerometer calibration details:

Channel name:	[PARAM:Ch1Name]
	[PARAM:Ch2Name]
	[PARAM:Ch3Name]
	[PARAM:Ch4Name]
	[PARAM:Ch5Name]
	[PARAM:Ch6Name]
	[PARAM:Ch7Name]
	[PARAM:Ch8Name]
	[PARAM:Ch9Name]
	[PARAM:Ch10Name]
	[PARAM:Ch11Name]
	[PARAM:Ch12Name]
	[PARAM:Ch13Name]
	[PARAM:Ch14Name]
	[PARAM:Ch15Name]
	[PARAM:Ch16Name]
mV/G sensitivity:	[PARAM:mVg1]
	[PARAM:mVg2]
	[PARAM:mVg3]
	[PARAM:mVg4]
	[PARAM:mVg5]
	[PARAM:mVg6]
	[PARAM:mVg7]
	[PARAM:mVg8]
	[PARAM:mVg9]
	[PARAM:mVg10]
	[PARAM:mVg11]
	[PARAM:mVg12]
	[PARAM:mVg13]
	[PARAM:mVg14]
	[PARAM:mVg15]
	[PARAM:mVg16]
Calibration data:	[PARAM:Cal1]
	[PARAM:Cal2]
	[PARAM:Cal3]
	[PARAM:Cal4]
	[PARAM:Cal5]
	[PARAM:Cal6]
	[PARAM:Cal7]
	[PARAM:Cal8]
	[PARAM:Cal9]
	[PARAM:Cal10]
	[PARAM:Cal11]

[PARAM:Cal12]
[PARAM:Cal13]
[PARAM:Cal14]
[PARAM:Cal15]
[PARAM:Cal16]

2.14.4 Field Data Replicator report parameter names

Values defining the test limits:

Max output voltage:	[PARAM:MaxOutputVoltageRMS]
Max Grms (Ch1):	[PARAM:MaxGrms1]
	[PARAM:MaxGrms2]
	[PARAM:MaxGrms3]
	[PARAM:MaxGrms4]
	[PARAM:MaxGrms5]
	[PARAM:MaxGrms6]
	[PARAM:MaxGrms7]
	[PARAM:MaxGrms8]
	[PARAM:MaxGrms9]
	[PARAM:MaxGrms10]
	[PARAM:MaxGrms11]
	[PARAM:MaxGrms12]
	[PARAM:MaxGrms13]
	[PARAM:MaxGrms14]
	[PARAM:MaxGrms15]
	[PARAM:MaxGrms16]

Values defining the control loop parameters:

Controller status:	[PARAM:Active]
Start averaging:	[PARAM:StartAveraging]
Startup rate:	[PARAM:StartRate]
Abort rate:	[PARAM:AbortRate]
Run averaging:	[PARAM:RunAveraging]
Spectrum Control SNR:	[PARAM:SNRSpectrum]
Phase Control SNR:	[PARAM:SNRPhase]
Spectrum Response time:	[PARAM:ResponseSpectrum]
Phase Response time:	[PARAM:ResponsePhase]
Minimum frequency:	[PARAM:MinFrequency]
Maximum frequency:	[PARAM:MaxFrequency]
Notch status:	[PARAM:Notch]
Notch start frequency:	[PARAM:MinNotch]
Notch end frequency:	[PARAM:MaxNotch]
Playback file:	[PARAM:PlaybackFileName]
File name:	[PARAM:FileName]
Test profile name:	[PARAM:TestName]
Sample rate:	[PARAM:SampleFrequency]
Analysis Window Type:	[PARAM:Window]
Analysis Lines:	[PARAM:Lines
Annotation line 1:	[PARAM:Note1]
Annotation line 2:	[PARAM:Note2]
Annotation line 3:	[PARAM:Note3]
Data directory:	[PARAM:DataDirectory]
Data storage:	[PARAM:DataStorage]

Schedule table: [PARAM:ScheduleTable]
Schedule table with dig. outputs: [PARAM:DigitalOutputTable]

Values giving the test results:

Stop code: [PARAM:StopCode]
Start time: [PARAM:StartTime]
Current time: [PARAM:Time]
Current level: [PARAM:CurrentLevel]
Memorized drive: [PARAM:MemorizedDrive]
Current multiplier: [PARAM:Multiplier]
Time running: [PARAM:RunTime]
Level duration: [PARAM:LevelDuration]
Time on this level: [PARAM:LevelTime]
Output drive voltage: [PARAM:OutputRMS]
Percent RMS error: [PARAM:PercentRMSError]
Channel measurement: [PARAM:Ch1]
[PARAM:Ch2]
[PARAM:Ch3]
[PARAM:Ch4]
[PARAM:Ch5]
[PARAM:Ch6]
[PARAM:Ch7]
[PARAM:Ch8]
[PARAM:Ch9]
[PARAM:Ch10]
[PARAM:Ch11]
[PARAM:Ch12]
[PARAM:Ch13]
[PARAM:Ch14]
[PARAM:Ch15]
[PARAM:Ch16]

Peak system gain: [PARAM:PeakSystemGain]

Unit names:

Acceleration units: [PARAM:UnitAcceleration]
Spectral density units: [PARAM:UnitDensity]

Accelerometer calibration details:

Channel name: [PARAM:Ch1Name]
[PARAM:Ch2Name]
[PARAM:Ch3Name]
[PARAM:Ch4Name]
[PARAM:Ch5Name]
[PARAM:Ch6Name]
[PARAM:Ch7Name]
[PARAM:Ch8Name]
[PARAM:Ch9Name]
[PARAM:Ch10Name]
[PARAM:Ch11Name]
[PARAM:Ch12Name]
[PARAM:Ch13Name]

	[PARAM:Ch14Name]
	[PARAM:Ch15Name]
	[PARAM:Ch16Name]
mV/G sensitivity:	[PARAM:mVg1]
	[PARAM:mVg2]
	[PARAM:mVg3]
	[PARAM:mVg4]
	[PARAM:mVg5]
	[PARAM:mVg6]
	[PARAM:mVg7]
	[PARAM:mVg8]
	[PARAM:mVg9]
	[PARAM:mVg10]
	[PARAM:mVg11]
	[PARAM:mVg12]
	[PARAM:mVg13]
	[PARAM:mVg14]
	[PARAM:mVg15]
	[PARAM:mVg16]
Calibration data:	[PARAM:Cal1]
	[PARAM:Cal2]
	[PARAM:Cal3]
	[PARAM:Cal4]
	[PARAM:Cal5]
	[PARAM:Cal6]
	[PARAM:Cal7]
	[PARAM:Cal8]
	[PARAM:Cal9]
	[PARAM:Cal10]
	[PARAM:Cal11]
	[PARAM:Cal12]
	[PARAM:Cal13]
	[PARAM:Cal14]
	[PARAM:Cal15]
	[PARAM:Cal16]

2.14.5 TEDS report parameter names

Typical values from TEDS accelerometers. Actual available values are dependent on the manufacturer of the accelerometer

Channel 1

Manufacturer:	[TEDS:Manufacturer]
Model number:	[TEDS:Model]
Version letter:	[TEDS:Ver_let]
Version number:	[TEDS:Ver_no]
Serial no.:	[TEDS:Serial]
Calibration date	[TEDS:CalDate]
Sensitivity @ Fref	[TEDS:Sens@Ref]
Fref Frequency	[TEDS:Reffreq]
Phase inversion (0: 0°, 1: 180°)	[TEDS:Sign]
Sensitivity direction (x,y,z)	[TEDS:Direction]
Measure position ID	[TEDS:MessID]
User data (ascii)	[TEDS:User]
Reference temperature (T ref)	[TEDS:RefTemp]

Channel 2

Manufacturer:	[TEDS:2,Manufacturer]
Model number:	[TEDS:2,Model]
Version letter:	[TEDS:2,Ver_let]
Version number:	[TEDS:2,Ver_no]
Serial no.:	[TEDS:2,Serial]
Calibration date	[TEDS:2,CalDate]
Sensitivity @ Fref	[TEDS:2,Sens@Ref]
Fref Frequency	[TEDS:2,Reffreq]
Phase inversion (0: 0°, 1: 180°)	[TEDS:2,Sign]
Sensitivity direction (x,y,z)	[TEDS:2,Direction]
Measure position ID	[TEDS:2,MessID]
User data (ascii)	[TEDS:2,User]
Reference temperature (T ref)	[TEDS:2,RefTemp]

Channel 3

Manufacturer:	[TEDS:3,Manufacturer]
Model number:	[TEDS:3,Model]
Version letter:	[TEDS:3,Ver_let]
Version number:	[TEDS:3,Ver_no]
Serial no.:	[TEDS:3,Serial]
Calibration date	[TEDS:3,CalDate]
Sensitivity @ Fref	[TEDS:3,Sens@Ref]
Fref Frequency	[TEDS:3,Reffreq]
Phase inversion (0: 0°, 1: 180°)	[TEDS:3,Sign]
Sensitivity direction (x,y,z)	[TEDS:3,Direction]
Measure position ID	[TEDS:3,MessID]
User data (ascii)	[TEDS:3,User]
Reference temperature (T ref)	[TEDS:3,RefTemp]

Channel 4

Manufacturer:	[TEDS:4,Manufacturer]
Model number:	[TEDS:4,Model]
Version letter:	[TEDS:4,Ver_let]
Version number:	[TEDS:4,Ver_no]
Serial no.:	[TEDS:4,Serial]
Calibration date	[TEDS:4,CalDate]
Sensitivity @ Fref	[TEDS:4,Sens@Ref]
Fref Frequency	[TEDS:4,Reffreq]
Phase inversion (0: 0°, 1: 180°)	[TEDS:4,Sign]
Sensitivity direction (x,y,z)	[TEDS:4,Direction]
Measure position ID	[TEDS:4,MessID]
User data (ascii)	[TEDS:4,User]
Reference temperature (T ref)	[TEDS:4,RefTemp]

2.14.6 Sine report graph types

Graph types for Sine reports may be one of the following:

[GRAPH:Accel]	Acceleration vs. frequency
[GRAPH:Vel]	Velocity vs. frequency
[GRAPH:Disp]	Displacement vs. frequency
[GRAPH:Drive]	Drive signal vs. frequency
[GRAPH:Phase]	Control and/or Ch2-Ch1 Phase vs. frequency
[GRAPH:Xmiss]	Transmissibility (ChX/ChY) vs. frequency
[GRAPH:SystemGain]	System Gain (Output voltage / Input Acceleration)

[GRAPH:t_Accel]	Acceleration time history
[GRAPH:t_Vel]	Velocity time history
[GRAPH:t_Disp]	Displacement time history
[GRAPH:t_Drive]	Drive signal time history
[GRAPH:t_Phase]	Control and/or Ch2-Ch1 Phase time history
[GRAPH:t_Xmiss]	Transmissibility (ChX/ChY) time history
[GRAPH:t_SystemGain]	System Gain (Output voltage / Input Acceleration) time history
[GRAPH:t_Frequency]	Output frequency time history

Flags that may be applied to enable traces:

Ch1	Channel 1
Ch2	Channel 2
Ch3	Channel 3
Ch4	Channel 4
Ch5	Channel 5
Ch6	Channel 6
Ch7	Channel 7
Ch8	Channel 8
Ch9	Channel 9
Ch10	Channel 10
Ch11	Channel 11
Ch12	Channel 12
Ch13	Channel 13
Ch14	Channel 14
Ch15	Channel 15
Ch16	Channel 16
Control	Control channel
Demand	Demand level
Tolerance	Tolerance lines
Abort	Abort lines

Note: for Transmissibility graphs the first channel listed is the numerator and the last is the denominator.

Flags that may be applied to set the formatting of the graph:

W150	Select graph width, in mm (default is 150)
H100	Select graph height, in mm (default is 100)
LogX	Use a logarithmic horizontal axis (default is linear)
LogY	Use a logarithmic vertical axis (default is linear)
BW	Create a black-and-white plot (default is color)
XMin10.0	Force the minimum value on the X axis to 10.0
XMax100.0	Force the maximum value on the X axis to 100.0
YMin10.0	Force the minimum value on the Y axis to 10.0
YMax100.0	Force the maximum value on the Y axis to 100.0

Example:

[GRAPH:Accel,Ch1,Ch2,LogX,LogY,BW]

2.14.7 Random report graph types

Graph types for Random reports may be one of the following:

[GRAPH:Accel]	Acceleration vs. frequency

[GRAPH:Drive]	Drive signal vs. frequency
[GRAPH:Filter]	Filter weight vs. frequency
[GRAPH:Xmiss]	Transmissibility (ChX/ChY) vs. frequency

Flags that may be applied to enable traces:

Ch1	Channel 1
Ch2	Channel 2
Ch3	Channel 3
Ch4	Channel 4
Ch5	Channel 5
Ch6	Channel 6
Ch7	Channel 7
Ch8	Channel 8
Ch9	Channel 9
Ch10	Channel 10
Ch11	Channel 11
Ch12	Channel 12
Ch13	Channel 13
Ch14	Channel 14
Ch15	Channel 15
Ch16	Channel 16
Control	Control channel
Demand	Demand level
Tolerance	Tolerance lines
Abort	Abort lines

Note: for Transmissibility graphs the first channel listed is the numerator and the last is the denominator.

Flags that may be applied to set the formatting of the graph:

W150	Select graph width, in mm (default is 150)
H100	select graph height, in mm (default is 100)
LogX	Use a logarithmic horizontal axis (default is linear)
LogY	Use a logarithmic vertical axis (default is linear)
BW	Create a black-and-white plot (default is color)
ActiveLines	Display only the frequency range being controlled
XMin10.0	Force the minimum value on the X axis to 10.0
XMax100.0	Force the maximum value on the X axis to 100.0
YMin10.0	Force the minimum value on the Y axis to 10.0
YMax100.0	Force the maximum value on the Y axis to 100.0

Example:

[GRAPH:Accel,Ch1,Ch2,LogX,LogY,BW]

2.14.8 Shock report graph types

Graph types for Sine reports may be one of the following:

[GRAPH:Accel]	Acceleration vs. time
[GRAPH:Vel]	Velocity vs. time
[GRAPH:Disp]	Displacement vs. time
[GRAPH:Frequency]	Spectral content of the acceleration signal
[GRAPH:Phase]	Phase vs. frequency
[GRAPH:Drive]	Drive signal vs. time

[GRAPH:Spectrum]	Drive signal vs. frequency

Flags that may be applied to enable traces:

Ch1	Channel 1
Ch2	Channel 2
Ch3	Channel 3
Ch4	Channel 4
Ch5	Channel 5
Ch6	Channel 6
Ch7	Channel 7
Ch8	Channel 8
Ch9	Channel 9
Ch10	Channel 10
Ch11	Channel 11
Ch12	Channel 12
Ch13	Channel 13
Ch14	Channel 14
Ch15	Channel 15
Ch16	Channel 16
Control	Control channel
Demand	Demand level
Tolerance	Tolerance lines
Abort	Abort lines

Flags that may be applied to set the formatting of the graph:

W150	Select graph width, in mm (default is 150)
H100	Select graph height, in mm (default is 100)
LogX	Use a logarithmic horizontal axis (default is linear)
LogY	Use a logarithmic vertical axis (default is linear)
BW	Create a black-and-white plot (default is color)
Full	Plot the full time duration (including pre and post-pulse compensation)
XMin10.0	Force the minimum value on the X axis to 10.0
XMax100.0	Force the maximum value on the X axis to 100.0
YMin10.0	Force the minimum value on the Y axis to 10.0
YMax100.0	Force the maximum value on the Y axis to 100.0

Example:

[GRAPH:Accel,Demand,Control,Tolerance,BW]

2.14.9 Field Data Replicator report graph types

Graph types for Field Data Replicator reports may be one of the following:

[GRAPH:Freq]	Input signal vs. frequency
[GRAPH:Time]	Input signal vs. time
[GRAPH:Drive]	Drive signal vs. frequency
[GRAPH:DriveTime]	Drive signal vs. time
[GRAPH:Filter]	Filter function vs. frequency
[GRAPH:Xmiss]	Transmissibility (ChX/ChY) vs. frequency
[GRAPH:t_Accel]	Acceleration time history
[GRAPH:t_Drive]	Drive Time history

Flags that may be applied to enable traces: (Input signal graphs only)

Ch1	Channel 1
Ch2	Channel 2
Ch3	Channel 3
Ch4	Channel 4
Ch5	Channel 5
Ch6	Channel 6
Ch7	Channel 7
Ch8	Channel 8
Ch9	Channel 9
Ch10	Channel 10
Ch11	Channel 11
Ch12	Channel 12
Ch13	Channel 13
Ch14	Channel 14
Ch15	Channel 15
Ch16	Channel 16

Note: for Transmissibility graphs the first channel listed is the numerator and the last is the denominator.

Flags that may be applied to set the formatting of the graph:

W150	select graph width, in mm (default is 150)
H100	select graph height, in mm (default is 100)
LogX	Use a logarithmic horizontal axis (default is linear)
LogY	Use a logarithmic vertical axis (default is linear)
BW	Create a black-and-white plot (default is color)
XMin10.0	Force the minimum value on the X axis to 10.0
XMax100.0	Force the maximum value on the X axis to 100.0
YMin10.0	Force the minimum value on the Y axis to 10.0
YMax100.0	Force the maximum value on the Y axis to 100.0

Example:

[GRAPH:Freq,Ch1,Ch2,LogX,LogY,BW]

3 Menu Commands

List of menus:

- File Menu
- Configuration Menu
- Test Menu
- Graph Menu
- Cursor Menu
- View Menu
- Window Menu
- Help Menu

3.1 File menu commands

The file menu is used for loading, saving, copying and printing data files and graphs. It offers the following commands:

Edit Test Settings	Change settings in current test profile
Define New Test	Create a new test profile.
Open Test Profile	Open an existing test profile.
Open Recent Test	Select from recently loaded test profiles.
New Graph	Creates a new graph.
Copy Graph	Copy the selected graph to the windows clipboard.
Save as...	
...Data File	Saves the active data set to the disk.
...Document File	Saves the document file using a defined template
...Text File	Saves the active data set as a tab delimited file
...Spreadsheet File	Saves the active data set as a csv delimited file
...Matlab File	Saves the active data set as a Matlab file
...Meta File	Saves the selected graph in emf format
...Bitmap File	Saves the selected graph in bmp format.
...PNG File	Saves the selected graph in png format
Open Data	Read a stored data file from the disk.
Previous Data	Read the previous data file from the disk.
Next Data	Read the next data file from the disk.
Close	Closes an opened graph.
Save Report	Saves the document file using a defined template
Print Report	Creates a printed report using a defined template
Print Active Graph	Prints the selected graph.
Printer Setup	Selects a printer and printer connection.
Print Options	Changes the graph scaling options used for printing.
Explore Data	Starts Windows Explorer
Exit	Exits *VibrationVIEW*.

3.1.1 Edit Test Settings command (File menu)

This command opens the Define Test dialog box allowing the user to edit the currently loaded test.

You can also open a previously defined test, or save the test under a different name from the Define Test dialog box.

Depending on which test type is currently active, one of the following Define Tests dialog boxes will be opened:

- Sine Define
- Random Define
- Classical Shock Define
- Field Data Replicator Define

Shortcuts

Toolbar: 

3.1.2 Define New Test command (File menu)

Use this command to have the program guide you through a series of dialog boxes to define a new test. All values are initialized to their defaults and may be changed as required by your test.

If your new test is substantially similar to another test you have already defined, open the similar test using the Test..Open Test menu command, and then use the Test..Edit Test menu command to bring up the Define Test dialog box. Use the "Save As..." button in that dialog box to save the test under a new name.

Shortcuts

Toolbar: 

3.1.3 Open Test Profile command (File menu)

Use this command to open any previously defined and saved test profile.

Shortcuts

Toolbar: 

3.1.4 Open Recent Test command (File menu)

This command opens a pop-up submenu with a list of the 10 most recently used tests. To open one of the tests listed, click on it.

3.1.5 New Graph command (File menu)

Use this command to create a new graph in *VibrationVIEW*. A dialog box will prompt you with the types of graphs available.

For more information on the graph dialog boxes and graph types for the various tests, see:

- System Check Graph Settings dialog box
- Sine Graph Settings dialog box
- Random Graph Settings dialog box
- Shock Graph Settings dialog box
- Field Data Replicator Graph Settings dialog box

Shortcuts

Toolbar: 
Keys: Ctrl+N

3.1.6 Copy Graph command (File menu)

Use this command to copy the selected graph onto the clipboard. Once on the clipboard, it may be pasted into another application, such as a word processor, using that program's Paste command.

Copying data to the clipboard replaces the contents previously stored there.

Shortcuts

Toolbar: 
Keys: Ctrl+C

3.1.7 Save Data command (File menu)

Use this command to store the active data set to a file that may then be loaded back into *VibrationVIEW* using the File..Open Data menu command for viewing and printing. This command will bring up a Save As... dialog box with a default file name derived from the date and time the test began and an index that is incremented every time a data set is stored. The file name may be changed if you so desire. Click on the "Save" button to save the data to a file.

Note: The live data stored in the file is that data that is displayed when the "Save" button is pressed in the Save As... dialog box. If you want to save a particular set of data (for example, a transient occurrence in an FDR playback file), click the "Save" button as soon as you see the data you want to save appear in the graphs. This data may then be loaded again for analysis. In this way, the Save Data menu command may be used as a form of freeze-frame for the live data.

Shortcuts

Toolbar: 
Keys: Ctrl+S

3.1.8 Save as Document File command (File Menu)

This command will save a document with a variety of useful information and graphs. The information saved depends on the template that you choose.

To create a report select the File..Save as Document File menu command, choose a template, decide where to store the report and under what name and click the "OK" button.

See also: How to create customized reports

Shortcuts

Toolbar: 
Keys: Ctrl+R
Menus: File..Save Report

3.1.9 Save as Text File command (File menu)

Use this command to store the active data set to a file that may then be loaded into a word processor for viewing and printing. This command will bring up a Save As... dialog box with Save as type options of "Tab-delimited text file (*.txt)". "Tab-delimited files" are typically used with word processors. Click on the "Save" button to save the data to a file.

Note: The Save as text file command is an export only function. All data is exported. If you wish to use the data with *VibrationVIEW* use the Save as ... Data File command instead.

3.1.10 Save as Spreadsheet File command (File menu)

Use this command to store the active data set to a file that may then be loaded into a spreadsheet for viewing and printing. This command will bring up a Save As... dialog box with Save as type options of "Comma-separated-values file (*.csv)". Comma separated files are typically used with spreadsheets. Click on the "Save" button to save the data to a file.

Note: The Save as text file command is an export only function. All data is exported. If you wish to use the data with *VibrationVIEW* use the Save as ... Data File command instead.

3.1.11 Save as *Matlab* File command (File menu)

Use this command to store the active data set to a file that may then be loaded by a MATLAB or MATLAB compatible technical computing package. This command will bring up a Save As... dialog box with Save as type options of "Matlab Files (*.mat)".

Note: The Save as *Matlab* File command is an export only function. All data is exported. If you wish to use the data with *VibrationVIEW* use the Save as ... Data File command instead.

3.1.12 Save as Meta File command (File Menu)

Use this command to save the currently selected graph to a file using the Meta File (emf) format. Graphs saved in this form may then be imported into another application, such as a word processor, but *VibrationVIEW* cannot read these files, nor can the graph scales be changed.

Select the File..Save Data menu command to save the raw data for later loading and viewing in *VibrationVIEW*, with full control over the scaling and format of the graphs.

Select the File..Copy Graph menu command to copy the graph image to the windows clipboard so that it can be directly pasted into another application.

3.1.13 Save as Bitmap File command (File menu)

Use this command to save the currently selected graph to a file using the Bitmap File (bmp) format. Graphs saved in this form may then be imported into another application, such as a word processor, but *VibrationVIEW* cannot read these files, nor can the graph scales be changed.

Select the File..Save Data menu command to save the raw data for later loading and viewing in *VibrationVIEW*, with full control over the scaling and format of the graphs.

Select the File..Copy Graph menu command to copy the graph image to the windows clipboard so that it can be directly pasted into another application.

3.1.14 Save as PNG File command (File menu)

Use this command to save the currently selected graph to a file using the portable network graphics (png) format. Graphs saved in this form may then be imported into another application, such as a word processor, but *VibrationVIEW* cannot read these files, nor can the graph scales be changed.

Select the File..Save Data menu command to save the raw data for later loading and viewing in *VibrationVIEW*, with full control over the scaling and format of the graphs.

Select the File..Copy Graph menu command to copy the graph image to the windows clipboard so that it can be directly pasted into another application.

3.1.15 Open Data command (File menu)

Use this command to read a data set previously saved with *VibrationVIEW* using the File..Save Data menu command. An Open File dialog box will be displayed, allowing selection of the data set you wish to view. Use the Window menu to switch among the multiple open documents. See the Window 1, 2,.. menu command.

Shortcuts

Toolbar: 
Keys: Ctrl+O

3.1.16 Previous Data command (File menu)

Use this command to read and plot the data file stored in the same directory as the current file, but stored immediately prior to the current file. This command is available only when a graph of a stored data file is active.

This command, along with the Next Data menu command, is useful for quickly scanning through a set of data files in a directory in chronological order.

Shortcuts

Keys: Left arrow
Down arrow
Keypad Minus (-)

Hint: To see multiple types of graphs of each data file, select the Window..New menu command and the Window..Tile menu command to create and arrange multiple graphs before using this command to scan through the files.

3.1.17 Next Data command (File menu)

Use this command to read and plot the data file stored in the same directory as the current file, but stored immediately after the current file. This command is only available when a graph of a stored data file is active.

This command, along with the Previous Data menu command, is useful for quickly scanning through a set of data files in a directory in chronological order.

Shortcuts

Keys: Right arrow
Up arrow
Keypad Plus (+)

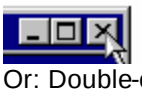
Hint: To see multiple types of graphs of each data file, select the Window..New menu command and the Window..Tile menu command to create and arrange multiple graphs before using this command to scan through the files.

3.1.18 Close command (File menu)

Use this command to close the active graph.

Shortcuts:

Mouse Click the X in the upper right corner of the graph window:



Or: Double-click the graph's system menu.
Or: Select Close from the graph's system menu. Clicking the icon in the upper left corner of the window, as shown below accesses the graph's system menu:

3.1.19 Save Report command (File Menu)

This command will save a document with a variety of useful information and graphs. The information saved depends on the template that you choose.

To create a report select the File..Save as Document File menu command, choose a template, decide where to store the report and under what name, and click the "OK" button.

See also: How to create customized reports

Shortcuts

Toolbar: 
Keys: Ctrl+R
Menus: File..Save as .. Report Document File

3.1.20 Print Report command (File Menu)

This command will create a report using the same templates that the Create Report Command uses but instead of storing it, the report will be printed on the default printer.

To print a report, select the File..Print Report menu command, choose a template and click the "OK" button.

See also: How to create customized reports

3.1.21 Email Report command (File Menu)

This command will create a report using the same templates that the Create Report Command uses but instead of storing it, the report will be sent to the e-mail recipient(s) you specify.

To Email a report, select the File..Email Report menu command, choose a template and click the "OK" button, then select recipient.

See also: How to create customized reports
File Send Email

3.1.22 Print Active Graph command (File menu)

Use this command to print a graph to the printer. A printer selection dialog box (see Printer Setup for a description of this dialog) will be presented, allowing selection of the printer and layout options. See also the Printer Setup and the Print Options menu commands.

Shortcuts

Toolbar: 
Keys: Ctrl+P

3.1.23 Print Screen command (File Menu)

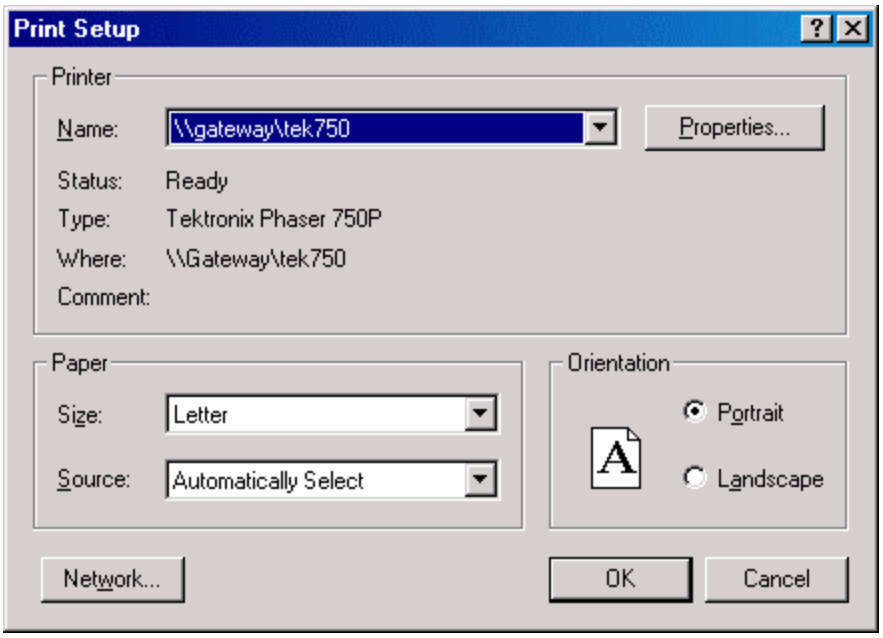
This command will send the entire display including toolbars, to the selected printer.

Shortcuts

Keys: F12

See also:
Print Active Graph
Printer Setup

3.1.24 Printer Setup command (File menu)



The following options allow you to select the destination printer and its connection. Changes made using this command will be used as the default settings for all subsequent Print commands.

Name

Select the printer you want to use. Select the Default Printer; or select a specific printer option from the drop-down list. You can install printers and configure ports using the Windows Control Panel.

Properties

Displays a dialog box where you can make additional choices about printing, specific to the type of printer you have selected.

Paper Size

Select the size of the paper on which the document is to be printed.

Paper Source

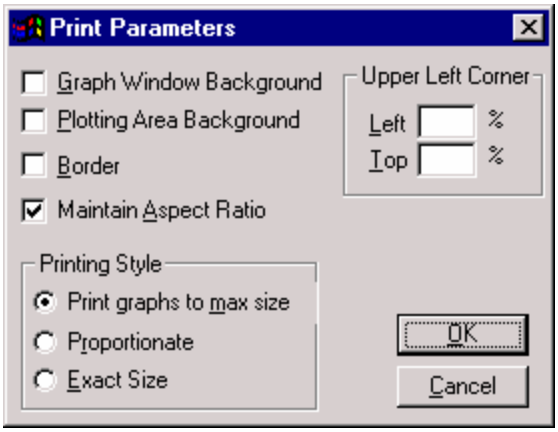
Some printers offer multiple trays for different paper sources. Select the tray here.

Orientation

Select Portrait or Landscape orientation.

3.1.25 Print Options command (File menu)

This command brings up the Print Parameters dialog box that may be used to change the way graphs are printed.



The following settings may be made:

Graph Window Background
If this box is checked, the background color of a graph is printed. Otherwise it is ignored. By default this box is not checked.

Plotting Area Background
If this box is checked, the background color of a graph's plotting area is printed. Otherwise it is ignored. By default this box is not checked.

Border
If this box is checked, graphs are printed with a rectangular border around them. By default this box is not checked.

Maintain Aspect Ratio
If this box is checked, the aspect ratio is maintained when printing with Print graphs to max size and Proportionate printing styles. Otherwise the graph is mapped to the full printed page and the aspect ratio may change. By default this box is checked.

Upper Left Corner
X- and Y-position of the left-upper corner of the page relative to the paper sheet width. A Left value of 0.0 indicates that the page will be printed starting at the left side of the paper sheet. A Top value of 0.0 indicates that the page will be printed starting at the left side of the paper sheet. This parameter is only used if the Exact Size printing style is selected.

Print graphs to max size and **Proportionate**
(These two options are equivalent for graphs printed by *VibrationVIEW*.)
Print the graphs with the maximum size that a the paper in the printer allows. These options will scale the text and symbols so that their size relative to other objects remains constant.

Exact Size
Printed graphs will maintain their original sizes. Only this option guarantees that text and symbols will be printed with their specified size.

3.1.26 Explore Data command (File Menu)

This command will open a Windows Explorer window to allow you to copy/move/rename files.

3.1.27 Exit command (File menu)

Use this command to end your *VibrationVIEW* session. You can also use the Close command on the application system menu.

Shortcuts

Mouse: Click the X in the upper right corner of the *VibrationVIEW* window

or double-click the application's system menu.

Keys: Alt+F4

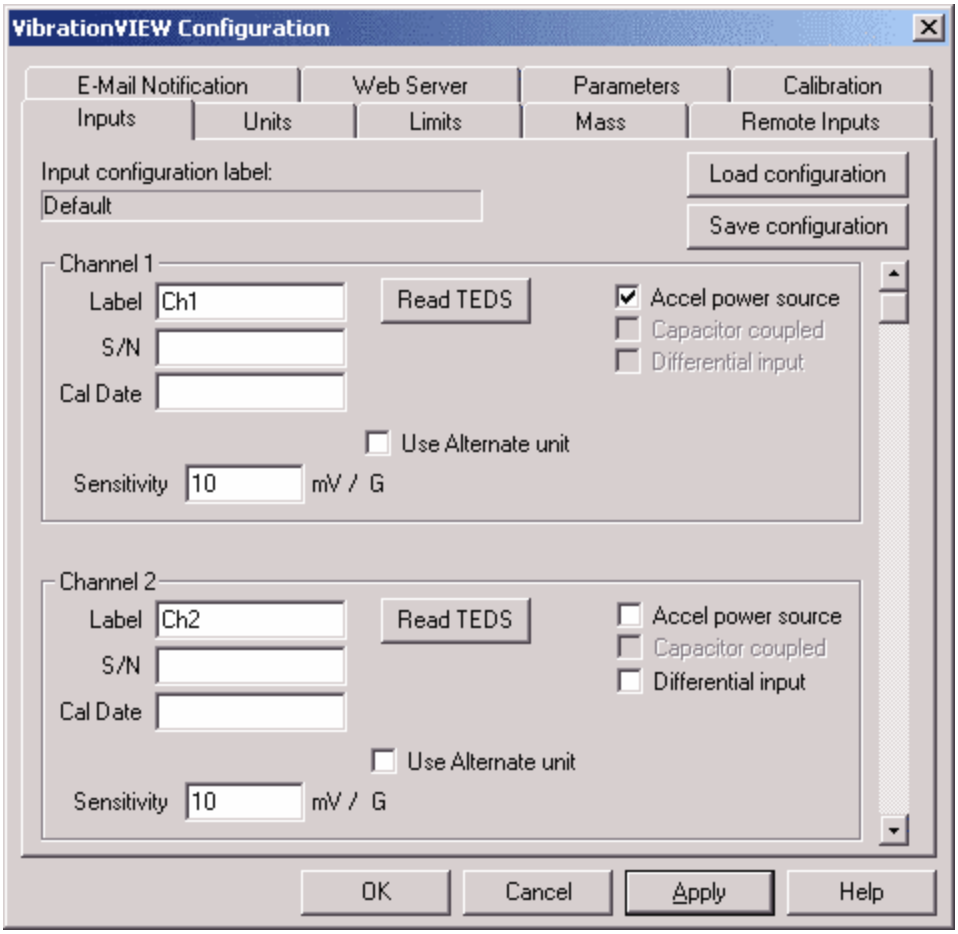
3.2 Configuration menu commands

The configuration menu is used to set the characteristics and limits of the shaker system. It offers the following commands:

Inputs	Enter the accelerometer parameters
Units	Select the units to use in the program
System Limits	Enter the physical limits of your shaker
System Mass	Enter the masses for system limitations
Remote Inputs	Configure the Remote Inputs parameters
E-Mail Notification	Configure the E-Mail notification parameters
Web Server	Configure the Web Server parameters
Parameters	General system configuration parameters
Calibration	Used for the instrument calibration procedure

3.2.1 Inputs command (Configuration menu)

This command opens the Inputs tab on the *VibrationVIEW* Configuration dialog box.



Use this dialog box to enter the calibrated sensitivity ratings for your accelerometers in mV/G. If you have a TEDS compatible accelerometer, click the "Read TEDS" button to have *VibrationVIEW* read this information from your accelerometer. ***Do not use this button if you do not have TEDS compatible accelerometers.***

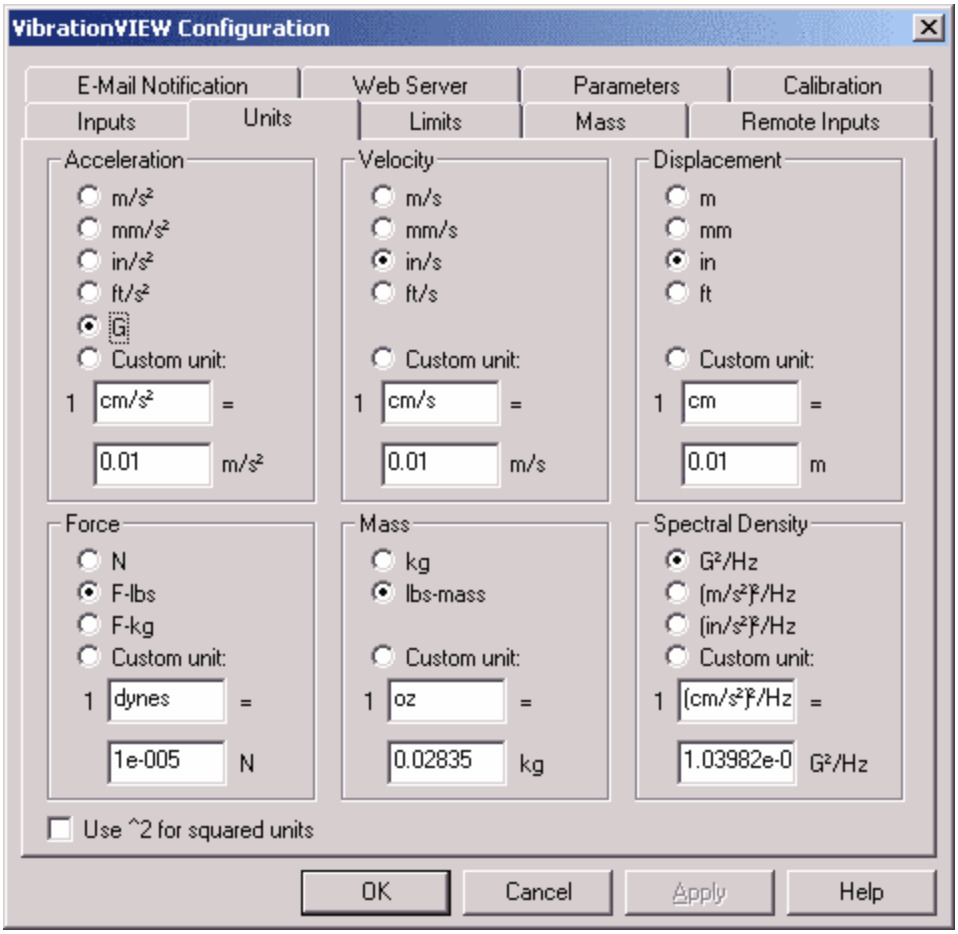
Check the "Accel power source" button to enable the accelerometer power source built into the *VibrationVIEW* I/O box.

Check the "Use alternate unit" checkbox if your accelerometer ratings are listed in some other unit.

To verify that the system is operating properly, and that the accelerometer sensitivity settings (along with the other system parameters) are correct, follow the procedure found under "How to verify shaker system operation."

3.2.2 Units command (Configuration menu)

This command opens the System Units tab on the *VibrationVIEW* Configuration dialog box.



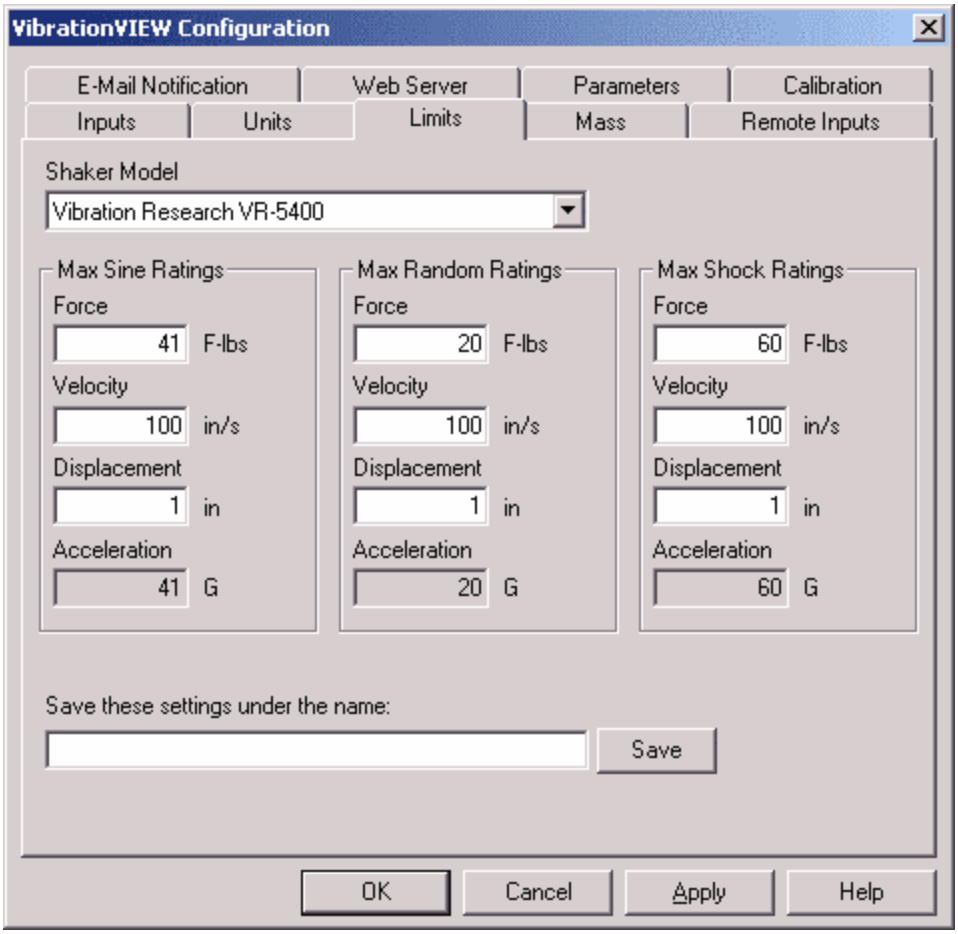
Use this dialog box to select and define the desired units to be used by the program. Changing the units will stop any active test.

Select the desired units for Acceleration, Velocity, Displacement, Force, Mass and Acceleration Spectral Density. If the listed units are not appropriate for your test, custom units may be defined by entering a label in the upper box (between the 1 and the =) and the appropriate scaling factor from the standard international unit in the lower box. For example:

1 oz = 0.02835 kg

3.2.3 System Limits command (Configuration menu)

This command opens the System Limits tab on the *VibrationVIEW* Configuration dialog box.

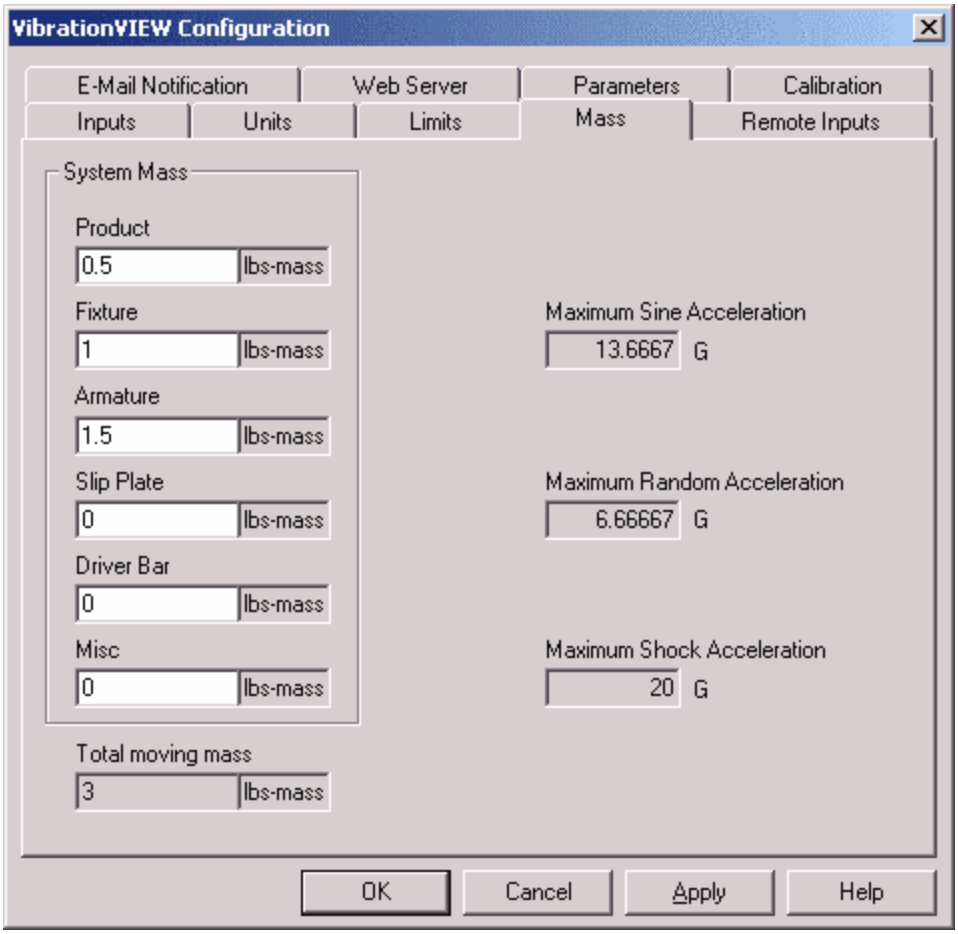


Use this dialog box to define the limits for your shaker system. These parameters are checked at run time to determine if the test you are about to run is within the specifications of your shaker system. The acceleration and velocity limits are measured 0-to-peak while displacement limits are measured peak-to-peak.

Note: The System Mass parameters are used to compute the acceleration rating limits.

3.2.4 System Mass command (Configuration menu)

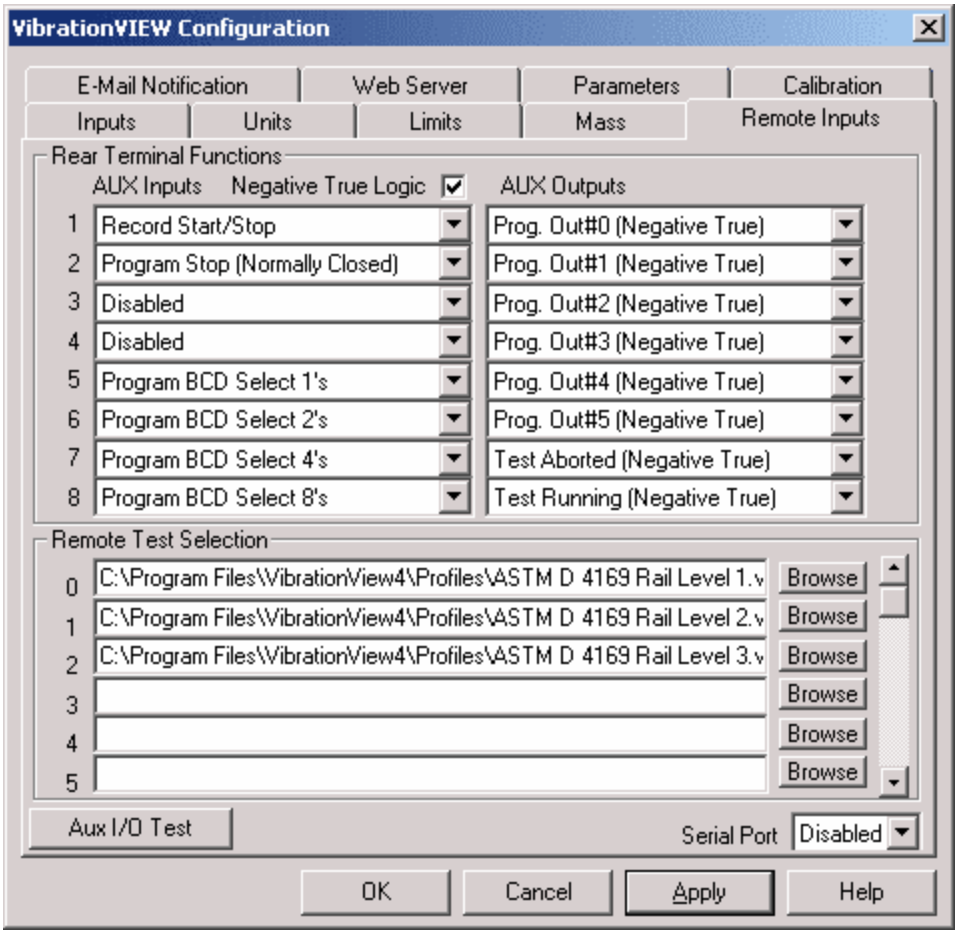
This command opens the System Mass tab on the *VibrationVIEW* Configuration dialog box.



Use this dialog box to enter the respective weights (mass) for the different elements of your system. These mass entries are totaled and used in conjunction with the System Limits to determine whether your tests can be run within the specifications of your shaker system. The units used for the masses are selected under the System Units tab.

3.2.5 Remote Inputs command (Configuration menu)

This command opens the Remote Inputs tab on the *VibrationVIEW* Configuration dialog box. Selecting Configuration..Remote Inputs menu command displays this dialog box.



Use this command to enable and configure the remote input and output option The remote inputs and outputs are an optional feature of *VibrationVIEW* that must be enabled in the software key.

