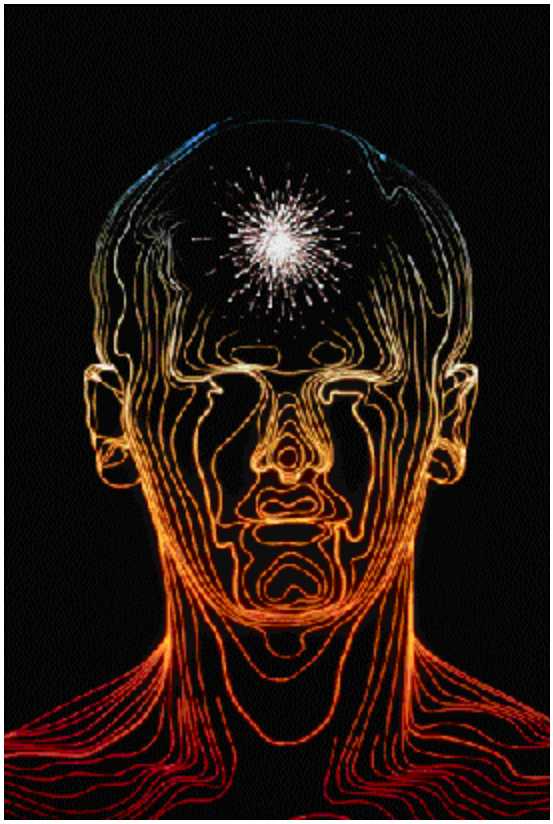
VibrationVIEW

Operators Guide

January 5, 2001



Vibration Research Corporation
2385 Wilshere Dr. Suite A
Jenison, Michigan 49428
U.S.A.

Phone: (616) 669-3028
Fax: (616) 669-5337
Email: support@vibrationresearch.com

1. VibrationVIEW Help Index

1.1. How To ...

Set up the system
Set the shaker system limits
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
Connect and use the Remote Input option
Convert tests created using the version 1.x software
Create customized reports

Call Vibration Research Corporation at (616) 669-3028 with additional questions.

1.2. Menu Commands

File menu
Configuration menu
Test menu
Graph menu
View menu
Window menu
Help menu

Control Center	**Test Definition**	**Stop Codes**
System Check		System Check
Sine	Sine	Sine
Random	Random	Random
Sine-on-Random	Sine-on-Random	Sine-on-Random
Shock	Shock	Shock
Field Data Replicator	Field Data Replicator	Field Data Replicator

2. Quick Setup Instructions

1. If you are installing the hardware into 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". 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 default values by selecting the Test..New Test menu command. Click the "Next >" button to advance through the new test wizard dialogs, accepting the default values for each. Repeat step 9. If the system is "looped on itself", this default test should certainly 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" checkbox 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.

3. Computer requirements

A Pentium computer (We recommend 100MHz or faster). The program will run on 486 machines, but the graph display speed will be unacceptable.
A full size AT style (ISA) expansion slot. Most PCs have these slots, but sometimes the memory or CPU interfere with full-length cards. One full size slot without obstruction is required.
VGA video card & monitor. We recommend using a screen resolution of 1024 x 768 or higher.
Mouse and keyboard
3.5" floppy disk drive or CD-ROM drive or network connection.
Hard disk drive with 50 MB of free space.
Microsoft Windows 95, 98, NT 4.0 or 2000 operating system.
Parallel port for the software protection key and printer.
16 MB RAM minimum (64MB recommended).
Any Windows-compatible printer (optional).
Vibration Research Corp. supplies the DSP card, the analog I/O unit, and the software key.

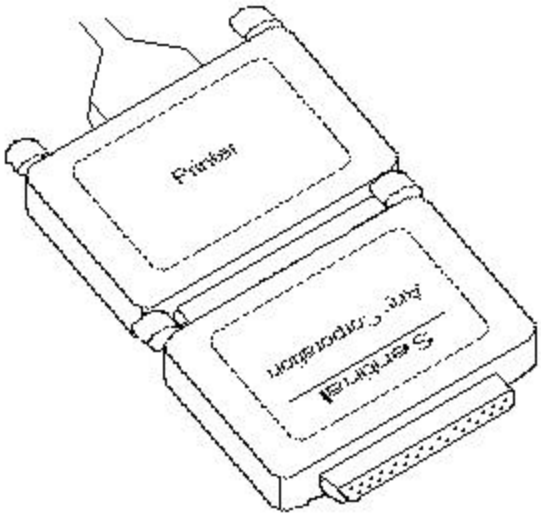
4. How to install the hardware

4.1. About the Key

The software key is shipped in an anti-static bag. It is a small white 25-pin connector with your company name on one side and the test types to which it gives you access on the other. This key is essential to the proper functioning of VibrationVIEW. It is the key that unlocks the software, allowing you to run the test types that you have purchased. The key must be plugged into the parallel port (printer port) on the rear of the controller computer for the controller to run a test.

You can install and run the VibrationVIEW program on a computer without the key attached (for example, your office computer) to read in data files created by the controller computer, and print graphs and reports of that data. All functions will work without the key attached, with the exception of actually running a test.

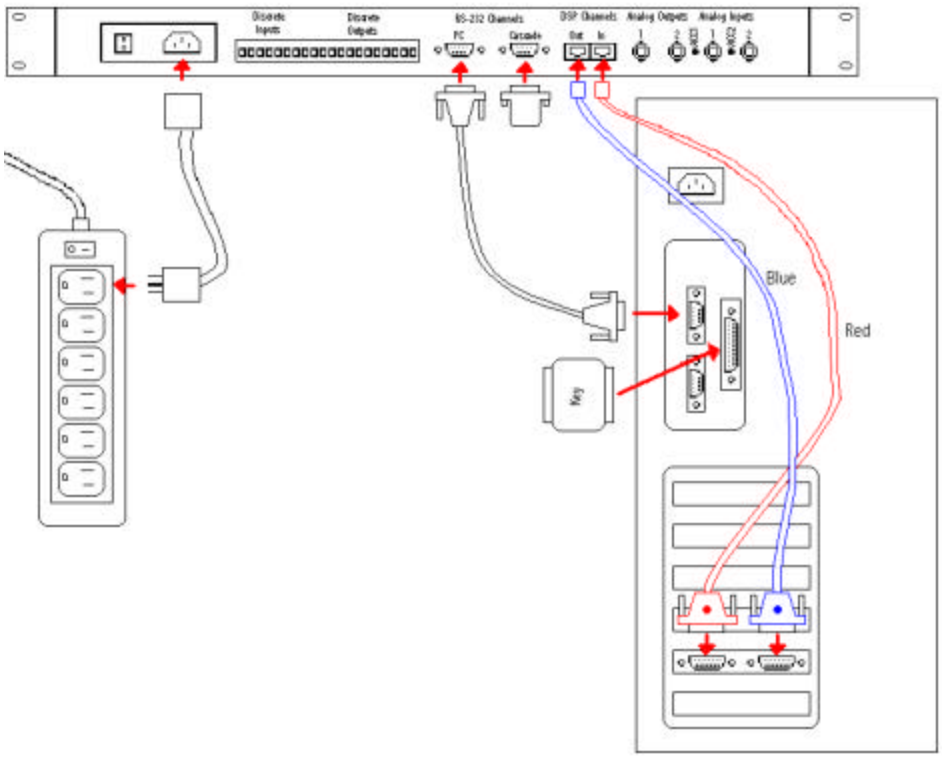
The key can be connected in-line with your printer cable, allowing you to use both the key and the printer at the same time. You can attach any peripheral (usually a printer) that you desire by connecting it to the open, female end of the key.



4.2. Connecting the VibrationVIEW I/O Hardware:

1. Set up the computer in the conventional configuration, with power cables, mouse, keyboard, and monitor.
2. Plug the Security Key into the parallel port of the computer. The Security Key is shipped in the front cover of the manual. If you have a printer installed, install the key between the computer's parallel port and the printer cable.
3. Connect the two DSP card cables between the DSP card in the computer and the VibrationVIEW I/O unit. Match the blue and red cables with the blue and red dots as shown in the figure below for correct connections.
4. Connect the serial cable between Com1 of the computer and the serial port labeled "PC" on the VibrationVIEW I/O unit.
5. Plug the gray terminating plug into the serial port labeled "Cascade" on the VibrationVIEW I/O unit.
6. Connect the 120 VAC power cord to the VibrationVIEW I/O unit.

7. Connect the shaker amplifier's input to Analog Output 1 on the VibrationVIEW I/O unit.
8. Connect accelerometers to Analog Inputs 1 and 2 of the VibrationVIEW I/O unit. For ICP-type accelerometers, turn on the ACC1 and/or ACC2 switches to provide a constant current source for the accelerometers. (The *On* position for the ACC switches is the *Up* position-towards the top of the box.) For other accelerometer types, use an external conditioner and connect the conditioned signals to Analog Inputs 1 and 2 of the VibrationVIEW I/O unit and turn off the ACC1 and/or ACC2 switches. (The *Off* position for the ACC switches is the *Down* position-towards the bottom of the box.)



Connecting Two or More VibrationVIEW I/O units

1. Connect the first VibrationVIEW I/O unit (the unit with two switches, "Start" and "Stop" on the front panel) as described above.
2. The second VibrationVIEW I/O unit (the unit without switches on the front panel) connects to a second DSP board in the controller computer, using another pair of red and blue cables.
3. Connect the serial port labeled "Cascade" on the first I/O unit to the serial port labeled "PC" on the second I/O unit using the supplied 6 inch long interconnect cable.
4. Plug the gray terminating plug into the serial port labeled "Cascade" on the *second* VibrationVIEW I/O unit.
5. Connect 120 VAC power cords to each of the VibrationVIEW I/O units.
6. Connect the shaker amplifier's input to Analog Output 1 on the *first* VibrationVIEW I/O unit. For systems controlling two shakers simultaneously, connect the second shaker amplifier's input to Analog Output 1 on the *second* VibrationVIEW I/O unit.
7. Connect accelerometer channels 1 and 2 to Analog Inputs 1 and 2 of the first VibrationVIEW I/O unit and accelerometer channels 3 and 4 to Analog Inputs 1 and 2 on the second VibrationVIEW I/O unit.

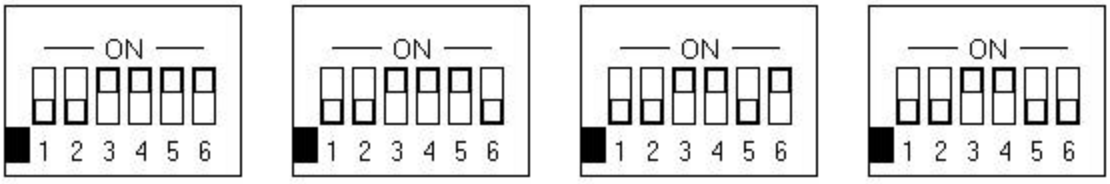
Additional boxes are connected in the same manner, connecting the "Cascade" serial port of one box to the "PC" serial port of the following box. The gray terminator plug must be connected to the "Cascade" serial port of the last box.

Additional Hardware Information:

The hardware for the VibrationVIEW System consists of one digital signal processing (DSP) card and one VibrationVIEW I/O unit for every pair of input channels. The DSP card occupies a full size ISA slot in your computer. The dipswitches on the card with the exception of the address selector should be left in their factory default settings. The address selector (SW3) is set by default to address 300H (768 decimal). For other address settings, refer to the following table:

Hex Addr.	(Decimal)	SW3-1	SW3-2	SW3-3	SW3-4	SW3-5	SW3-6
300H	(768)	OFF	OFF	ON	ON	ON	ON
310H	(784)	OFF	OFF	ON	ON	ON	OFF
320H	(800)	OFF	OFF	ON	ON	OFF	ON
330H	(816)	OFF	OFF	ON	ON	OFF	OFF

300H 310H 320H 330H

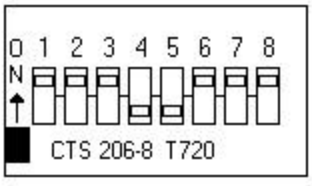


By default the software looks for the first board at 300H, the second board at 310H, the third board at 320H, and the fourth board at 330H. If other addresses must be used due to address conflicts with other interface cards in your computer, the actual addresses used must be set in the file "C:\Program Files\VibrationVIEW\Shaker.ini", in the "DSP Board Address" section. (The board addresses stored in that file are in decimal form.)

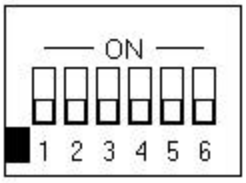
[DSP Board Address]
Board 1=768
Board 2=784
Board 3=800
Board 4=816

The other switch settings will never change. They are the factory default settings and are as follows:

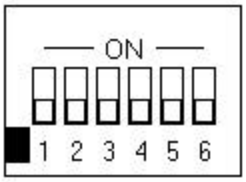
SW1 Positions 1,2,3, 6,7,8 ON 4,5 Off



SW2 Positions 1,2,3,4,5,6 OFF



SW4 Positions 1,2,3,4,5,6 OFF



For ZPD1007 IO Hardware (used in older systems only) use the following procedure:

1. Install the computer in the conventional configuration, with power cables, mouse, keyboard and monitor.
2. Install the Security Key in the parallel port of the computer. The Security Key is shipped in the front cover of the manual. If you have a printer installed, install the key between the computer's parallel port and the printer cable.
3. Install the two DSP card cables between the DSP card in the computer and the ZPD1007 IO unit. The cables can only be connected in the correct polarity. (Note: the connector is the same size as a game port connector. Do not connect the DSP cables to a game port).
4. Connect the external power supply to the ZPD1007 IO unit.
5. Connect the shaker amplifier's input to the "Drive" output of the ZPD1007 IO Unit.
6. Connect *conditioned* accelerometers to the Analog Inputs of the ZPD1007 IO unit (labeled "Channel 1" and "Channel 2", etc)

5. How to install the software

This software package runs on Microsoft Windows 95, 98, NT 4.0, and 2000.

5.1. Installation in Windows 95 and Windows 98

If you purchased this system with a computer, the software will already be installed. Otherwise, install the software using the following steps if the install program does not start automatically:

1. A CD-ROM containing the VibrationVIEW software is supplied with your system. 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 installation is complete, click the "Finish" button.

To run the software, double click the VibrationVIEW icon on the desktop.

5.2. Installation in Windows NT 4.0 and Windows 2000

Windows NT and Windows 2000 require Administrator privileges to install the programs and the required drivers for accessing the DSP card and software protection key. In the following steps it is assumed that the VibrationVIEW CD 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 logged in as Administrator, perform the following steps:

1. Install the VibrationVIEW program according to steps (1) and (2) above.
2. Copy the file "R:\Extra\WinNT Drivers\winrt.sys" (**) to "C:\WINNT\system32\drivers\winrt.sys".
3. Double-click the "R:\Extra\WinNT Drivers\winrt.reg" file to enter the configuration into the registry
4. Run the program "C:\Program Files\VibrationVIEW\Rainbow\Win_NT\setupx86.exe" and

select the Functions..Install drivers menu command to install the Rainbow Sentinel SuperPro drivers.

5. Reboot the computer.

** Note: some of the files mentioned are system files that may be hidden from view in the Windows Explorer. To show system files:

in Windows NT 4.0, select the "View..Options..." menu command, click the "View" tab, and then check the "Show All Files" check box. Then click the "OK" button to close the dialog box.
in Windows 2000, select the "Tools..Folder Options..." menu command, click the "View" tab, and then click the "Show hidden files and folders" radio button. Then click the "OK" button to close the dialog box.

6. 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, either select 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

7. How to set the accelerometer calibration

The calibrated mV/g settings for your accelerometers are entered by selecting the Configuration..Accelerometer Sensitivity 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 accel calibration sheet. You may also (optionally) enter a serial number and a calibration date for each accelerometer to aid in tracing the current calibration settings.

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

8. 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 Help..About menu command, and verify that there is a "Key serial number" listed there. If the About box says "No key found", verify that the hardware key is connected to the computer's parallel port, and then exit and restart the VibrationVIEW program.
2. Select the Test..System Check menu command to switch the system into System Check mode.

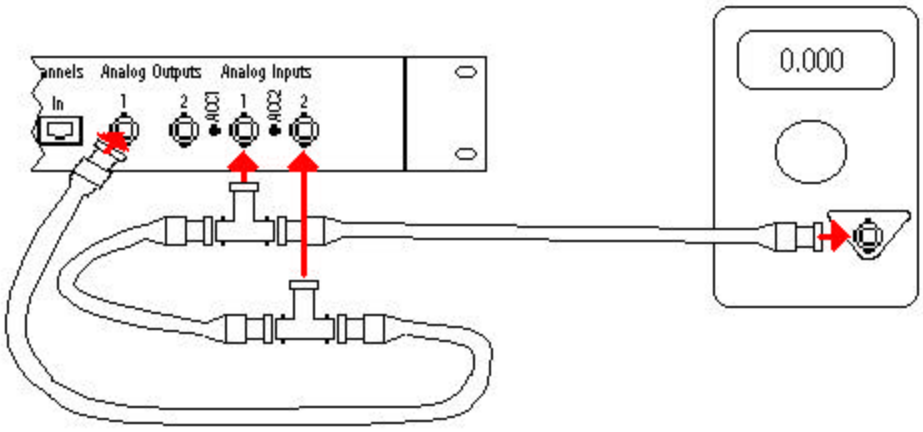
3. Enter a frequency of 30 Hz and a displacement of 0.1 inches (peak-to-peak).
4. Verify that the input levels are near zero. If the input offset is large, select the System Calibration Procedure menu command to set the input offset values.
5. Click the "Auto" button. The output level should slowly ramp 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, and then verify that the channel 1 output from the VibrationVIEW I/O unit 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 output of the VibrationVIEW I/O box to an oscilloscope or 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 this reading is incorrect, select the Configuration..System Calibration menu command to verify that the VibrationVIEW I/O hardware selection matches your hardware, that the Output Drive Gain is 3.3333, and, for VibrationVIEW I/O hardware, and that the Serial Port is set to the port to which the VibrationVIEW I/O unit is connected. If you change any of these settings, click the "OK" button, exit and restart VibrationVIEW, and return to step 3.
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 there still is no input signal, connect the conditioned accelerometer signal to an oscilloscope or voltmeter and verify that you get 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 simply due to bad cable connections.
9. Once you get both an output and an input, 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..Accelerometer Sensitivity menu command to verify that the accelerometer sensitivity settings match the calibrated values for the accelerometers you are using.
10. If the sensitivity values are correct, connect a frequency counter to the output signal, and verify that the output frequency is correct (30 Hz). If the output frequency is not correct and you are using a ZPD1007 I/O unit, select the Configuration..System Calibration menu command to verify that the sample frequency setting matches that used by the VibrationVIEW I/O unit. The ZPD1007 unit should have a calibration sticker on it indicating the internal sample frequency setting.

9. 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 back cover of the manual.***

Note: When the calibration password is enabled, you will get password prompts whenever applying calibration changes. The password for these prompts is "Control" where the 'C' is upper-case and the rest is lower-case.

1. Connect a power cord to the VibrationVIEW I/O unit.
2. Connect 3 coaxial cables together with two T-connectors.
3. Connect one end of the cable to Analog Output 1 of the VibrationVIEW I/O unit.
4. Connect the other end of the cable to the Digital Volt Meter (DVM).



5. Turn the unit on, and allow 1 hour for the temperature to stabilize before performing calibration.
6. Run VibrationVIEW.
7. Set the Accelerometer Sensitivity for each channel to 100 mV/g by selecting the Configuration..Accelerometer Sensitivity menu command. These may already be set.
8. Click the "OK" button.
9. Select the Test..System Check menu command.
10. Select the Configuration..Parameters menu command.
11. Set the Default Sample Frequency to 8000 and the Output Drive Gain to 3.33333. This may already be set.
12. Click the Calibration tab.
13. Set the Gain Multipliers for each channel to 1.
14. Click the "Apply" button to apply the changes. Select "No" when asked if you want to make these changes permanent.
15. Click the "AZ" (auto-zero) buttons to set the output offset values for each channel.
16. Click the "OK" button.
17. Select "No" when asked if you want to make these changes permanent.
18. Connect the T-connectors to Analog Inputs 1 and 2 of the VibrationVIEW I/O box.
19. In the System Check Control Center set the output frequency to 30 Hz and the Volts (0-pk) to 1. Click outside the Volts (0-pk) box.
20. Click the "Yes" button when asked to raise the output voltage to 1.
21. Adjust the Volts (0-pk) until the DVM reads exactly 707 mV (RMS). Write the Volts (0-pk) setting down. This is the Output Gain Multiplier for channel 1.
22. For each input channel in use, record the Ch N Accel value to four significant digits.
23. Calculate the new Input Gain Multiplier values by using 10/(Ch N Accel).
24. Click the "Stop" button to turn the output off.
25. Select the Configuration..System Calibration menu command.
26. Enter the Input Gain Multiplier values you calculated for each channel.
27. Enter the Output Gain Multiplier for channel 1 (from step 20, the 0-pk voltage required to get .707 mV (RMS) on the DVM.
28. Click the "OK" button in the System Calibration Parameters dialog box to apply the new settings.
29. Click the "Yes" button if you want to permanently change the calibration settings. ***Warning!! This will overwrite the previous calibration information in the VibrationVIEW I/O unit, so be sure 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.
30. Restart VibrationVIEW.
31. Set the Desired Value for Frequency to 30 Hz. This may already be set.
32. Adjust the Volts (0-pk) until the DVM reads 707 mV (RMS).

33. Make sure that the Desired Value for Frequency matches the frequency reading on the DVM (30 Hz +/- .01Hz).
34. Repeat steps 30-32 using frequencies of 500, 1000 and 2000 Hz.
35. Disconnect the T-connectors from the Analog Inputs.
36. Connect the coaxial cable to Analog Output 2 of the VibrationVIEW I/O box.
37. Select the Test..Sine menu command.
38. DVM should read about 707 mV (RMS).

Calibration is now complete.

10. How to enter tests

10.1. How to enter a Sine test

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

Select the Test..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 parameter (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 terms of time, 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 Parameters The feedback control parameters 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 for the test.
4. 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 (yellow dashed lines), and are used to determine when to switch to "Run" mode when starting a test.
   The abort lines 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 "How to tune Sine controller parameters" for more information on how to determine the appropriate Max Output and Max

System Gain parameters for your test.

5. Sine Channels The control channels are selected here, as well as how to combine the selected control channels, and whether or not to use tracking filters for either the control channel or the monitor channel inputs.
6. Sine Data Storage The directory into which the data from this test will be stored is entered here. Data can be automatically stored on a regular time interval, sweep interval, at the end of each level (levels are defined using the Sine 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.

10.2. How to enter a Random test

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

Select the Test..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 of the test are entered here. Durations are entered in terms of time, using the format hours:minutes:seconds.
3. Random Parameters The feedback control parameters 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 for the test.
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 (yellow dashed line), and are used to determine when to switch to "Run" mode when starting a test.
   When the "Max Outlier %" of the control lines exceeds the abort lines, 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 "How to tune Random controller parameters" for more information on how to determine the appropriate values for these parameters for your system.
   The maximum output voltage limit is also entered here. This parameter is a safety limit that limits the maximum output voltage produced by the controller.
5. Random Channels The control channels are selected here, as well as selecting 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 Random Schedule dialog), 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.

10.3. How to enter a Sine-On-Random test

Select the Test..Sine On Random menu command to switch into Sine On Random Test Mode.

Select the Test..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. New Test Name You will be prompted for the name to assign to this new test. Enter the desired name, and click the "Save" button.
2. Sine On Random Limits The tolerance and abort limits for the test, measured in dB relative to the demand curve, are entered here. The tolerance lines are reference lines shown on the graphs, and are used to determine when to switch from "Start" mode to "Run" mode when starting a test. When the "Max Outlier %" of the control lines exceeds the abort lines, the test will be aborted. The Drive limits (the maximum output voltage allowed), and the RMS limits (the limits on the RMS value of acceleration, measured in dB relative to the RMS value of the demand signal) are also entered here.
3. Sine On Random Schedule The duration of the test is entered here. Durations may be entered in terms of time only. The schedule also is used to schedule test levels that are scaled to different amplitudes, and to enable or disable the memorized drive.
4. Sine On 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 Sine-on-Random Schedule dialog box), or at the end of the test. Select the check boxes for your desired data storage times, and enter the desired interval times.
5. Sine On Random Parameters The parameters to adjust the behavior of the control loop are entered here. In general, the default values will be sufficient. Refer to "How to tune Sine On Random controller parameters" for more information on how to determine the appropriate parameters for your test.
   The Maximum V/G parameters for both the starting and the running of the test are safety limits that will cause the test to abort when something goes wrong (for example, when an accelerometer cable comes loose.) Suitable values for Max V/G depend on the gain of your amplifier and shaker system. If your shaker system requires 0.5-volt RMS output to achieve 1 G RMS of acceleration, then the typical V/G level of your system would be 0.5 V/G. To allow some variation, set the Max V/G setting to double the typical value, i.e. 1 V/G
6. Sine On Random Tones The Freq Multiplier column determines the multiple of the fundamental frequency for each of the tones. This multiplier may be any floating-point value. The Amplitude G peak column determines the output level for the tone. If either the Freq Multiplier or the Amplitude G peak values for a tone is zero, that tone will be disabled. The tones may be either fixed, or sweep through a frequency range. For a fixed frequency test, enter the desired fundamental frequency in the "Sweep Start" group, and select the Fixed radio . For a swept frequency test, select the frequency range in the "Fund. Sweep" group, the sweep speed in the "Sweep Rate" group, and the starting frequency and direction in the "Sweep Start" group.
7. Sine On Random Define This is the main dialog used for editing Sine-on-Random tests. The frequencies and amplitudes of the breakpoint table are entered here. Up to 200 breakpoints can be entered. Use the scrollbar to scroll through the defined segments, and Insert/Delete buttons to add or remove segments.

You can use the buttons to the right of this dialog box to view and edit all of the parameters that we entered earlier, starting with "Limits..." and ending with "Sine Tones...".

At this point the test definition is complete. Click "Finish" 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.

10.4. How to enter a Shock test

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

Select the Test..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 are entered here. Also the allowable pre-pulse and post-pulse acceleration levels, as a percentage of the pulse peak acceleration level are entered here.
2. Schedule The duration of the test, in terms of number of pulses, is entered here. The schedule also may be used to schedule levels scaled to different amplitude levels.
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 "How to tune Shock controller parameters" 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 yellow reference lines shown on the graphs, and are used to determine when to switch from "Start" mode to "Run" mode when starting a test. The abort lines 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 this test will be stored is selected here. Data can be automatically stored on a regular pulse interval, at the end of each level (levels are defined in the Schedule section), 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.

10.5. 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 Test..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 sample frequency 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 "How to import and export Field Data Replicator waveforms."

2. 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 amplitude levels.
3. Parameters The feedback control parameters are entered here. In most cases the default values will be sufficient. Refer to "How to tune Field Data Replicator controller parameters" for more information on how to tune the parameters specifically for your test.
4. Limits The abort limits for the test are entered here. The control (Ch1) abort limit applies to the controlled 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 input to the input channel 2 on the back 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.
5. 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.
6. Data Storage The directory into which the data from this test will be stored is slected here. Data can be automatically stored on a regular time interval, at the end of each level (levels are defined using the Schedule tab), 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.

To access the waveform recording or importing features, select the Test..Edit Test menu command, and click the Record or Import tabs.

11. How to run tests

11.1. 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 desired setting may be used.

1. Place a displacement wedge on your shaker table.
2. Put VibrationVIEW into System Check mode by selecting the Test..System Check menu command.
3. In VibrationVIEW 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 window and 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.

11.2. 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 also 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.

11.3. 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 dialog box. 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.

11.4. How to run a Sine-on-Random test

To run a sine-on-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 Sine-on-Random Profiles (*.vop)" and select the test you wish to run. Then click the "Open" button to load that test.

To run the test, click the "Run" button in the Sine-on-Random Control Center dialog box. 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.

11.5. 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. Then click the "Open" button to load that test.

To run the test, click the "Run" button in the Shock Control Center dialog box. 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 with making control updates to the waveform.

11.6. 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. Then click the "Open" button to load that test.

To run the test, click the "Run" button in the Field Data Replicator Control Center dialog box. 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.

12. 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 commandor 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
Sine-on-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-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 then 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 it 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 by itself.

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. The graph will then be printed to the selected printer.

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

Laser printers that only print in black and white will produce the best graph output if you select BLACK AND WHITE mode rather than 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 clicking the "OK" button.

13. How to store and retrieve data

13.1. How to store Data

To store the data for the current test, select the File..Save Data menu command or click the "Save Current Data" 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 then click the Data Storage tab 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.

13.2. 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. Using this feature you can quickly scan through many stored data sets.

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

14. How to use memorized drives to instantly start tests

The default behavior of the controller is to slowly ramp up the output level until the desired acceleration level 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.

14.1. Enabling memorized drive

To enable the memorized drive, first start the test, and wait until the output reaches the desired level. Then select Test..Save Current Drive. You will be prompted 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.)

14.2. 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.

14.3. 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 then 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.

15. 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

15.1. How to tune Sine controller parameters

Click 

on the toolbar.

Open the Sine Parameters dialog box.

Start with the settings:

Startup Time	10 seconds
Startup Max System Gain	1 Volt/G
Startup Max Output	0.5 Volts
Running Gain	0.1
Running Max System Gain	1 Volt/G
Running Max Output	1 Volt
Averaging Ch1	10 cycles
Averaging Ch2	10 cycles
Averaging Control	10 cycles
System lag	0 ms

The ***Startup Time*** 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 ***Startup Max System Gain*** and ***Startup Max Output*** values are safety limits that are used to turn off 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. Switch to System Check mode
2. Set the desired frequency value to your startup frequency, and the desired acceleration value to the acceleration level at your startup frequency.
3. Click the "Auto" button and wait for the shaker table to reach the desired level.
4. Copy down the output voltage (0-pk) value, and then click the "Stop" button.
5. Switch back to the Sine Test mode, and edit the test parameters as described at the beginning of this section..
6. Set the ***Startup Max Output*** value to 1.25 times the output voltage value you wrote down.
7. Divide the ***Startup Max Output*** value you just entered by the desired acceleration value you just used in System Check mode. Enter this ratio as the ***Startup Max System Gain*** value.

The ***Running Gain*** value adjusts the responsiveness of the control loop. The larger the value is, the faster the control reacts. If this value is too small, the controller will appear sluggish and slow to respond. If the gain value is too large, the control loop will oscillate (indicated by a wow-wow sound coming from the shaker). If you hear the controller oscillating, reduce the ***Running Gain*** value.

The ***Running Max System Gain*** is a safety limit that is used to turn off the system when something goes wrong (e.g. an accelerometer cable falls off). This value typically is double the ***Startup Max System Gain*** value, but may need to be increased if a 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*** can be determined by running a constant-acceleration test over the desired frequency range. 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.) Divide this voltage level by the acceleration level, and then multiply by 1.25. Enter the computed value in the ***Running Max System Gain*** parameter.

The ***Running Max System Output*** is a safety limit that is also used to turn off the system when something goes wrong. This value typically is double the ***Startup Max System Output***, but may need to be increased if a 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.) Multiply this voltage value by 1.25 and enter it in the ***Running Max System Output*** parameter.

The ***Averaging Factor*** values set the minimum number of sine wave cycles over which to average the measurements. The higher the number is, the more averaging is done. Normally these values should be set to 10 cycles. Reducing the ***Control Averaging Factor*** value will result in noisier readings, but the controller will respond faster to changing amplitudes. If the controller tends to overshoot or not respond quickly enough to changes (for example, during fast sweeps with systems that have a large resonance), this value may need to be reduced to 2 or 4.

15.2. How to tune Random controller parameters

Click 

on the toolbar.

Click the Parameters tab.

Start with the following values:

Starting Average	3	Running Average	10
Starting Gain	0.1	Running Gain	0.04
Starting Loop Time	Auto	Running Loop Time	Auto
Sample Frequency	Auto	Meas. Average RMS	10
Lines	800	Sigma Clipping	6

There are two sets of control parameters:

> The Starting values 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 values are used once the output has reached the desired level, and generally have higher averaging and lower gain values than the corresponding Startup values, because in this mode the system has already been equalized, so only small changes need to be made.

The Gain values are multiplied by the error signal each control loop. The larger the number is, the faster the control will react. If the controller tends to overshoot on startup, you should reduce

the ***Starting Gain***. If the controller takes too long to get up to level, you should increase the ***Starting Gain***. The Gain values should never be more than 1.0.

The ***Starting*** and ***Running Average*** values control the amount of averaging done on the frequency spectrum. The higher the number, the more averaging is done and therefore the smoother the control line will be. However, high amounts of averaging also result in slower controller response. If the averaging setting is large, the gain will have to be low, and conversely, if you want faster system response, you can increase the gain, but must also decrease the averaging. 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 ***Loop Time*** is the time in seconds between each control loop update. This setting should always be left in automatic mode to allow the controller to automatically select a suitable loop time.

The ***Measurement Average RMS*** specifies the amount of averaging used for the RMS values shown in the Random Control Center. It does not affect the response of the controller in any way, and should in general be left at the default value of 10.

The ***Sigma Clipping*** level 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.

The ***Sample Frequency*** in general should be set to automatic to allow the controller to automatically select the optimum sample frequency for the test. If the sample frequency is set manually, it must be greater than 2.4 times the maximum frequency in the test specification.

The ***Lines*** setting 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, decreasing the number of lines can help improve the responsiveness of the controller because less data needs to be sampled to compute the spectrum. When running tests with maximum frequencies below 200 Hz which are not displacement limited, the responsiveness of the controller can be increased by reducing the number of lines down to 400 or 200.

15.2.1. 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*** value 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 typically 0.050 Volts for a Servo-Hydraulic shaker. To determine this value 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.

15.3. How to tune Sine-on-Random controller parameters

Click 

on the toolbar.

Open the Sine-on-Random Parameters dialog box.

Start with the following values:

Sine Average	5
Sine Gain	0.3
Sine Loop Time	4
Start Average	3
Start Gain	0.1
Start Loop Time	Auto
Run Average	10
Run Gain	0.04
Run Loop Time	Auto

Note that there are three sets of control parameters:

> The Sine values are used for controlling the sine tones.
> The Starting values 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 values are used once the output has reached the desired level, and generally higher ***Run Average*** and a lower ***Run Gain*** values (than the corresponding Starting values) are used because in run mode the system has already been equalized, so only small changes need to be made.

The ***Average*** values control the amount of averaging done on the frequency spectrum. The higher the number, the more averaging is done and therefore the smoother the control line will be. However, high amounts of ***Start Averaging*** (above 5) also result in slower controller response.

The ***Gain*** values are multiplied by the error signal 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***. If the controller takes too long to get up to level, you should increase the ***Starting Gain***. The Gain values should never be more than 1.0, and typically should be between 0.02 and 0.5.

The ***Loop Time*** is the time in seconds between each control loop. If the averaging is large, either the loop time will have to be longer, or the gain will have to be lower. In general, you should keep the loop time set to the values recommended above, and decrease the gain level to reduce overshoot.

The ***Start*** and ***Run Loop Time*** parameters in general should both be set to Auto to allow the controller to automatically adjust the loop times to match the Sample Rate and number of lines used for control.

The ***Sine Loop Time*** should generally be one to two times the ***Run Loop Time***.

15.4. 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
Startup pulse interval	1
System lag	0
Sample Frequency	Automatic

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

The ***Max Frequency*** value 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 ***Startup Pulse Interval*** controls how rapidly the pulses are output when the test is starting and the controller is equalizing the output. One second between pulses is generally a good value.

The ***Lag*** value 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 Frequency*** parameter generally should always be set to Automatic to have the controller automatically select a suitable sample frequency. Advanced users may switch this to manual mode to force the system to use a specific sample frequency if they so desire.

Click the "OK" button to save the test, and then 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 startup max output value typically should be more conservative (smaller than) the running

max output value, but the startup max system gain value should typically be higher than the running max system gain. This is because, during startup, the output is increasing, and the startup max system gain 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.

15.5. Using a memorized drive signal

Note: Once a satisfactory pulse has been achieved, you can select the Test..Save Current Drive menu command to memorize the drive signal. After doing this, 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.

15.6. 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:

RMS Error Gain	0.1	RMS Error SNR	2
Spectrum Error Gain	0.1	Spectrum Error SNR	2
Phase Error Gain	0.1	Phase SNR	40
Start Averaging	3	Run Averaging	10
Update Interval	1		

If the output increases to slowly at startup, increase the ***RMS Error Gain*** value. This gain value is used up until the point where the Control RMS value is a factor of 10 below the Reference RMS value. This value should not exceed 1.0.

If the control converges slowly after the Control RMS value is within a factor of 10 of the Reference RMS value, increase the ***Spectrum Error Gain***. This value should not exceed 1.0.

If the controller tends to overshoot the desired values, reduce the ***averaging*** values or increase the ***Update Interval***.

If the sampling rate used by the test is less than 3 kHz, set the ***Update Interval*** to (3000 / SamplingRate). The update interval should always be 1 second or higher. Select the Configuration..System Calibration menu command to check the sampling rate used by the test.

16. 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 channel 2 input of the VibrationVIEW I/O box
Import a waveform from a text (ASCII) file
Import a waveform from a Windows .WAV file
Export a waveform to a Windows .WAV file

16.1. Record a waveform from the channel 2 input of the VibrationVIEW I/O box

To record a signal from the Channel 2 input on the VibrationVIEW IO box, 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 "Record..." button in the Field Data Replicator Define dialog box
5. Connect the signal to be recorded to the channel 2 input.
6. Choose the file into which you wish to store the FDR waveform. 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 "Input sensitivity" to the mV/G rating of the accelerometer connected to the channel 2 input.
8. Set the "Sample Frequency" to the desired sample frequency. This value should be at least 3 times the highest frequency of interest in the input waveform.
9. Click the "Start" button to start recording.
10. When the recording is complete, click the "Stop" button to end recording.
11. Click the "OK" button to close the "Digital Record" dialog box. The new playback file, record level, and sample frequency will be automatically entered into the Field Data Replicator Define dialog box
12. Click the "OK" button to close the Define dialog box and save the test to the hard drive.

16.2. How to import a Field Data Replicator waveform from a text file

To convert an ASCII format file to the VibrationVIEW internal format (FDR 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 FDR 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 value to the sampling rate used when sampling the data in 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 will show the progress of the conversion process.
12. Click the "OK" button in the message box that tells you that the conversion was successfully

completed.

13. Click the "Exit" button to close the ASCII -> FDR dialog box. The new playback file, record level, and sample frequency 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.

16.3. How to import a Field Data Replicator waveform from a Windows .WAV file

To convert from WAV to FDR, 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 "WAV -> FDR" button in the Field Data Replicator Define dialog box
5. Choose the .WAV file that you want to read, and click the "Open" button.
6. Choose the file into which you wish to store the FDR waveform into, and click the "Save" button. 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. The new playback file and sample frequency will be automatically entered in the Field Data Replicator Define dialog box
8. Set the Record level to the mV/G level used when recording this waveform file.
9. Click the "OK" button to close the Define dialog box and save the test to the hard drive.

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

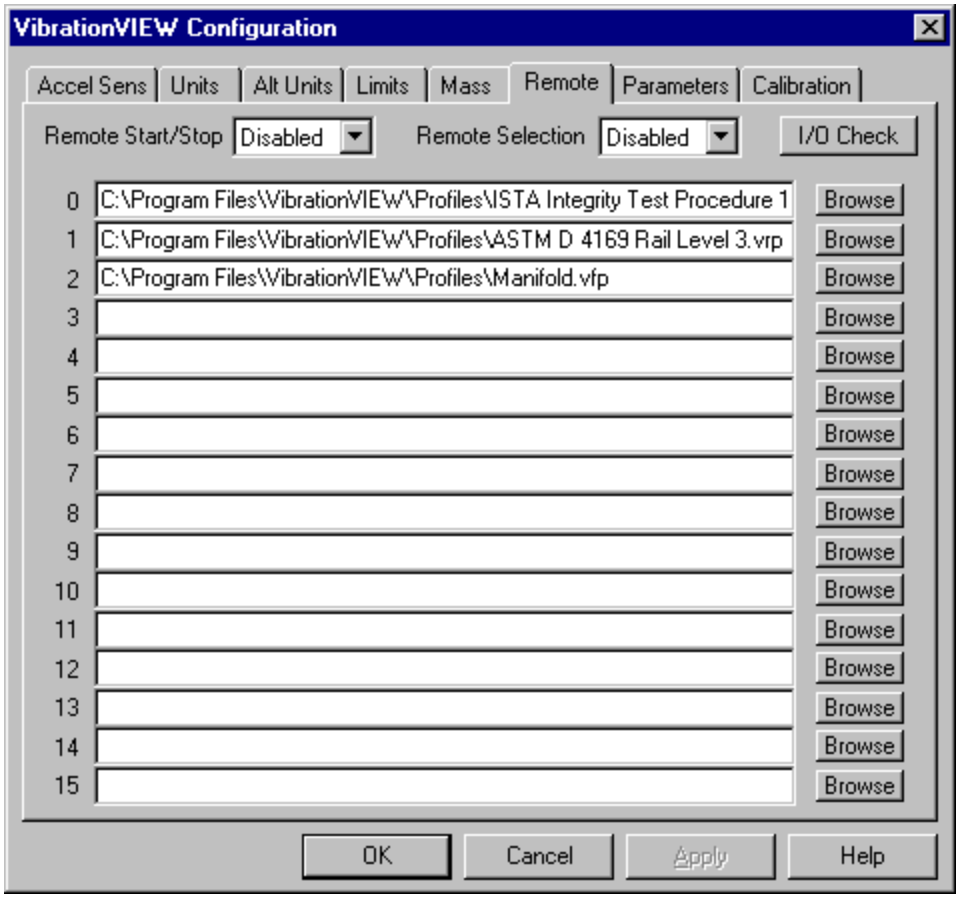
To convert from FDR to WAV, use the following procedure:

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 Define dialog box
5. Choose the .FDR file that you want to read, and click the "Open" button.
6. Choose the .WAV file into which you wish to store waveform, and click the "Save" button.

You may then load the .WAV file using a waveform editor program, and manipulate the waveform. To import the modified waveform back into the Field Data Replicator test, use the WAV -> FDR conversion procedure.

17. Remote Inputs dialog box

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



Use this dialog box to enable and configure the remote input option.

Note: Remote Input is an optional feature that requires that the software key be programmed at the factory. If you haven't purchase the remote input option, you may ignore this configuration dialog box.

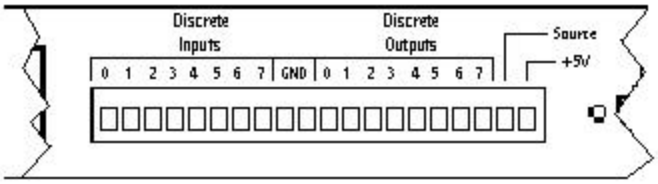


Figure: Input terminals on the rear of the VibrationVIEW IO box.

