

be certain.

STEX (Seismic Test Execution) Software User's Manual



Copyright information

© 2009, 2010 MTS Systems Corporation. All rights reserved.

Trademark information

MTS is a registered trademark of MTS Systems Corporation within the United States. These trademarks may be protected in other countries.

All other trademarks or service marks are property of their respective owners.

Proprietary information

Software use and license is governed by MTS's End User License Agreement which defines all rights retained by MTS and granted to the End User. All Software is proprietary, confidential, and owned by MTS Systems Corporation and cannot be copied, reproduced, disassembled, decompiled, reverse engineered, or distributed without express written consent of MTS.

Software validation and verification

MTS software is developed using established quality practices in accordance with the requirements detailed in the ISO 9001 standards. Because MTS-authored software is delivered in binary format, it is not user accessible. This software will not change over time. Many releases are written to be backwards compatible, creating another form of verification.

The status and validity of MTS's operating software is also checked during system verification and routine calibration of MTS hardware. These controlled calibration processes compare the final test results after statistical analysis against the predicted response of the calibration standards. With these established methods, MTS assures its customers that MTS products meet MTS's exacting quality standards when initially installed and will continue to perform as intended over time.

Publication information

MANUAL PART NUMBER	PUBLICATION DATE
100-216-955 A	August 2009
100-216-955 B	July 2010
100-216-955 C	November 2010

Contents

Introduction 9

STEX Software Capabilities	12
Group Options	12
Test Options	13
Analysis Options	13
Hardware Options	14
Help Options	14
Defining and Executing a Test	15
Acquisition Only Test	15
Model Measurement Test	15
Uncompensated Earthquake Testing	16
Compensated Earthquake Testing	17

STEX Operation 19

Starting the STEX Software	19
Quitting the STEX Software	19
STEX Main Dialog	20
Pulldown Menu Options	21
Tests	22
Analysis Results	23
STEX On-line Help	24

STEX Operation (continued)

Viewing Tests and Results	25
Viewing Test Parameters	25
Generating a Test Report	28
Viewing Result Parameters	28
Time History and Real Valued Spectra Statistics Dialog	29
Random Time History Statistics Dialog	31
Complex Spectrum Statistics Dialog	33
Response Spectrum Statistics Dialog	34
Vector Statistics Dialogs	36
Matrix Statistics Dialog	37
Model Statistics Dialog	39
Using the Plotting Option	41
Plot Setup Dialog	43
Using the Printing Option	47
Using the Select Option	48

Test Groups 51

Group Menu Options	52
Opening an Existing Group	53
Creating a New Group	54
Creating a Group Archive File	55
Restoring a Group Archive File	56
Deleting an Existing Group	57
Quitting the STEX Application	58

Tests 59

Test Types	60
Test Menu Options	61
Configuring New Tests	63
Test Setup Dialogs	68
Acquisition Only Test Setup Dialog	68
Model Measurement Test Setup Dialog	69
Time History and Compensated Time History Test Setup Dialog	70
Compensated Test Loopback	72
Executing Tests	74
Test Execution - No Data Acquisition Hardware	74
Test Execution - Data Acquisition Hardware	76
Reconciliation Dialogs	76
Control DOF Channel Discrepancy	77
Specimen Channel Discrepancy	77
Specimen Channel Discrepancy Dialog – Pacific Instruments Data Acquisition	77
Specimen Channel Discrepancy Dialog – MTS 498 and 493 Data Acquisition	80
Channels to Monitor Dialog	81
Pretest Checks Dialog	82
Test Excitation Dialog	86
Compensated Test Dialogs	88
Channel Setup Dialogs	93
Channel Setup – Pacific Instruments Data Acquisition	93
Channel Setup - MTS 498/493 Data Acquisition System	95
Viewing Test Parameters	101
Deleting a Test	104

Analysis Results 105

User vs. System Results	106
Result Types	107
Time Histories and Vectors	107
Spectra	107
Response Spectra	108
Matrices and Models	108
Analysis Menu Options	109
Creating a New Result	111
Creating Time Histories	112
Addition and Scaling of Time Histories	113
Multiplying Time Histories	114
Time History Differential Operations	115
Inverse Shock Time History Synthesis	117
Inverse FFT of Spectrum	120
Random Time History	121
Filtering a Time History	122
Creating a Sine Sweep	123
Tapering Time Histories	124
Concatenation of Time Histories	125
Harmonic Distortion	126
Kinematic Conversion	127
Correlation	128
Accel Conversion	129
Sine Waves	129
Adjust Time Base	131
Create Subrange	132