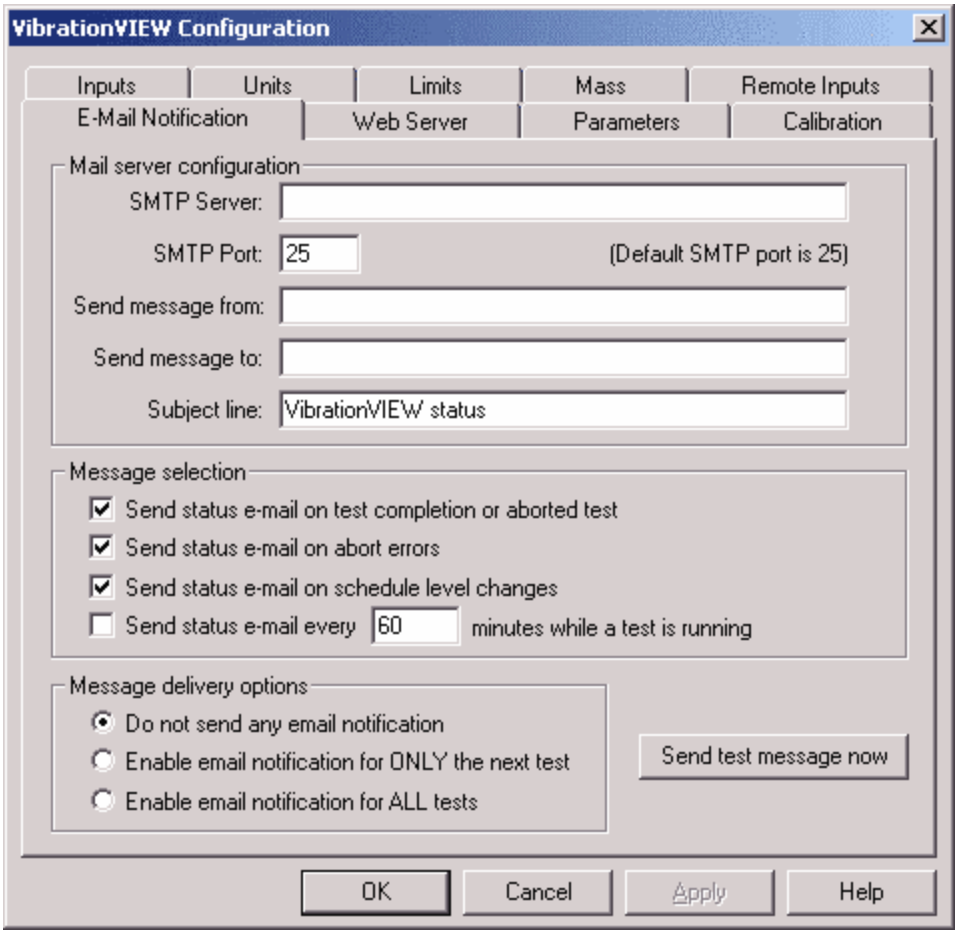
The serial port option allows control of *VibrationVIEW* from an optional RF remote. The RF remote frees you from the computer keyboard and allows control of your test from the palm of your hand.

See also

How to use Remote Inputs to control *VibrationVIEW*

3.2.6 E-Mail Notification command (Configuration menu)

This command opens the Config..Email tab on the *VibrationVIEW* Configuration dialog box.



Use this dialog box to configure E-Mail Notification. E-Mail notification is an optional component of *VibrationVIEW*, the software key must be programmed to enable it.

E-Mail Notification allows tests to be monitored from a remote location with any email client.

To configure the E-Mail Notification, complete the following steps:

1. Enter your SMTP Server into the form. Your SMTP Server name can often be determined from your email client configuration (Outlook or similar). If you are not sure what SMTP server to use, ask your Internet Service Provider, or your Network Administrator.
2. The SMTP port is almost always 25, which is the officially recognized port number for mail transport servers. Only change this if your Network Administrator has given you a different port number to use.
3. Enter an email address into the 'Send message from' field. This can be your e-mail address, if you wish to receive any replies to the automatically sent messages. Some SMTP servers will only accept mail if the "from" address is a valid email address on the SMTP server.
4. Enter a destination email address in the 'Send message to' field. Multiple e-mail addresses can be entered with a ";" separator.
5. Enter a subject line to identify the message. This can be any convenient text, although it is useful to use a unique subject line that can be used by an email program such as

Outlook to filter the messages to a separate folder. If you are sending email to a pager or cell phone you may want to leave the subject line blank to keep the message length shorter.

Select your message options:

- "Send status e-mail on test completion or aborted test" will send e-mail every time a test is stopped.
- "Send status e-mail on abort errors" will send an e-mail every time a test aborts.
- "Send status e-mail on schedule level changes" will send an e-mail each time the level in the test schedule changes.
- "Send status email every n minutes while a test is running" will send an e-mail every n minutes to keep you updated on test status continually.

Select Message delivery Options:

- "Do not send any email notification" turns the notification off.
- "Enable email notification for ONLY the next text" turns the notification on but shuts it back off after the next test is complete.
- "Enable email notification for ALL tests" turns the notification on.

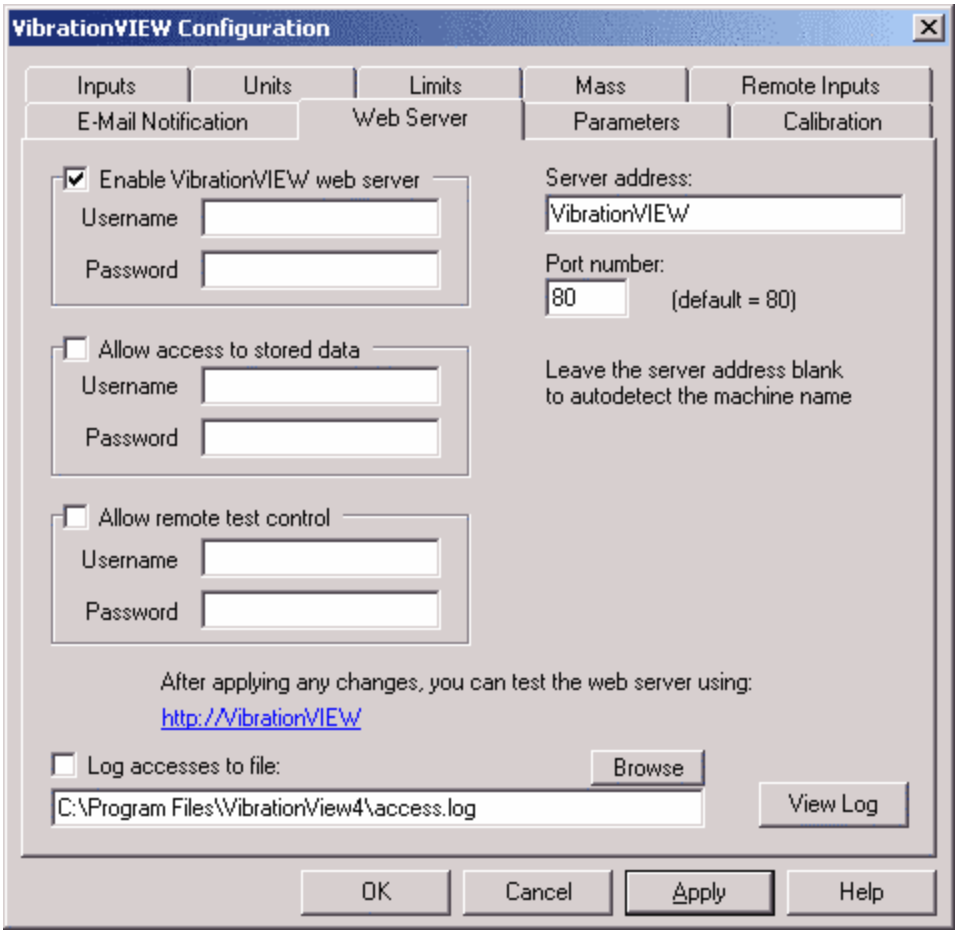
You can test the email configuration by pressing the "Send test message now" button. This will send a test message using the configured addresses. If you don't receive the message or get an error message, check the configuration to make sure all of the settings are correct.

Shortcuts

Keys: Ctrl+E

3.2.7 Web Server command (Configuration menu)

This command opens the Config..Web tab on the *VibrationVIEW* Configuration dialog box.



Use this dialog box to configure the *VibrationVIEW* Web server. The Web server is an optional component of *VibrationVIEW*, the software key must be programmed to enable it.

The Web server allows tests to be monitored and controlled from a remote location with a web client such as *Netscape Navigator* or *Internet Explorer*.

To enable the web server check the box labeled "Enable *VibrationVIEW* web server."
Fill in the fields labeled ***Username*** and ***Password*** to restrict access to the web features. If these fields are left blank, your web client will allow anyone to access the live test data.

To allow web access to data stored on this machine check the box labeled "Allow access to stored data". This allows remote access of stored data from previous tests run on this machine.
Fill in the field labeled ***Username*** and ***Password*** to restrict access to stored data. If these fields are left blank, your web client will allow anyone to access the stored data.

To allow web access to starting and stopping of tests on this machine check the box labeled "Allow remote test control".
Fill in the field labeled ***Username*** and ***Password***. Test control requires the use of a login Username and Password.

Normally leave the ***Server Address*** parameter blank. The local computer name will automatically be

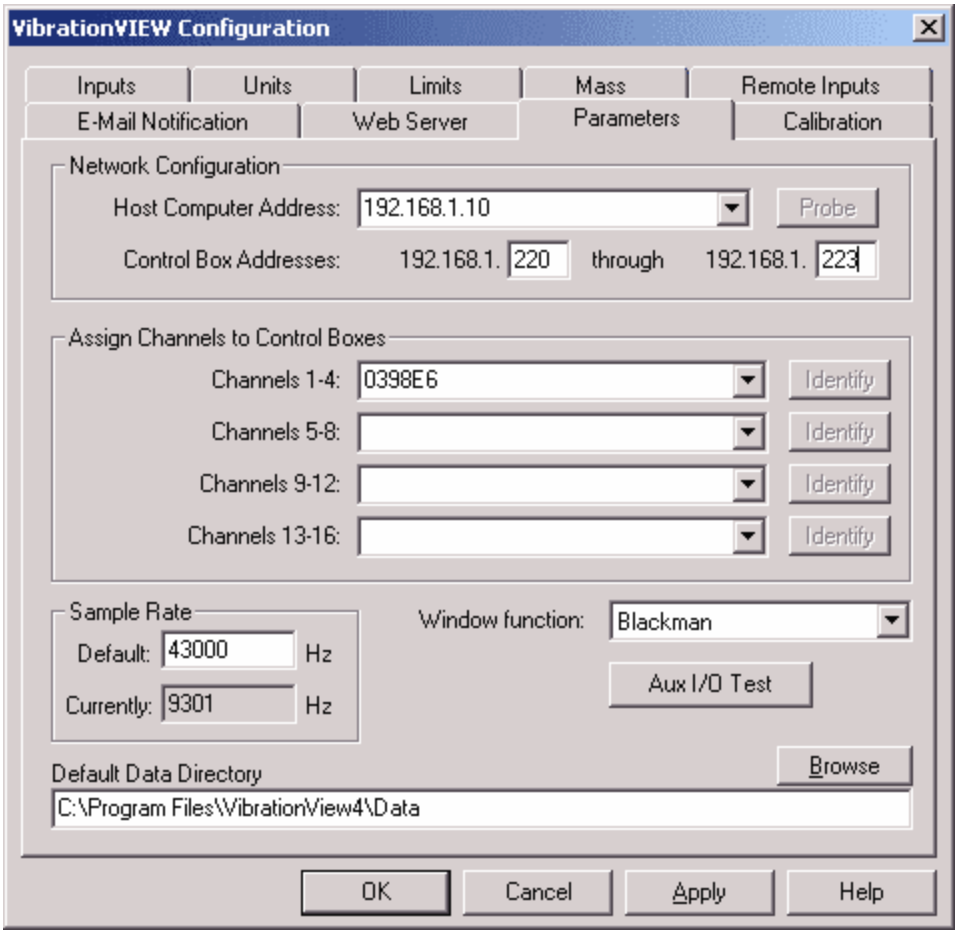
filled in here for you. If you wish to bind the interface to a specific IP address you can enter the IP address here. The standard port number for a web server is 80. However, you can set the port address to any number between 1 and 65535 if you have another web server running on this machine, or don't want the server visible on the standard port. If the port number is not 80, then you will have to include the port number when typing the web server address in your web browser. For example, if you set the port number to 10000, then in your web browser you would have to enter the address 'http://*VibrationVIEW*:10000'

After you enter all the fields and click "Apply" you can test your connection by clicking on the blue http://name label. This will launch your default browser on your machine and, if all is configured properly, display the Web interface. This is also the address you use on any remote machines to access this machine remotely.

Check the box labeled "Log accesses to file:" and specify a log file if you need debugging information. The logfile will contain a log of all the remote access attempts.

3.2.8 Parameters command (Configuration menu)

This command opens the Parameters tab on the *VibrationVIEW* Configuration dialog box.



Use this dialog box to select Simulation/Demo Mode, File Viewer Mode, or to initially configure the software initializing your VR8500 I/O unit(s). The Simulation/Demo and File Viewer modes can be selected as special "Host Computer Address". These modes do not require a VR8500 be attached to this system.

When the VR8500 is connected to this computer, select the network card connected to the VR8500. Pick the network card's address with the "Host Computer Address" drop list. The selected network card can be probed by pressing the "Probe" pushbutton. This command will display details for each VR8500 found on the network segment, as well as the selected network card's configuration.

Control Box Addresses are used to assign IP address to the VR8500(s) attached to this computer.

The VR8500 attached to this computer must be selected for channel assignment. When your network card and control box assignments are all valid, the VR8500 serial numbers will be in the drop list(s) labeled Channels 1-4, 5-8. 9-12, and 13-16. A special serial number "ANY" may be used in single box systems. "ANY" assignments will connect to the first VR8500 found on the selected network.

After the box(es) are assigned and initialize, the Identify button for each box is active. Clicking the "Identify" causes the assigned box to wink, alternating the Status and Fault lights on the front panel.

In the dialog shown, the computer network card has been previously been assigned to 192.168.1.10. This card is networked to VR8500 serial number 0398E6, which is automatically assigned the address 192.168.1.220. If a second, third, and forth VR8500 are added to the system they will be automatically assigned addresses 192.168.1.221, 192.168.1.222, and 192.168.1.223.

This dialog box may also be used to set the sampling rate and window function used in System Check mode and to see the current sampling rate used by the controller. Note that each test has a sample rate configuration setting, so the Default rate set in this dialog box applies only to System Check mode.

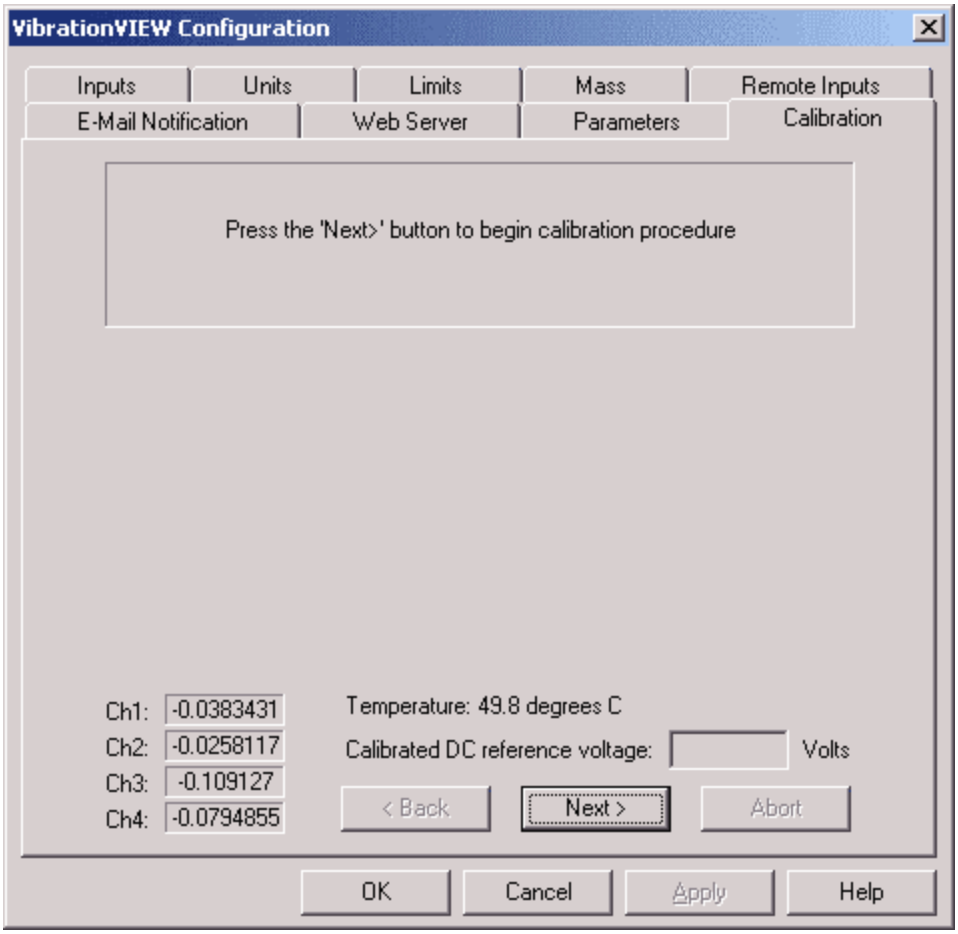
The Aux I/O Test button is used to test the auxiliary inputs and outputs on the rear of the 8500. This button is only available if you have purchased the Remote Inputs or Remote Outputs options. See Help..About *VibrationVIEW* to see a list of the options enabled on your controller.

See Also:

How to Install the Software

3.2.9 System Calibration command (Configuration menu)

This command opens the System Calibration tab on the *VibrationVIEW* Configuration box.



See Also:

How to calibrate the system

3.3 Test menu commands

The test menu is used to select, load, save, start, and stop tests. It offers the following commands:

Edit Test Settings	Edit current test settings.
Select Test Type	Select sine, random, shock etc. type test.
Run Test	Start running the current test.
Stop Test	Stop the current test.
Advance to next level	Skip to the beginning of the next level.
Test Schedule	Create a list of tests to run in sequence.
System Check	Switch to System Check mode.
Sine	Switch to Sine mode.
Random	Switch to Random mode.
Shock	Switch to Shock mode.
Field Data Replicator	Switch to Field Data Replicator mode.
Record	

...Record	Start recording input data.
...Stop	Stop recording input data and save to a file.
...Pause	Pause recording input data.
...Edit	Edit last recorded input data file.
...Settings	Setup recorder
Reset Filter Values	Reset the FDR filter values.
Random Profile from Ch2	Create a random spectrum from the Ch2 input.
Save Current Drive	Save the drive signal for rapid restart.
Create Standard Drive File	Save the drive signal for use with other tests.

3.3.1 Select Test Type command (Test menu)

Use this command to select the type of test you will be running. The *VibrationVIEW* Test Type dialog box will appear. The different test types are software options that are protected by a hardware key. If your hardware key does not have an option enabled, you will still be able to switch to that mode, define profiles and load and plot data sets, but you will not be able to run the test. Contact Vibration Research Corp. for information on options that are not enabled.

Shortcuts

Toolbar: 

3.3.2 Run Test command (Test menu)

Use this command to start the current test.

Shortcuts

Toolbar: 

Controls: 

Keys: F9

See also: Stop Test

3.3.3 Stop Test command (Test menu)

Use this command to stop any running test.

Shortcuts

Toolbar: 

Controls: 

Keys: F10

See also: Run Test

3.3.4 Advance to next level command (Test menu)

Use this command while a test is running to skip the remaining time in the current level, and to begin the next level. Levels are defined in the "Schedule..." option of the Edit Test dialog boxes.

Shortcuts

Keys: F11

3.3.5 Test Schedule command (Test menu)

Use this command to build a schedule of tests. To create a test schedule, first define the tests and then use this command to bring up the Test Schedule dialog box to add the tests to the test sequence.

The tests will be run in the defined order, with each test running for the duration specified within the individual test's schedule. There will be a pause of approximately 5 seconds between each test.

The schedule will be aborted if a test is shut down prior to completion of the schedule. (For example, if the operator presses the "Stop" button or a test encounters an error condition.)

3.3.6 System Check command (Test menu)

Use this command to switch to the System Check test mode.

3.3.7 Sine command (Test menu)

Use this command to switch to the Sine test mode.

3.3.8 Random command (Test menu)

Use this command to switch to the Random test mode.

3.3.9 Shock command (Test menu)

Use this command to switch to the Shock test mode.

3.3.10 Field Data Replicator command (Test menu)

Use this command to switch to the Field Data Replicator test mode.

3.3.11 Record command (Test menu)

This command starts recording real time input data to for data storage directory specified with the Recorder Buttons control bar.

Use View .. Recorder Buttons to display the Recorder Buttons control bar.

Shortcuts

Controls: 

Keys: Shift + F9

3.3.12 Record Stop command (Test menu)

This command stops recording real time input data and closes the current file.

Use View .. Recorder Buttons to display the Recorder Buttons control bar.

Shortcuts

Controls: 
Keys: Shift + F10

Record Pause (Test menu)

This command stops recording real time input data, but holds the input file open allowing the data storage to be resumed by pressing the run button..

Use View .. Recorder Buttons to display the Recorder Buttons control bar.

Shortcuts

Controls: 
Keys: Shift + F11

3.3.13 Record edit command (Test menu)

Edit command launches the *CoolEdit*2000 file editor, with the most recent recorded file.

Shortcuts

Controls: 

3.3.14 Record settings command (Test Menu)

Settings command displays the Record Settings dialog box. This allows channel selection, as well as selection of default record directory.

Shortcuts

Controls: 

3.3.15 Reset Filter Values command (Test menu)

Use this command to reset the filter values used by a Field Data Replicator test. Also use this command to reset the FDR controller back to its initial state in case the filter values are corrupted as a result of a loose or broken cable. In general, this command does not need to be used, because the filter will be automatically re-equalized every time a test is started.

3.3.16 Random Profile from Ch2 (Test menu)

This command creates a Random test spectrum from the channel 2 input. First input your test signal into the channel 2 input (this could be done while running a test). When the averaging is complete, select this menu option to transfer the spectral shape of the channel 2 waveform into the Edit Test dialog box. If it isn't already open, the Edit Test dialog box will automatically open and you can then adjust the test parameters and save the test.

Note: This is an optional component that must be programmed into your key. If you did not purchase this option, this command will be disabled in the menu.

Shortcuts
Keys: Ctrl+2

3.3.17 Save Current Drive command (Test menu)

Use this command to save the drive signal that is currently being output by the controller, and use it as the drive signal when starting the test. This avoids the need to re-equalize the system when starting the test, and is useful for starting a test quickly when the test conditions and amplifier gains are fixed.

The program will ask you if you want to save the new drive signal to the disk or use it for the current session only (reverting to the original settings the next time this test is opened). Generally you would want to click the "Yes" button to permanently save this drive signal with the test.

Procedure for using Memorized Drives:

1. Disable memorized drive for the test (edit the test, click the Schedule... tab and uncheck "Enable memorized drive").
2. Run the test until the drive signal has been well equalized.
3. Select Test..Save Current Drive. Click the "Yes" button to save these changes to the disk when prompted. This will store the current drive signal with the test and enable the "Enable memorized drive" option for this test.
4. The next time you run this test, it will start running with the memorized drive signal.

Notes:

- This command is available only in Random, Shock and Field Data Replicator test modes.
- To switch back to starting the test from a low level signal and equalizing the system on startup (i.e. disable the memorized drive function), remove the check mark next to the "Enable memorized drive" option in the Random Level Schedule, Shock Level Schedule or Field Data Replicator Level Schedule dialog box, as appropriate for your test.
- For random tests, see also Create Standard Drive for exchanging drive signals between different tests.

Shortcuts
Keys: Ctrl+M

3.3.18 Create Standard Drive File command (Test menu)

Use this command to save the current random drive signal to the standard drive file. The standard drive may then be imported into other tests to be used as a memorized drive signal.

The difference between this command and the Save Current Drive command is that the standard drive file is used to exchange drive signals between tests. The standard drive file is created using one test, and then imported into a second test to be used as a memorized drive in that test.

This command is only available for Random tests.

Procedure for using Standard Drive signals:

1. Run a test until the drive signal is well equalized.
2. Select the Test..Create Standard Drive File menu command.
3. Stop the test.
4. Open a different random test, or define a new random test.
5. Edit the test, and click the "Schedule..." button in the Random Define dialog box.
6. Click the "Start with standard drive" button in the Random Level Schedule dialog box. This will read the standard drive file, shape it to match the test profile, and save it to the

test's memorized drive signal. It will also enable the "Enable memorized drive" check box.

7. The next time you run this test, it will start running with the drive signal created from the standard drive file.

Notes:

- To switch back to starting the test from a low level signal and equalizing the system on startup (i.e. disable the memorized drive function), remove the check mark next to the "Enable memorized drive" option in the Random Level Schedule dialog box.
- The standard drive option is useful for getting a reasonable approximation to the drive signal required for a new test. However, the Save Current Drive command will get the memorized drive much closer because it will use a drive signal tailored to a particular test. Therefore, after starting and equalizing a new test using a standard drive signal, we recommend selecting the Save Current Drive menu command to save a more accurate memorized drive signal.

3.4 Graph menu commands

The graph menu is used to adjust the display of graphs and cursors. It offers the following commands:

Graph Colors	Set the default colors used by the graphs.
Graph Update Time	Change the graph update interval.
Pause Graph Updates	Pause/resume the automatic updating of the graphs
Edit Graph Settings	Change the lines and type of graph for the selected graph.
Y-Axis Autoscale	Autoscale only the vertical axis of the selected graph.
Full Autoscale	Autoscale both axes of the selected graph.
Autoscale All Graphs	
Refresh Graph	Redraw the selected graph.
Save Graph Layout	Save the current graph layout
Restore Graph Layout	Restore a previously saved graph layout
Reset Graph Data	Reset the Hi/Lo data for the current sine test, or PSD for Random Test.

3.4.1 Graph Colors command (Graph menu)

Use this command to bring up the Graph Colors dialog box. This dialog box may be used to customize the colors of your graphs. This will only affect graphs created after changing the colors; already open graphs will retain the old settings. To update the colors on any already-open graph, select the graph and press Ctrl+G (Edit Graph Settings command), and click the "OK" button.

3.4.2 Graph Update Time command (Graph menu)

Use this command to open the Graph Update Time dialog box. This dialog box may be used to set the number of milliseconds between graph updates. The default value is 1000 ms (i.e. 1 second).

3.4.3 Pause Graph Updates (Graph menu)

Use this command to manually pause or resume the automatic graph refreshes. This is most useful in Field Data Replicator mode because it allows the operator to closely examine the demand and control waveforms.

Shortcuts

Keys: Ctrl+P

3.4.4 Edit Graph Settings command (Graph menu)

Use this command to open the Graph Settings dialog box appropriate for the currently selected graph. You may then modify the type of graph, the traces displayed on the graph and the axis settings. Click the "OK" button to have the graph rebuilt with these new settings.

Shortcuts

Toolbar: 
Keys: Ctrl+G

See also:

- System Check Graph Settings dialog box
- Sine Graph Settings dialog box
- Random Graph Settings dialog box
- Shock Graph Settings dialog box
- Field Data Replicator Graph Settings dialog box

3.4.5 Y-Axis Autoscale command (Graph menu)

Use this command to automatically set the vertical (Y-axis) scale so that the data shown in the current plot fits on the display.

Shortcuts

Keys: Ctrl+A

See also: Full Autoscale

3.4.6 Full Autoscale command (Graph menu)

Use this command to automatically set both horizontal and vertical axis scales of the active graph so that the traces fit on the screen.

Note: You can zoom into a region of the graph by clicking with the right mouse button on one corner of the desired view, holding the button down while moving the mouse to enclose the desired region and then releasing the right mouse button.

Shortcuts

Toolbar: 
Keys: Ctrl+F
Mouse: Double-click the right mouse button

See also: Y-Axis Autoscale

3.4.7 Autoscale All Graphs command (Graph menu)

Use this command to automatically set both horizontal and vertical axis scales of all graphs so that the traces fit on the screen.

Shortcuts

Keys: Ctrl-Q

See also:
Y-Axis Autoscale
Full Autoscale

3.4.8 Refresh Graph command (Graph menu)

Use this command to force a refresh of the active graph.

Shortcuts
Keys: F5

3.4.9 Save Graph Layout command (Graph menu)

Use this command to save the graph arrangement from the screen to a disk file. Use Restore Graph Layout to restore the screen layout.

3.4.10 Restore Graph Layout command (Graph menu)

Use this command to restore the graph arrangement to the screen from a disk file. Use Save Graph Layout to create this disk file.

3.4.11 Reset Graph Data command (Graph menu)

Use this command to reset the peak- and valley-hold data for a Sine graph to the demand levels. The data is cleared automatically when a sine test is started. Also, depending on the test settings, the hi/lo data can be cleared automatically after auto-saving data. (See the Sine Data Storage settings.)

Use this command to clear the retained time history data for Random and Field Data Replicator time history graphs.

3.5 Cursor Menu commands

This menu is used to configure cursors. It offers the following commands:

Display Cursor Window	Toggle the cursor window on and off
Set Base Cursor	Set the fixed cursor at the current mouse position
Next Differential Cursor	Cycle through the various cursor modes
Prev Differential Cursor	Cycle backwards through the various cursor modes
Standard Cursor	Display a window with the data at the current cursor location
Delta Cursor	Display the difference between two cursors
Slope Cursor	Display rate of change between two cursors
Minimum Cursor	Display minimum value between two cursors
Maximum Cursor	Display maximum value between two cursors
Mean Cursor	Display mean value between two cursors
Dual Cursor	Display values at each of two cursors
RMS Cursor	Compute the RMS value between two cursors
Add Annotation	Put a graph annotation at the current cursor location
Move Annotation	Move the currently selected graph annotation
Remove Annotation	Remove the currently selected graph annotation
Remove All Annotations	Remove all annotation strings from the active graph

Tips:

- To use the dual cursor modes (Delta, Slope, Minimum, Maximum, Mean, Dual and RMS) move the cursor the FIRST cursor location.
- Use the <Home> key to lock the cursor in this position.
- Holding down the <Shift> key while moving the mouse snaps the cursor to the nearest peak. Pressing the <Home> key while holding down the <Shift> key locks the first cursor at the location of the peak.
- Holding down the <Ctrl> key while moving the mouse snaps the cursor to the nearest valley. Pressing the <Home> key while holding down the <Ctrl> key locks the first cursor at the location of the valley.
- After locking the FIRST cursor, use the mouse to move the primary cursor. The calculated values between the two cursors are displayed in the cursor window (depending on the cursor mode).
- Use the <Home> key at any point to move the FIRST cursor to the current primary cursor location.
- Use <PgUp> and <PgDn> to cycle through the cursor modes.
- Use annotations to display multiple cursor intervals on the screen. Annotations will display any of the special mode cursors and update on live graphs.

3.5.1 Cursor Window command (Cursor menu,View Menu)

Use this command to show or hide the Cursor Display box that shows the values for the graph traces at the current cursor position.

Holding down the <Shift> key while moving the cursor with the mouse snaps the cursor to the nearest peak.

Holding down the <Ctrl> key while moving the cursor with the mouse snaps the cursor to the nearest valley.

Shortcuts

Keys: Ctrl+D

3.5.2 Set Base Cursor command (Cursor menu)

Use this command to lock the dual cursor at the current cursor position. The primary cursor then moves freely with the mouse.

Holding down the <Shift> key while moving the cursor with the mouse snaps the cursor to the nearest peak.

Holding down the <Ctrl> key while moving the cursor with the mouse snaps the cursor to the nearest valley.

Shortcuts

Keys: <Home>

3.5.3 Next Differential Cursor command (Cursor menu)

Use this command to cycle the dual cursor to the next cursor mode.

The cursor modes are:

Standard Cursor	Display a window with the data at the current cursor location
Delta Cursor	Display the difference between two cursors
Slope Cursor	Display rate of change between two cursors
Minimum Cursor	Display minimum value between two cursors
Maximum Cursor	Display maximum value between two cursors
Mean Cursor	Display mean value between two cursors
Dual Cursor	Display values at each of two cursors
RMS Cursor	Compute the RMS value between two cursors

Shortcuts

Keys: <PgUp>

3.5.4 Previous Differential Cursor command (Cursor menu)

Use this command to cycle the dual cursor to the previous cursor mode.

The cursor modes are:

Standard Cursor	Display a window with the data at the current cursor location
Delta Cursor	Display the difference between two cursors
Slope Cursor	Display rate of change between two cursors
Minimum Cursor	Display minimum value between two cursors
Maximum Cursor	Display maximum value between two cursors
Mean Cursor	Display mean value between two cursors
Dual Cursor	Display values at each of two cursors
RMS Cursor	Compute the RMS value between two cursors

Shortcuts

Keys: <PgDn>

3.5.5 Standard Cursor command (Cursor menu)

Display a window with the data of the current cursor location.

Shortcuts

Keys: <End>

3.5.6 Delta Cursor command (Cursor menu)

Display the difference between two cursors.

3.5.7 Slope Cursor command (Cursor menu)

Display rate of change between two cursors.

3.5.8 Minimum Cursor command (Cursor menu)

Display minimum value between two cursors.

3.5.9 Maximum Cursor command (Cursor menu)

Display maximum value between two cursors.

3.5.10 Mean Cursor command (Cursor menu)

Display mean value between two cursors.

3.5.11 Dual Cursor command (Cursor menu)

Display values at each of two cursors.

3.5.12 RMS Cursor command (Cursor menu)

Compute the RMS value between two cursors.

3.5.13 Add Annotation command (Cursor menu)

Use this command to annotate the current graph with the trace values at the current cursor position. To display a vertical cursor bar to help with the selection of an insertion point, turn on the cursor display using the Graph..Cursor Display menu command.

Tips:

- To accurately place an annotation, first use the right mouse button (click, drag, release) to zoom in on the region of interest. Add the annotation by positioning the cursor and pressing the <Insert> key. Zoom out by double-clicking the right mouse button.
- To move an annotation, click on the annotation with the left mouse button, hold the button down, move the mouse to the new location, and release the left mouse button.
- Holding down the <Shift> key snaps the cursor to the nearest peak. Pressing the <Insert> key while holding down the <Shift> key puts an annotation at the location of the peak.
- Holding down the <Ctrl> key snaps the cursor to the nearest valley. Pressing the <Insert> key while holding down the <Ctrl> key puts an annotation at the location of the valley.
- After creating an annotation you can double-click it (using the left mouse button) to edit the text.

See also: Cursor Display, Remove Annotation, Move Annotation

Shortcuts

Keys:	Insert	
	F8	
	Shift-Insert	Insert annotation at the nearest peak
	Ctrl-Insert	Insert annotation at the nearest valley

3.5.14 Move Annotation command (Cursor menu)

Use this command to change the position of the currently selected annotation. First click on the annotation text (a black rectangle will briefly encircle the text), and then use this command to start the move. Move the mouse pointer to the desired location for the annotation, and click the left mouse button to complete the move.

Tip: You can also move an annotation by pointing to the annotation text with the mouse, clicking the left mouse button, holding that button down while moving the mouse cursor to the desired location and releasing the button to complete the move.

See also: Add Annotation, Remove Annotation

Shortcuts

Mouse:	Click and drag the annotation
Keys:	F4

3.5.15 Remove Annotation command (Cursor menu)

Use this command to remove the currently selected annotation. First click on the annotation (a black rectangle will briefly encircle the text) and then use this command to remove the annotation.

See also: Add Annotation, Move Annotation

Shortcuts

Keys:	Delete
	F7

3.5.16 Remove All Annotations command (Cursor menu)

This command will remove ALL annotations previously placed on the graph.

3.6 View menu commands

The View menu offers the following commands:

Toolbar	Show or hide the toolbar.
Status Bar	Show or hide the status bar.
ToolTips	Enable or disable display of tool tips in the toolbars
Toolbar Text	Enable or disable button captions in the toolbars
Control buttons	Show or hide the active Control Center
Accelerometers	Toggle display of the accelerometers toolbar
Motor Controls	Display motor controls for optional motor simulation module.
Cursor Window	Show the cursor information dialog box.
Sine Big Display	Display the sine frequency and amplitude information with large numbers.
Sine COLA Slip	Adjust the sine COLA slip frequency.
Reset Control buttons	Reset the control window to the default location

3.6.1 Toolbar command (View menu)

Use this command to show or hide the Toolbar. A check mark appears next to the menu item when the Toolbar is shown.

See Toolbar for help on using the toolbar.

3.6.2 Status Bar command (View menu)

Use this command to show or hide the Status Bar. A check mark appears next to the menu item when the Status Bar is shown.

See Status Bar for help on using the status bar.

3.6.3 Tool Tips command (View menu)

Use this command to enable or disable display of tool tips in the toolbar and the Control Centers. A tool tip is a small window that displays context sensitive help information when you move the mouse cursor over a button on the toolbar. A check mark appears next to this menu item when tool tips are enabled.

3.6.4 Tool Bar Text command (View menu)

Use this command to turn the tool bar button captions on or off. When text is off, only the graphical icons are displayed on the toolbar.

3.6.5 Control buttons command (View menu)

Use this command to show or hide the Control Centers. A check mark appears next to the menu item when the Control Center is shown.

Shortcuts
Keys: F2

See the following for help on the various Control Centers:

- System Check
- Sine
- Random
- Classical Shock
- Field Data Replicator

3.6.6 Accelerometers command (View menu)

Use this command to show or hide the accelerometers toolbar. A check mark appears next to the menu item when the status bar is shown.

Shortcuts
Keys: F3

See the following for help on the various status bars:

- System Check
- Sine
- Random
- Classical Shock
- Field Data Replicator

3.6.7 Recorder command (View menu)

Use this command to show or hide the Recorder control center. The data recorder an optional package.

3.6.8 Motor Control command (View menu)

Use this command to show or hide the Motor Controls toolbar Motor controls are used for the optional motor simulator package.

3.6.9 Sine Big Display command (View menu)

Use this command to show or hide the Sine Big Display dialog box that displays the frequency and amplitude parameters in numbers large enough to be seen from a distance. This is useful when examining a product on the shaker for resonance.

The dialog box also provides a feature for remote control (using the mouse) of the sweep direction. When used with a mouse extension cable, the sine frequency may be controlled from a distance.

This display is available only in Sine test mode.

Shortcuts

Keys: Ctrl+B

3.6.10 Sine COLA Slip command (View menu)

Use this command to bring up the Channel 2 Slip Frequency that may be used to vary the channel 2 output frequency shift.

This command is only available in Sine test mode.

3.6.11 Reset Control buttons command (View menu)

Use this command to reset the size and position of the Control Center. Use this command if you accidentally move the Control Center to the edge of the screen and aren't able to retrieve it.

3.7 Window menu commands

The Window menu offers the following commands that allow you to arrange multiple views of multiple documents in the application window:

New Window	Create a new window that duplicates the active graph.
Cascade	Arrange windows in an overlapped fashion.
Tile Horizontal	Arrange windows in non-overlapped horizontal tiles.
Tile Vertical	Arrange windows in non-overlapped vertical tiles.
Arrange Icons	Arrange icons of minimized windows.
Window 1, 2, ...	Go to specified window (graph).

3.7.1 New Window command (Window menu)

Use this command to open a new window with the same data as the active window. You can open multiple document windows to display different views of a data set at the same time. If you change the contents in one window, all other windows based on the same data set reflect those changes. When you open a new window, it becomes the active window and is displayed on top of all other open windows.

3.7.2 Cascade command (Window menu)

Use this command to arrange multiple open windows in an overlapped fashion.

3.7.3 Tile Horizontal command (Window menu)

Use this command to arrange multiple open windows in a non-overlapping fashion with the graphs stacked one above the other.

3.7.4 Tile Vertical command (Window menu)

Use this command to arrange multiple open windows in a non-overlapping fashion with the graphs placed one beside the other.

3.7.5 Arrange Icons command (Window menu)

Use this command to arrange the icons for minimized windows at the bottom of the main window. If there is an open document window at the bottom of the main window, then some or all of the icons may not be visible.

3.7.6 1, 2,..command (Window menu)

VibrationVIEW displays a list of currently open document windows at the bottom of the Window menu. A check mark appears by the document name of the active window. Choose a document from this list to make its window active.

3.8 Help menu commands

The Help menu offers the following commands that provide assistance with *VibrationVIEW*:

Help	Open the help file to the overview page
Contents and Index	Open an index of topics for which you can get help
Context Help	Use this to get context-sensitive help information.
Enter Registration	Use this to register the optional File Viewer and Simulation/Demo modes
About *VibrationVIEW*...	Display application information and update key.

3.8.1 Help command (Help menu)

Use this command to display the opening screen of Help. From the opening screen, you can jump to step-by-step instructions for using *VibrationVIEW* and various types of reference information.

Once you open Help, you can click the "Contents" button to return to the opening screen.

Shortcut

Toolbar: 
Keys: F1

3.8.2 Contents and Index command (Help menu)

Use this command to open the help file's table of contents, index, and search functions. You can browse through or search the help file for information on a particular subject.

3.8.3 What's This? command (Help menu)

Use the Context Help command to obtain help on some portion of *VibrationVIEW*. When you choose the Toolbar's Context Help button, the mouse pointer will change to an arrow and question mark. Click somewhere in the *VibrationVIEW* window, such as another Toolbar button or menu item. The Help topic for the item you clicked will be shown.

Shortcut

Toolbar: 
Keys: Shift+F1

3.8.4 Enter Registration Code (Help Menu)

Display the Enter Registration dialog box. Registration is not required for normal system operation, but is necessary to enable the optional file viewer and demonstration modes.

See also:

How to .. Simulation Demo
How to .. File Viewer

3.8.5 About command (Help menu)

Use this command to display the copyright notice and version number of *VibrationVIEW*.

This dialog box also lists the key serial number and the installed packages.

This dialog box also provides the "Update Key" button used to install software key updates that unlock additional packages. To update the key settings press the "Update Key" button and search for the file with the extension .vrckey provided by Vibration Research. Press the "Open" button to install the key update.

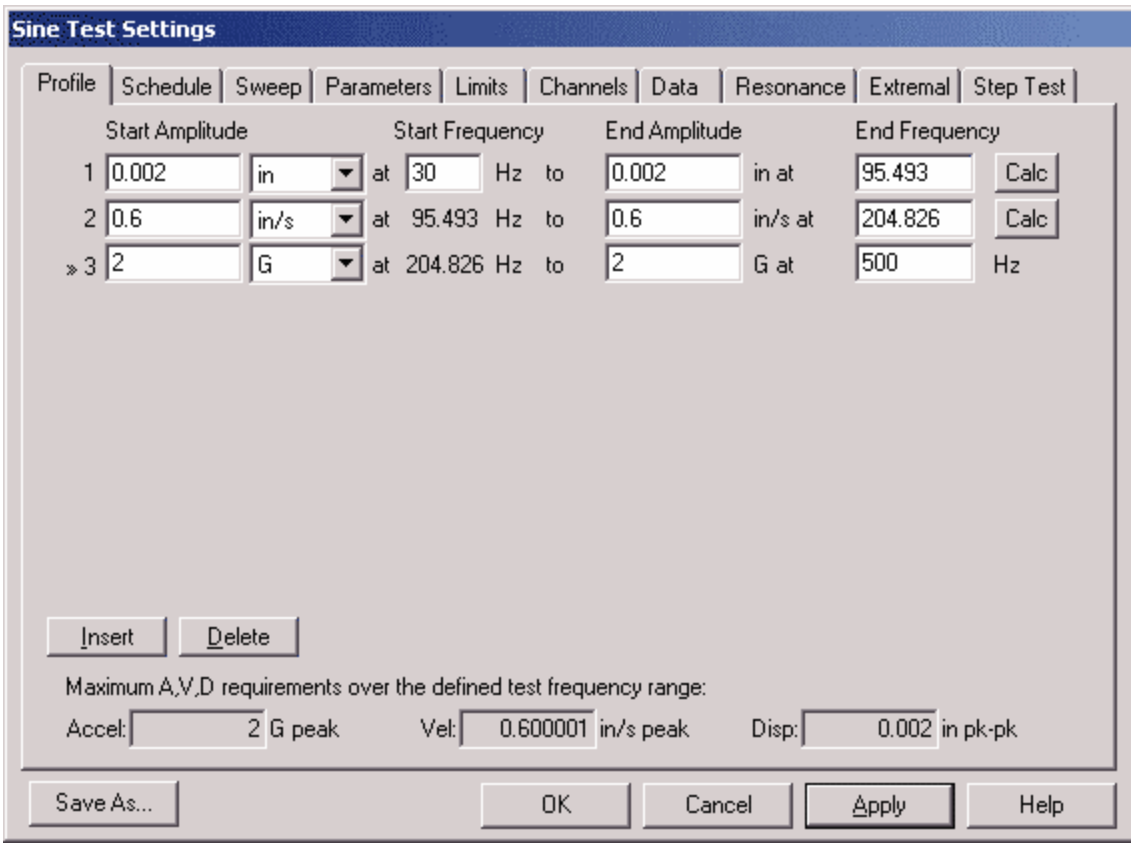
4 Test Definition

Test Definition dialogs

- Sine Test Definition
- Random Test Definition
- Classical Shock Test Definition
- Field Data Replicator Test Definition

4.1 Sine Define dialog box

Selecting the Test..Edit Test menu command while a Sine test is open displays this dialog box.



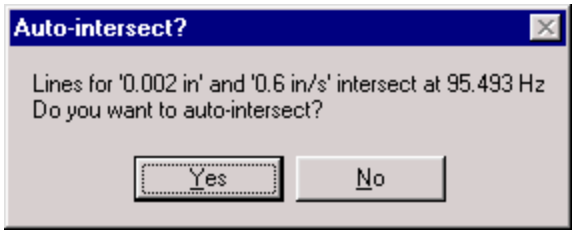
The ***Start*** and ***End Frequency*** parameters define the frequency span of the segment. You selects the first ***Start Frequency*** and the other ***Start Frequencies*** follow from the previous ***End Frequency***. The ***Start*** and ***End Amplitude*** parameters set the starting and ending points for each segment and do not need to be constant for a given segment or continuous at the segment endpoints.

The ***Units*** parameter selected for the ***Start Amplitude*** defines what type of control to use for each level. For example, if "in" (inches) is selected, displacement will be controlled, if "in/sec" is selected, velocity will be controlled or if "G" is selected, acceleration will be controlled. Therefore, if you need a constant displacement section, set the units selection to a displacement unit (e.g. "in").

If the frequency sweep is linear (Hz/min), ramped amplitude sweeps will be straight lines on linear graphs. If the frequency sweep is logarithmic, ramped amplitude sweeps will be straight lines on log-log graphs.

To have the program automatically compute the frequency at which two constant-amplitude lines intersect, click the "Calc" button next to the ***End Frequency*** parameter that needs to be computed. For example, to find the intersection between 0.002 in displacement and 0.6 in/s velocity:

1. In the first segment, enter 0.002 for the ***Start Amplitude*** and select "in" for the units
2. Set the starting frequency (e.g. 30 Hz)
3. In the second segment, enter 0.6 for the ***End Amplitude*** and select "in/s" for the units.
4. Enter the second ***End Frequency*** (e.g. 200 Hz)
5. Click the "Calc" button next to the first ***End Frequency*** parameter. A dialog box will appear:



6. Click the "Yes" button to automatically select this frequency as the end frequency of the first segment and to set the two lines to constant displacement and constant velocity, respectively.

The "Insert" and "Delete" buttons allow you to add and remove segments in the breakpoint table. The currently selected segment (indicated by the small arrow on the left side of the dialog box) will be deleted when the "Delete" button is pressed. Inserted segments will immediately follow the currently selected segment.

At the bottom of the dialog box are the maximum acceleration, velocity and displacement requirements for the test. These boxes will be highlighted in red if the test exceeds the System Limits and in yellow if the test is within 5% of the system limits.

The tabs along the top of the dialog box are used to select the following parameter entry screens:

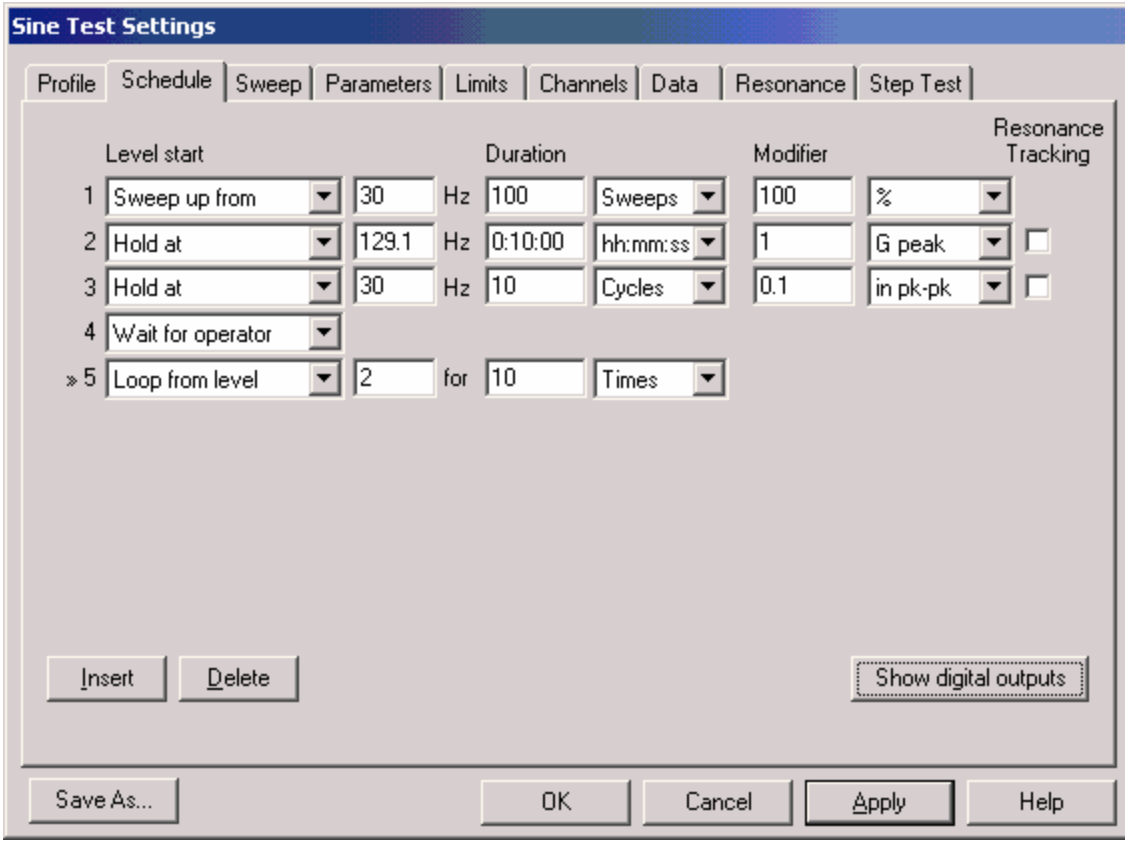
Profile	Enter the frequency and amplitude breakpoints.
Schedule	Set test duration and amplitude scaling level.
Sweep	Enter sine sweep rates
Parameters	Enter control loop parameters.
Limits	Enter the safety limits.
Channels	Select tracking filters and active control channel.
Data Storage	Select automatic data storage parameters.
Resonance	Enable/disable the Resonance Search feature.
Extremal	Enter Extremal (extreme conditions, AVD) parameters. (optional module)
Step Tests	Enter sine step test parameters. (optional module)

The buttons at the bottom of the dialog box perform the following operations:

Save As...	Save the current settings under a new test name.
OK	Save the changes to this test and close the dialog box
Cancel	Close the dialog box and abandon all changes
Apply	Apply the changes to the currently open test (does not save changes to the hard disk).
Help	Get help information about these parameters.

4.1.1 Sine Define Schedule tab

Clicking the "Schedule" tab on the Sine Define dialog box accesses this dialog box.