Discrete Input 0: used for remote start/stop control, is on the far left of the terminal block.
Discrete Input 1: used for the emergency stop input, is the next input.
Discrete Inputs 4,5,6,7 are in the middle of the terminal block.
Note: all discrete inputs have internal resistors pulling them to ground.
Discrete Output 7: used for the test running indicator, is the third connection from the

right of the terminal block.
Discrete Output 6: used for the test aborted indicator, is the fourth connection from the right.

Discrete Input 0: Remote Start/Stop Signal

The Remote Input options allow a remote TTL level input signal to start and stop the test. To use this feature, connect the remote input signal to Discrete Input 0 on the back of the VibrationVIEW IO unit. Then enable remote input by selecting "Enabled" next to the *Remote Start/Stop* label of the above dialog (in the upper left corner of the dialog).

Once this feature has been enabled, the test may be started and stopped using a remote input. When the input signal to Discrete Input 0 makes a transition from 0 volts to +5 volts, the test will start. When the input signal to Discrete Input 0 makes a transition from +5 volts to 0 volts, the test will stop.

You may also operate the remote input through a relay, using the relay to switch the Discrete Input 0 signal between +5V and ground.

Discrete Input 1: Emergency Stop Signal

The emergency stop input is an optional feature (part of the Remote Input package) that connects to Digital Input 1 on the rear of the VibrationVIEW I/O unit. When this input is low (below 2 volts) the outputs of the I/O unit are immediately brought to 0 volts to stop the drive signal going to the amplifier. Typically this input is connected to +5V through a normally closed red mushroom button. When the mushroom button is pressed, the connection to +5V is opened and Digital Input 1 is pulled to ground by a pull-down resistor inside the VibrationVIEW I/O unit. This causes the shaker drive signal to be immediately shut down.

Discrete Inputs 4,5,6,7: Remote Test Selection

When *Remote Selection* is enabled in the above dialog box (upper right corner of the dialog), 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 4, 5, 6, 7 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.

DI7	DI6	DI5	DI4	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

Note: In *Standalone-Mode* (a special operating mode that may be enabled at the factory), the remote test selection is also loaded when the start switch on the front of the VibrationVIEW I/O unit is used to start a test.

Discrete Output 7: Test Running Indicator

When a test is running, Discrete Output 7 on the rear of the VibrationVIEW I/O unit will be connected to the Source input on the same terminal block. When a test is NOT running, Discrete Output 7 will be connected to ground. To get a +5V output signal while the test is running, connect the Source input line to the +5V output on the terminal block.

Discrete Output 6: Test Abort Indicator

When an error occurs causing the test to abort, Discrete Output 6 on the rear of the VibrationVIEW I/O unit will be connected to the Source input on the same terminal block. While no error conditions exist, Discrete Output 6 will be connected to ground. This output mirrors the Red background color for the Stop Code indication in the control buttons dialog box. When the stop code is highlighted with red, this output will be high.

18. How to convert tests created using the version 1.x software

Select the File..Read Old File Formats..Test Profiles menu command to bring up a File Open dialog box. Select the type of version 1.x format test to read (Sine, Random, etc.), and then select the test file from those listed in the dialog box. Click the "Open" button and the test will be loaded.

If a test of the same name with the new format is not present, the converted test will be automatically saved to the same directory with the same file name, but with the new file format's extension. The old format test will not be changed by the version 2.x software, so it can still be used with the old software.

To ensure that the converted test is saved, and to change the name and directory into which it is stored, select the Test..Edit Test menu command to open up the edit test dialog box, and click the "Save As..." button to select a directory and filename into which to save the test.

19. 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
Sine-on-Random data values
Shock data values
Field Data Replicator data values

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

19.1. 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 Frequency:	[PARAM:SampleFrequency]
Startup time:	[PARAM:StartPeriod]
Startup Threshold Voltage:	[PARAM:ThresholdVoltage]
Startup gain limit:	[PARAM:StartSystemGainLimit]
Startup max output:	[PARAM:StartMaxOutputVoltage]
Run control gain:	[PARAM:RunControlGain]
Run low-frequency control gain:	[PARAM:RunLowFreqGain]
Run gain limit:	[PARAM:RunSystemGainLimit]
Run max output:	[PARAM:RunMaxOutputVoltage]
Tracking filter:	[PARAM:TrackFilter]
Ch1,2 Tracking filter:	[PARAM:TrackCh12]
Sweep type:	[PARAM:Sweep]
Expected system lag:	[PARAM:Lag]
Step test setting:	[PARAM:StepTest]
Step frequency:	[PARAM:StepFrequency]
Step on time:	[PARAM:StepOnTime]
Step off time:	[PARAM:StepOffTime]
Resonance setting:	[PARAM:Resonance]
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]
Extremal velocity limits:	[PARAM:MaxV1]
	[PARAM:MaxV2]
	[PARAM:MaxV3]
	[PARAM:MaxV4]
	[PARAM:MaxV5]
	[PARAM:MaxV6]
	[PARAM:MaxV7]
	[PARAM:MaxV8]
Extremal displacement limits:	[PARAM:MaxD1]
	[PARAM:MaxD2]
	[PARAM:MaxD3]
	[PARAM:MaxD4]
	[PARAM:MaxD5]
	[PARAM:MaxD6]
	[PARAM:MaxD7]
	[PARAM:MaxD8]
Control averaging:	[PARAM:CtrlAveraging]
Channel averaging:	[PARAM:Ch1Ave]
	[PARAM:Ch2Ave]
	[PARAM:Ch3Ave]
	[PARAM:Ch4Ave]
	[PARAM:Ch5Ave]
	[PARAM:Ch6Ave]
	[PARAM:Ch7Ave]
	[PARAM:Ch8Ave]
Channel 1 aborts:	[PARAM:Ch1Abort]
Channel 2 aborts:	[PARAM:Ch2Abort]
Channel 3 aborts:	[PARAM:Ch3Abort]
Channel 4 aborts:	[PARAM:Ch4Abort]
Channel 5 aborts:	[PARAM:Ch5Abort]
Channel 6 aborts:	[PARAM:Ch6Abort]
Channel 7 aborts:	[PARAM:Ch7Abort]
Channel 8 aborts:	[PARAM:Ch8Abort]
Channel moment arms:	[PARAM:Ch1MomentArm]
	[PARAM:Ch2MomentArm]
	[PARAM:Ch3MomentArm]
	[PARAM:Ch4MomentArm]
	[PARAM:Ch5MomentArm]
	[PARAM:Ch6MomentArm]
	[PARAM:Ch7MomentArm]
	[PARAM:Ch8MomentArm]
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]
Control phase:	[PARAM:CtrlPhase]
Current demand level:	[PARAM:Demand]
Channel acceleration:	[PARAM:Ch1Accel] [PARAM:Ch2Accel] [PARAM:Ch3Accel] [PARAM:Ch4Accel] [PARAM:Ch5Accel] [PARAM:Ch6Accel] [PARAM:Ch7Accel] [PARAM:Ch8Accel]
Channel velocity:	[PARAM:Ch1Vel] [PARAM:Ch2Vel] [PARAM:Ch3Vel] [PARAM:Ch4Vel] [PARAM:Ch5Vel] [PARAM:Ch6Vel] [PARAM:Ch7Vel] [PARAM:Ch8Vel]
Channel displacement:	[PARAM:Ch1Disp] [PARAM:Ch2Disp] [PARAM:Ch3Disp] [PARAM:Ch4Disp] [PARAM:Ch5Disp] [PARAM:Ch6Disp] [PARAM:Ch7Disp] [PARAM:Ch8Disp]
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]

Accelerometer calibration details:

mV/G sensitivity:	[PARAM:mVg1]
	[PARAM:mVg2]
	[PARAM:mVg3]
	[PARAM:mVg4]
	[PARAM:mVg5]
	[PARAM:mVg6]
	[PARAM:mVg7]
	[PARAM:mVg8]
Calibration data:	[PARAM:Cal1]
	[PARAM:Cal2]
	[PARAM:Cal3]
	[PARAM:Cal4]
	[PARAM:Cal5]
	[PARAM:Cal6]
	[PARAM:Cal7]
	[PARAM:Cal8]

19.2. 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 frequency:	[PARAM:SampleFrequency]
Control channels:	[PARAM:Channels]
Sigma clipping:	[PARAM:SigmaClipping]
Control lines:	[PARAM:Lines]
Max outlier percent:	[PARAM:MaxOutlierPercent]
Max drive voltage:	[PARAM:MaxDrive]

Start loop time:	[PARAM:StartLoopTime]
Start control gain:	[PARAM:StartControlGain]
Start averaging:	[PARAM:StartAveraging]
Start max system gain:	[PARAM:StartMaxSystemGain]
Start threshold volts:	[PARAM:ThresholdVoltage]
Run loop time	[PARAM:RunLoopTime]
Run control gain:	[PARAM:RunControlGain]
Run averaging:	[PARAM:RunAveraging]
Run max system gain:	[PARAM:RunMaxSystemGain]
Drive averaging:	[PARAM:RMSAveraging]
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]
In-band RMS measurements:	[PARAM:ibRMS1] [PARAM:ibRMS2] [PARAM:ibRMS3] [PARAM:ibRMS4] [PARAM:ibRMS5] [PARAM:ibRMS6] [PARAM:ibRMS7] [PARAM:ibRMS8]
Peak system gain:	[PARAM:PeakSystemGain]

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]

Accelerometer calibration details:

mV/G sensitivity:	[PARAM:mVg1]
	[PARAM:mVg2]
	[PARAM:mVg3]
	[PARAM:mVg4]
	[PARAM:mVg5]
	[PARAM:mVg6]
	[PARAM:mVg7]
	[PARAM:mVg8]
Calibration data:	[PARAM:Cal1]
	[PARAM:Cal2]
	[PARAM:Cal3]
	[PARAM:Cal4]
	[PARAM:Cal5]
	[PARAM:Cal6]
	[PARAM:Cal7]
	[PARAM:Cal8]

19.3. Sine-on-Random report parameter names

Values defining the Random 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 frequency:	[PARAM:SampleFrequency]
Control channels:	[PARAM:Channels]
Sigma clipping:	[PARAM:SigmaClipping]
Control lines:	[PARAM:Lines]
Max outlier percent:	[PARAM:MaxOutlierPercent]
Max drive voltage:	[PARAM:MaxDrive]
Start loop time:	[PARAM:StartLoopTime]
Start control gain:	[PARAM:StartControlGain]

Start averaging:	[PARAM:StartAveraging]
Run loop time	[PARAM:RunLoopTime]
Run control gain:	[PARAM:RunControlGain]
Run averaging:	[PARAM:RunAveraging]
Drive averaging:	[PARAM:RMSAveraging]
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: Time, Level)

Values defining the Sine profile and controller parameters:

Sine loop time:	[PARAM:SineLoopTime]
Sine control gain:	[PARAM:SineControlGain]
Sine averaging:	[PARAM:SineAveraging]
Sine low frequency:	[PARAM:SineLowFrequency]
Sine high frequency:	[PARAM:SineHighFrequency]
Sine sweep:	[PARAM:SineSweep]
Sine Multipliers:	[PARAM:Sine1M]
	[PARAM:Sine2M]
	[PARAM:Sine3M]
	[PARAM:Sine4M]
	[PARAM:Sine5M]
	[PARAM:Sine6M]
	[PARAM:Sine7M]
	[PARAM:Sine8M]
	[PARAM:Sine9M]
Sine Demand:	[PARAM:Sine1D]
	[PARAM:Sine2D]
	[PARAM:Sine3D]
	[PARAM:Sine4D]
	[PARAM:Sine5D]
	[PARAM:Sine6D]
	[PARAM:Sine7D]
	[PARAM:Sine8D]
	[PARAM:Sine9D]
Sine tone table:	[PARAM:SineTones]

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]
Channel 1 measurement:	[PARAM:Ch1]
Channel 2 measurement:	[PARAM:Ch2]
Channel 3 measurement:	[PARAM:Ch3]
Channel 4 measurement:	[PARAM:Ch4]
Channel 5 measurement:	[PARAM:Ch5]
Channel 6 measurement:	[PARAM:Ch6]
Channel 7 measurement:	[PARAM:Ch7]
Channel 8 measurement:	[PARAM:Ch8]

Values giving the sine results:

Sine control readings:	[PARAM:Sine1C]
	[PARAM:Sine2C]
	[PARAM:Sine3C]
	[PARAM:Sine4C]
	[PARAM:Sine5C]
	[PARAM:Sine6C]
	[PARAM:Sine7C]
	[PARAM:Sine8C]
	[PARAM:Sine9C]
Sine frequencies:	[PARAM:Sine1F]
	[PARAM:Sine2F]
	[PARAM:Sine3F]
	[PARAM:Sine4F]
	[PARAM:Sine5F]
	[PARAM:Sine6F]
	[PARAM:Sine7F]
	[PARAM:Sine8F]
	[PARAM:Sine9F]
Fundamental frequency:	[PARAM:SineFundamentalFrequency]
Sine reading table:	[PARAM:SineReadings]

Unit names:

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

Accelerometer calibration details:

mV/G sensitivity:	[PARAM:mVg1]
	[PARAM:mVg2]
	[PARAM:mVg3]
	[PARAM:mVg4]
	[PARAM:mVg5]
	[PARAM:mVg6]
	[PARAM:mVg7]
	[PARAM:mVg8]
Calibration data:	[PARAM:Cal1]
	[PARAM:Cal2]
	[PARAM:Cal3]
	[PARAM:Cal4]
	[PARAM:Cal5]

[PARAM:Cal6]
[PARAM:Cal7]
[PARAM:Cal8]

19.4. 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 Frequency:	[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:

mV/G sensitivity:	[PARAM:mVg1]
	[PARAM:mVg2]
	[PARAM:mVg3]
	[PARAM:mVg4]
	[PARAM:mVg5]
	[PARAM:mVg6]
	[PARAM:mVg7]
	[PARAM:mVg8]
Calibration data:	[PARAM:Cal1]
	[PARAM:Cal2]
	[PARAM:Cal3]
	[PARAM:Cal4]
	[PARAM:Cal5]
	[PARAM:Cal6]
	[PARAM:Cal7]
	[PARAM:Cal8]

19.5. Field Data Replicator report parameter names

Values defining the test limits:

Max output voltage:	[PARAM:MaxOutputVoltageRMS]
Max Grms (Ch1):	[PARAM:MaxGrms1]
Max Grms (Ch2):	[PARAM:MaxGrms2]
Max Grms (Ch3):	[PARAM:MaxGrms3]
Max Grms (Ch4):	[PARAM:MaxGrms4]
Max Grms (Ch5):	[PARAM:MaxGrms5]
Max Grms (Ch6):	[PARAM:MaxGrms6]
Max Grms (Ch7):	[PARAM:MaxGrms7]
Max Grms (Ch8):	[PARAM:MaxGrms8]

Values defining the control loop parameters:

Controller status:	[PARAM:Active]
Start averaging:	[PARAM:StartAveraging]
Run averaging:	[PARAM:RunAveraging]
Update interval:	[PARAM:UpdateInterval]
RMS Control Gain:	[PARAM:GainRMS]
Spectrum Control Gain:	[PARAM:GainSpectrum]
Phase Control Gain:	[PARAM:GainPhase]
RMS Control SNR:	[PARAM:SNRRMS]
Spectrum Control SNR:	[PARAM:SNRSpectrum]
Phase Control SNR:	[PARAM:SNRPhase]
Minimum frequency:	[PARAM:MinFrequency]
Maximum frequency:	[PARAM:MaxFrequency]
Notch status:	[PARAM:Notch]
Notch start frequency:	[PARAM:MinNotch]

Notch end frequency:	[PARAM:MaxNotch]
Playback mode:	[PARAM:Mode]
Playback file:	[PARAM:PlaybackFileName]
File name:	[PARAM:FileName]
Test profile name:	[PARAM:TestName]
Sample frequency:	[PARAM:SampleFrequency]
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 1 measurement:	[PARAM:Ch1]
Channel 2 measurement:	[PARAM:Ch2]
Channel 3 measurement:	[PARAM:Ch3]
Channel 4 measurement:	[PARAM:Ch4]
Channel 5 measurement:	[PARAM:Ch5]
Channel 6 measurement:	[PARAM:Ch6]
Channel 7 measurement:	[PARAM:Ch7]
Channel 8 measurement:	[PARAM:Ch8]
Peak system gain:	[PARAM:PeakSystemGain]

Unit names:

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

Accelerometer calibration details:

mV/G sensitivity:	[PARAM:mVg1]
	[PARAM:mVg2]
	[PARAM:mVg3]
	[PARAM:mVg4]
	[PARAM:mVg5]
	[PARAM:mVg6]
	[PARAM:mVg7]
	[PARAM:mVg8]
Calibration data:	[PARAM:Cal1]
	[PARAM:Cal2]
	[PARAM:Cal3]
	[PARAM:Cal4]

[PARAM:Cal5]
[PARAM:Cal6]
[PARAM:Cal7]
[PARAM:Cal8]

19.6. 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 histor
[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
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)
MinX10.0	Force the minimum value on the X axis to 10.0
MaxX100.0	Force the maximum value on the X axis to 100.0
MinY10.0	Force the minimum value on the Y axis to 10.0
MaxY100.0	Force the maximum value on the Y axis to 100.0

Example:

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

19.7. 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
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
MinX10.0	Force the minimum value on the X axis to 10.0
MaxX100.0	Force the maximum value on the X axis to 100.0
MinY10.0	Force the minimum value on the Y axis to 10.0
MaxY100.0	Force the maximum value on the Y axis to 100.0

Example:

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

19.8. Sine-on-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
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)
MinX10.0	Force the minimum value on the X axis to 10.0
MaxX100.0	Force the maximum value on the X axis to 100.0
MinY10.0	Force the minimum value on the Y axis to 10.0
MaxY100.0	Force the maximum value on the Y axis to 100.0

Example:

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

19.9. 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
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)
MinX10.0	Force the minimum value on the X axis to 10.0
MaxX100.0	Force the maximum value on the X axis to 100.0
MinY10.0	Force the minimum value on the Y axis to 10.0
MaxY100.0	Force the maximum value on the Y axis to 100.0

Example:

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

19.10. 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:FiltTime]	Filter function vs. time
[GRAPH: Xmiss]	Transmissibility (ChX/ChY) vs. frequency

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

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)
MinX10.0	Force the minimum value on the X axis to 10.0
MaxX100.0	Force the maximum value on the X axis to 100.0
MinY10.0	Force the minimum value on the Y axis to 10.0
MaxY100.0	Force the maximum value on the Y axis to 100.0

Example:

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

20. Menu Commands

20.1. File menu commands

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

New Graph	Creates a new graph.
Copy Graph	Copy the selected graph to the windows clipboard.
Save Meta File	Saves the selected graph in emf format
Save Bitmap File	Saves the selected graph in bmp 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.
Save Data	Save the active data set to the disk.
Close	Closes an opened graph.
Read Old File Formats	Read test profiles and data created with version 1.x software.
Create Report	Creates a saved report 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.
Explorer	Starts Windows Explorer
Exit	Exits VibrationVIEW.

20.1.1. 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 dialogs and graph types for the various tests, see:

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

Shortcuts

Toolbar: 

Keys: Ctrl+N

20.1.2. 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

20.1.3. Save Meta File (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 such that it can be directly pasted into another application.

20.1.4. Save Bitmap File (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 such that it can be directly pasted into another application.

20.1.5. Open Data command (File menu)

Use this command to read a data set previously saved in 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

20.1.6. 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 the currently selected graph type.

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.

20.1.7. 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 the currently selected graph type.

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.

20.1.8. Save Data command (File menu)

Use this command to store the data 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

20.1.9. 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. The graph's system menu is accessed by clicking the icon in the upper left corner of the window, as shown below:

20.1.10. Read Old File Formats command (File menu)

Use this nested menu to read test profiles or data files created using version 1.x of the VibrationVIEW software. Two options are given:

Test Profiles
Data

20.1.11. Old Test Profiles command (File menu)

Use this command to read test profiles created using version 1.x of VibrationVIEW. An Open File dialog box will be displayed, allowing selection of the test to load.

If the test has not previously been converted to the format used in version 2.x of VibrationVIEW, a new file will be created with the same base file name, but with the filename extension changed according to the following table:

Test Type	Old	New
Sine	sin	vsp
Random	ran	vrp
Sine-on-Random	sor	vop
Shock	shk	vkp
FDR	drp	vfp

This command will not change the version 1.x format file, so that file may still be used with the older software. Also, this command will not overwrite any files. To force VibrationVIEW to overwrite a test file, load the file into VibrationVIEW, then use the Test..Edit Test menu command to edit the test and click the "Save As..." button in the Edit Test dialog box.

20.1.12. Old Data command (File menu)

Use this command to read data sets created using version 1.x of the VibrationVIEW software. An Open File dialog box will be presented, allowing selection of the data set to read.

20.1.13. Create Report Command (File Menu)

This command will create 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..Create Report 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

20.1.14. 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

20.1.15. 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

20.1.16. Printer Setup..command (File menu)

The following options allow you to select 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 the Specific Printer option and select one of the current installed printers shown in the dialog box. 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.

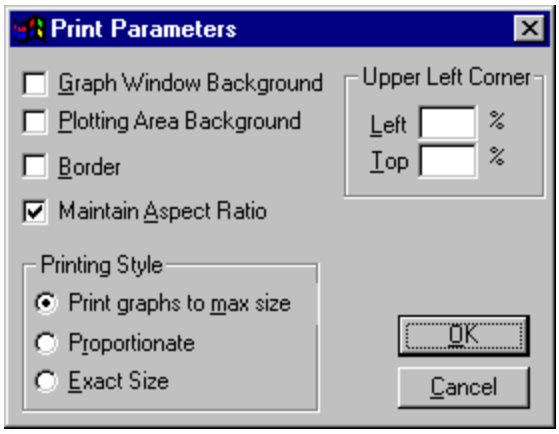
Paper Size
Select the size of the paper on which the document is to be printed.

Paper Source
Some printers offer multiple trays for different paper sources. Specify the tray here.

Orientation
Select Portrait or Landscape.

20.1.17. 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, a graph is printed with a rectangular border around it. 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
Proportionate
(These two options are equivalent for graphs printed by VibrationVIEW.)
Print the graphs with the maximal size that a paper sheet 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.

20.1.18. Explorer Command (File Menu)

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

20.1.19. 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

20.2. Configuration menu commands

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

Accelerometer Sensitivity	Enter the accelerometer mV/g levels
Units	Select units to use in the program
Alternate Units	Define and Enable Alternate Unit display
System Limits	Enter the physical limits of your shaker
System Mass	Enter the masses for system limitations
Remote Inputs	Configure the Remote Input options
Parameters	General system configuration parameters
Calibration	Used for the instrument calibration procedure

20.2.1. Accelerometer Sensitivity command (Configuration menu)

This command brings up the Accelerometer Sensitivity tab on the VibrationVIEW Configuration dialog box.

Use this command to enter the sensitivity values associated with each of your accelerometers. You should use the sensitivity values assigned by the calibration laboratory.

20.2.2. Units command (Configuration menu)

This command brings up the System Units tab on the VibrationVIEW Configuration dialog box.

Use this command to select and define the desired units to be used by the program.

20.2.3. Alternate Units command (Configuration menu)

This command brings up the Alt Units tab on the VibrationVIEW Configuration dialog box.

Use this command to define and enable the alternate units for each input channel.

20.2.4. System Limits command (Configuration menu)

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

Use this command to enter the respective shaker system limitations. These parameters are checked at run time to determine if you are able to run a test.

20.2.5. System Mass command (Configuration menu)

This command brings up the System Mass tab on the VibrationVIEW Configuration dialog box.

Use this command 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 settings

to determine whether the test can be run within the specifications of your shaker system.

20.2.6. Remote Inputs command (Configuration menu)

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

Use this command to enable and configure the Remote Input options.

20.2.7. Parameters command (Configuration menu)

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

Use this command to enter general controller parameters such as I/O hardware and serial port number.

20.2.8. System Calibration command (Configuration menu)

This command brings up the System Calibration tab on the VibrationVIEW Configuration box.

This command is used to calibrate the input scaling factors and offsets, the output scaling factor.

20.3. Test menu commands

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

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 and run a list of tests to run in sequence.
New Test	Create a new test profile.
Open Test	Open an existing test profile.
Open Recent Test	Submenu displays recently used test
Edit Test	Edit the current test.
System Check	Switch to System Check mode.
Sine	Switch to Sine mode.
Random	Switch to Random mode.
Sine on Random	Switch to Sine-on-Random mode.
Classical Shock	Switch to Classic Shock mode.
Field Data Replicator	Switch to Field Data Replicator mode.
Reset Filter Values	Reset the FDR filter values.
Random Profile from Ch2	Create a random spectum 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.
Test DSP Memory	Checks operation of the DSP card.

20.3.1. Select Test Type command (Test menu)

Use this command to select type of test you will be running. The VibrationVIEW Test Type dialog box will appear, allowing selection of the test type. 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: 

20.3.2. Run Test command (Test menu)

Use this command to start the current test. See also: Stop Test

Shortcuts

Toolbar: 

Controls: 

Keys: F9

20.3.3. Stop Test command (Test menu)

Use this command to stop any running test. See also: Run Test

Shortcuts

Toolbar: 

Controls: 

Keys: F10

20.3.4. 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 dialogs.

Shortcuts

Keys: F11

20.3.5. 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.)

20.3.6. New Test command (Test 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 then be changed as required by your test.

If your new test is substantially similar to another test you have already defined, open a 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. Use the "Save As..." button in that dialog box to save the test under a new name.

Shortcuts

Toolbar: 

20.3.7. Open Test command (Test menu)

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

Shortcuts

Toolbar: 

20.3.8. Open Recent Test command (Test menu)

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

20.3.9. Edit Test command (Test 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.

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

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

Shortcuts

Toolbar: 

20.3.10. System Check command (Test menu)

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

20.3.11. Sine command (Test menu)

Use this command to switch directly to the Sine test mode.

20.3.12. Random command (Test menu)

Use this command to switch directly to the Random test mode.

20.3.13. Sine on Random command (Test menu)

Use this command to switch directly to the Sine-on-Random test mode.

20.3.14. Classical Shock command (Test menu)

Use this command to switch directly to the Classical Shock test mode.

20.3.15. Field Data Replicator command (Test menu)

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

20.3.16. 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.

20.3.17. Random Profile from Ch2 (Test menu)

This command creates a Random test spectrum from the Channel 2 input. To use this, 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 which must be programmed into your key. If you did not purchase this option, this command will be disabled in the menu.

20.3.18. 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 to use 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 just use it for the current session (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... button, and uncheck

"Enable memorized drive").
2. Run the test, and let it run 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 also will 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, Sine-on-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.

20.3.19. 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 different 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, and let it run 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 to selecting the Save Current Drive menu command to save a more accurate memorized drive signal.

20.3.20. Test DSP Memory command (Test menu)

Use this command to test the memory on the DSP card. This is a diagnostic tool, and is only available when the program is set to the System Check mode. You need to use this option only if instructed to do so by Vibration Research Corporation.

20.4. 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 the amplitude axis of the selected graph.
Full Autoscale	Autoscale both axes of the selected graph.
Refresh Graph	Redraw the selected graph.
Reset Sine Hi/Lo Data	Reset the Hi/Lo data for the current sine test.
Sine Big Display	Display the sine frequency and amplitude information with large numbers.
Sine COLA Slip	Adjust the sine COLA slip frequency.
Cursor Display	Show the cursor information dialog box.
Add Annotation	Add an annotation to the selected graph.
Remove Annotation	Click on an annotation and select this to remove it.
Move Annotation	Click on an annotation and select this to change its position.

20.4.1. Graph Colors command (Graph menu)

Use this command to bring up the Graph Colors dialog box that 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.

20.4.2. Graph Update Time command (Graph menu)

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

20.4.3. 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

20.4.4. 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.

See also:

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

Shortcuts

Toolbar: 

Keys: Ctrl+G

20.4.5. Y-Axis Autoscale command (Graph menu)

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

See also: Full Autoscale

Shortcuts

Keys: Ctrl+A

20.4.6. Full Autoscale command (Graph menu)

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

See also: Y-Axis Autoscale

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

20.4.7. Refresh Graph command (Graph menu)

Use this command to redraw the selected graph.

Shortcuts

Keys: F5

20.4.8. Reset Sine Hi/Lo Data command (Graph menu)

Use this command to clear the peak- and valley-hold data for a Sine graph. The data is reset 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.)

20.4.9. Sine Big Display command (Graph menu)

This command will open the Sine Big Display dialog box that displays the frequency and amplitude parameters in numbers large enough to be seen from across the room. 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

20.4.10. Sine COLA Slip command (Graph 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.

20.4.11. Cursor Display command (Graph menu)

Use this command to bring up 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

20.4.12. Add Annotation command (Graph 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 selection of data, 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.

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. Press the <Insert> key while holding down the <Shift> key to put an annotation at the location of the peak.

Holding down the <Ctrl> key snaps the cursor to the nearest valley. Press the <Insert> key while holding down the <Ctrl> key to put 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

20.4.13. Remove Annotation command (Graph 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

20.4.14. Move Annotation command (Graph 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

20.5. View menu commands

The View menu offers the following commands:

Toolbar	Show or hide the toolbar.
Status Bar	Show or hide the status bar.
Control buttons	Show or hide the active Control Center
Reset Control buttons	Reset the control window to the default location
ToolTips	Enable or disable display of tool tips in the toolbars

20.5.1. Toolbar command (View menu)

Use this command to show or hide the Toolbar that includes buttons for some of the most common commands in VibrationVIEW, such as the File..Open Data menu command. A check mark appears next to the menu item when the Toolbar is shown.

See Toolbar for help on using the toolbar.

20.5.2. Status Bar command (View menu)

Use this command to show or hide the Status Bar that describes the action to be executed by the selected menu item or depressed toolbar button and keyboard latch state. 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.

20.5.3. Control buttons command (View menu)

Use this command to show or hide the active Control Center that contains buttons for starting and stopping tests, and also contains the current test measurements. A check mark appears next to the menu item when the Control Center is shown.

See the following for help on the various Control Centers:

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

20.5.4. Reset Control buttons command (View menu)

Use this command to reset the size and position of the Control Center that contains buttons for starting and stopping tests, and also contains the current test measurements. Use this command if you accidentally move the Control Center to the edge of the screen and aren't able to select it to move it.

20.5.5. ToolTips command (View menu)

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

20.6. 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 views the same document.
Cascade	Arrange windows in an overlapped fashion.
Tile Horizontal	Arrange windows in non-overlapped tiles, horizontally.
Tile Vertical	Arrange windows in non-overlapped tiles, vertically.
Arrange Icons	Arrange icons of closed windows.
Window 1, 2, ...	Go to specified window (graph).

20.6.1. 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 containing 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.

20.6.2. Cascade command (Window menu)

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

20.6.3. Tile Horizontal command (Window menu)

Use this command to arrange multiple open windows in a non-overlapping fashion, with the preference being that the graphs be stacked one above the other in a horizontal fashion.

20.6.4. Tile Vertical command (Window menu)

Use this command to arrange multiple open windows in a non-overlapping fashion, with the preference being that the graphs be stacked one beside the other in a vertical fashion.

20.6.5. 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 because they will be underneath this document window.

20.6.6. 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 in front of the document name of the active window. Choose a document from this list to make its window active.

20.7. Help menu commands

The Help menu offers the following commands that provide you assistance with this application:

Help	Open the help file to the overview page
Contents and Index	Offer you an index to topics for which you can get help
What's This?	Use this to get context-sensitive help information.
About VibrationVIEW...	Display application information.

20.7.1. 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 whenever you want to return to the opening screen.

Shortcut

Toolbar: 
Keys: F1

20.7.2. 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.

20.7.3. 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. Then 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

20.7.4. About command (Help menu)

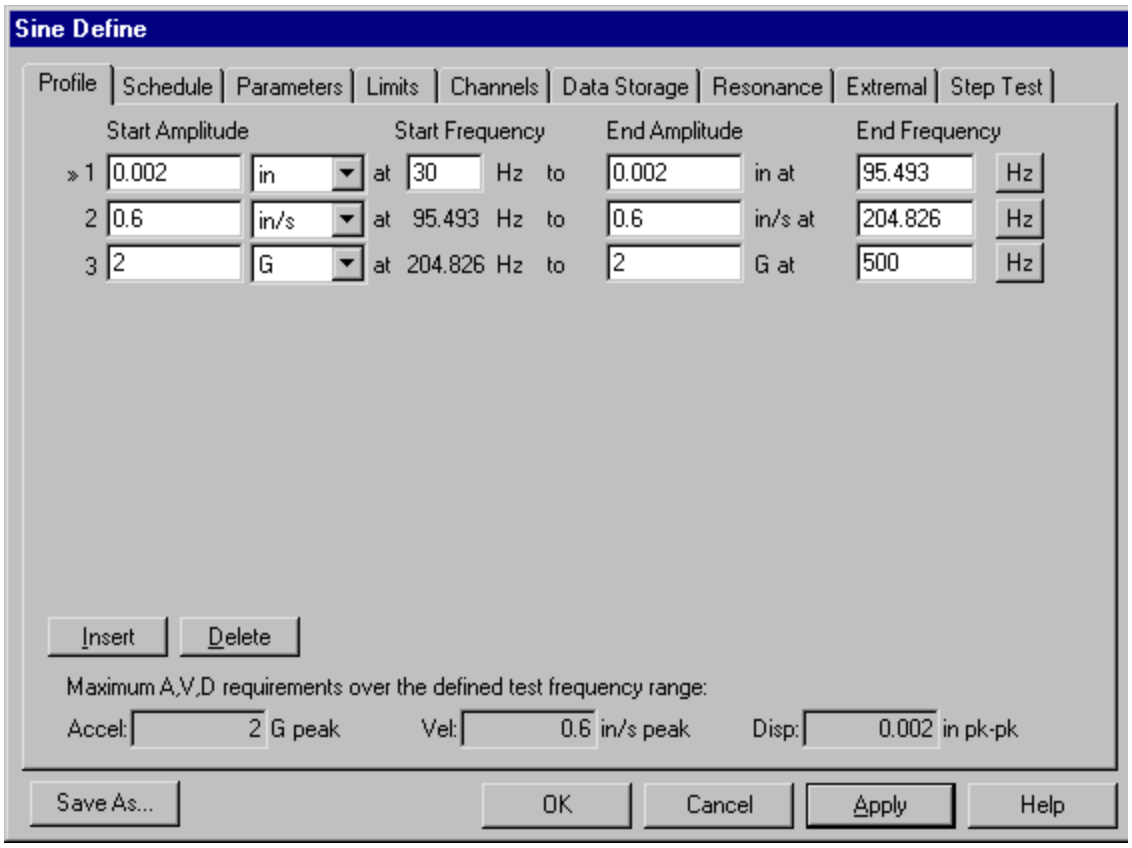
Use this command to display the copyright notice and version number of your copy of VibrationVIEW.

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

21. Dialog Boxes

21.1. Sine Define dialog box

This dialog box is accessed by selecting the Test..Edit Test menu command while a Sine test is open.



The *Start* and *End Frequencies* define the frequency span of the segment. The operator selects the first *Start Frequency*; the other start frequencies follow from the previous *End Frequency*. The amplitude parameters set the starting and ending points for each segment, and do not need to be constant for a given segment, nor do they need to be continuous at the segment endpoints.

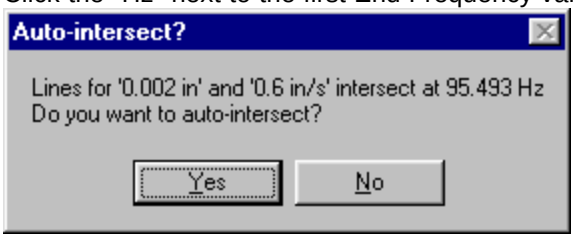
The *units* selected for the start amplitude will determine what type of control is used for that segment of the sweep. For example, if "in" (inches, in English units) is the unit selected, the parameter controlled will be displacement, if "in/sec" is selected, velocity will be controlled, if "G" is 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 "Hz" button next to the End Frequency that needs to be computed. For example, to find the intersection between 0.002 in displacement and 0.6 in/s velocity:

1. Enter 0.002 in the first Start Amplitude, and then select "in" for the units

2. Set the starting frequency (e.g. 30 Hz)
3. Select "in/s" for the units in the second line, and then enter 0.6 for the second End Amplitude.
4. Select second End Frequency (e.g. 200 Hz)
5. Click the "Hz" next to the first End Frequency value. 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 automatically set the two lines to constant displacement and constant velocity, respectively.

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

At the bottom of the dialog box there are maximum acceleration, velocity, and displacement ratings required to run the test. These boxes will be highlighted in red if the test exceeds the System Limits, and will be highlighted 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:

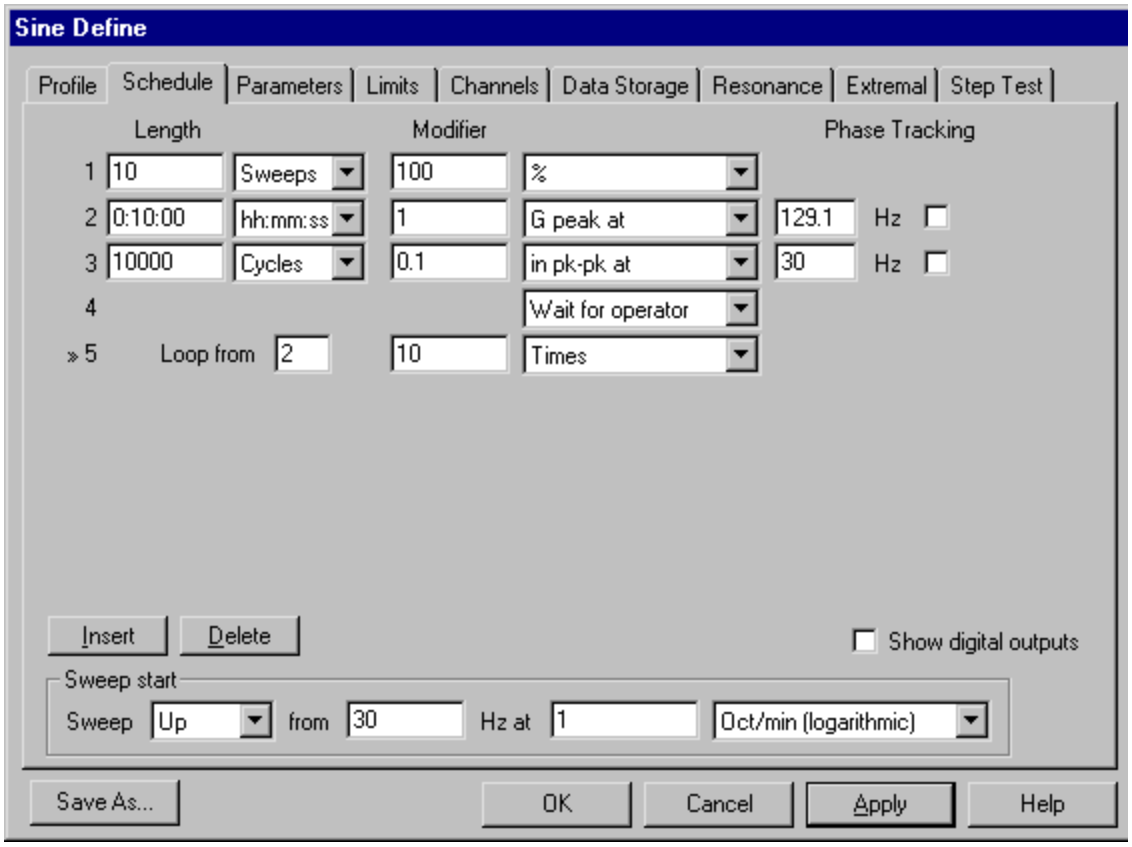
Profile	Enter the frequency and amplitude breakpoints.
Schedule	Set test duration and amplitude scaling level.
Parameters	Enter control loop parameters.
Limits	Enter the safety limits.
Control Channel	Select tracking filters and active control channel.
Data Store	Select automatic data storage parameters.
Resonance	Enable/disable the Resonance Search feature.
Extremal	Enter Extremal (extreme conditions, AVD) parameters. (optional)
Steps	Enter sine step test parameters. (optional)

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
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.

21.1.1. Sine Define Schedule tab

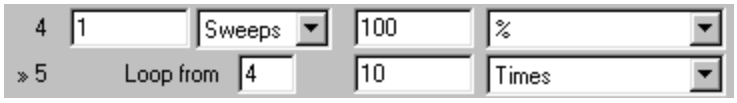
This dialog box is accessed by clicking the "Schedule" tab on the Sine Define dialog box.



Use this dialog box to set how long your test will sweep at each amplitude level, or how long the test will run at each fixed-frequency. Use the drop-down box to select type of duration measurement: number of sweeps, number of sine wave cycles, or the length of time. Then enter the duration parameter in the Length column.

The *Sweep Start* section 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 line in the dialog box shown above will run 10000 Cycles at 0.1 in pk-pk at a fixed frequency of 30 Hz).

The number of *sweeps* 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 to get 10 consecutive sweeps in the same direction:



The *hh:mm:ss* setting is the amount of time (using the format of hours, minutes, and seconds

separated by colons) for the indicated level number to run. When entering time parameters, you may use forms 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 number of *cycles* setting is 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 a before the test continues.
Resonance Table		Automatically detect the Ch2/Ch1 transmissibility peaks, and allow the operator to select time to test at each peak.