Analysis Results (continued)

Creating Spectra	133
Addition and Scaling of Existing Spectra	134
Multiplying Spectra	135
Spectra Differential Operations	136
Smoothing Spectra	137
Shock Response Calculation	138
FFT of an Existing Time History	139
Spectral Analysis of Existing Time Histories	140
Designing a Digital Filter	142
Editing Existing User Results	144
Editing Time History and Real Spectrum	145
Editing Complex Spectrum	146
Editing a Shock Response	148
Editing a Model	149
Deleting Results	150
Data Import and Export	151
Data Import	152
Copying Results From Another Group	154
Data Export	155
Exporting All Results	156
Exporting Multiple Results	157
Creating Shape Results	158
Creating a Model	162
Editing the Inverse Transfer Function	163

Naming Conventions 169

Random Signal Architecture 171

Model Creation 173

File Formats 177

STEX FILE FORMATS (Import/Export) 177

Example of real valued time history 179

Example of word valued time history: 180

Example of real valued spectrum: 180

Example of complex valued spectrum: 181

Example of complex valued spectrum: 182

Result Units 183

Introduction

Overview

MTS seismic test execution (STEX) software provides the setup, program definition, test execution and analysis capabilities required to conduct seismic tests using MTS seismic test systems.

This manual describes how to use the STEX software to define and execute compensated or uncompensated seismic tests on MTS seismic test systems. It is designed to provide both the experienced and first-time operator with information about the use of STEX software.

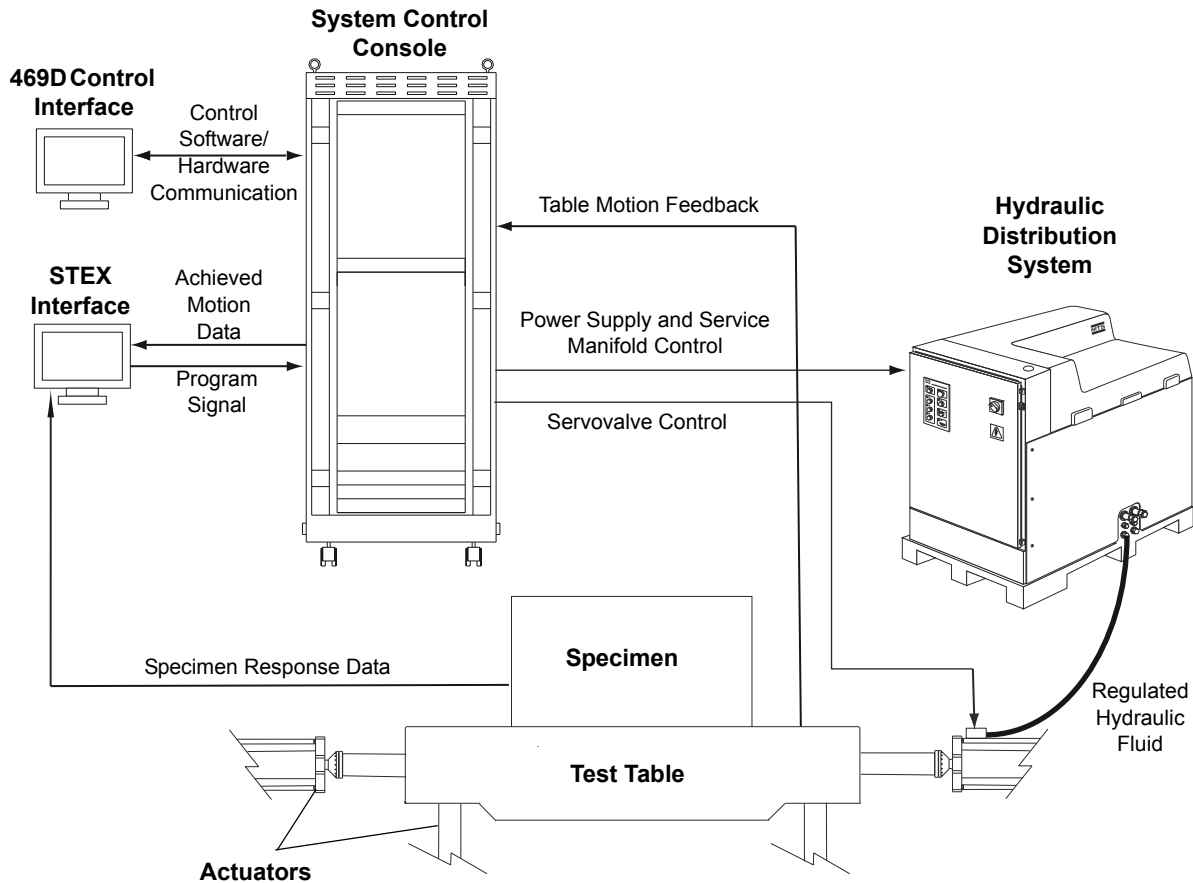
Before Using this Manual

This manual does not describe standard Windows operating procedures. Before using the STEX software, you should become familiar with your Windows operating system and the procedures related to using the menus and controls. Refer to the Windows documentation provided with your system.