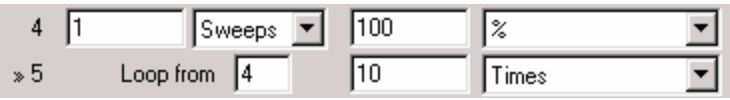


Use this dialog box to define how long your test will sweep at each level or how long the test will run at each fixed-frequency. Use the drop-down box to select the type of duration measurement: number of

sweeps, number of sine wave cycles or the length of time. Then enter the ***Duration*** parameter in the ***Duration*** column.

The ***Level Start*** parameter defines the frequency at which the sweep will begin, whether it will start sweeping up or down from this value and the rate at which the sweep will run. To enter a fixed-frequency test, define one or more levels at fixed frequencies using the Modifier type (for example, the third segment in the dialog box shown above will run 10 Cycles at 0.1 in pk-pk at a fixed frequency of 30 Hz).

The ***Sweeps*** parameter defines how many sweeps will be run at the specified level. One sweep will be counted down every time the test hits the end of the sweep and turns around. The frequency will sweep back-and-forth between the minimum and maximum frequencies. Note: If you want to repeat a sweep in the same direction every time, set the desired direction on the Profile tab of this dialog box and then set two consecutive test levels as follows:



This will produce 10 consecutive sweeps in the same direction.

The ***hh:mm:ss*** parameter defines the amount of time (using the format of hours, minutes and seconds separated by colons) for the indicated level to run. When entering time parameters, you may use formats such as "10 hours" or "1h30m" or "100m" as shortcuts. The program will automatically convert these to the hh:mm:ss format. This type of setting may be used for both sweep and fixed-frequency levels.

The ***Cycles*** parameter defines how many cycles of a sine wave will be counted while the test runs at the specified level. Remember, 100 Hz = 100 cycles/second, so a level running at a fixed 100 Hz tone for 12000 cycles will be 120 seconds long. This type of setting may be used for both sweep and fixed-frequency levels.

The modifier type can be set to one of the following values:

Modifier	**Level type**	**Amplitude**
%	Sweep	Scale the defined profile by the specified percentage
dB	Sweep	Scale the defined profile by the specified dB level, where +3dB increases the level by a factor of 2, and -3dB decreases the level by a factor of 2.
x amp	Sweep	Multiply the defined profile by the specified scale factor
% at	Fixed	Scale the defined profile by the specified percentage
Accel. pk at	Fixed	Use the specified acceleration amplitude (0-to-peak) at the specified frequency
Velocity pk at	Fixed	Use the specified velocity amplitude (0-to-peak) at the specified frequency
Disp. pk-pk at	Fixed	Use the specified displacement amplitude (peak-to-peak) at the specified frequency
Times		Repeat part of the test schedule the specified number of times
Wait for operator		Wait for the operator to press the "Continue" button to resume the test.
Resonance Table		Automatically detect the Ch2/Ch1 transmissibility peaks, and allow the operator to select time to test at each peak.

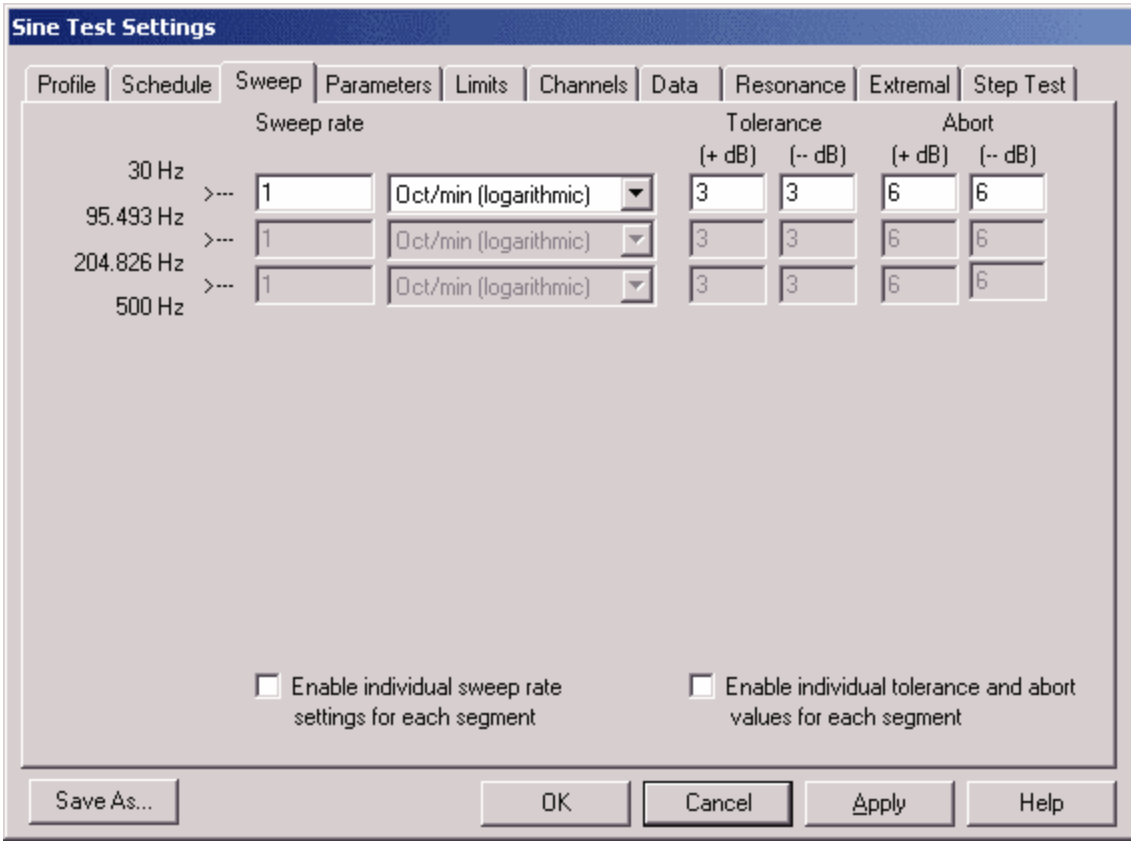
Note: Use of the Resonance Table or the Phase Tracking settings requires that the (optional) Sine Search-and-Dwell package be programmed into the software key. See the section on the Select Resonance Frequencies dialog box for more details on the use of these settings.

To repeat part of the test schedule a number of times, insert a level immediately following the level(s) you want to repeat and set its modifier type to "Times". Enter the level number of the first level in the loop in the "Loop from" entry and the total number of passes to perform in the "Modifier" column. For example, the entries in the above dialog box will run in the order 1,2,3,4,2,3,4,2,3,4,..,2,3,4 (where levels 2, 3, and 4 are each run a total of 10 times). Loops can be nested up to 10 levels deep to create more complex repeating patterns.

A loop pointing to a resonance table will repeat only the tones in that table and the Select Resonance Frequencies will appear only on the first pass. If you want to select resonance frequencies on every pass, make the "Loop to" point to a sweep level immediately before the "Resonance Table" level. Then each pass of the loop will run a sweep followed by the dialog box to select resonance frequencies, followed by the selected fixed-frequencies.

4.1.2 Sine Define Sweep tab

Clicking the "Sweep" tab in the Sine Define dialog box accesses this dialog box.

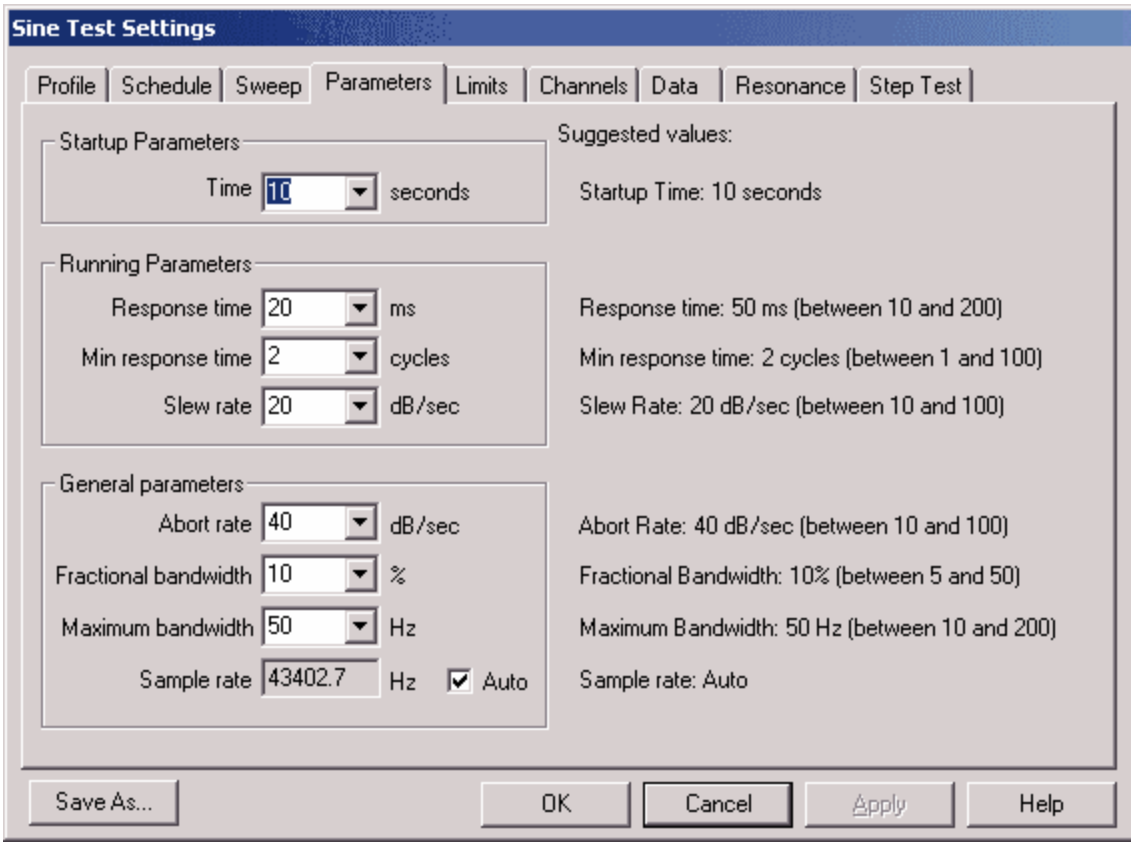


Use this dialog box to define your sweep rate. Enter the rate in the ***Sweep rate*** box and select the desired units from the drop-down list.

If a fixed frequency is defined (see the sweep functions below), and number of sweeps is selected, the controller will calculate how long the test must run to complete the number of cycles (sweeps) specified. Example: at 10 Hz fixed frequency test, a 1000 "sweep" test would run for 1000/10=100 seconds.

4.1.3 Sine Define Parameters tab

Clicking the "Parameters" tab in the Sine Define dialog box accesses this dialog box.



Use this dialog box to define the control loop parameters. These numbers determine how the control algorithm reacts.

The ***Startup Time*** parameter defines the approximate amount of time the controller will take to go from no output to the desired level. Having a long ***Startup Time*** (e.g. 10 to 20 seconds) is a good safety precaution because it allows the operator more time to abort the test if he sees that the shaker is moving but the input signals are not registering any acceleration (for example, when an accelerometer cable is not connected.)

The **Response Time** *and* **Min Response Time** *parameters are the primary loop tuning parameters.*
The ***Response Time*** parameter defines high frequency loop response. Lower response times provide faster control response. Higher response times provide loop stability.

The ***Min Response Time*** parameter defines low frequency loop response. If the number of cycles specified in ***Min Response Time***, at a given frequency, is greater than the specified response time, it determines loop response.

The ***Slew Rate*** parameter defines the maximum slew rate during normal operation.

The ***Fractional Bandwidth*** parameter defines the bandwidth of the tracking filters at low frequencies. It is specified as a percentage of frequency.

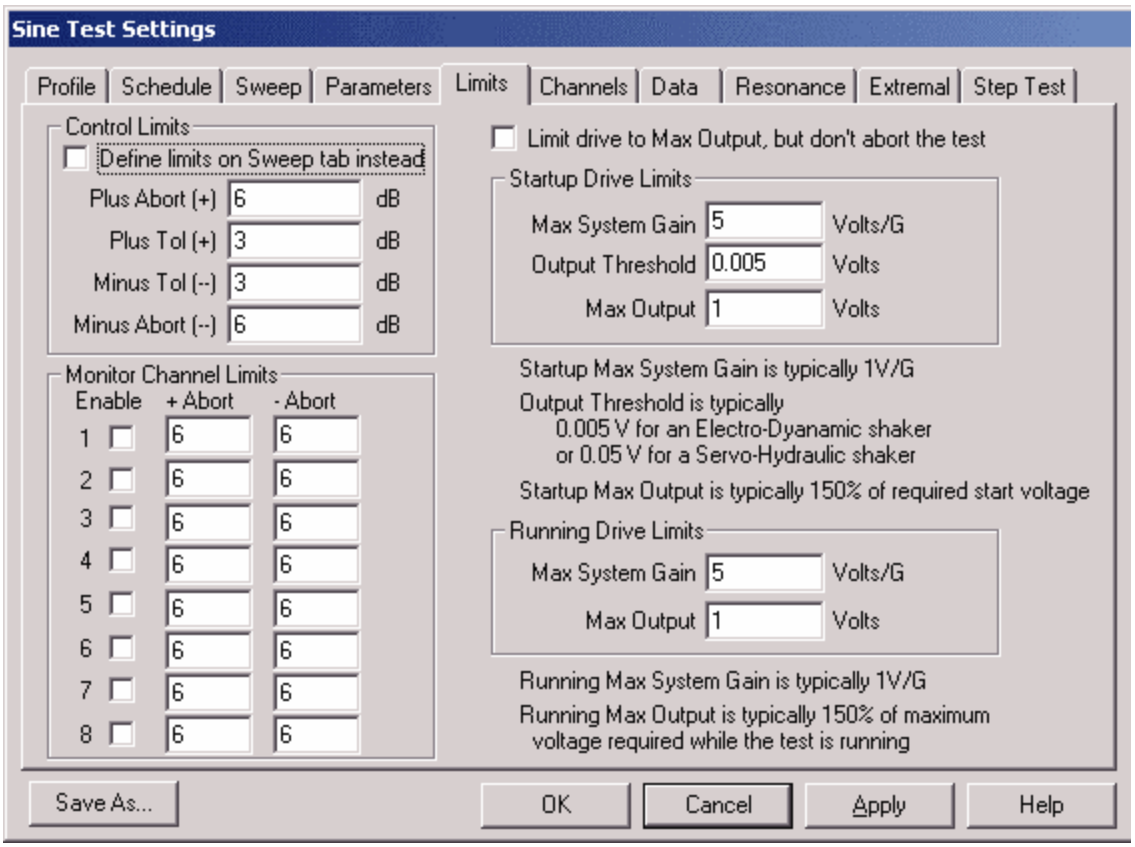
The ***Maximum Bandwidth*** parameter defines the bandwidth of the tracking filters at high frequencies. The ***Fractional Bandwidth*** is not allowed to exceed the ***Maximum Bandwidth***.

The ***Sample Rate*** parameter should in nearly all cases be set to Auto to allow the controller to select an appropriate sample rate. However, experienced users have the option of selecting the sample rate used by the controller.

See also: How to tune Sine controller parameters.

4.1.4 Sine Define Limits tab

Clicking the "Limits" tab on the Sine Define dialog box accesses this dialog box.



Use this dialog box to define the upper/lower abort/tolerance limits for the control channel, the upper/lower limits for the monitor channels and the upper limits on the starting and running drive signals.

The ***+/- Tol* (dB)** parameters define the number of decibels the tolerance lines will be above (+) and below (-) the desired acceleration level. These lines are typically drawn as brown lines on the graphs. The tolerance levels are used to determine when the control switches from startup mode to run mode (i.e. when the frequency sweep begins and the timers begin counting).

The ***+/- Abort (dB)*** parameters define the number of decibels the abort lines will be above (+) and below (-) the desired acceleration level. These lines are typically drawn as red lines on the graphs. If the controlled accelerometer ever exceeds these lines, the test will abort.

The ***Monitor Channel +/- Abort*** parameters define the number of decibels the monitor channels are allowed to be above (+) or below (-) the desired acceleration level. These abort lines may be enabled individually for each input channel by selecting the check box next to the channel number. When one of the monitor channels exceeds these limits (and that channel has the abort limits enabled), the test will abort.

The startup ***Max System Gain*** parameter defines the maximum allowed ratio of output volts to measured acceleration while a test is starting. Setting this value to a large number (e.g. 100 volts/G) disables this limit. This safety limit prevents the drive output from increasing unless some response is measured on the accelerometer. Refer to "How to Tune Sine Controller Parameters" for details on how to determine a suitable value for your system.

The startup ***Output Threshold*** parameter defines the base output voltage on the control loop when a test is starting.

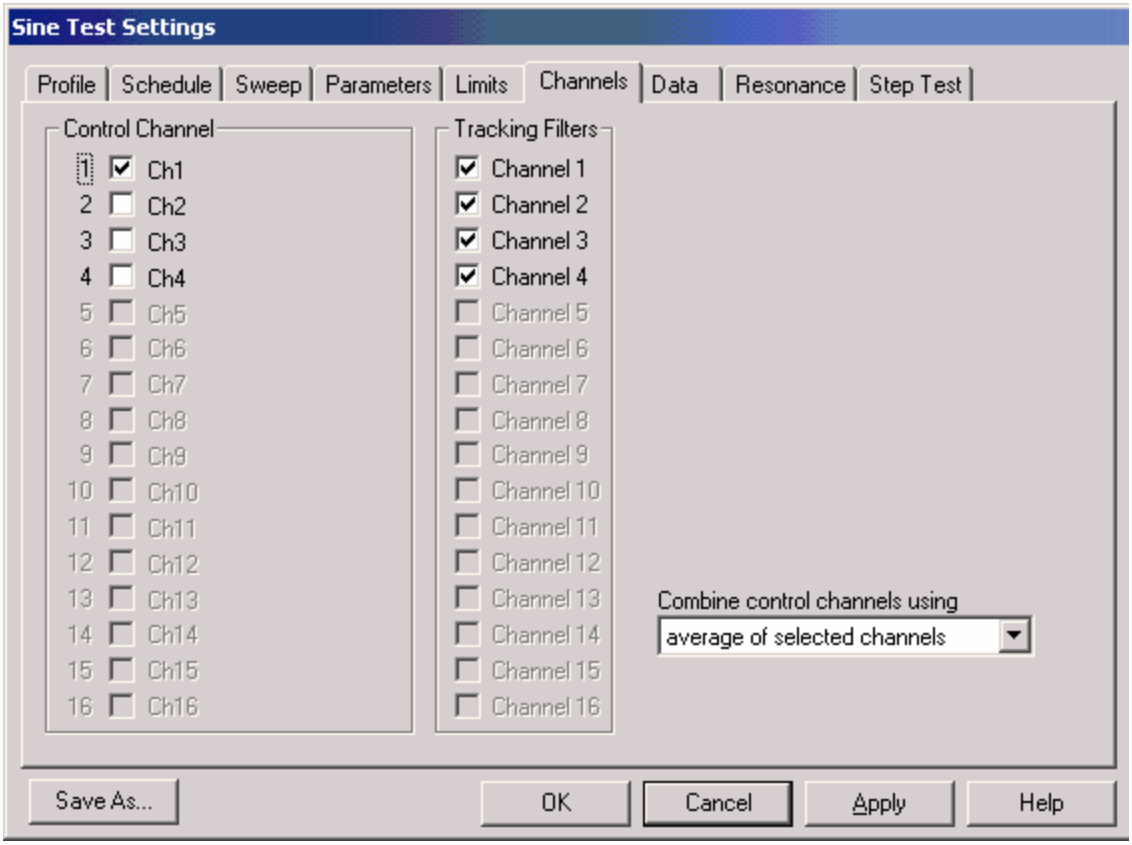
The startup ***Max Output*** parameter defines the maximum output voltage allowed from the control loop when a test is starting. If the channel 1 drive signal output exceeds this value when starting a test, the test will abort. We recommend setting this at about 150% of the normal start up voltage required.

The running ***Max System Gain*** parameter defines the maximum allowed ratio of output volts to measured acceleration while a test is running. Setting this value to a large number (e.g. 100 volts/G) disables this limit. This is a safety limit that restricts the drive output from increasing unless some response is measured on the accelerometer. Refer to "How to Tune Sine Controller Parameters" for details on how to determine a suitable value for your system.

The running ***Max Output*** parameter defines the maximum output voltage allowed from the control loop while the test is in Run mode. If the channel 1 output ever exceeds this value, the test will abort. We recommend setting this at about 150% of the normal running maximum.

4.1.5 Sine Define Channels tab

Clicking the "Channels" tab in the Sine Define dialog box accesses this dialog box.



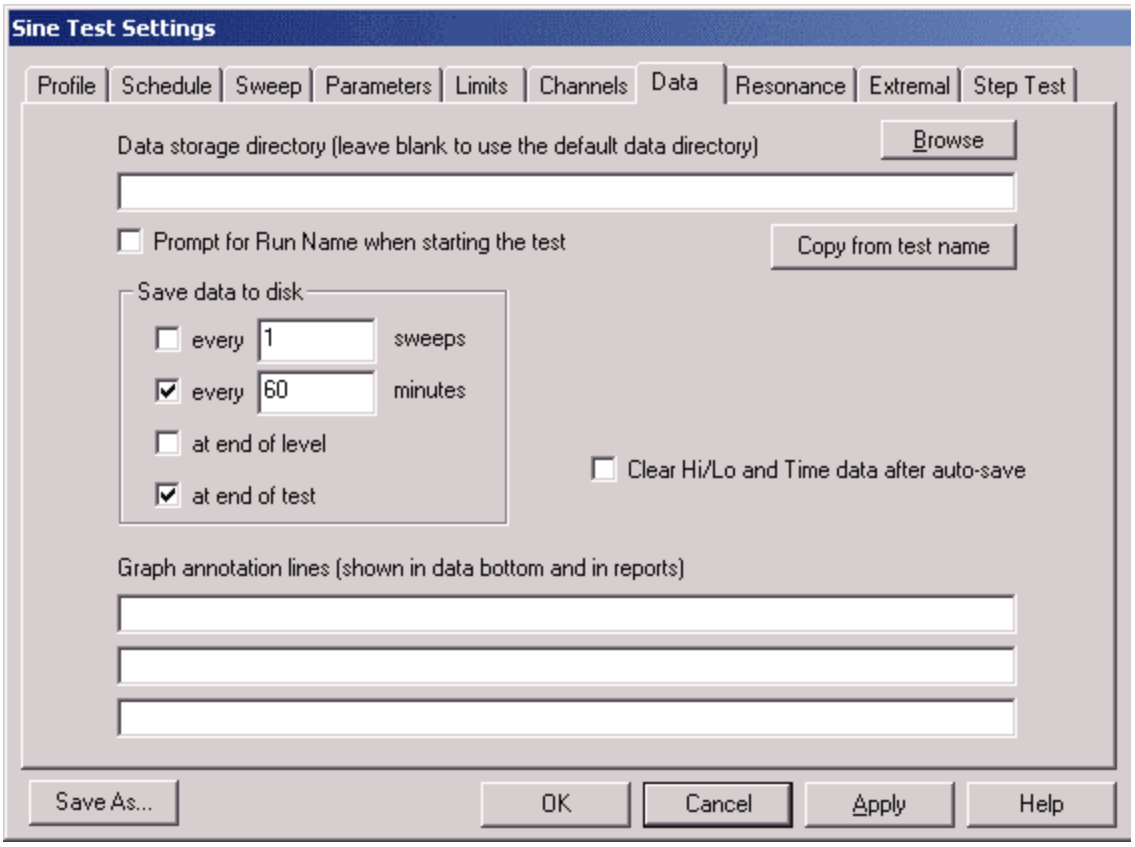
Use this dialog box to define the input channel(s) used for sine control, and also to enable/disable the tracking filters for both the control signal and the other input channels. The input channels must include either channel 1 or 2 and may include any further combinations of the available channels. Channels may be combined by averaging the amplitudes together (standard mode) or by using the largest of the selected input channels (extremal mode). The extremal mode is an optional feature; the software key must be programmed to enable it.

Note: a tracking filter is a filter that extracts only the frequency of interest from the measured signal. This is useful for controlling in low signal-to-noise ratio environments and also for removing the harmonics from the measured response (for example, if the shaker is producing a distorted sine wave).

When input channel 2 is selected as the control channel the Sine System Limits are checked only against **channel 1**. This is useful for situations where channel 1 is measuring the acceleration of the shaker head, but the acceleration to be controlled (channel 2) is near the end of a highly resonant beam. The end of the beam may see accelerations that are much higher than those measured on the shaker head, but the system limits only apply to the accelerometer on the shaker head.

4.1.6 Sine Define Data Storage tab

Clicking the "Data" tab on the Sine Define dialog box accesses this dialog box.



Use this dialog box to automatically save data to disk. You can save the data at the end of a level, at the end of a test, at the end of a sweep and/or at the end of a time period. All data is saved as a time stamped file, and can be viewed at a later time open the data file(s) by selecting the File..Open Date menu command or by clicking the Open Data button on the toolbar.

You may also select the directory into which to store the data. Click the "Copy from test name" button to automatically create an output directory based on the name of the current test. Check "Prompt for Run

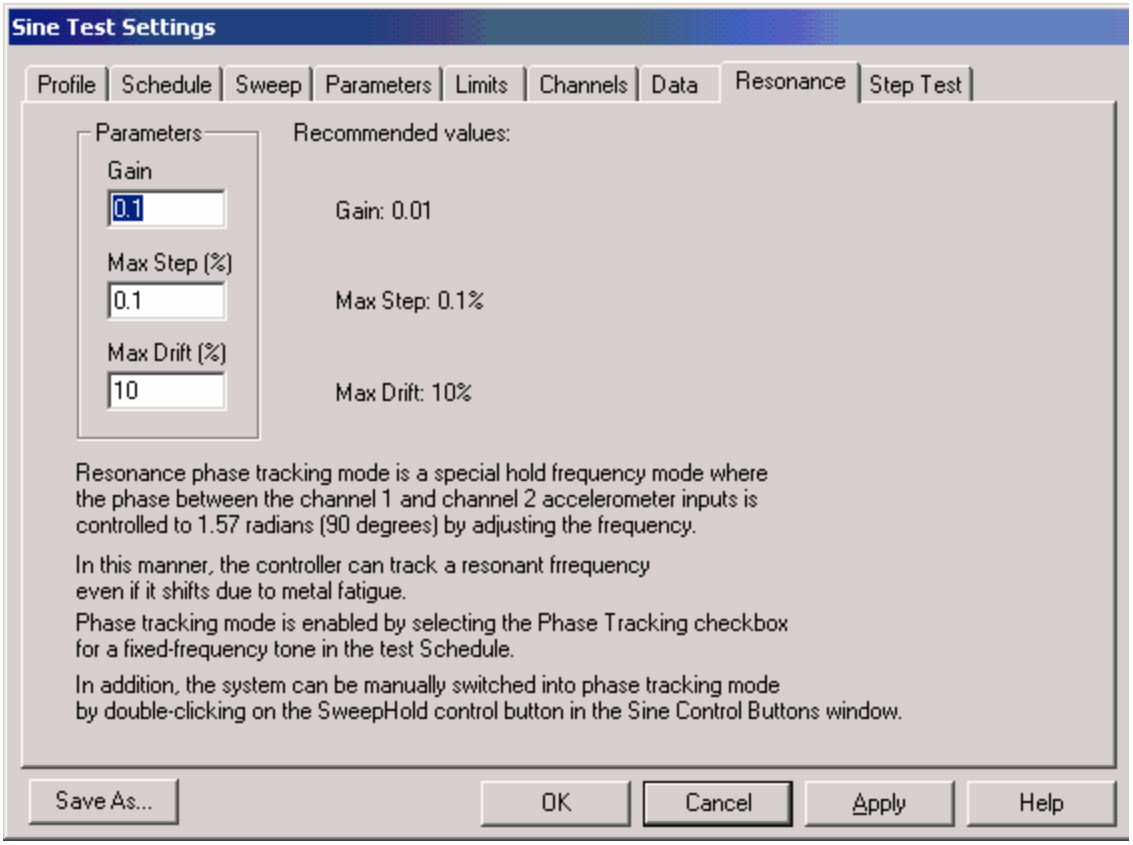
Name when starting the test" to display a prompt dialog allowing the operator to enter a new directory each time the test is started.

The Hi/Lo peak hold values may be automatically cleared after the data has been stored so that the Hi/Lo lines for each data set will indicate the extreme values seen between each stored data set.

The Graph annotation lines are displayed on the Data Bottom portion of a graph (if Data Bottom is selected for the graph) and also will appear in reports generated from this test data.

4.1.7 Sine Define Resonance tab

Clicking the "Resonance" tab in the Sine Define dialog box accesses this dialog box.



This feature is optional and requires the software key be programmed to enable it.

Sine Resonance control tracks a resonance frequency by adjusting the driving frequency so that the relative phase between input channels 1 and 2 is 90 degrees (1.57 radians). This type of test is usually done with the channel 1 accelerometer mounted on the fixture and the channel 2 accelerometer mounted on the product at the location at which you want resonance induced.

The ***Gain*** parameter defines the speed at which the frequency will be adjusted when finding the resonance.

The ***Max Step*** parameter defines the maximum allowable step per control interval and limits the rate at which the driving frequency will be adjusted. If the resonance control is not stable, decrease this value to improve the stability of the control loop.

The ***Max Drift*** parameter defines the maximum allowable drift in the resonance frequency before the test

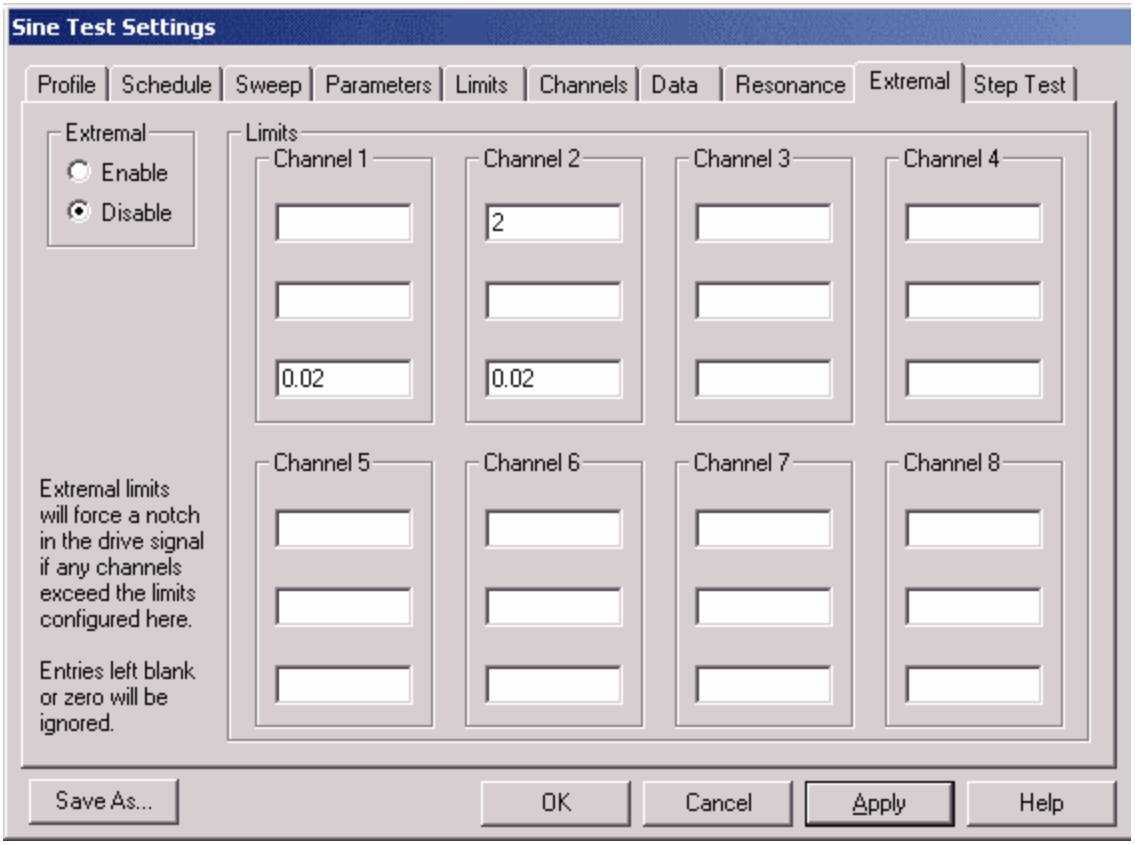
is automatically aborted. This is a safety factor that will shut the test down, if, for example, the product breaks (and therefore the resonance frequency changes drastically.)

Recommended values are:

- Gain: 0.1
- Max Drift: 10%
- Max Step: 0.1%

4.1.8 Sine Define Extremal tab

Clicking the "Extremal" tab in the Sine Define dialog box accesses this dialog box.

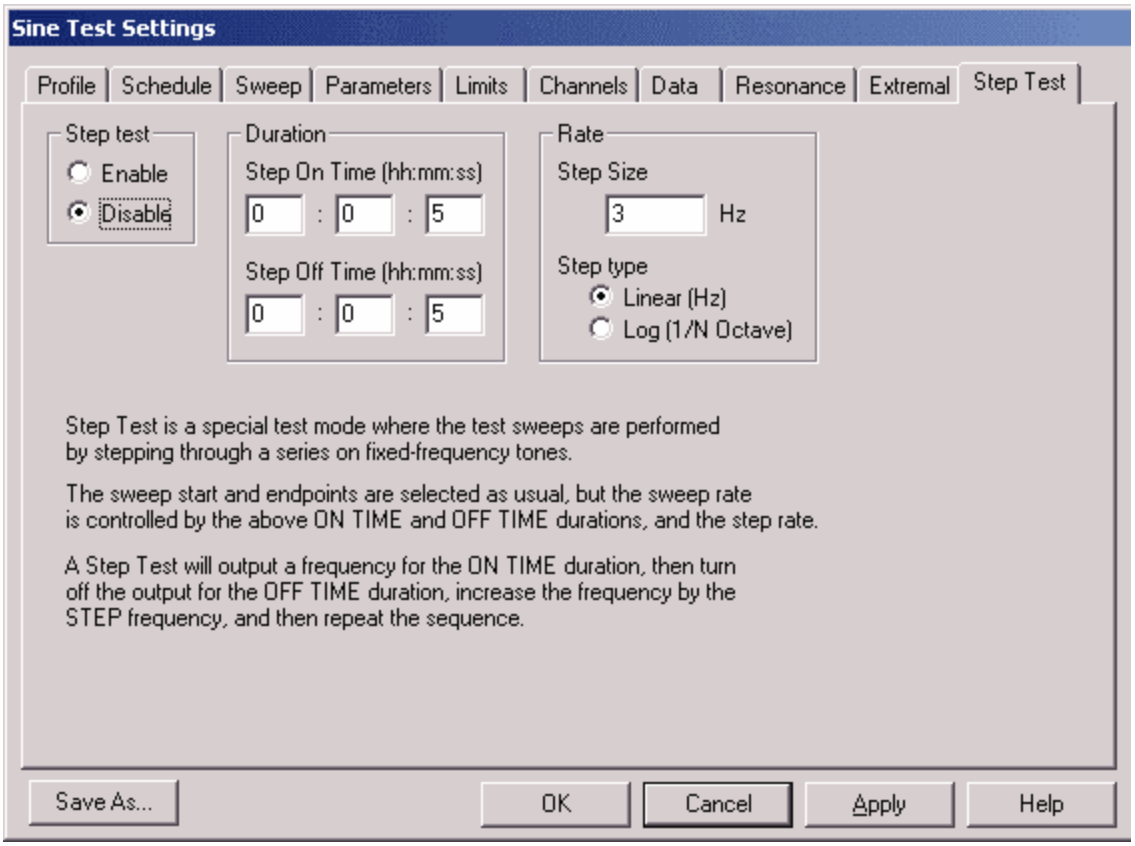


This feature is optional and requires the software key be programmed to enable it.

Use this dialog box to define the maximum acceleration, velocity and displacement parameters for each channel. If the controller exceeds these parameters while running and the Extremal control feature is enabled, the controller will reduce the drive to the shaker to keep the shaker at or below the specified maximum values. Blank or zero entries are ignored.

4.1.9 Sine Define Step Test tab

Clicking the "Step Test" tab in the Sine Define dialog box accesses this dialog box.



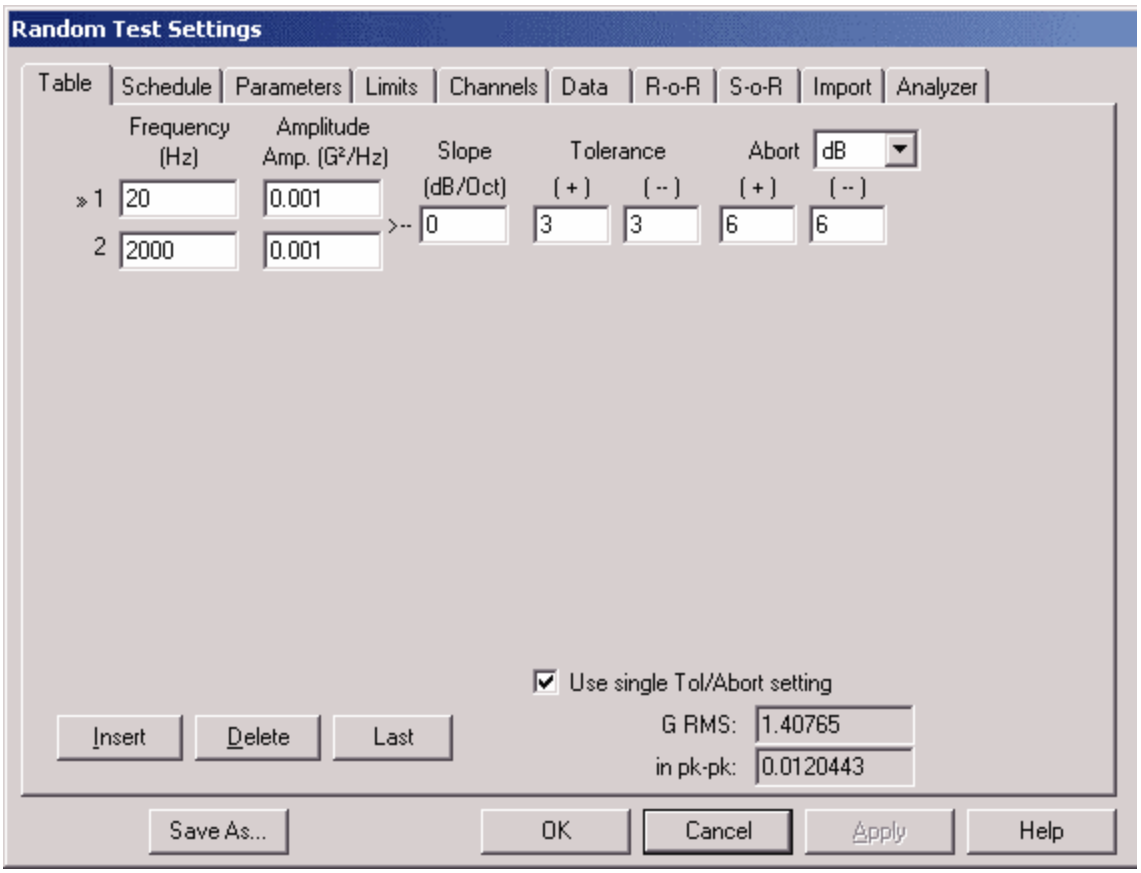
This feature is optional and requires the software key be programmed to enable it.

Use this dialog box to define a stepped-frequency (instead of continuously swept frequency) test. This type of test will run at a fixed frequency for the programmed "on" time. Then it will shut down for the programmed "off" time. Next, the frequency will step up or down by the amount specified by the ***Rate Step Size*** parameter. This will continue for the duration of the test, stepping with a series of discrete frequency values instead of the standard continuously varying sweep frequency. Certain test specifications require this type of operation.

The ***Rate Step Size*** parameter defines the size of the steps and the ***Step On Time*** and ***Step Off Time*** parameters define how much time to spend at each frequency. Click the "Enable" radio button to turn the step function on or the "Disable" radio button to turn it off.

4.2 Random Define dialog box

Selecting the Test..Edit Test menu command while a Random test is open displays this dialog box.



The ***Frequency*** and ***Amplitude*** parameters define the breakpoints for the spectral density profile. If you want the computer to calculate an amplitude based on a dB/Octave slope value, first enter both ***Frequencies*** and the known ***Amplitude***. Click in the box with the known ***Amplitude*** and then in the ***Slope*** parameter box. The amplitude parameter that will be calculated based on the frequencies and slope will be grayed out while you enter the slope value. After entering the ***Slope*** parameter, press the Tab key and the controller will automatically compute the dependent amplitude value.

If more than 10 ***Frequency***/***Amplitude*** breakpoints are entered, a scrollbar will appear allowing the operator to scroll through the full breakpoint table. Up to 201 breakpoints may be entered in the table.

The "Insert" and "Delete" buttons allow you to add and remove segments in the breakpoint table. The currently selected segment (indicated by the small arrow on the left side of the dialog box) will be deleted when the "Delete" button is clicked. Inserted segments will immediately follow the currently selected segment.

The RMS acceleration and peak-to-peak displacement requirements for a 100% level is indicated in the lower right corner of the dialog box. Either a percentage modifier or a fixed G level (using the Schedule tab) can set the actual control level.

The tabs along the top of the dialog box are used to select the following parameter entry screens:

Table	Enter the frequency and amplitude breakpoints.
Schedule	Set test duration and amplitude scaling level.
Parameters	Enter control loop parameters.
Limits	Enter the safety limits.
Channels	Select control channels, and averaging technique
Data Store	Select automatic data storage parameters.

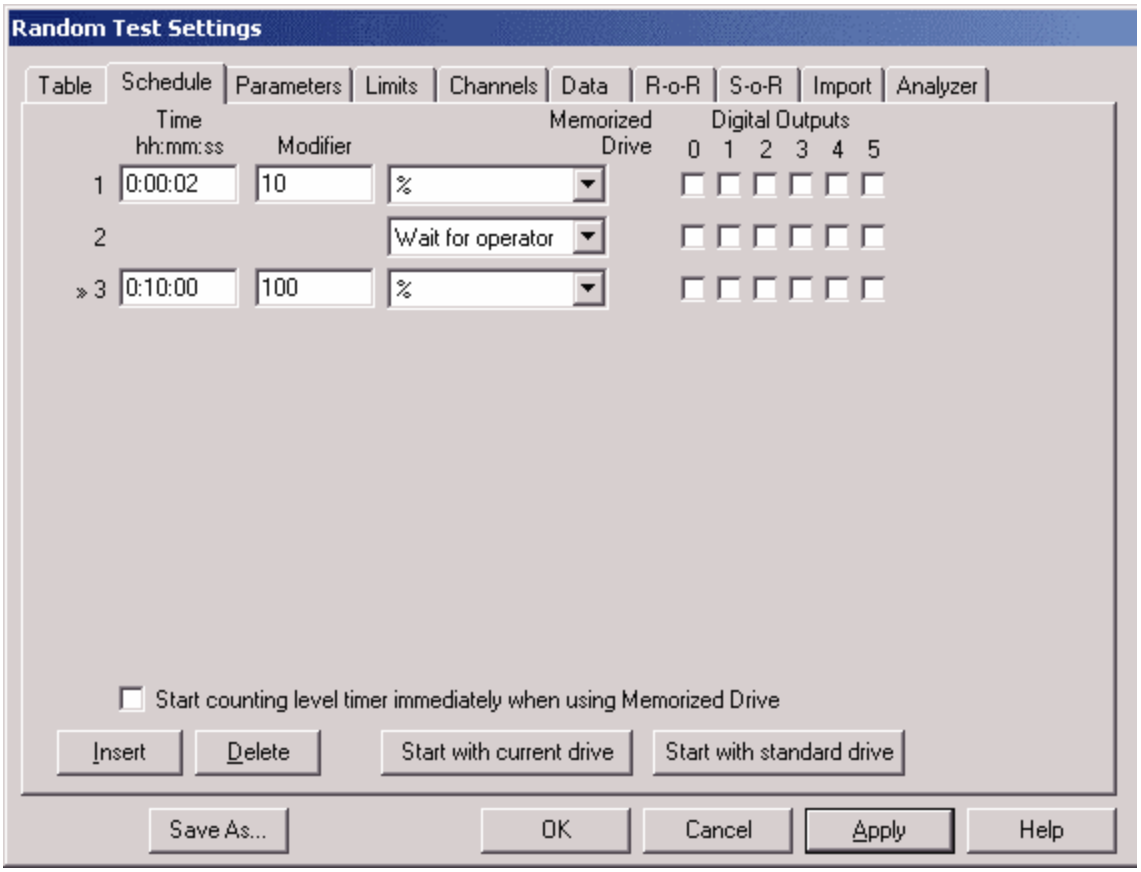
R-o-R	Define Random-on-Random "tones"
S-o-R	Define Sine-on-Random tones
Import	Import a spectrum from a text file
Analyzer	Configure the link to the VibrationAnalyzer Program

The buttons at the bottom of the dialog box perform the following operations:

Save As...	Save the current settings under a new test name.
OK	Save the changes to this test and close the dialog box
Cancel	Close the dialog box and abandon all changes
Apply	Apply the changes to the currently running test (does not save changes to the hard disk).
Help	Get help information about these parameters.

4.2.1 Random Define Schedule tab

Clicking the "Schedule" tab on the Random Define dialog box accesses this dialog box.



Use this dialog box to define how long your test will run at each output level. You can define up to 200 separate levels and add looping to repeat level sequences. If you have more than 10 levels defined, a scroll bar will appear on the right side of the dialog box. Use it to scroll through the defined levels.

The ***Time*** parameter defines the duration (in the form hours:minutes:seconds) for the indicated level number. You can also enter time durations by following the hour value with an 'h', the minute value with an 'm' and the second value with an 's'. For example, '5 hours', '5h', '300m' are all valid entries and will be converted to 05:00:00 (5 hours).

The ***Modifier*** parameter defines the amplitude for each level You can modify the entire profile to match the desired RMS acceleration level or scale the entered spectrum by a % or dB modifier. Enter the ***Modifier*** parameter in the box and select the appropriate ***Modifier*** type from the drop-down list box.

To repeat part of the test schedule a number of times, insert a level immediately following the level(s) you want to repeat and set its modifier type to "Times". Enter the number of the first level to be repeated in the ***Loop from*** parameter and the total number of passes to perform in the ***Modifier*** column. Loops can be nested up to 10 levels deep to create more complex repeating patterns.

The last modifier type is "Wait for operator". When this level type is reached, the controller output will be turned off and a dialog box will appear asking you to click the "Continue" button. When the "Continue" button is clicked, the test will resume with the next level. If "Memorized drive" is checked for the next level, the equalized drive from the previous level will be used to quickly start the output at the new demand level. If "Memorized drive" is not selected for the next level, the controller will ramp the output up slowly to the new demand amplitude.

Memorized drives:
To speed up the test start-up process, the drive signal required by the test can be memorized. To memorize a drive, perform the following steps:

1. Start the test and wait until the output has reached the desired amplitude (the test switches to Run mode with a green "Run" button).
2. Edit the test schedule and click the "Start with current drive" button to memorize the drive currently being output by the system. Note: if the test is currently stopped, the last drive output before the "Stop" button was clicked will be memorized.
3. Make sure the "Enable memorized drive" check box is checked for level number 1.
4. Save the test.

Note: the current drive signal can also be saved by selecting Test..Save Current Drive, without going to the edit test dialogs.

If you have many similar tests, the drive required for the current test may be estimated from a standard drive signal. To do this, perform the following steps:

1. Run one of the tests and wait until the output has reached the desired amplitude (i.e. the test switches to Run mode and the "Run" button turns green).
2. Select the Test..Create Standard Drive File menu command to save that test's drive output as a standard output.
3. Open another test (or create a new test)
4. Edit the test schedule and click the "Start with standard drive" button. This will read the standard drive signal used by the first test and reshape it to match the profile of the second test.
5. Repeat steps 3 and 4 for any other similar tests.

Note: the standard drive signal only needs to be created once, as long as the system configuration (amplifier gain, shaker system, etc.) does not change.

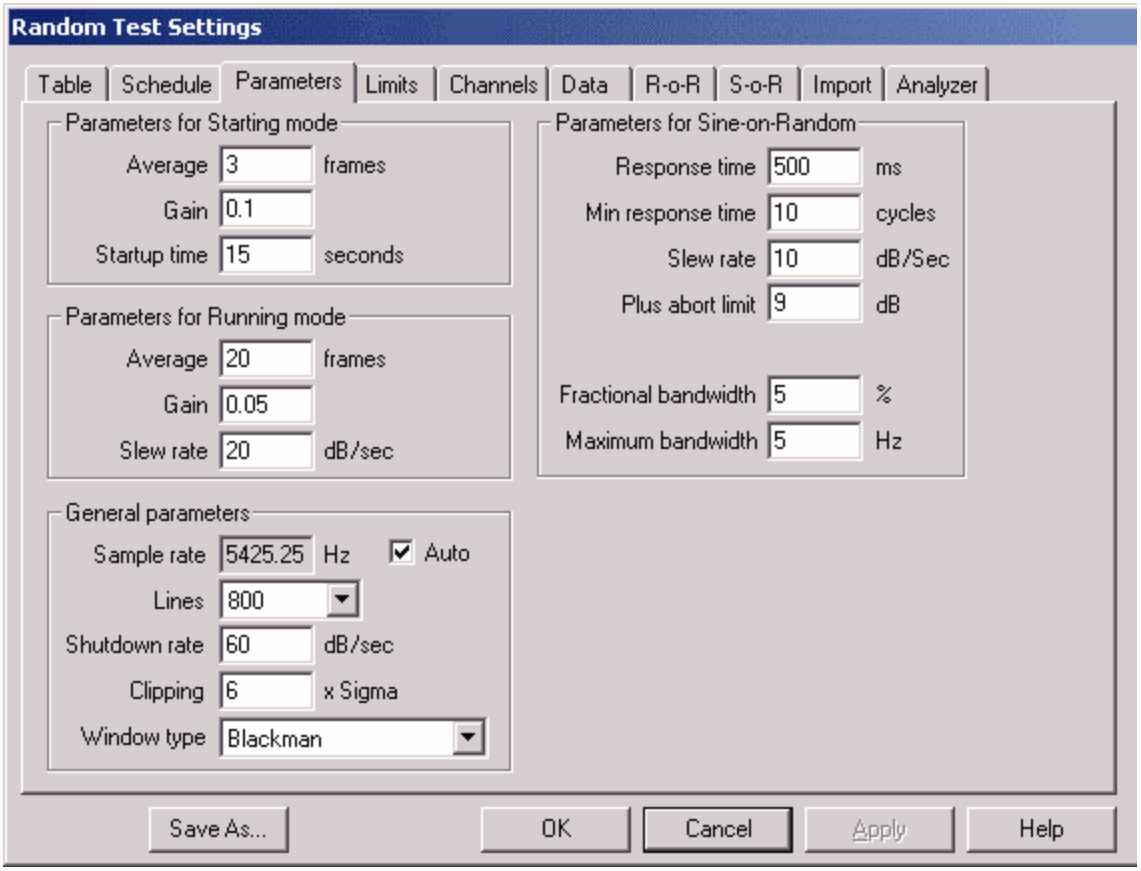
To disable the use of the memorized drive signal and revert to the slow test start-up, uncheck the "Enable memorized drive" box.