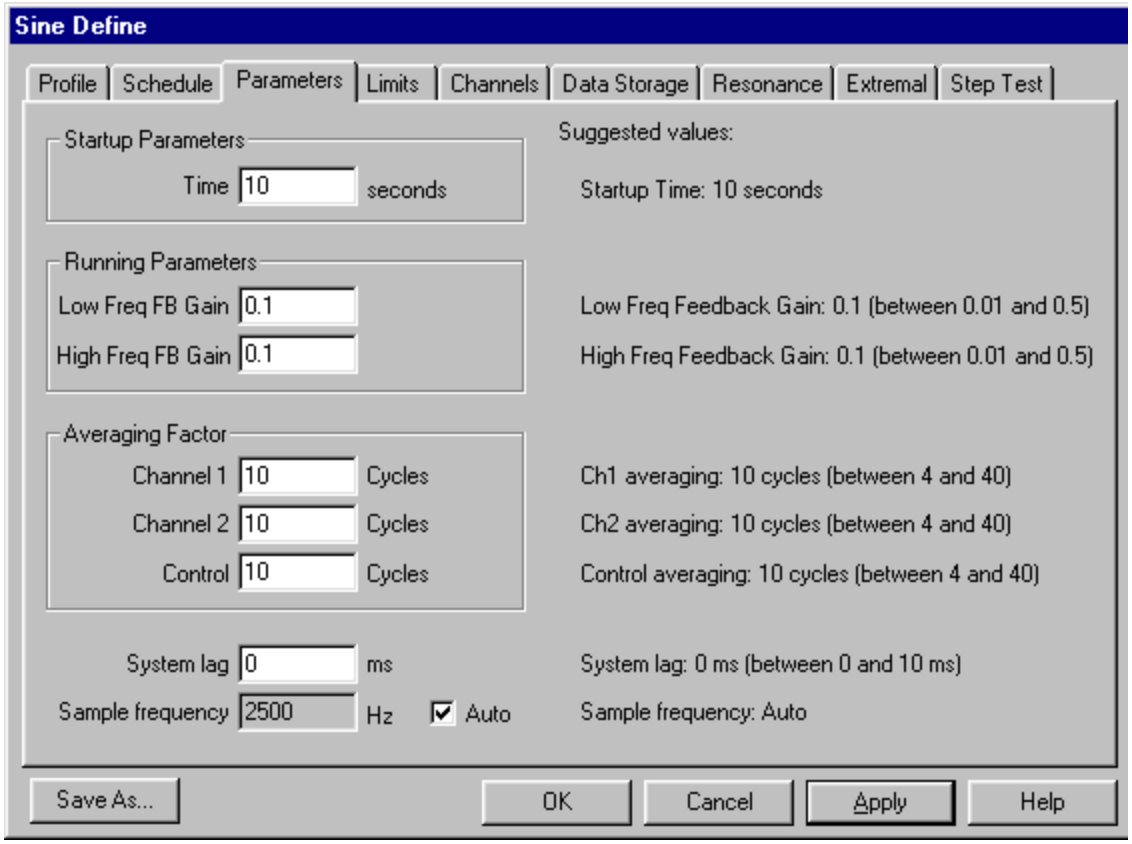
Note: Use of the Resonance Table or the Phase Tracking settings requires that the (optional) Sine Search-and-Dwell package is programmed into the software key at the factory. See the section on the Select Resonance Frequencies dialog box for more details on the use of these settings.

To repeat part of the test schedule a number of times, insert a level immediately following the section 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 deep to create more complex repeating patterns.

A loop pointing to a resonance table will repeat only the tones in that table; 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.

21.1.2. Sine Define Parameters tab

This dialog box is accessed by clicking the "Parameters" tab in the Sine Define dialog box.



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

The *low and high frequency feedback gain* values are the multiplier numbers by which the error is multiplied each control loop. The bigger the number 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 the control does not overshoot the intended level.

There are both *low and high frequency feedback gain* values. Normally these are the same but, under some conditions, you may need different feedback gain values at the lower and higher frequencies. Typically, both low and high frequency feedback gains should be 0.1. If the high frequency feedback gain value is too large, the controller will oscillate, causing a wow-wow sound. If it is too small, the control will lag behind the desired point. If the control is lagging too far behind the desired point at low frequencies, either increase the low frequency feedback gain, or decrease the control averaging cycles.

The *averaging factor* is the number of sine wave cycles averaged together to get the control data. The higher the number of cycles is, the more averages there will be. 10 averages is the default. When running at very low frequencies, the *control averaging factor* may need to be reduced (e.g. use 4 cycles) if the control signal lags behind the desired value.

The *system lag* parameter adjusts the controller to account for the time delay between a signal output on channel 1 and the resulting shaker response measured by the accelerometer. For most cases this value may be left at 0 ms (no delay) but for high-speed frequency sweeps, setting

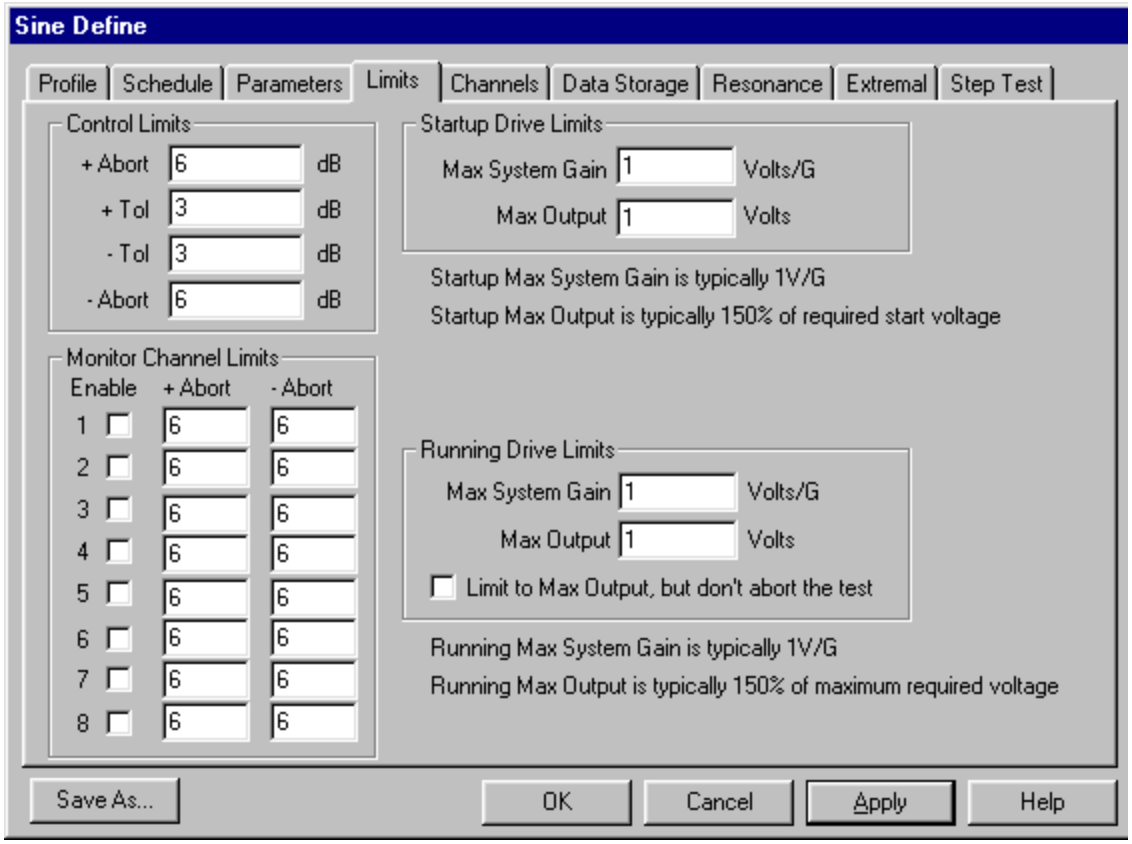
this value to the actual delay value may improve the controller response. The lag can be seen on a graph of the control signal phase (phase of the control signal relative to the channel 1 accelerometer signal). A lag will appear as control phase increasing with frequency -- when the system lag parameter matches the true system lag, the control signal phase should be nearly constant with respect to frequency.

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

See also: How to tune Sine controller parameters.

21.1.3. Sine Define Limits tab

This dialog box is accessed by clicking the "Limits" tab on the Sine Define dialog box.



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

The +/- *Tol* (dB) settings specify the number of decibels the tolerance lines will be above (+) and below (-) the desired acceleration level. These lines are typically drawn as yellow 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) settings specify 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* settings specify 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 drive limit parameter *Max Output Voltage* is 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.

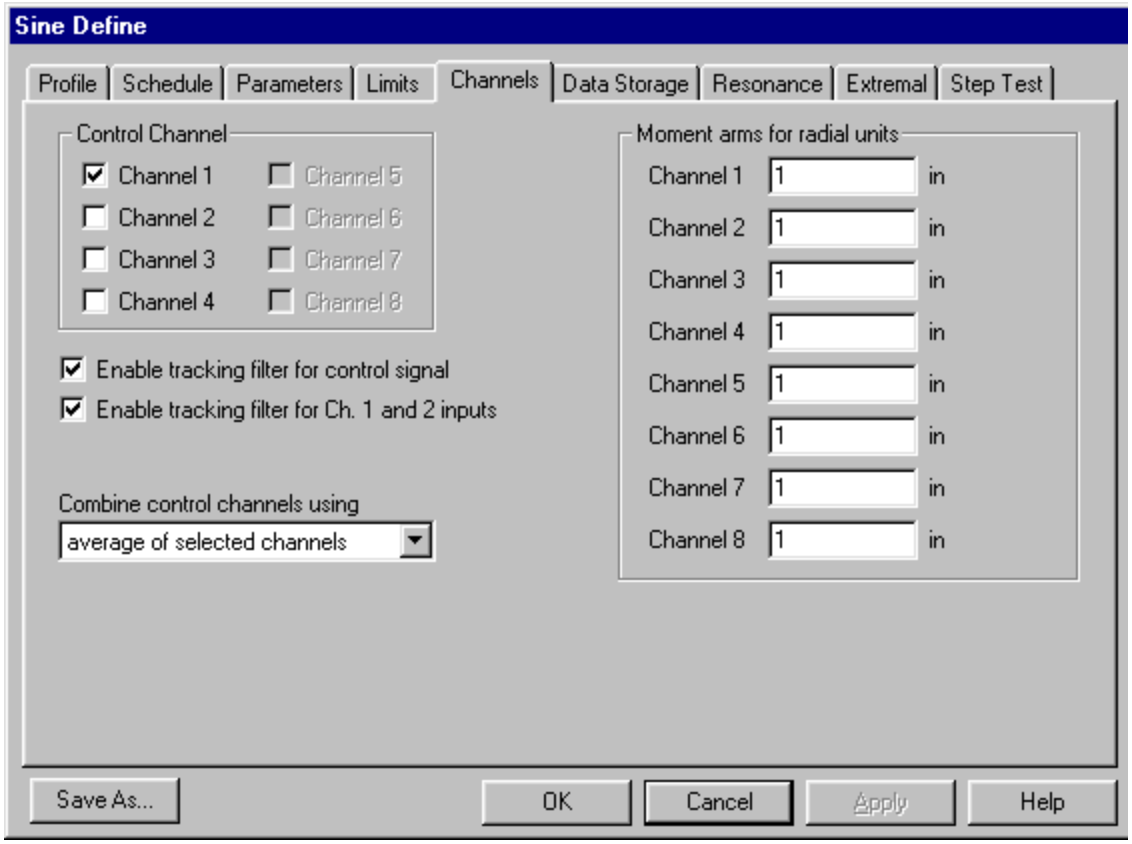
The startup drive limit parameter *Max System Gain* is the maximum allowed ratio of output volts to measured acceleration while a test is starting. Setting this value to a large number (e.g. 100 volts/G) disables this limit. This is a safety limit that restricts the drive output from increasing unless 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 drive limit parameter *Max Output Voltage* is 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.

The running drive limit parameter *Max System Gain* is 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.

21.1.4. Sine Define Channels tab

This dialog box is accessed by clicking the "Channels" tab in the Sine Define dialog box.



Use this dialog box to select 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 at least one of channels 1 and 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 which requires that the software key be programmed at the factory to enable this feature.

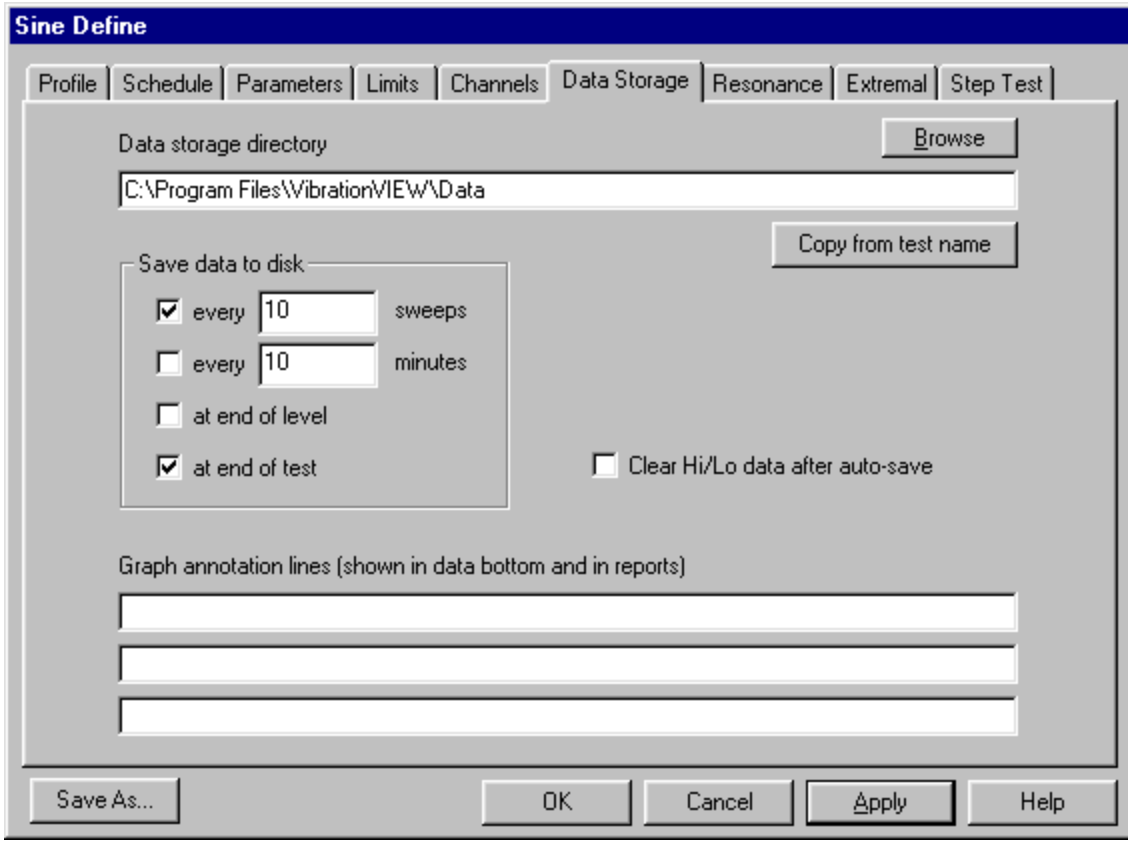
Note: a tracking filter is a filter that extracts only the frequency of interest from the measured signal. This is useful for controlling in low signal-to-noise environments, and also for removing the harmonics from the measured response (for example, if the shaker is producing a distorted sine wave).

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.

The *Moment arms for radial units* settings are used to convert linear values measured from accelerometers to angular measurements. The angular acceleration, in radians/sec^2, is defined as the acceleration measurement divided by the moment arm distance. Radial units can be used in the Transmissibility graphs to show radial transmissibility instead of linear transmissibility.

21.1.5. Sine Define Data Storage tab

This dialog box is accessed by clicking the "Data" tab on the Sine Define dialog box.



Use this dialog box to enable automatic data saving 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, 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 by selecting the Window..New window..Stored data graph display menu command.

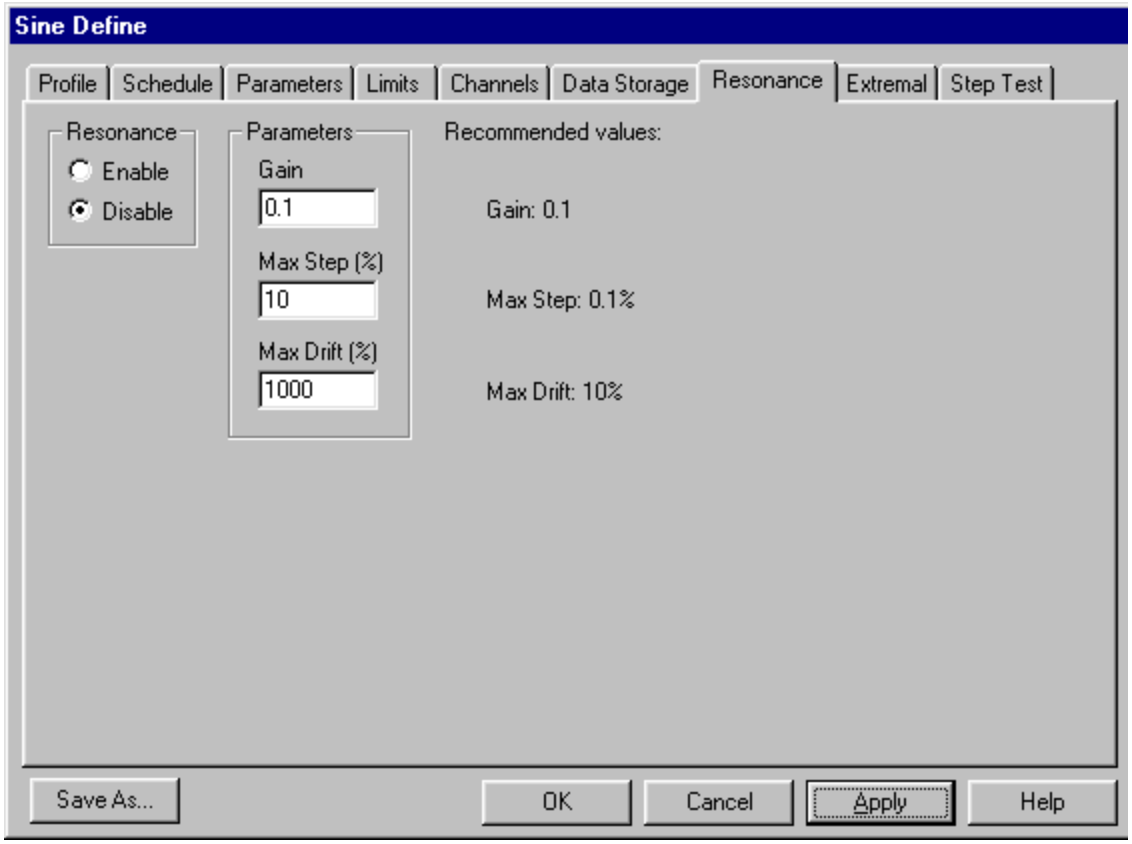
You may also select the directory into which to store the data. Click the "Copy from test name" button to automatically create an output directory based on the name of the current test.

The Hi/Lo peak hold values may also be automatically reset 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.

21.1.6. Sine Define Resonance tab

This dialog box is accessed by clicking the "Resonance" tab in the Sine Define dialog box.



This feature is optional and requires that the software key be programmed at the factory to allow it.

Sine Resonance control tracks a resonance frequency by adjusting the driving frequency such that the relative phase between the 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 second accelerometer mounted on the product at the location at which you want resonance induced.

The *Gain* value sets the speed at which the frequency will be adjusted when finding the resonance.
The *Dead Band* value sets the smallest desired frequency change. If the correction is smaller than this value, no change will be made to the driving frequency.
The *Max Drift* value sets 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, for example when the product breaks (and therefore the resonance frequency changes drastically.)
The *Max Step* value sets 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.

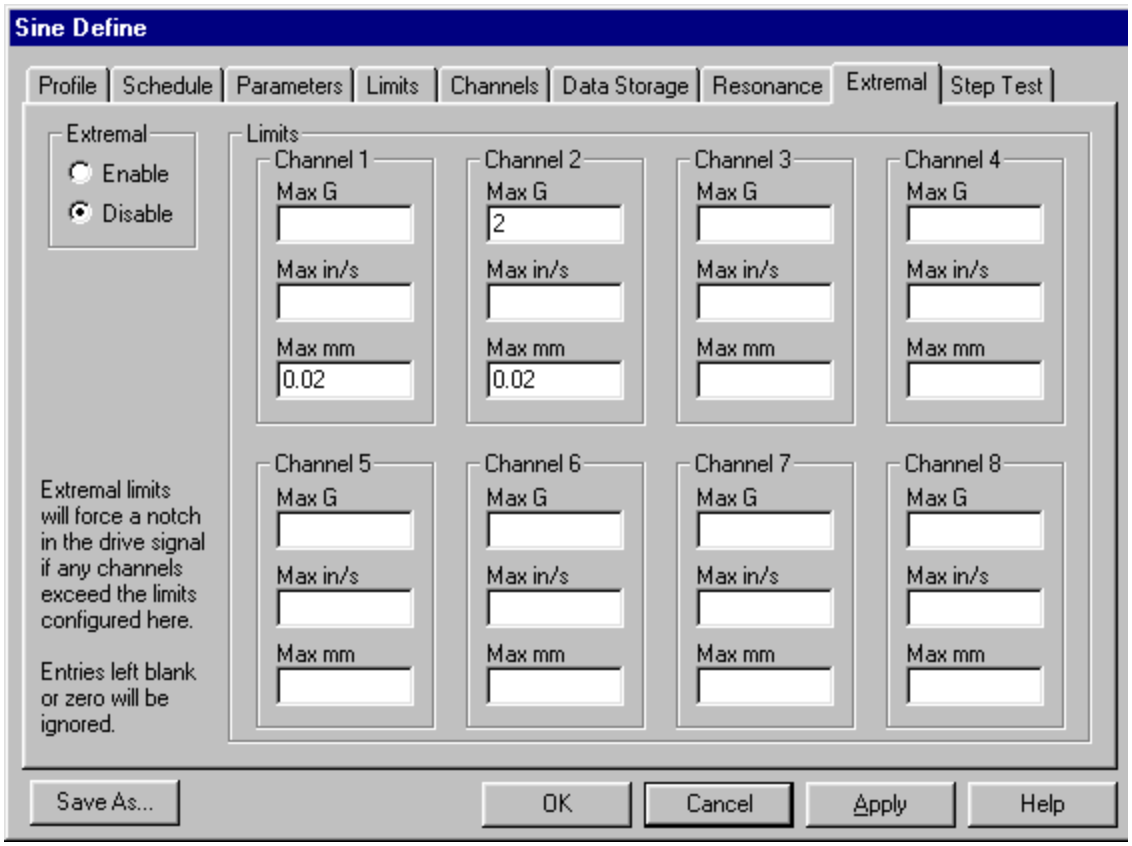
Recommended values are:

Gain: 0.1

Max Drift: 10%
Dead Band: 0%
Max Step: 0.1%

21.1.7. Sine Define Extremal tab

This dialog box is accessed by clicking the "Extremal" tab in the Sine Define dialog box.

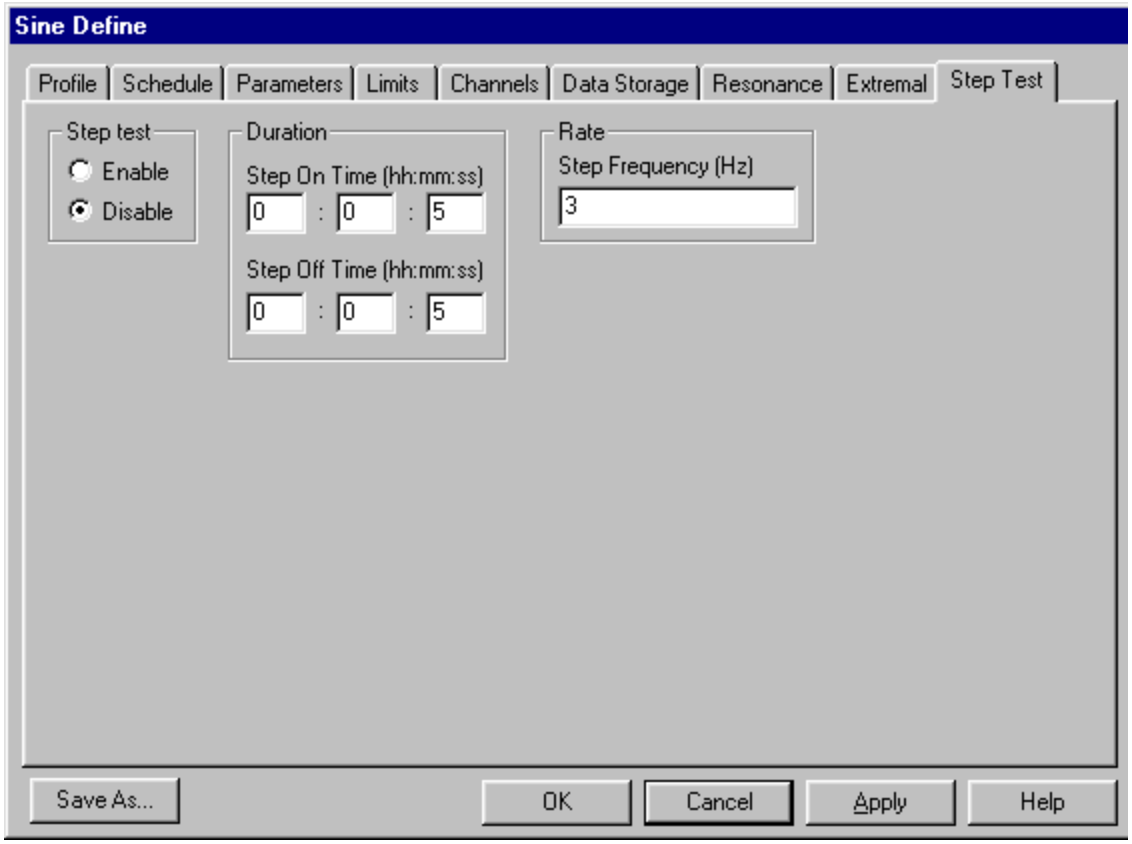


This feature is optional and requires that the software key be programmed at the factory to allow it.

Use this dialog box to set the maximum acceleration, velocity, and displacement parameter for each channel. If, while running, the controller exceeds these settings, 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. Entries left as blank or zero are ignored.

21.1.8. Sine Define Step Test tab

This dialog box is accessed by clicking the "Step Test" tab in the Sine Define dialog box.



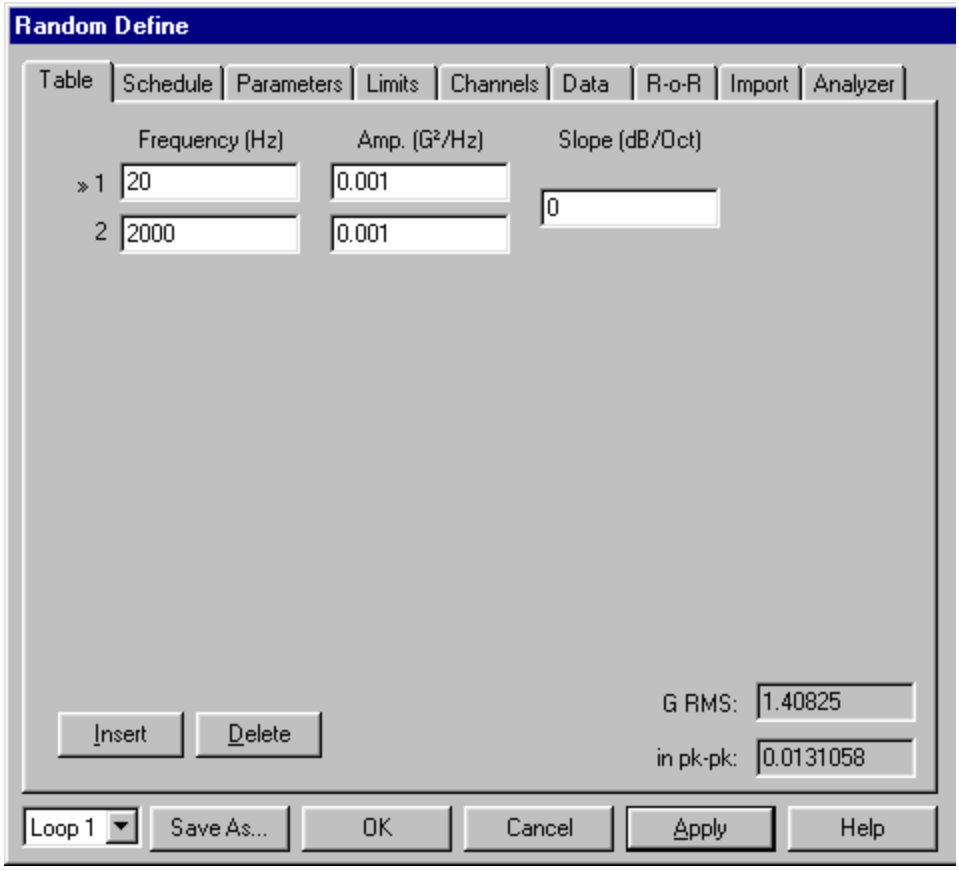
This feature is optional and requires that the software key be programmed at the factory to allow it.

Use this dialog box to configure a stepped-frequency (instead of continuously swept frequency) test. In this type of test, the sine test will run at a fixed frequency for the programmed "on" time. Then, it will shut off for the programmed "off" time. Next, the frequency will step up or down by the amount programmed into the step frequency. This will continue for the duration of the programmed sine test, sweeping with a series of discrete frequency values instead of the standard continuously varying sweep frequency. This is used for certain test specifications that require this type of operation.

Enter the frequency step interval to define how large the steps are, and enter the on and off times to determine how much time to spend at each frequency. Click the "Enable" button to turn the step function on, or the "Disable" button to turn it off.

21.2. Random Define dialog box

This dialog box is accessed by selecting the Test..Edit Test menu command while a Random test is open.



The frequency and amplitude values 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 box with the Slope. The amplitude box that will be calculated based on the frequencies and slope will be grayed out while you enter the slope value. After entering the slope value, 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 lines in the breakpoint table. The currently selected line (indicated by the small arrow on the left side of the dialog box) will be deleted when the "Delete" button is clicked. Inserted lines will immediately follow the currently selected line.

The RMS acceleration and required peak-to-peak displacement for a 100% level is indicated in the lower right corner of the dialog. The actual control level can be set by either a percentage modifier or a fixed G level, using the Schedule tab.

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

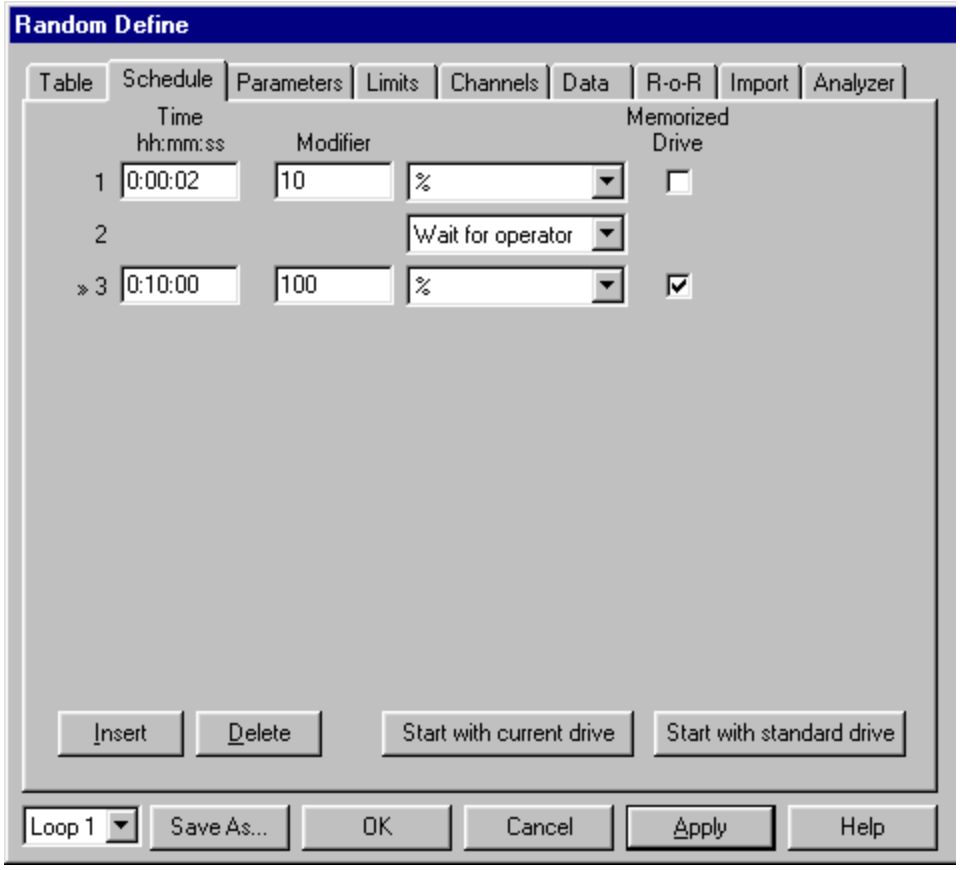
Table	Enter the frequency and amplitude breakpoints.
Schedule	Set test duration and amplitude scaling level.
Parameters	Enter control loop parameters.
Limits	Enter the safety limits.
Data Store	Select automatic data storage parameters.
R-o-R	Define Random-on-Random tones
Import	Import a spectrum from 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
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.

21.2.1. Random Define Schedule tab

This dialog box is accessed by clicking the "Schedule" tab in the Random Define dialog box.



Use this dialog box to set 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. 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 setting is 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 box allows you to change the test levels for each level. You can automatically modify the entire profile to match the desired RMS acceleration level, or scale the entered spectrum by a % or dB modifier. Enter the modifier value in the box, and select the appropriate modified type from the drop-down list box.

To repeat part of the test schedule a number of times, insert a level immediately following the section 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 clicked, 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 selected for the following level, the controller will ramp the output up slowly to the new demand level.

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 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 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 level (i.e. the test switches to Run mode with a green "Run" button).
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 (e.g. amplifier gain, shaker system) does not change.

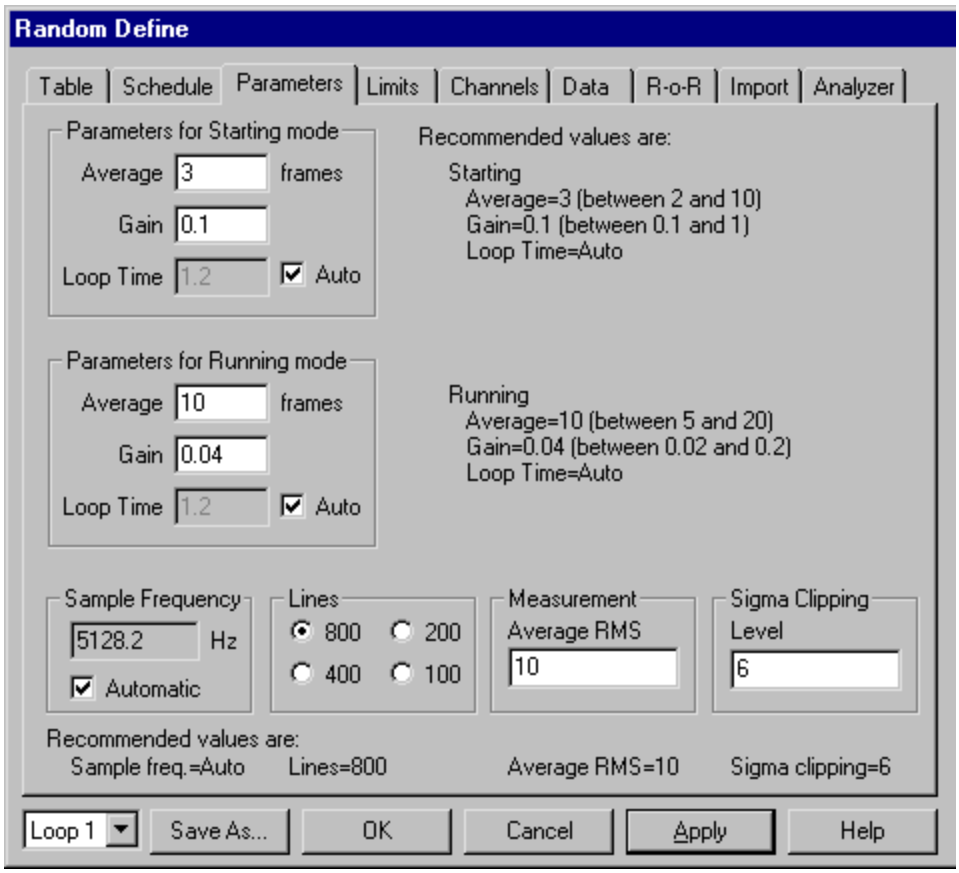
To disable the use of the memorized drive signal and revert to the slow test start-up, uncheck the "Enable memorized drive" box.

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.

21.2.2. Random Define Parameters tab

This dialog box is accessed by clicking the "Parameters" tab in the Random Define dialog box.



See also: How to tune Random controller parameters.

Use this dialog box to set 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 Loop Time is set to "Automatic".

The **Starting** control parameters 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 responsive to quickly equalize the output.

The **Running** control parameters 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 frames*** is the number of data frames averaged together to get the frequency data. The higher the number is, the smoother the line is but the slower the controller will respond.

The ***control gain*** is 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 ***loop time*** is the time in seconds between each control loop. Generally this value should be set to Auto to allow the controller to automatically select the appropriate loop time. Experienced users have the option of selecting the loop time for the controller.

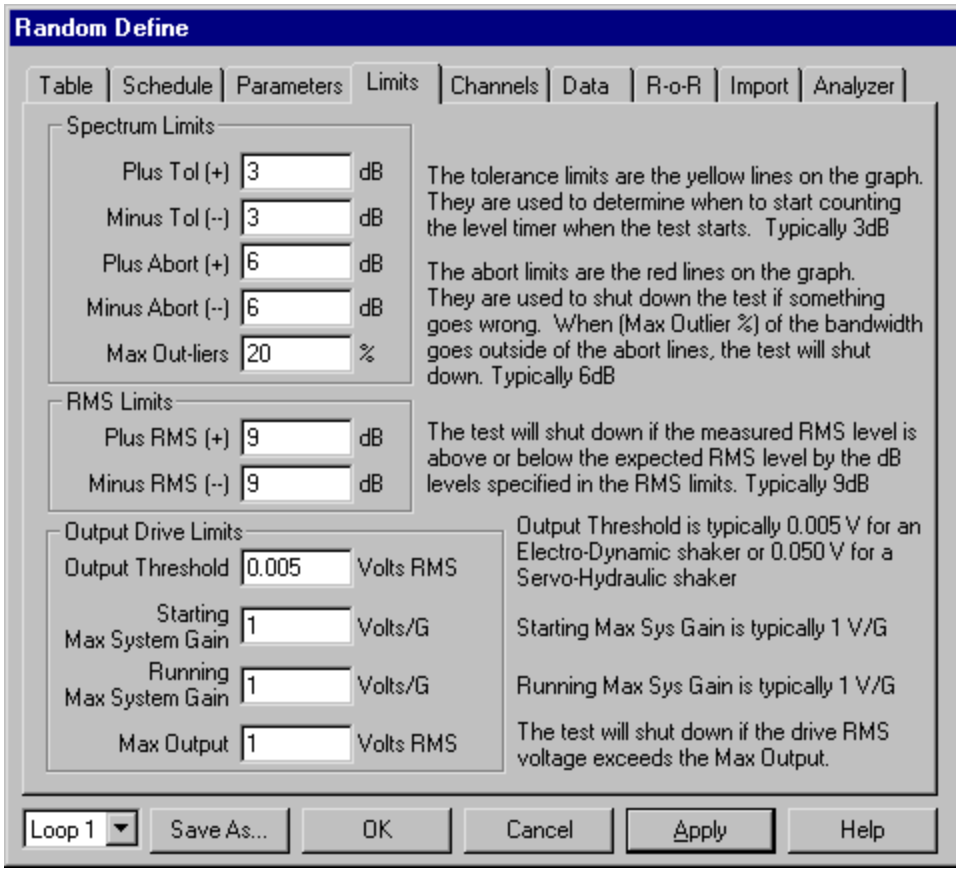
The ***measurement average RMS*** is the number of frames averaged together to arrive at the RMS number used in the display boxes under the Stop and Run buttons. By default, the number is 10. If more averaging is desired, for a smoother display, this number can be increased.

The ***sigma clipping level*** 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 also 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 frequency*** parameter should in nearly all cases be set to Auto to allow the controller to automatically select an appropriate sample frequency. However, experienced users have the option of entering the sample frequency used by the controller. If a sample frequency 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.

21.2.3. Random Define Limits tab

This dialog box is accessed by clicking the "Limits" tab in the Random Define dialog box.



Use this dialog box to set 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*** values set the maximum/minimum spectral density tolerance line levels indicated on the graphs. The tolerance lines are shown as dashed yellow lines on the graph, and are use to determine when the controller should switch from starting mode to running mode.

The ***+/- Abort*** values set the maximum/minimum spectral density abort line levels indicated on the graphs. The abort lines are shown as red lines on the graph. These levels 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 %*** is 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** values set 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*** settings 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 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 value is typically 0.005 Volts for an Electro-Dynamic shaker, and typically 0.050 Volts for a Servo-Hydraulic shaker. The Starting Max System Gain value depends on the shaker system and the amplifier gain setting. To determine this value, start the test, and after the demand levelnote both the "Volts rms" and the "Demand G RMS" readings in the Random Control Center. Set the Starting Max System Gain value to approximately:

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

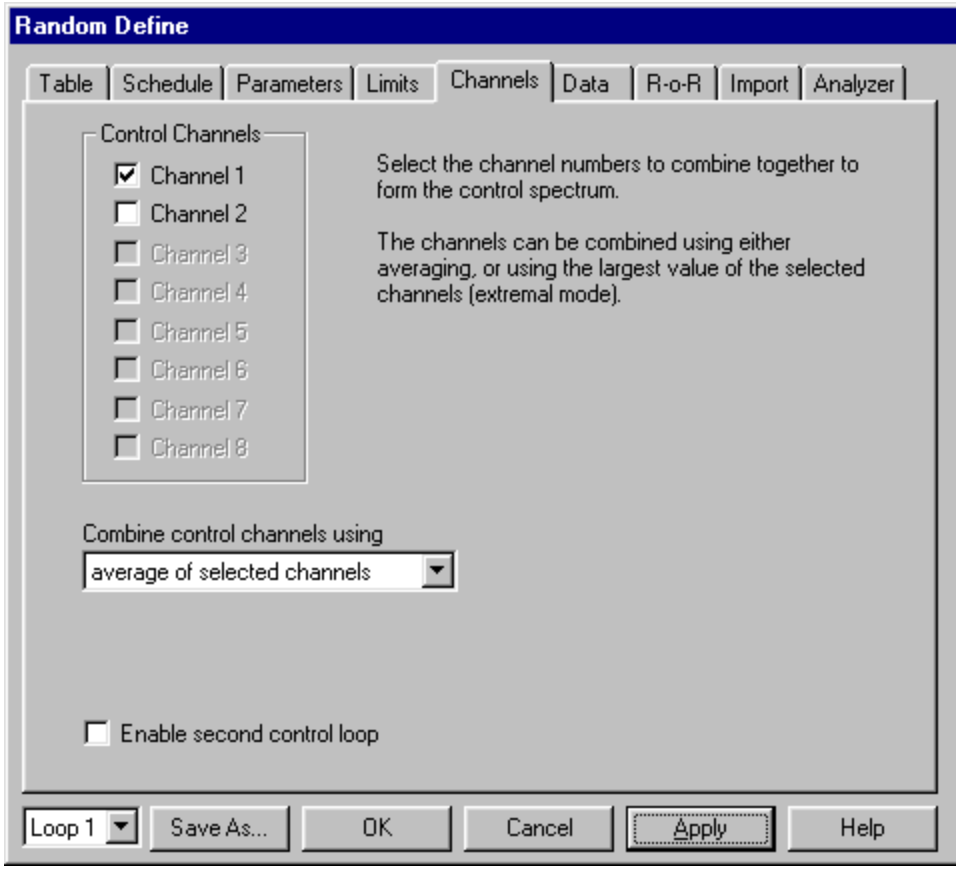
To disable this limit, set the Starting Max System Gain value to a large number (1000 V/G), or set the Output Threshold value to 10 Volts.