System Introduction

Seismic test systems are designed by MTS Systems Corporation to investigate the characteristics of specimens undergoing simulated seismic events. This simulation is controlled through MTS Seismic Test Execution (STEX) software.

The following figure identifies the basic configuration for a typical test system.



System Configuration

As illustrated, the major test system components include hydraulic actuators, servovalves, HSM, HPU, the test table, and the control consoles, software, and interfaces. During a test, the actuators are driven by their electronically-controlled servovalves to apply the desired specimen displacement and test forces. Measurements are made of the actuator motion and specimen response.

The STEX software resides on a PC workstation and provides the electronic program, or drive, signal that is sent to the control console components, where servovalve control signals are generated. It also provides achieved motion and specimen response data acquisition.

The function and operation of the hardware components contained in the test system are described in the system Operation and Reference Manuals. This manual focuses on STEX software operation.

STEX Software Capabilities

STEX software provides control of multi-axial table motion, electronic storage of test definitions and test results, and analysis of desired motion, achieved motion, and specimen response data.

STEX software is used to create tests and results. Available test types are Time History, Compensated Time History, Model Measurement, and Acquisition Only. Results consist of time histories, spectra, response spectra and models.

The MTS Main Dialog appears when you start the STEX application and displays the currently active group of tests and results. The STEX functions are accessed by using the MTS pulldown menu options located on the STEX main dialog. The following table identifies the five pulldown menu options.

MTS Main Dialog Pulldown Menu Options	
OPTION NAME	DESCRIPTION
GROUP	Use to open, create, or delete a test group. Can also create and restore group archive files.
TEST	Use to create, execute, or delete a test from the group. Can also view the state of a test before or after execution.
ANALYSIS	Use to create, edit, delete, or export a result.
HARDWARE	Use to perform hardware related activities such as setting up hardware channels and filters.
HELP	Use to access the STEX on-line help capability.

Group Options

The Group options allow you to create, open, delete, and archive test groups.

The test group represents the highest level of data organization for a test database. A test group contains test records and analysis results. All tests and results within a group must share the same sample rate (time histories) or Nyquist frequency (spectra).

Test Options

The Test options allow you to define and execute uncompensated and compensated earthquake tests. You can also execute model measurement and acquisition only tests. Before test execution, this option allows you to check the limits of and scale the drive signal. When a test has been executed and data stored, you can perform test motion analysis on achieved and specimen response test data using the Analysis options.

Acquisition Only Tests	You define acquisition only tests by selecting specimen channels and acquisition duration.
Model Measurement Tests	You define model measurement tests using random shaping functions.
Uncompensated Earthquake Tests	You define uncompensated earthquake tests using time histories.
Compensated Earthquake Tests	A compensated earthquake test uses a drive signal that has been modified to allow for system dynamics. You define these tests using time histories and a system model.

Further drive signal adjustment can be made on a compensated earthquake test through a series of actual test iterations. In these iterations, STEX measures the difference between desired motion and achieved motion and allows you to redefine, or “tune,” the drive signal.

Analysis Options

- The Analysis options allow you to create, edit and delete results. You can also create model elements.
- You can create time histories that define the drive signal (or target) for a test. You can create different time histories for each control axis that is to be active in the test.
- Analysis options can be used to analyze specimen response and achieved motion test data. You can also perform shock spectrum analysis, and create new results by performing operations on existing results.
- You can create spectra that define the spectral content for random time history generation used in model measurement tests.
- You may import and export data using files external to the STEX database. Imported files are typically earthquake time histories defined in text files residing on the STEX workstation. Test results may be exported and analyzed on a remote workstation.

You may copy test results between test groups within the STEX database.

You may load Plot files that describe the various settings of a detailed data plot. STEX has a feature-rich plot setup environment that allows you to configure a customized representation of test data. Once a plot has been configured, the setup may be saved to a Plot file and reloaded for viewing at a later date.