For levels 2 and above, the "Enable memorized drive" check box will enable fast switching of output levels between the previous level and the current level. The system will compute the difference in the desired output levels, and scale the drive signal accordingly. If "Enable memorized drive" is not checked for levels 2 and above, the transition will be gradual, with the transition speed controlled by the Start Gain defined for the test.

See also Test..Save Current Drive and Test..Create Standard Drive File for additional information.

4.2.2 Random Define Parameters tab

Clicking the "Parameters" tab on the Random Define dialog box accesses this dialog box.



See also: How to tune Random controller parameters.

Use this dialog box to define the parameters that influence the Random control loop.

There are two sets of control parameters. The first set (starting) is used when starting a test, to get the output from zero up to the demand level. By default, the ***Average*** is 3, the ***Gain*** is 0.1 and the ***Startup time xx seconds*** is 15.

The **Parameters for Starting mode** are used while the tests is ramping up to the desired output level, indicated when the "Run" button is Yellow (**Yellow**=normal start) or Blue (**Blue**=saved drive). There will typically will be less averaging (2 to 6 frames) and more gain (0.1 to 0.2) during the starting period because this is when the controller must be able to quickly equalize the output.

The **Parameters for Running mode** are used after the test has reached the desired output level, indicated when the "Run" button is Green (**Green**=running). There will typically be more averaging (10 to 20 frames) and less gain (0.02 to 0.05) during the running period because the controller has already equalized the system, and needs only make small adjustments to the drive signal.

The ***Average XX frames*** parameter defines the number of data frames averaged together to get the frequency data. The higher the number is, the smoother the line will be but the slower the controller will

respond.

The ***Gain*** parameter defines the scaling factor by which the error is multiplied each control loop. The bigger the gain value is, the faster the control reacts. You should keep this number large enough to get to level in a reasonable time, but small enough so that the control does not overshoot the intended level. Typical values for the starting control gain are between 0.1 and 1.0. If the averaging value is increased, the gain value must be decreased to avoid overshoot.

The ***Slew rate*** parameter defines the maximum slew rate during operation. The test will abort if the output rate exceeds this parameter

The ***Clipping X x Sigma*** parameter adjusts the allowable peak output level relative to the RMS output level. Since the output signal is random, it will typically have an occasional high peak value. The sigma clipping level limits the amplitude of these peaks and is typically used to prevent over-voltage tripping of the amplifier input and to limit the peak displacement value required by tests. A sigma clipping level of 6 essentially disables the sigma-clipping. When sigma-clipping is used, it is typically set to a value of 3. Note: Gaussian-distributed random noise as used in a random vibration controller rarely exceeds the 4-sigma level.

The ***Sample Rate*** parameter should in nearly all cases be set to Auto to allow the controller to select an appropriate sample rate. However, experienced users have the option of entering the sample rate used by the controller. If a sample rate is selected manually, it must be at least double the highest frequency in the test, and typically would be 2.6 times the highest test frequency.

The parameters for the optional Sine on Random module are also adjusted here.

The **Response Time** *and* **Min Response Time** *parameters are the primary loop tuning parameters.*
The ***Response Time*** parameter defines high frequency loop response. Lower response times provide faster control response. Higher response times provide loop stability.

The ***Min Response Time*** parameter defines low frequency loop response. If number of cycles specified in ***Min Response Time***, at a given frequency, is greater than the specified response time, it determines loop response.

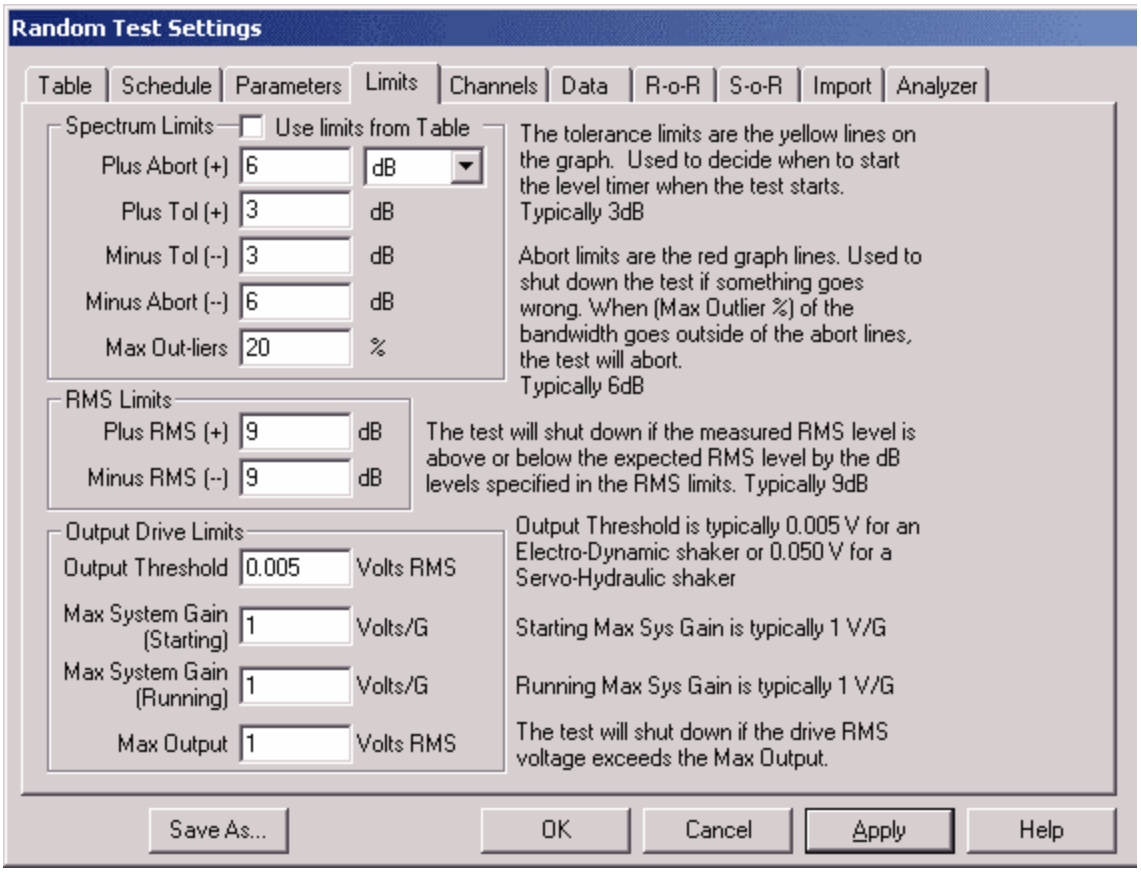
The ***Slew Rate*** parameter defines the maximum slew rate during normal operation.

The ***Fractional Bandwidth*** parameter defines the bandwidth of the tracking filters at low frequencies. It is specified as a percentage of frequency.

The ***Maximum Bandwidth*** parameter defines the bandwidth of the tracking filters at high frequencies.
The ***Fractional Bandwidth*** parameter is not allowed to exceed the ***Maximum Bandwidth*** parameter.

4.2.3 Random Define Limits tab

Clicking the "Limits" tab on the Random Define dialog box accesses this dialog box.



Use this dialog box to define critical controller limits for a random vibration test. These safety limits cause the controller to abort a test when it detects an inconsistent measurement, for example, if an accelerometer falls off during the test.

The ***+/- Tol*** parameters define the maximum/minimum spectral density tolerance line levels indicated on the graphs. The tolerance lines are shown as dashed brown lines on the graph and are use to determine when the controller should switch from starting mode to running mode.

The ***+/- Abort*** parameters define the maximum/minimum spectral density abort line levels indicated on the graphs. The abort lines are shown as red lines on the graph. These level are measured in dB from the demand line, where 6dB sets the tolerance/abort line to a factor of 4 above the demand spectral density level. This corresponds to a doubling of the RMS acceleration level because the spectral density levels are proportional to the acceleration squared.

The ***Max Outlier %*** parameter defines the maximum percentage of the test bandwidth allowed to exceed the abort limits before a shutdown will occur. Since the test is inherently random, an occasional line outside of the abort lines is typically allowed, but large numbers of lines exceeding the abort limits indicate a problem requiring the test to be aborted. This parameter sets the threshold between acceptable occasional outliers and excessive outliers.

The **+/- RMS** parameters define the maximum/minimum RMS acceleration level, measured in dB relative to the demand level, allowed for each of the control channels. If there is excessive out-of-band energy in the signal (for example, if a part is rattling, causing harmonics) this limit will cause the test to shut down.

Also, if the instantaneous RMS level on one of the control channels falls this far below the demand RMS level, the test will abort with an Accel Reading too Low stop code.

The ***Output Threshold*** and ***Starting Max System Gain*** parameters are safety limits that define an output voltage limit as a function of the measured RMS acceleration level. When starting a test, the output voltage will be limited to:

(Output Voltage) <= (Output Threshold) + (Starting Max System Gain)*(Measured RMS Acceleration)

If you start a test *without* an accelerometer connected, these settings will limit the maximum output allowed and the controller will abort the test. When an accelerometer *is* connected, the output voltage will be allowed to increase as the measured RMS acceleration level from the accelerometer increases.

The ***output threshold*** parameter is typically 0.005 Volts for an Electro-Dynamic shaker and 0.050 Volts for a Servo-Hydraulic shaker. The ***Starting Max System Gain*** parameter depends on the shaker system and the amplifier gain setting. To determine this value, start the test and after the output reaches the demand amplitude note both the "Volts rms" and the "Demand G RMS" readings in the Random Control Center. Set the ***Starting Max System Gain*** parameter to approximately:

Starting Max System Gain = 4 x (Volts rms) / (Demand G RMS)

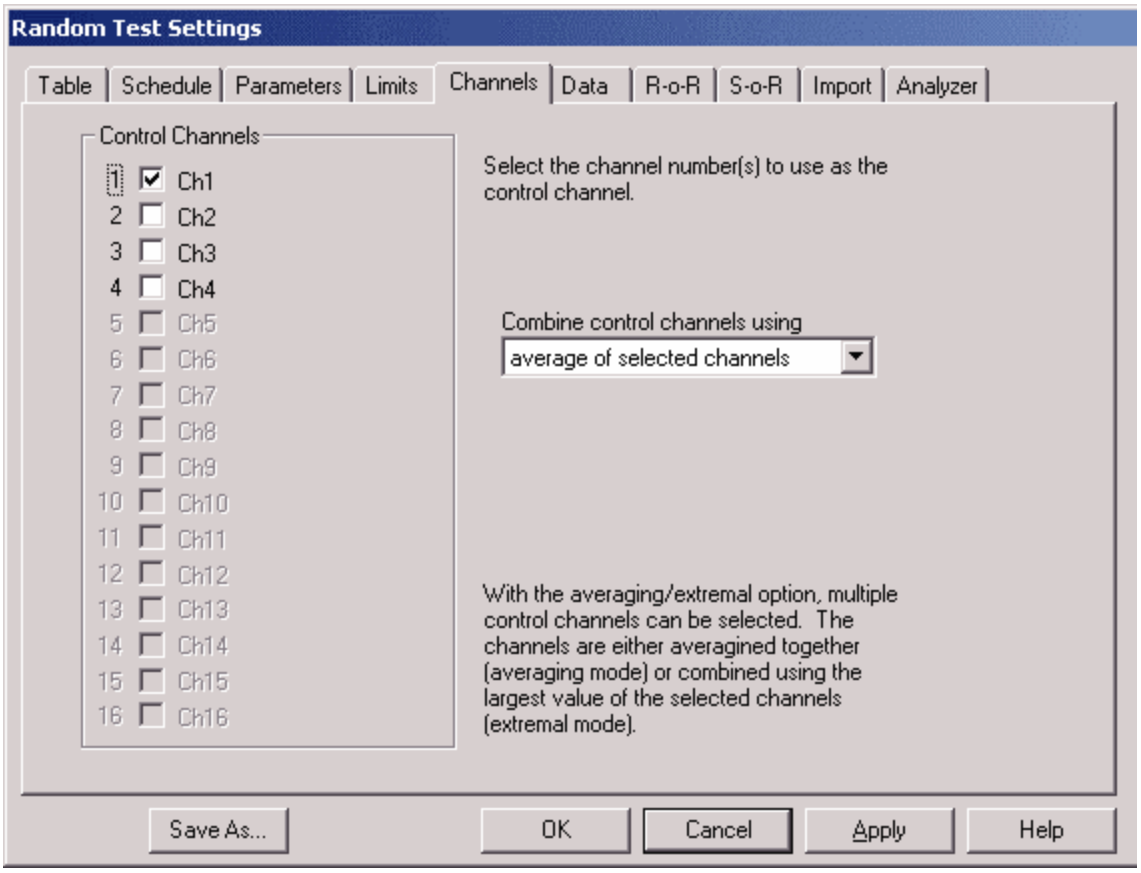
To disable this limit, set the ***Starting Max System Gain*** parameter to a large number (1000 V/G), or set the ***Output Threshold*** parameter to 10 Volts.

The ***Running Max System Gain*** parameter is another safety limit that limits the output voltage as a function of the measured RMS acceleration while the test is running. This value should typically be set to double the V/G level required by your system. To determine your required V/G setting, set the ***Running Max System Gain*** parameter to a high level (e.g. 1000 V/G), click the "Apply" button, run the test and note the RMS output voltage level required by the test, as indicated in the Random Control Center. Divide this voltage level by the Demand G RMS level (also shown in the Random Control Center) and multiply by 2.

The ***Max Output*** parameter defines the maximum RMS output voltage allowed from the controller. If the drive signal output ever exceeds this value, the test will abort. To determine an appropriate ***max output*** setting, run the test and look at the "Volts rms" reading displayed on the Random Control Center. Multiply that value by 1.5 (to give a 50% cushion) to get an appropriate ***Max Output*** setting.

4.2.4 Random Define Channels tab

Clicking the "Channels" tab on the Random Define dialog box accesses this dialog box.

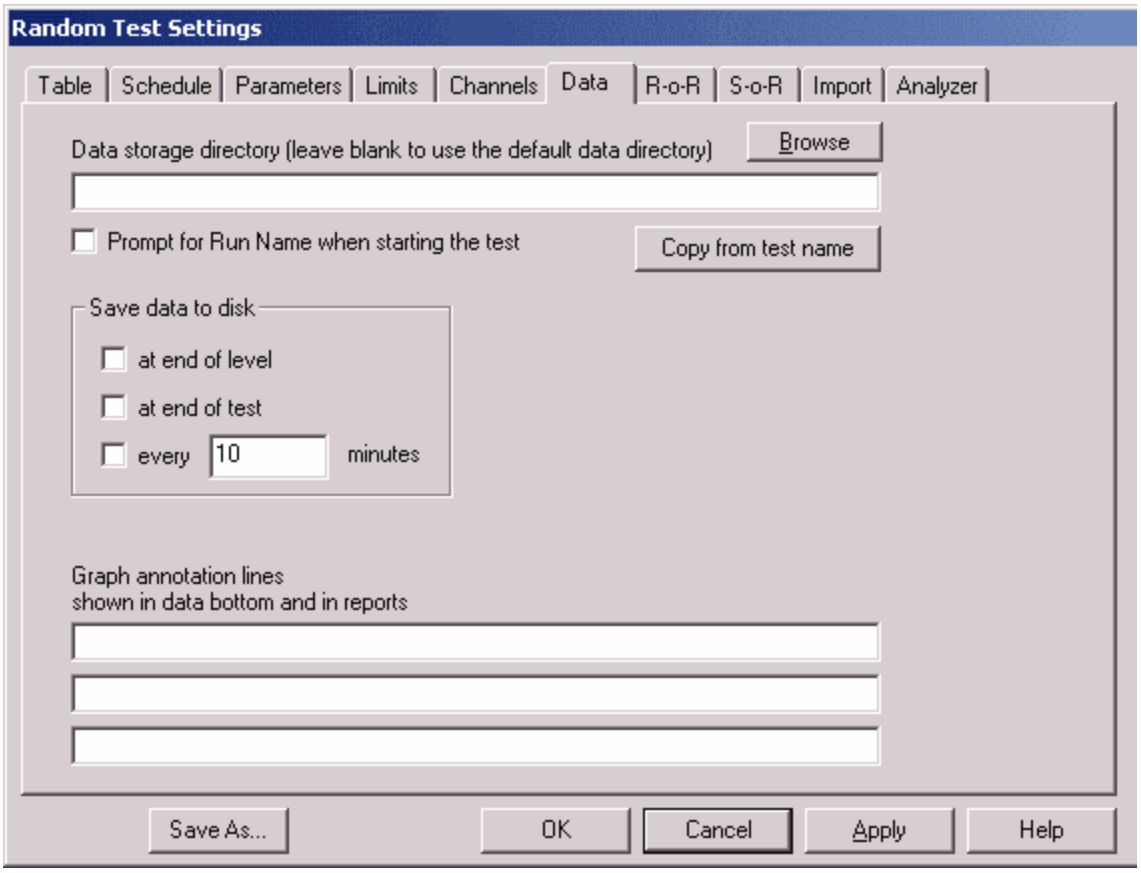


Use this dialog box to define the input channels to be used in the random control loop.

If multiple channels are selected, they can be combined either by averaging the selected channels' spectral lines or by using the maximum value from each of the selected channels. Note that the extremal mode is an optional feature that is available only if the software key has been programmed to enable it. If this feature is not enabled in your software key, the "combine control channels using" will always be "average of selected channels".

4.2.5 Random Define Data Storage tab

Clicking the "Data" tab on the Random Define dialog box accesses this dialog box.



Use this dialog box to configure the automatic data storage options for a test.

You can save the data automatically at the end of a level (as defined on the "Schedule" tab), at the end of a test and/or at a regular time interval. Check the appropriate boxes to activate the automatic data store function. Check "Prompt for Run Name when starting the test" to display a prompt dialog allowing the operator to enter a new directory each time the test is started.

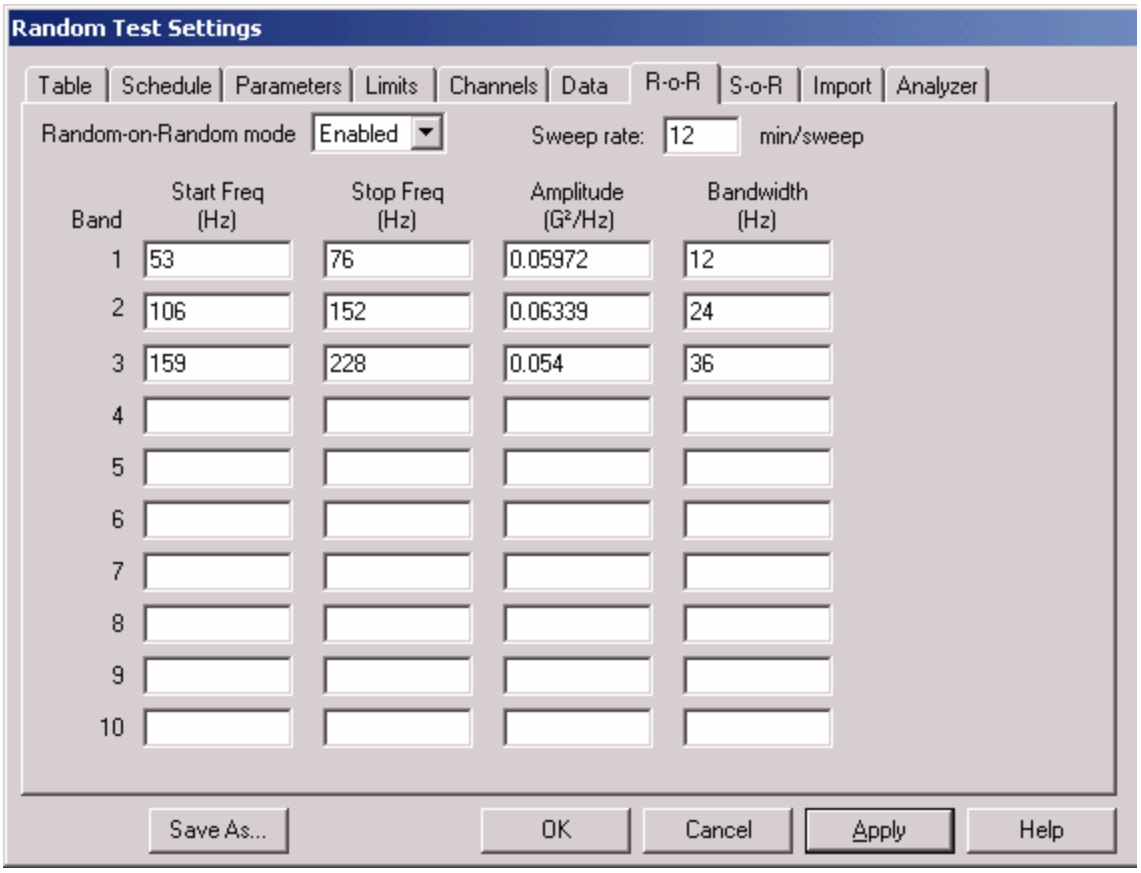
You can also enter three lines that will appear in the Data Bottom portion of a graph. Editing a graph's settings and checking the "Data at Bottom" check box in the Random Graph Settings dialog box enables this display.

Editing a graph's settings and checking the "Data at Bottom" check box in the Graph Settings dialog box can enable this display.

All data is saved as a time stamped file that can be viewed at a later time by selecting the File..Open Data menu command.

4.2.6 Random Define R-o-R tab

Clicking the "R-o-R" tab on the Random Define dialog box accesses this dialog box.

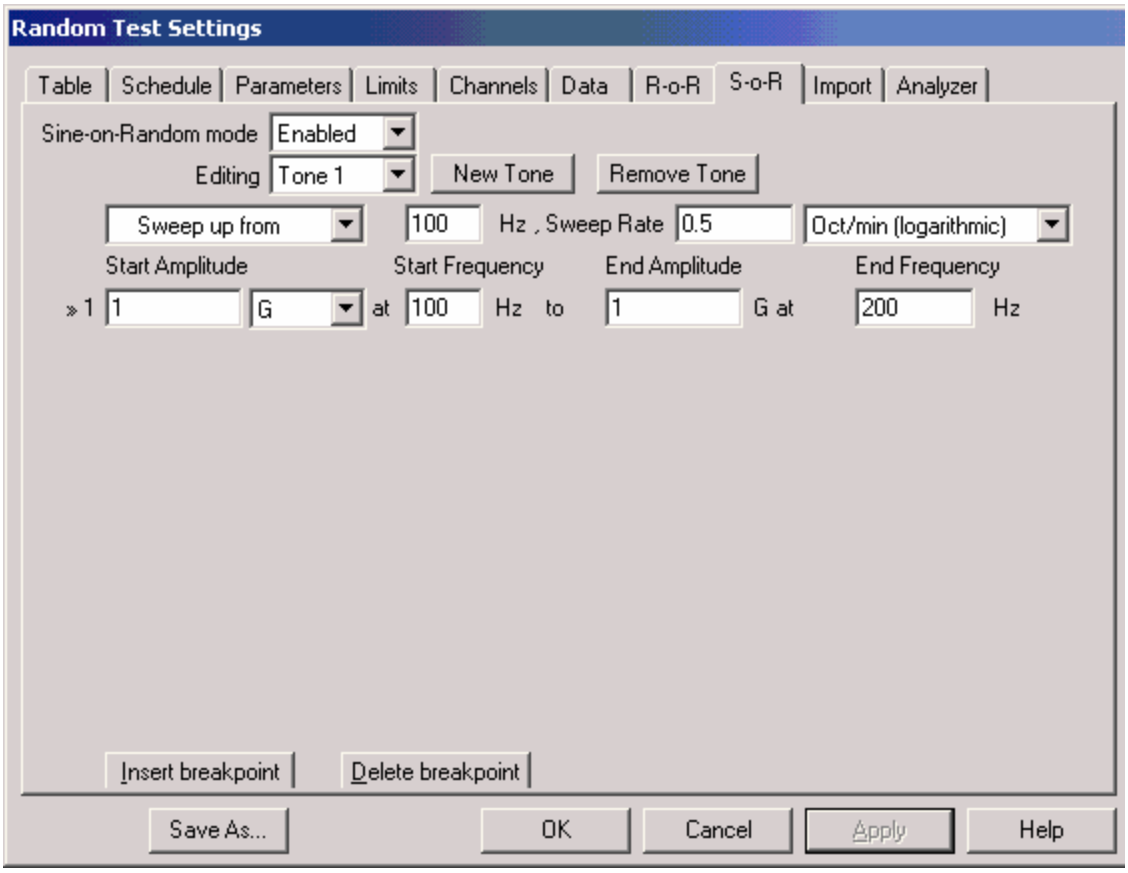


Use this dialog box to define up to 10 narrow-band random "tones" to add to the Random profile. The defined tones will sweep from the ***Start Frequency*** to the ***End Frequency*** in the specified number of minutes. After the ***End Frequency*** is reached, the sweep will continue, sweeping from the ***End Frequency*** back to the ***Start Frequency***. To sweep up in frequency, set the ***End Frequency*** higher than the ***Start Frequency***. To sweep down in frequency, set the ***End Frequency*** lower than the ***Start Frequency***.

The Random-on-Random control mode is an optional feature; the software key must be programmed to enable it.

4.2.7 Random Define S-o-R tab

Clicking the "S-o-R" tab on the Random Define dialog box accesses this dialog box.



The ***Start*** and ***End Frequency*** parameters define the frequency span of the level. The operator selects the first ***Start Frequency***; the other ***Start Frequencies*** follow from the previous ***End Frequency***. The ***Start*** and ***End Amplitude*** parameters set the starting and ending points for each level and do not need to be constant for a given level or continuous at the level endpoints.

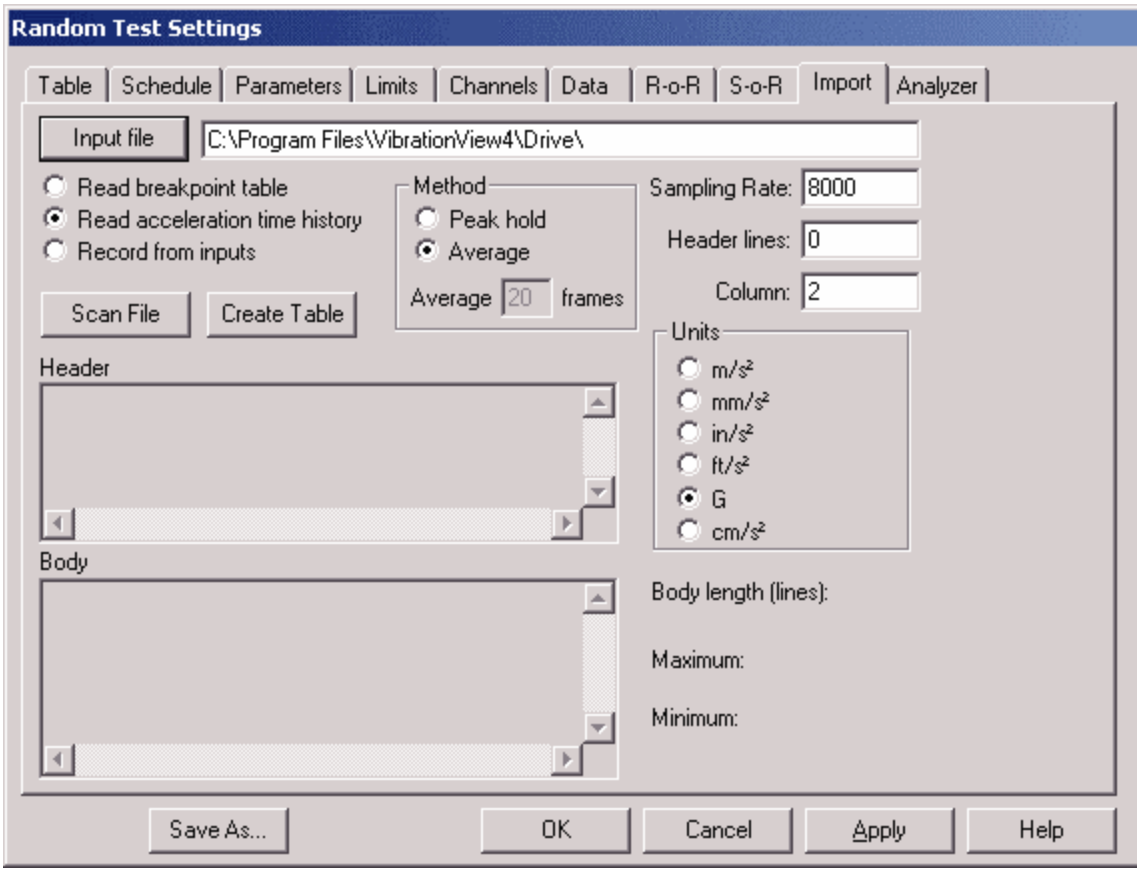
The units selected for the ***Start Amplitude*** will determine what type of control is used for that segment of the sweep. For example, if "in" (inches, in English units) is the unit selected, displacement will be controlled, if "in/sec" is selected, velocity will be controlled and if "G" is selected, acceleration will be controlled. Therefore, if you need a constant displacement section, set the units selection to a displacement unit (e.g. "in").

If the frequency sweep is linear (Hz/min), ramped amplitude sweeps will be straight lines on linear graphs. If the frequency sweep is logarithmic, ramped amplitude sweeps will be straight lines on log-log graphs. The frequency can be swept up, swept down, or held at the start frequency.

The Sine-on-Random control mode is an optional feature; the software key must be programmed to enable it.

4.2.8 Random Define Import tab

Clicking the "Import" tab on the Random Define dialog box accesses this dialog box.



Use this dialog box to automatically generate a spectrum profile from input data. The input can be a set of frequency/amplitude breakpoints, a time/acceleration history stored in a text file, or can be measured from an analog input channel on the *VibrationVIEW* I/O unit.

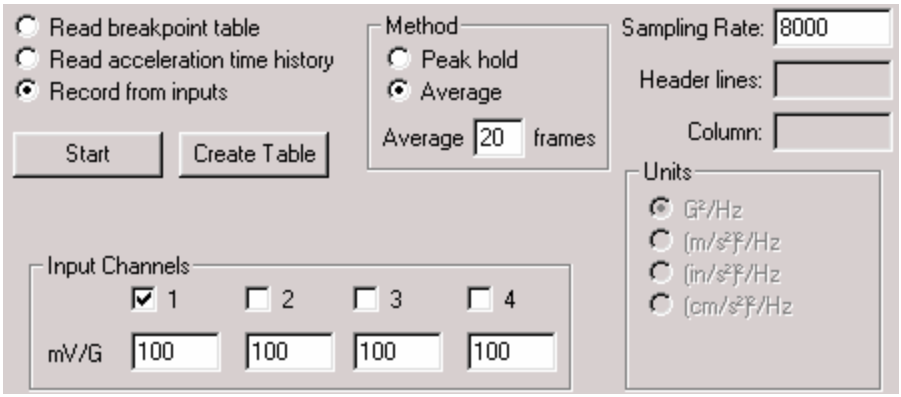
To input data from a file, perform the following procedure:

1. Select the input type using the radio buttons directly below the "Input file" button.
2. Click the "Input file" button to select the file from which to read the data. The file will be scanned, and the dialog box will be updated to reflect the contents of the file.
3. Set the sampling rate, number of header lines, the column number for the acceleration data, and the acceleration units used in the file.
4. Select the spectrum generation method. The import procedure divides the file into 1024-sample blocks, computes the spectrum for each block, and then combines them according to the selected method. "Average" will average all of the spectra together using a uniform average. "Peak hold" will extract the peak values over all of the blocks at each frequency.
5. Click the "Create Table" button to process the file and create the frequency/amplitude breakpoint table from the input file.

To input data from an analog input channel on the *VibrationVIEW* I/O unit, perform the following procedure:

1. Select the "Record from inputs" radio button, as shown in the figure below.
2. Select the spectrum generation method. "Peak hold" will keep the peak acceleration level seen at each frequency. "Average" will perform an exponentially-weighted average of the input data, with the amount of averaging determined by the ***Average XX frames*** parameter. Higher averaging values will result in more averaging and therefore smoother spectra.
3. Select the input channel, mV/G setting and desired sampling rate. The sampling rate should be 2.6 times the highest desired frequency to be used in the test. An anti-aliasing filter on the input will remove the higher frequency components from the signal.
4. Connect the input source to the selected analog input channel on the *VibrationVIEW* I/O unit.
5. Start playback on the input source (e.g. if using a DAT record, click the Play button).
6. Click the "Start" button to start recording. When finished, click the "Stop" button.
7. Click the "Create Table" button to process the input and create the frequency/amplitude breakpoint table from the data.

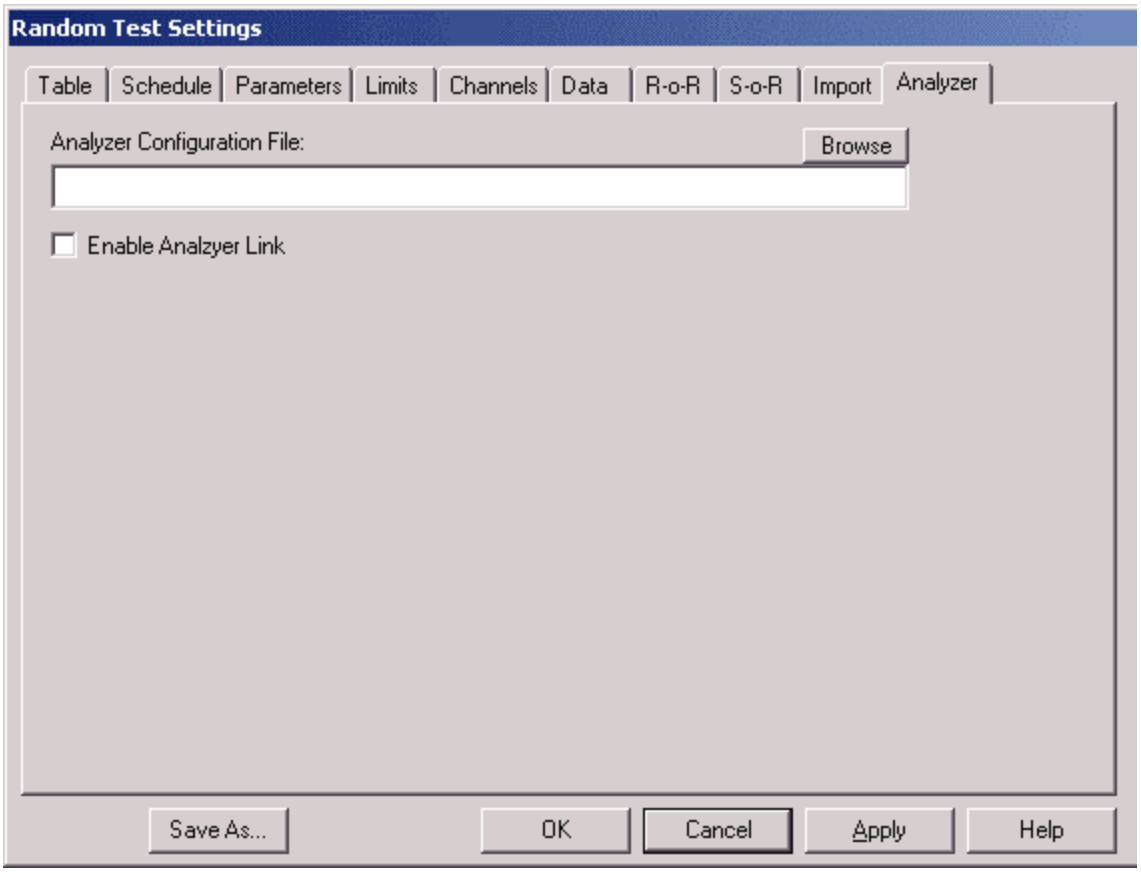
Figure: Generate a random test profile table from the analog input signals.



The Random Import feature is an optional feature and requires the software key be programmed to enable it.

4.2.9 Random Define Analyzer tab

Clicking the "Analyzer" tab on the Random Define dialog box accesses this dialog box.

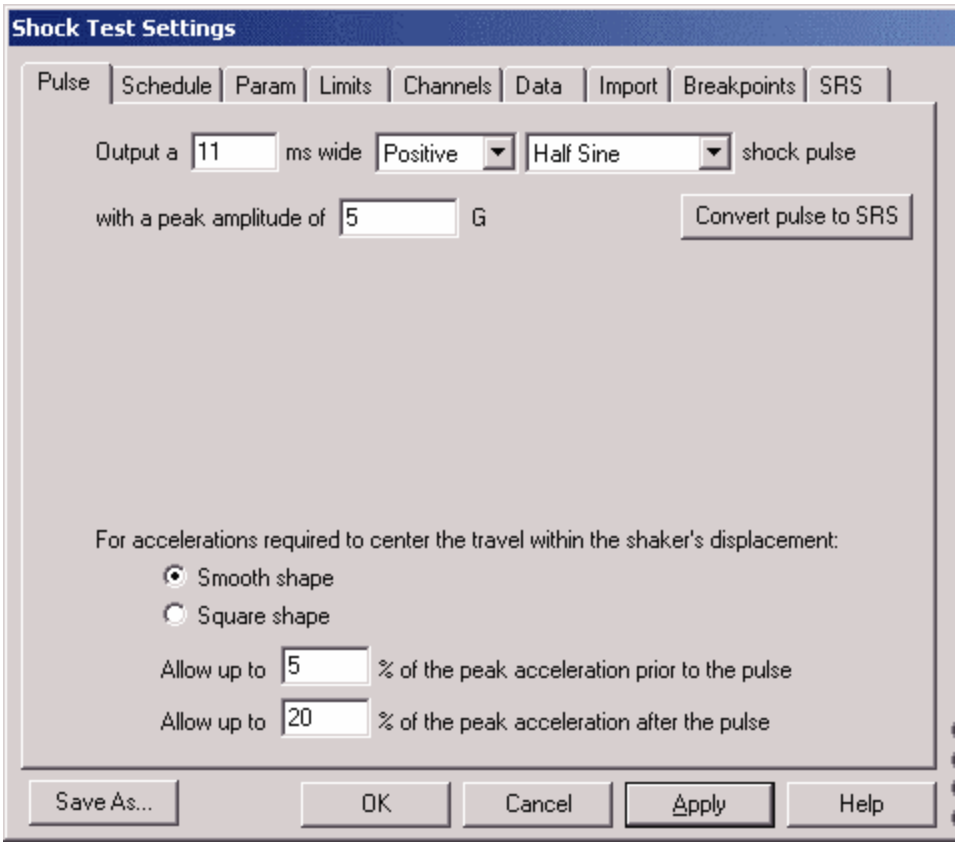


Use this dialog box to configure the link between *VibrationVIEW* and VibrationAnalyzer. VibrationAnalyzer is a separate application available from Vibration Research Corporation that allows monitoring of up to 16 channels with comparison to custom abort limits. If VibrationAnalyzer detects one of the monitor channels outside of the limit lines specified for that channel, it will pass this information through the configured Analyzer Link and *VibrationVIEW* will abort the current test.

Enter the appropriate configuration file for VibrationAnalyzer in the "Analyzer Configuration File" box. When *VibrationVIEW* runs a test, this configuration will be automatically loaded into VibrationAnalyzer. The configuration file must be defined separately using VibrationAnalyzer. Refer to the documentation for VibrationAnalyzer for details on how to create a configuration file.

4.3 Shock Define dialog box

Selecting the Test..Edit Test menu command while a Shock test is open displays this dialog box.



Use this dialog box to define the type of output pulse.

The ***Width*** parameter defines the duration of the pulse in milliseconds.

The ***Polarity*** parameter defines the direction, either positive or negative, of the pulse.

The ***Type*** parameter defines the pulse shape. Available shapes are: Half Sine, Initial Peak, Terminal Peak, Triangle, Trapezoid, Square, Haversine, and User-Defined.

For the Trapezoid pulse type, you must also enter the Ramp-Up and Ramp-Down times, in milliseconds.

To create non-oscillatory shock waveforms on shakers, it is necessary to add acceleration components before and/or after the pulse to ensure that the shaker begins and ends with zero displacement and zero velocity. You must enter the maximum acceptable acceleration values before and after the pulse, in terms of a percentage of the pulse's peak amplitude. To have compensation only after the pulse, enter 0% for the peak acceleration allowed prior to the pulse. To have compensation only prior to the pulse, enter 0% for the peak acceleration allowed after the pulse. If compensation is allowed both prior to and after the pulse, the compensation waveforms will be optimized to center the displacement waveform.

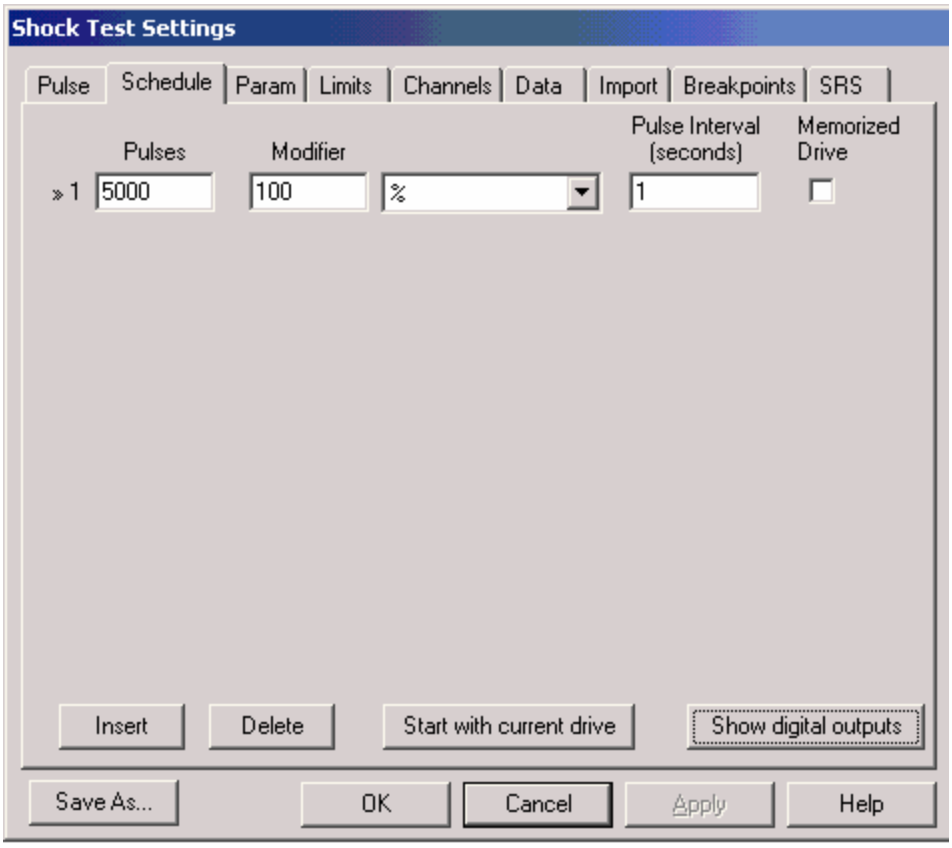
The "Convert pulse to SRS" button will compute the SRS of the currently selected pulse type, and transfer those settings to the SRS tab.

The tabs along the top of the dialog box are used to select the following parameter entry screens:

Pulse	Select pulse type
Schedule	Enter number of pulses and scaling factors.
Parameters	Enter control loop parameters
Limits	Enter the safety limits
Channels	Select control channel(s)
Data Storage	Set automatic data storage settings
Import	Import user-defined waveforms from an ASCII file
Breakpoints	Define the Frequency/Amplitude breakpoints of the desired SRS curve
SRS	Define the SRS analysis parameters, and synthesize waveform

4.3.1 Shock Define Schedule tab

Clicking the "Schedule" tab on the Shock Define dialog box accesses this dialog box.



Use this dialog box to define how long your test will run at each of a set of output levels. You can define up to 200 separate levels, plus looping to repeat level sequences multiple times. If you have more than 10 levels defined, a scroll bar will appear on the right side of the dialog box. Use it to scroll through the defined levels.

The ***Pulses*** parameter defines the duration of the level in number of pulses.

The ***Modifier*** parameter defines the peak amplitude for each level. You can set a desired peak acceleration value, or scale the value entered on the Pulse tab by a % or dB modifier. Enter the Modifier value in the box and select the appropriate modifier type from the drop-down list box.

To repeat part of the test schedule a number of times, insert a level immediately following the level(s) you want to repeat and set its modifier type to "Times". Enter the level number of the first level in the loop in

the "Loop from" entry and the total number of passes to perform in the "Modifier" column. Loops can be nested up to 10 levels deep to create more complex repeating patterns.

The last modifier type is "Wait for operator". When this level type is reached, the controller output will be turned off and a dialog box will appear on the controller asking you to click the "Continue" button to continue the test. When the "Continue" button is pressed, the test will resume with the following level. If "Memorized drive" is checked for the following level, the equalized drive from the previous level will be used to quickly start the output at the new demand level. If "Memorized drive" is not checked for the following level, the controller will ramp the output up slowly to the new demand level.

The ***Pulse Interval*** parameter defines the time, in seconds, between pulses. Setting a value of 0 will output pulses as fast at the controller is able.

Memorized drives:
To make tests start immediately without the equalization steps, the drive signal required by the test can be memorized. To memorize a drive, perform the following steps:

1. Start the test and wait until the output has reached the desired level (the test switches to Run mode with a green Run button).
2. Edit the test schedule and click the "Start with current drive" button to memorize the drive currently being output by the system. Note: if the test is currently stopped, the last drive output before the "Stop" button was clicked will be memorized.
3. Make sure the "Enable memorized drive" check box is checked for level number 1.
4. Save the test.

Note: the current drive signal can also be saved by selecting the Test..Save Current Drive menu command.

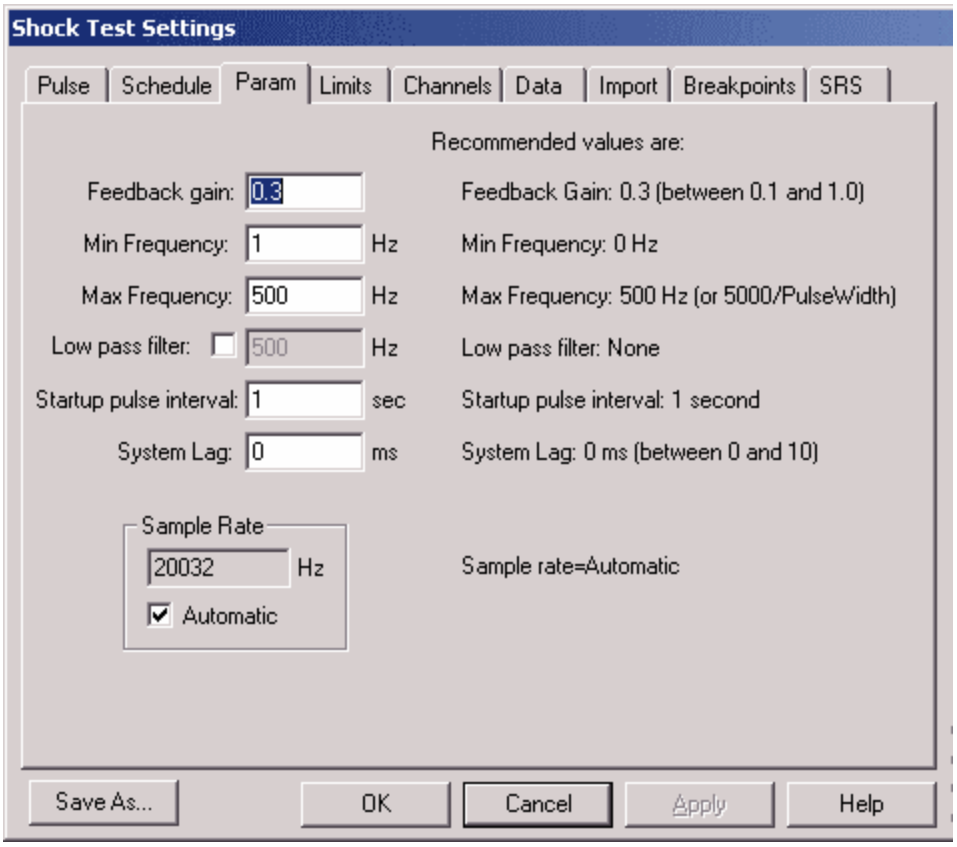
To disable the use of the memorized drive signal and revert to the graduated test start-up, uncheck the "Enable memorized drive" box.

For levels 2 and above, the "Enable memorized drive" check box will enable fast switching of output levels between the previous level and the current level. The system will compute the difference in the desired output levels, and scale the drive signal accordingly. If "Enable memorized drive" is not checked for levels 2 and above, the transition will be gradual, with the transition speed controlled by the Start Gain defined for the test.

See also Test..Save Current Drive for additional information

4.3.2 Shock Define Parameters tab

Clicking the "Parameters" tab on the Shock Define dialog box accesses this dialog box.



See also: How to tune shock controller parameters.

Use this dialog box to define the parameters used by the shock control loop.

The ***Feedback Gain*** parameter defines the rate at which the controller adjusts the drive output and the step size used during the equalization pulses. For example, with a feedback gain value of 0.3, the controller will first output a low-level training pulse and then pulses at 30%, 60% and finally 100% of the desired amplitude. We recommend using a value of 0.3. If the pulse starts up too quickly or the output pulse is not consistent, reduce this value.

The ***Min Frequency*** parameter defines the lowest frequency adjusted by the controller. This value typically should be left at 0Hz to control over the entire low frequency band. In some situations with User-Defined pulse waveforms, this setting may be used to remove some of the low frequency content from the waveform.

The ***Max Frequency*** parameter defines the maximum frequency used in the feedback control. The controller will not drive frequencies higher than this value. This value should typically be set to the maximum frequency that the shaker can output or the frequency at which the demand amplitude goes below 10 times the noise level. A rule of thumb is to set the ***Max Frequency*** parameter to be at least 5000 divided by the Pulse Width (in milliseconds). For example, a 10ms pulse should use a ***Max Frequency*** value of 5000/10 = 500 Hz.

The ***Low pass filter*** parameter filters all frequencies above the specified frequency.

The ***Startup Pulse Interval*** parameter defines the time (in seconds) between output pulses while the controller is equalizing the system. A value of 0 will result in pulses output as fast as the controller can output them.