The ***Running Max System Gain*** setting 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 setting 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*** setting is 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.

21.2.4. Random Define Channels tab

This dialog box is accessed by clicking the "Channels" tab in the Random Define dialog box.

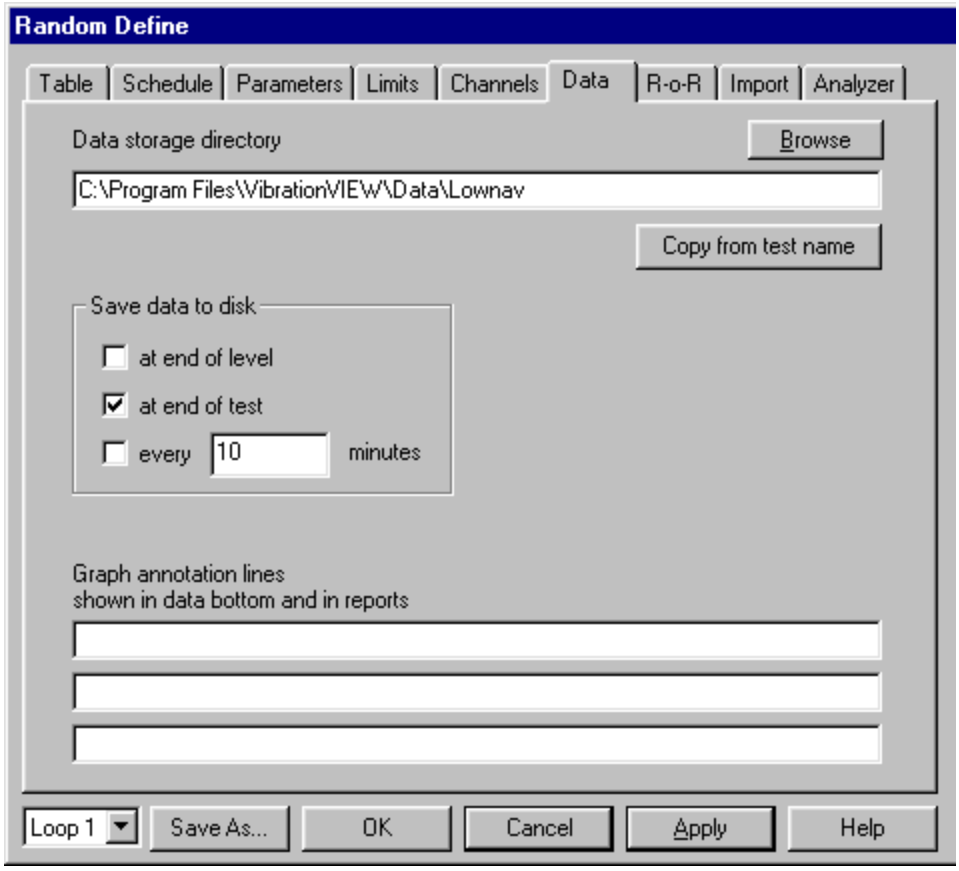


Use this dialog box to select the input channels to be used in the random control loop.

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 at the factory to enable it. If this feature is not enabled in your software key, the "combine control channels using" will always have the setting "average of selected channels".

21.2.5. Random Define Data Storage tab

This dialog box is accessed by clicking the "Data" tab in the Random Define dialog box.



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

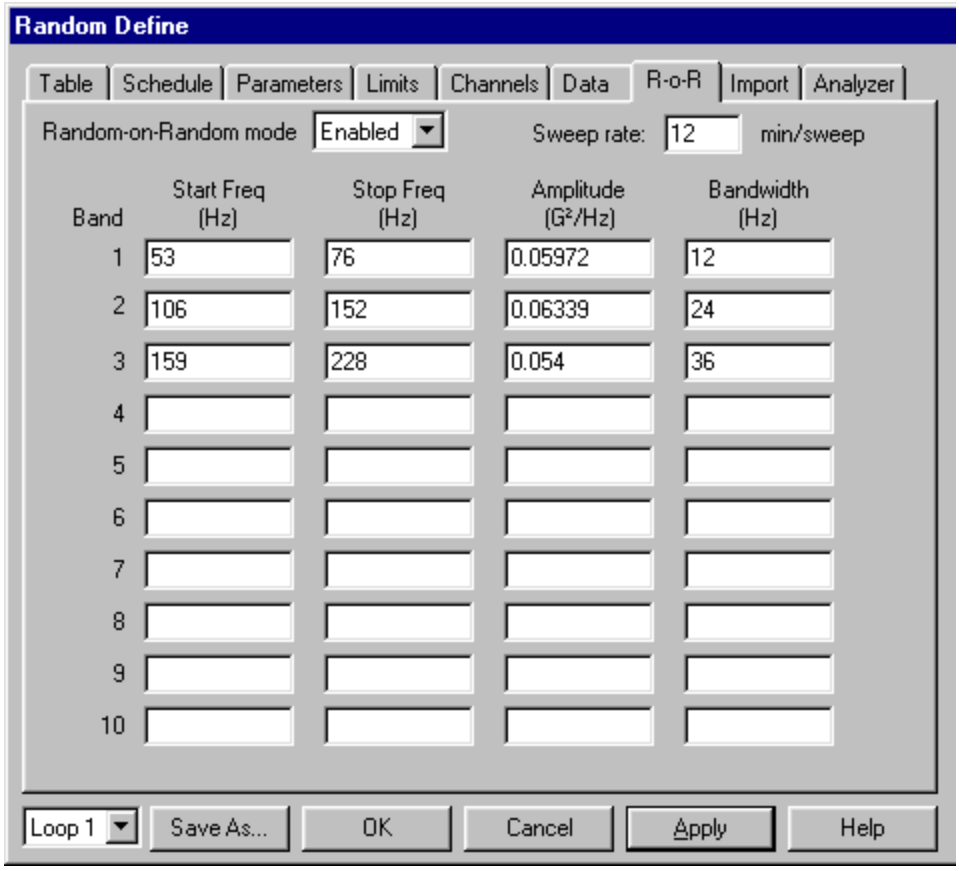
You can save the data automatically at the end of a level (as defined on the "Schedule" tab), at the end of a test, or at a regular time interval. Check the appropriate boxes to activate the automatic data store function.

You can also enter three lines that will appear in the Data Bottom portion of a graph. This display can be enabled for a graph by editing a graph's settings, and checking the "Data Bottom" check box in the Random Graph Settings dialog box.

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.

21.2.6. Random Define R-o-R tab

This dialog box is accessed by clicking the "RoR" tab in the Random Define dialog box.

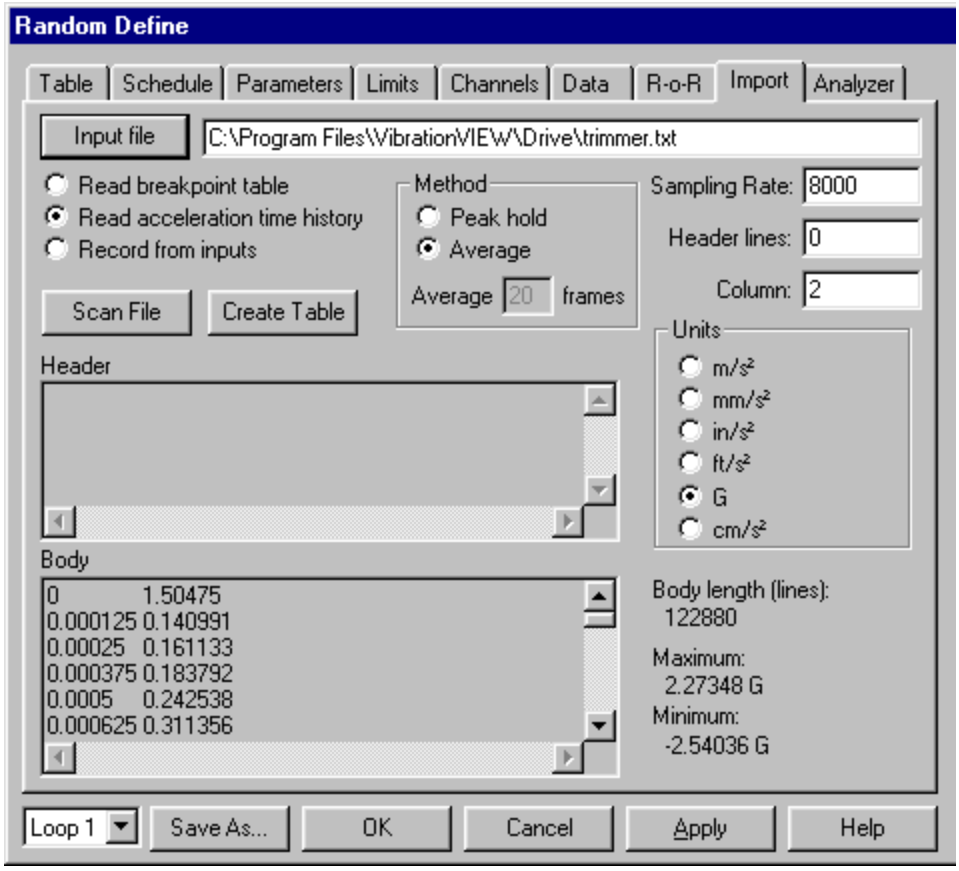


Use this dialog box to define up to 10 narrow-band random tones to add to the Random profile. The defined tones will sweep from the start frequency to the end frequency in the selected number of minutes. After the end frequency is reached, the sweep will continue, sweeping from the end frequency back 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 and requires that the software key be programmed at the factory to enable it.

21.2.7. Random Define Import tab

This dialog box is accessed by clicking the "Import" tab in the Random Define 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 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.

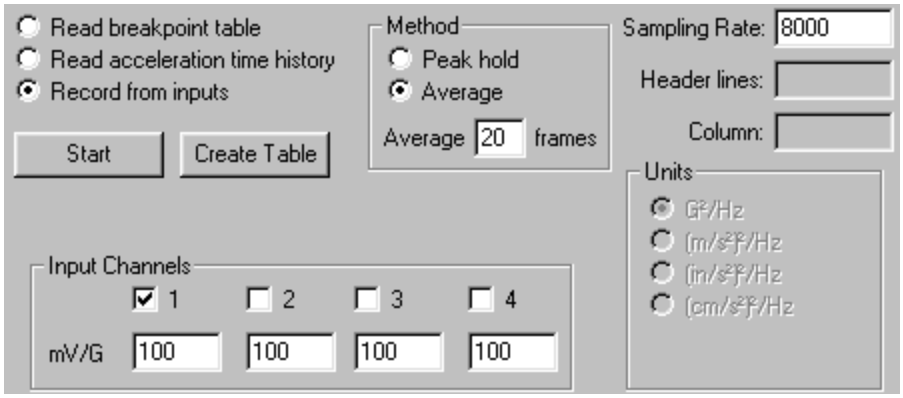
To input from an analog input channel on the VibrationVIEW I/O unit, perform the following procedure:

1. Select "Record from inputs" radio button, as shown in the figure below.
2. Select spectrum 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" setting. 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.

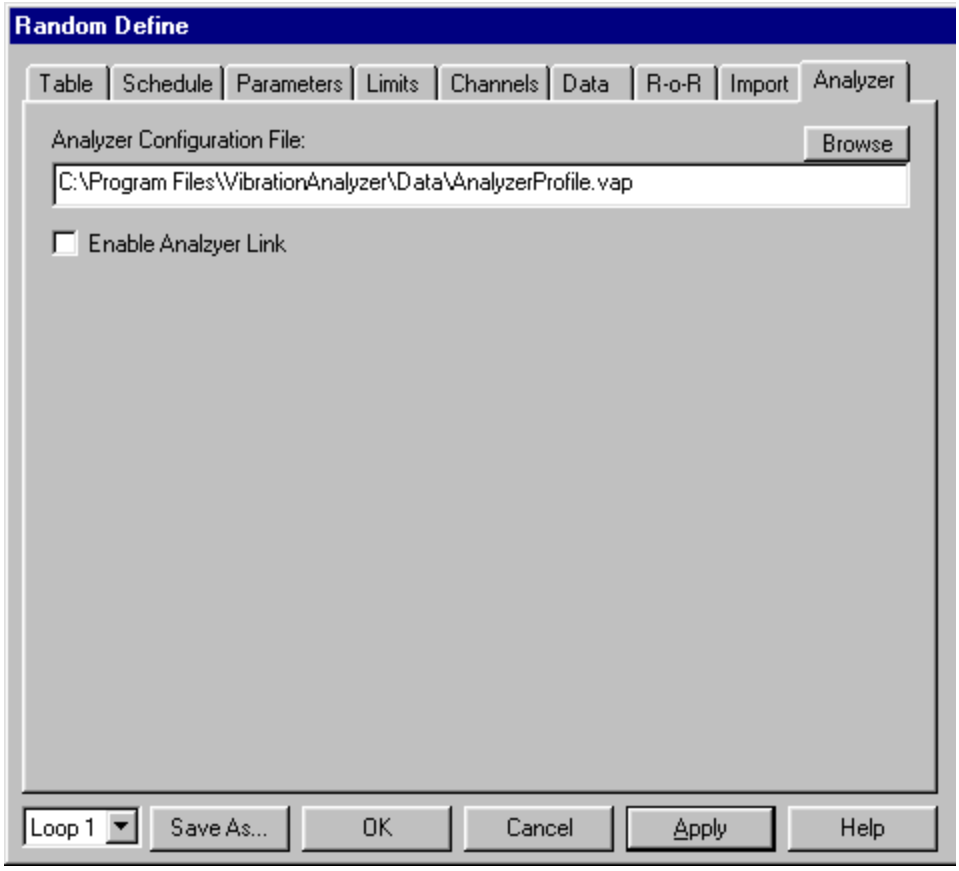
Figure: Generate a random test profile table from the analog input signals.



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

21.2.8. Random Define Analyzer tab

This dialog box is accessed by selecting the "Analyzer" tab in the Random Define dialog box.

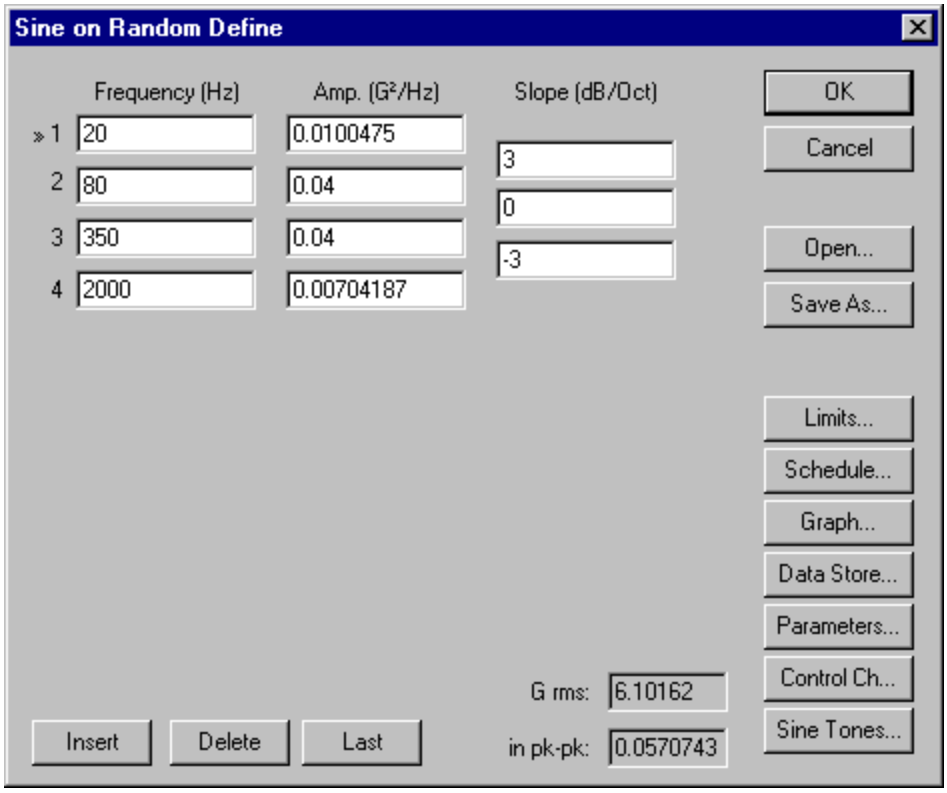


Use this dialog box to configure the link between VibrationVIEW and VibrationAnalyzer. VibrationAnalyzer is a separate application available from Vibration Research Corporation that allows monitoring of up to 16 channels with comparison to custom abort limits. If, while the test is running, VibrationAnalyzer 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.

21.3. Sine-on-Random Define dialog box

This dialog box is accessed through the Test..Edit Test command.



The following buttons are available:

Limits	Enter the safety limits here.
Schedule	Enter run time here.
Graph	Enter graph parameters here.
Data Store	Enter automatic data storage parameters here.
Parameters	Enter control parameters here.
Control Channels	Enter control channels here.
Sine Tones	Define to sine tone frequencies and amplitudes here.

The frequencies define the segment frequency span. Enter the numbers.

The amplitudes determine where the control will take place. Enter the desired amplitude.

The starting and ending amplitude do not need to be the same. If you want the computer to calculate a slope, enter the amplitudes around the slope. The computer will fill in the missing value.

The Insert, Delete and Last buttons allow editing of the test.

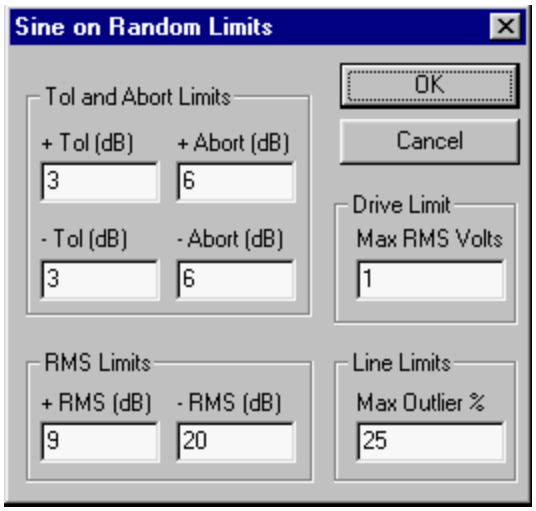
Important note: click the Last button to clear out segments after the currently active displayed interval. Use this button when you are entering your last interval.

The Open... button will open a previously saved test.

The "Save as..." button will save a test under the entered name.

21.3.1. Sine-on-Random Define Limits dialog box

This dialog box is accessed by clicking the "Limits..." button in the Sine-on-Random Define dialog box.



Use this dialog box to set critical limits in the sine-on-random software.

The +/- Tol (dB) for the channels sets the maximum/minimum G tolerance line levels indicated on the graphs for each channel.

The +/- Abort (dB) for the channels sets the maximum/minimum G abort line levels indicated on the graphs for each channel.

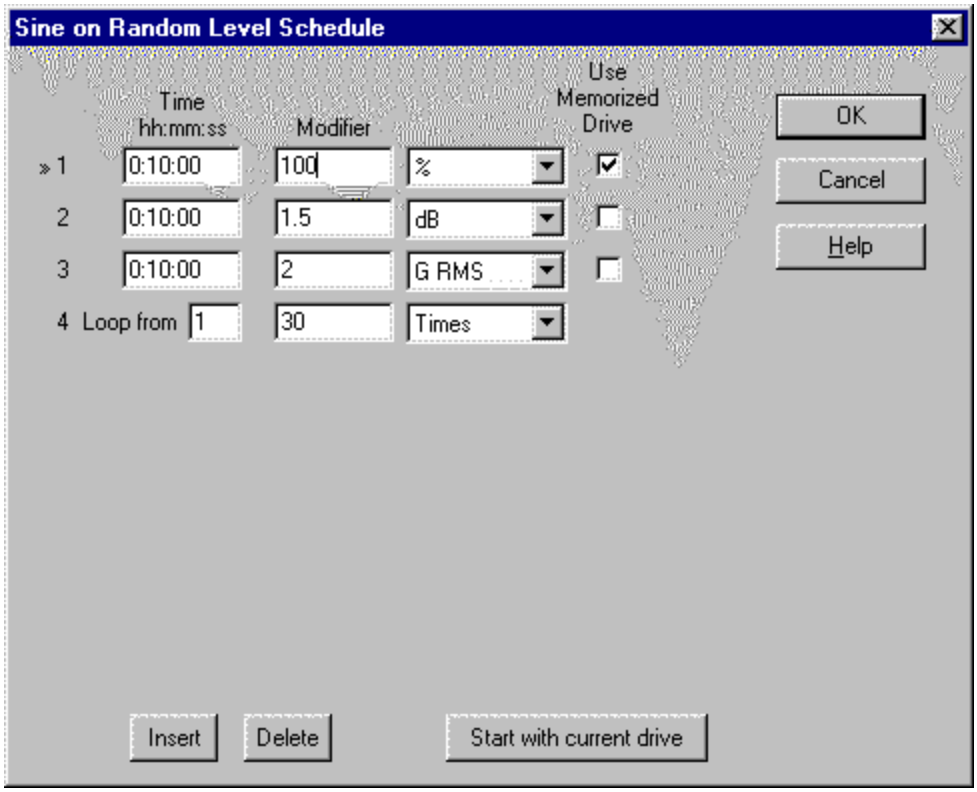
The +/- RMS (dB) for the channels sets the maximum/minimum G rms level allowed into each channel.

The Max Outlier % is the maximum percentage of the test bandwidth allowed to exceed the abort limits before a shut-down will occur.

The Drive Limit (Max RMS Volts) is the maximum 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 running maximum.

21.3.2. Sine-on-Random Define Schedule dialog box

This dialog box is accessed by clicking the "Schedule..." button in the Sine-on-Random Define dialog box.



Use this dialog box to determine how long your test will run. Up to 200 levels can be specified. Click the scroll bar arrows to review the selections.

The time setting is the duration (hours:minutes:seconds) for the indicated level number.

The modifier box allows you to change the test levels for each level number. You can automatically modify the entire profile to match the desired g rms value, or scale the entered spectrum by a % or dB modifier. Enter the modifier value in the box, and click the appropriate check box to select unit.

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

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

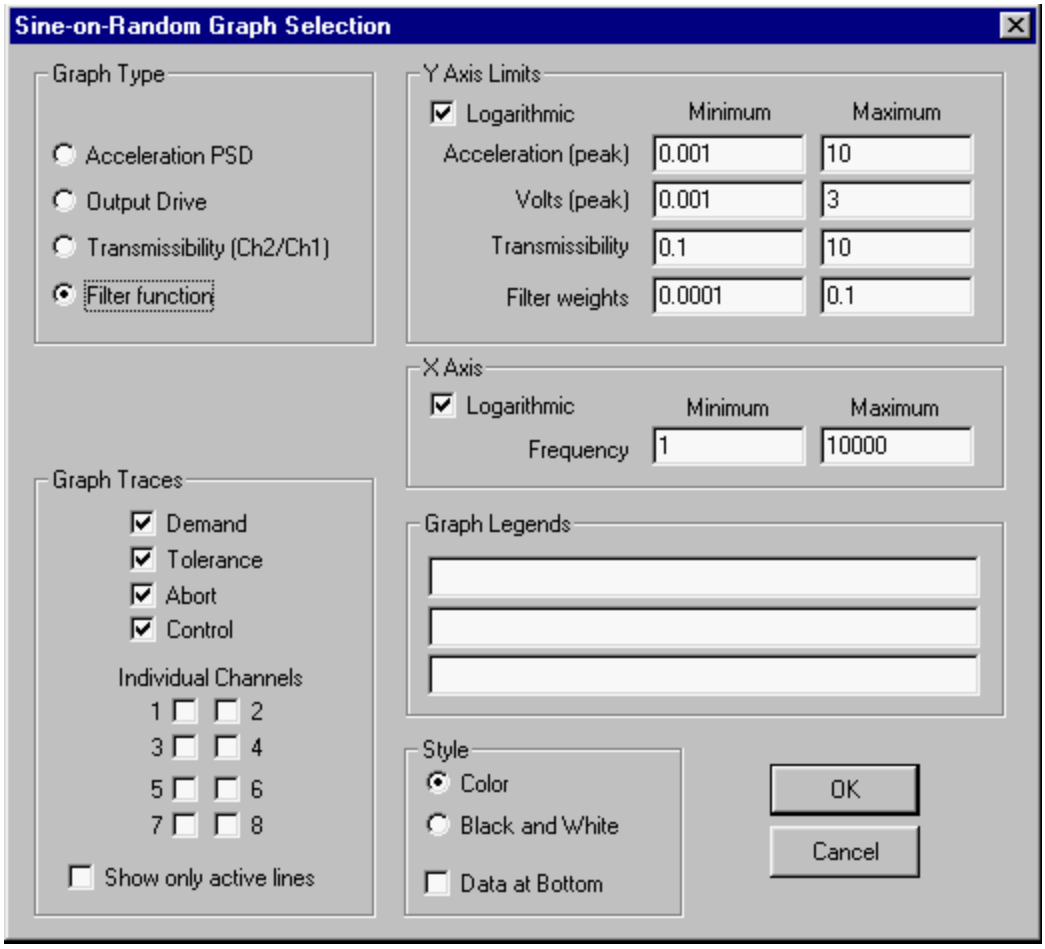
Note: the current drive signal can also be saved by selecting the Test..Save Current Drive menu command, without using the edit test dialog boxes.

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

21.3.3. Sine-on-Random Graph Settings dialog box



This dialog box is used when creating a new Sine-on-Random graph by selecting the File..New Graph or Window..New Window menu commands, when changing the settings of the selected graph by selecting the Graph..Edit Graph Settings menu command, or when setting the default graph settings by clicking the "Graph" button in the Sine-on-Random Define dialog box.

21.3.4. Sine-on-Random Define Data Storage dialog box

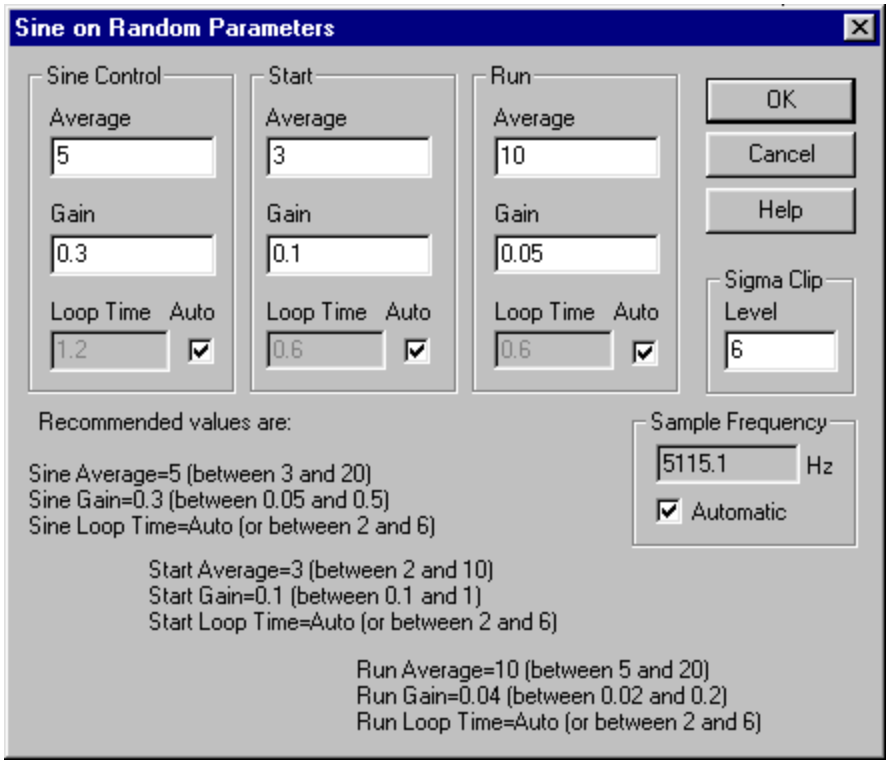
This dialog box is accessed by clicking the "Data..." button in the Sine-on-Random Define dialog box.

Use this dialog box to save data to disk as the test proceeds. Note that 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 Window..New window..Stored Data graph display menu command.

Check a box to activate the store function.

21.3.5. Sine-on-Random Define Parameters dialog box

This dialog box is accessed by clicking the "Parameters..." button in the Sine-on-Random Define dialog box.



Use this dialog box to set the parameters used by the Sine-on-Random control algorithm.

There are three sets of control parameters. The first set (Sine Control) is used to control the sine tones. By default, the averaging is set to 5, the loop time to 4, and the gain to 0.3.

The second set (Start) is used to get the random controller running. By default, the average is 3, the gain is 0.1, and the loop time is (6000/SampleRate) seconds. Typically you would use a higher gain (relative to the Run values) to get the output up to level quickly. Also, you should use a low averaging value (e.g. 3 to 5) because the output will be changing significantly as the controller brings the output up to level. The starting mode is indicated by a Yellow (**Yellow**=normal start) or Blue (**Blue**=memorized drive) Run button in the Sine-on-Random Control Center.

The third set (Run) is used to define the parameters used when the random control loop is near the running level. By default, the average is 10, the gain is 0.05, and the loop time is (6000/SampleRate) seconds. Typically you would use a lower gain and higher averaging value here (relative to the Start values) so that the spectrum is smoothed through a lot of averaging. However, in the case of ***swept*** sine tests, this averaging value should be low (5 to 10) so that the sine tones do not interfere with the random control loop. The running mode is indicated by a green Run button in the Sine-on-Random Control Center.

The *control gain* is 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 the control does not overshoot the intended level. Typical values are between 0.1 and 1.0. Higher averaging values will require lower gain

values to avoid overshoot.

The *loop time* is the time in seconds between each control loop. Generally this value should be set to Auto to allow the controller to automatically select appropriate loop time. Experienced users have the option of entering the loop time for the controller.

The *average frames* is the number of data frames averaged together to get the frequency data. The higher the number of frames is, the smoother the line will be. This relates to degrees of freedom by doubling the number. Example: 50 averages is 100 degrees of freedom. 77 averages is 154 d.o.f.

The *measurement average RMS* is the number of frames averaged to arrive at the RMS number used in the display boxes under the Stop and Run buttons. By default, the number is 10. If more averaging is desired, for a smoother display, this number can be increased.

The *sigma clipping level* 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 also 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.

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

21.3.6. Sine-on-Random Define Control Channels dialog box

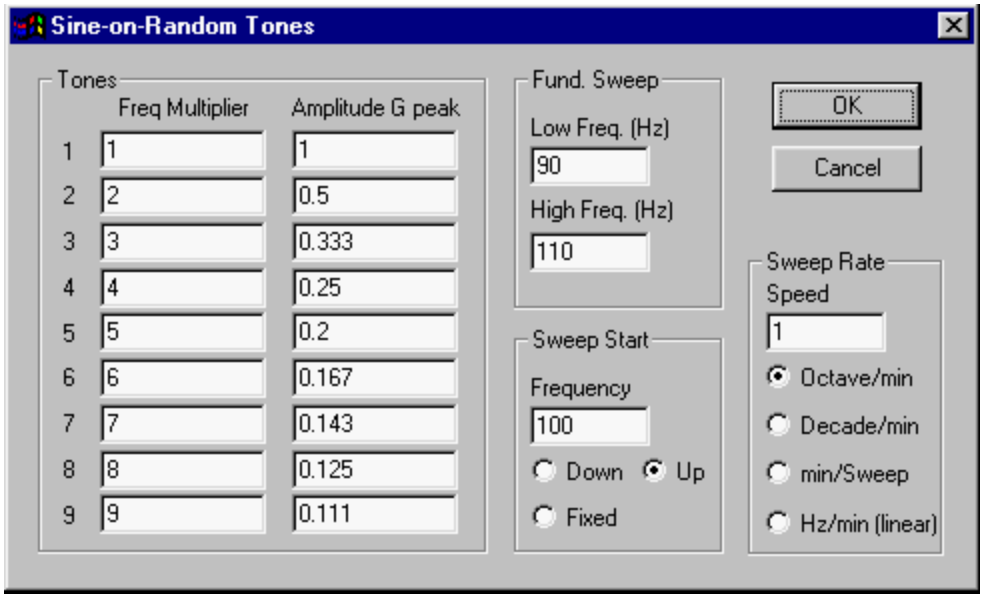
This dialog box is accessed by clicking the "Control Ch..." button in the Sine-on-Random Define dialog box.

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

If multiple channels are selected, they will be combined either by averaging the selected channels' spectral lines (Extremal=Disable) or by using the maximum value for each spectral line (Extremal=Enable)

21.3.7. Sine-on-Random Define Sine Tones dialog box

This dialog box is accessed by clicking the "Sine Tones" button in the Sine-on-Random Define dialog box.

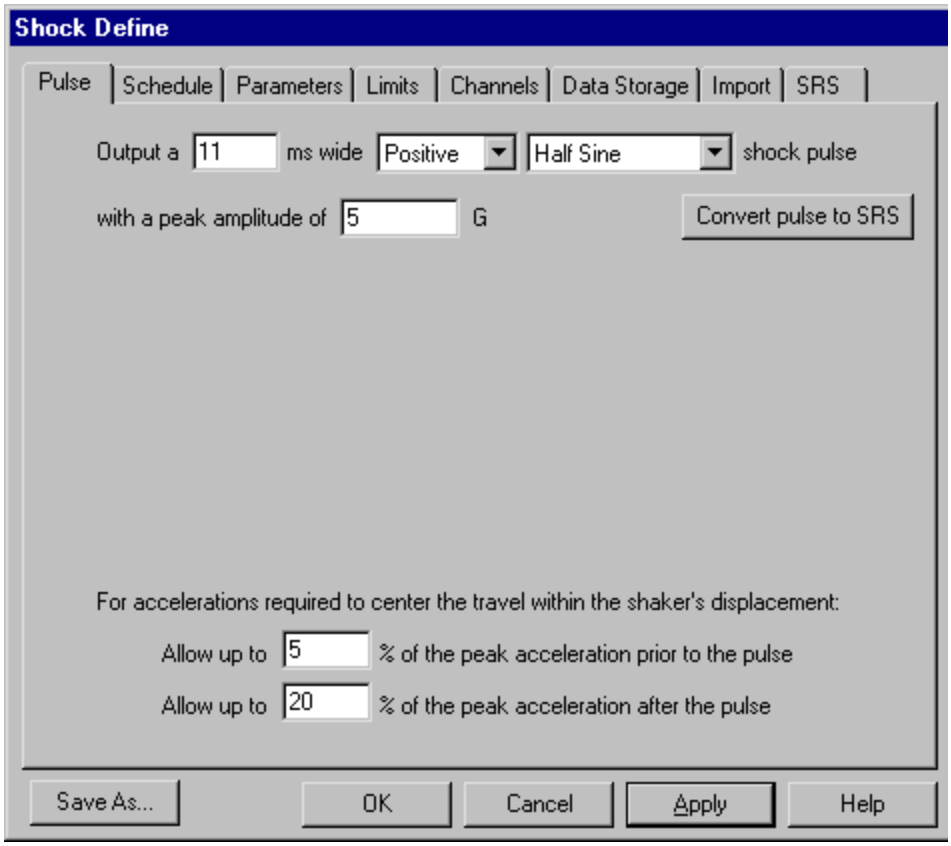


Use this dialog box to configure up to 9 simultaneous sine tones to add to the random signal. The Freq Multiplier column determines the multiple of the fundamental frequency for each of the tones (not necessarily an integer multiple). The Amplitude G peak column determines the output level for the tone. If either the Freq Multiplier or the Amplitude G peak values are zero, that tone will be disabled.

The test may be either a fixed tone, or sweep through a frequency range. For a fixed frequency test, enter the desired fundamental frequency in the "Sweep Start" group, and select the Fixed radio button. For a swept frequency test, select the frequency range in the "Fund. Sweep" group, the sweep speed in the "Sweep Rate" group, and the starting frequency and direction in the "Sweep Start" group.

21.4. Shock Define Pulse Dialog Box

This dialog box is accessed by selecting the Test..Edit Test menu command while a Shock test is open.



Use this dialog box to select the type of pulse to output. The pulse parameters are

Width	The duration of the pulse, in milliseconds
Polarity	Whether the pulse is in the Positive or Negative direction
Type	Selects the pulse shape. Available shapes are Half Sine, Initial Peak, Terminal Peak, Triangle, Trapezoid, Square, Haversine, and User-Defined.
Amplitude	Selects the peak acceleration value of the pulse.

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

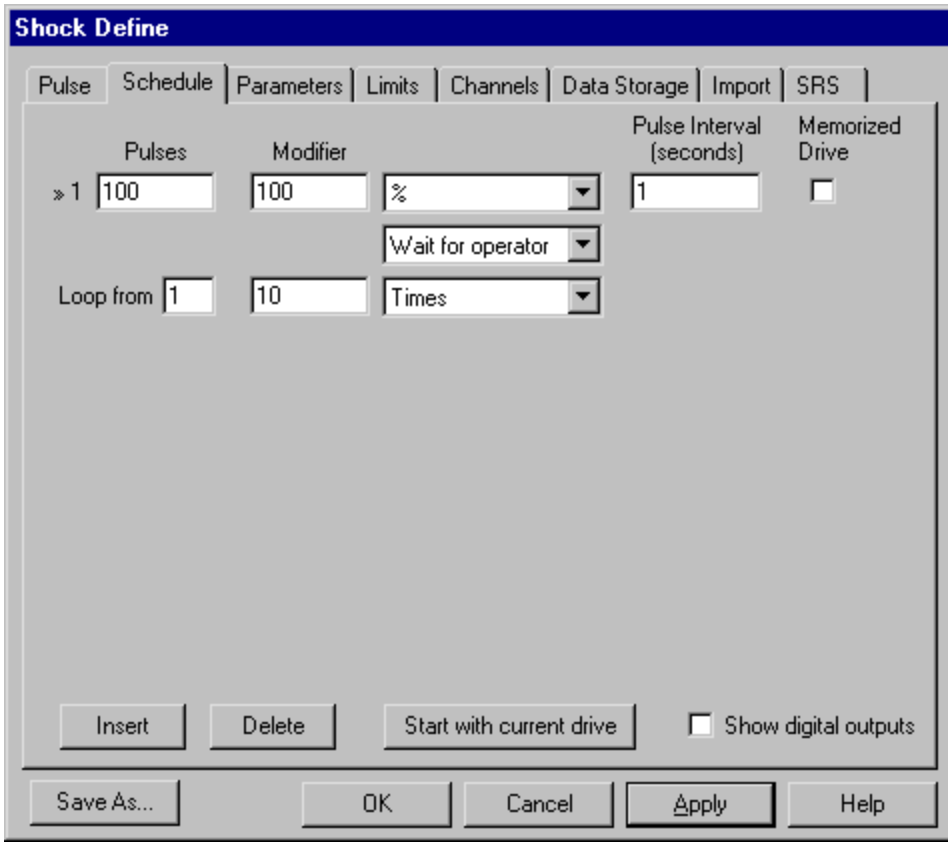
transfer those settings to the SRS tab.

The tabs along the top of the dialog box are used to select from 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

21.4.1. Shock Define Schedule tab

This dialog box is accessed by clicking the "Schedule" tab in the Shock Define dialog box.



Use this dialog box to set 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* column selects the duration of the segment in terms of the number of pulses.

The *Modifier* column allows you to modify 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 section 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* column sets 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, without going to the edit test dialogs.

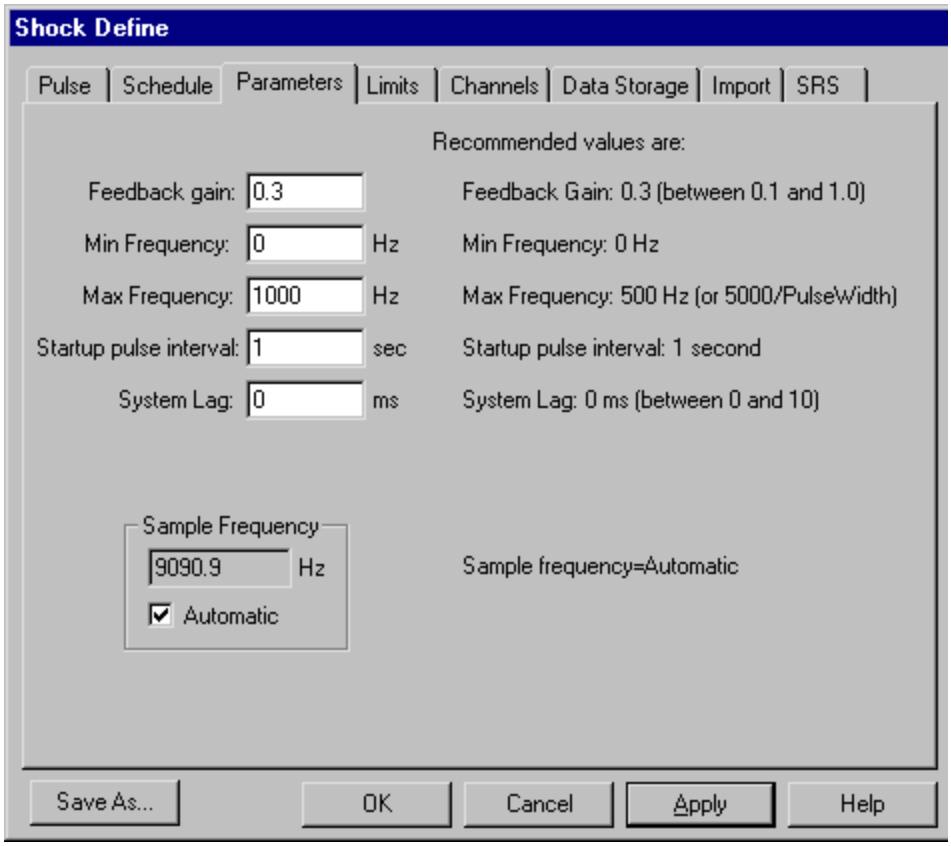
To disable the use of the memorized drive signal and revert to the graduated test start-up, uncheck the "Enable memorized drive" box.

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.

21.4.2. Shock Define Parameters tab

This dialog box is accessed by clicking the "Parameters" tab in the Shock Define dialog box.