Hardware Options

The Hardware options allow you to perform functions related to the hardware, such as setting up hardware channels. These options are dependent upon the hardware configuration and will vary with each system.

Help Options

The Help options allow you to invoke the STEX on-line help capability, which can be used to obtain information about a specific STEX related topic.

Defining and Executing a Test

As described in the preceding subsections, the options on the pulldown menus perform unique functions that collectively provide test configuration and execution for four types of tests:

- acquisition only test
- model measurement tests
- uncompensated earthquake testing
- compensated earthquake testing

The following subsections describe the processes used for each type of test definition and execution.

Acquisition Only Test

You define an acquisition only test by selecting the desired specimen channels and the duration. There are three basic steps to complete for definition and execution of an acquisition only test.

1. Setup specimen acquisition channels.

Use the “New” Test Option

2. Input the desired duration.

Use the “New” Test Option

3. Execute the test.

Use the “Execute” Test Option

Each of the steps are described in more detail in the sections describing the option indicated in the Steps.

Model Measurement Test

You define model measurement tests based on spectral content described in a spectrum result. A test defined using random motion is used to generate a system model for compensated testing. There are five basic steps to complete for definition and execution of a model measurement test.

1. Determine the control DOFs for the test.

2. Create random shape spectrum(s) for the DOFs (more than one DOF may use the same random shape result).

Use the “Shape” Analysis Option.

3. Define the test by specifying the DOFs.

Use the “New” Test Option.

4. Specify a random shape result for each DOF. Input the excitation level and test duration.

Use the “Define Random Motions” dialog

5. Execute the test.

Use the “Execute” Test Option

After test execution, you can use the New Model option to create a model from the results of a model measurement test.

Each of the steps are described in more detail in the sections describing the option identified in the Step.

Uncompensated Earthquake Testing

You define an uncompensated earthquake test using time histories. There are five basic steps to complete for definition and execution of an uncompensated earthquake test.

1. Determine the control DOFs for the test

2. Create a drive time history function for each DOF

Use the Analysis Option

3. Define the test by specifying the DOFs.

Use the “New” Test Option

4. Specify a time history for each DOF.

Use the “Specify Target Time Histories” dialog

5. Execute the test.

Use the “Execute” Test Option

After test execution, you can use the Analysis options to analyze specimen response and achieved motion test data.

Each of the steps are described in more detail in the sections describing the option indicated in the Steps.

Compensated Earthquake Testing

You define these tests using target time histories and a control system model. There are nine basic steps to complete for definition and execution of a compensated earthquake test.

1. Determine the control DOFs for the test.
2. Create a target time history result for each DOF.
Use the Analysis Option
3. Create a system model from data achieved during a model measurement test
Use the “New Model” Test Option
4. Define the test by specifying the DOFs and model.
Use the “New” Test Option
5. Specify a target time history result for each DOF.
Use the “Specify Target Time Histories” dialog
6. Execute the test.
Use the “Execute” Test Option
7. Tune the drive signal by comparing achieved and desired motion signals, and then correcting the signal.
Use the “Execute” Test Option
8. Execute the test
Use the “Execute” Test Option
9. Repeat Steps 7 and 8 as desired.
Use the “Execute” Test Option

After test execution, you can use Analysis Options to analyze the specimen response and achieved motion data.

Each of the steps are described in more detail in the sections describing the option indicated in the Steps.

STEX Operation

Overview

This section describes some basic information that you need to know in order to begin operating the MTS STEX software, including:

- Starting the STEX software application
- Quitting the STEX software application
- Using the STEX software including:
 - using the STEX Main Dialog
 - selecting menu options
 - test and result definitions
 - displaying test and result parameters
 - using the plot dialog
 - using the Select button
- Using STEX On-line Help

This manual does not describe standard Windows procedures. Before using the STEX software, you should be familiar with the information provided in the *Windows User's Guide*.

Starting the STEX Software

Use the following procedure to start the STEX software application:

- start the computer system
- log in
- double click on the STEX icon

The STEX application will open with the STEX Main Dialog.