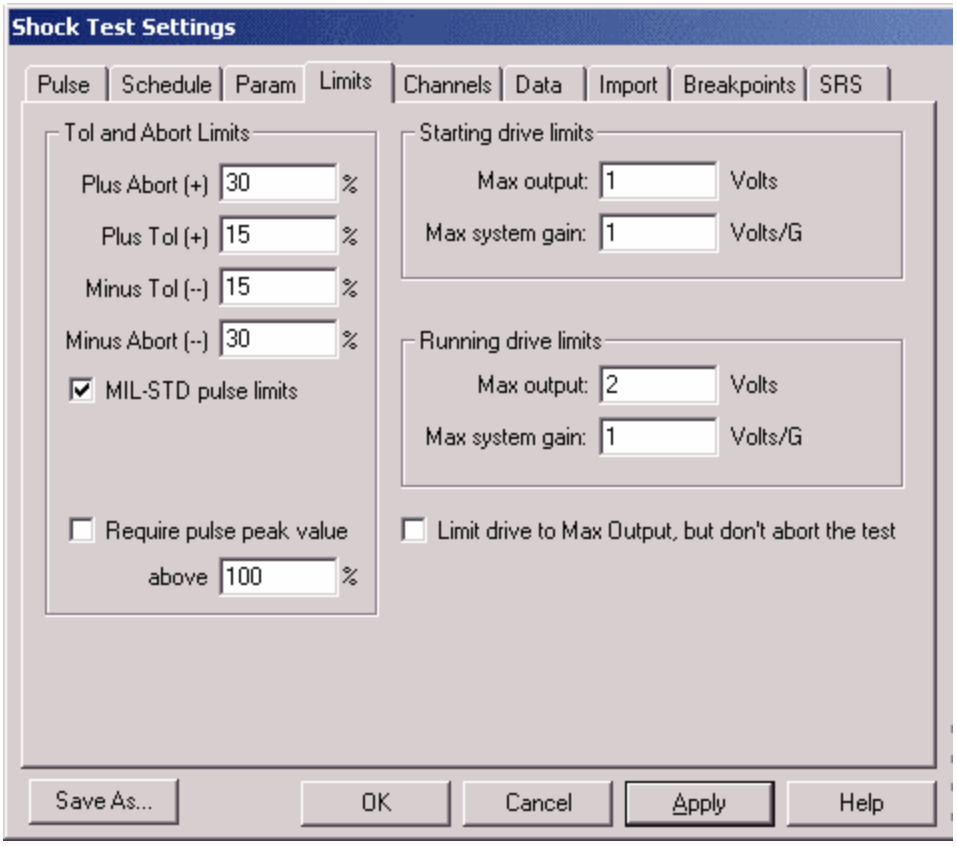
The ***System Lag (ms)*** parameter defines the expected lag (time-delay) in the amplifier and shaker system. This will adjust the initial timing of the drive pulse, and therefore can be used to improve the start-up time. It will not affect the steady-state behavior of the controller. A value of 0 is fine for most situations.

The ***Sample Rate*** parameter should in nearly all cases be set to Auto to allow the controller to automatically select an appropriate sample rate. However, experienced users have the option of entering the sample rate used by the controller.

In the case of user-defined pulses, setting the sample rate to Auto will be defined by the sample rate entered on the Import tab when the waveform was imported from an ASCII file. Manually setting the sample rate for user-defined pulses will result in playback of the waveform at different rate, changing the frequency content of the pulse, and generally should only be done by an experienced user. To get the same pulse shape and frequency content using a different sample rate, use a waveform editor program to resample/interpolate the original data, and then use the Import tab to import the resampled pulse.

4.3.3 Shock Define Limits tab

Clicking the "Limits" tab on the Shock Define dialog box accesses this dialog box.



Use this dialog box to define critical controller limits for a shock test

The ***+/- Tol*** parameters define the percentage of peak pulse acceleration at which the controller switches from startup mode to run mode.

The ***+/- Abort*** parameters define the percentage of peak pulse acceleration at which the controller aborts the test

Select ***MIL-STD Pulse Limits*** check box to enable MIL-STD constraints on the pulse. These limits enable the following tolerances:

- Tolerance is +/- 5% of the peak acceleration for one pulse width prior to the pulse.
- Tolerance is +/- 15% of the peak acceleration during the pulse
- Tolerance is +20% -30% of the peak acceleration for one pulse width after the pulse

The *starting drive limits* and *running drive limits* are also entered here. The starting drive limits apply while the controller is equalizing the pulse (i.e. while the test is starting). The running drive limits apply after the output level has reached the demand level (i.e. while the test is running).

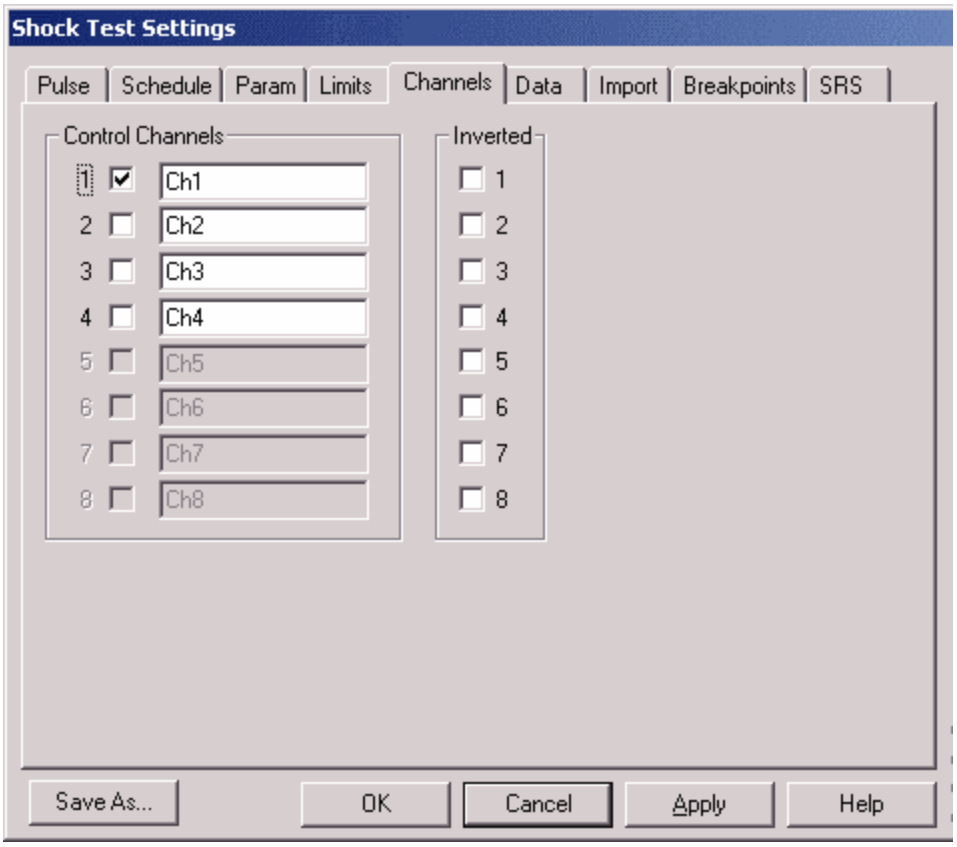
The ***Max output*** parameter defines an absolute voltage limit on the output. If this voltage is exceeded (in either the positive or negative direction) the test will abort.

The ***Max system gain*** parameter defines a limit relative to the output. The peak output voltage will not be allowed to exceed the peak acceleration value of the input signal multiplied by this system gain value. This is a safety limit that will prevent the control from outputting a pulse if there is no accelerometer connected to the system.

Appropriate values for the drive limits depend on your shaker and amplifier system. Typically you should first run the test to determine when peak output voltage is required. Then set the ***Max Output*** parameter to about 150% of the required level and set the ***Max System Gain*** parameter to 2 times the peak output voltage divided by the peak acceleration.

4.3.4 Shock Define Control Channels tab

Clicking the "Channels" tab on the Shock Define dialog box accesses this dialog box.

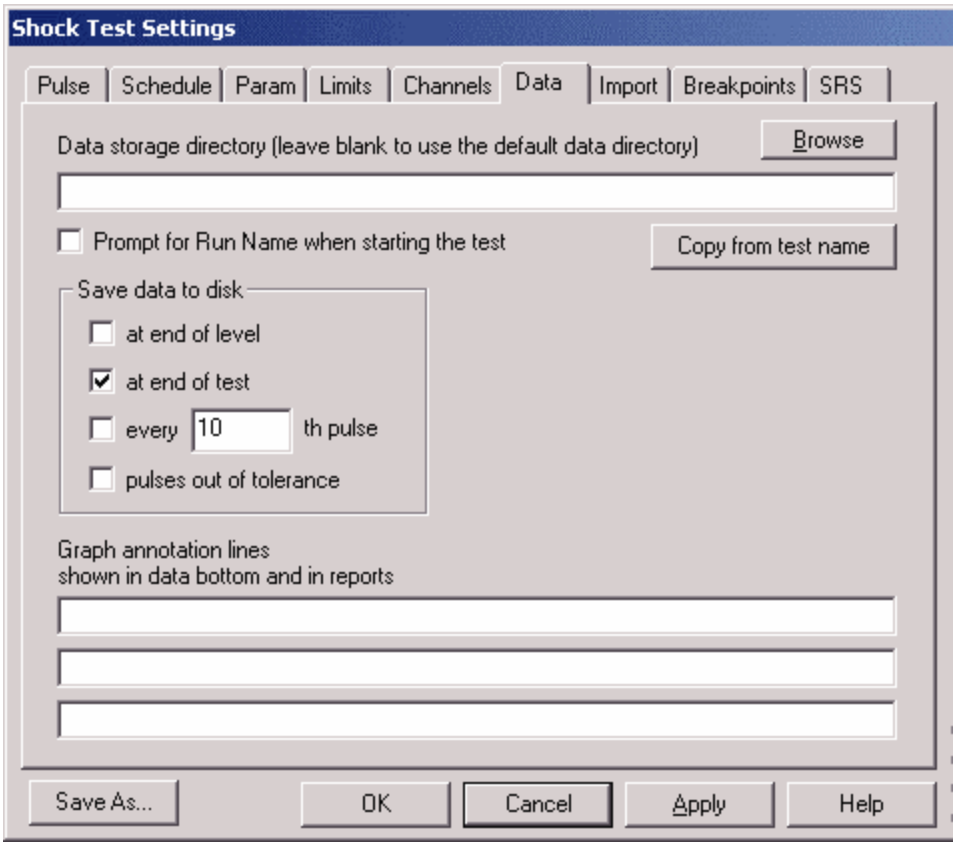


Use this dialog box to define which channel(s) will be used as the control signal. When multiple channels are selected, the control waveform will be the average of the selected input channels. This averaging is done on the time-domain signals.

You can also invert some of the input channels if some of your accelerometers have reversed polarity or must be mounted upside-down.

4.3.5 Shock Define Data Storage tab

Clicking the "Data Storage" tab on the Shock Define dialog box accesses this dialog box.



Use this dialog box to define the directory in which you want to store the data files generated by this test and the conditions under which a data file will be automatically stored.

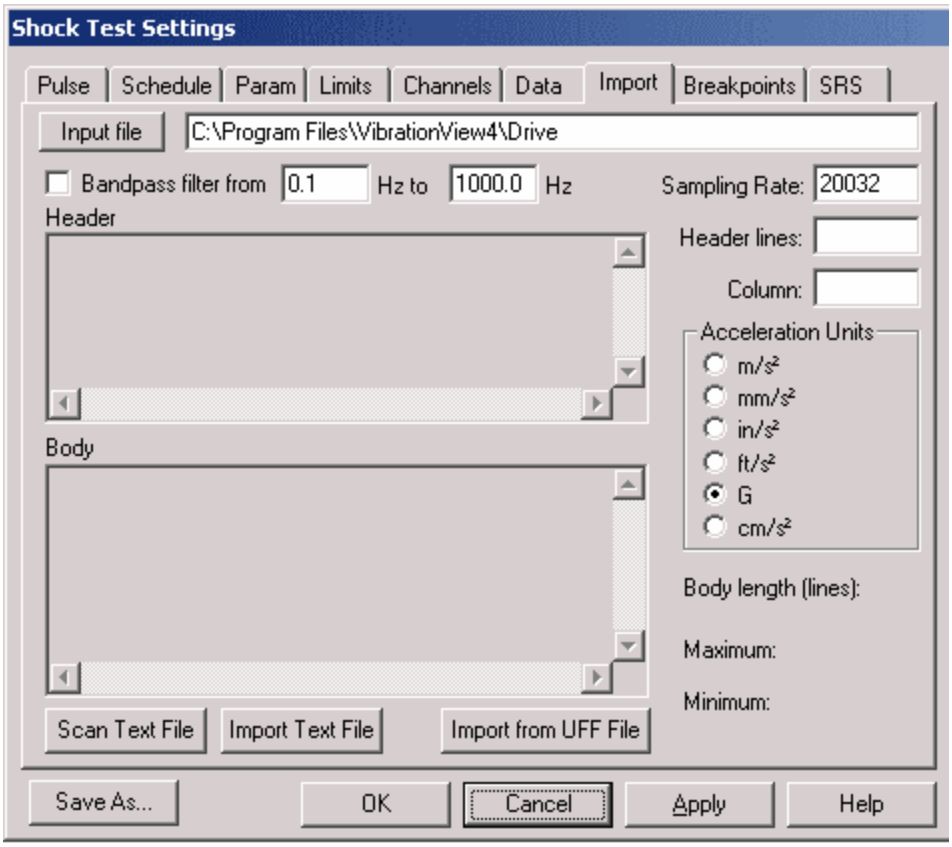
You can save the data automatically at the end of a level (as defined on the "Schedule" tab), at the end of a test, at a regular pulse interval and/or all pulses that are outside of the tolerance lines. Check the appropriate boxes to activate the automatic data store function(s). Check "Prompt for Run Name when starting the test" to display a prompt dialog box allowing the operator to enter a new directory each time the test is started.

You can also enter three lines that will appear in the Data Bottom portion of a graph. Editing a graph's settings and checking the "Data at Bottom" check box in the Shock Graph Settings dialog box enables this display.

All data is saved as time stamped files that can be viewed at a later time by selecting the File..Open Data menu command.

4.3.6 Shock Define Import tab

Clicking the "Import" tab on the Shock Define dialog box accesses this dialog box.



This dialog box allows the operator to import an arbitrary pulse shape, up to 20,000 samples long. The input must be a time/acceleration history stored as a text file.

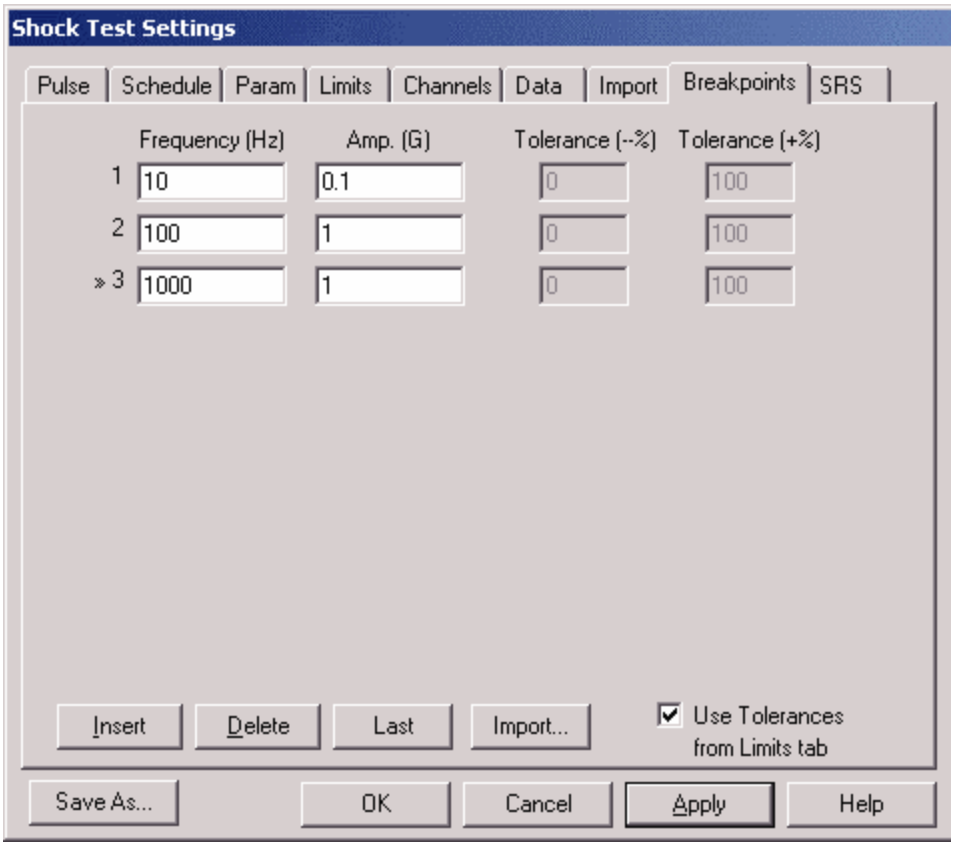
To input a waveform from an ASCII file, perform the following procedure:

1. Click the "Input file" button to select the file from which to read the data. The file will be scanned and the dialog box will be updated to reflect the contents of the file.
2. Set the sampling rate, number of header lines, the column number for the acceleration data, and the acceleration units used in the file.
3. Click the "Import Test File" button to copy the contents of the file into the user-defined waveform buffer. The program will switch to the Pulse tab. Select User-defined pulse type, and the graph will show the imported waveform.

Note that the User-defined shock pulse feature is an optional feature and requires the software key be programmed to enable it.

4.3.7 Shock Define Breakpoints tab

Clicking the "Breakpoints" tab on the Shock Define dialog box accesses this dialog box.



This dialog box is used to define the ***Frequency***/***Amplitude*** breakpoints of the desired Shock Response Spectrum (SRS) curve.

To delete a row, click a box in the row you want to delete and click the "Delete" button.

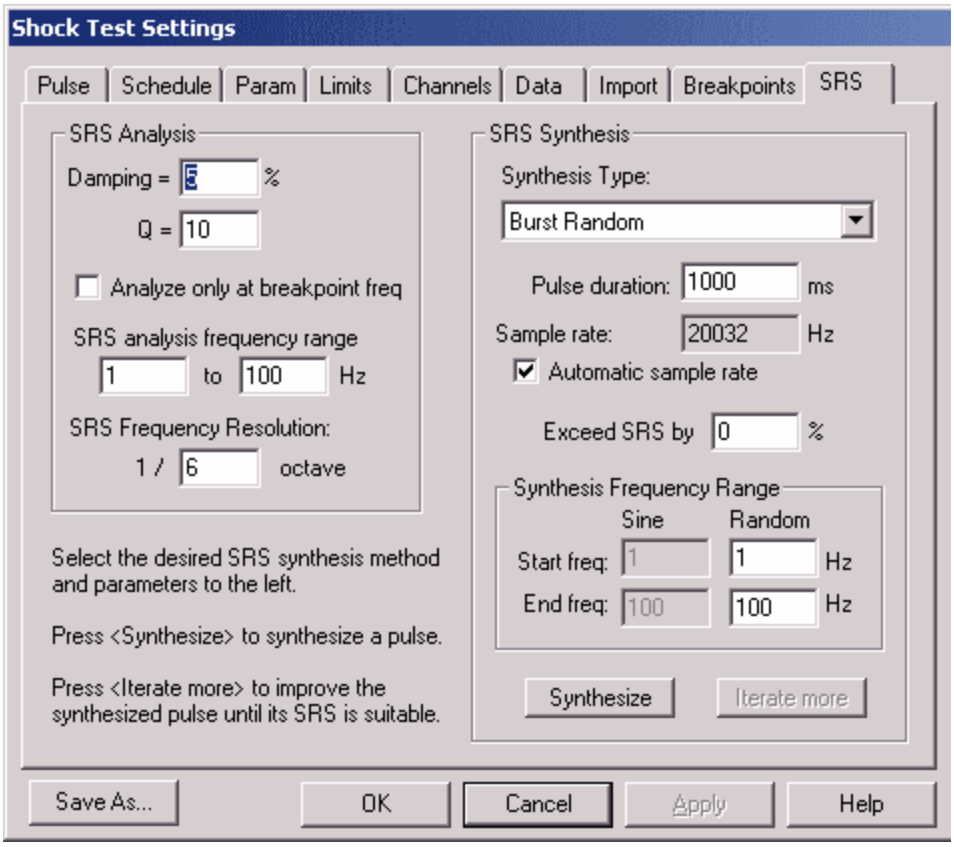
The "Last" button will remove all rows after the selected row. To use this, select the last row you want to keep and press the Last button. There will be a prompt to confirm that you want to delete *all* of the following rows.

The "Import..." button allows you to read a set of delimited (either whitespace or tab delimited) frequency and amplitude breakpoints from a text file.

You can also enter individual tolerance bands for each frequency in the table. If you want to use a single tolerance setting for all frequencies, check the "Use Tolerances from Limits tab" check box and enter the desired tolerance level on the Limits tab.

4.3.8 Shock Define SRS tab

Clicking the "SRS" tab on the Shock Define dialog box accesses this dialog box.



Use this dialog box to define the SRS (Shock Response Spectrum) analysis parameters and to synthesize a pulse to meet a specified SRS curve. Note: the SRS test method is an optional feature ,your software key be programmed to enable it.

The SRS Analysis parameters define the ***Damping*** and ***Q*** values used in the SRS calculation and the frequencies at which the SRS analysis is performed. The SRS calculations will be performed at the reference frequency and at frequencies with (1/N) octave spacing relative to the selected reference frequency. The frequency range used for SRS analysis will be from a lower frequency of 100/(Pulse Duration in ms) to up to half of the sample rate.

Tip: To force the analysis to a certain set of frequencies, check the "Analyze only at breakpoint freq" box, and enter the desired analysis frequencies in the SRS Breakpoints list. When that box is checked, only the frequencies listed in the SRS Breakpoints list will be used for SRS analysis.

The ***SRS Synthesis*** parameters are used to synthesize a pulse to match a specified SRS curve. Select the type of pulse with the ***Synthesis Type*** selection box. The ***Pulse Duration*** should be longer than the period of the lowest frequency in the SRS specification and optimally should about 10 times as long. The ***Sample Rate*** should be at least 4 times the highest frequency in the SRS specification and optimally should be 10 times the highest frequency.

The ***Start Frequency*** and ***End Frequency*** parameters control the chirp (frequency sweep) and burst random synthesis methods.

For frequency sweeps, to sweep up in frequency, set the ***End Frequency*** higher than the ***Start***

Frequency. To sweep down in frequency, set the ***End Frequency*** lower than the ***Start Frequency***. The lower frequency should be one tenth of the lowest frequency in the SRS specification and the higher frequency should be double the highest frequency in the SRS specification.

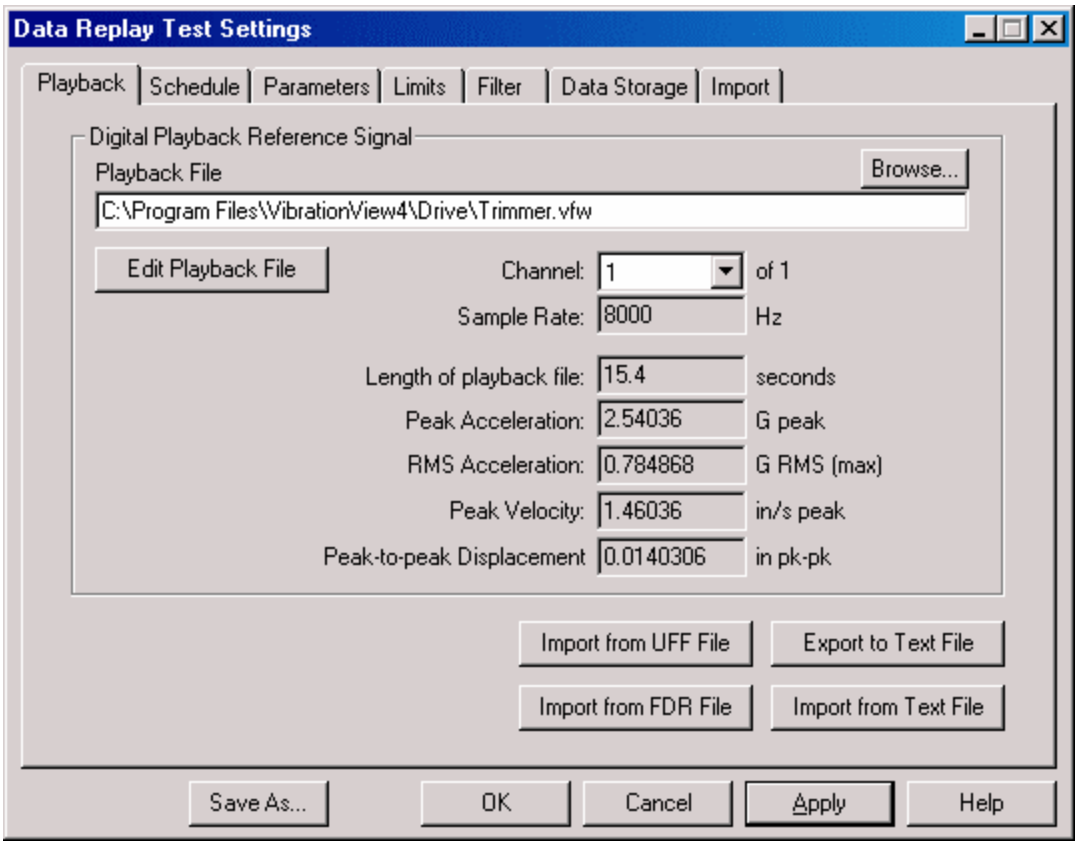
For burst random, the ***Start Frequency*** should be equal to or greater than the lowest frequency in the SRS specification and the ***End Frequency*** should be equal to or less than the highest frequency in the SRS specification.

After you have entered the SRS specification and synthesis parameters, press the ***Synthesize*** button to synthesize a pulse to meet the specification. The synthesis uses an iterative technique so each time you click the ***Iterate more*** button, the SRS of the synthesized pulse will get closer to the specified curve. Press the ***Synthesize*** button to restart the synthesis with a new set of parameters.

To see the synthesized acceleration, velocity and displacement curves, select the "Accel", "Vel" or "Disp" radio buttons under the bottom left corner of the graph. The "SRS" button will restore the SRS Profile Definition graph.

4.4 Field Data Replicator Playback dialog box

Selecting the Test..Edit Test menu command while a Field Data Replicator test is open displays this dialog box.



Use this dialog box to select the playback file (from the disk), along with the associated sample rate and recording level. Click the "Browse..." button to make a selection of previously recorded data.

The "Edit Playback File" launches the *CoolEdit*2000 file editor, with the currently selected playback file. If the test is running the playback file will be read locked. The test must be stopped prior to overwriting the playback file, or it can be saved to a new file.

The "Record Playback File" is available if a new test is under construction using File .. Define New Test wizard, and the optional data-recording package is installed. Recording is only available from the new test wizard. Record Playback File pushbutton activates the record dialog box

The "Import from UFF File" and "Export To Text File" buttons are used to convert from "Universal File Format" files.

The "Import from Text File" and "Export To Text File" buttons are used to convert to and from "Ascii Text" files.

The "Import from FDR File" button is used to convert files from FDR (Version 2 and 3 *VibrationVIEW* internal format).

The following tabs are available:

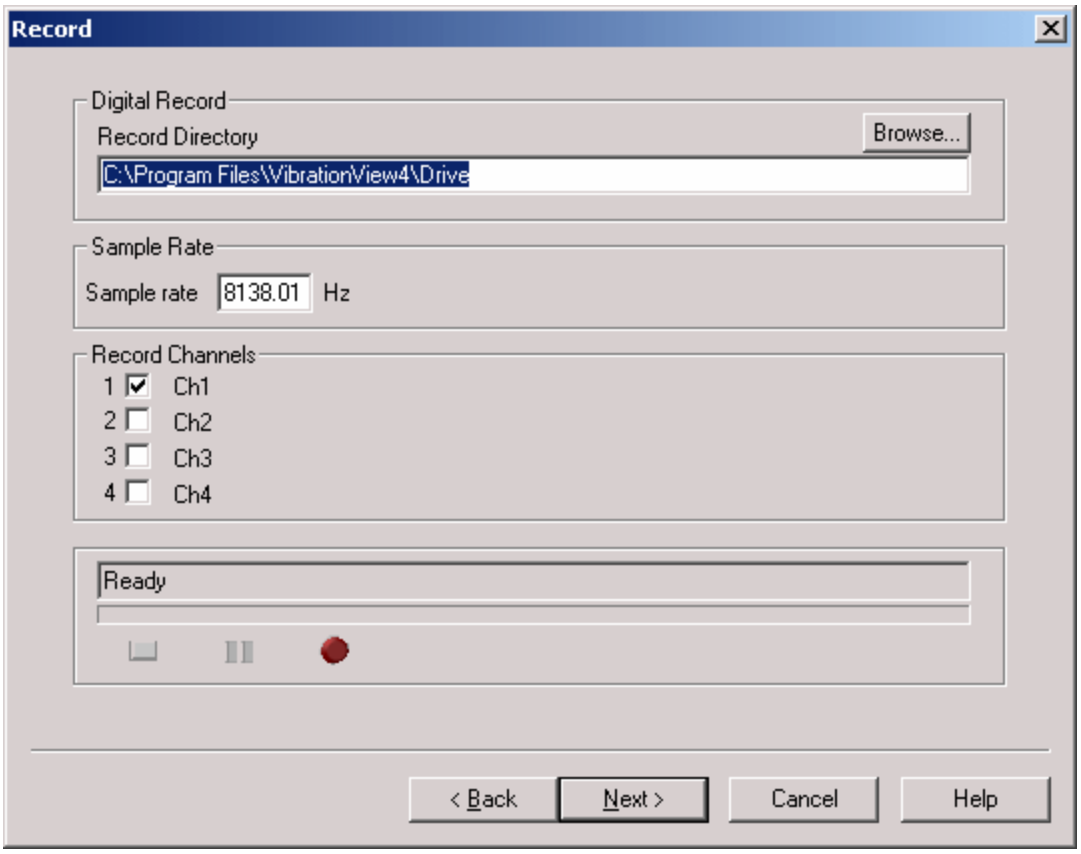
Playback	Enter the filename of and settings for the playback file
Schedule	Enter the test duration here
Parameters	Enter control parameters here.
Limits	Enter the safety limits here.
Filter	Enter filter frequency settings here.
Data Storage	Enter automatic data storage setting here.
Import	Convert ASCII files to FDR files

The following buttons are available:

Save As...	Save the current settings under a new test name.
OK	Save the changes to this test, and close the dialog
Cancel	Close the dialog box and abandon all changes
Apply	Apply the changes to the currently running test (does not save changes to the hard disk).
Help	Get help information about these parameters.

4.4.1 Data Record dialog box

Selecting the File..Define New Test menu command while a Field Data Replicator test is open, then pressing "Record Playback File" pushbutton displays this dialog box. Data record is available from existing tests using the "Settings" pushbutton on the Data Recorder tool bar. Access this toolbar using the View Data Recorder command.



Sample Rate: Recorder sample rate is available when creating a new Field Data Recorder test from the File Define New Test wizard, and also in System Check mode. Sine, Random, and Shock tests record at the sample rate defined for the loaded test.
Record Channels: Select which channels to record. Channel sensitivity is configured at the Configuration Inputs menu.

Record status and buttons are available when recording a new Field Data Recorder playback file from the File Define New Test wizard. To record when not setting up a new test use the Data Recorder Control Center.

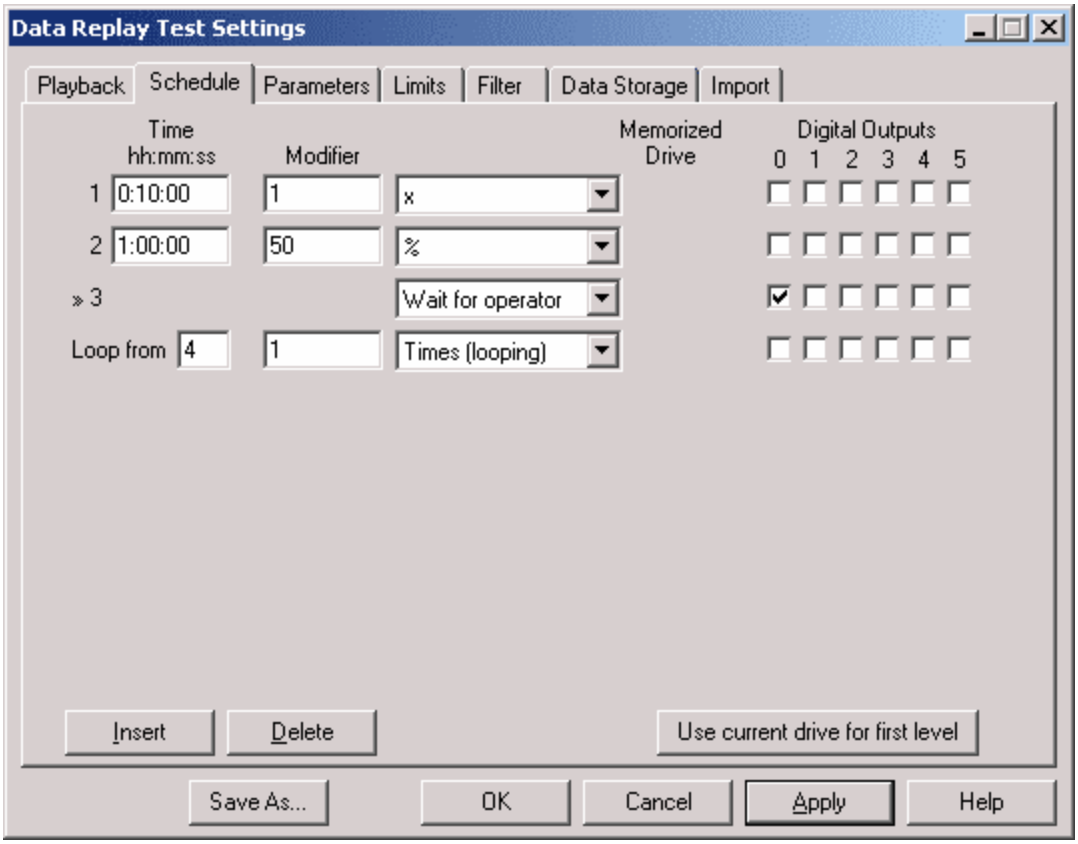
To import a new Field Data Replication test from and external source (e.g. DAT recorder), setup an input for the sensitivity required for the playback device with the Configuration Inputs menu. Any available input can be setup for recording.

Press the "Next >" button once to start recording.

Press the "Next >" button again to stop recording, and select your saved file. You are automatically returned to the playback tab with the recorded file setup for playback.

4.4.2 Field Data Replicator Define Schedule tab

Clicking the "Schedule" tab on the Field Data Replicator Define dialog box accesses this dialog box.



Use this dialog box to define how long your test will run. You may enter up to 200 levels, each of which has an associated amplitude gain (modifier) level. Modifier types are "x" (linear multiple), % (percentage level) and dB (logarithmic multiple). These three types of modifiers are related in the following manner:

x	%	dB
0.25	25	-6
0.5	50	-3
1	100	0
2	200	3
4	400	6

The modifier level is the multiplier applied to the reference input. For example, if the control is running at 1.5 G rms, and the multiplier is set to "2 x", the controller will control the shaker at 3 G rms (double the reference level).

The time setting, entered in the form hh:mm:ss, is the amount of time in hours, minutes and seconds that the controller will spend at each amplitude level.

There are also two other types of modifiers: "Times" and "Wait for operator". The "Times" modifier is used to enter a repeated loop in the test schedule, if you want to run a test that cycles repeatedly through a certain sequence of amplitude levels. The "Wait for operator" modifier will pause the test until the operator clicks the "Continue" or "abort Test" button in the dialog box that appears when the test reaches the "Wait for operator" level.

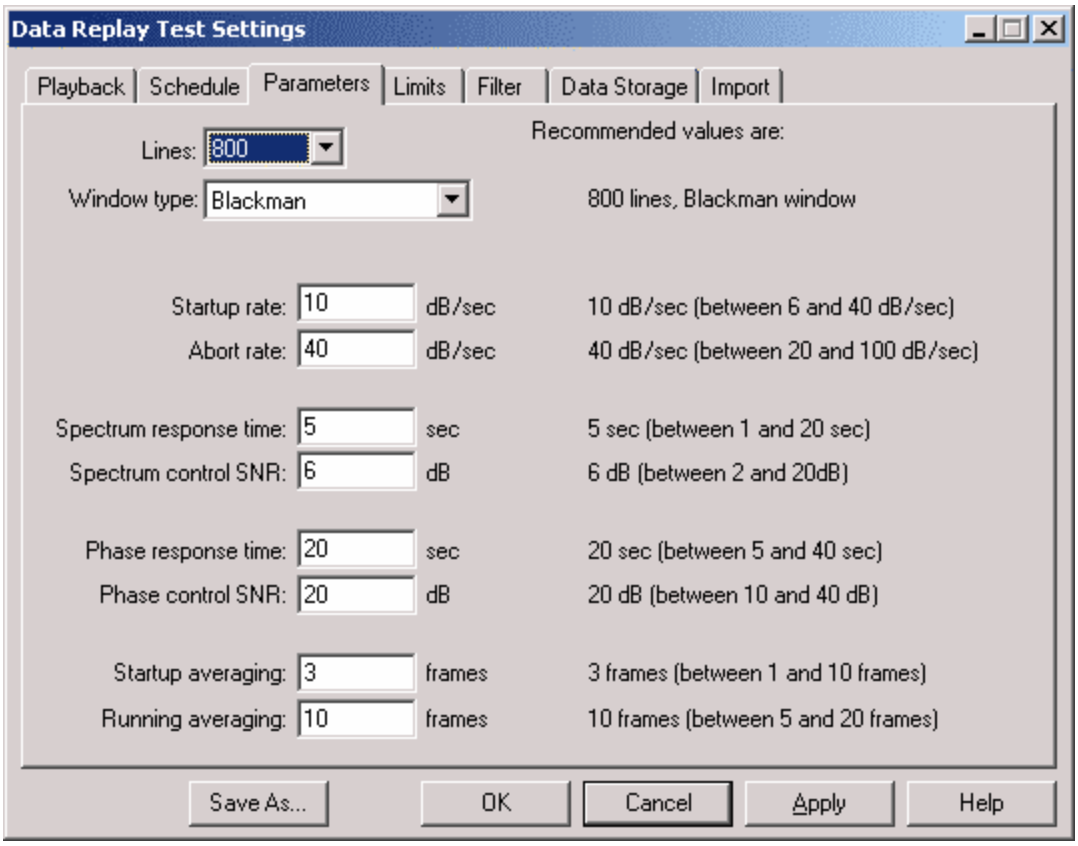
The "Memorized Drive" setting enables fast startup and fast switching from one level to another. If the memorized drive check box is not checked, the controller will shut down the output when switching to that

level, and then slowly bring the output up to the desired level. If the memorized drive check box is checked, the controller will quickly switch the output to the new level.

To use a memorized drive for the first level, you must first start up the controller so that it learns the frequency response function for your system, and then click the "Use current drive for first level" button to store this drive function with the test.

4.4.3 Field Data Replicator Define Parameters tab

Clicking the "Parameters" tab on the Field Data Replicator Define dialog box accesses this dialog box.



Use this dialog box to set the control function parameters. These numbers determine how the control algorithm reacts.

The error gain is the multiplier by which the error is multiplied each control loop. The bigger the multiplier is, the faster the control will react. You should keep this number large enough to get to level in a reasonable time, but small enough so that sudden changes in level (potholes) do not adversely affect the control. The gain values should not be greater than 1.0, and typically should be around 0.1.

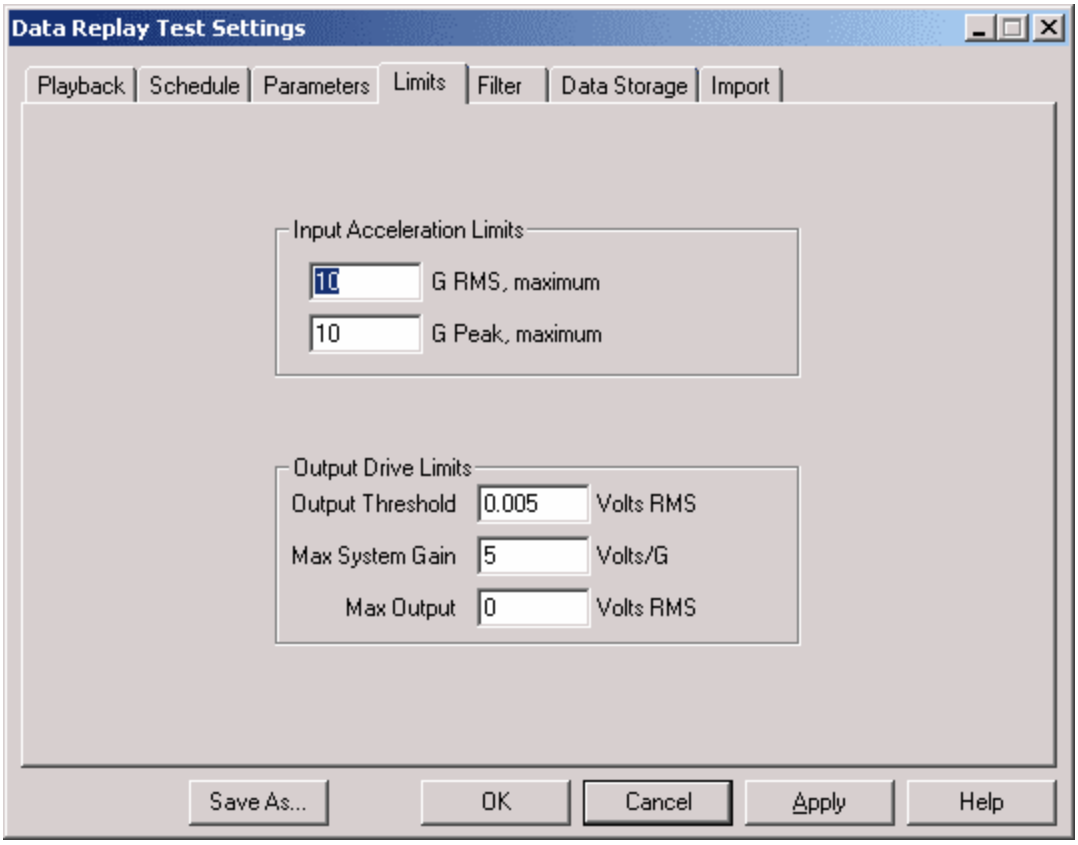
The averaging values set the amount of temporal averaging applied to the spectra during startup and normal operation. A typical value for startup is 3, and for run is 10.

The SNR values for RMS Error, Spectrum Error, and Phase Error should be left at 2, 2, and 40, respectively.

See also: How to tune Field Data Replicator controller parameters.

4.4.4 Field Data Replicator Define Limits tab

Clicking the "Limits" tab on the Field Data Replicator Define dialog box accesses this dialog box.



Use this dialog box to set the abort limits in the Field Data Replicator software.

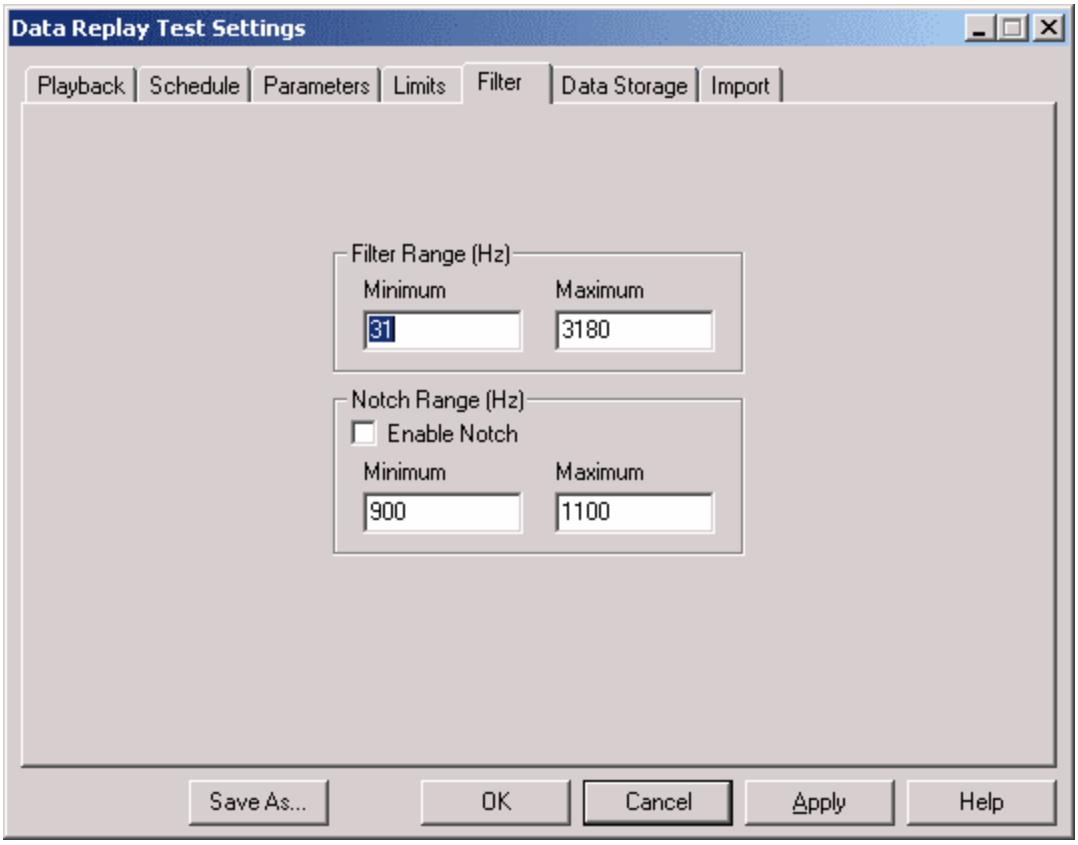
The Control (Ch1) Max is the maximum RMS acceleration level allowed on the control channel. (Channel 1 is the control input from the shaker accelerometer) If this level is ever exceeded the test will abort.

The Reference (Ch2) Max is the maximum RMS acceleration level allowed on the reference channel (Channel 2 is the reference input). If this level is ever exceeded the test will abort.

The drive limit is the maximum RMS output voltage allowed from the control loop. If the channel 1 output exceeds this value, the test will abort. We recommend setting this at about 150% of the normal maximum voltage level required to run your test.

4.4.5 Field Data Replicator Define Filter tab

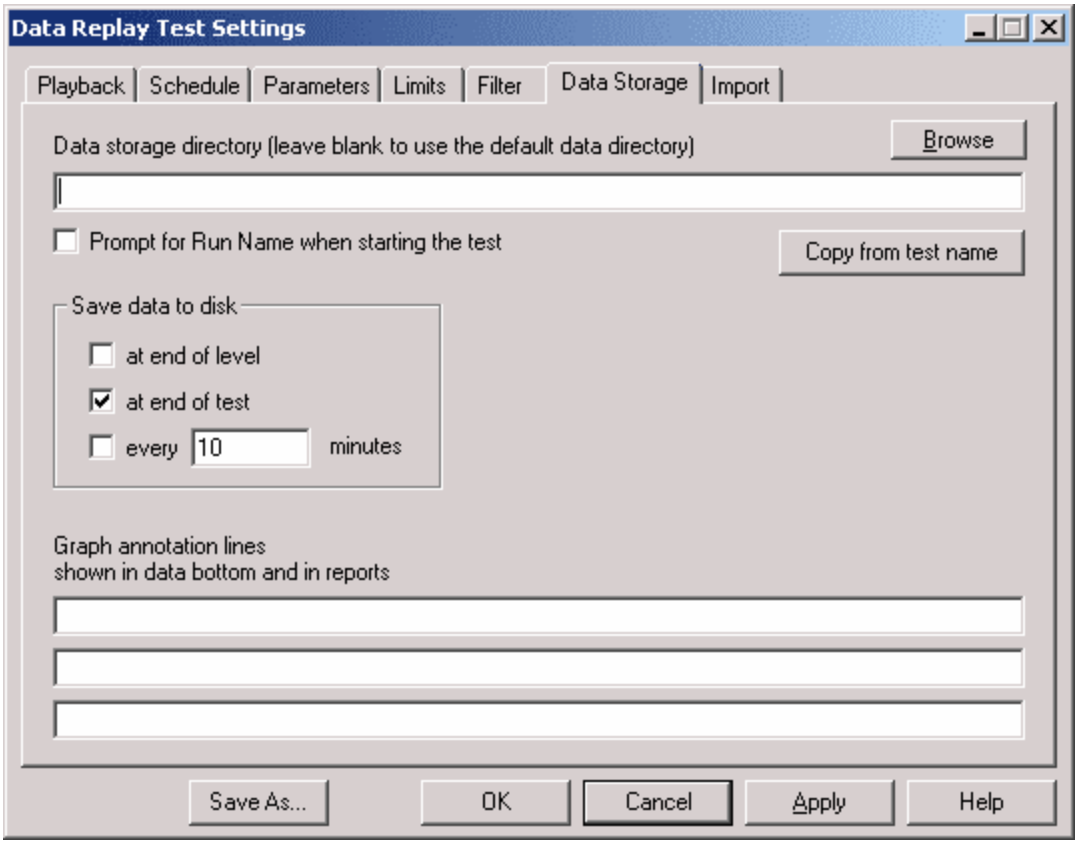
Clicking the "Filter" tab on the Field Data Replicator Define dialog box accesses this dialog box.



Use this dialog box to display and adjust the filter frequency settings, i.e. to set the frequency range over which the feedback control will operate. You can also enable a notch band to filter out a frequency range in the middle of the spectrum. This is useful for such things as a shake-and-rattle test, where you can see which frequencies are exciting a rattle mode.

4.4.6 Field Data Replicator Define Data Storage tab

Clicking the "Data Storage" tab on the Field Data Replicator Define dialog box accesses this dialog box.