See also: How to tune shock controller parameters.

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

Feedback Gain sets the rate at which the controller adjusts the drive output, and also 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.

Min Frequency sets 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.

Max Frequency sets 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 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.

Lag (ms) sets 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, but will

not affect the steady-state behavior of the controller. A value of 0 is fine for most situations.

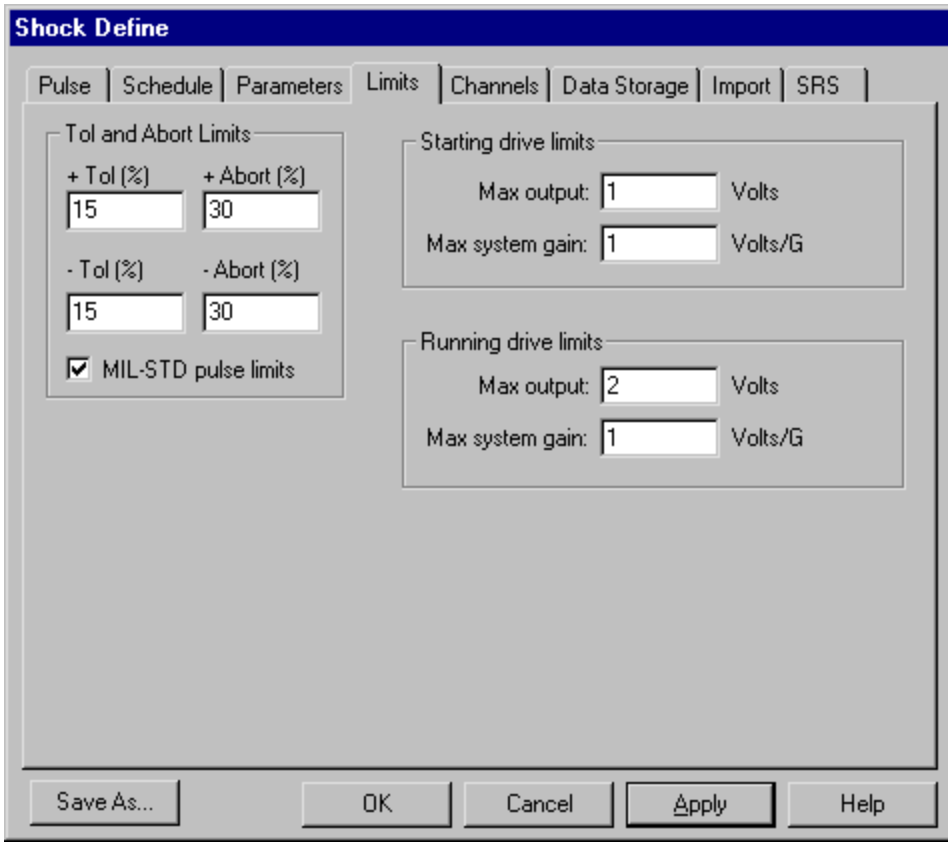
The *Startup Pulse Interval* parameter sets the time (in seconds) between pulses output while the controller is equalizing the system. A value of 0 will result in pulses output as fast as the controller can output them.

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

In the case of user-defined pulses, setting the sample frequency to Auto will select the sample frequency entered on the Import tab when the waveform was imported from an ASCII file. Manually setting the sample frequency 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.

21.4.3. Shock Define Limits tab

This dialog box is accessed by clicking the "Limits" tab in the Shock Define dialog box.



Use this dialog box to specify the positive and negative *tolerance* and *abort* limits, entered as a percentage of the peak pulse acceleration.

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).

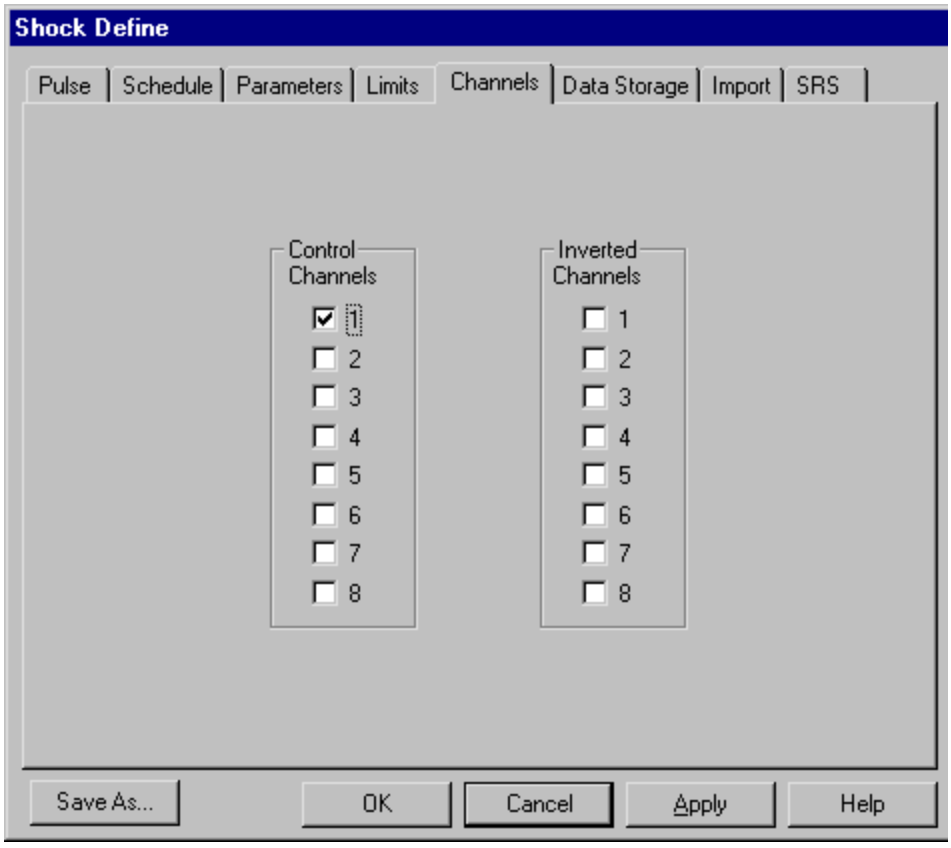
The *Max output* limit is an absolute voltage limit on the output. If this voltage is exceeded (in either the positive or negative direction) the test will abort.

The *Max system gain* limit is a relative limit on 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 maximum output to about 150% of the required level, and set the max system gain value to 2 times the peak output voltage divided by the peak acceleration.

21.4.4. Shock Define Control Channels tab

This dialog box is accessed by clicking the "Channels" tab in the Shock Define dialog box.



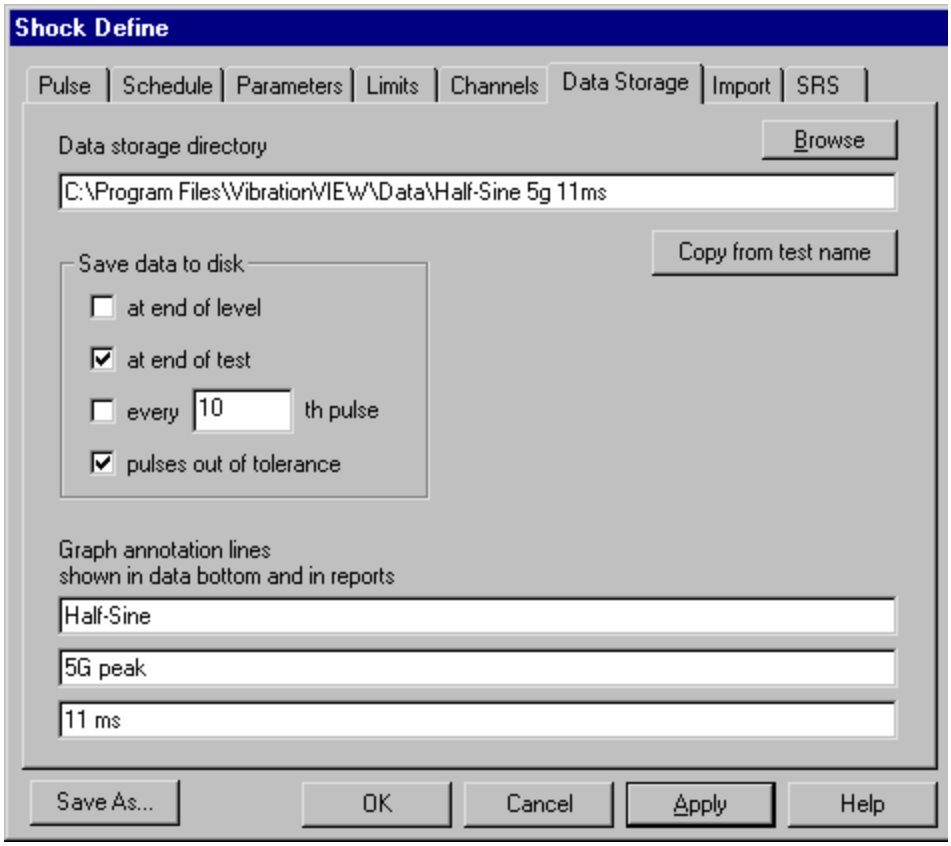
Use this dialog box to select which channel (or channels) will be used as the control signal. When multiple channels are selected, the control waveform will be the average of each of the

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.

21.4.5. Shock Define Data Storage tab

This dialog box is accessed by clicking the "Data Storage" tab in the Shock Define dialog box.



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

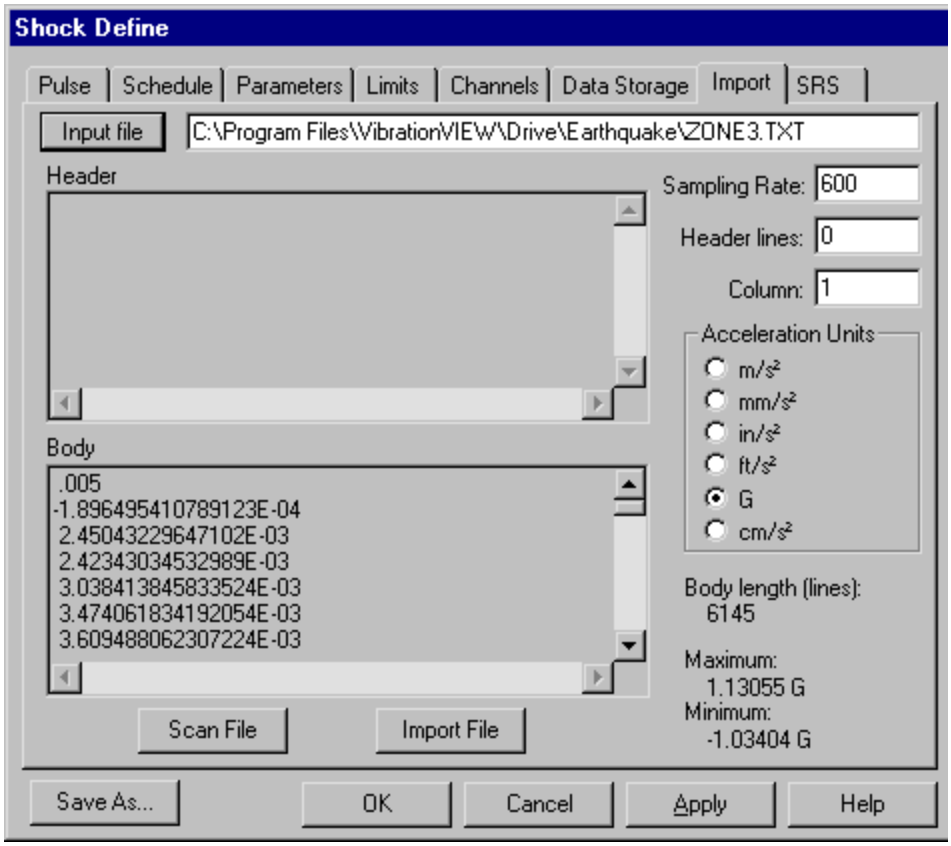
You can save the data automatically at the end of a level (as defined on the "Schedule" tab), at the end of a test, at a regular pulse interval, and/or all pulses that are outside of the tolerance lines. Check the appropriate boxes to activate the automatic data store function(s).

You can also enter three lines that will appear in the Data Bottom portion of a graph. This display can be enabled for a graph by editing a graph's settings, and selecting the "Data Bottom" check box in the Shock Graph Settings dialog box.

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.

21.4.6. Shock Define Import tab

This dialog is accessed by clicking the "Import" tab in the Shock Define 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 a text file.

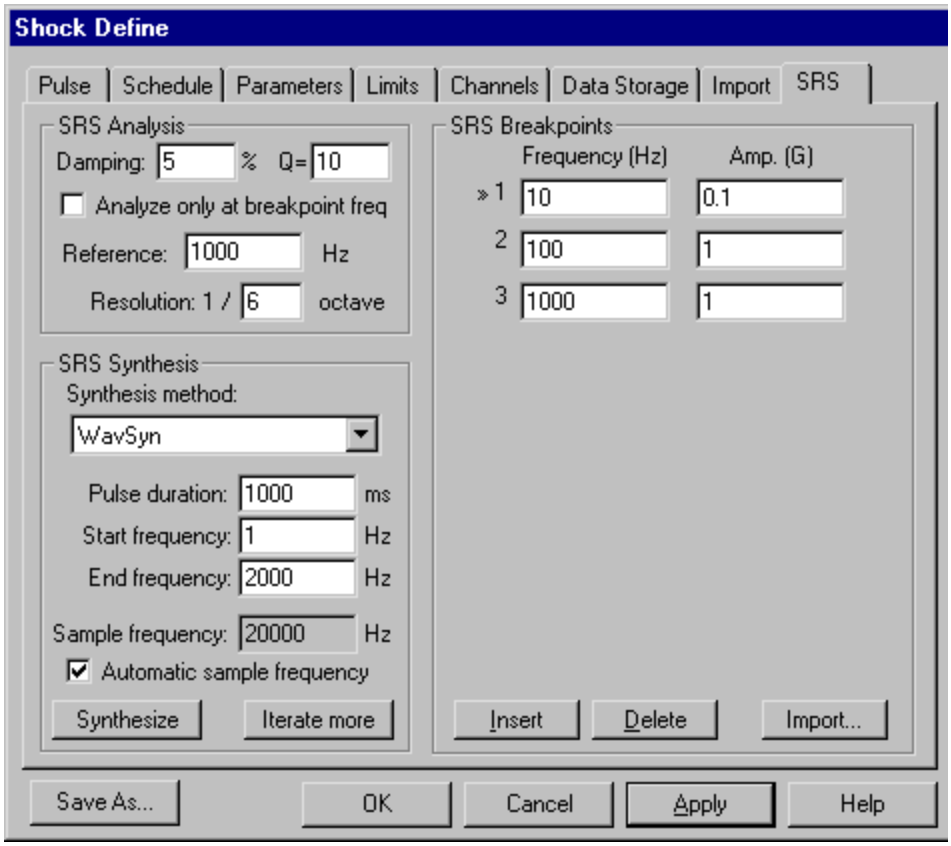
To input a waveform from an ASCII file, perform the following procedure:

1. Click the "Input file" button to select 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 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 that the software key be programmed at the factory to enable it.

21.4.7. Shock Define SRS tab

This dialog is accessed by clicking the "SRS" tab in the Shock Define dialog box.



Use this dialog box to define the SRS analysis parameters, and to synthesize a pulse to meet a specified SRS curve. Note: the SRS test method is an optional feature that must be enabled in your software key in order to access this dialog box.

The ***SRS Analysis*** parameters configure the damping/Q value 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) up to an upper frequency of half of the sample frequency.

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 method*** 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 Frequency 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.

The ***SRS Breakpoints*** defines the amplitude vs. frequency curve of the SRS specification. Click the "Insert" button to add breakpoints to the list. Select a line (indicated by the double-arrow next to the line number) and then click the "Delete" button to remove breakpoints from the list.

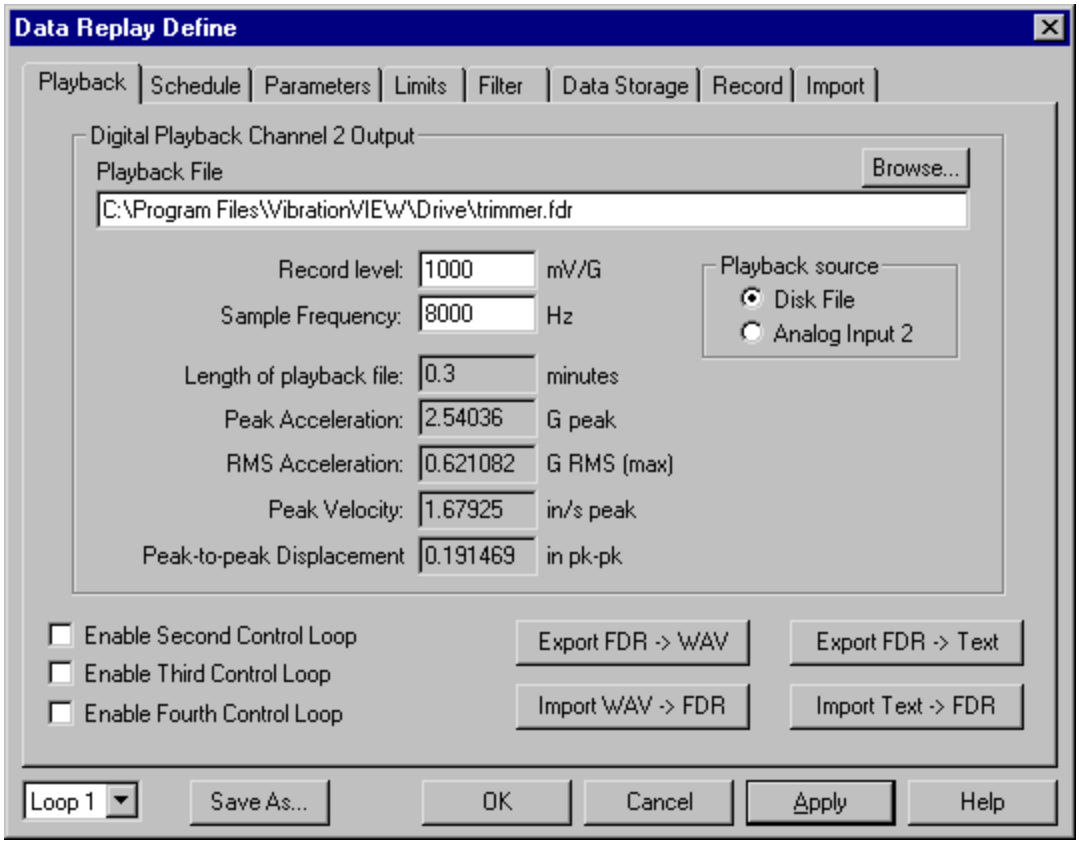
The "Import..." button allows you to import a breakpoint table from a text file. The text file must list one frequency value (in Hz) and one amplitude value (in the currently selected Acceleration Unit) per line, separated by either spaces, tabs, commas (,) or semicolons (;).

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" buttons in the bottom left corner of the graph. The "SRS" button will restore the SRS Profile Definition graph.

21.5. Field Data Replicator Define dialog box

This dialog box is accessed by selecting the Test..Edit Test menu command while a Field Data Replicator test is open.



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

The digital playback file can also be turned on/off by selecting the "On" or "Off" radio button. When playback is turned off, the controller uses the channel 2 input for the reference signal. Using this feature you can play a DAT tape directly into the channel 2 input (without first recording to disk) to have the recording replayed on the shaker with proper equalization. Note that the channel 2 input limits are +/- 2.75 volts.

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.
Data Store	Enter automatic data storage setting here.
Filter	Enter filter frequency settings here.
Record	Record a playback file from the channel 2 input
Import	Import ASCII files to FDR files

The "Export FDR -> WAV" and "Import WAV -> FDR" buttons are used to convert files back-and-

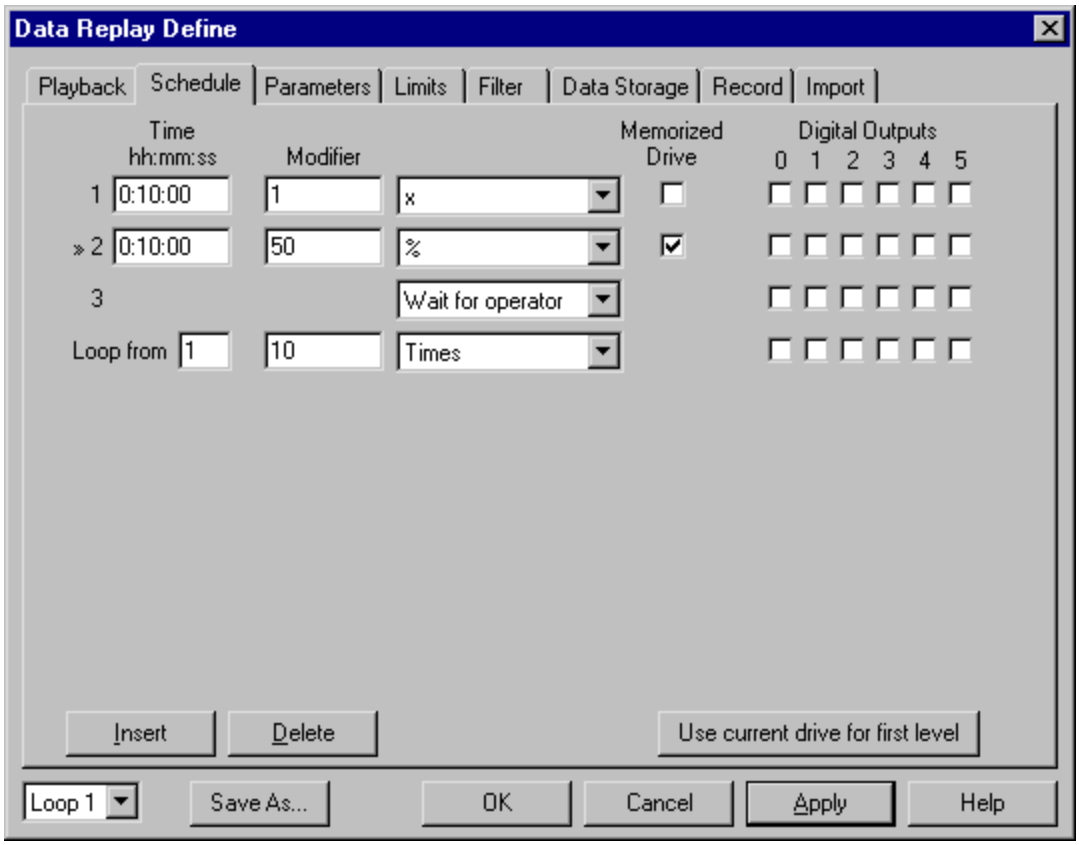
forth between FDR (VibrationVIEW internal format) and WAV (Windows audio file format) files. Standard Windows waveform editor programs can be used to filter, crop, or otherwise manipulate waveforms saved as .WAV files.

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.

21.5.1. Field Data Replicator Define Schedule tab

This dialog box is accessed by clicking the "Schedule" tab in the Field Data Replicator Define dialog box.



Use this dialog box to set 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.

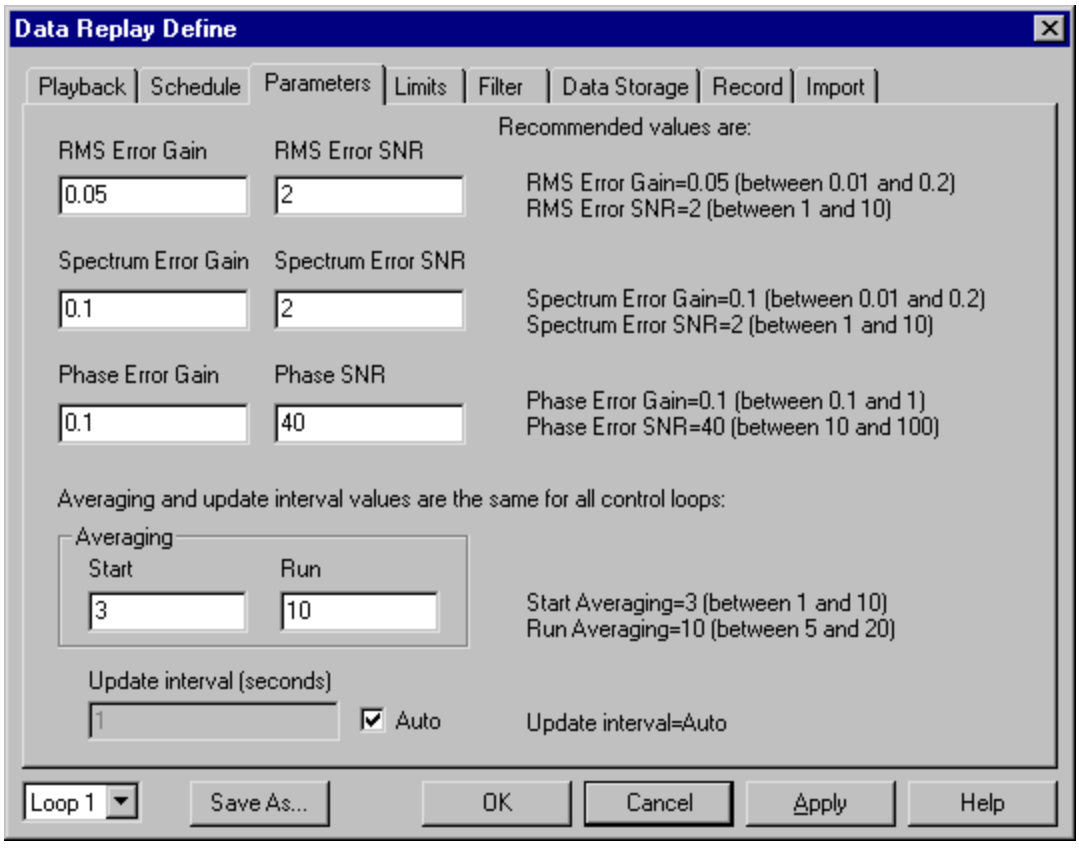
There are also two other types of modifiers: "Times" and "Wait for operator". The "Times" modifier type is used to enter a repeated loop in the test schedule, if you want to run a test that cycles repeatedly through a certain sequence of amplitude levels. The "Wait for operator" modifier type will pause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

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 immediately 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.

21.5.2. Field Data Replicator Define Parameters tab

This dialog box is accessed by clicking the "Parameters" tab in the Field Data Replicator Define 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.

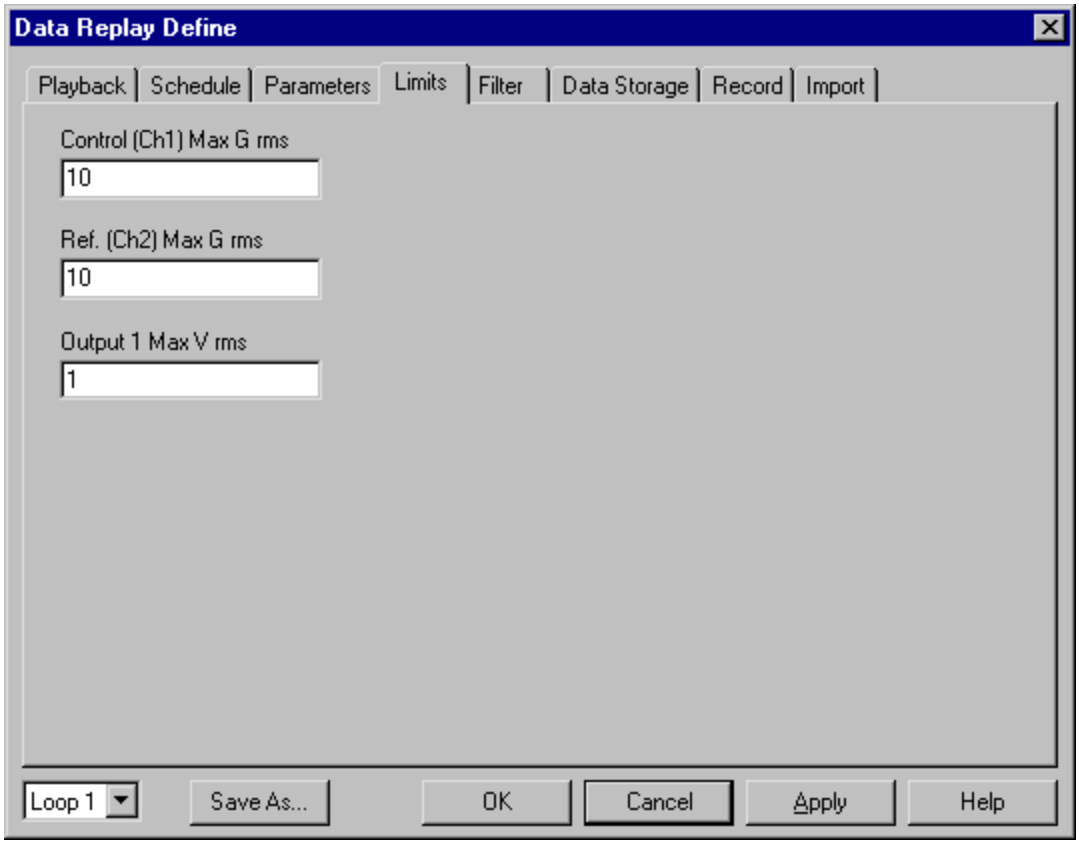
The averaging values set the amount of temporal averaging applied to the spectra during startup and normal operation. A typical value for startup is 3, and for run is 10.

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.

21.5.3. Field Data Replicator Define Limits tab

This dialog box is accessed by clicking the "Limits" tab in the Field Data Replicator Define 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.

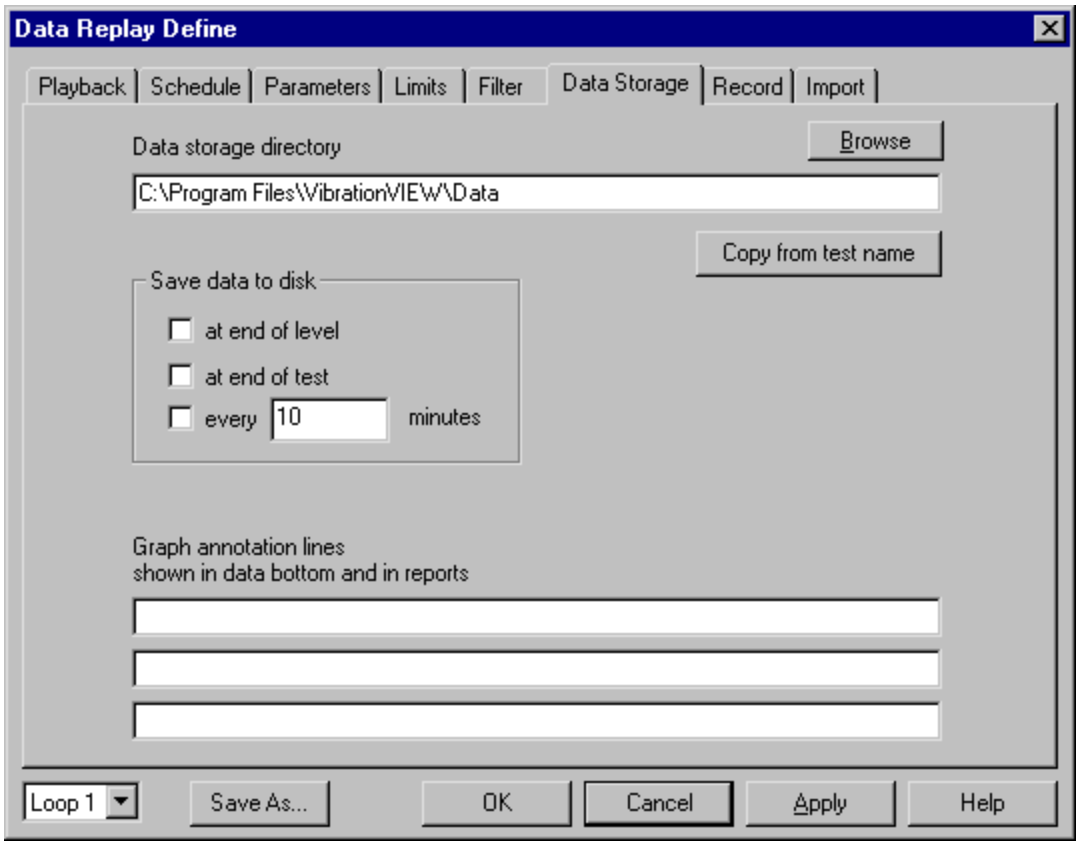
The Reference (Ch2) Max is the maximum RMS acceleration level allowed on the reference

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.

21.5.4. Field Data Replicator Define Data Storage tab

This dialog box is accessed by clicking the "Data Storage" tab in the Field Data Replicator Define dialog box.

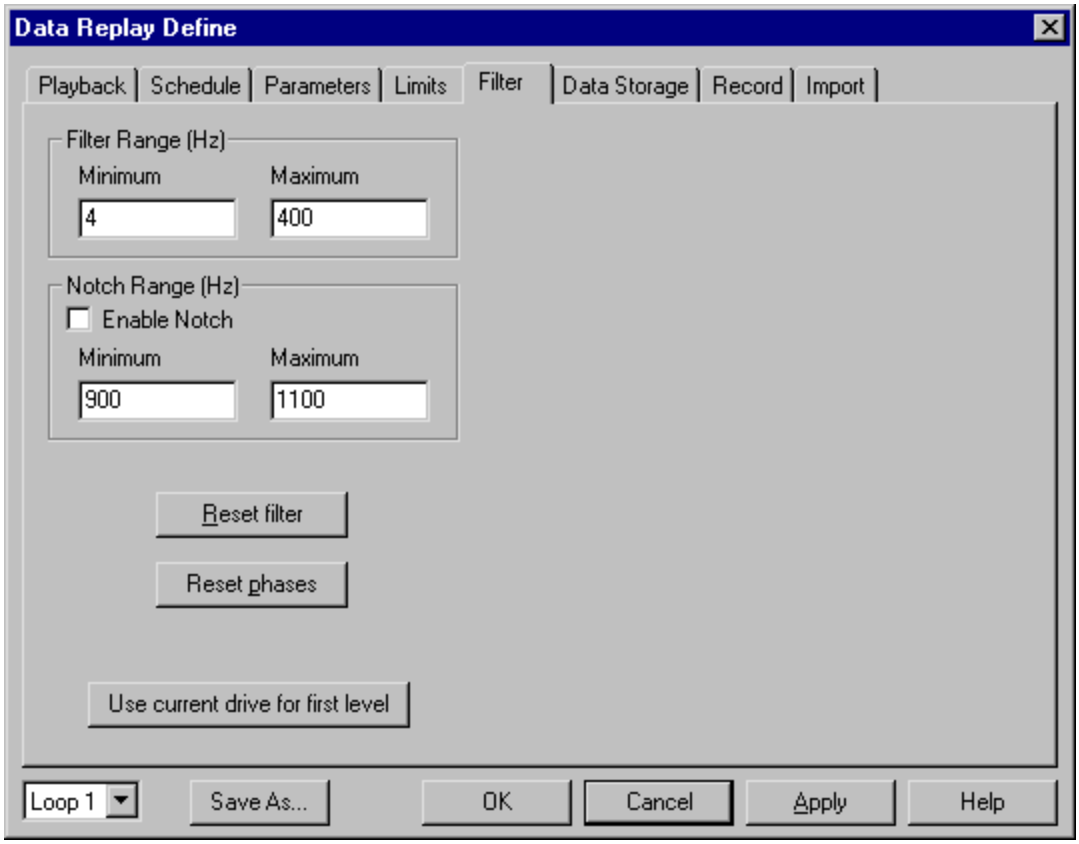


Use this dialog box to set the times at which the test data is automatically saved while the test runs. You can save the data at the end of a level, at the end of a test, or at the end of a time period. All data is saved as a time stamped file that can be viewed at a later time by selecting the File..Open Data menu command.

Check the boxes to activate the desired data storage functions.

21.6. Field Data Replicator Define Filter tab

This dialog box is accessed by clicking the "Filter" tab in the Field Data Replicator Define dialog box.



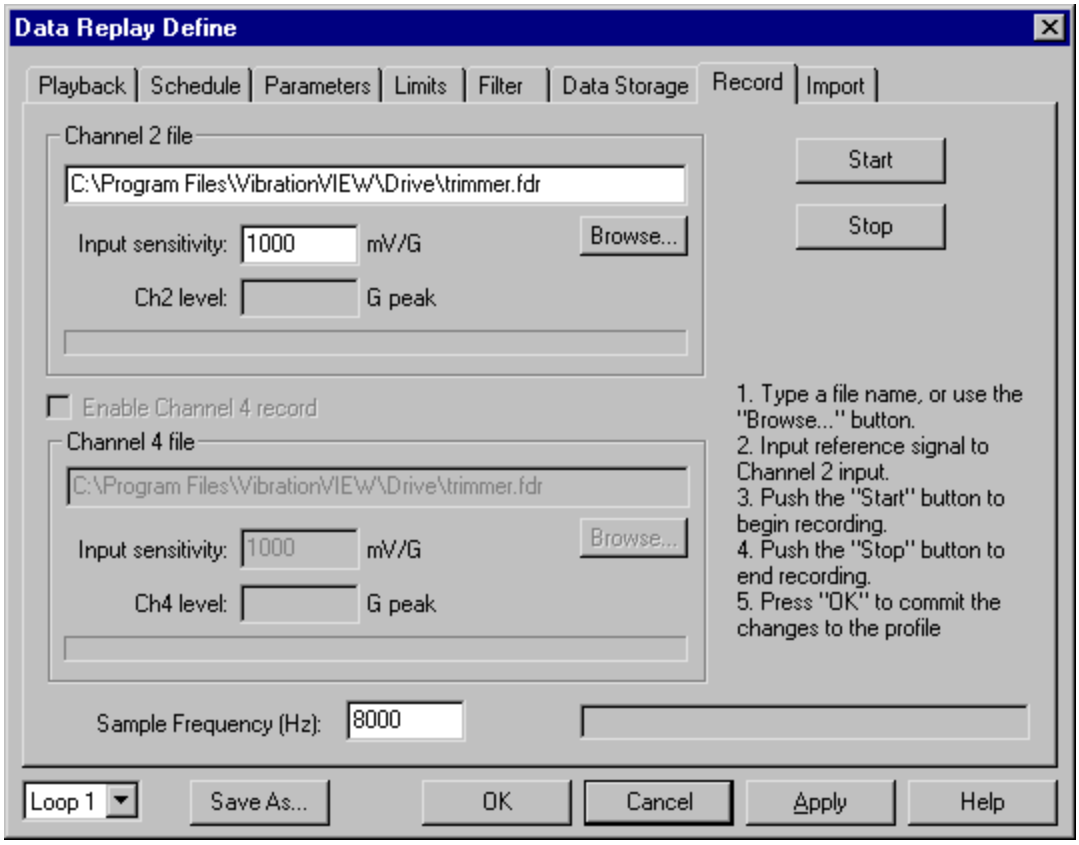
Use this dialog box to display and adjust the filter frequency settings, i.e. to set the frequency range over which the feedback control will operate. You can also enable a notch band to filter out a frequency range in the middle of the spectrum. This is useful for such things as a shake-and-rattle test,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

Click the "Reset filter" button to force the controller to relearn the Frequency Response Function of the system.

Click the "Reset phases" button to force the controller to relearn the phases of the Frequency Response Function of the system (but retain the learned amplitudes).

21.6.1. Field Data Replicator Define Record tab

This dialog box is accessed by clicking the "Record" tab in the Field Data Replicator Define dialog box.

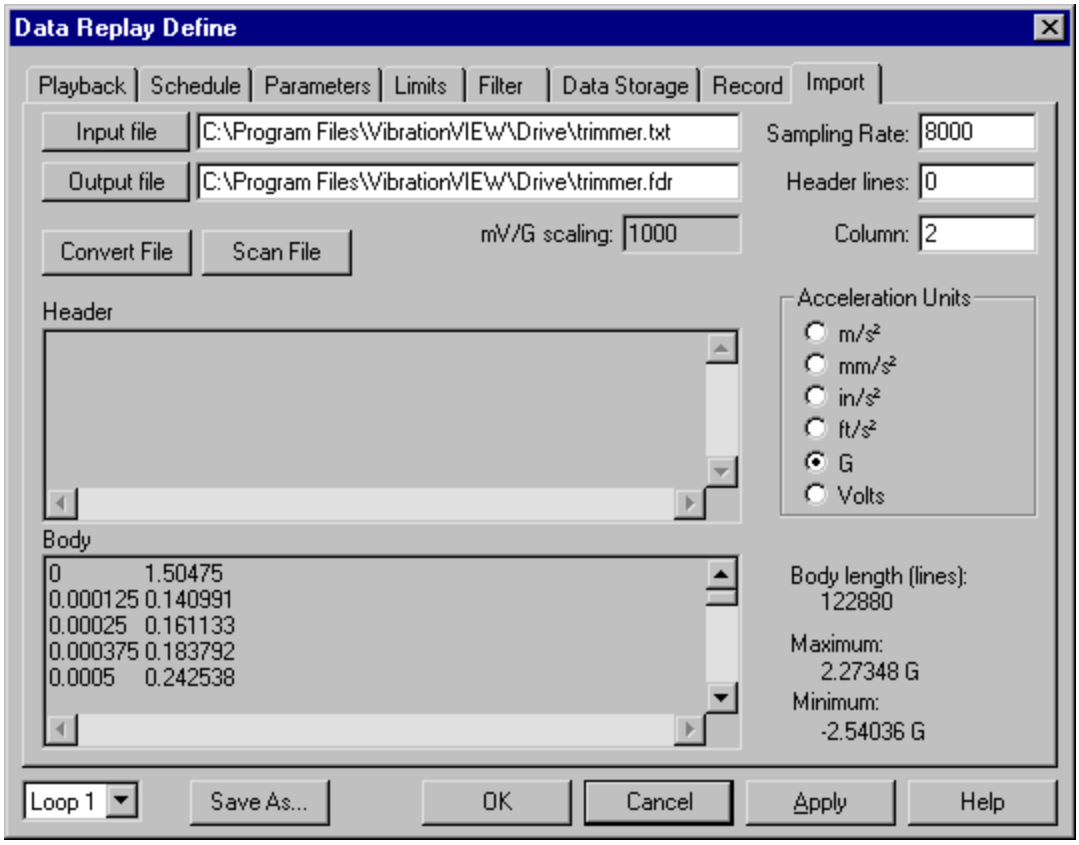


This dialog box is used to record a new playback file from the channel 2 input. Typical use for this dialog box is to connect the output of some playback device (e.g. a DAT recorder) to the Channel 2 input of the VibrationVIEW I/O unit, and then to record this data to the hard drive.

Hint: If your recording device has the ability to store data to a wav file, you can also import the data using the "WAV -> FDR".