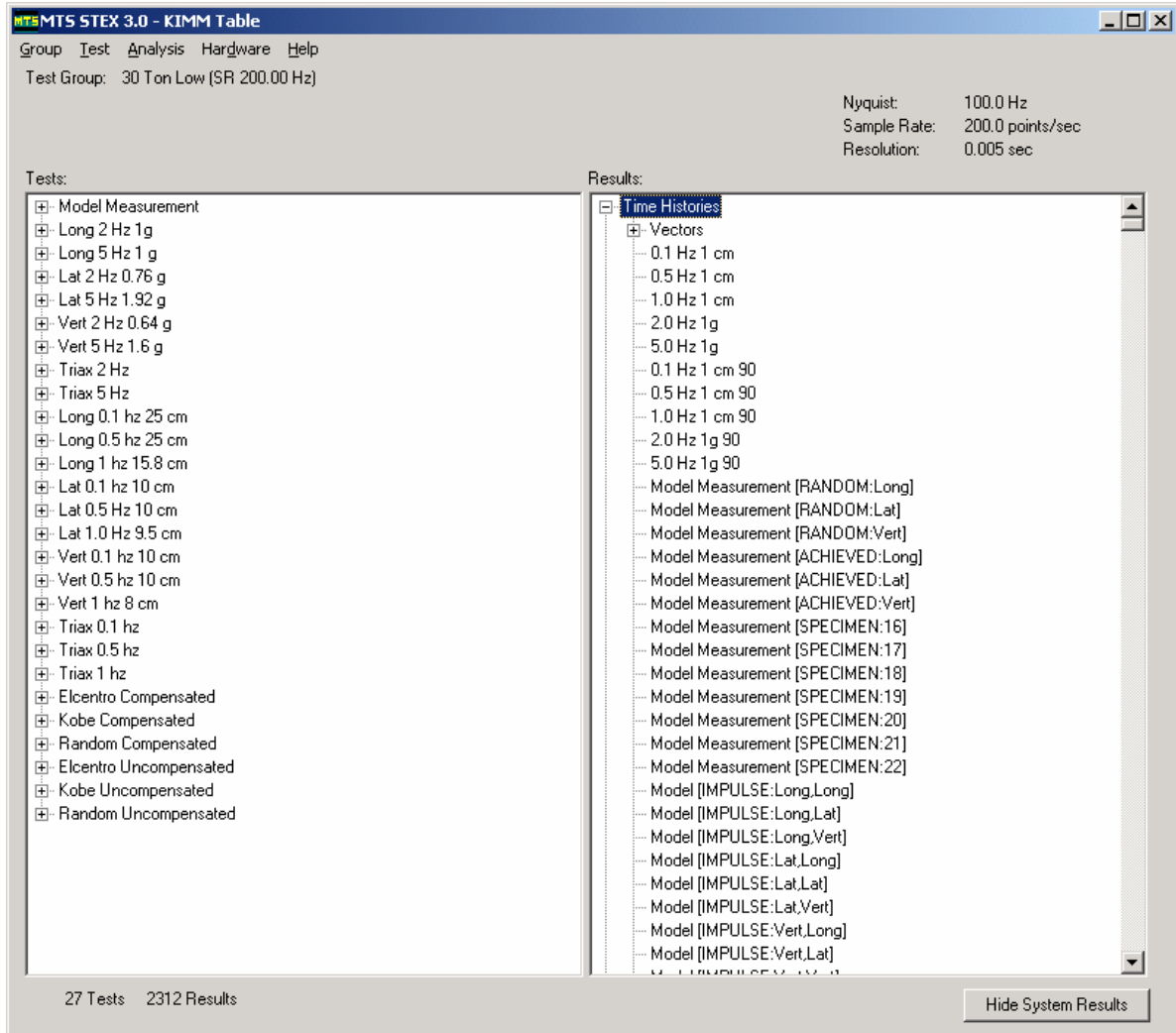
Quitting the STEX Software

To exit the STEX application, select “**EXIT**” under the Group pulldown menu located on the Main Dialog.

You may also exit the STEX application, like most Windows applications, by clicking on the  button on the right end of the main dialog title bar, depressing the “**Escape Key**”, or selecting “**Close**” from the system dialog. The system dialog is activated as a pulldown menu on the left end of the title bar.

STEX Main Dialog

The STEX main dialog opens automatically when you start the STEX application.