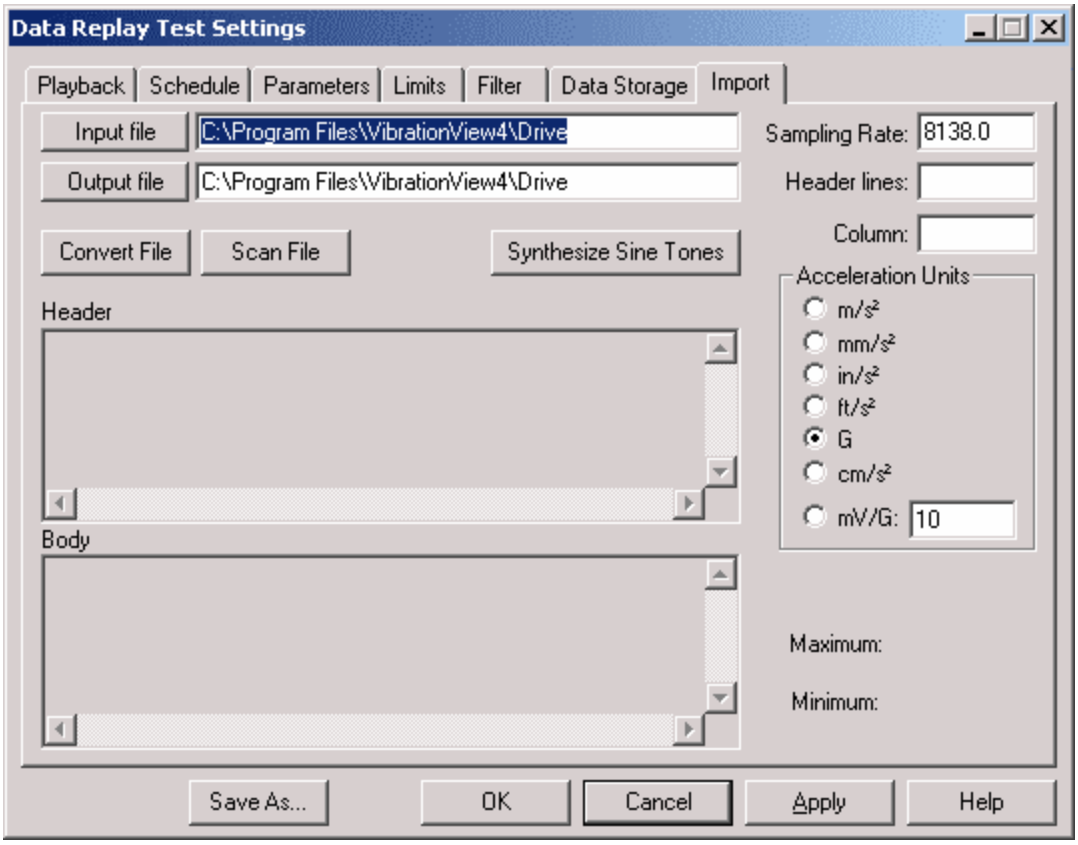


Use this dialog box to set the times at which the test data is automatically saved when the test is running. You can save the data at the end of a level, at the end of a test, or at the end of a time period. All data is saved as a time stamped file that can be viewed at a later time by selecting the File..Open Data menu command. Check "Prompt for Run Name when starting the test" to display a prompt dialog box allowing the operator to enter a new directory each time the test is started.

Check the boxes to activate the desired data storage functions.

4.4.7 Field Data Replicator Define Import tab

Clicking the "Import" tab on the Field Data Replicator Define dialog box accesses this dialog box.



Use this dialog box to import a text file with comma, semicolon, or white space-delimited data into an FDR format file for use as the playback file in Field Data Replicator tests. Select an input file by clicking the "Input file" button. Set the appropriate sampling rate, column number and acceleration units. Click the "Convert File" button to convert the file to the *VibrationVIEW* internal format (FDR format).

5 Control Centers

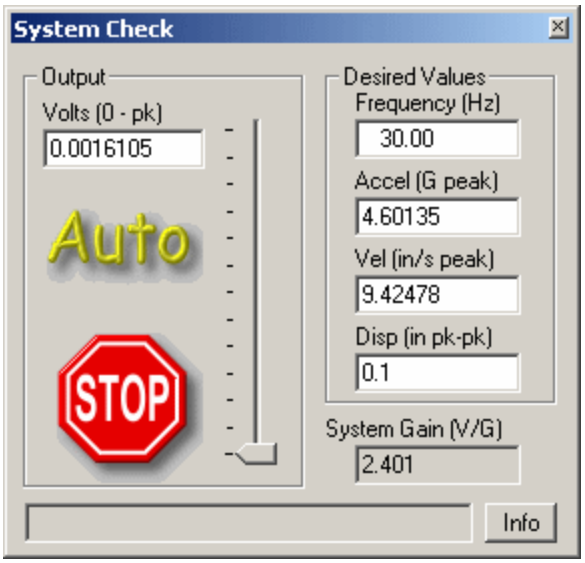
The control centers are docking toolbars that can be:

- Attached to either the left or right side of the VibrationVIEW window
- Detached and moved independently
- Turned on/off by selecting the View..Control buttons menu command.

Test Control Centers

- System Check Control Center
- Sine Control Center
- Random Control Center
- Classical Shock Control Center
- Field Data Replicator Control Center

5.1 System Check Control Center



Use this dialog box to check the system operation. We recommend that you perform this function as a prelude to any new testing operation. This is a good way to verify that the accelerometer, amplifier, shaker and control system are all functioning. To use, place a displacement wedge on your shaker table. The operation defaults are 0.10 inches at 30 Hz. Turn the shaker on. Click the "Auto" button. Watch the shaker table carefully. It should begin to oscillate at a visible level of 0.10 inches.

Auto: This button automatically adjusts the output voltage so that the channel 1 input matches the Desired Values of Acceleration, Velocity, and Displacement.

Stop: This button immediately returns the output voltage to 0 (i.e. turn the output off).

Volts (0 - pk): The output voltage level (peak value of the sine wave) is entered here.

Voltage level: You can manually set this with the mouse. Although it is not as accurate as the Volts (0 - pk) box, it is useful when moving quickly from a high voltage to a low one (or vice versa).

Frequency: The Frequency to use for the output signal is entered here. The default of 30 Hz works well on most systems.

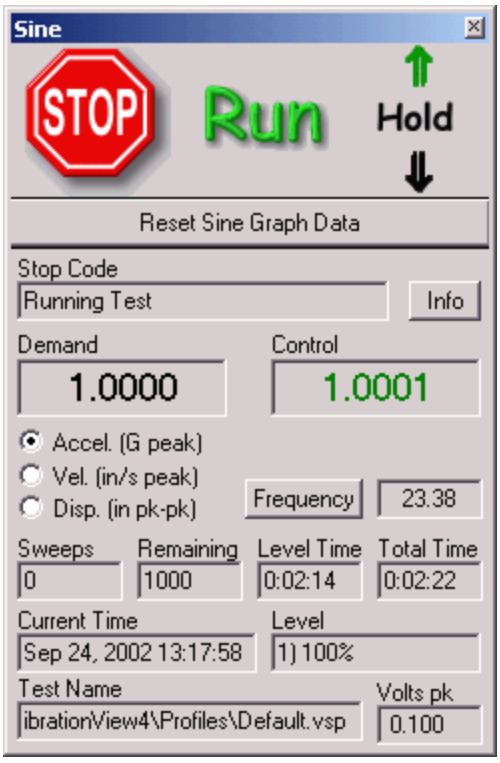
Accel: The desired peak acceleration for input channel 1 is entered here. This is used when Automatic mode is enabled.

Velocity: The desired peak velocity for input channel 1is entered here. This is used when Automatic mode is enabled.

Disp: The desired peak-to-peak displacement for input channel 1 is entered here. This is used when Automatic mode is enabled.

See Also: System Check Accelerometers Toolbar

5.2 Sine Control Center



Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test and want to continue where you left off, click the "Run" button and click the "Yes" button in the dialog box that asks if you want to continue or start from the beginning.

Run: This button starts the test. Click this button when you are ready to start your test. This button will be gray when you are not running, yellow when you are starting, light blue if you are starting with a memorized drive and green when you are running.

Sweep Up: This button causes the frequency sweep to proceed with increasing frequencies.

Sweep Hold: This button freezes the output frequency at the current value.

Sweep Down: This button causes the frequency sweep to proceed with decreasing frequencies.

Reset Sine Graph Data: This button clears the historical high/low/last pass data from the graphs.

Mode Field: These radio buttons allow you to easily change what type of information the Sine Accelerometers Toolbar is displaying. You can choose acceleration, velocity or displacement. The mode affects both the Control and Demand display and the Sine Accelerometers Toolbar. The units can also be changed for these readings (see Units).

Frequency: This box displays the current operating frequency (when running). You can use the button next to this display to define a new starting frequency.

Sweeps: This box displays how many sweeps the test has run.

Remaining: This box displays how many sweeps are remaining.

Current Time: This box displays the current date and time.

Level Time: This box displays how long you have been at the current level.

Test Name: This box displays the name of the test file that you are running.

Total Time: This box displays how much time has elapsed since the test started.

Demand: This box displays the desired acceleration/velocity/displacement level for the test, as selected using the Mode radio buttons.

Control: This box displays the measured acceleration/velocity/displacement level for the control signal, as selected using the Mode radio buttons.

Volts Pk: This box displays the output voltage level.

Level: This box displays the current level at which the test is running.

Stop Code/Info: This box displays why the test shut down. Click the "Info" button for more information on the current status of the test. See also: Sine Stop Codes

See Also: Sine Accelerometers Toolbar

5.3 Random Control Center

Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test and want to continue where you left off, click the "Run" button and click the "Yes" button in the dialog box that asks if you want to continue or start from the beginning.

Run: This button starts the test. Click this button when you are ready to start your test. This button will be gray when you are not running, yellow when you are starting, light blue if you are starting with a

memorized drive and green when you are running.

Hold Timer: When this button is yellow, the level time counter is paused. This is useful in combination with the Open Loop button to pause the test in order to adjust the item under test.

Open Loop: When this button is yellow, the output drive spectrum remains fixed (the control loop updates are disabled). This is useful for disabling the control loop temporarily in order to turn off the shaker amplifier and adjust the item under test.

Current Time: This box displays the current date and time.

Remain: This box displays how much time is remaining at the current level of the test schedule.

Lev. Time: This box displays how long the test has been running at the current level of the test schedule.

Total Time: This box displays how much time has elapsed since the test started.

Test Name: This box displays the name of the test file that you are running.

Demand: This box displays the expected RMS acceleration and pk-pk displacement level for the test.

Control: This box displays the measured RMS acceleration and pk-pk displacement level of the control signal.

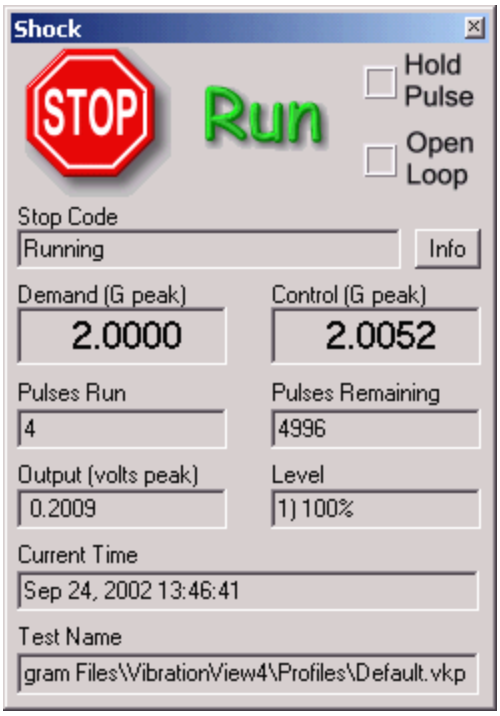
Volts rms: This box displays the output voltage (RMS) level.

Level: This box displays the current level number and modifier value that the test is running.

Stop Code/Info: This box displays why the test shut down. Click the "Info" button for more information on the current status of the test. See also: Random Stop Codes.

See Also: Random Accelerometers Toolbar

5.4 Shock Control Center



Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test and want to continue where you left off, click the "Run" button and click the "Yes" button in the dialog box that asks if you want to continue or start from the beginning.

Run: This button runs the test. Click this button when you are ready to start your test. This button will be gray when you are not running, yellow when you are starting, light blue if you are starting with a memorized drive and green when you are running.

Hold Pulse: This button stops the pulses and thus temporarily pauses the test. This allows you to adjust or exchange the part under test without stopping the test.

Open Loop: This button repeats the output signal continuously, without control loop updates. This allows you to maintain a constant output drive signal throughout the test.

Test Name: This box displays the name of the file that you are running.

Pulses Run: This box displays how many shock pulses have been outputted since this test level began. When a long pulse is being output, this box will also indicate how much of the current pulse is complete.

Pulses Remaining: This box displays how many shock pulses the test still has to run at this level.

Output: This box displays the current peak output voltage of the test.

Level: This box displays the current level number and modifier value the test is running.

Demand: This box displays the desired peak acceleration value.

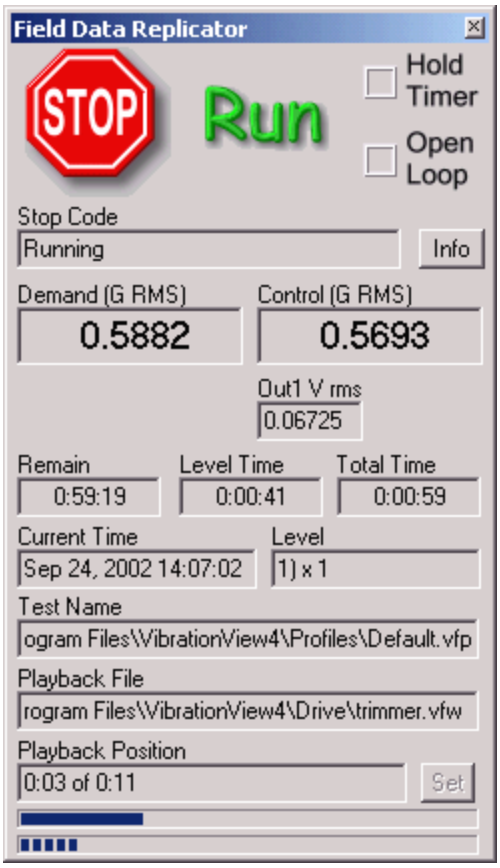
Control: This box displays the measured peak acceleration value.

Stop Code/Info: This box displays why the test shut down. Click the "Info" button for more information

on the current status of the test. See also: Classical Shock Stop Codes

See Also: Classical Shock Accelerometers Toolbar

5.5 Field Data Replicator Control Center



Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test and want to continue where you left off, click the "Run" button and click the "Yes" button at the dialog box that asks if you want to continue or start from the beginning.

Run: This button runs the test. Click this button when you are ready to start your test. This button will be gray when you are not running, yellow when you are starting, light blue if you are starting with a memorized drive and green when you are running.

Current Time: This box displays the current date and time

Test Name: This box displays the name of the file that you are running.

Playback File: This box displays the file that you are reading from in your test.

Ctrl: This box displays the RMS acceleration measurement of the control input.

Level Time: This box displays how long you have been at the current level.

Ref: This box displays the RMS acceleration level of the reference signal.

Total Time: This box displays how much time has elapsed since the test started.

V rms: This box displays what the current output voltage level is.

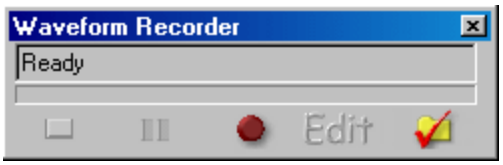
Multiplier: This box displays the factor applied to the reference level to increase/decrease the control level.

Level: This box displays the current level that the test is running.

Stop Code/Info: This box displays why the test shut down. Click the "Info" button for more information on the current status of the test. See also: Field Data Replicator Stop Codes.

See Also: Field Data Accelerometers Toolbar

5.6 Data Record Control Center



Stop: Stop the record and prompt for a file to save waveform data.
Pause: Pause the record, and hold the file open, resume paused record by pressing again
Record: Start recording the selected data to a new file, or resume paused record if previously paused.
Edit: Edit the most recent recorded file with optionally installed Cool Edit waveform editor.
Settings: Adjust the recorder settings including default directory and selected record channels.

6 Accelerometers Toolbars

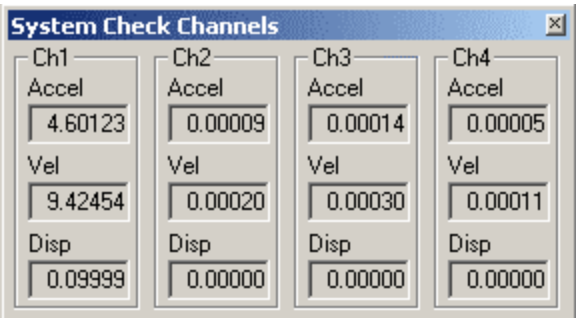
The Accelerometer Toolbars are docking toolbars that can be:

- Attached to either any side of the VibrationVIEW window.
- Detached and moved independently.
- **Turned on/off by selecting the View..Accelerometers menu command.**

Accelerometer Toolbars

- System Check
- Sine
- Random
- Classical Shock
- Field Data Replicator

6.1 System Check Accelerometers Toolbar

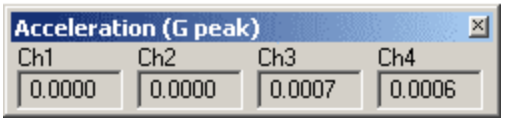


This toolbar displays the acceleration, velocity and displacement values for the active channels in your System Check test.

The channel label is highlighted **RED** if an accelerometer problem is detected.

See also: System Check Control Center

6.2 Sine Accelerometers Toolbar



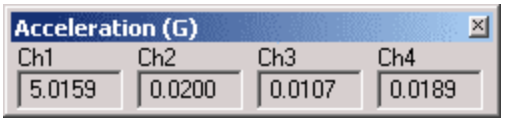
This toolbar displays the acceleration, velocity or displacement values for the active channels in your System Check test.

To switch the display between acceleration, velocity and displacement, click the desired radio button in the Sine Control Center.

The channel label is highlighted **RED** if an accelerometer problem is detected.

See Also: Sine Control Center

6.3 Random **Accelerometers Toolbar**

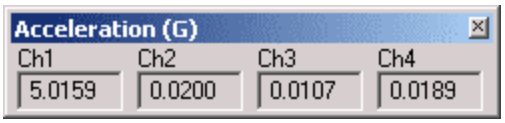


This toolbar displays the measured g RMS acceleration levels for each of the input channels

The channel label is highlighted **RED** if an accelerometer problem is detected.

See Also: Random Control Center

6.4 Classical Shock Accelerometers Toolbar

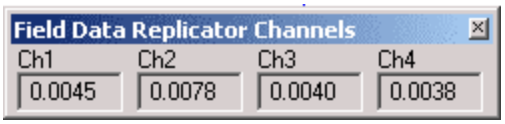


This toolbar displays the measured peak acceleration levels for each of the input channels

The channel label is highlighted **RED** if an accelerometer problem is detected.

See Also: Classical Shock Control Center

6.5 Field Data Replicator Accelerometers Toolbar



This toolbar displays the measured g RMS acceleration levels for each of the input channels

The channel label is highlighted **RED** if an accelerometer problem is detected (open or shorted accelerometer).

See Also: Field Data Replicator Control Center

7 Stop Codes

- System Check Stop Codes
- Sine Stop Codes
- Random Stop Codes
- Classical Shock Stop Codes
- Field Data Replicator Stop Codes

7.1 System Check Stop Codes

The following codes indicate that something has occurred to cause the controller to abort the current test.

Input channel n is clipping
No acceleration detected on channel n
Open accel on channel n
Remote Start
Remote Stop
Shorted accel on channel n
Stop Button Pressed
Waiting for box n

Daq reset (Box n)
Desired Acceleration is too high
Desired Displacement is too high
Desired Velocity is too high
Emergency Stop
Lost contact with I/O box n
Max Sine Acceleration on channel n
Max Sine Displacement on channel n
Max Sine Velocity on channel n
Test Parameters Changed
Watchdog Timeout

7.1.1 Input channel N is clipping (System Check Stop Code)

This code indicates that the input signal to channel n is exceeding the allowable input voltage range of +/- 10 volts, and is being distorted by clipping. To solve this problem, use an accelerometer with a lower mV/G sensitivity rating.

The maximum allowable acceleration readings for various accelerometer sensitivities are shown in the table below.

Sensitivity	Maximum Acceleration
10 mV/G	1000 G peak
20 mV/G	500 G peak
50 mV/G	200 G peak
100 mV/G	100 G peak
1000 mV/G	10 G peak

7.1.2 No acceleration detected on channel n (System Check Stop Code)

This code indicates that "Auto" mode disabled itself because the output was increasing, but there was no acceleration measured on the channel 1 input. This is just a safety precaution to prevent the controller from increasing the output without limit when no accelerometer feedback is present.

Verify that:

- There is an accelerometer connected to Channel n on the VibrationVIEW I/O box.
- The cable connections are all secure.
- Drive from the VibrationVIEW I/O box is connected to the amplifier.
- The amplifier gain is set to a reasonable value.

If you can see the shaker moving, but don't see a sine wave on the Input Waveform graph (i.e. the Input

Waveform graph just shows noise), double-check your accelerometer connections.

If everything appears to be correct, press the "Auto" button again. Each time you press the "Auto" button, the output voltage will increase to double it's previous value, and the controller will again check to see if acceleration is measured on the channel n input.

If your shaker has a threshold voltage which must be exceeded before the shaker starts moving (servo-hydraulic shakers typically exhibit this behavior) you may need to press the "Auto" button several times before the shaker starts to move and the controller detects the acceleration on input channel 1. Once the threshold is overcome, the "Auto" button will remain green, and the controller will adjust the output voltage to reach the desired acceleration level.

7.1.3 Open accel on channel n(System Check Stop Code)

This code indicates that an accelerometer connection is open.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.1.4 Remote Start (System Check Stop Code)

This code indicates that the test was switched into Automatic mode using the Remote Start/Stop option.

7.1.5 Remote Stop (System Check Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option.

7.1.6 Shorted Accelerometer (System Check Stop Code)

This code indicates that an accelerometer connection is shorted.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.1.7 Stop Button Pressed (System Check Stop Code)

This code indicates that the output was turned off because the stop button was pressed.

7.1.8 Waiting for box n (System Check Stop Code)

This code indicates that more than one control box is defined and the system is waiting for all the boxes to reach the ready state.

If you have a single control box system (4 channels or fewer), check the Configuration..Parameters dialog box and remove any boxes assigned to higher channels.

7.1.9 Daq Reset (System Check Stop Code)

This code indicates a failure initializing the D/A and A/D converters in the 8500 I/O box. Under normal conditions this message should never appear. If you get this error, please contact Vibration Research Corp. at (616) 669-3028 or support@vibrationresearch.com for assistance.

7.1.10 Desired Acceleration is too high (System Check Stop Code)

This code indicates that the acceleration value entered in the Desired Values section exceeds the maximum acceleration allowed for the system. This limit is defined in the System Limits dialog box. It is a combination of the Force limits of the shaker system and the mass of the shaker and fixture (defined in the System Mass dialog box) using Newton's first law:

$$a = F/m$$

where:

a = acceleration limit of the shaker
F = Force limit of the shaker
m = mass that the shaker must move (sum of the system masses)

If you get this error message, either the parameters in the System Limits and/or System Mass dialog boxes need to be adjusted to reflect the capabilities of your shaker or the current test requires a larger shaker than you are using.

7.1.11 Desired Displacement is too high (System Check Stop Code)

This code indicates that the displacement value entered in the Desired Values section exceeds the maximum displacement allowed for the system. This limit is defined in the System Limits dialog box.

If you receive this error message, either the displacement limit parameter in the System Limits dialog box needs to be adjusted to reflect the capabilities of your shaker or the current test requires a shaker with a larger displacement range than you are using.

7.1.12 Desired Velocity is too high (System Check Stop Code)

This code indicates that the velocity value entered in the Desired Values section exceeds the maximum velocity allowed for the system. This limit is defined in the System Limits dialog box.

If you get this error message, either the velocity limit parameter in the System Limits dialog box needs to be adjusted to reflect the capabilities of your shaker or the current test requires a larger shaker than you are using.

7.1.13 Emergency Stop (System Check Stop Code)

This code indicates that the emergency stop button was pressed, causing the controller to shut down. The emergency stop button is an optional accessory that connects to a special connector on the front of the *VibrationVIEW* I/O unit.

7.1.14 Lost contact with I/O box n (System Check Stop Code)

This code indicates that VR8500 is no longer communicating with the host computer.

Verify that:

- The Ethernet cables are connected properly.
- The box is powered.

7.1.15 Max Sine Acceleration on channel n (System Check Stop Code)

This code indicates that the maximum acceleration allowed for the system has been exceeded. This limit is defined in the System Limits dialog box. It is a combination of the Force limits of the shaker system and the mass of the shaker and fixture (defined in the System Mass dialog box) using Newton's first law:

a = F/m

where:

a = acceleration limit of the shaker
F = Force limit of the shaker
m = mass that the shaker must move (sum of the system masses)

If you get this error message, either the parameters in the System Limits and/or System Mass dialog boxes need to be adjusted to reflect the capabilities of your shaker or the current test requires a larger shaker than you are using.

7.1.16 Max Sine Displacement on channel n (System Check Stop Code)

This code indicates that the maximum displacement allowed for the system has been exceeded. This limit is defined in the System Limits dialog box.

If you receive this error message, either the displacement limit parameter in the System Limits dialog box needs to be adjusted to reflect the capabilities of your shaker or the current test requires a shaker with a larger displacement range than you are using.

7.1.17 Max Sine Velocity (System Check Stop Code)

This code indicates that the maximum velocity allowed for the system has been exceeded. This limit is defined in the System Limits dialog box.

If you receive this error message, either the velocity limit parameter in the System Limits dialog box needs to be adjusted to reflect the capabilities of your shaker or the current test requires a larger shaker than you are using.

7.1.18 Test Parameters Changed (System Check Stop Code)

This code indicates that the output was turned off because a critical test parameter such as the sampling rate was changed. Press the "Auto" button to resume the test.

7.1.19 Watchdog Timeout (System Check Stop Code)

This code indicates that the communications link between the *VibrationVIEW* program and the Signal Processing hardware was lost. Under normal operations this should never happen. Click the "Run" button to reset the timeout.

The most likely cause of this error is a loss of power to the *VibrationVIEW* I/O unit.

Other events that could cause this to happen:

- The VibrationVIEW I/O unit is turned on after the program is started.
- The Ethernet cable connecting the computer with the VibrationVIEW I/O unit comes loose.
- The computer freezes.
- The computer crashes.

7.2 Sine Stop Codes

The following codes indicate the status of the controller, and occur during normal controller operation:

Changing Level
End of Cycle Count Test

End of Sweep Test
End of Timed Test
Holding Frequency
Input channel n is clipping
NI-DAQ error code
Open accel on channel n
Paused
Record Active
Record Exclusive
Remote Start
Remote Stop
Resonance Dwell
Record Active
Record Exclusive
Running Test
Select Resonance Frequencies
Shorted accel on channel n
Starting
Starting Scheduled Test
Step Off
Step On
Stop Button Pressed
Waiting for box n

The following codes indicate that something has occurred to cause the controller to abort the current test.

Channel n Minus Abort
Channel n Plus Abort
Control Minus Abort
Control Plus Abort
Daq reset (Box n)
Didn't reach demand level
Emergency Stop
Invalid Remote Test
Lost contact with I/O box n
Max Run Drive
Max Sine Acceleration
Max Sine Displacement
Max Sine Velocity
Max Start Drive
Resonance Max Drift Limit
Resonance Min Drift Limit
Run System Gain Limit
Startup System Gain Limit
Test Parameters Changed
Watchdog Timeout

7.2.1 Changing Level (Sine Stop Code)

This code indicates that the test has finished one level and is starting another. The test continues to run.

7.2.2 End of Cycle Count Test (Sine Stop Code)

This code indicates that the test was run for the scheduled number of sine wave cycles (as defined in the Test Schedule dialog box). When this code is displayed, the test is complete and the output is turned off.

7.2.3 End of Sweep Test (Sine Stop Code)

This code indicates that the test was run for the scheduled number of frequency sweeps (as defined in the Test Schedule dialog box). When this code is displayed, the test is complete and the output is turned off.

7.2.4 End of Timed Test (Sine Stop Code)

This code indicates that the test was run for the scheduled time duration (as defined in the Test Schedule dialog box). When this code is displayed, the test is complete and the output is turned off.

7.2.5 Holding Frequency (Sine Stop Code)

This code indicates that the output frequency is held at a fixed value. The frequency sweep can be frozen using either mouse control in the Sine Big Display dialog box or by clicking the "Hold Sweep" button in the Sine Control Center.

7.2.6 Input channel n is clipping (Sine Stop Code)

This code indicates that the input signal to channel n is exceeding the allowable input voltage range of +/- 10 volts and is being distorted by clipping. To solve this problem, use an accelerometer with a lower mV/G sensitivity rating.

The maximum allowable acceleration readings for various accelerometer sensitivities are shown in the table below.

Sensitivity	**Maximum Acceleration**
10 mV/G	1000 G peak
20 mV/G	500 G peak
50 mV/G	200 G peak
100 mV/G	100 G peak
1000 mV/G	10 G peak

7.2.7 Max Run Drive (Sine Stop Code)

This code indicates that the maximum allowable voltage was reached while the test was running. This error typically results from having a disconnected accelerometer, not having the amplifier turned on or having the amplifier gain level turned to a low setting.

If the accelerometer and amplifier are all properly connected, it may be that the Running Max Output Voltage limit for this test is too conservative. This parameter is set in the Sine Parameters dialog box. Typically the maximum output voltage allowed during the test startup should be about 125% to 150% of the maximum output voltage level required over the entire frequency range of the test.

The Max Run Drive limit is a safety limit used to prevent the output from reaching excessive levels while the test is running, in such cases as when the accelerometer falls off the test product. You can disable this safety feature (***not recommended!!!***) by selecting a high voltage level (e.g. 10 Volts) for the Max Run Drive parameter.

7.2.8 NI-DAQ error code (Sine Stop Code)

This code indicates that an error occurred while reading input channels from the National Instruments data acquisition card. This error typically happens when another program (e.g. VibrationAnalyzer) is attempting to read data from the data acquisition card at the same time as the *VibrationVIEW* program. Do not run the VibrationAnalyzer program while a sine test is running.

To clear this error condition, stop and resume the test. That is, click the "Stop" button, click the "Run" button and select "Yes" to continue the current test.

7.2.9 Open accel on channel n (Sine Stop Code)

This code indicates that an accelerometer connection is open.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.2.10 Waiting for operator (Sine Stop Code)

This code indicates that the test is paused because it has reached a "Wait for operator" level in the test schedule.

The test will continue if the operator clicks either the "Start" button in the Sine Control Center or the "Continue" button in the "Wait for operator" dialog box.

The test will abort if the operator clicks either the "Stop" button in the Sine Control Center or the "Abort test" button in the "Wait for operator" dialog box.

For more details on the test schedule, refer to the Define Schedule tab of the Sine Define dialog box.

7.2.11 Record Active (Sine Stop Code)

This code indicates that the test was selected using the Remote Start/Stop option, but the recorder is running preventing the test from loading.

7.2.12 Record Exclusive (Sine Stop Code)

This code indicates that the test was selected using the Remote Start/Stop option, but the recorder is running at a sample rate which prevents the test from starting.

7.2.13 Remote Start (Sine Stop Code)

This code indicates that the test was started using the Remote Start/Stop option. While this code is displayed, the system is in the starting mode, as described in the Starting Test stop code. Once the desired level has been reached, the code will change to Running Test.

7.2.14 Remote Stop (Sine Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option. When this code is displayed, the test has been stopped and the output is turned off.

7.2.15 Resonance Dwell (Sine Stop Code)

This code indicates that Sine Resonance Mode is enabled (see the Sine Resonance Control dialog box) and the frequency sweep is frozen during a swept sine test. When this code is displayed, the output frequency is adjusted to keep the phase shift between channels 1 and 2 at +/- PI/2. The frequency sweep can be frozen using either mouse control in the Sine Big Display dialog box or by clicking the "Hold Sweep" button in the Sine Control Center.

7.2.16 Running (Sine Stop Code)

This code indicates that the test has completed the startup phase and is now running. While the test is running, the Level Time and Sweep counters shown in the Sine Control Center will count up/down as appropriate for the current test.

The controller behavior during the running period is defined in the Running Parameters group in the Sine Parameters dialog box.

The output voltage is limited by two settings: "Max System Gain" and "Max Output". The maximum allowed output voltage is defined by the smaller of the following values:

(Max System Gain) * (Desired Acceleration)
Max Output

The "Max System Gain" parameter is also used to detect open-accelerometer conditions. If the measured acceleration level is less than

(Current Output) / (Max System Gain)

the test will automatically be stopped, with a Max Run System Gain stop code.

7.2.17 Select Resonance Frequencies (Sine Stop Code)

This code is displayed when the Select Resonance Frequencies dialog box is displayed. When this code is displayed, the test is paused and is waiting for the operator to select desired resonance frequencies.

7.2.18 Shorted Accel on channel n (Sine Stop Code)

This code indicates that an accelerometer connection is shorted.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.2.19 Starting (Sine Stop Code)

This code indicates that the test is currently in the startup mode, where the output is slowly increased until the desired acceleration level is reached.

The controller behavior during the startup period is defined in the Startup Parameters group in the Sine Parameters dialog box.

The "Time" parameter defines the approximate time it takes to reach the desired running level. The output voltage is limited by two settings: "Max System Gain" and "Max Output". The maximum allowed output voltage is defined by the smaller of the following values:

(Max System Gain) * (Desired Acceleration)
Max Output

The "Max System Gain" parameter is also used to detect open-accelerometer conditions. If the measured acceleration level is less than

(Current Output) / (Max System Gain)

the test will automatically be stopped, with a Max Start System Gain stop code.

7.2.20 Starting Scheduled Test (Sine Stop Code)

This code indicates that the current test is one of several listed in a test schedule and was started using the Test Schedule dialog box.

7.2.21 Step Off (Sine Stop Code)

This code indicates that the test is running in the Stepped-Frequency mode and that the output is currently inactive. When running a Stepped Frequency test, the stop code will alternate between "Step On" and "Step Off".

7.2.22 Step On (Sine Stop Code)

This code indicates that the test is running in the Stepped-Frequency mode and that the output is currently active. When running a Stepped Frequency test, the stop code will alternate between "Step On" and "Step Off".

7.2.23 Stop Button Pressed (Sine Stop Code)

This code indicates that the test was halted through operator control, by selecting the Test..Stop Test menu command, clicking the "Stop" button on the toolbar or clicking the "Stop" button in the Sine Control Center.

7.2.24 Waiting for box n(Sine Stop Code)

This code indicates that more than one control box is defined and the system is waiting for all the boxes to reach the ready state.

If you have a single control box system (4 channels or fewer), select the Configuration Parameters menu command and remove any boxes assigned to higher channels.

7.2.25 Channel n Minus Abort (Sine Stop Code)

This code indicates that the measured acceleration has exceeded the minus abort lines set for channel n in the Sine Limits dialog box.

Typically, this results from an accelerometer or cable being loose or from the controller having difficulty controlling the system as the output sweeps through a resonance frequency.

If the problem is with the controller parameters, see the section "How to tune Sine controller parameters" for information on how to improve the control.

7.2.26 Channel n Plus Abort (Sine Stop Code)

This code indicates that the measured acceleration has exceeded the plus abort lines set for channel n in the Sine Limits dialog box.

Typically, this results from an accelerometer or cable being loose or from the controller having difficulty controlling the system as the output sweeps through a resonance frequency.

If the problem is with the controller parameters, see the section "How to tune Sine controller parameters" for information on how to improve the control.

7.2.27 Control Minus Abort (Sine Stop Code)

This code indicates that the measured acceleration has exceeded the minus abort lines set for the test in the Sine Limits dialog box.

Typically, this results from an accelerometer or cable being loose or from the controller having difficulty controlling the system as the output sweeps through a resonance frequency.

If the problem is with the controller parameters, see the section "How to tune Sine controller parameters" for information on how to improve the control.

7.2.28 Control Plus Abort (Sine Stop Code)

This code indicates that the measured acceleration has exceeded the plus abort lines set for the test in the Sine Limits dialog box.

Typically, this results from an accelerometer or cable being loose or from the controller having difficulty controlling the system as the output sweeps through a resonance frequency.

If the problem is with the controller parameters, see the section "How to tune Sine controller parameters" for information on how to improve the control.

7.2.29 Daq Reset (Sine Stop Code)

This code indicates a failure initializing the D/A and A/D converters in the 8500 I/O box. Under normal conditions this message should never appear. If you receive this error, please contact Vibration Research Corp. at (616) 669-3028 or support@vibrationresearch.com for assistance.

7.2.30 Didn't Reach Demand Level (Sine Stop Code)

This code indicates that the controller was either starting a test or switching test levels, and failed to reach the demand level within 30 minutes. The controller should, even in the worst case, be able to reach the demand level within one minute. If you see this error, it is likely that the controller parameters are set incorrectly. See the section "How to tune Sine controller parameters" for information on how to improve the control.

7.2.31 Emergency Stop (Sine Stop Code)

This code indicates that the emergency stop button was pressed, causing the controller to shut down. The emergency stop button is an optional accessory that connects to a special connector on the front of the *VibrationVIEW* I/O unit.

7.2.32 Invalid Remote Test (Sine Stop Code)

This code indicates that the "Remote Start/Stop" and "Remote Selection" options are both enabled but the test number selected using the remote test selection input (Discrete Inputs 4,5,6,7) does not have a valid test name configured for the current input settings. Select the Configuration..Remote Inputs menu command to verify that the test names configured there are valid.

If you do not want the program to automatically load a test when the remote input signal goes high, set Remote Selection to "Disabled" in the Remote Inputs configuration dialog box. That is, if you want the remote input signal to simply start/stop the current test, enable "Remote Start/Stop" and disable "Remote Selection".

7.2.33 Lost contact with I/O box n (Sine Stop Code)

This code indicates that the *VibrationVIEW* I/O box is no longer communicating with the host computer.

Verify that:

- The Ethernet cables are connected properly.
- The box is powered.

7.2.34 Max Sine Acceleration (Sine Stop Code)

This code indicates that the maximum acceleration allowed for the system has been exceeded. This limit is defined in the System Limits dialog box. It is a combination of the Force limits of the shaker system and the mass of the shaker and fixture (defined in the System Mass dialog box) using Newton's first law:

$$a = F/m$$

where:

a = acceleration limit of the shaker
F = Force limit of the shaker
m = mass that the shaker must move (sum of the system masses)

If you receive this error message, either the parameters in the System Limits and/or System Mass dialog boxes need to be adjusted to reflect the capabilities of your shaker or the current test requires a larger shaker than you are using.

7.2.35 Max Sine Displacement (Sine Stop Code)

This code indicates that the maximum displacement allowed for the system has been exceeded. This limit is defined in the System Limits dialog box.

If you receive this error message, either the displacement limit parameter in the System Limits dialog box needs to be adjusted to reflect the capabilities of your shaker or the current test requires a shaker with a larger displacement range than you are using.

7.2.36 Max Sine Velocity (Sine Stop Code)

This code indicates that the maximum velocity allowed for the system has been exceeded. This limit is defined in the System Limits dialog box.

If you receive this error message, either the velocity limit parameter in the System Limits dialog box needs to be adjusted to reflect the capabilities of your shaker or the current test requires a larger shaker than you are using.

7.2.37 Max Start Drive (Sine Stop Code)

This code indicates that the maximum allowable voltage was reached when starting the test. This error typically results from having a disconnected accelerometer, not having the amplifier turned on or having the amplifier gain level turned to a low setting.

If the accelerometer and amplifier are all properly connected, it may be that the Startup Max Output Voltage limit for this test is too conservative. This parameter is set in the Sine Parameters dialog box. Typically the maximum output voltage allowed during the test startup should be about 125% to 150% of the output voltage level required at the starting frequency.

The Max Start Drive limit is a safety limit used to prevent the output from reaching excessive levels during system startup, in such cases as when the accelerometer is not connected or has fallen off the test product. You can disable this safety feature (***not recommended!!!***) by selecting a high voltage level (e.g. 10 Volts) for the Max Start Drive parameter.

7.2.38 Resonance Min Drift Limit (Sine Stop Code)

This code indicates that the resonance frequency has drifted below the lower limit specified in the Sine Resonance Control dialog box. Typically this is a result of failure (e.g. breakage) of the product being tested.

7.2.39 Resonance Max Drift Limit (Sine Stop Code)

This code indicates that the resonance frequency has drifted above the upper limit specified in the Sine Resonance Control dialog box. Typically this is a result of failure (e.g. breakage) of the product being tested.

7.2.40 Max Run System Gain (Sine Stop Code)

This code indicates that the maximum allowable voltage/acceleration ratio was reached while the test was running. This error typically results from having a disconnected accelerometer, not having the amplifier turned on or having the amplifier gain level turned to a low setting.

If the accelerometer and amplifier are all properly connected, it may be that the running Max System Gain limit for this test is too conservative. This parameter is set in the Sine Parameters dialog box. Typically the maximum system gain allowed while the test is running should be about 125% to 150% of the maximum voltage/acceleration ratio required over the entire frequency range of the test.

The Max System Gain limit is a safety limit used to prevent the output from reaching excessive levels while the test is running, in such cases as when the accelerometer is not connected or has fallen off the test product. You can disable this safety feature (***not recommended!!!***) by selecting a high gain level (e.g. 10 Volts/G) for the Max System Gain parameter.

7.2.41 Max Start System Gain (Sine Stop Code)

This code indicates that the maximum allowable voltage/acceleration ratio was reached when starting the test. This error typically results from having a disconnected accelerometer, not having the amplifier turned on or having the amplifier gain level turned to a low setting.

If the accelerometer and amplifier are all properly connected, it may be that the starting Max System Gain limit for this test is too conservative. This parameter is set in the Sine Parameters dialog box. Typically the maximum system gain allowed during the test startup should be about 125% to 150% of the expected voltage/acceleration ratio required at the starting frequency.

The Max System Gain limit is a safety limit used to prevent the output from reaching excessive levels during system startup, in such cases as when the accelerometer is not connected or has fallen off the test product. You can disable this safety feature (***not recommended!!!***) by selecting a high gain level (e.g. 10 Volts/G) for the Max System Gain parameter.

7.2.42 Test Parameters Changed (Sine Stop Code)

This code indicates that the test parameters were changed, by selecting either the Test..Edit Test or the File..Define New Test menu commands, or their respective toolbar buttons. As a result of the test parameter changes, the test was halted.

7.2.43 Watchdog Timeout (Sine Stop Code)

This code indicates that the communications link between *VibrationVIEW* and the Signal Processing hardware was lost. Under normal operations this should never happen. Click the "Run" button to reset the timeout.

The most likely cause of this error is a loss of power to the *VibrationVIEW* I/O unit.

Other events that could cause this to happen:

- The VibrationVIEW I/O unit is turned on after the program is started.
- The Ethernet cable connecting the computer with the VibrationVIEW I/O unit comes loose.
- The computer freezes.
- The computer crashes.

7.3 Random Stop Codes

These are status messages that tell you what the controller is doing.

Changing Level
End of Test
Input channel n is clipping
Open accel on channel n
Paused
Record Active
Record Exclusive
Remote Start
Remote Stop
Running
Shorted accel on channel n
Starting
Starting Scheduled Test
Starting with Memorized Drive
Stop Button Pressed
Waiting for box n

These are stop codes that tell you why the controller stopped the test. A brief description follows explaining what to do about the problem.

Analyzer Link Failed (Err=n)
Analyzer Trace Outside Aborts
Ch n Accel Reading Too High
Ch n Accel Reading Too Low
Control Lines Exceeded (Loop n)
Control Minus RMS (Loop n)
Control Plus RMS (Loop n)
Daq reset (box n)
Didn't reach demand level
Emergency Stop
Invalid Analyzer configuration file
Invalid Remote Test
Lost contact with I/O box n
Max Random Acceleration (Loop n)
Max Random Displacement (Loop n)
Max Random Velocity (Loop n)
Max Run Drive (Loop n)
Max Run System Gain (Loop n)
Max Start Drive (Loop n)
Max Run System Gain
Sine Plus abort limit (Loop n)
Test Paramaters Changed
Watchdog Timeout

7.3.1 Changing Level (Random Stop Code)

This code indicates that the test has finished one level and starting another. The test continues to run.

7.3.2 End of Test (Random Stop Code)

This code indicates the normal shut down condition. Use the Random Schedule dialog box to change how long a test runs.

7.3.3 Input channel N is clipping (Random Stop Code)

This code indicates that the input signal to channel N is exceeding the allowable input voltage range of +/- 10 volts, and is being distorted by clipping. To solve this problem use an accelerometer with a lower mV/G sensitivity rating.

The maximum allowable acceleration readings for various accelerometer sensitivities are shown in the table below.

Sensitivity	**Maximum Acceleration**
10 mV/G	1000 G peak
20 mV/G	500 G peak
50 mV/G	200 G peak
100 mV/G	100 G peak
1000 mV/G	10 G peak

7.3.4 Waiting for operator (Random Stop Code)

This code indicates that the test is paused because it has reached a "Wait for operator" level in the test schedule.

The test will continue if the operator clicks either the "Start" button in the Random Control Center or the "Continue" button in the "Wait for operator" dialog box.

The test will abort if the operator clicks either the "Stop" button in the Random Control Center or the "Abort test" button in the "Wait for operator" dialog box.

For more details on the test schedule, refer to the Define Schedule tab of the Random Define dialog box.

7.3.5 Record Active (Random Stop Code)

This code indicates that the test was selected using the Remote Start/Stop option, but the recorder is running preventing the test from loading.

7.3.6 Record Exclusive (Random Stop Code)

This code indicates that the test was selected using the Remote Start/Stop option, but the recorder is running at a sample rate which prevents the test from starting.

7.3.7 Remote Start (Random Stop Code)

This code indicates that the test was started using the Remote Start/Stop option. When this code is displayed, the system is in the starting mode, as described in the Starting Test stop code. Once the desired level has been reached, the test mode will change to Running Test.

7.3.8 Remote Stop (Random Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option. When this code is displayed, the test has been stopped and the output is turned off.

7.3.9 Running (Random Stop Code)

This code indicates that the test has completed the startup phase and is now running. While the test is running, the Level Time will count.

The controller behavior during the running period is defined in the "Running" group in the Random Parameters dialog box.

7.3.10 Shorted Accel on channel n (Random Stop Code)

This code indicates that an accelerometer connection is shorted.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.3.11 Starting Test (Random Stop Code)

This code indicates that the test is currently in the startup mode, where the output is slowly increased until the desired acceleration level is reached.

The controller behavior during the startup period is defined in the "Starting" group in the Random

Parameters dialog box.

7.3.12 Starting Scheduled Test (Random Stop Code)

This code indicates that the current test is one of several listed in a test schedule and was started using the Test Schedule dialog box.

7.3.13 Starting with Memorized Drive (Random Stop Code)

This code indicates that the test is starting with a memorized drive.

Checking the box next to "Enable memorized drive" in the Random Level Schedule dialog box will turn on this option. You can start with either the current drive (the one you most recently saved) or the standard drive (the default drive).

In order to set the current drive you must have a test started. Then select the Test..Save Current Drive menu command.

7.3.14 Stop Button Pressed (Random Stop Code)

This code indicates that the test was halted through operator control, by selecting the Test..Stop Test menu command, clicking the "Stop" button on the toolbar or clicking the "Stop" button in the Sine Control Center.

7.3.15 Waiting for box n (Random Stop Code)

This code indicates that more than one control box is defined and the system is waiting for all the boxes to reach the ready state.

If you have a single control box system (4 channels or fewer), select the Configuration Parameters menu command and remove any boxes assigned to higher channels.

7.3.16 Analyzer Link Failed (Random Stop Code)

This code indicates that there was a problem starting the link between *VibrationVIEW* and VibrationAnalyzer. This might occur if you do not have VibrationAnalyzer installed or you do not have VibrationAnalyzer 1.0.6 or higher installed or if you exit VibrationAnalyzer before a test is complete.

7.3.17 Analyzer Trace Outside Aborts (Random Stop Code)

This code indicates that Vibration Analyzer detected that the test was no longer running within the specified abort limits. When the Vibration Analyzer status indicator light shows red while the test is running then this stop code will appear.

7.3.18 Ch. n Accel Reading Too High (Random Stop Code)

This code indicates that the RMS acceleration level on channel n is above the maximum prescribed. The limit is set as a dB level above the expected RMS acceleration level. The value for this limit is the "+ RMS dB" limit, configured in the Random Limits dialog box.

7.3.19 Ch. n Accel Reading Too Low (Random Stop Code)

This code indicates that the RMS acceleration level on Channel N is below the minimum prescribed. The limit is set as a dB level below the expected RMS acceleration level. The value for this limit is the "- RMS dB" limit, configured in the Random Limits dialog box.

In most cases, this error indicates that one of the accelerometers is not working or has fallen off the test fixture.

7.3.20 Control Lines Exceeded (Random Stop Code)

A random test is comprised of narrow filters called "lines". This code indicates that more lines than the allowed amount have exceeded the plus and minus abort limits. This abort usually occurs if the shaker is shut down without the controller knowing or if the accelerometer falls off or is disconnected. The number of lines and abort limits are adjusted through the Random Parameters dialog box.

7.3.21 Control Minus RMS (Random Stop Code)

This code indicates that the RMS acceleration level of the Control signal is below the minimum prescribed. The limit is set as a dB level below the expected RMS acceleration level. The value for this limit is the "- RMS dB" limit, configured in the Random Limits dialog box.

7.3.22 Control Plus RMS (Random Stop Code)

This code indicates that the RMS acceleration level of the Control signal is above the maximum prescribed. The limit is set as a dB level above the expected RMS acceleration level. The value for this limit is the "+ RMS dB" limit, configured in the Random Limits dialog box.

7.3.23 Daq Reset (Random Stop Code)

This code indicates a failure initializing the D/A and A/D converters in the 8500 I/O box. Under normal conditions this message should never appear. If you receive this error, please contact Vibration Research Corp. at (616) 669-3028 or support@vibrationresearch.com for assistance.

7.3.24 Didn't Reach Demand Level (Random Stop Code)

This code indicates that the controller was either starting a test or switching test levels, and failed to reach the demand level within 30 minutes. The controller should, even in the worst case, be able to reach the demand level within 30 minutes. If you see this error, it is likely that the controller parameters are set incorrectly. See the section "How to tune Random controller parameters" for information on how to improve the control.

7.3.25 Emergency Stop (Random Stop Code)

This code indicates that the emergency stop button was pressed, causing the controller to shut down. The emergency stop button is an optional accessory that connects to a special connector on the front of the *VibrationVIEW* I/O unit.

7.3.26 Invalid Analyzer configuration file (Random Stop Code)

This code indicates that the configuration file you selected in the Analyzer Link dialog box does not exist. Check the filename to make sure it exists.

7.3.27 Invalid Remote Test (Random Stop Code)

This code indicates that the "Remote Start/Stop" and "Remote Selection" options are both enabled but the test number selected using the remote test selection input (Discrete Inputs 4,5,6,7) does not have a valid test name configured for the current input settings. Select the Configuration..Remote Inputs menu command to verify that the test names configured there are valid.

If you do not want the program to automatically load a test when the remote input signal goes high, set

Remote Selection to "Disabled" in the Remote Inputs configuration dialog box. That is, if you want the remote input signal to simply start/stop the current test, enable "Remote Start/Stop" and disable "Remote Selection".

7.3.28 Lost contact with I/O box n (Random Stop Code)

This code indicates that the *VibrationVIEW* I/O box is no longer communicating with the host computer.

Verify that:

- The Ethernet cables are connected properly.
- The box is powered.

7.3.29 Max Random Acceleration (Random Stop Code)

This code indicates that you have exceeded the maximum acceleration allowed for the system. See the section "How to set the Shaker System Limits". You may need to modify the parameters or get a bigger shaker. Note that maximum acceleration is calculated from the mass installed on the shaker.