21.6.2. Field Data Replicator Define Import tab

This dialog box is accessed by clicking the "Import" tab in the Field Data Replicator Define 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, and then click the "Convert File" button to convert the file to the VibrationVIEW internal format (FDR format).

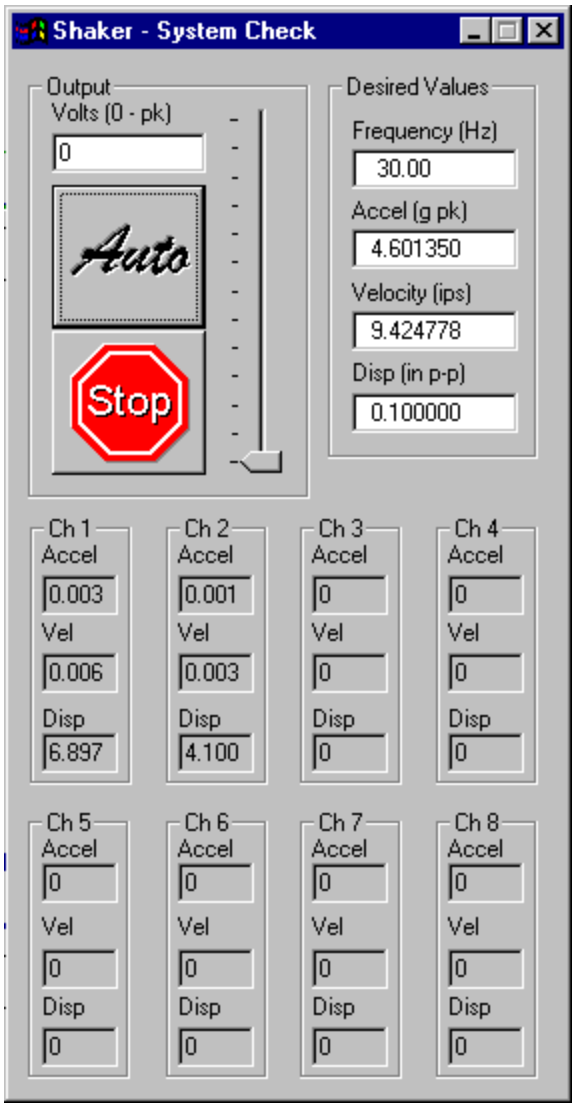
The arrows in the body section indicate the select column. If you change the column number or number of header lines, click the "Scan file" button to update the header, body, maximum, and minimum display values.

22. Control Centers

22.1. System Check Control Center

The control centers are docking toolbars, and may 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.



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, stopping at the specified 0.10 inches.

Auto: This button automatically adjusts the output voltage such that the channel 1 input matches the Desired Values of Acceleration, Velocity, and Displacement.

Stop: This button immediately return the output voltage to 0 volts (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 is 30 Hz

that works well on most systems.

Accel: The desired peak acceleration for input channel 1 is entered here; used when Automatic mode is enabled.

Velocity: The desired peak velocity for input channel 1is entered here; used when Automatic mode is enabled.

Disp: The desired peak-to-peak displacement for input channel 1 is entered here; used when Automatic mode is enabled.

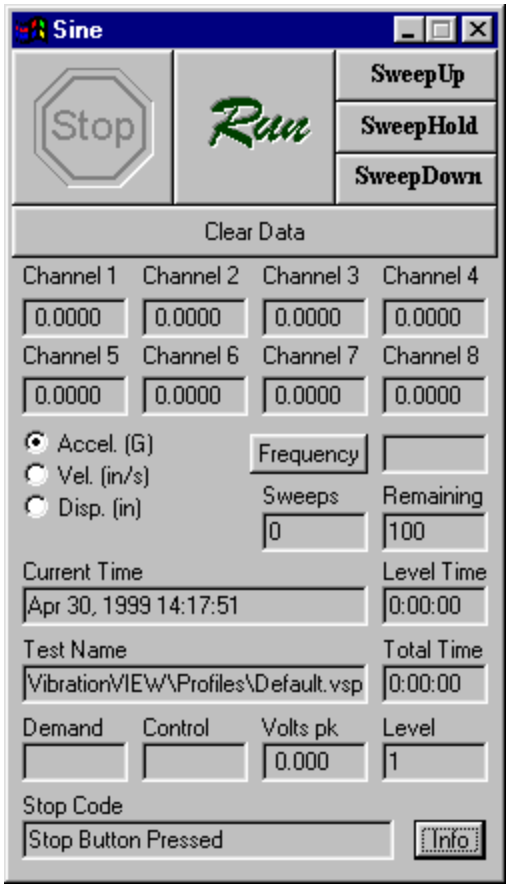
Channels: The values reached by each channel:

Accel	measured peak acceleration value
Vel	measured peak velocity
Disp	measured peak-to-peak displacement value. This value should match the displacement of the shaker (as measured using a displacement indicator such as a displacement wedge).

22.2. Sine Control Center

The control centers are docking toolbars, and may 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.



Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test for some reason and desire 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. Clicking this button is the equivalent of flipping the stop switch on the front of the I/O box.

Run: This button starts the test. Click this button when you are ready to start your test. This button will show dark green when you are not running, yellow when you are starting, light blue if starting with a memorized drive and green when you are running. This button is the equivalent of flipping the start switch on the front of the VibrationVIEW I/O unit.

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.

Channels: These boxes are the input channels. They show you what the system is reading for the shaker acceleration, velocity, or displacement (depending on which mode it is in. See below.).

Mode Field: These radio buttons allow you to easily change what type of information the channels are showing. You can choose acceleration, velocity, or displacement. The units can also be changed for these readings (see Operators Manual or help file).

Frequency: This box shows the current operating frequency (when running). You can use the button next to this display to define a new starting frequency.

Sweeps: This box shows how many times up and down (in frequency) the test has run.

Remaining: This box shows how many times the test has to sweep before it finishes.

Current Time: This box shows the current date and time.

Level Time: This box shows how long you have been at the current level.

Test Name: This box shows the name of the test file that you are running.

Total Time: This box shows how much time has elapsed since the test started.

Demand: This box shows the desired acceleration/velocity/displacement level for the test, as selected using the Mode radio buttons.

Control: This box shows the measured acceleration/velocity/displacement level for the control signal, as selected using the Mode radio buttons.

Volts Pk: This box shows what the output voltage level is.

Level: This box shows the current level at which the test is running.

Stop Code/Info: This box shows why the test shut down. Click this button for more information on the current status of the test. See also: Sine Stop Codes

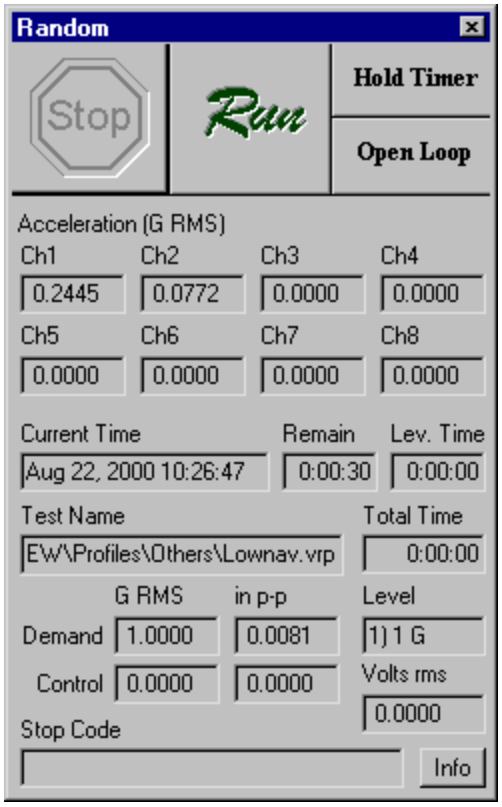
The sine control box shows 8 channels. All installed channels will have the current value, indicated by the radio button selected (acceleration, velocity, displacement). To change units

(AVD) just click the radio button next to the desired parameter. This also affects the control and demand variables.

22.3. Random Control Center

The control centers are docking toolbars, and may 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.



Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test for some reason and desire 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. Clicking this button is the equivalent of flipping the stop switch on the front of the VibrationVIEW I/O unit.

Run: This button starts the test. Click this button when you are ready to start your test. This button will show dark green when you are not running, yellow when you are starting, light blue if starting with a memorized drive and green when you are running. Clicking this button is the equivalent of flipping the start switch on the front of the VibrationVIEW I/O unit.

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.

Channels: These boxes show the measured RMS acceleration levels for each of the input channels.

Current Time: This box indicates the current date and time.

Remain: This box indicates how much time is remaining at the current level of the test schedule.

Lev. Time: This box indicates how long the test has been running at the current level of the test schedule.

Total Time: This box shows how much time has elapsed since the test started.

Test Name: This box shows is the name of the test file that you are running.

Demand: This box shows the expected RMS acceleration and pk-pk displacement level for the test.

Control: This box shows the measured RMS acceleration and pk-pk displacement level of the control signal.

Volts rms: This box shows the output voltage (RMS) level.

Level: This box shows the current level number and modifier value that the test is running.

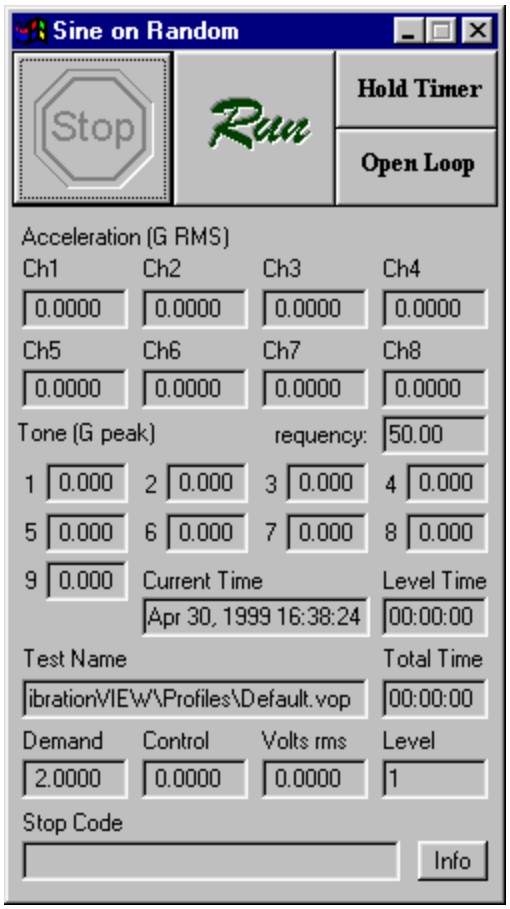
Stop Code/Info: This box shows why the test shut down. Click this button for more information on the current status of the test. See also: Random Stop Codes.

The random control box shows 8 channels. All installed channels will have the current rms acceleration value, indicated in g RMS.

22.4. Sine-on-Random Control Center

The control centers are docking toolbars, and may 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.



Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test for some reason and desire 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. Clicking this button is the equivalent of flipping the stop switch on the front of the VibrationVIEW I/O unit.

Run: Click this button when you are ready to start your test. This button will show dark green 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. Clicking this button is the equivalent of flipping the start switch on the front of the VibrationVIEW I/O unit.

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.

Channels: These boxes show the RMS acceleration measurements for each of the input channels.

Frequency: This box shows the current operating frequency (when running). It sweeps through the sine tones (See Operators Manual - Sine Define).

Tone: These boxes show the measured acceleration value of the Sine Tones that the test is running.

Current Time: This box shows the current date and time.

Level Time: This box shows how long you have been at the current level.

Total Time: This box shows how much time has elapsed since the test started.

Test Name: This box shows the name of the file for the test that you are running.

Demand: This box shows the desired RMS acceleration level.

Control: This box shows the measured RMS acceleration level of the control signal.

Volts rms: This box shows the output voltage (RMS) level.

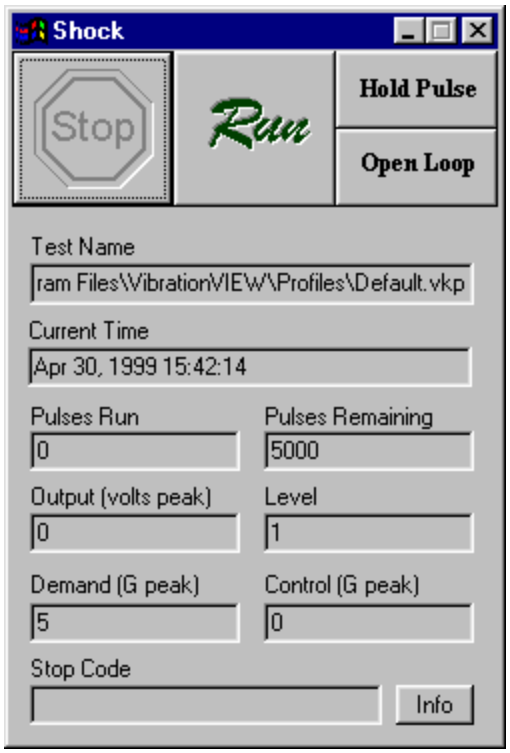
Level: This box shows the current level number and modifier value the test is running.

Stop Code/Info: This box shows why the test shut down. Click this button for more information on the current status of the test. See also: Sine-On-Random Stop Codes.

22.5. Shock Control Center

The control centers are docking toolbars, and may 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.



Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test for some reason and desire 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. Clicking this button is the equivalent of flipping the stop switch on the front of the VibrationVIEW I/O unit.

Run: This button runs the test. Click this button when you are ready to start your test. This button will show dark green when you are not running, yellow when you are starting, light blue if you start with a memorized drive, and green when you are running. Clicking this button is the equivalent of flipping the start switch on the front of the VibrationVIEW I/O unit.

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 shows the name of the file that you are running.

Pulses Run: This box shows how many shock pulses have been output 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 shows how many shock pulses the test still has to run at this level.

Output: This box shows the current peak output voltage of the test.

Level: This box shows the current level number and modifier value the test is running.

Demand: This box shows the desired peak acceleration value.

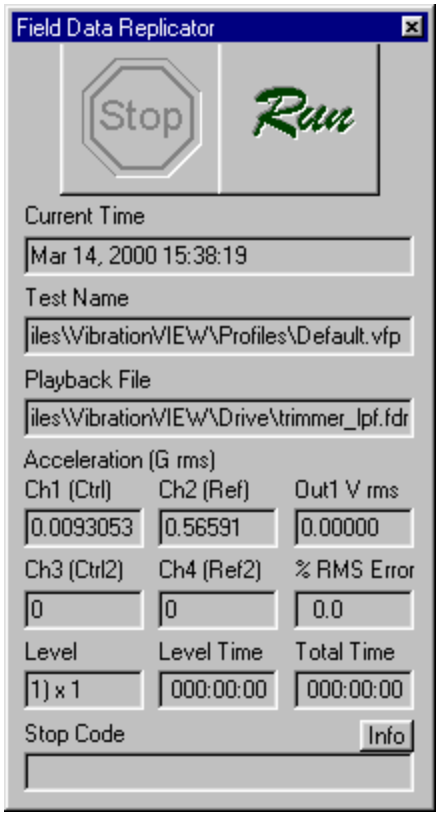
Control: This box shows the measured peak acceleration value.

Stop Code/Info: This box shows why the test shut down. Click this button for more information on the current status of the test. See also: Classical Shock Stop Codes

22.6. Field Data Replicator Control Center

The control centers are docking toolbars, and may 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.



Stop: This button stops the test. You should stop the test before opening a new test. If you stop a test for some reason and desire 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. Clicking this button is the equivalent of flipping the stop switch on the front of the VibrationVIEW I/O unit.

Run: This button runs the test. Click this button when you are ready to start your test. This button will show dark green when you are not running, yellow when you are starting, light blue if you start with a memorized drive, and green when you are running. Clicking this button is the equivalent of flipping the start switch on the front of the VibrationVIEW I/O unit.

Current Time: This box shows the current date and time

Test Name: This box shows the name of the file that you are running.

Playback File: This box shows the file that you are reading from in your test.

Ctrl: This box shows the RMS acceleration measurement of the control input.

Level Time: This box shows how long you have been at the current level.

Ref: This box shows the RMS acceleration level of the reference signal.

Total Time: This box shows how much time has elapsed since the test started.

V rms: This box shows what the current output voltage level is.

Multiplier: This box shows the factor applied to the reference level to increase/decrease the control level.

Level: This box shows the current level that the test is running.

Stop Code/Info: This box shows why the test shut down. Click this button for more information on the current status of the test. See also: Field Data Replicator Stop Codes.

23. Stop Codes

23.1. System Check Stop Codes

The following codes indicate that something has occurred to cause the controller to abort the current test.

Max Sine Acceleration	Desired Acceleration is too high	Input channel N is clipping
Max Sine Velocity	Desired Velocity is too high	Remote Start
Max Sine Displacement	Desired Displacement is too high	Remote Stop
No acceleration detected	Input is Clipping	Watchdog Timeout
Test Parameters Changed	Stop Button Pressed	Emergency Stop

23.1.1. Stop Button Pressed (System Check Stop Code)

This stop code indicates that the output was turned off because the stop button was pressed.

23.1.2. 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.

23.1.3. Max Sine Acceleration (System Check Stop Code)

This code indicates that the maximum acceleration allowed for the system has been exceeded.

This limit is a combination of the Force limits of the shaker system (defined in the System Limits dialog box) and the mass of the shaker and fixture (defined in the System Mass dialog box) using the 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 abilities of your shaker, or the current test requires a larger shaker than you are using.

23.1.4. 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 abilities of your shaker, or the current test requires a larger shaker than you are using.

23.1.5. Max Sine Displacement (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 abilities of your shaker, or the current test requires a shaker with a larger displacement range than you are using.

23.1.6. 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 acceleration limit is a combination of the Force limits of the shaker system (defined in the System Limits dialog box) and the mass of the shaker and fixture (defined in the System Mass dialog box) using the 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 abilities of your shaker, or the current test requires a larger shaker than you are using.

23.1.7. 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 velocity 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 abilities of your shaker, or the current test requires a larger shaker than you are using.

23.1.8. 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 displacement 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 abilities of your shaker, or the current test requires a shaker with a larger displacement range than you are using.

23.1.9. 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 +/- 2.75 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	270 G peak
20 mV/G	135 G peak
50 mV/G	54 G peak
100 mV/G	27 G peak
1000 mV/G	2.7 G peak

23.1.10. Remote Start (System Check Stop Code)

This code indicates that the test was switched into Automatic mode using the Remote Start/Stop option.

23.1.11. Remote Stop (System Check Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option.

23.1.12. Emergency Stop (System Check Stop Code)

This code indicates that the emergency stop input was pressed, causing the controller to shut down. The emergency stop input is an optional accessory that connects to the remote input terminal block on the rear of the VibrationVIEW I/O unit. This option is enabled and disabled using the "Emergency stop input" selection in the VibrationVIEW Configuration Parameters dialog box.

23.1.13. 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. Clicking on the "Run" button will reset the timeout.

The most likely cause of this error is a loss of power to the VibrationVIEW I/O unit, either because it is turned off, or because of a temporary power outage.

Other events that could cause this to happen are:

the VibrationVIEW I/O unit is turned on after the program is started.

the serial cable connecting the computer with the VibrationVIEW I/O unit comes loose.
the computer freezes.
the computer crashes.

23.2. Sine Stop Codes

The following codes indicate the status of the controller, and occur during normal controller operation:

Starting Test	Running Test	Starting Schedule
Remote Start	Remote Stop	Step On
Front Panel Start	Front Panel Stop	Step Off
End of Timed Test	End of Sweep Test	End of Cycle Count Test
Holding Frequency	Resonance Dwell	Select Resonance Frequencies
Changing Level	Stop Button Pressed	Waiting for operator
NI-DAQ error code		

The following codes indicate that something has occurred to cause the controller to abort the current test.

Test Parameters Changed	Max Start Drive	Max Start System Gain
Max Sine Acceleration	Max Run Drive	Max Run System Gain
Max Sine Velocity	Control Plus Abort	Control Minus Abort
Max Sine Displacement	Resonance Min Drift Limit	Resonance Max Drift Limit
Channel N Minus Abort	Channel N Plus Abort	Input channel N is clipping
Didn't Reach Demand Level	Emergency Stop	Invalid Remote Test

23.2.1. Starting Test (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.

23.2.2. Running Test (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.

23.2.3. Starting Schedule (Sine Stop Code)

This code indicates that this test is one of several tests listed in a test schedule, and was started using the Test Schedule dialog box.

23.2.4. Waiting for operator (Sine 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" 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.

23.2.5. 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 test mode will change to Running Test.

23.2.6. 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 is complete and the output is turned off.

23.2.7. Invalid Remote Test (Sine Stop Code)

This stop code indicates that the "Remote Start/Stop" and "Remote Selection" options are both enabled, but that 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".

23.2.8. Emergency Stop (Sine Stop Code)

This code indicates that the emergency stop input was pressed, causing the controller to shut

down. The emergency stop input is an optional accessory that connects to the remote input terminal block on the rear of the VibrationVIEW I/O unit. This option is enabled and disabled using the "Emergency stop input" selection in the VibrationVIEW Configuration Parameters dialog box.

23.2.9. 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. Clicking the "Run" button will reset the timeout.

The most likely cause of this error is a loss of power to the VibrationVIEW I/O unit, either because it is turned off, or because of a temporary power outage.

Other events that could cause this to happen are:

the VibrationVIEW I/O unit is turned on after the program is started.
the serial cable connecting the computer with the VibrationVIEW I/O unit comes loose.
the computer freezes.
the computer crashes.

23.2.10. 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".

23.2.11. 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".

23.2.12. Front Panel Start (Sine Stop Code)

This code indicates that the test was started using the "Start" switch on the front panel of the VibrationVIEW I/O unit. 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.

23.2.13. Front Panel Stop (Sine Stop Code)

This code indicates that the test was stopped using the "Stop" switch on the front panel of the VibrationVIEW I/O unit. When this code is displayed, the test is complete and the output is turned off.

23.2.14. 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.

23.2.15. 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.

23.2.16. 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.

23.2.17. Holding Frequency (Sine Stop Code)

This code indicates that the output frequency is held at a fixed value. The frequency sweep can be frozen either using mouse control in the Sine Big Display dialog box, or by clicking the "Hold Sweep" button in the Sine Control Center.

23.2.18. Select Resonance Frequencies (Sine Stop Code)

This code is displayed when the Select Resonance Frequencies dialog box is shown. When this code is displayed, the test will be paused waiting for the operator to select desired resonance frequencies.

23.2.19. Resonance Dwell (Sine Stop Code)

This code indicates that Sine Resonance Mode is enabled (see the Sine Resonance Control dialog) 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 either using mouse control in the Sine Big Display dialog box, or by clicking the "Hold Sweep" button in the Sine Control Center.

23.2.20. Changing Level (Sine Stop Code)

This code indicates that the test is finishing one level and starting another. The test continues to run.

23.2.21. Stop Button Pressed (Sine Stop Code)

This code indicates that the test was halted through operator control, by selecting either the Test..Stop Test menu command, clicking the "Stop" button on the toolbar, or clicking the "Stop" button in the Sine Control Center.

23.2.22. Test Parameters Changed (Sine Stop Code)

This code indicates that the test parameters were changed, using either the Test..Edit Test or the Test..New Test menu commands, or their respective toolbar buttons. As a result of the test parameter changes, the test was halted.

23.2.23. 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 either 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 specimen. You can disable this safety feature (***not recommended!!!***) by selecting a high voltage level (e.g. 10 Volts) for the Max Start Drive parameter.

23.2.24. 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 either 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 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 specimen. 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.

23.2.25. Max Run Drive (Sine Stop Code)

The maximum allowable voltage was reached while the test was running. This error typically results from having either 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 specimen. You can disable this safety feature (***not recommended!!!***) by selecting a high voltage level (e.g. 10 Volts) for the Max Run Drive parameter.

23.2.26. 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 either 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 specimen. 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.

23.2.27. Max Sine Acceleration (Sine Stop Code)

This code indicates that the maximum acceleration allowed for the system has been exceeded. This limit is a combination of the Force limits of the shaker system (defined in the System Limits dialog box) and the mass of the shaker and fixture (defined in the System Mass dialog box) using the 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 abilities of your shaker, or the current test requires a larger shaker than you are using.

23.2.28. 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 abilities of your shaker, or the current test requires a larger shaker than you are using.

23.2.29. 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 abilities of your shaker, or the current test requires a shaker with a larger displacement range than you are using.

23.2.30. Control Plus Abort (Sine Stop Code)

This code indicates that the measured acceleration 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 because the controller is 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 adjusting them to improve the control.

23.2.31. Control Minus Abort (Sine Stop Code)

This code indicates that the measured acceleration 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 because the controller is 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 adjusting them to improve the control.

23.2.32. 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.

23.2.33. 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.

23.2.34. Channel N Minus Abort (Sine Stop Code)

This code indicates that the measured acceleration 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 because the controller is 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 adjusting them to improve the control.

23.2.35. Channel N Plus Abort (Sine Stop Code)

This code indicates that the measured acceleration 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 because the controller is 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 adjusting them to improve the control.

23.2.36. 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 +/- 2.75 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	270 G peak
20 mV/G	135 G peak
50 mV/G	54 G peak
100 mV/G	27 G peak
1000 mV/G	2.7 G peak

23.2.37. 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 a couple minutes. 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 adjusting the controller parameters to improve the control.

23.2.38. 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 results when another program (e.g. VibrationAnalyzer) is attempting to read data from the data acquistion 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 the test, and then resume the test. That is, press the "Stop" button, then the "Run" button, and select "Yes" to continue the current test.

23.3. Random Stop Codes

These are status messages that tell you what the controller is doing.

Starting Test	Running Test	End of Test
Starting With Memorized Drive	Starting Schedule	Changing Level
Front Panel Stop	Front Panel Start	Stop Button Pressed
Remote Start	Remote Stop	
Max Start System Gain	Waiting for operator	

These are stop codes that tell you why the controller stopped the test. A brief description follows explaining what to do about the problem.

Test Parameters Changed	Max Start Drive	Max Run Drive
Input channel N is clipping	Didn't Reach Demand Level	Max Run System Gain
Control Lines Exceeded	Control Plus RMS	Ch N Accel Reading Too High
Max Random Acceleration	Control Minus RMS	Ch N Accel Reading Too Low
Max Random Displacement	Emergency Stop	Invalid Remote Test
Analyzer Trace Outside Aborts	Analyzer Link Failed	Bad Analyzer Configuration

23.3.1. 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.

23.3.2. Running Test (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.

23.3.3. 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.

23.3.4. Starting with Memorized Drive (Random Stop Code)

This code indicates that the test is starting with a memorized drive.

This option can be turned on by checking the box next to "Enable memorized drive" in the Random Level Schedule dialog box. You can either start with 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. By doing this you can start a test where you last saved it (the timer starts over).

23.3.5. Starting Schedule (Random Stop Code)

This code indicates that this test is one of several tests listed in a test schedule, and was started using the Test Schedule dialog box.

23.3.6. Changing Level (Random Stop Code)

This code indicates that the test is finishing one level and starting another. The test continues to run.

23.3.7. 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.

23.3.8. Front Panel Start (Random Stop Code)

This code indicates that the test was started using the "Start" switch on the front panel of the VibrationVIEW I/O unit. 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.

23.3.9. Front Panel Stop (Random Stop Code)

This code indicates that the test was stopped using the "Stop" switch on the front panel of the VibrationVIEW I/O unit. When this code is displayed, the test is complete and the output is turned off.

23.3.10. Remote Start (Random 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.

23.3.11. Remote Stop (Random Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option.

23.3.12. Invalid Remote Test (Random Stop Code)

This stop code indicates that the "Remote Start/Stop" and "Remote Selection" options are both enabled, but that 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".

23.3.13. Stop Button Pressed (Random Stop Code)

This code indicates that you clicked the "Stop" button, telling the test to shut down.

23.3.14. Emergency Stop (Random Stop Code)

This code indicates that the emergency stop input was activated, causing the controller to shut down. The emergency stop input is an optional accessory that connects to the remote input terminal block on the rear of the VibrationVIEW I/O unit. This option is enabled and disabled using the "Emergency stop input" selection in the VibrationVIEW Configuration Parameters dialog box.

23.3.15. 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. Clicking the "Run" button will reset the timeout.

The most likely cause of this error is a loss of power to the VibrationVIEW I/O unit, either because it is turned off, or because of a temporary power outage.

Other events that could cause this to happen are:

the VibrationVIEW I/O box is turned on after the program is started.
the serial cable connecting the computer with the VibrationVIEW I/O unit comes loose.
the computer freezes.
the computer crashes.

23.3.16. Test Parameters Changed (Random Stop Code)

This code indicates that the test parameters were changed, using either the Test..Edit Test or the Test..New Test menu commands, or their respective toolbar buttons. As a result of the test parameter changes, the test was halted.

23.3.17. Max Start Drive (Random Stop Code)

This code indicates that the maximum allowable voltage was reached when starting your test. Perhaps the accelerometer is not turned on, or your amplifier is not turned on. If necessary, you may need more voltage. Modify the appropriate parameter found in the Random Define dialog

box.

23.3.18. Max Run Drive (Random 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. If necessary, you may need more voltage. Modify the appropriate parameter found in the Random Define dialog box.

23.3.19. 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 +/- 2.75 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	90 G RMS
20 mV/G	45 G RMS
50 mV/G	18 G RMS
100 mV/G	9 G RMS
1000 mV/G	0.9 G RMS

23.3.20. Max Start System Gain (Random Stop Code)

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. If necessary, you may need to adjust the maximum start 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 maximum start system gain limit will limit the rate at which the output increases, and this message will appear as a yellow notice code while this limiting is active.

23.3.21. 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 starting the test. Perhaps the accelerometer is disconnected, or your amplifier is not turned on. If necessary, you may need to adjust the maximum start system gain parameter, found in the Random Parameters dialog box.

23.3.22. Control Lines Exceeded (Random Stop Code)

A random test is comprised of "lines" that are narrow filters. This code indicates that more lines than the allowed amount have exceeded the plus and minus abort points. This abort usually occurs if the shaker is shut down without the controller knowing and also if the accelerometer falls off or is disconnected. The number of lines and abort limits are adjusted through the Random Parameters dialog box.

23.3.23. 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.

23.3.24. 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.

23.3.25. Ch. N Accel Reading Too High (Random Stop Code)

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.

23.3.26. Ch. N Accel Reading Too Low (Random Stop Code)

The RMS acceleration level on Channel N is below the maximum 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 of the test fixture.

23.3.27. 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.

23.3.28. 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.

23.3.29. 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 adjusting the controller parameters to improve the control.

23.3.30. 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.

23.3.31. Analyzer Link Failed (Random Stop Code)

This stop 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 Vibration Analyzer 1.0.6 or higher, or if you exit VibrationAnalyzer before a test is complete.

23.3.32. Bad Analyzer Configuration (Random Stop Code)

This code indicates that the configuration you selected in the Analyzer Link dialog box does not exist. Check the filename to make sure it exists.

23.4. Sine-on-Random Stop Codes

These are status messages, to let you know what the controller is doing.

Starting Test	Running Test	End of Test
Starting Schedule	Stop Button Pressed	Front Panel Start
Front Panel Stop	Remote Start	Remote Stop
Changing Level	Starting with Memorized Drive	

These are stop codes, telling you why the controller stopped the test. A brief description follows explaining what to do about the problem.

Test Parameters Changed	Max Start Drive	Max Run Drive
Input channel N is clipping	Control Plus RMS	Control Minus RMS
Max Random Acceleration	Max Random Displacement	Control Lines Exceeded
Didn't Reach Demand Level	Emergency Stop	Invalid Remote Test

23.4.1. Starting Test (Sine-On-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 "Start" group in the Sine-On-Random Parameters dialog box.

23.4.2. Running Test (Sine-On-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 "Run" group in the Sine-On-Random Parameters dialog box.

23.4.3. End of Test (Sine-On-Random Stop Code)

This code indicates the normal shut down condition. Use the Sine-On-Random Schedule dialog box to change how long a test runs.

23.4.4. Starting Schedule (Sine-On-Random Stop Code)

This code indicates that this test is one of several tests listed in a test schedule, and was started using the Test Schedule dialog box.

23.4.5. Stop Button Pressed (Sine-On-Random Stop Code)

This code indicates that you clicked the "Stop" button, telling the test to shut down.

23.4.6. Front Panel Start (Sine-On-Random Stop Code)

This code indicates that the test was started using the "Start" switch on the front panel of the VibrationVIEW I/O unit. 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.

23.4.7. Front Panel Stop (Sine-On-Random Stop Code)

This code indicates that the test was stopped using the "Stop" switch on the front panel of the VibrationVIEW I/O unit. When this code is displayed, the test is complete and the output is turned off.

23.4.8. Remote Start (Sine-On-Random 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.

23.4.9. Remote Stop (Sine-On-Random Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option.

23.4.10. Invalid Remote Test (Sine-on-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".

23.4.11. Emergency Stop (Sine-on-Random Stop Code)

This code indicates that the emergency stop input was pressed, causing the controller to shut down. The emergency stop input is an optional accessory that connects to the remote input terminal block on the rear of the VibrationVIEW I/O unit. This option is enabled and disabled using the "Emergency stop input" selection in the VibrationVIEW Configuration Parameters dialog box.

23.4.12. Watchdog Timeout (Sine-on-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. Clicking the "Run" button will reset the timeout.

The most likely cause of this error is a loss of power to the VibrationVIEW I/O unit, either because it is turned off, or because of a temporary power outage.

Other events that could cause this to happen are:

the VibrationVIEW I/O unit is turned on after the program is started.
the serial cable connecting the computer with the VibrationVIEW I/O unit comes loose.
the computer freezes.
the computer crashes.

23.4.13. Changing Level (Sine-On-Random Stop Code)

This code indicates that the test is finishing one level and starting another. The test continues to run.

23.4.14. Starting with Memorized Drive (Sine-On-Random Stop Code)

This code indicates that the test is starting with a memorized drive.

The option can be turned on by checking the box next to "Enable memorized drive" in the Sine-On-Random Level Schedule dialog box.

In order to set the current drive you must have a test started. Then select the Test..Save Current Drive menu command. By doing this you can start a test where you last saved it (the timer starts over).

23.4.15. Test Parameters Changed (Sine-On-Random Stop Code)

This code indicates that the test parameters were changed using either the Test..Edit Test or the Test..New Test menu commands or their respective toolbar buttons. As a result of the test parameter changes the test was halted.

23.4.16. Max Start Drive (Sine-On-Random 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. If necessary, you may need more voltage. Modify the appropriate parameter found in the Sine-On-Random Define dialog box.

23.4.17. Max Run Drive (Sine-On-Random 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. If necessary, you may need more voltage. Modify the appropriate parameter found in the Sine-On-Random Define dialog box.

23.4.18. Input channel N is clipping (Sine-on-Random Stop Code)

This code indicates that the input signal to channel N is exceeding the allowable input voltage range of +/- 2.75 volts, and is being distorted by clipping. To solve this problem, use an accelerometer with a lower mV/G sensitivity rating.

The maximum allowable random acceleration readings for various accelerometer sensitivities are shown in the table below.

Sensitivity	**Maximum Acceleration**
10 mV/G	90 G RMS
20 mV/G	45 G RMS
50 mV/G	18 G RMS
100 mV/G	9 G RMS
1000 mV/G	0.9 G RMS

23.4.19. Max Random Acceleration (Sine-On-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.

23.4.20. Max Random Displacement (Sine-on-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.

23.4.21. Control Plus RMS (Sine-On-Random Stop Code)

This code indicates that the RMS acceleration level exceeded the maximum prescribed. This value is set in the Sine-On-Random Parameters dialog box.

23.4.22. Control Minus RMS (Sine-On-Random Stop Code)

This code indicates that the RMS acceleration level fell below the minimum prescribed. This value is set in the Sine-On-Random Parameters dialog box.

23.4.23. Control Lines Exceeded (Sine-On-Random Stop Code)

A random test is comprised of "lines" that are narrow filters. This code indicates that more lines than the allowed amount have exceeded the plus and minus abort points. This abort usually occurs if the shaker is shut down without the controller knowing and also if the accelerometer falls off or is disconnected.

23.4.24. Didn't Reach Demand Level (Sine-on-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 Sine-on-Random controller parameters" for information on adjusting the controller parameters to improve the control.

23.5. Shock Stop Codes

The following codes indicate the status of the controller, and occur during normal controller operation:

Starting Test	Running Test	End of Test
Starting Schedule	Stop Button Pressed	Front Panel Start
Front Panel Stop	Remote Start	Remote Stop
Changing Level	Starting with Memorized Drive	Max Starting System Gain
Waiting for operator		

The following codes indicate that something has occurred to cause the controller to abort the current test.

Test Parameters Changed	Max Starting Voltage	Max Run Voltage
Max Shock Acceleration	Didn't Reach Demand Level	Max Run System Gain
Max Shock Velocity	Input Signal is Clipped	Abort Limit
Max Shock Displacement	Emergency Stop	Invalid Remote Test

23.5.1. 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.

23.5.2. Running Test (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.

23.5.3. 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.

23.5.4. Starting Schedule (Shock Stop Code)

This code indicates that this test is one of several tests listed in a test schedule and was started using the Test Schedule dialog box.

23.5.5. Stop Button Pressed (Shock Stop Code)

This code indicates that the test was halted through operator control by selecting either the Test..Stop Test menu command, clicking the "Stop" button on the toolbar, or pressing the "Stop" button in the Classical Shock Control Center.

23.5.6. Front Panel Start (Shock Stop Code)

This code indicates that the test was started using the "Start" switch on the front panel of the VibrationVIEW I/O unit. 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.

23.5.7. Front Panel Stop (Shock Stop Code)

This code indicates that the test was stopped using the "Stop" switch on the front panel of the VibrationVIEW I/O unit. When this code is displayed, the test is complete and the output is turned off.

23.5.8. 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.

23.5.9. 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 is complete and the output is turned off.

23.5.10. 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".

23.5.11. Emergency Stop (Shock Stop Code)

This code indicates that the emergency stop input was pressed, causing the controller to shut down. The emergency stop input is an optional accessory that connects to the remote input terminal block on the rear of the VibrationVIEW I/O unit. This option is enabled or disabled using the "Emergency stop input" selection in the VibrationVIEW Configuration Parameters dialog box.

23.5.12. 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. Clicking the "Run" button will reset the timeout.

The most likely cause of this error is a loss of power to the VibrationVIEW I/O unit, either because it is turned off, or because of a temporary power outage.

Other events that could cause this to happen are:

the VibrationVIEW I/O unit is turned on after the program is started.
the serial cable connecting the computer with the VibrationVIEW I/O unit comes loose.
the computer freezes.
the computer crashes.

23.5.13. Changing Level (Shock Stop Code)

This stop code indicates that the test is finishing one level and starting another. The test continues to run.

23.5.14. 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.

23.5.15. Starting with Memorized Drive (Shock Stop Code)

This stop code indicates that the test is starting with a memorized drive.

The option can be turned on by checking the box next to "Enable memorized drive" in the Shock Level Schedule dialog box.

In order to set the current drive you must have a test started. Then go to the Test Save Current Drive menu command. By doing this you can start a test where you last saved it (the timer starts over).

23.5.16. Test Parameters Changed (Shock Stop Code)

This code indicates that the test parameters were changed, using either the Test..Edit Test or the Test..New Test menu commands, or their respective toolbar buttons. As a result of the test parameter changes, the test was halted.

23.5.17. 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 either 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 specimen. 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.

23.5.18. Max Start System Gain (Shock Stop Code)

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. If necessary, 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 maximum start system gain limit will limit the rate at which the output increases, and this message will appear as a yellow notice code while this limiting is active.

23.5.19. 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 either 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 specimen. 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.

23.5.20. Max Run System Gain (Shock Stop Code)

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. If necessary, you may need to adjust the running max system gain

parameter, found in the Shock Parameters dialog box.

23.5.21. 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 that your shaker could not perform. If you need to continue with this acceleration you need a bigger shaker. Or you can tone down your test by setting the Modifier to a lower value in the Shock Schedule dialog box.

23.5.22. 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 that your shaker could not perform. If you need to continue with this velocity you need a bigger shaker. Or you can tone down your test by setting the Modifier to a lower value in the Shock Schedule dialog box.

23.5.23. 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 that your shaker could not perform. If you need to continue with this displacement you need a bigger shaker. Or you can tone down your test by setting the Modifier to a lower value in the Shock Schedule dialog box.

23.5.24. 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.

23.5.25. 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 adjusting the controller parameters to improve the control.

23.5.26. 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 +/- 2.75 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	270 G peak
20 mV/G	135 G peak
50 mV/G	54 G peak
100 mV/G	27 G peak
1000 mV/G	2.7 G peak

23.6. Field Data Replicator Stop Codes

These are status messages, to let you know what the controller is doing.

Starting Test	Running Test	Stop Button Pressed
Front Panel Start	Front Panel Stop	Remote Start
Remote Stop	Starting Schedule	Changing Level
End of Test	Starting With Memorized Drive	Waiting for operator

These are stop codes, telling you why the controller stopped the test. A brief description follows explaining what to do about the problem.

Test Parameters Changed	Max Starting Voltage	Max Run Voltage
Maximum G RMS (ChN)	Doing Digital Record	Input channel N is clipping
ChN Exceeded Sine Acceleration Limit		Didn't Reach Demand Level
ChN Exceeded Random Acceleration Limit		Emergency Stop
ChN Exceeded Shock Acceleration Limit		Invalid Remote Test

23.6.1. 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.

23.6.2. 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.

23.6.3. Stop Button Pressed (Field Data Replicator Stop Code)

This code indicates that you clicked the "Stop" button, telling the test to shut down.

23.6.4. Front Panel Start (Field Data Replicator Stop Code)

This code indicates that the test was started using the "Start" switch on the front panel of the VibrationVIEW I/O box. 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.

23.6.5. Front Panel Stop (Field Data Replicator Stop Code)

This code indicates that the test was stopped using the "Stop" switch on the front panel of the VibrationVIEW I/O unit. When this code is displayed the test is complete and the output is turned off.

23.6.6. 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.

23.6.7. Remote Stop (Field Data Replicator Stop Code)

This code indicates that the test was stopped using the Remote Start/Stop option.

23.6.8. 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".

23.6.9. Emergency Stop (Field Data Replicator Stop Code)

This code indicates that the emergency stop input was pressed, causing the controller to shut down. The emergency stop input is an optional accessory that connects to the remote input terminal block on the rear of the VibrationVIEW I/O unit. This option is enabled and disabled using the "Emergency stop input" selection in the VibrationVIEW Configuration Parameters dialog box.

23.6.10. 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. Clicking on the "Run" button will reset the timeout.

The most likely cause of this error is a loss of power to the VibrationVIEW I/O unit, either because it is turned off, or because of a temporary power outage.