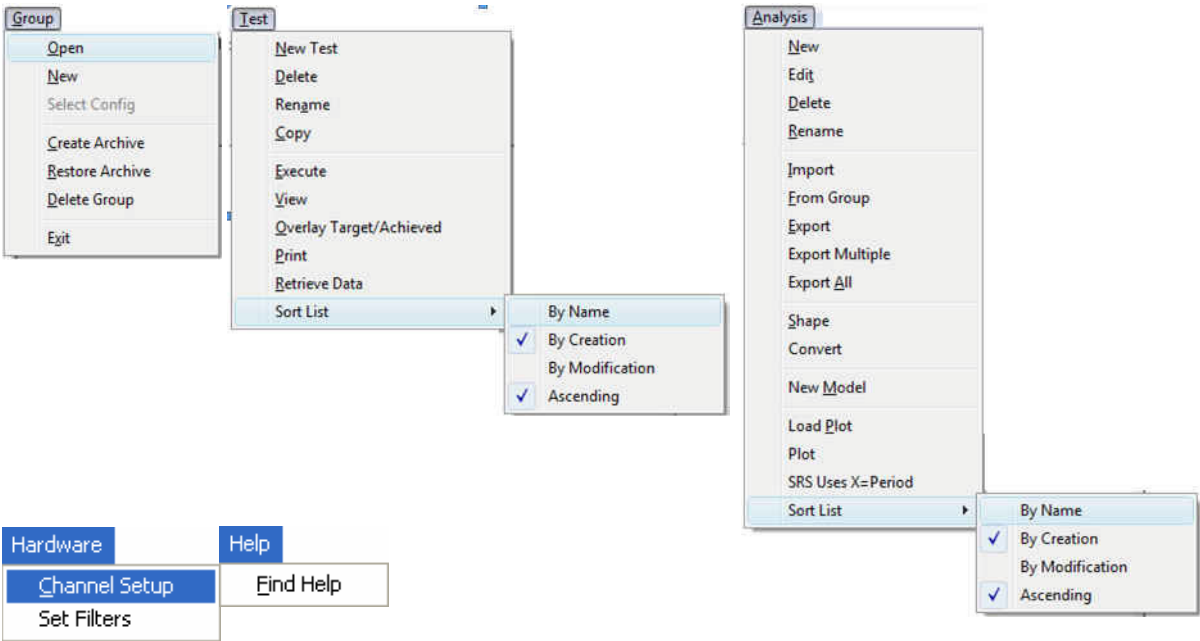
The STEX main dialog provides information relating to the test group that is currently active. Test groups represent the highest level of data organization for a test database. Refer to [Test Groups](#) on [page 51](#), for more information about the function and use of test group options. The main dialog displays the name of the active group and the following group parameters; Nyquist frequency, sample rate, and sample resolution. Two scrolling lists display the tests and analysis results contained by the active group. Also, the number of tests and results are displayed below the test list box. Pulldown menus provide access to group, test, and analysis functions, as well as hardware and help options.

A button exists that toggles between “Hide System Results” and “Show System Results”. STEX automatically generates system results during certain operations. However, user results are explicitly created by the user through the Analysis menu. Refer to, [“Analysis Results,”](#) on [page 23](#).

The following subsections further describe the functionality of the main dialog.

Pulldown Menu Options

The STEX main dialog has five pulldown menus which open other dialogs when selected.



The Group pulldown menu provides options that are used to setup and manipulate test groups. Refer to [Test Groups](#) beginning on [page 51](#).

The Test pulldown menu provides options that are used to configure, execute and view tests. Refer to [Tests](#) beginning on [page 59](#).

The Analysis pulldown menu provides options that are used to create and edit analysis results including time histories, spectra and models. Refer to [Analysis Results](#) beginning on [page 105](#).

The Hardware pulldown menu provides options that are used to set up hardware related functions. Refer to, [“Channel Setup Dialogs,”](#) on [page 93](#).

The Help pulldown menu provides options that are used to invoke STEX On-line Help. Refer to, [“STEX On-line Help,”](#) on [page 24](#).

Tests

Tests are created using the Test pulldown menu and are listed in the Test list box on the Main Dialog. When a test is created, it must be given a unique name.

Test types are:

- Acquisition only (no drive motion)
- Model Measurement (for creating models)
- Time History (time history drive for each DOF)
- Compensated Time History (uses model for compensation)

An acquisition only test is defined by a set of specimen acquisition channels.

The model measurement, time history, and compensated time history tests provide drive signal capability along with the DOF and specimen acquisition parameters.

A time history test includes drive signals for each control DOF. Adding compensation to a time history test involves using a model to adjust the drive signal based on experience gained during a model measurement test.

A model measurement test uses shaped random noise to drive each DOF independently (i.e. consecutively). The drive signals are specially designed to provide accurate model information so that compensation may be used to improve system response and drive correlation.

Note that time histories associated with a test (e.g. drive and achieved signals) are listed in the result list box; the result names are derived from the test name.

When you double click on a test in the test list box, a test viewing dialog will open, allowing you to review the test parameters. Refer to, [“Viewing Test Parameters,”](#) on page 25.