7.3.30 Max Random Displacement (Random Stop Code)

This code indicates that you have exceeded the maximum displacement allowed for the system. See the section "How to set the Shaker System Limits". You may need to modify the parameters or get a shaker with a larger displacement range in order to run this test.

7.3.31 Max Random Velocity (Random Stop Code)

This code indicates that you have exceeded the maximum velocity allowed for the system. See the section "How to set the Shaker System Limits". You may need to modify the parameters or get a bigger shaker.

7.3.32 Max Run Drive (Random Stop Code)

This code indicates that the maximum allowable voltage was reached while the test was running. Perhaps the accelerometer is disconnected or your amplifier is not turned on. You may need more voltage. Modify the appropriate parameter found in the Random Define dialog box.

7.3.33 Max Run System Gain (Random Stop Code)

This code indicates that the ratio between the output voltage and the measured acceleration exceeds the maximum value allowed while the test is running. Perhaps the accelerometer is disconnected or your amplifier is not turned on. You may need to adjust the running Max System Gain parameter, found in the Random Parameters dialog box.

7.3.34 Max Start Drive (Random Stop Code)

This code indicates that the maximum allowable voltage was reached when starting your test. Perhaps the accelerometer is disconnected or your amplifier is not turned on. You may need more voltage. Modify the appropriate parameter found in the Random Define dialog box.

7.3.35 Max Start System Gain (Random Stop Code)

This code indicates that the ratio between the output voltage and the measured acceleration exceeds the maximum value allowed when starting the test. Perhaps the accelerometer is disconnected or your amplifier is not turned on. You may need to adjust the starting Max System Gain parameter, found in the

Random Parameters dialog box.

Note that it is typical for this message to be displayed when a test is started and then disappear after a few seconds. The starting Max System Gain parameter will limit the rate at which the output increases. The stop code box will be yellow while this limiting is active.

7.3.36 Sine Plus abort limit (Loop n) (Random Stop Code)

This code indicates that one of the Sine tones exceeded the Sine Abort Limits as defined on the Random Define Limits dialog box. If you get this stop code increase the sine abort limits, use a narrower tracking filter settings to better filter out the random background or decrease the sweep rate if you are sweeping rapidly through a resonance.

7.3.37 Test Parameters Changed (Random Stop Code)

This code indicates that the test parameters were changed, by selecting either the Test..Edit Test or the File..Define New Test menu commands or their respective toolbar buttons. As a result of the test parameter changes, the test was halted.

7.3.38 Watchdog Timeout (Random Stop Code)

This code indicates that the communications link between *VibrationVIEW* and the Signal Processing hardware was lost. Under normal operations this should never happen. Click the "Run" button to reset the timeout.

The most likely cause of this error is a loss of power to the *VibrationVIEW* I/O unit.

Other events that could cause this to happen:

- The VibrationVIEW I/O box is turned on after the program is started.
- The Ethernet cable connecting the computer with the VibrationVIEW I/O unit comes loose.
- The computer freezes.
- The computer crashes.

7.4 Shock Stop Codes

The following codes indicate the status of the controller, and occur during normal controller operation:

Changing Level
End of Test
Open accel on channel n
Paused
Record Active
Record Exclusive
Remote Start
Remote Stop
Running
Shorted accel on channel n
Start System Gain Limit
Starting
Starting Scheduled Test
Starting with Memorized Drive
Stop Button Pressed
Waiting for box n

The following codes indicate that something has occurred to cause the controller to abort the current test.

Abort Limit
Daq reset (box n)
Didn't reach demand level
Emergency Stop
Input Signal is Clipped
Invalid Remote Test
Lost contact with I/O box n
Max Run Voltage
Max Shock Acceleration
Max Shock Displacement
Max Shock Velocity
Max Starting Voltage
Run System Gain Limit
Test Parameters Changed
Watchdog Timeout

7.4.1 Changing Level (Shock Stop Code)

This code indicates that the test has finished one level and starting another. The test continues to run.

7.4.2 End of Test (Shock Stop Code)

This code indicates the normal shut down condition. Use the Shock Schedule dialog box to change how long a test runs.

7.4.3 Open Accel on channel n (Shock Stop Code)

This code indicates that an accelerometer connection is open.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.4.4 Waiting for operator (Shock Stop Code)

This stop code indicates that the test is paused because it has reached a "Wait for operator" level in the test schedule.

The test will continue if the operator clicks either the "Start" or "Hold Pulse" buttons in the Shock Control Center or the "Continue" button in the "Wait for operator" dialog box.

The test will abort if the operator clicks either the "Stop" button in the Shock Control Center or the "Abort test" button in the "Wait for operator" dialog box.

For more details on the test schedule, refer to the Define Schedule tab of the Shock Define dialog box.

7.4.5 Record Active (Shock Stop Code)

This code indicates that the test was selected using the Remote Start/Stop option, but the recorder is running preventing the test from loading.

7.4.6 Record Exclusive (Shock Stop Code)

This code indicates that the test was selected using the Remote Start/Stop option, but the recorder is running at a sample rate which prevents the test from starting.

7.4.7 Remote Start (Shock Stop Code)

This code indicates that the test was started using the Remote Start/Stop option. While this code is displayed, the system is in the starting mode, as described in the Starting Test stop code. Once the desired level has been reached, the test mode will change to Running Test.

7.4.8 Remote Stop (Shock Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option. When this code is displayed, the test has been stopped and the output is turned off.

7.4.9 Running (Shock Stop Code)

This code indicates that the test has completed the startup phase and is now running. While the test is running, the Level Time and the Pulses will count.

The controller behavior during the running period is defined in the Shock Parameters dialog box and "Run Pk Volts" in the Shock Limits dialog box.

7.4.10 Shorted Accelerometer (Shock Stop Code)

This code indicates that an accelerometer connection is shorted.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.4.11 Max Start System Gain (Shock Stop Code)

This code indicates that the ratio between the output voltage and the measured acceleration exceeds the maximum value allowed while starting the test. Perhaps the accelerometer is disconnected or your amplifier is not turned on. You may need to adjust the starting Max System Gain parameter, found in the Shock Parameters dialog box.

Note that it is typical for this message to be displayed when a test is started and then disappear after a few pulses. The starting Max System Gain parameter will limit the rate at which the output increases. Thestop code box will be yellow while this limiting is active.

7.4.12 Starting Test (Shock Stop Code)

This code indicates that the test is currently in the startup mode, where the output is slowly increased until the desired acceleration level is reached.

The controller behavior during the startup period is defined in the "Start Pk Volts" in the Shock Limits dialog box.

7.4.13 Starting Scheduled Test (Shock Stop Code)

This code indicates that the current test is one of several listed in a test schedule and was started using the Test Schedule dialog box.

7.4.14 Starting with Memorized Drive (Shock Stop Code)

This stop code indicates that the test is starting with a memorized drive.

Checking the box next to "Enable memorized drive" in the Shock Level Schedule dialog box will turn on this option.

In order to set the current drive you must have a test started. Then select the Test Save Current Drive menu command.

7.4.15 Stop Button Pressed (Shock Stop Code)

This code indicates that the test was halted through operator control by selecting the Test..Stop Test menu command, clicking the "Stop" button on the toolbar or clicking the "Stop" button in the Classical Shock Control Center.

7.4.16 Waiting for box n (Shock Stop Code)

This code indicates that more than one control box is defined and the system is waiting for all the boxes to reach the ready state.

If you have a single control box system (4 channels or fewer), select the Configuration Parameters menu command and remove any boxes assigned to higher channels.

7.4.17 Abort Limit (Shock Stop Code)

This code indicates that the maximum or minimum acceleration was exceeded by the test. These abort limits are the red lines (as originally colored) on the graph. The abort limits can be set in the Shock Limits dialog box.

7.4.18 Daq Reset (Shock Stop Code)

This code indicates a failure initializing the D/A and A/D converters in the 8500 I/O box. Under normal conditions this message should never appear. If you receive this error, please contact Vibration Research Corp. at (616) 669-3028 or support@vibrationresearch.com for assistance.

7.4.19 Didn't Reach Demand Level (Shock Stop Code)

This code indicates that the controller was either starting a test or switching test levels and failed to reach the demand level within 100 pulses. The controller should, even in the worst case, be able to reach the demand level within a few pulses. If you see this error, it is likely that the controller parameters are set incorrectly. See the section "How to tune Shock controller parameters" for information on how to improve the control.

7.4.20 Emergency Stop (Shock Stop Code)

This code indicates that the emergency stop button was pressed, causing the controller to shut down. The emergency stop button is an optional accessory that connects to a special connector on the front of the *VibrationVIEW* I/O unit.

7.4.21 Input Signal is Clipped (Shock Stop Code)

This code indicates that the input signal to channel N is exceeding the allowable input voltage range of +/- 10 volts, and is being distorted by clipping. To solve this problem, use an accelerometer with a lower mV/G sensitivity rating.

The maximum allowable acceleration readings for various accelerometer sensitivities are shown in the table below.

Sensitivity	Maximum Acceleration
10 mV/G	1000 G peak
20 mV/G	500 G peak
50 mV/G	200 G peak
100 mV/G	100 G peak
1000 mV/G	10 G peak

7.4.22 Invalid Remote Test (Shock Stop Code)

This code indicates that the "Remote Start/Stop" and "Remote Selection" options are both enabled but the test number selected using the remote test selection input (Discrete Inputs 4,5,6,7) does not have a valid test name configured for the current input settings. Select the Configuration..Remote Inputs menu command to verify that the test names configured there are valid.

If you do not want the program to automatically load a test when the remote input signal goes high, set Remote Selection to "Disabled" in the Remote Inputs configuration dialog box. That is, if you want the remote input signal to simply start/stop the current test, enable "Remote Start/Stop" and disable "Remote Selection".

7.4.23 Lost ontact with I/O box n (Shock Stop Code)

This code indicates that the *VibrationVIEW* I/O box is no longer communicating with the host computer.

Verify that:

- The Ethernet cables are connected properly.
- The box is powered.

7.4.24 Max Starting Voltage (Shock Stop Code)

This code indicates that the maximum allowable voltage was reached when starting the test. This error typically results from having a disconnected accelerometer, not having the amplifier turned on or having the amplifier gain level turned to a low setting.

The Max Start Drive limit is a safety limit used to prevent the output from reaching excessive levels during system startup, in such cases as when the accelerometer is not connected or has fallen off the test product. You can disable this safety feature (***not recommended!!!***) by selecting a high voltage level (e.g. 10 Volts) for the Start Pk Volts parameter in the Shock Limits dialog box.

7.4.25 Max Shock Acceleration (Shock Stop Code)

This code indicates that the Maximum Shock Acceleration (G peak) for the test was reached.

This means that your test sent a shock acceleration to your shaker that it could not perform. If you need to continue with this acceleration you need a bigger shaker. You can also tone down your test by setting the Modifier to a lower value in the Shock Schedule dialog box.

7.4.26 Max Shock Displacement (Shock Stop Code)

This code indicates that the Maximum Shock Displacement for the test was reached.

This means that your test sent a shock displacement to your shaker that it could not perform. If you need

to continue with this displacement you need a bigger shaker. You can also tone down your test by setting the Modifier to a lower value in the Shock Schedule dialog box.

7.4.27 Max Shock Velocity (Shock Stop Code)

This code indicates that the Maximum Shock Velocity for the test was reached.

This means that your test sent a shock velocity to your shaker that it could not perform. If you need to continue with this velocity you need a bigger shaker. You can also tone down your test by setting the Modifier to a lower value in the Shock Schedule dialog box.

7.4.28 Max Run Voltage (Shock Stop Code)

This code indicates that the maximum allowable voltage was reached while the test was running. This error typically results from having a disconnected accelerometer, not having the amplifier turned on or having the amplifier gain level turned to a low setting.

The Max Run Drive limit is a safety limit used to prevent the output from reaching excessive levels while the test is running, in such cases as when the accelerometer falls off the test product. You can disable this safety feature (***not recommended!!!***) by selecting a high voltage level (e.g. 10 Volts) for the Run Pk Volts parameter in the Shock Limits dialog box.

7.4.29 Max Run System Gain (Shock Stop Code)

This code indicates that the ratio between the output voltage and the measured acceleration exceeds the maximum value allowed while the test is running. Perhaps the accelerometer is disconnected or your amplifier gain is turned down. You may need to adjust the running Max System Gain parameter, found in the Shock Parameters dialog box.

7.4.30 Test Parameters Changed (Shock Stop Code)

This code indicates that the test parameters were changed, by selecting either the Test..Edit Test or the File..Define New Test menu commands or their respective toolbar buttons. As a result of the test parameter changes, the test was halted.

7.4.31 Watchdog Timeout (Shock Stop Code)

This code indicates that the communications link between *VibrationVIEW* and the Signal Processing hardware was lost. Under normal operations this should never happen. Click the "Run" button to reset the timeout.

The most likely cause of this error is a loss of power to the *VibrationVIEW* I/O unit.

Other events that could cause this to happen:

- The VibrationVIEW I/O unit is turned on after the program is started.
- The Ethernet cable connecting the computer with the VibrationVIEW I/O unit comes loose.
- The computer freezes.
- The computer crashes.

7.5 Field Data Replicator Stop Codes

These are status messages, to let you know what the controller is doing.

Changing Level
Doing Digital Record
End of Test
Input channel n is clipping
Open accel on channel n
Paused
Record Active
Record Exclusive
Remote Start
Remote Stop
Running
Shorted accel on channel n
Starting
Starting Scheduled Test
Starting with Memorized Drive
Stop Button Pressed
Waiting for box n

These are stop codes, telling you why the controller stopped the test. A brief description follows explaining what to do about the problem.

Control Minus RMS
Control Plus RMS
Daq reset (box n)
Didn't reach demand level
Emergency Stop
Exceeded Displacement Limit
Exceeded Peak Acceleration Limit
Exceeded RMS Acceleration Limit
Exceeded Velocity Limit
Invalid Remote Test
Lost contact with I/O box nn
Max Run System Gain
Max Run Voltage
Max Start System Gain
Max Starting Voltage
Network congestion (loop n)
Test Parameters Changed
Watchdog Timeout

7.5.1 Changing Level (Field Data Replicator Stop Code)

This code indicates that the test has finished one level and starting another. The test continues to run.

7.5.2 Doing Digital Record (Field Data Replicator Stop Code)

This code indicates that the program is recording the data from the input channel.

You can access this in the Field Data Replicator Define dialog box by clicking the "Record..." button, selecting a file to record to and clicking the "Start" button in the Digital Record dialog box.

7.5.3 End of Test (Field Data Replicator Stop Code)

This code indicates the normal shut down condition. Use the Field Data Replicator Schedule dialog box to change how long a test runs.

7.5.4 Input channel n is clipping (Field Data Replicator Stop Code)

This code indicates that the input signal to channel nN is exceeding the allowable input voltage range of +/- 10 volts and is being distorted by clipping. To solve this problem, use an accelerometer with a lower mV/G sensitivity rating.

The maximum allowable acceleration readings for various accelerometer sensitivities are shown in the table below.

Sensitivity	Maximum Acceleration
10 mV/G	1000 G peak
20 mV/G	500 G peak
50 mV/G	200 G peak
100 mV/G	100 G peak
1000 mV/G	10 G peak

7.5.5 Open Accel on channel n (Field Data Replicator Stop Code)

This code indicates that an accelerometer connection is open.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.5.6 Waiting for operator (Field Data Replicator Stop Code)

This code indicates that the test is paused because it has reached a "Wait for operator" level in the test schedule.

The test will continue if the operator clicks either the "Start" button in the Field Data Replicator Control Center or the "Continue" button in the "Wait for operator" dialog box.

The test will abort if the operator clicks either the "Stop" button in the Field Data Replicator Control Center or the "Abort test" button in the "Wait for operator" dialog box.

For more details on the test schedule, refer to the Define Schedule tab of the Field Data Replicator Define dialog box.

7.5.7 Remote Start (Field Data Replicator Stop Code)

This code indicates that the test was started using the Remote Start/Stop option. While this code is displayed, the system is in the starting mode, as described in the Starting Test stop code. Once the desired level has been reached, the test mode will change to Running Test.

7.5.8 Record Active (Field Data Replicator Stop Code)

This code indicates that the test was selected using the Remote Start/Stop option, but the recorder is running preventing the test from loading.

7.5.9 Record Exclusive (Field Data Replicator Stop Code)

This code indicates that the test was selected using the Remote Start/Stop option, but the recorder is running at a sample rate which prevents the test from starting.

7.5.10 Remote Stop (Field Data Replicator Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option.

7.5.11 Running Test (Field Data Replicator Stop Code)

This code indicates that the test has completed the startup phase, and is now running. While the test is running, the Level Time will count during the test.

The controller behavior during the running period is defined in "Run" in the Field Data Replicator Parameters dialog box.

7.5.12 Shorted Accel on channel n (Field Data Replicator Stop Code)

This code indicates that an accelerometer connection is shorted.

Verify that:

- The cable connections are all secure.
- The input configuration is set up correctly. If you are using external conditioning turn off "Accel Power Source" in the Configuration Inputs dialog box.

7.5.13 Starting Test (Field Data Replicator Stop Code)

This code indicates that the test is currently in the startup mode, where the output is slowly increased until the desired acceleration level is reached.

The controller behavior during the startup period is defined in the "Start" group in the Field Data Replication Parameters dialog box.

7.5.14 Starting Scheduled Test (Field Data Replicator Stop Code)

This code indicates that the current test is one of several listed in a test schedule and was started using the Test Schedule dialog box.

7.5.15 Starting With Memorized Drive (Field Data Replicator Stop Code)

This code indicates that the test is starting with a memorized drive.

Checking the box next to "Enable memorized drive" in the Field Data Replicator Level Schedule dialog box will turn on this option. You can start with either the current drive (the one you most recently saved) or the standard drive (the default drive).

In order to set the current drive you must have a test started. Then select the Test Save Current Drive menu command.

7.5.16 Stop Button Pressed (Field Data Replicator Stop Code)

This code indicates that you clicked the "Stop" button, telling the test to shut down.

7.5.17 Waiting for box n (Field Data Replicator Stop Code)

This code indicates that more than one control box is defined and the system is waiting for all the boxes to reach the ready state.

If you have a single control box system (4 channels or fewer), select the Configuration Parameters menu command and remove any boxes assigned to higher channels.

7.5.18 Control Minus RMS (Field Data Replicator Stop Code)

This code indicates that the RMS level of the control accelerometer was too low relative to the demand signal, causing the test to abort. This indicates a loss of closed loop control. Check your accelerometer and drive cables to make sure all of the connections are secure.

7.5.19 Control Plus RMS (Field Data Replicator Stop Code)

This code indicates that the RMS level of the control accelerometer was too high relative to the demand signal, causing the test to abort. This indicates a loss of closed loop control. Check your accelerometer and drive cables to make sure all of the connections are secure.

7.5.20 Daq Reset (box n) (Field Data Replicator Stop Code)

This code indicates a failure initializing the D/A and A/D converters in the *VibrationVIEW* I/O box. Under normal conditions this message should never appear. If you receive this error, please contact Vibration Research Corp. at (616) 669-3028 or support@vibrationresearch.com for assistance.

7.5.21 Didn't Reach Demand Level (Field Data Replicator Stop Code)

This code indicates that the controller was either starting a test or switching test levels and failed to reach the demand level within 30 minutes. The controller should, even in the worst case, be able to reach the demand level within a few minutes. If you see this error, it is likely that the controller parameters are set incorrectly. See the section "How to tune Field Data Replicator controller parameters" for information on how to improve the control.

7.5.22 Emergency Stop (Field Data Replicator Stop Code)

This code indicates that the emergency stop button was pressed, causing the controller to shut down. The emergency stop button is an optional accessory that connects to a special connector on the front of the *VibrationVIEW* I/O unit.

7.5.23 Ch n Exceeded Displacement Limit (Field Data Replicator Stop Code)

This code indicates that you have exceeded the maximum displacement allowed for the system. See the section "How to set the Shaker System Limits". You may need to modify the parameters, or get a bigger shaker.

7.5.24 Ch n Exceeded Velocity Limit (Field Data Replicator Stop Code)

This code indicates that you have exceeded the maximum velocity allowed for the system. See the section "How to set the Shaker System Limits". You may need to modify the parameters, or get a bigger shaker.

7.5.25 Invalid Remote Test (Field Data Replicator Stop Code)

This code indicates that the "Remote Start/Stop" and "Remote Selection" options are both enabled but the test number selected using the remote test selection input (Discrete Inputs 4,5,6,7) does not have a

valid test name configured for the current input settings. Select the Configuration..Remote Inputs menu command to verify that the test names configured there are valid.

If you do not want the program to automatically load a test when the remote input signal goes high, set Remote Selection to "Disabled" in the Remote Inputs configuration dialog box. That is, if you want the remote input signal to simply start/stop the current test, enable "Remote Start/Stop" and disable "Remote Selection".

7.5.26 Exceeded Peak Acceleration Limit (Field Data Replicator Stop Code)

This code indicates that the peak acceleration level of the demand or control signal exceeded the Peak Acceleration Limit defined on the Limits tab of the Field Data Replicator test settings dialog box.

7.5.27 Exceeded RMS Acceleration Limit (Field Data Replicator Stop Code)

Maximum Control (Ch1) G rms. This code indicates that the maximum allowable control Grms was reached while running your test. Perhaps the accelerometer current source is off or your amplifier is not turned on. You may need more Gs. Modify the appropriate parameter found in the Field Data Replicator Limits dialog box.
Maximum Reference (Ch2) G rms. This code indicates that the maximum allowable control Grms was reached while running your test. You may have a large spike in your data or the channel two sensitivity is not set correctly. You may need more Gs. Modify the appropriate parameter found in the Field Data Replicator Limits dialog box.

7.5.28 Lost contact with I/O box n (Field Data Replicator Stop Code)

This code indicates that the *VibrationVIEW* I/O box is no longer communicating with the host computer.

Verify that:

- The Ethernet cables are connected properly.
- The box is powered.

7.5.29 Maximum Run System Gain (Field Data Replicator Stop Code)

This code indicates that the ratio between the output voltage and the measured acceleration exceeds the maximum value allowed while starting the test. Perhaps the accelerometer is disconnected or your amplifier is not turned on. You may need to adjust the running Max System Gain parameter, found in the Field Data Replicator Limits dialog box.

7.5.30 Max Run Voltage (Field Data Replicator Stop Code)

This code indicates that the maximum allowable voltage was reached when running your test. Perhaps the accelerometer is disconnected or your amplifier is not turned on. You may need more voltage. Modify the appropriate parameter found in the Field Data Replicator Limits dialog box.

7.5.31 Maximum Start System Gain (Field Data Replicator Stop Code)

This code indicates that the ratio between the output voltage and the measured acceleration exceeds the maximum value allowed while starting the test. Perhaps the accelerometer is disconnected or your amplifier is not turned on. You may need to adjust the starting Max System Gain parameter, found in the Field Data Replicator Limits dialog box.

7.5.32 Max Starting Voltage (Field Data Replicator Stop Code)

This code indicates that the maximum allowable voltage was reached when starting your test. Perhaps

the accelerometer is disconnected or your amplifier is not turned on. You may need more voltage. Modify the appropriate parameter found in the Field Data Replicator Limits dialog box.

7.5.33 Network Congestion (Field Data Replicator Stop Code)

This code indicates the network connection was too slow to keep up with the data transfer rate to stream the demand waveform from the PC's hard drive to the *VibrationVIEW* I/O box. In Field Data Replicator mode it is necessary to have a dedicated 100 Megabit network card for the connection between the PC and the *VibrationVIEW* I/O box.

7.5.34 Test Parameters Changed (Field Data Replicator Stop Code)

This code indicates that the test parameters were changed, by selecting either the Test..Edit Test or the File..Define New Test menu commands or their respective toolbar buttons. As a result of the test parameter changes, the test was halted.

7.5.35 Watchdog Timeout (Field Data Replicator Stop Code)

This code indicates that the communications link between the *VibrationVIEW* program and the Signal Processing hardware was lost. Under normal operations this should never happen. Click the "Run" button to reset the timeout.

The most likely cause of this error is a loss of power to the *VibrationVIEW* I/O unit.

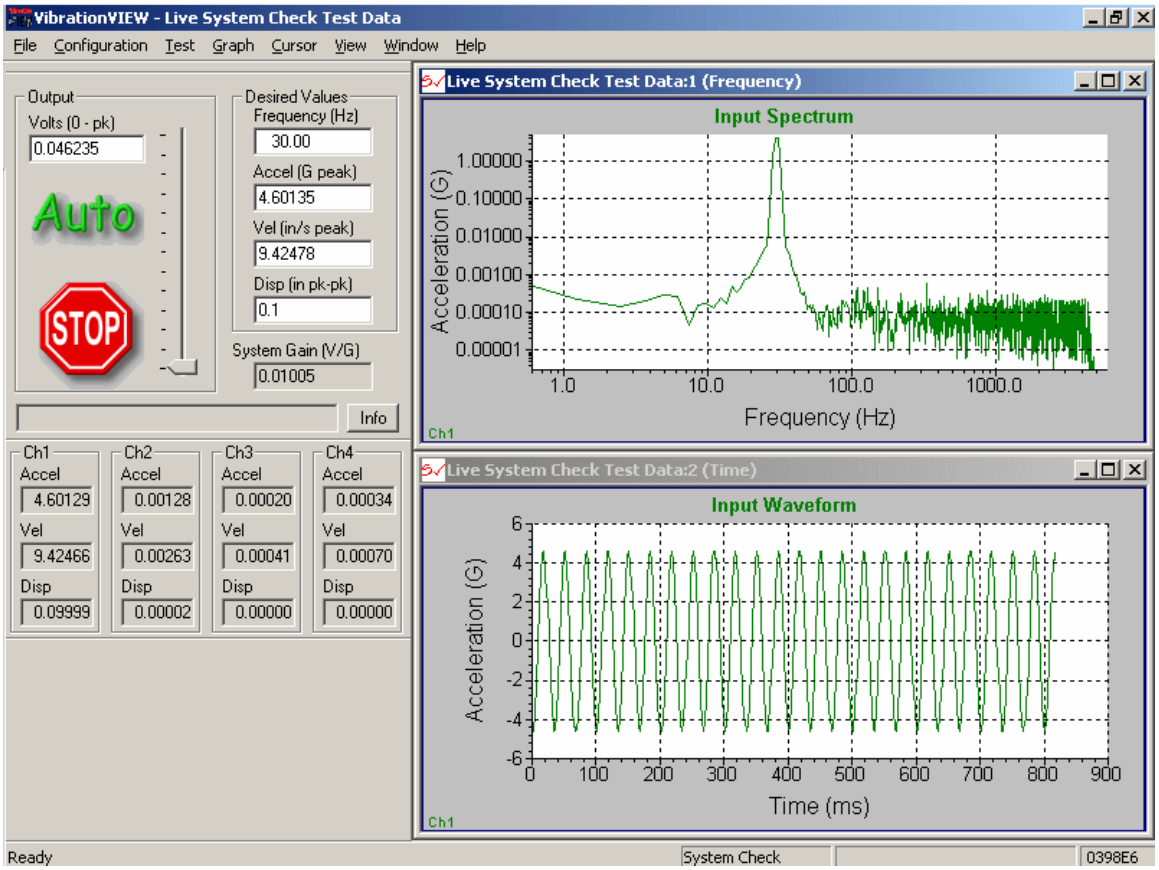
Other events that could cause this to happen:

- The VibrationVIEW I/O unit is turned on after the program is started.
- The Ethernet cable connecting the computer with the VibrationVIEW I/O unit comes loose.
- The computer freezes.
- The computer crashes.

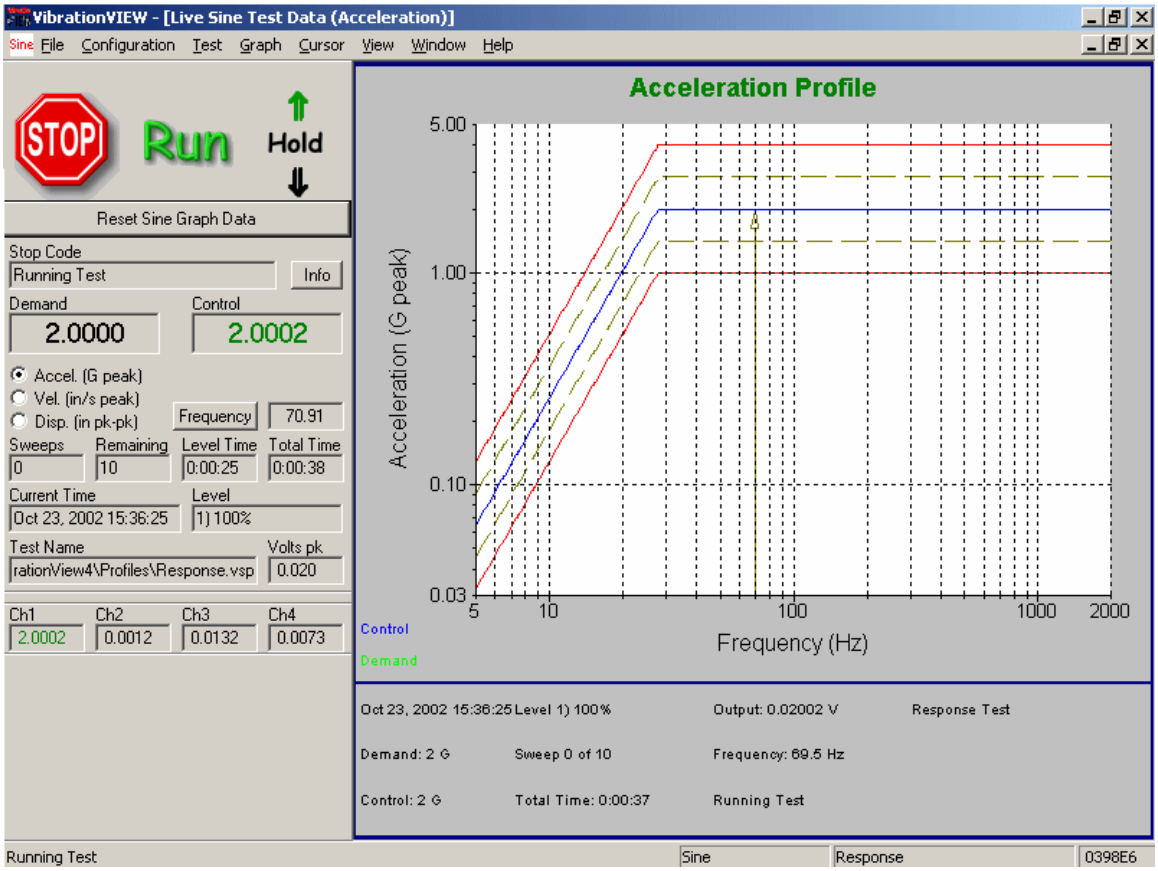
8 Typical Views

System Check
Sine
Random
Classical Shock
Field Data Replicator

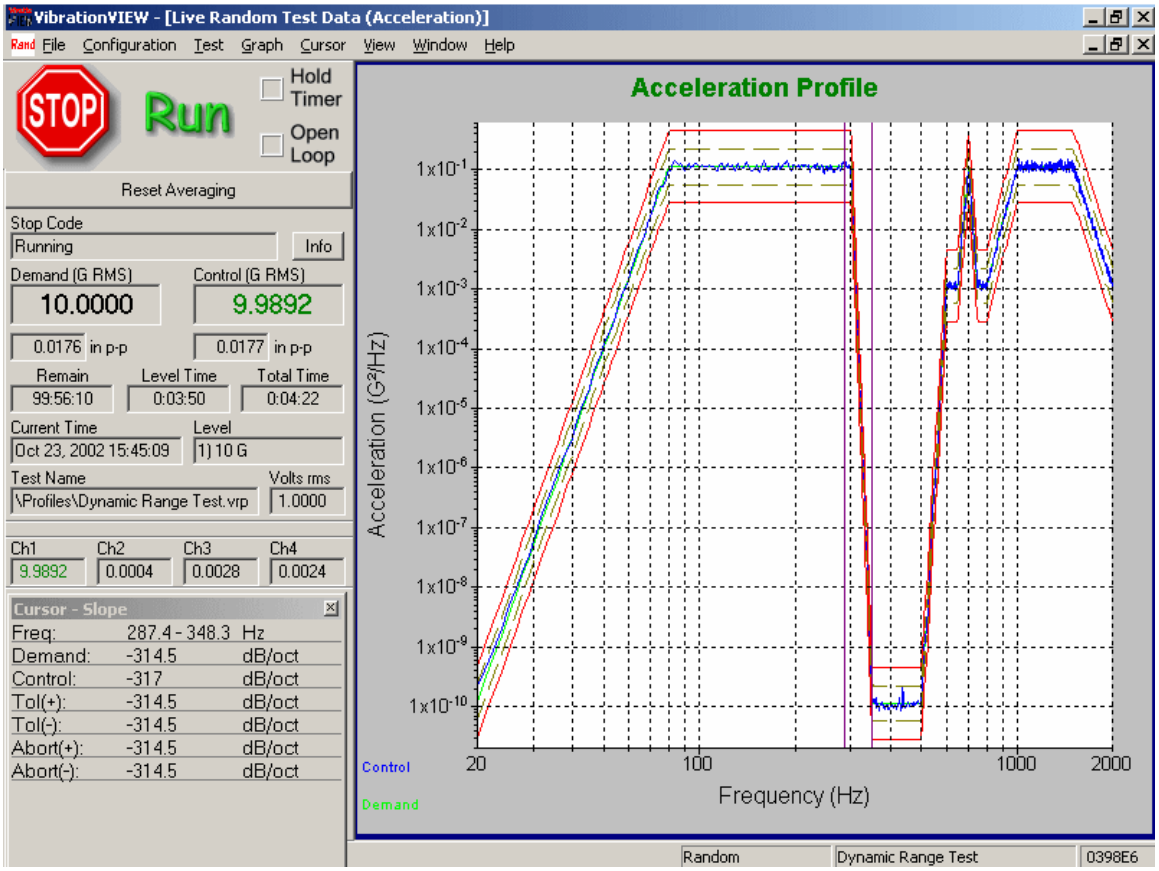
8.1 System Check CRT View



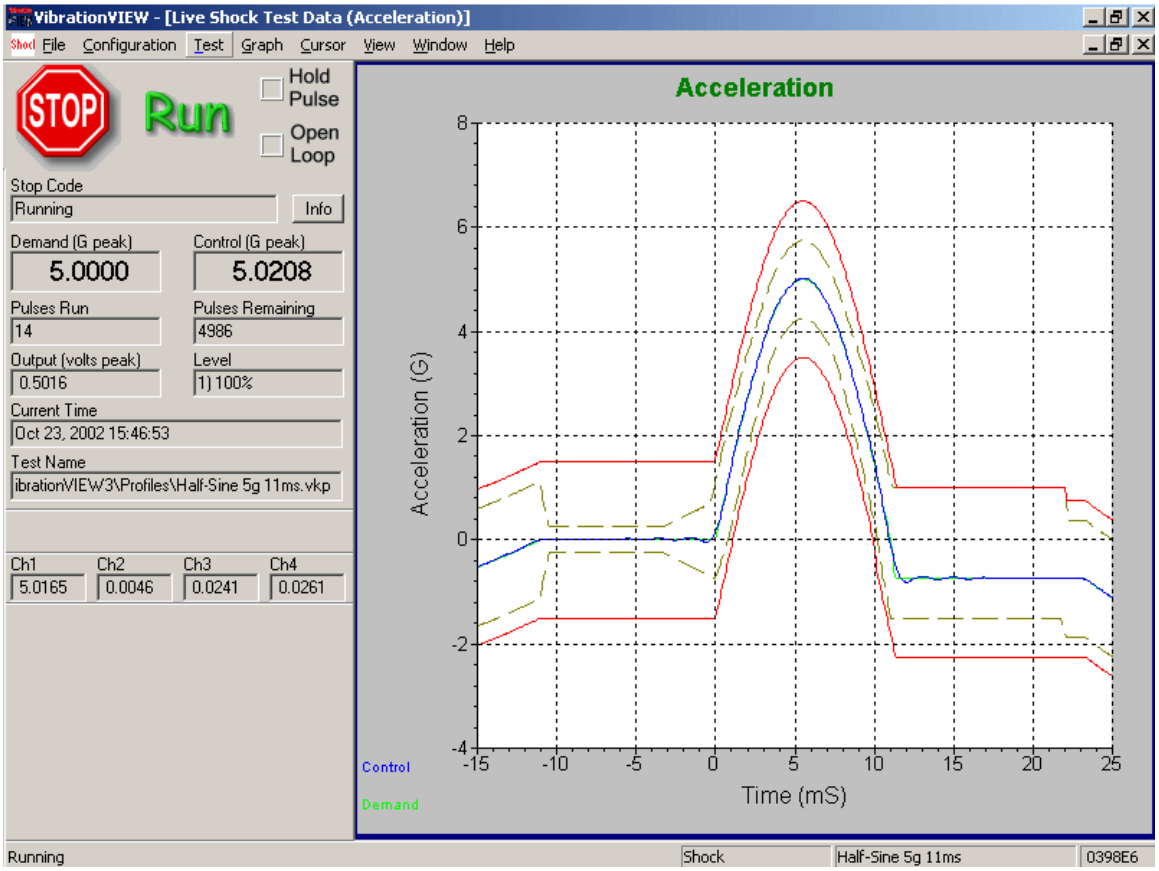
8.2 Sine CRT View



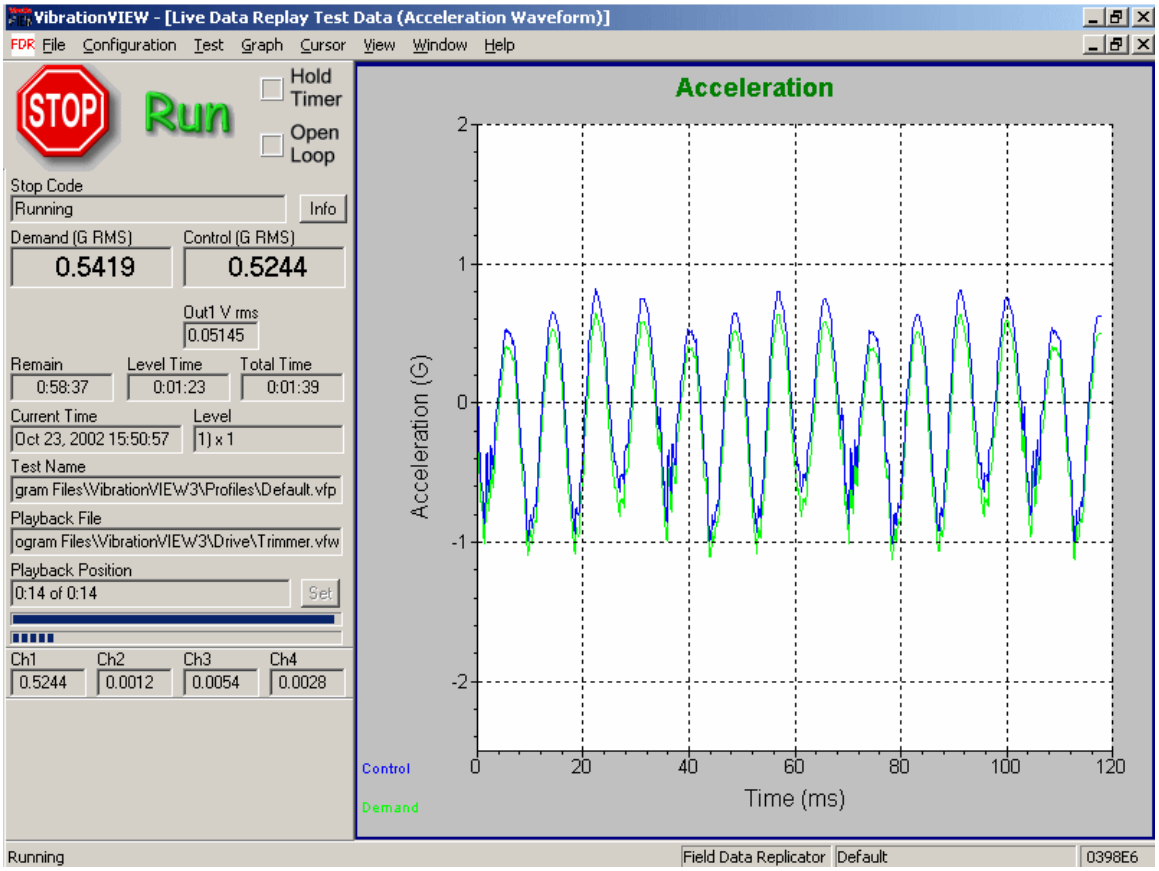
8.3 Random CRT View



8.4 Classical Shock CRT View



8.5 Field Data Replicator CRT View



9 Dialog Boxes

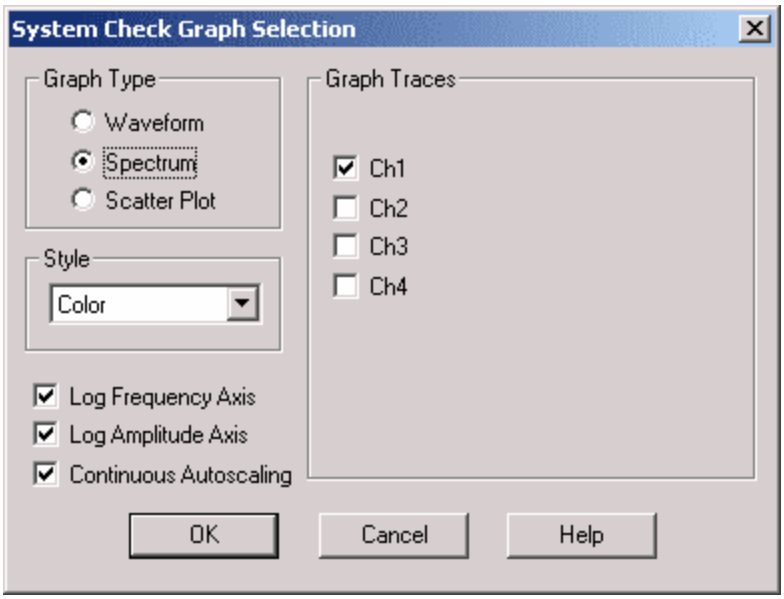
- System Check dialog boxes
- Sine Dialog boxes
- Random dialog boxes
- Shock dialog boxes
- Field Data Replicator dialog boxes
- Other dialog boxes

9.1 System Check dialog boxes

- System Check Graph Settings

9.1.1 System Check Graph Settings dialog box

This dialog box is accessed by selecting the Graph..Edit Graph menu command or clicking the Edit Graph toolbar button.



In the Graph Type" selection area, you can define the type of graph you want to display.

In the Style area, you can define the style of graph you want to display, either Black-and-White or Color.

If the "Log XX Axis" check box is checked, the XX axis will have a log scale. If it is unchecked, the XX will have a linear scale.

In the Graph Traces selection area, you can define which traces you want to be displayed on your graphs.

9.1.2 System Check Graph

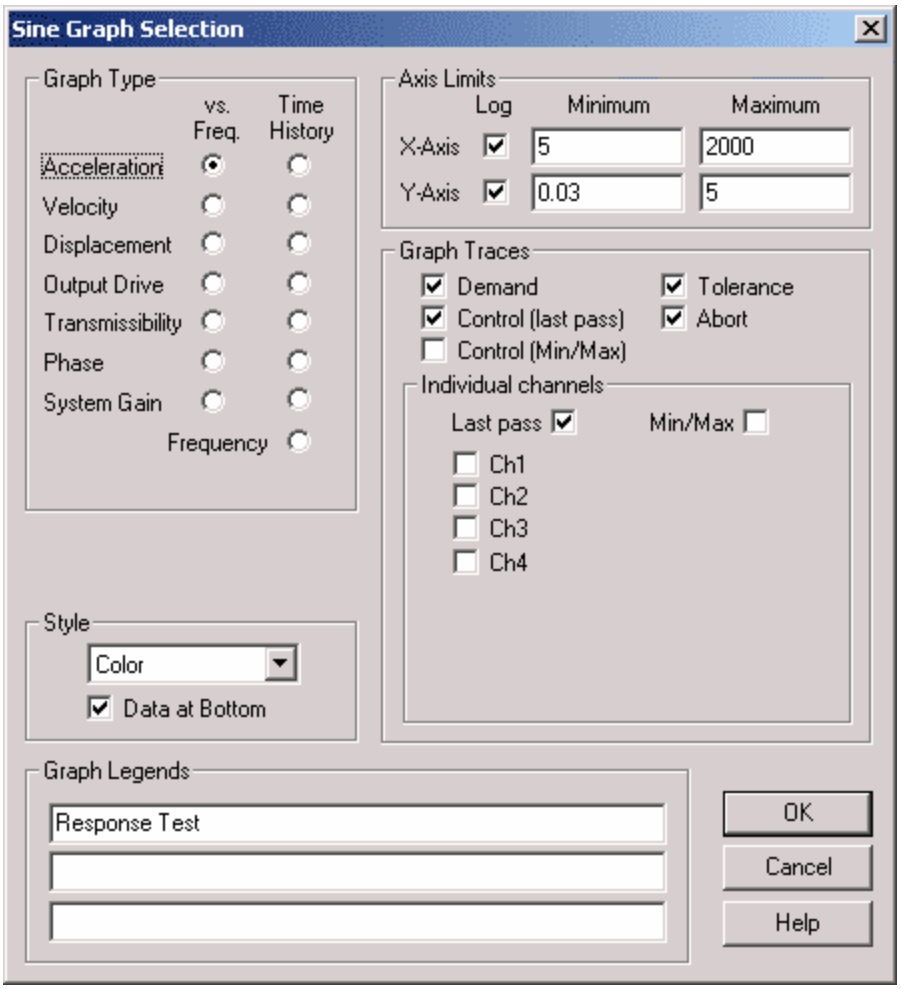
Use the Graph..Edit Graph menu command to change the settings of this graph.

9.2 Sine dialog boxes

- Sine Graph Settings
- Channel 2 Slip Frequency
- Edit Frequency
- Select Resonance Frequencies
- Sine Big Display
- Sine Graph

9.2.1 Sine Graph Settings dialog box

In Sine test mode selecting File..New Graph, Window..New Window or the Graph..Edit Graph Settings menu command or clicking the "Edit Graph" toolbar button displays this dialog box.



In the Graph Type selection area, the first column selects plots of acceleration, velocity, etc. vs. frequency. The second column selects plots of acceleration, velocity, etc vs. time.

In the Axis Limits selection area, you can define the minimum and maximum limits of the axes for the graphs. You can also define the type of axis, either logarithmic or linear.

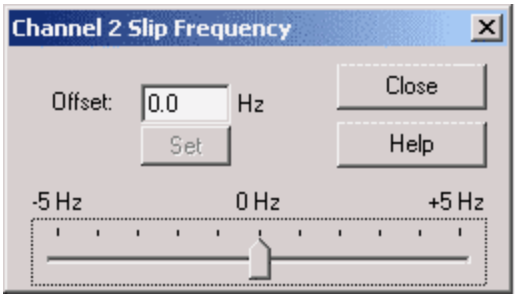
In the Graph Traces selection area, you can define which traces you want to be displayed on your graphs.

You can create Black-and-White or Color graphs using the Style drop-down list.

In the Graph Legends area, you can define up to three lines of text you want to display in the "Data at Bottom" area of the graph. This "Data at Bottom area can be displayed by checking the "Data at Bottom" check box in the Style area.

9.2.2 Channel 2 Slip Frequency dialog box

In Sine test mode selecting the selecting Graph..Sine COLA Slip menu command displays this dialog box.

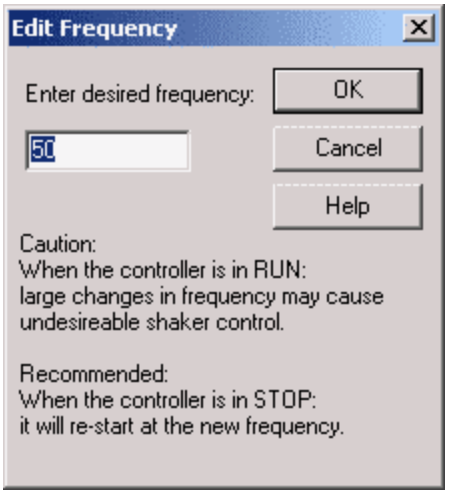


Use this dialog box to adjust the frequency shift for the Constant Output Level Adapter (COLA) output. This is useful for triggering stroboscopes at a frequency slightly shifted from the excitation frequency so that the stroboscope may be used to see device vibration in slow motion.

Frequency shifts between -5 Hz and +5 Hz may be selected by dragging the indicator, or by typing a value in the box.

9.2.3 Edit Frequency dialog box

In Sine test mode selecting the clicking the "Frequency" button in the Sine Control Center displays this dialog box.



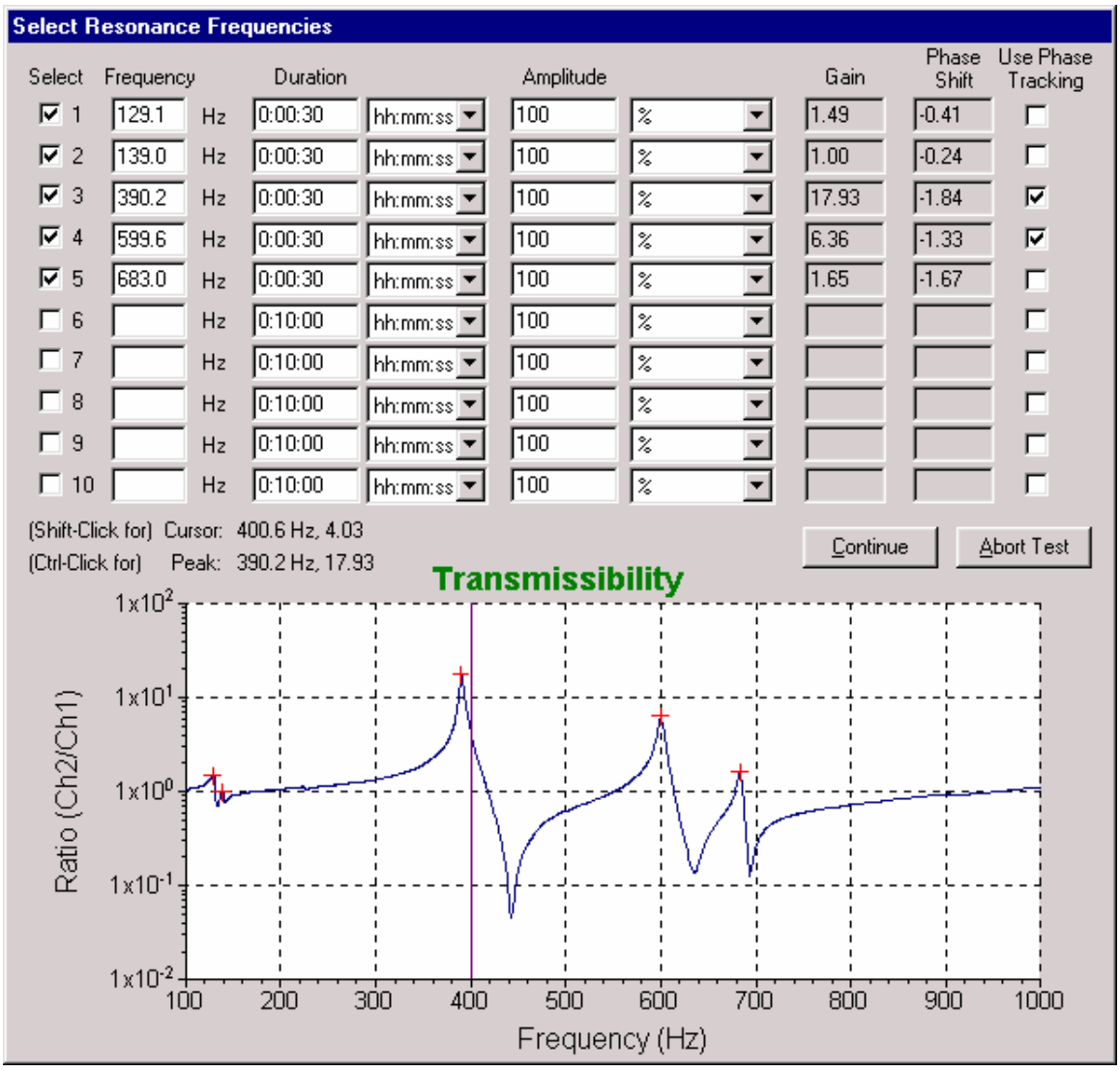
Use this dialog box to set the current and/or the starting frequency in Sine Tests.