Other events that could cause this to happen are:

the VibrationVIEW I/O unit is turned on after the program is started.
the serial cable connecting the computer with the VibrationVIEW I/O unit comes loose.
the computer freezes.
the computer crashes.

23.6.11. Starting Schedule (Field Data Replicator Stop Code)

This code indicates that this test is one of several tests listed in a test schedule, and was started using the Test Schedule dialog box.

23.6.12. Changing Level (Field Data Replicator Stop Code)

This code indicates that the test is finishing one level and starting another. The test continues to run.

23.6.13. Waiting for operator (Field Data Replicator 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" 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.

23.6.14. 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.

23.6.15. Starting With Memorized Drive (Field Data Replicator Stop Code)

This code indicates that the test is starting with a memorized drive.

The option can be turned on by checking the box next to "Enable memorized drive" in the Field Data Replicator Level Schedule dialog box. You can either start with 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 and then go to the Test..Save Current Drive menu command. By doing this you can start a test where you last saved it (the timer starts over).

23.6.16. Test Parameters Changed (Field Data Replicator Stop Code)

This code indicates that the test parameters were changed, using either the Test..Edit Test or the Test..New Test menu commands, or their respective toolbar buttons. As a result of the test parameter changes, the test was halted.

23.6.17. 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. If necessary, you may need more voltage. Modify the appropriate parameter found in the Field Data Replicator Limits dialog box.

23.6.18. Max Run 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. If necessary, you may need more voltage. Modify the appropriate parameter found in the Field Data Replicator Limits dialog box.

23.6.19. ChN Exceeded Sine Acceleration Limit (Field Data Replicator 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.

23.6.20. ChN Exceeded Random Acceleration Limit (Field Data Replicator 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.

23.6.21. ChN Exceeded Shock Acceleration Limit (Field Data Replicator 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.

23.6.22. Maximum G RMS (chN) (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 is off, or your amplifier is not turned on. If necessary, 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 must have a bigger spike in your data than you thought. Or, the channel two sensitivity is not set correctly. If necessary, you may need more Gs. Modify the appropriate parameter found in the the Field Data Replicator Limits dialog box.

23.6.23. Doing Digital Record (Field Data Replicator Stop Code)

This code indicates that the program is recording the data from the input channel.

You can get to this in the Field Data Replicator Define dialog box by clicking the "Record..." button, selecting a file to record to and pressing the "Start" button in the Digital Record dialog box.

23.6.24. Input channel N is clipping (Field Data Replicator Stop Code)

This code indicates that the input signal to channel N is exceeding the allowable input voltage range of +/- 2.75 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.

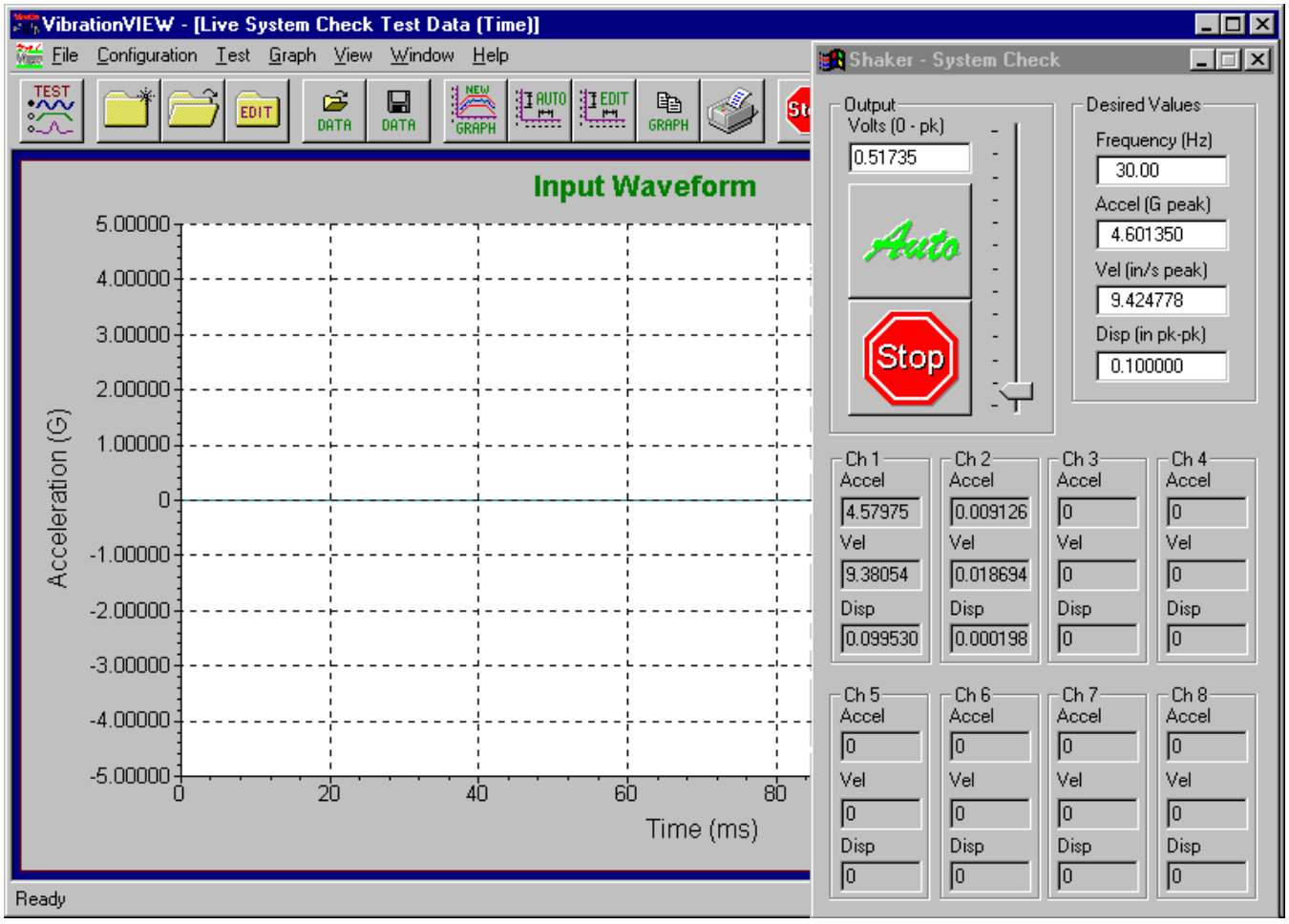
Sensitivity	**Maximum Acceleration**
10 mV/G	270 G peak
20 mV/G	135 G peak
50 mV/G	54 G peak
100 mV/G	27 G peak
1000 mV/G	2.7 G peak

23.6.25. Didn't Reach Demand Level (Field Data Replicator Stop Code)

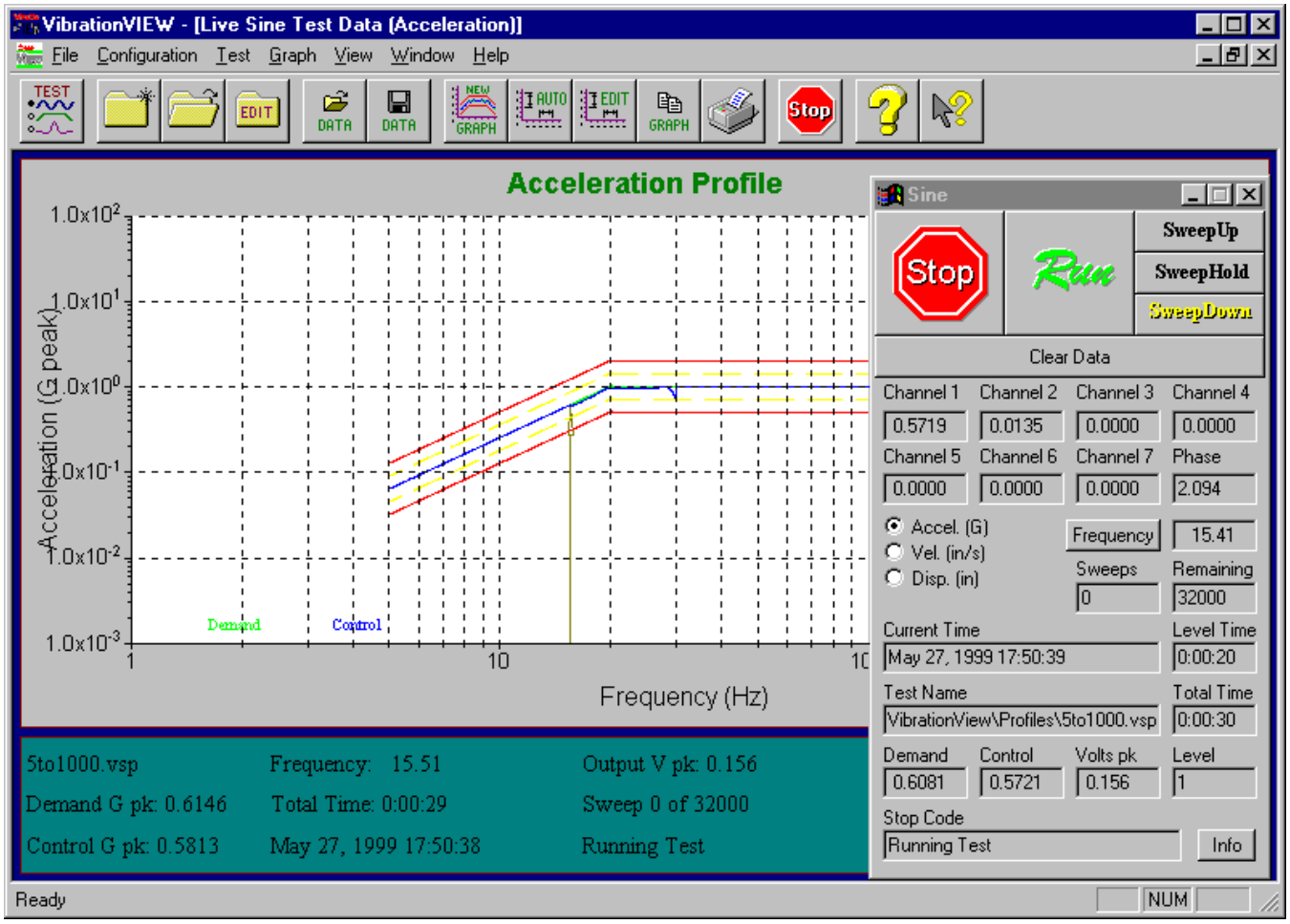
This code indicates that the controller was either starting a test or switching test levels, and failed to reach the demand level within 30 minutes. The controller should, even in the worst case, be able to reach the demand level within a few minutes. If you see this error, it is likely that the controller parameters are set incorrectly. See the section "How to tune Field Data Replicator controller parameters" for information on adjusting the controller parameters to improve the control.

24. Typical Views

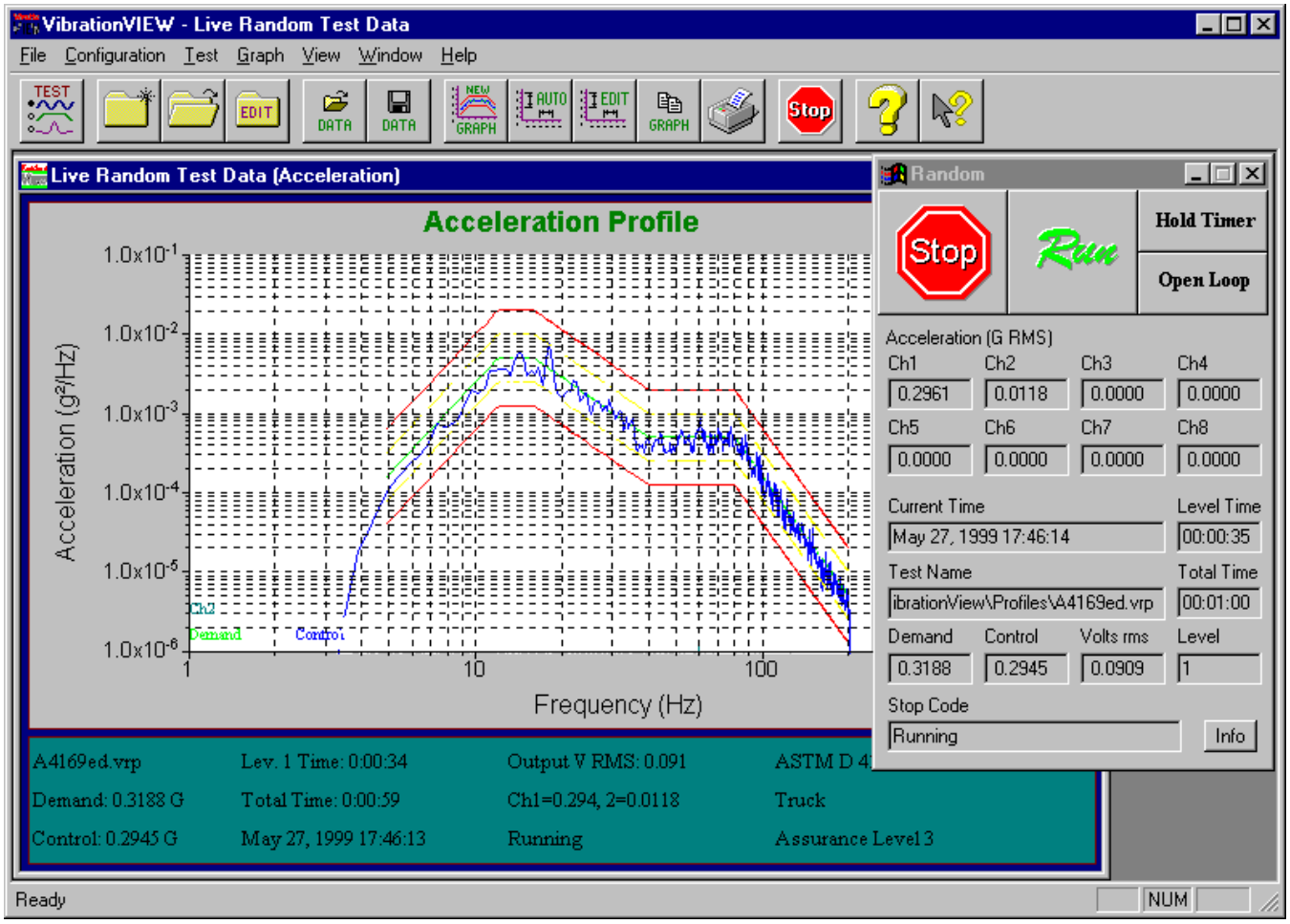
24.1. System Check CRT View



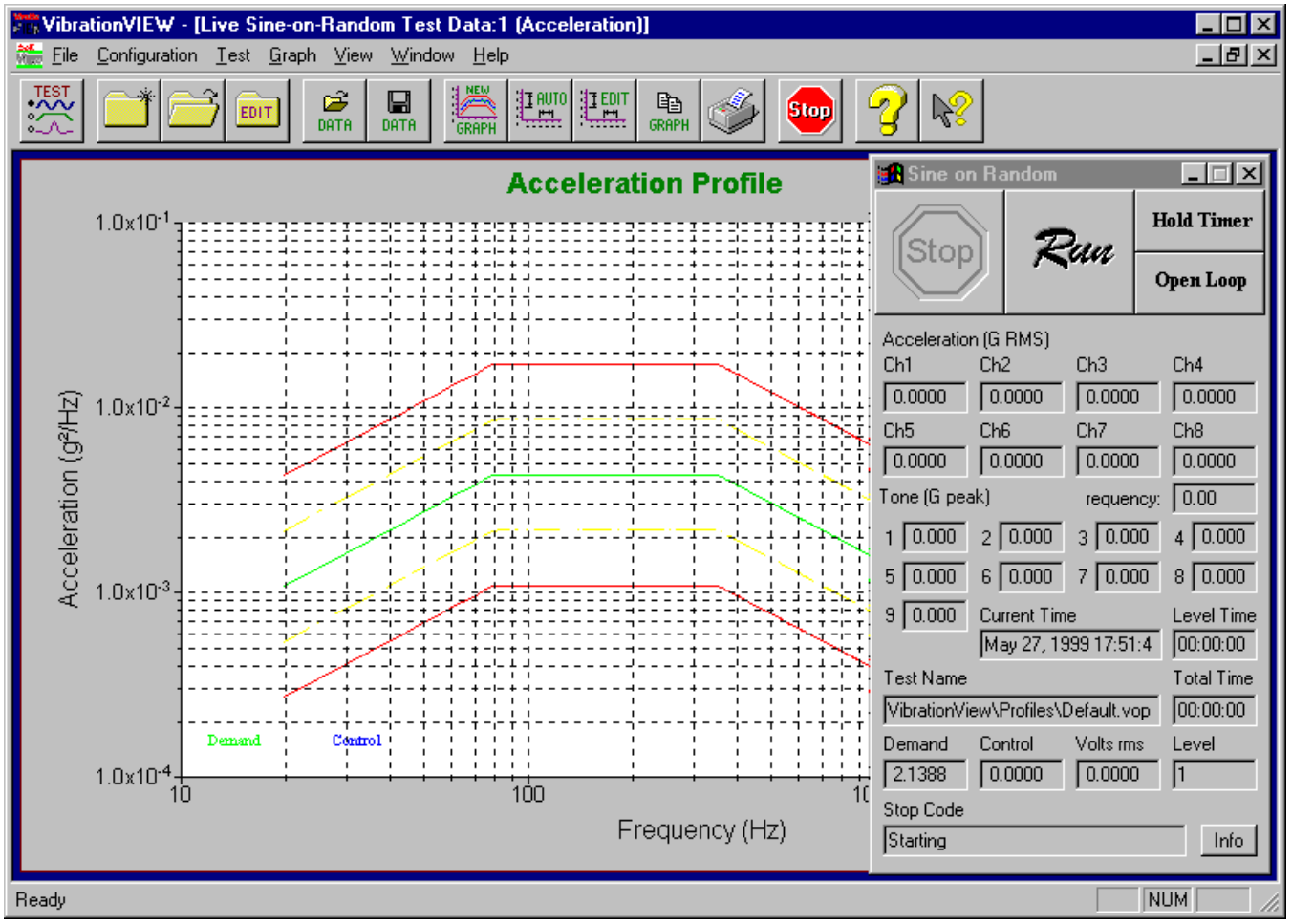
24.2. Sine CRT View



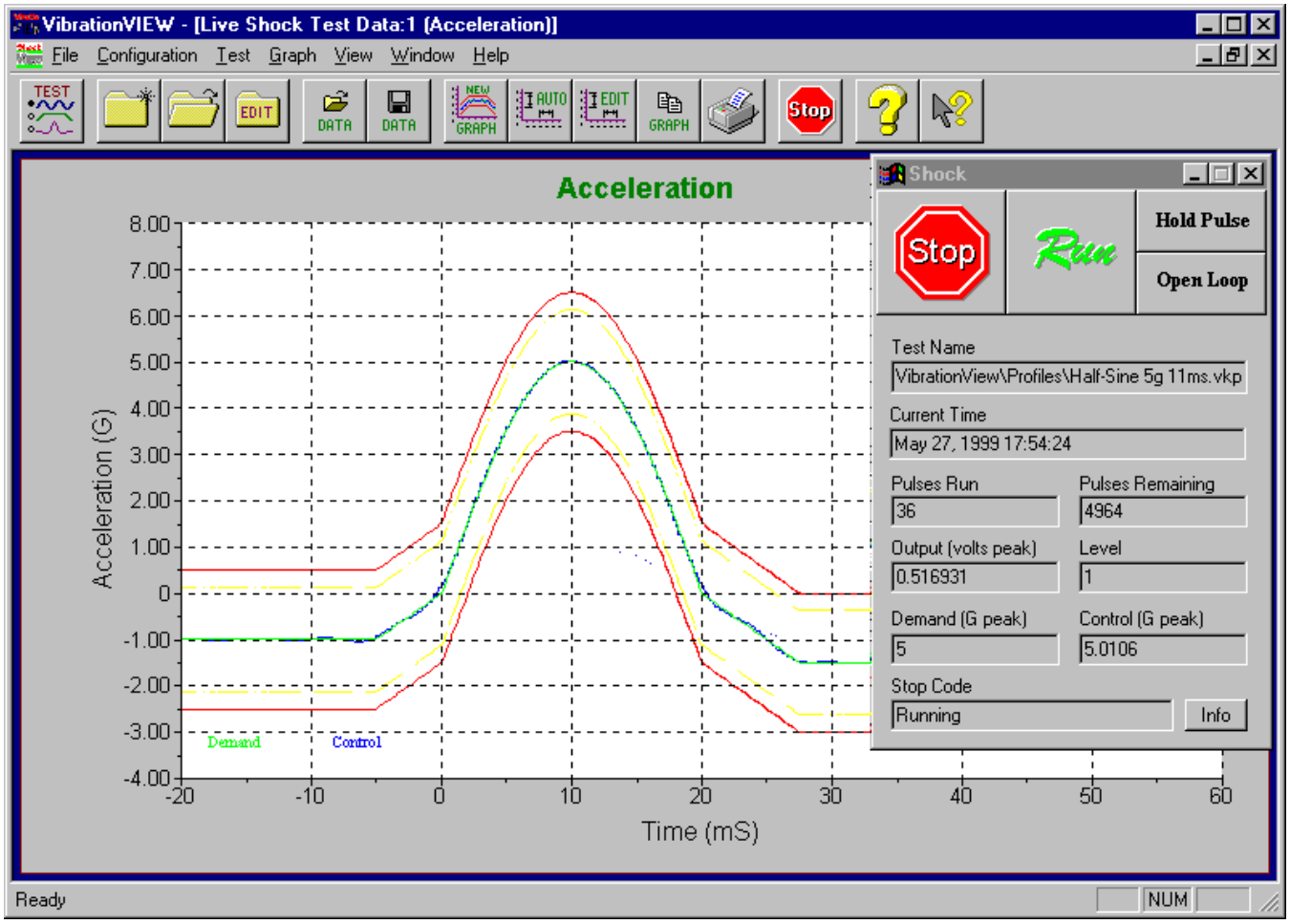
24.3. Random CRT View



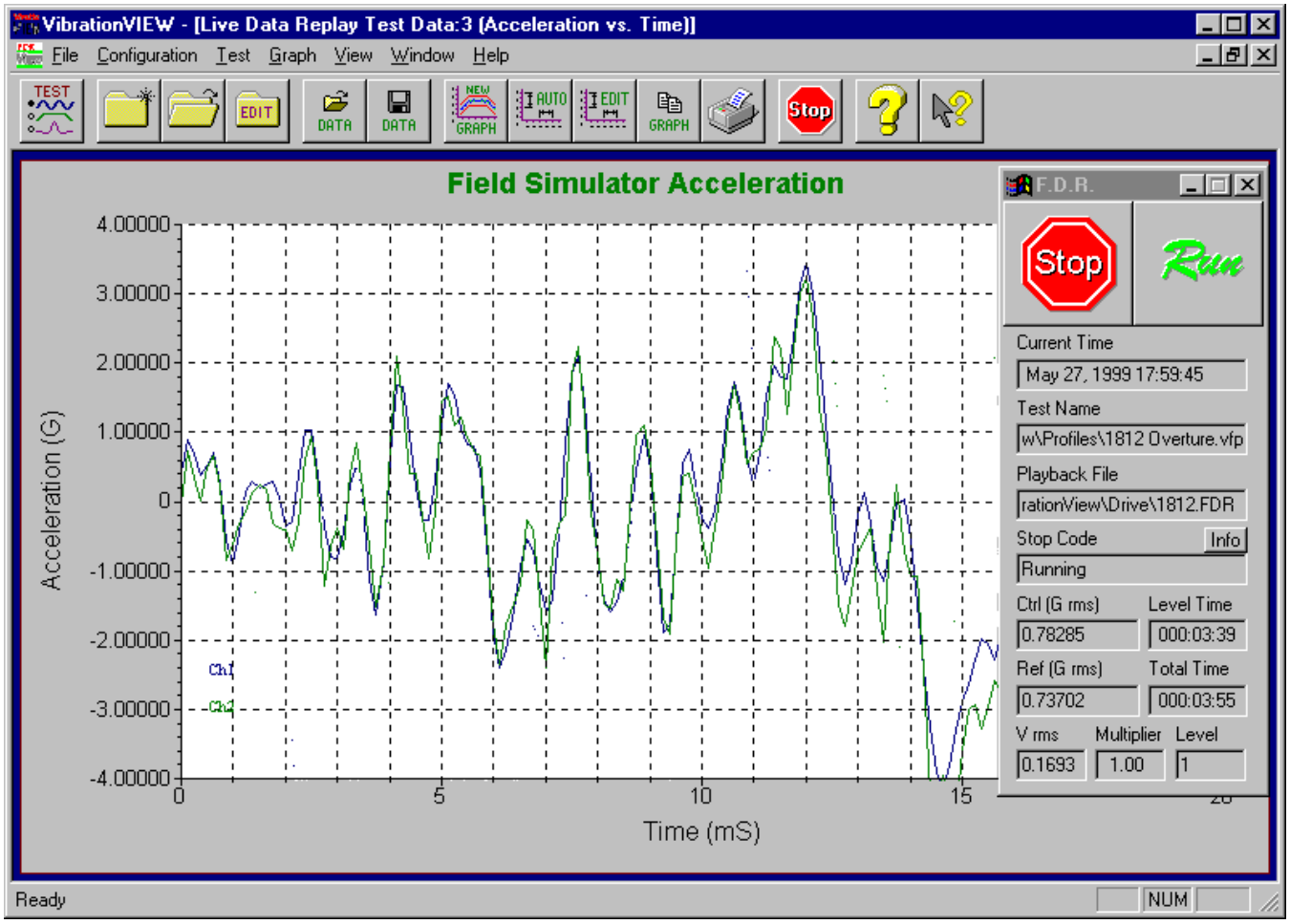
24.4. Sine-on-Random CRT View



24.5. Classical Shock CRT View

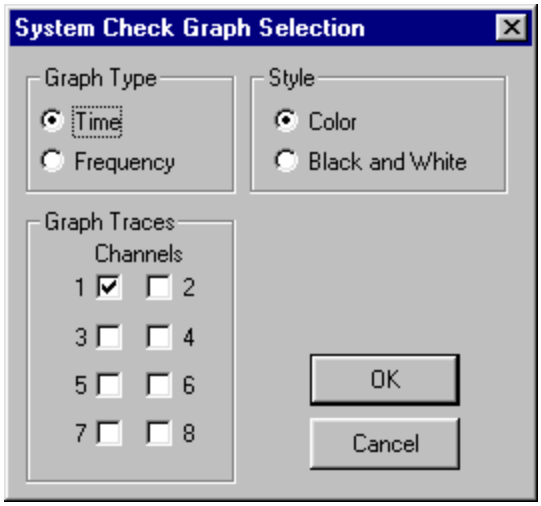


24.6. Field Data Replicator CRT View



25. Other Dialog Boxes

25.1. System Check Graph Settings dialog box



25.2. System Check Graph

Use the Graph..Edit Graph menu command to change the settings of this graph.

25.3. Define Sweep Rate dialog box

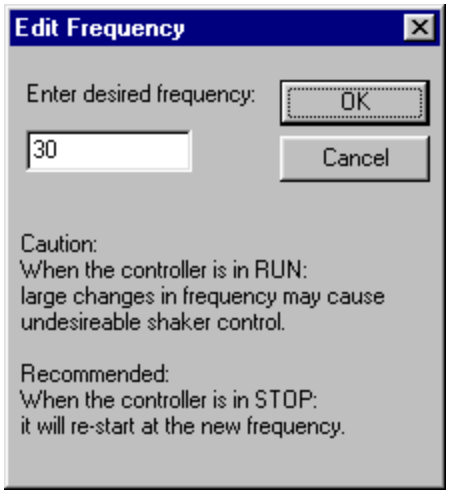
This dialog box is accessed by clicking the "Sweep..." button in the Sine Define dialog box.

Use this dialog box to set your sweep rate. Enter the rate in the "Speed" box, and click the radio button for the desired units.

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 run the number of cycles (sweeps) specified. Example: at 10 Hz fixed frequency test, a 1000 "sweep" test would run for 1000/10=100 seconds.

25.4. Edit Frequency dialog box

This dialog box is accessed by clicking the "Frequency" button in the Sine Control Center.



Use this dialog box to set the current and/or the starting frequency in Sine Tests.

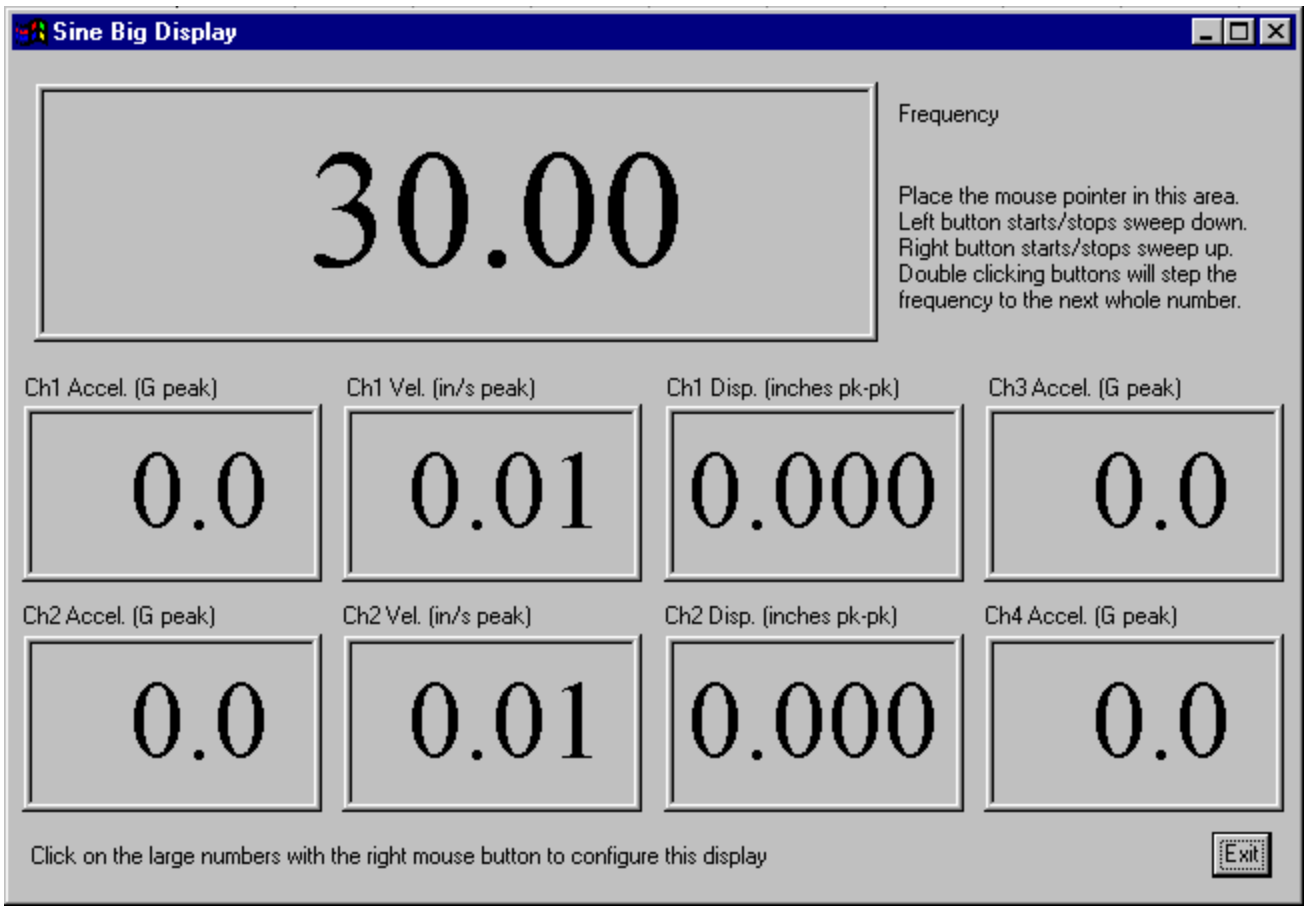
When the controller is running, changing the frequency value will put the controller in Hold mode and then modify the frequency value to the value set with this dialog box, and remain in sweep

hold mode at the new frequency.

When the controller is stopped, changing the frequency value will change the starting frequency for the sweep.

25.5. Sine Big Display dialog box

This dialog box is accessed by selecting the Graph..Sine Big Display menu command.



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.

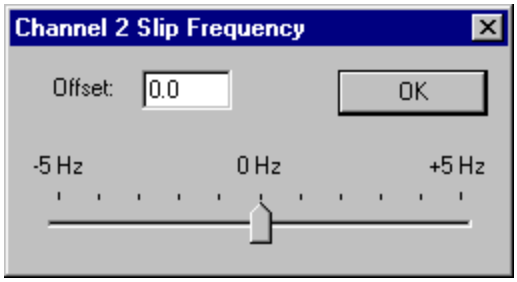
This dialog box also provides a feature for remote control (using the mouse) of the sweep direction. Move the cursor into the region in the upper right corner of the dialog box (the area with the small text) and then use the mouse buttons as follows:

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.

When used with a mouse extension cable, the sine frequency may be controlled from a distance.

25.6. Channel 2 Slip Frequency dialog box

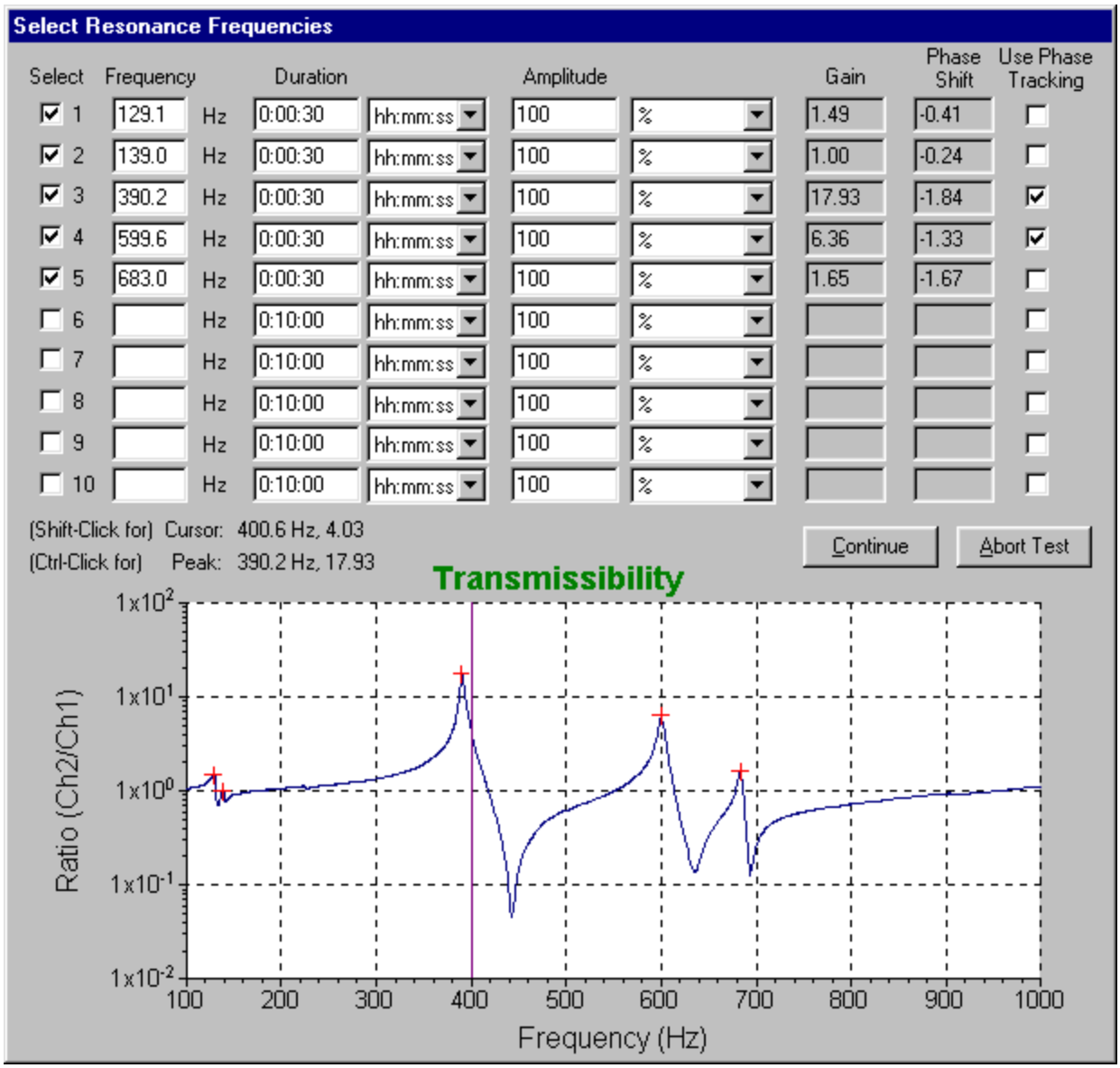
This dialog box is accessed by selecting the Graph..Sine COLA Slip menu command.



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.

25.7. Select Resonance Frequencies dialog box



This dialog box appears when a test is running and a "Resonance Table" level is reached in the 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.

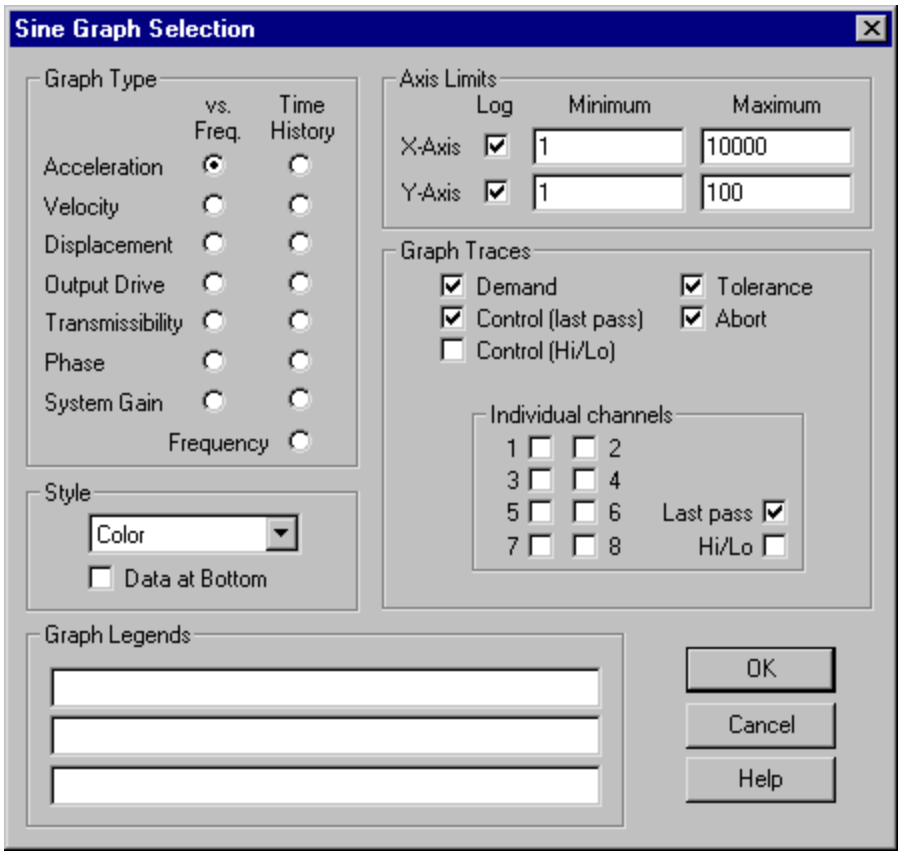
For resonant frequencies that exhibit a 180-degree change in phase between input channels 1 and 2 (typical of sharp resonance peaks), the phase tracking selection will be checked by default. When "Use Phase Tracking" is enabled, the controller will automatically adjust the output frequency to keep the 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.

25.8. Sine Graph Settings dialog box

This dialog box is used when creating a new Sine graph using the File..New Graph or Window..New Window menu commands, when changing the settings of the selected graph using the Graph..Edit Graph Settings menu command, or when setting the default graph settings by clicking the "Graph" button in the Sine Define dialog box.

In the Graph Type selection area, the first column selects plots of the data vs. frequency. The second column selects plost of the data vs. time.

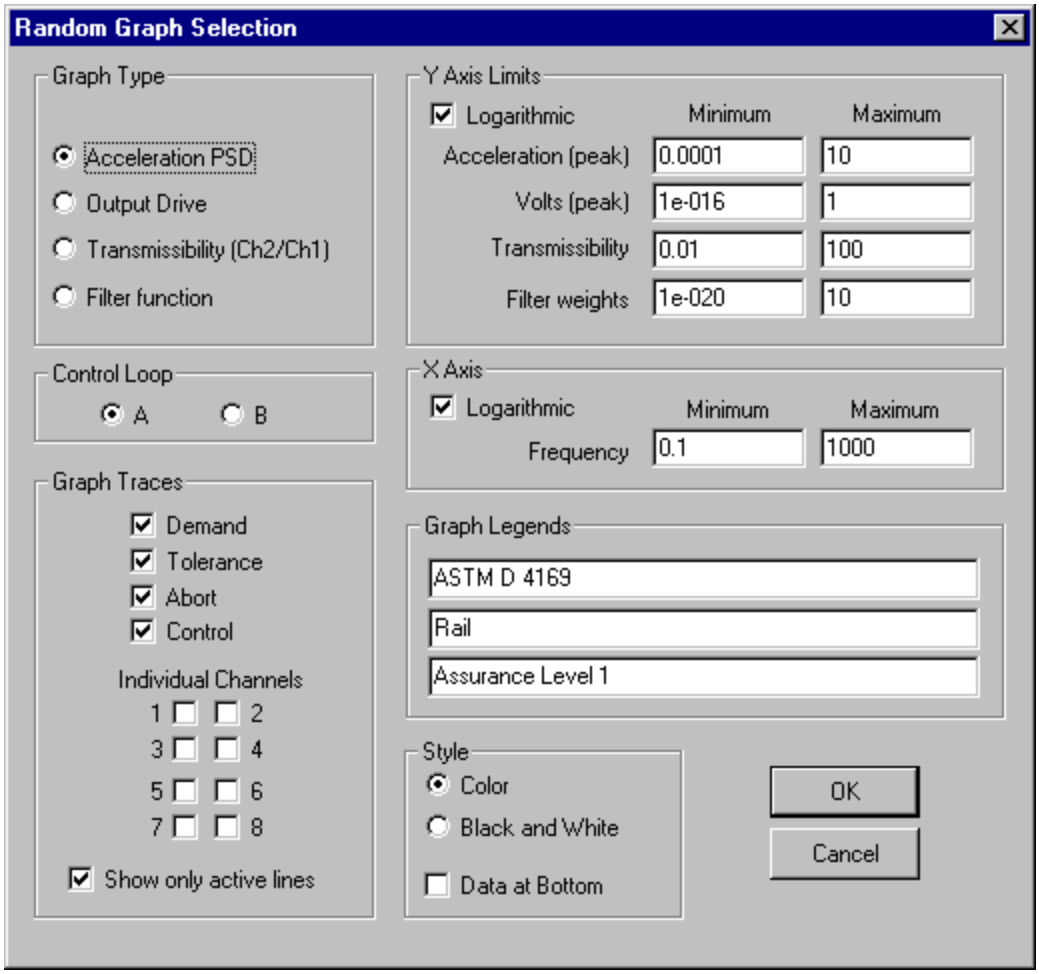


25.9. Sine Graph

Select the Graph..Edit Graph menu command to change the settings of this graph.