When you double click on a result in the result list, a statistic dialog will open, allowing you to review the result parameters. Refer to, [“Viewing Result Parameters,”](#) on page 28.

Analysis Results

Analysis results are listed in the Results list box on the Main Dialog. There are user results and system results. Each result has a unique name.

User results are created by the user (i.e. a time history, spectrum or a model) using options on the Analysis pulldown menu. User result names cannot include the brackets [and].

System results are created by STEX (i.e. time histories of a test or elements of a model). When the system creates a result, it inserts a special description at the beginning of the result name, enclosed in brackets:

Model Measurement Test [RANDOM:Lat]

The button “Hide (Show) System Results” located on the Main Dialog will remove (or include) the system results from the list box, depending on the current state. This can be helpful when there is a large number of system results.

Results are listed by type on the Main Dialog.:

- Time Histories (including Vectors)
- Spectra
- Response Spectra
- Models (including Matrices)

Results include functions of time (time histories) and functions of frequency (spectra) but also include vectors, matrices, and models.

A vector is a group of one or more time history results. For example, the set of achieved signals from a test consists of several individual time histories grouped into a single vector. A vector can only be created by the system.

A matrix is a two-dimensional grouping of spectra or time histories. For example, the impulse response function of a model is a matrix of time histories. A matrix can only be created by the system.

A model result consists of several matrices including the transfer function and the expanded inverse transfer function. A model is used by a compensated test to iteratively improve the response of the control system to more closely match the target motion.

When you double click on a result in the result list box, an appropriate statistic dialog will open, allowing you to view and plot result parameters. Each result type provides a uniquely formatted statistic dialog presenting information derived from the result content. Refer to, [“Viewing Result Parameters,”](#) on page 28.

STEX On-line Help

STEX On-line help may be initiated by one of the following techniques:

- The Help menu on the main dialog
- Pressing F1 within any dialog

The help menu may be used to select **“Find Help”**; this invokes the Windows Help application.

Pressing F1 activates the **“Context Sensitive Help”**. The Windows Help application will appear with the topic set to describe the current STEX dialog. If a menu item is highlighted on the main dialog, the help topic will describe the item.

Viewing Tests and Results

You can view test and result parameters by double-clicking on the desired test or result in the appropriate list box on the main dialog. A test viewing dialog or result statistic dialog appears which displays the parameters.

You can also open the test-viewing dialog by highlighting the test and selecting **“View”** from the Test pulldown menu.

Viewing Test Parameters

When you double-click on a test in the test list box or highlight the test and select **“View”** from the Test pulldown menu, the **View Test** dialog will appear.

When you double-click a test name in the test list for viewing, STEX checks to see if test setup was completed. If not completed, a message box appears before the test-viewing dialog asking if setup should continue. If you choose not to continue test setup, the View Test dialog appears. Otherwise, setup continues (loopback step, for example) and then the View Test dialog appears.

View Compensated Test

Name: Elcentro Compensated
 Test Type: Compensated Time History
 Created: 16-Jun-1997:23:28:05
 Modified: 10-Aug-2001:11:47:15
 Storage: 1687 (SAAAABK5.STX) 14.85 MB
 Duration: 66.555000 (PPF: 1024)

☐ Model: Model II

☐ Target: Elcentro Compensated [TARGET]
☐ Desired: Elcentro Compensated [DESIRED]
☐ Drive: Elcentro Compensated [DRIVE4]
☐ Specimen: Elcentro Compensated [SPECIMEN4]
☐ Achieved: Elcentro Compensated [ACHIEVED4]

DOF	Control Mode	Full Scale	Feedback Mode	Full Scale
Longitudinal	Acceleration	10.0 g	Acceleration	10.0 g
Lateral	Acceleration	10.0 g	Acceleration	10.0 g
Vertical	Acceleration	10.0 g	Acceleration	10.0 g