When the controller is running, changing the frequency value will put the controller in Hold mode at the frequency defined in this dialog box.

When the controller is stopped, changing the frequency value will change the starting frequency for the sweep.

9.2.4 Select Resonance Frequencies dialog box

This dialog box appears when running Sine test and the "Resonance Table" level is reached in the sine test schedule.



The peaks in the Ch2/Ch1 transmissibility curve are automatically detected and entered into the table. The operator may then select which frequencies to use, or add additional frequencies to the table. When the "Continue" button is clicked, the selected frequencies will be inserted into the test schedule and the test will continue with the first selected frequency.

If a sequence of fixed-frequency levels was already in the test schedule, the new frequencies will be matched to the nearest frequency in the test schedule. The duration and amplitude modifier values for that frequency will be used as the default values in the table. When the test continues, the previous sequence of fixed-frequency levels will be replaced with the new sequence.

Frequencies can be entered into the table manually, or by using the mouse to select a frequency in the transmissibility graph. To add or remove frequencies using the mouse, move the cursor to the desired

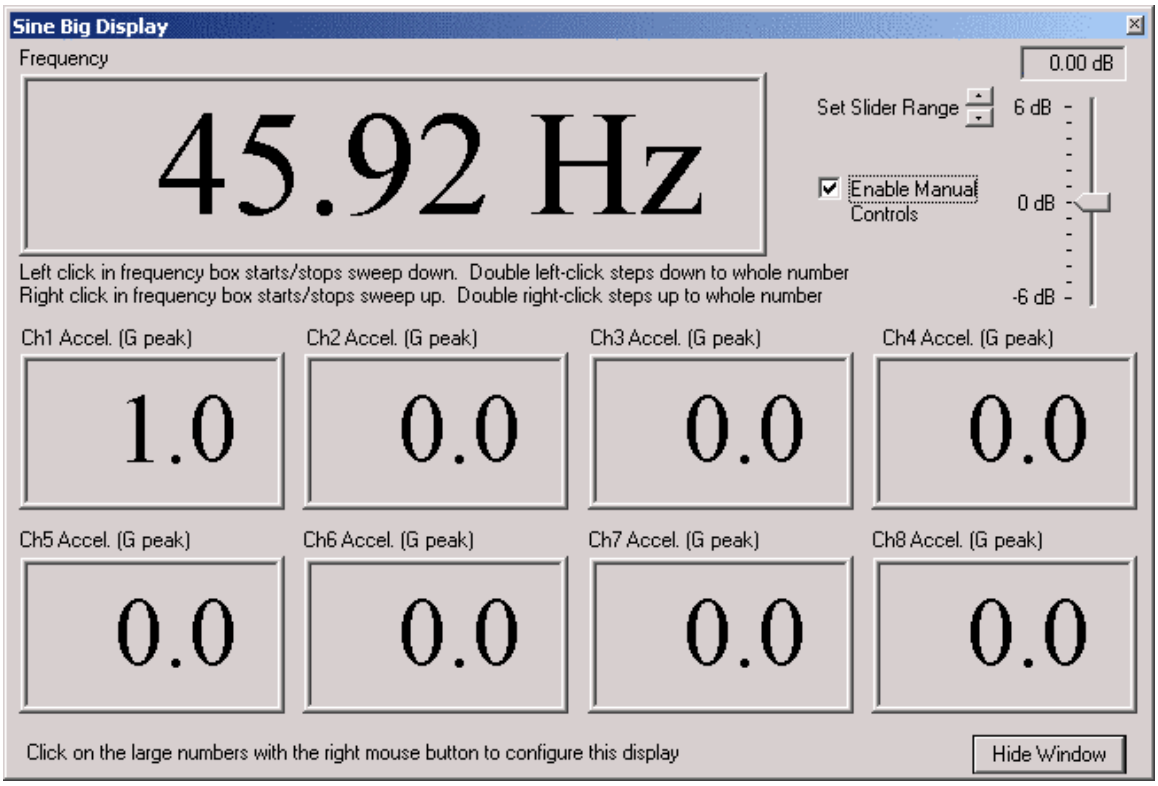
frequency in the graph, hold down either the <Shift> or <Ctrl> key, and click the left mouse button. If you hold down the <Shift> key while clicking on the graph, the exact frequency of the cursor will be used. If you hold down the <Ctrl> key while clicking on the graph, the frequency of the closest peak or valley will be used. Clicking on a frequency that is already in the table will toggle the "Select" check box.

For resonant frequencies that exhibit a 180-degree change in phase between input channels 1 and 2 (typical of sharp resonance peaks), the phase tracking selection will be checked by default. When "Use Phase Tracking" is enabled, the controller will automatically adjust the output frequency to keep the phase shift between channels 1 and 2 at 90-degrees (1.57 radians), and will track the resonance frequency even if it changes over time. The parameters used for this phase tracking of the resonance frequency are defined on the Resonance tab of the Edit Test dialog box. These parameters include a setting for the maximum allowed frequency change that is useful for testing products at a resonance until they fail. When the product fails, the resonance frequency will change significantly, causing the test to shut down.

Once the desired frequencies and appropriate durations are entered into the table, click the "Continue" button to run the test at each of the selected frequencies.

9.2.5 Sine Big Display dialog box

Selecting View..Sine Big Display menu command in Sine test mode displays this dialog box.



This display may be configured by right-clicking on one of the eight parameter display areas and selecting the desired channel and parameter type from the drop-down menu.

This dialog box also provides a feature for remote control (using the mouse) of the sweep direction. Check the "Enable Manual Controls" checkbox, move the mouse cursor into the frequency display box and then use the mouse buttons as follows:

- Left mouse button to start/stop sweep up.
- Right mouse button to start/stop sweep down.
- Double-click either mouse button to step the frequency to the next integer.

Use the scroll wheel (if available) on the mouse or drag the slider to adjust the output level. The "Set Slider Range" spinner control sets the maximum and minimum range of the slider.

When used with a mouse extension cable, the sine frequency may be controlled from a distance.

9.2.6 Sine Graph

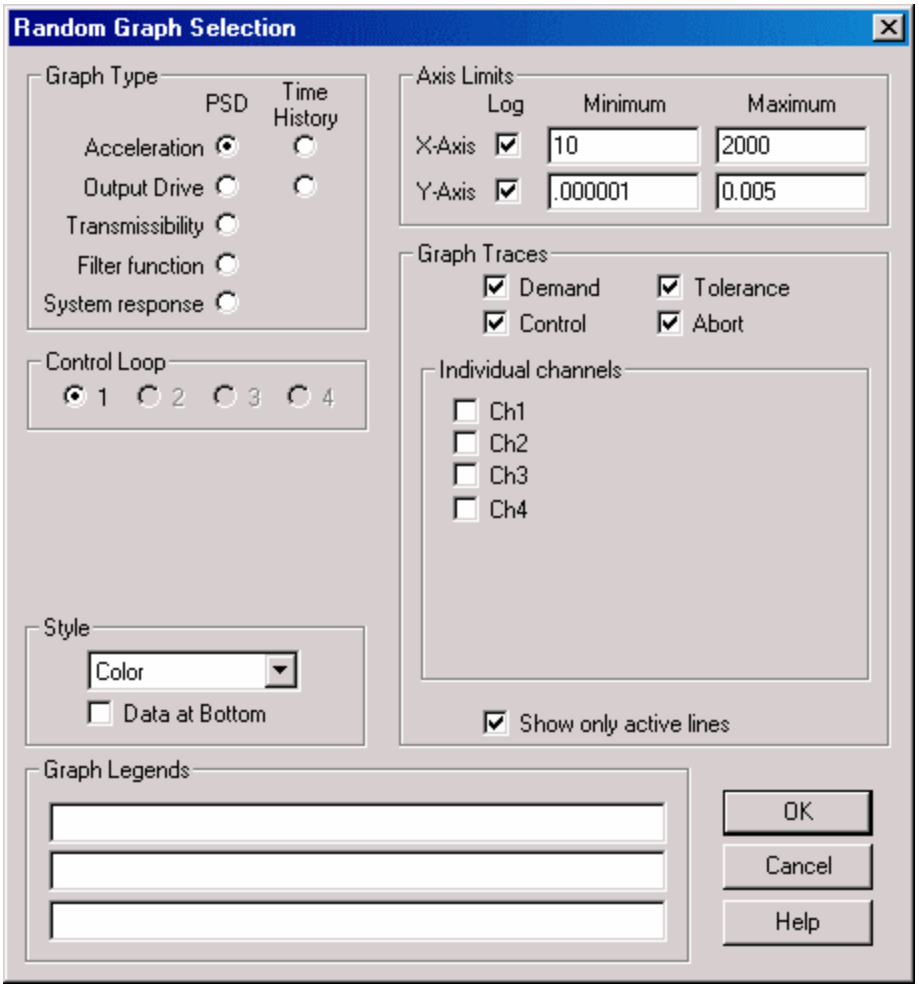
Select the Graph..Edit Graph menu command to change the settings of this graph.

9.3 Random dialog boxes

- Random Graph Settings

9.3.1 Random Graph Settings dialog box

In Random test mode selecting File..New Graph, Window..New Window or Graph..Edit Graph Settings menu command or clicking the "Edit Graph" toolbar button displays this dialog box.



In the Graph Type selection area, the first column selects plots of acceleration, output drive, etc. vs. frequency. The second column selects plots of acceleration, output drive, etc. vs. time.

In the Axis Limits selection area, you can define the minimum and maximum limits of the axes for the graphs. You can also define the type of axis, either logarithmic or linear.

In the Graph Traces selection area, you can define which traces you want to be displayed on your graphs.
To display only the active lines, check the "Show only active lines" checkbox.

You can create Black-and-White or Color graphs using the Style drop-down list.

In the Graph Legends area, you can define up to three lines of text you want to display in the "Data at Bottom" area of the graph. This "Data at Bottom area can be displayed by checking the "Data at Bottom" check box in the Style area.

If you purchased more than one control loop, you will be able to define which control loop you want to display.

9.3.2 Random Graph

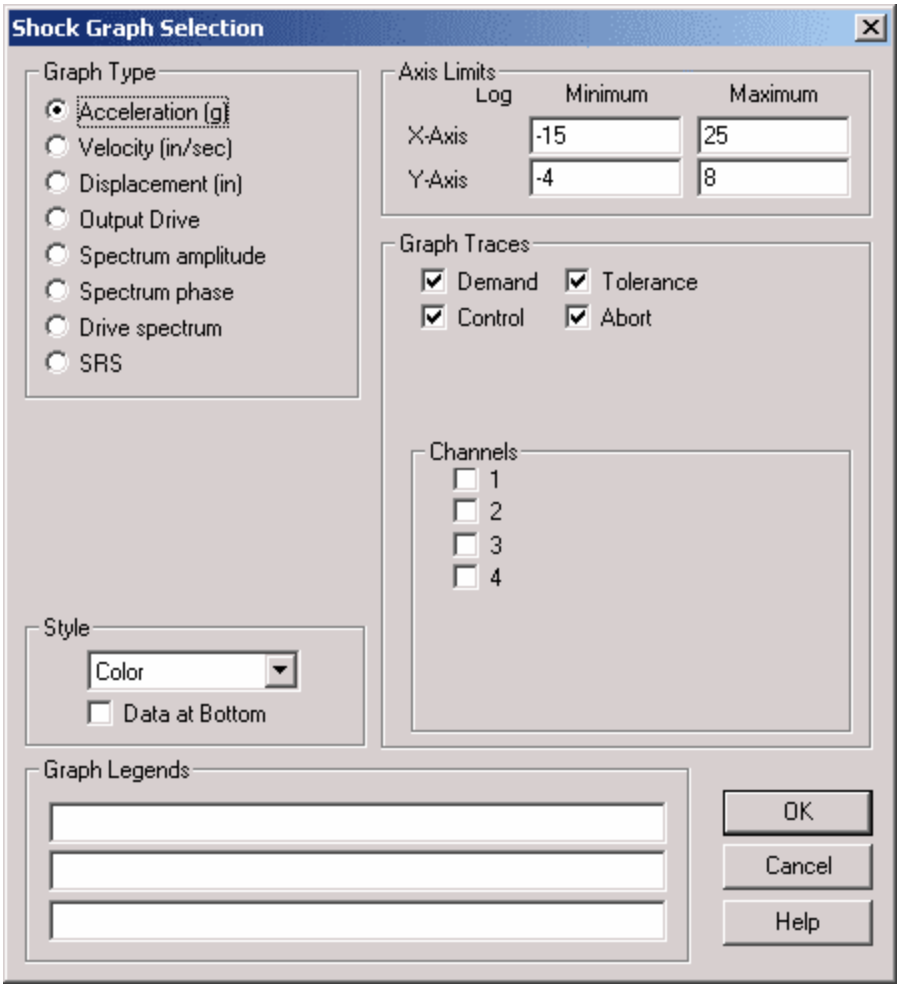
Select the Graph..Edit Graph menu command to change the settings of this graph.

9.4 Classical Shock dialog boxes

- Shock Graph Settings
- Shock Graph

9.4.1 Shock Graph Settings dialog box

In Shock test mode selecting File..New Graph, Window..New Window or Graph..Edit Graph Settings menu command or clicking the "Edit Graph" toolbar button displays this dialog box.



In the Graph Type selection area you can define which type of graph you want to display.

In the Axis Limits selection area, you can define the minimum and maximum limits of the axes for the graphs. You can also define the type of axis, either logarithmic or linear.

In the Graph Traces selection area, you can define which traces you want to be displayed on your graphs.
To display only the active lines, check the "Show only active lines" checkbox.

You can create Black-and-White or Color graphs using the Style drop-down list.

In the Graph Legends area, you can define up to three lines of text you want to display in the "Data at Bottom" area of the graph. This "Data at Bottom area can be displayed by checking the "Data at Bottom" check box in the Style area.

9.4.2 Shock Graph

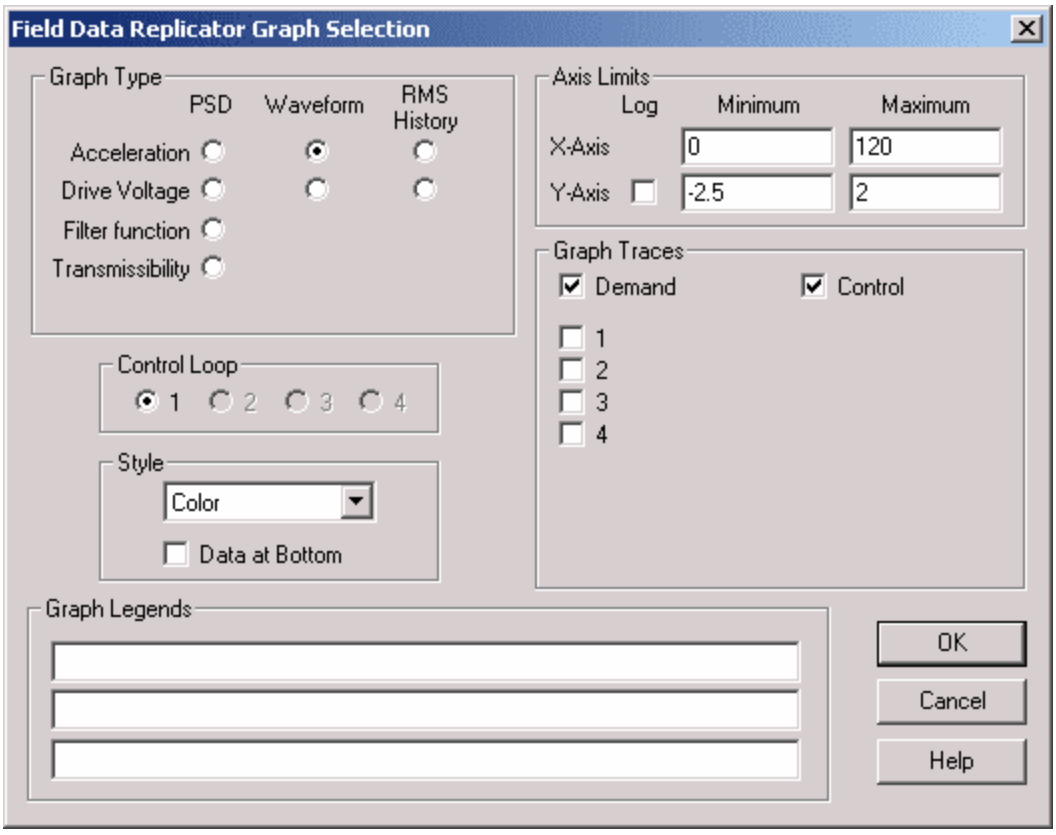
Use the Graph..Edit Graph menu command to change the settings of this graph.

9.5 Field Data Replicator dialog boxes

- Field Data Replicator Graph Settings
- Field Data Replicator Graph
- Drive Check
- Import Universal File Format

9.5.1 Field Data Replicator Graph Settings dialog box

In Field Data Replicator test mode selecting File..New Graph, Window..New Window or Graph..Edit Graph Settings menu command or clicking the "Edit Graph" toolbar button displays this dialog box.



In the Graph Type selection area you can define which type of graph you want to display.

In the Axis Limits selection area, you can define the minimum and maximum limits of the axes for the graphs. You can also define the type of axis, either logarithmic or linear.

In the Graph Traces selection area, you can define which traces you want to be displayed on your graphs.
To display only the active lines, check the "Show only active lines" checkbox.

You can create Black-and-White or Color graphs using the Style drop-down list.

In the Graph Legends area, you can define up to three lines of text you want to display in the "Data at Bottom" area of the graph. This "Data at Bottom area can be displayed by checking the "Data at Bottom" check box in the Style area.

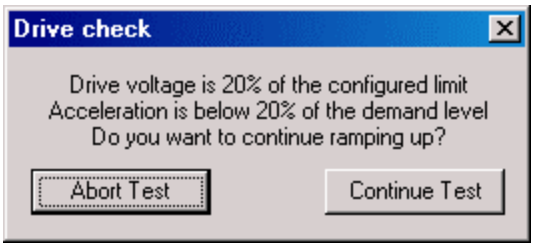
If you purchased more than one control loop, you will be able to define which control loop you want to display.

9.5.2 Field Data Replicator Graph

Use the Graph..Edit Graph menu command to change the settings of this graph.

9.5.3 Drive Check dialog box

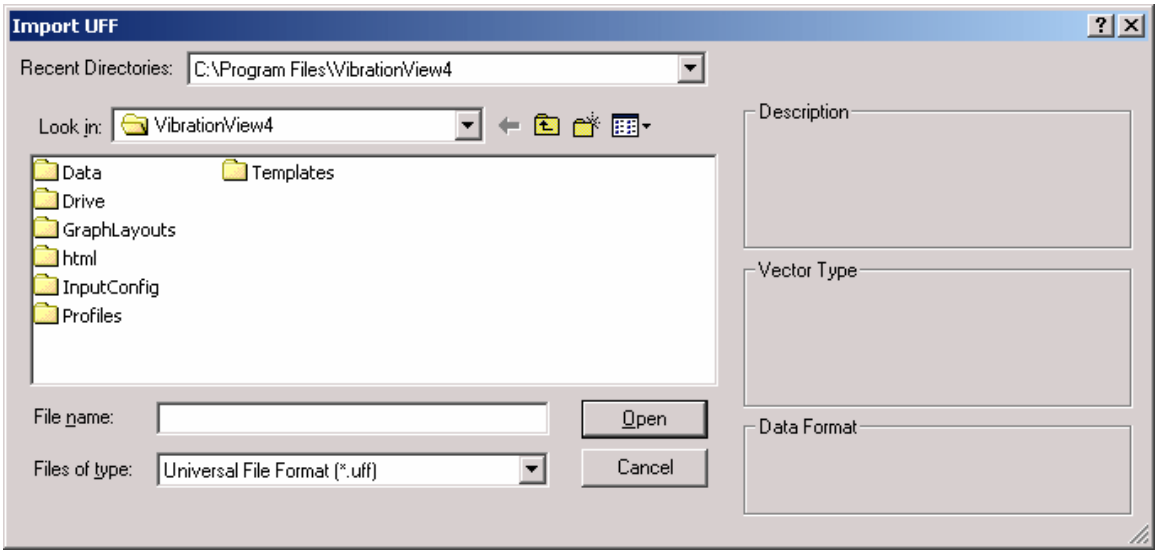
This dialog box is automatically is displayed when Field Data Replicator tests reach 20% and again at 50% of configured maximum drive level during the startup phase of the test. It is an indication your test may be unable to reach level with programmed test limits.



Abort Test: Abort the test and check your setup
Continue Test: Continue to run startup phase.

9.5.4 Import Universal File Format dialog box

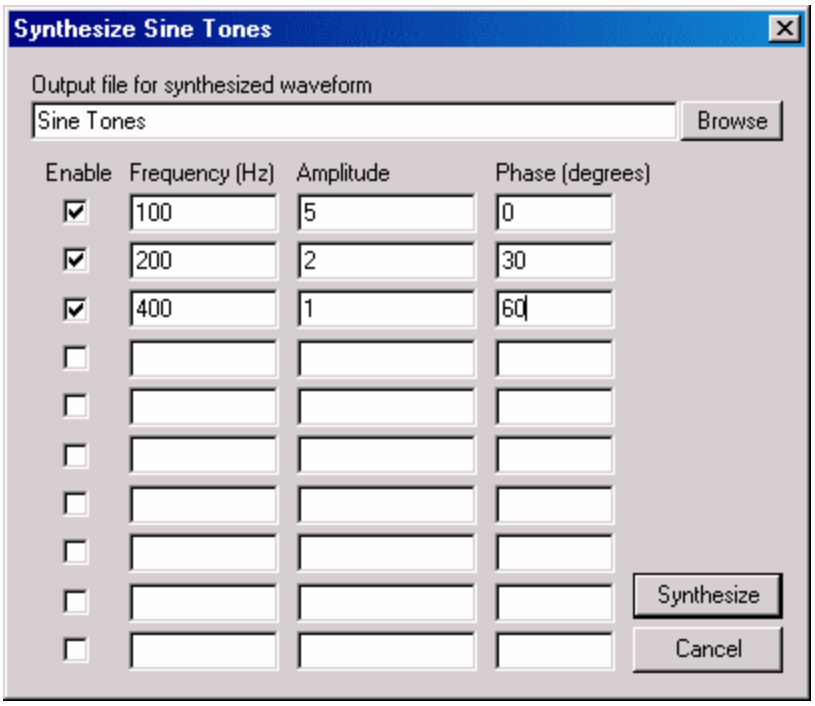
Selecting the File..Edit Test Settings menu command or clicking the Edit Test button on the toolbar, then clicking the "Import from UFF File" button, while an FDR test is open, accesses this dialog.



Use the Import UFF function to bring universal file format files from your data recorder into Field Data Replicator.

9.5.5 Synthesize Size dialog box

Selecting the File..Edit Test Settings menu command or clicking the Edit Test button on the toolbar, then clicking the "Synthesize Sine Tones" button on the "Import" tab, while an FDR test is open, accesses this dialog.



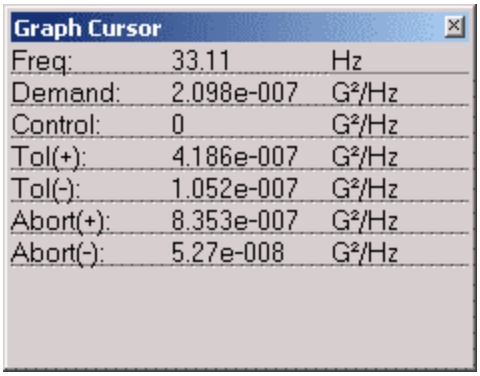
This utility allows you to create a field data replicator file based on superimposed sine tones. Each sine tone is defined by frequency, amplitude and phase. After you define your tones press the Synthesize button to create a new field data replicator replay file containing the specified sine tones.

9.6 Other dialog boxes

- Cursor
- Graph Colors
- Demo Registration
- Enter Registration Code
- Graph Update Time
- I/O Unit Initializing
- Key Not Found
- Memorized Drive
- Motor Controls
- Notice Message
- Password Entry
- Rear I/O Test Panel
- Run Name
- Schedule Loop
- Test Schedule
- VibrationVIEW Test Type
- Wait for operator
- Wait Message

9.6.1 Cursor Display dialog box

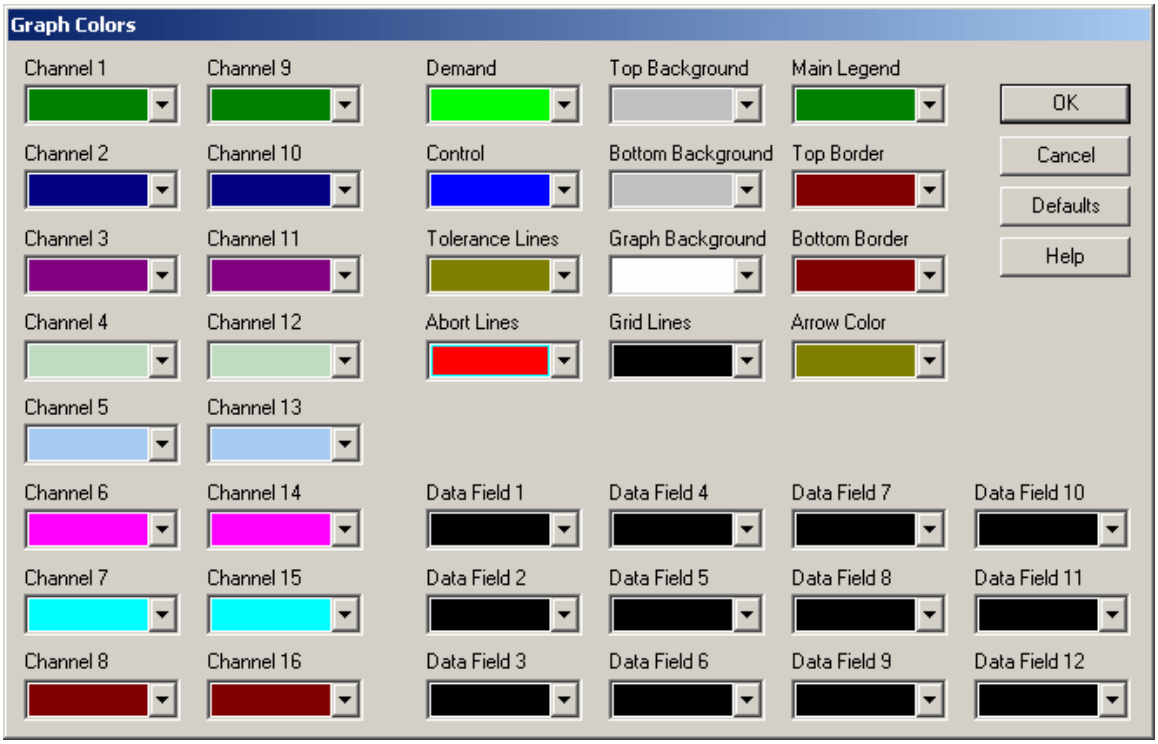
Selecting Graph..Cursor Display menu command displays this dialog box.



This dialog box will display the values of the graph data at the current cursor position in the active graph. The parameters displayed will be specific to the type of graph.

9.6.2 Graph Colors dialog box

Selecting Graph..Graph Colors menu command displays this dialog box.

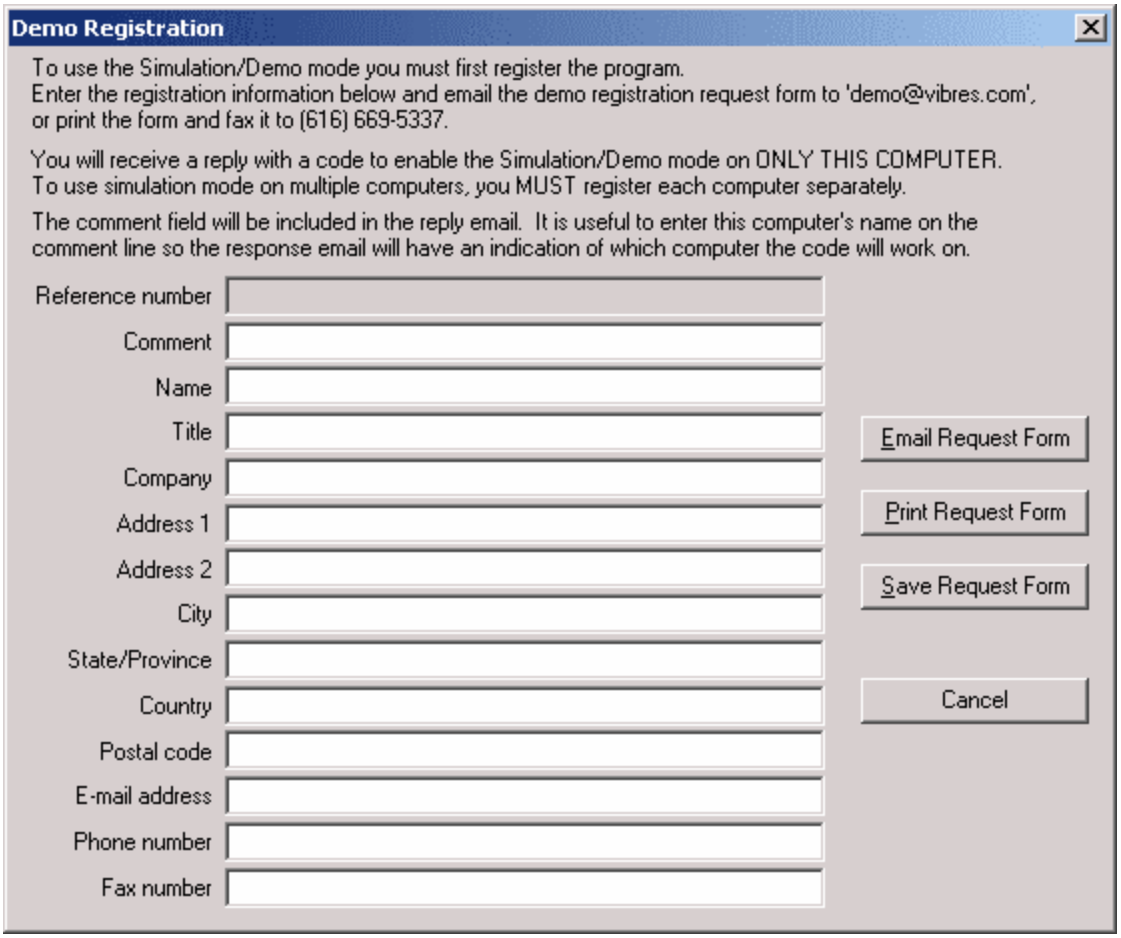


Use this dialog box to customize the colors for various components of the graphs.

Changes made here will only affect graphs that are created after changing the colors. Already open graphs will retain the old settings. To update the colors on any already-open graph, select the graph, press Ctrl+G (Edit Graph Settings command) and click the "OK" button.

9.6.3 Demo Registration dialog box

Selecting Help..Enter Registration Code menu command, then clicking "Request demo code" pushbutton on "Enter Registration" dialog box displays this dialog box.



This is the form which must be filled out to register demonstration mode..

Demonstration mode allows the software to run with a standard PC sound system to emulate a shaker. This mode works without any external equipment and is designed both as a sales demonstration and teaching tool.

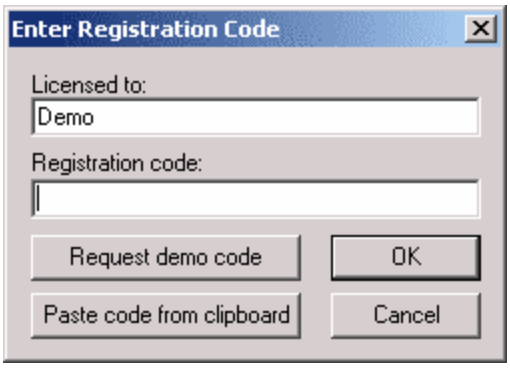
To request demonstration mode fill in ALL the fields on this form and press the "E-Mail Request Form" to send the form to Vibration Research.

You will receive registration e-mail from Vibration Research. Paste the received registration code into the Enter Registration Code dialog Box.

9.6.4 Enter Registration Code dialog box

Selecting Help..Enter Registration Code menu command displays this dialog box.

This dialog box is accessed by selecting the Help..Enter Registration Code menu command.



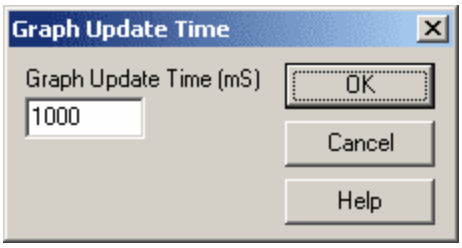
To register Simulation/Demo mode:

Click the "Request demo code" button to display the Demo Registration dialog box.

After you receive your response e-mail from Vibration Research, copy the entire email into clipboard and click the "Paste code from clipboard" button. Enter "Demo" into the "Licensed to" field. Press "OK" to activate demonstration mode

9.6.5 Graph Update Time dialog box

Selecting Graph..Graph Update Time menu command displays this dialog box.



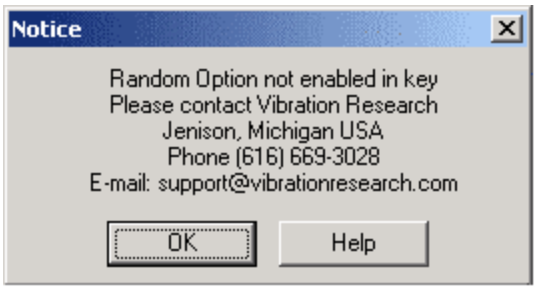
The value specified here is the number of milliseconds between graph updates. The default value is 1000 ms (i.e. 1 second).

9.6.6 I/O Unit Initializing dialog box

This dialog box shows progress of the initialization process of the *VibrationVIEW* I/O unit.

9.6.7 Key Not Found dialog box

This dialog box is displayed when you run a test type that is not enabled in your box, or a test type for which the demo timer has expired. If you get this error message, use the Help..About menu command to see the list of enabled options.

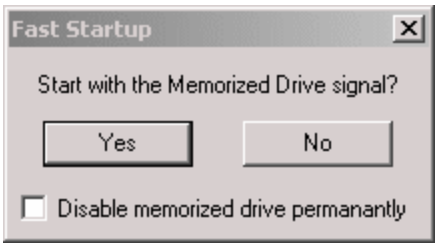


This picture is an example for Random being disabled.

Note that the "Key not found" notice is also displayed when you run a test that requires a feature that is not installed. For example, you could get a "Key not found" notice when you run a Sine test if you have the Extremal mode enabled in the test but the Extremal option is not enabled in the key. In this case, edit the test and disable the Extremal mode.

9.6.8 Memorized Drive dialog box

This dialog box is displayed if you have either a drive memorized for a particular test or a standard drive for your tests, Enable Memorized drive is checked in your test schedule and drive is saved Save Current Drive.



Check the "Disable memorized drive permanently" to disable fast startup for **this** test permanently.

9.6.9 Message dialog box

Miscellaneous dialog box displayed with informational message.

9.6.10 Motor Controls

Motor controls dialog is part of the optional motor simulator package.

9.6.11 Motor Define

Motor Define dialog defines motor operation, with the optional motor simulator package.

9.6.12 Notice Message dialog box

This dialog box is used to provide a variety of notice messages. The dialog box will stay open until the "OK" button is pressed but will not interrupt normal program operation.

9.6.13 Password Entry dialog box

Whenever a password-protected operation is performed this dialog box is displayed. Type in the password and click the "OK" button to continue.

Contact Vibration Research Corporation (616-669-3028) if you have forgotten or do not know the required password.

Note: Use the following procedure to disable the password protection feature of the program:

1. Open the file 'C:\Program Files\VibrationVIEW4\shaker.ini' using Notepad
2. Find the section labeled [Password]

This section should have two lines:

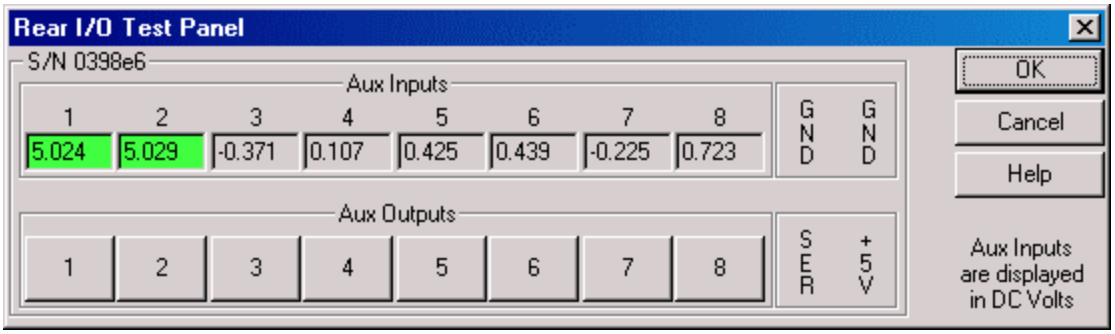
```
Calibrate=ThePasswordHere
Program=ThePasswordHere
```

Enter the passwords in either or both lines in place of ThePasswordHere and save the shaker.ini file.

The Program password is used to protect the test profiles from unauthorized or inadvertent changes. The Calibrate password is used to protect the system calibration dialog box from unauthorized changes.

9.6.14 Rear I/O dialog box

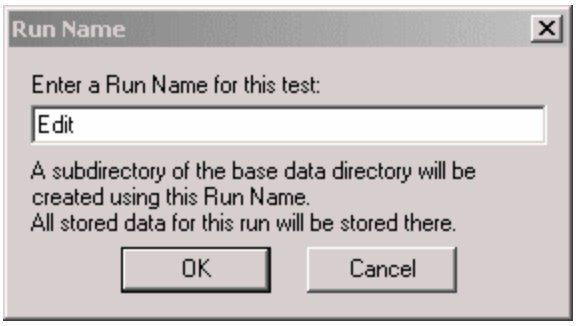
Clicking the "Aux I/O test" button on the System Parameters configuration tab displays this dialog.



This dialog box is used only for diagnostic purposes. It displays the input values for the 8 discrete inputs on the rear of the VR8500 unit. The Aux Outputs can be cycled by pressing the output indicator pushbuttons.

9.6.15 Enter Run Name dialog box

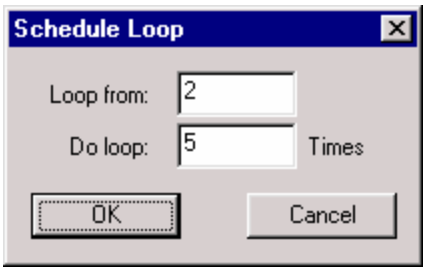
Check the box labeled "Prompt for Run Name when starting the test" on the "Data Storage" tab of the test definition, and this dialog box prompts you each time you run the test.



This will allow a different data directory be entered at the start of each test.

9.6.16 Schedule Loop dialog box

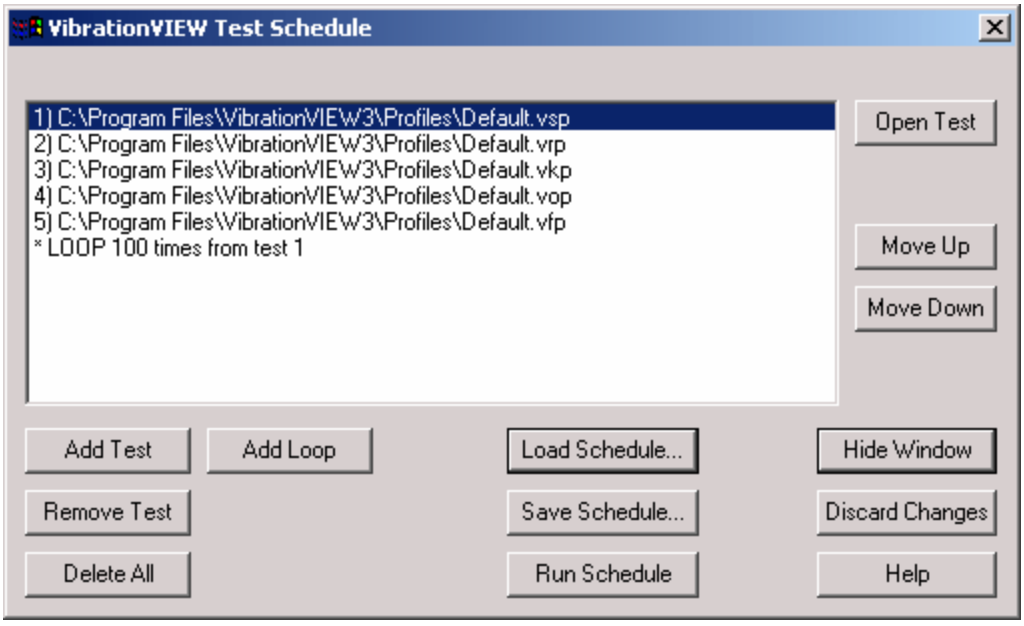
Click the "Add Loop" button in the *VibrationVIEW* Test Schedule menu command then this dialog box prompts for looping parameters.



Enter the test number of the first test you want to repeat in the "Loop from" box and the total number of times you want the sequence of tests performed in the "Do loop x Times" box. Click the "OK" button to enter the loop level into the test schedule.

9.6.17 Test Schedule dialog box

Selecting Test..Test Schedule menu command displays this dialog box..



Use this dialog box to build a schedule of tests to run in sequence. The tests will be run in the listed order, with each test running for the duration specified within the individual test's schedule. There will be a pause of approximately 5 seconds between each test.

The schedule will be aborted if a test is shut down prior to completion of the schedule. (For example, if the operator presses the "Stop" button, or a test encounters an error condition.)

Click the "Run Schedule" button to run all tests beginning with the first test. If you want to start the schedule at a specific test, first select that test and click the "Run Schedule" button.

To add programs to the test schedule, select the test you want the new test to follow and click the "Add Program" button. You will be prompted for a test name. When you click the "Open" button, that test will be added to the schedule.

To remove programs from the test schedule, select the test you want to remove and click the "Delete Program" button. Click the "Delete All" button to clear all entries in the list.

To repeat a sequence of tests multiple times, select the last test you want in the repeat sequence, and then click the "Add Loop" button. A Schedule Loop dialog box will appear prompting you for the first test in the sequence and the number of times you want the sequence performed.

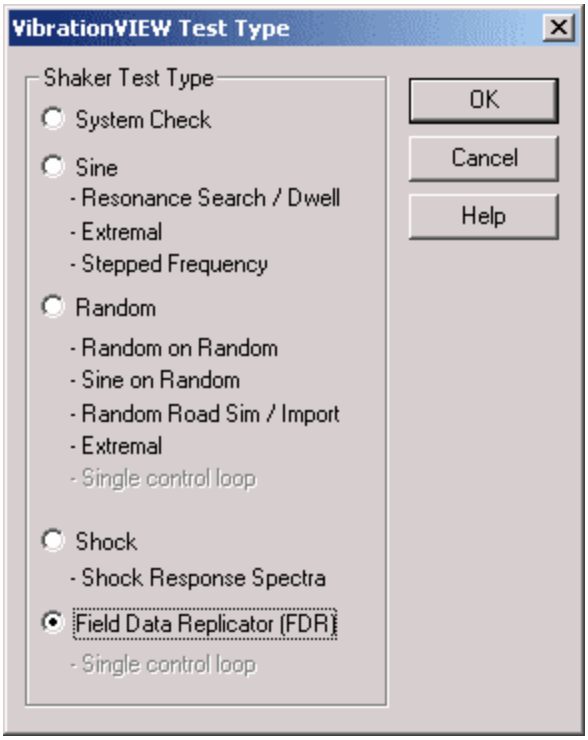
To change the settings of an existing loop, select the loop level in the schedule and click the "Add Loop" button.

Test schedules may be saved on the hard disk by clicking the "Save As..." button and later loaded back in by clicking the "Load..." button.

9.6.18 *VibrationVIEW* Test Type dialog box

Selecting Test..Select Test Type menu command or by clicking 

on the toolbar displays this dialog box.

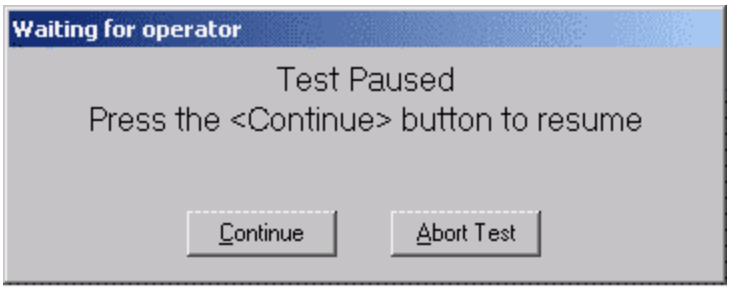


The different test types are software options that are protected by a hardware key. If your hardware key does not have an option enabled, you will still be able to switch to that mode, define profiles and load and plot data sets but you will not be able to run the test.

Contact Vibration Research Corp. for information on options that are not enabled.

9.6.19 Waiting for operator dialog box

A "Wait for operator" level in the test schedule tab pauses the test at the scheduled time until the "Continue" button is pressed.



The test pauses until the operator clicks either the "Continue" button in this dialog box or the "Run" button in the Control Center.

Clicking either the "Abort test" button in this dialog box or the "Stop" button in the Control Center aborts the current test.

9.6.20 Wait message

This dialog box indicates that a file transfer is in progress and provides feedback to the user as to how much remains before the transfer is complete. Please wait until the transfer operation is complete and dialog box closes before continuing.

10 Windows Elements

Close command (Control menus)

Use this command to close the active window or dialog box.

Double-clicking a Control-menu box is the same as choosing the Close command.

 Clicking on the Close icon in the upper right corner of the window is the same as choosing the Close command.

Note: If you have multiple windows open for a single document, the Close command on the document Control menu closes only one window at a time. You can close all windows at once with the Close command on the File menu.

Shortcuts

Keys: CTRL+F4 closes a document window
ALT+F4 closes the *VibrationVIEW* window or dialog box

Move command (Control menu)

Use this command to display a four-headed arrow that allows you to move the active window or dialog box with the arrow keys.

Note: This command is unavailable if you maximize the window.

Shortcut

Keys: CTRL+F7

Minimize command (application Control menu)

Use this command to reduce the *VibrationVIEW* window to the taskbar.

Shortcut

Mouse: Click the minimize icon  on the title bar.
Keys: ALT+F9

Maximize command (System menu)

Use this command to enlarge the active window to fill the available screen space.

Shortcut

Mouse: Click the maximize icon  on the title bar; or double-click the title bar.
Keys: CTRL+F10 enlarges a document window.

Next Window command (document Control menu)

Use this command to switch to the next open document window. *VibrationVIEW* determines what window is next according to the order in which you opened the windows.

Shortcut
Keys: CTRL+F6

Restore command (Control menu)

Use this command to return the active window to its former size and position before you chose the Maximize or Minimize command.

Size command (System menu)

Use this command to display a four-headed arrow that allows you to resize the active window with the arrow keys.

After the pointer changes to the four-headed arrow:

1. Press one of the DIRECTION keys (left, right, up, or down arrow key) to move the pointer to the border you want to move.
2. Press a DIRECTION key to move the border.
3. Press ENTER when the window is the size you want.

Note: This command is unavailable if you maximize the window.

Shortcut
Mouse: Drag the size bars at the corners or edges of the window.

10.1 Scroll bars

These are displayed at the right and bottom edges of the document window. The scroll boxes inside the scroll bars indicate your vertical and horizontal location in the document. You can use the mouse to scroll to other parts of the document.

10.2 Title Bar

The title bar is located along the top of a window. It contains the name of the application and current document.

To move the window, drag the title bar. Note: You can also move dialog boxes by dragging their title bars.

A title bar may contain the following elements:

- Application Control-menu
- Document Control-menu
- Maximize button
- Minimize button
- Name of the application
- Name of the document
- Restore button

10.3 Toolbar

The toolbar is displayed across the top of the application window, below the menu bar. The toolbar provides quick mouse access to many tools used in *VibrationVIEW*,

To hide or display the Toolbar, select the View..Toolbar menu command (ALT, V, T).

Click: **To:**


Change the test type (Test..Select Test Type)


Create a new test (File..Define New Test)


Open an existing test (Test..Open Test)


Edit the current test (Test..Edit Test)


Open a stored data file (File..Open Data)


Save the current data to a file (File..Save Data)


Print the selected graph (File..Print)


Create a report (File..Save as Document File)


Create a new graph (File..New Graph)


Autoscale the active graph (Graph..Full Autoscale)


Edit the active graph's settings (Graph..Edit Graph Settings)


Copy the active graph to the windows clipboard (File..Copy Graph)

Stop the current test (Test..Stop Test)

Start the current test (Test..Run Test)


Show the opening screen of the help file (Help..Help)


Context sensitive help (Help..What's This?)

10.4 Status Bar

The status bar is displayed at the bottom of the *VibrationVIEW* window. To display or hide the status bar, select the View..Status Bar menu command.

The left area of the status bar describes actions of menu items as you use the arrow keys to navigate through menus. This area also shows messages that describe the actions of toolbar buttons and the meaning of numbers as you move the mouse over the buttons and numbers, respectively. When the mouse is not over a toolbar or menu, this area displays the current stop code.

The three other areas of the status bar indicate, from left to right, the current test type (e.g. Field Data Replicator), the name of the currently loaded test (e.g. Default) and the serial number of your controller (e.g. 0x0001).

Insert Generated Index here