25.10. Random Graph Settings dialog box

This dialog box is used when creating a new Random graph by selecting the File..New Graph or Window..New Window menu commands or when changing the settings of the selected graph by selecting the the Graph..Edit Graph Settings menu command.



25.11. Random Graph

Select the Graph..Edit Graph menu command to change the settings of this graph.

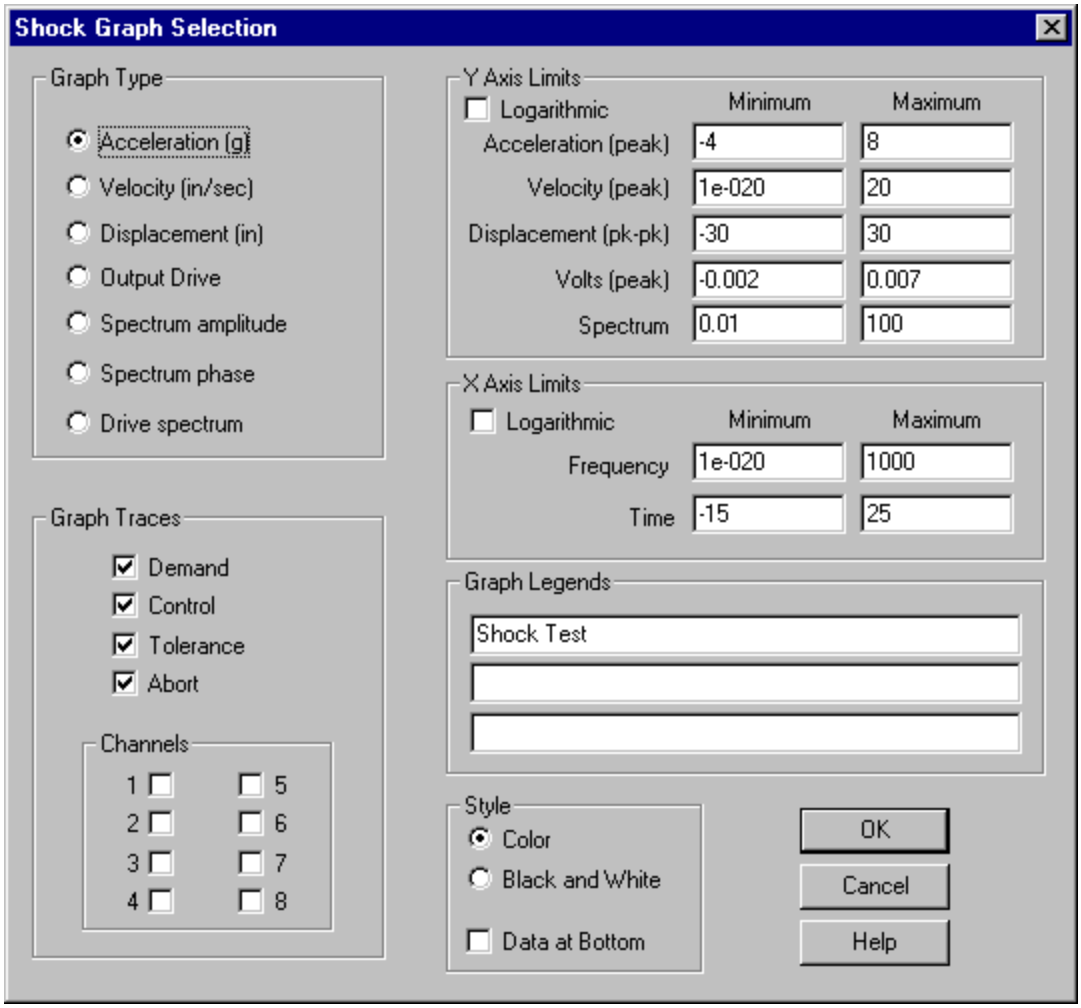
25.12. Sine-on-Random Graph

Use the Graph..Edit Graph menu command to change the settings of this graph.

25.13. Shock Graph Settings dialog box

This dialog box is used when creating a new Shock graph by selecting the File..New Graph or Window..New Window menu commands, when changing the settings of the selected graph by

selecting the Graph..Edit Graph Settings menu command, or when setting the default graph settings by clicking the "Graph" button in the Classical Shock Define dialog box.

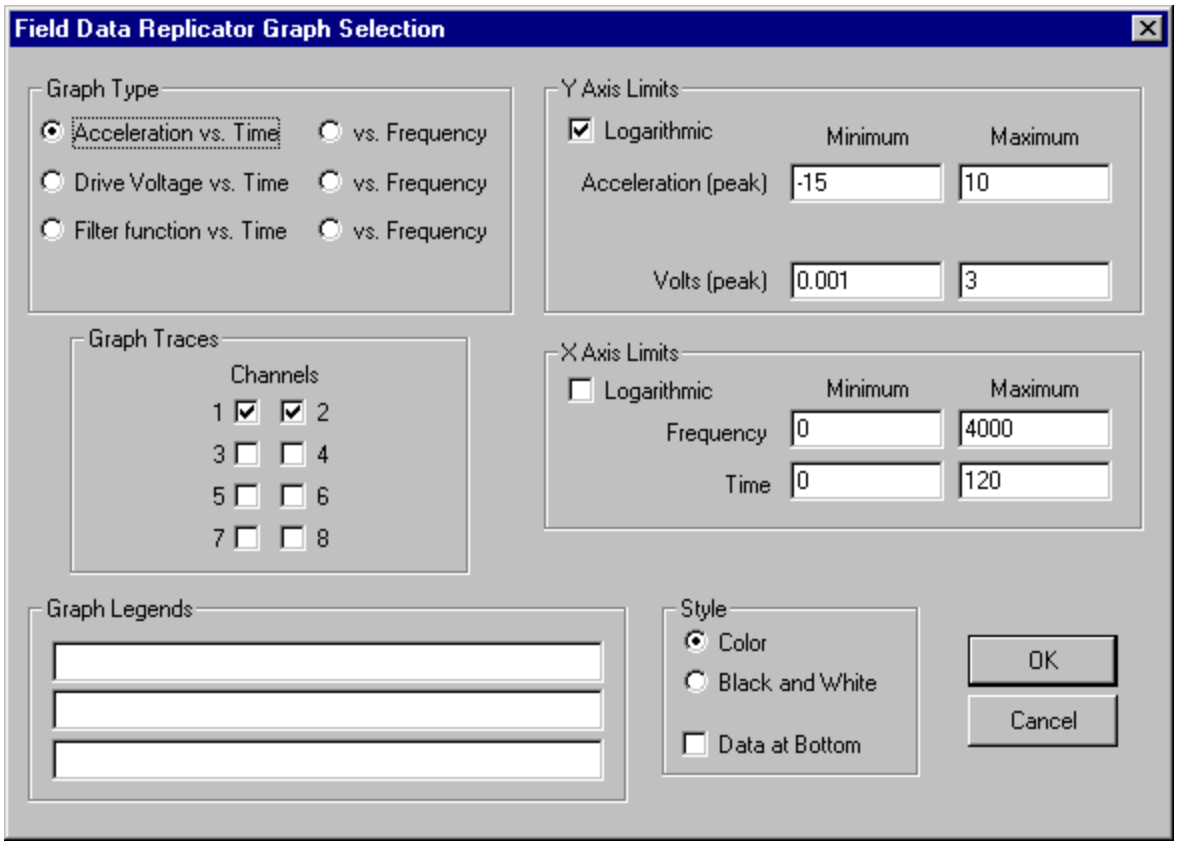


25.14. Shock Graph

Use the Graph..Edit Graph menu command to change the settings of this graph.

25.15. Field Data Replicator Graph Settings dialog box

This dialog box is used when creating a new Field Data Replicator graph by selecting the File..New Graph or Window..New Window menu commands, when changing the settings of the selected graph by selecting the Graph..Edit Graph Settings menu command, or when setting the default graph settings by clicking the "Graph" button in the Field Data Replicator Defi ne dialog box.

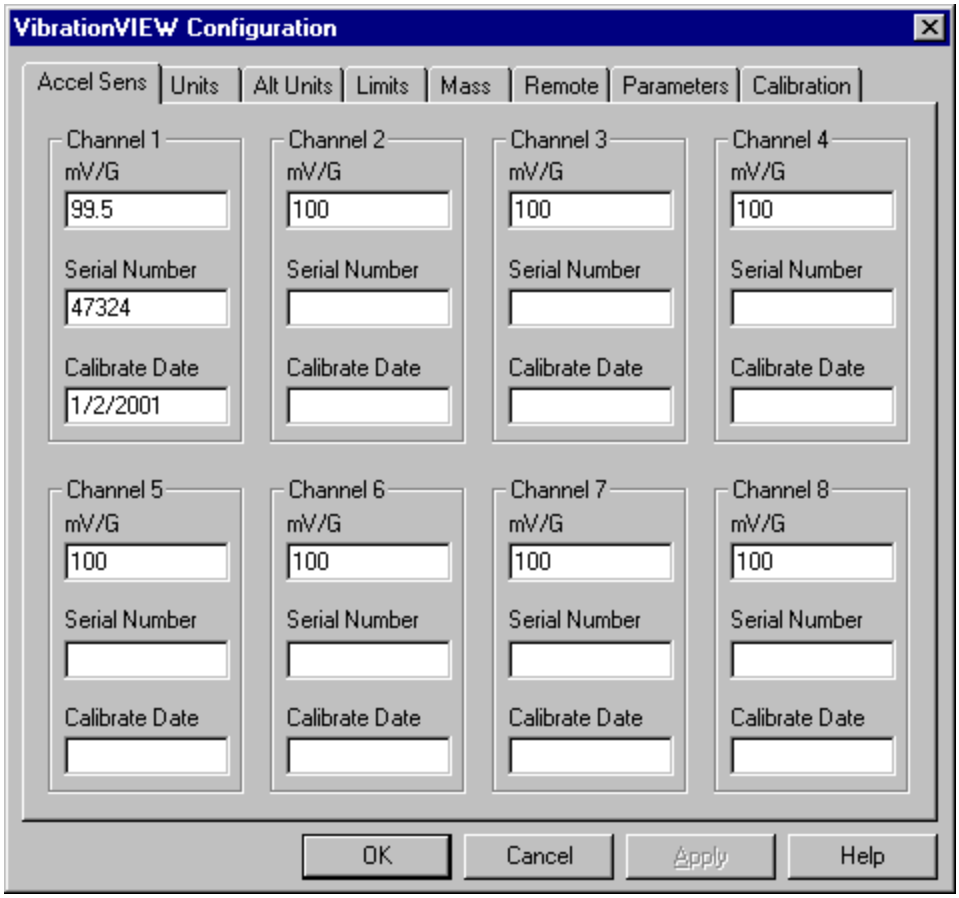


25.16. Field Data Replicator Graph

Use the Graph..Edit Graph menu command to change the settings of this graph.

25.17. Accelerometer Sensitivity dialog box

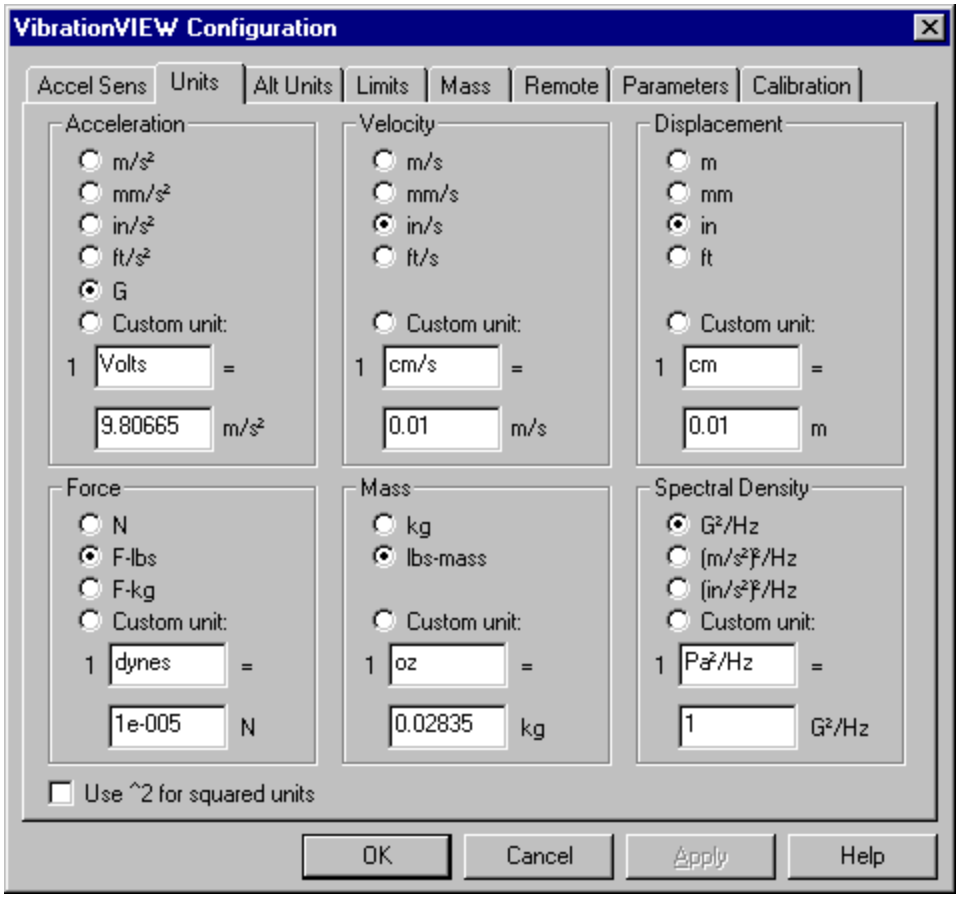
This dialog box is accessed by selecting the Configuration..Accelerometer Sensitivity menu command.



Use this dialog box to enter the calibrated sensitivity ratings for your accelerometers in mV/G. 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."

25.18. System Units dialog box

This dialog box is accessed by selecting the Configuration..Units menu command.



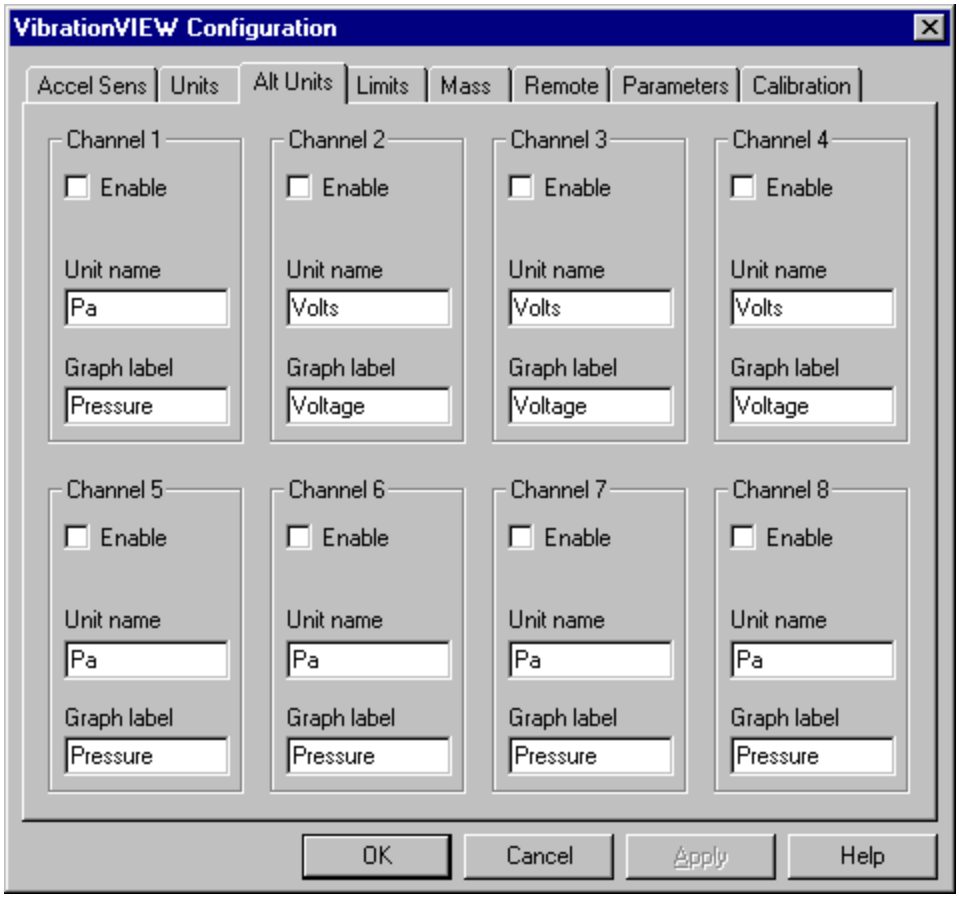
Use this dialog box to select and define the desired units to be used by the program. Changing the units will stop any active test.

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 SI unit in the lower box. The unit label and scale factor must make the equation correct. For example:

1 oz = 0.02835 kg

25.19. Alternate Units dialog box

This dialog box is accessed by selecting the Configuration..Alt Units menu command.



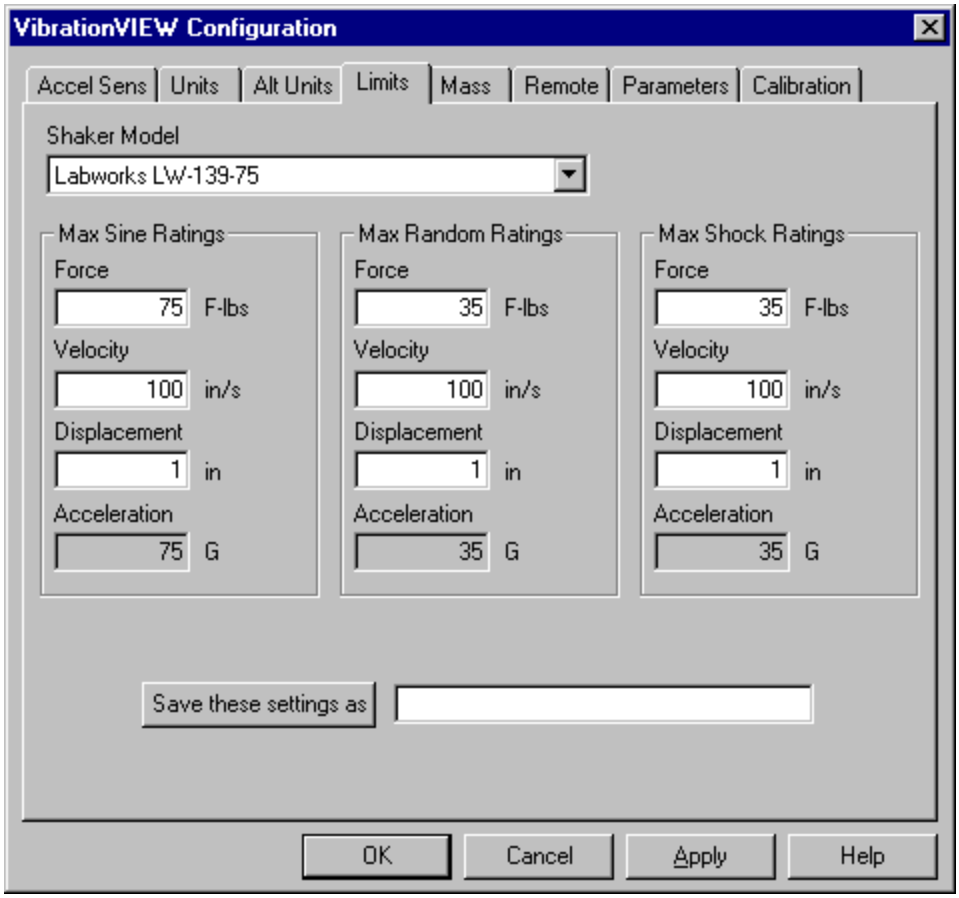
Use this dialog box to select and define the alternate units to be used by the program. Changing the units will stop any active test.

Note: Alternate Units is an optional feature that requires that the software key be programmed at the factory. If you haven't purchase the Alternate Units option, you may ignore this configuration dialog box.

Alternate Units can be used to define special units for individual input channels that are independant of the primary acceleration, velocity, and displacement units. When a channel is configured using an Alternate Unit, the input scaling of that unit is specified in the Accelerometer Sensitivity dialog box, in terms of mV/Unit, where Unit is the defined alternate unit. Alternate Units can be used for control, but acceleration/velocity/displacement conversions are not available. For example, you could connect a strain gauge, and define an alternate unit for the strain value.

25.20. System Limits dialog box

This dialog box is accessed by selecting the Configuration..System Limits menu command.

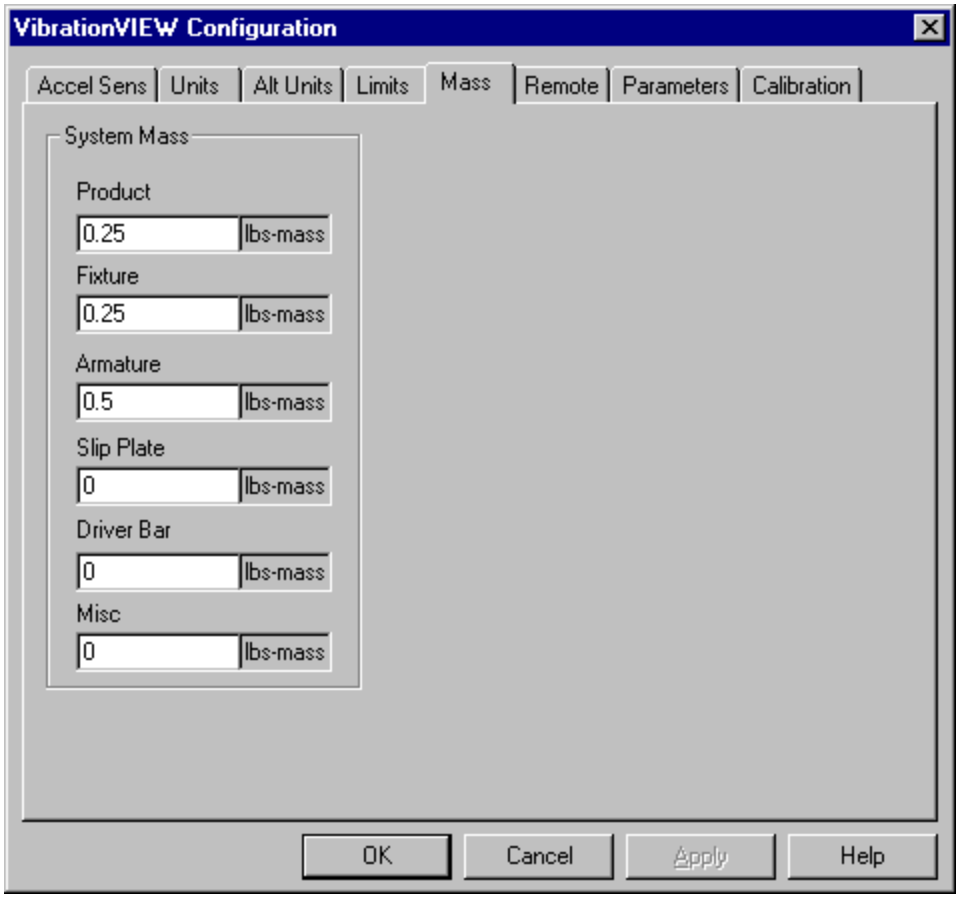


Use this dialog box to configure the limits for your shaker system. These parameters are checked at run time to determine if the test you are about to run is within the specifications of your shaker system. The acceleration and velocity limits are measured 0-to-peak while displacement limits are measured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

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

25.21. System Mass dialog box

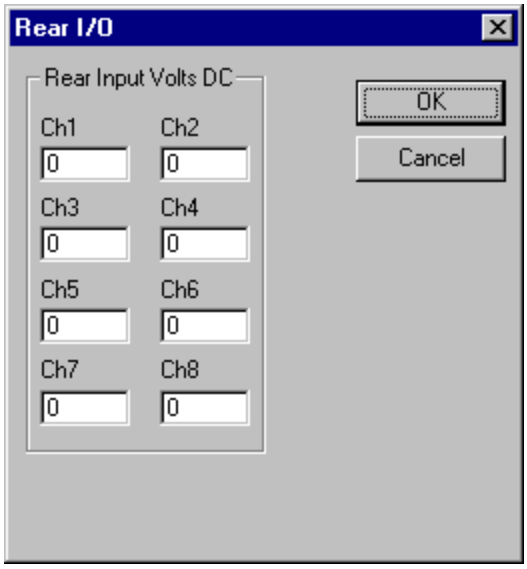
This dialog box is accessed by selecting the Configuration..System Mass menu command.



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 the test can be run within the specifications of your shaker system. The units used for the masses are selected under the System Units tab.

25.22. Rear I/O dialog box

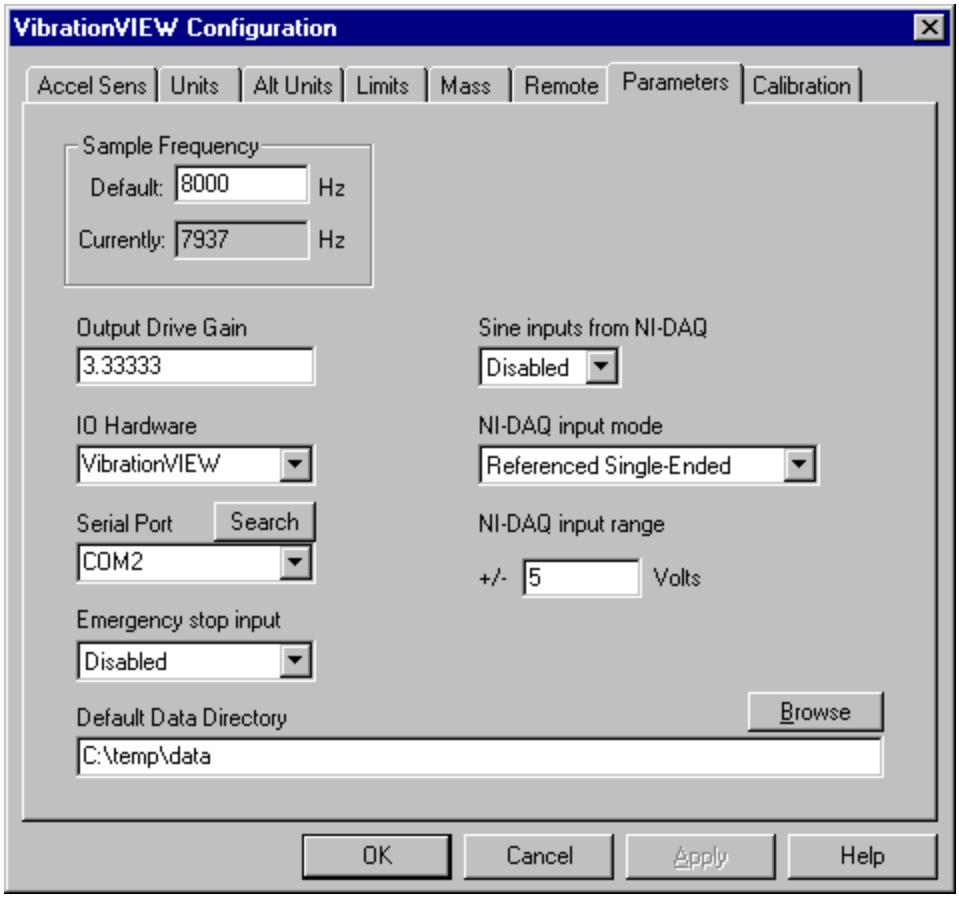
This dialog box is accessed by clicking the Rear I/O button in the System Calibration dialog box.



This dialog box is used only for diagnostic purposes. It displays the input values for the 8 discrete inputs on the rear of the VibrationVIEW I/O unit.

25.23. Parameters dialog box

This dialog box is accessed by selecting the Configuration..Parameters menu command.



Use this dialog box to set global system parameters including the type of I/O hardware and the serial port used by the controller. This dialog box also may be used to set the default sampling rate, and to see the current sampling rate used by the controller.

Most systems will use the settings shown in the dialog above, although the serial port might be different. If you are not sure of the correct serial port setting, turn on your VibrationVIEW I/O unit, and then click the Search button to have the software automatically detect the proper serial port.

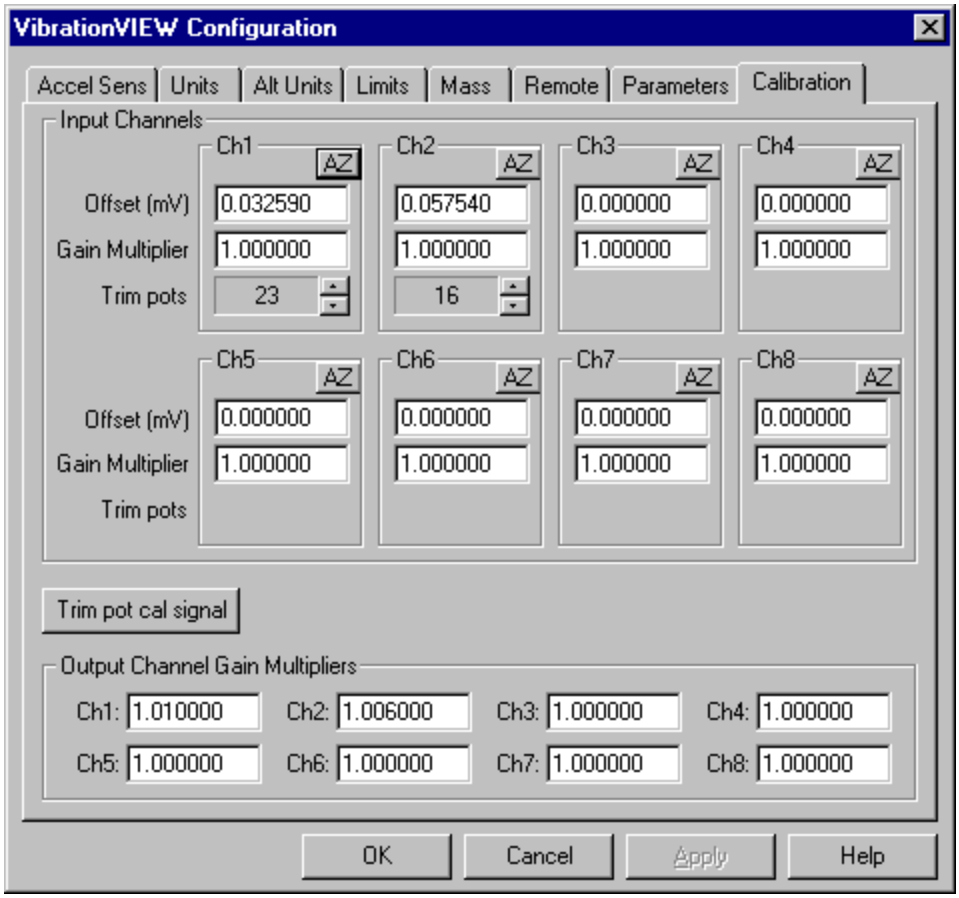
The emergency stop input is an optional feature (part of the Remote Input package) that connects to Discrete Input 1 on the rear of the VibrationVIEW I/O unit. When this input is low (below 2 volts) the outputs of the I/O unit are immediately brought to 0 volts to stop the drive signal going to the amplifier. Typically this input is connected to +5V through a normally closed red mushroom button. When the mushroom button is pressed, the connection to +5V is opened, and Digital Input 1 is pulled to ground by a pull-down resistor inside the VibrationVIEW I/O unit. This causes the shaker drive signal to be immediately shut down.

The "Sine Inputs from NI-DAQ" option is available only if you have purchased the VibrationAnalyzer software option. With this option you get extra monitor channels in sine mode using a National Instruments data acquisition card. Select either "Enabled" or "Disabled" to enable/disable these inputs in Sine mode. The input mode must be set to match the manner in which the inputs are connected. If you are using the BNC-2090 connector unit, there are switches to the left of each input which select Single-ended or Differential input modes (labeled

SE and DIFF), and also a switch to the right of ACH7 which selects Referenced or Non-referenced single-ended mode (labeled RSE and NRSE. Refer to the documentation for the National Instruments data acquisition card and for the BNC-2090 connector unit for details on the various input connection methods. Differential mode generally provides the cleanest input signals.

25.24. System Calibration dialog box

This dialog box is accessed by selecting the Configuration..System Calibration menu command.

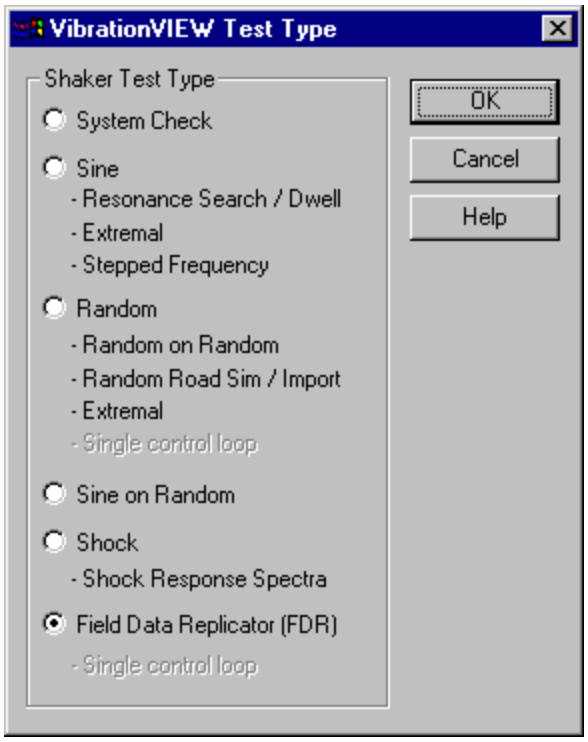


Refer to How to calibrate the system for details on how to use this dialog box to recalibrate the system.

25.25. VibrationVIEW Test Type dialog box

This dialog box is accessed by selecting the Test..Select Test Type menu command or by clicking 

on the toolbar.

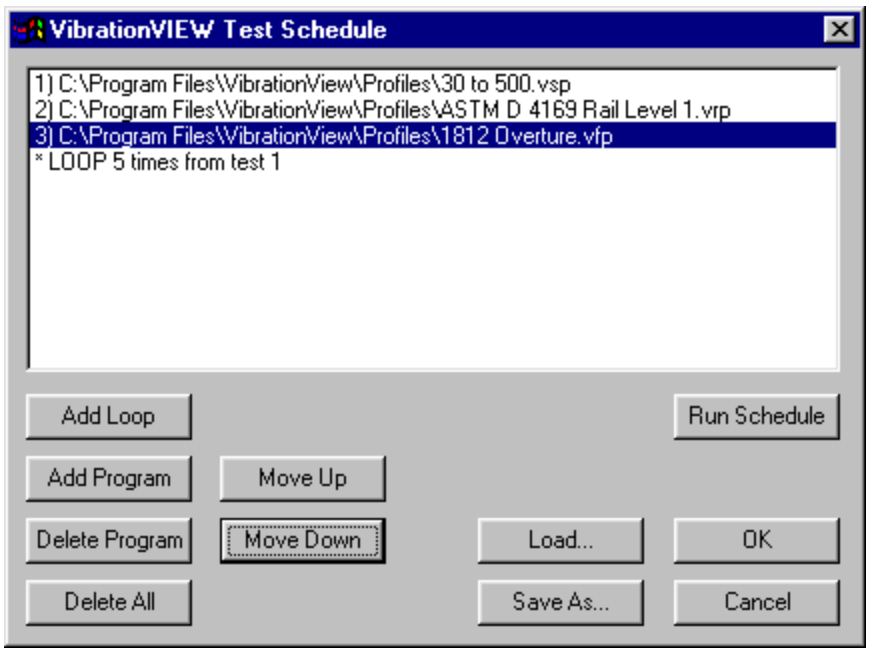


The different test types are software options that are protected by a hardware key. If your hardware key does not have an option enabled, you will still be able to switch to that mode, define profiles, and load and plot data sets, but you will not be able to run the test.

Contact Vibration Research Corp. for information on options that are not enabled.

25.26. Test Schedule dialog box

This dialog box is accessed by selecting the Test..Test Schedule menu command.



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 then 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 then click the "Delete Program" button. Click the "Delete All" button to clear all entries in the list.

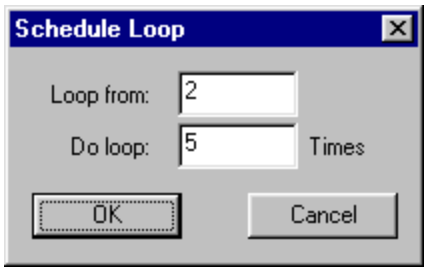
To repeat a sequence of tests multiple times, select the last test you want in the repeat sequence, and then click the "Add Loop" button. A Schedule Loop dialog will appear prompting you for the first test in the sequence and the number of times you want the sequence performed.

To change the settings of an existing loop, select loop in the schedule, and then 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.

25.27. Schedule Loop dialog box

This dialog box is accessed by clicking the "Add Loop" button in the VibrationVIEW Test Schedule dialog box.



Enter the test number of the first test in the sequence in the "Loop from" entry, and then the total number of times you want the sequence of tests performed in the "Do loop" entry. Then click the "OK" button to enter the loop parameters into the test schedule

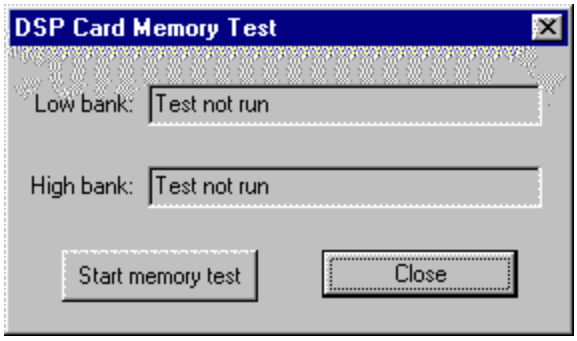
25.28. Cursor Display dialog box

This dialog box is accessed by selecting the Graph..Cursor Display menu command.

This dialog box will display the values of the graph data at the cursor position in the currently selected graph. The parameters displayed will be specific to the selected graph.

25.29. DSP Card Memory Test dialog box

This dialog box is accessed by selecting the Test..Test DSP Memory menu command.



Use this dialog box to verify that the DSP memory is operating properly, and that communication between the PC and the DSP card is working. This command is only available when running in System Check mode. In general, this command only needs to be used if you are instructed to do so by Vibration Research Corporation.

25.30. Graph Colors dialog box

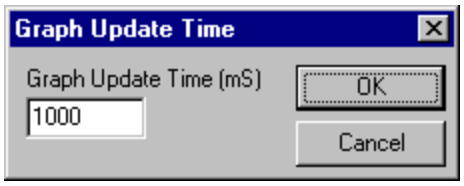
This dialog box is accessed by selecting the Graph..Graph Colors menu command.

Use this dialog box to set the default colors for various components of the graphs.

Changes made here 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, press Ctrl+G (Edit Graph Settings command), and then click the "OK" button.

25.31. Graph Update Time dialog box

This dialog box is accessed by selecting the Graph..Graph Update Time menu command.



The value specified here is the number of milliseconds between graph updates. The default value is 1000 ms (i.e. 1 second).

25.32. Message dialog box

This dialog box is used to provide status information for an operation that may take more than a couple of seconds.

25.33. 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 this dialog disappears

before continuing.

25.34. Wait for operator dialog box

This dialog box appears when the test is running, and it reaches a "Wait for operator" level in the test schedule. The test will pause 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 will abort the current test.

25.35. 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.

25.36. Key Not Found notice

This notice is given when you run a test test which is not enabled in the key, or a test which was set for demo mode, and the demo time has expired. If you get this error message, use the Help..About menu command to see the list of options enabled in the key.

The line underneath the date in the "About VibrationVIEW" dialog box gives the key number. If this line reads "No key found", then your key might have been disconnected. See 'How to install the hardware' for details on how to connect the key.

Note that the "Key not found" notice is also given when you run a test which 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, and you will be able to run the test.

25.37. Password Entry dialog box

This dialog box appears whenever a password-protected operation is performed. Type in the password, and click the "OK" button to continue.

A sheet of paper with the password instructions was attached to the manual you received with your system. Contact Vibration Research Corporation (616-669-3028) if you have forgotten the password.

Note: Use the following procedure to disable the password protection feature of the program:

1. Open the file 'C:\Program Files\VibrationVIEW\shaker.ini' using Notepad
2. Find the section labeled [Password]
3. This section should have two lines:

4. 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.

26. Windows Elements

26.1. Control Menu

26.1.1. Restore command (Control menu)

Use this command to return the active window to its size and position before you chose the Maximize or Minimize command.

26.1.2. 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

26.1.3. 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.

26.1.4. Minimize command (application Control menu)

Use this command to reduce the VibrationVIEW window to an icon.

Shortcut

Mouse: Click the minimize icon on the title bar.
Keys: ALT+F9

26.1.5. 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.

26.1.6. 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

26.1.7. 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

26.2. Title Bar

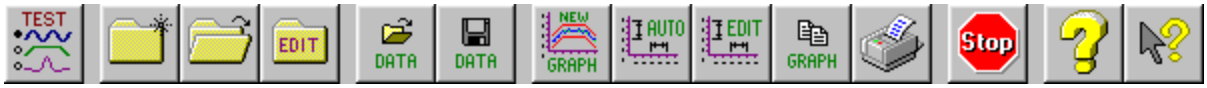
The title bar is located along the top of a window. It contains the name of the application and current document.

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
Minimize
Name of the application
Name of the document
Restore

26.3. 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 (Test..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)

Create a new graph (File..New Graph)

Autoscale the selected graph (Graph..Full Autoscale)

Edit the selected graph's settings (Graph..Edit Graph Settings)

Copy the selected graph to the windows clipboard (File..Copy Graph)


Print the selected graph (File..Print)

Stop the current test (Test..Stop Test)

Show the opening screen of the help file (Help..Help)

Context sensitive help (Help..What's This?)

26.4. 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 similarly 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 this controller machine (e.g. 0x0001).

26.5. 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.