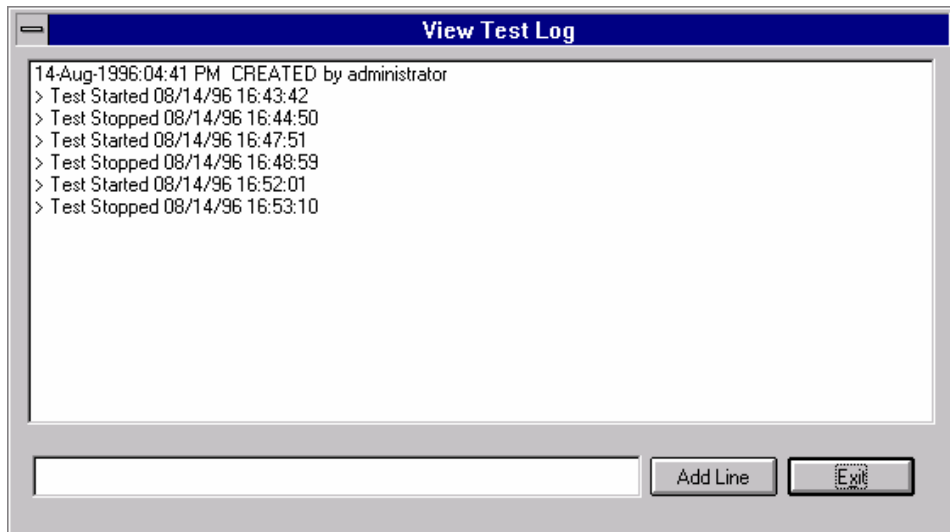
Channel	Full Scale	Identifier
18(18)	-110.6,110.6 g	(16) CHAN16:Unused:Unused
19(19)	-101.9,101.9 g	(17) CHAN17:B00218:Unused
20(20)	-42.4,42.4 g	(18) CHAN18:784823:Unused
21(21)	-43.6,43.6 g	(19) CHAN19:784820:Unused
22(22)	-105.0,105.0 g	(20) CHAN20:B00215:Unused
23(23)	-44.5,44.5 g	(21) CHAN21:784819:Unused
24(24)	-81.6,81.6 g	(22) CHAN22:796237:Unused

This dialog contains the following information:

- Test name and type
- Test creation and modification dates
- Test duration (in seconds)
- Model name (for compensated tests)
- Target motion vector name
- Desired motion vector name (for compensated tests)
- Specimen motion vector name (if any)
- A list detailing the DOFs controlled by the test, along with mode and full-scale information
- A list of specimen acquisition channels along with identifier and full-scale information

Selecting a button next to one of the vector names (target, achieved, drive, or specimen) will open an appropriate statistics dialog for that vector. Refer to, [“Viewing Result Parameters,”](#) on page 28.

Selecting the “**View Test Log**” button will open a View Test Log dialog, allowing you to scroll through the test log.



You may add one or more comment lines to the log by entering text in the edit field and selecting the “**Add Line**” button.

Generating a Test Report

Selecting the “**Report**” button on the test-viewing dialog will generate a test report in a text file. A filename is specified using the Save As dialog. The test report describes the test including channel information as well as the test log.

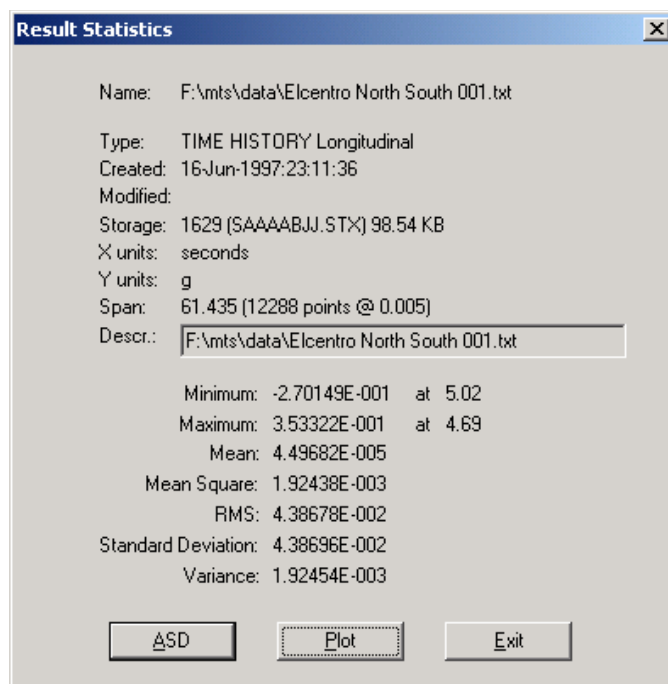
Viewing Result Parameters

When you double click on a result in the result list box, an appropriate statistic dialog will open. The statistic dialog may be viewed and exited, or you may plot the results.

Each result type provides a uniquely formatted statistic dialog presenting information derived from the result content. The following subsections describe these statistic dialogs.

Time History and Real Valued Spectra Statistics Dialog

When you double click on a time history or real-valued spectrum in the result list on the main dialog, the **Results Statistics** dialog appears. (Refer to, “[Random Time History Statistics Dialog](#),” on page 31 for information regarding *random* time histories.)



The dialog displays the following information:

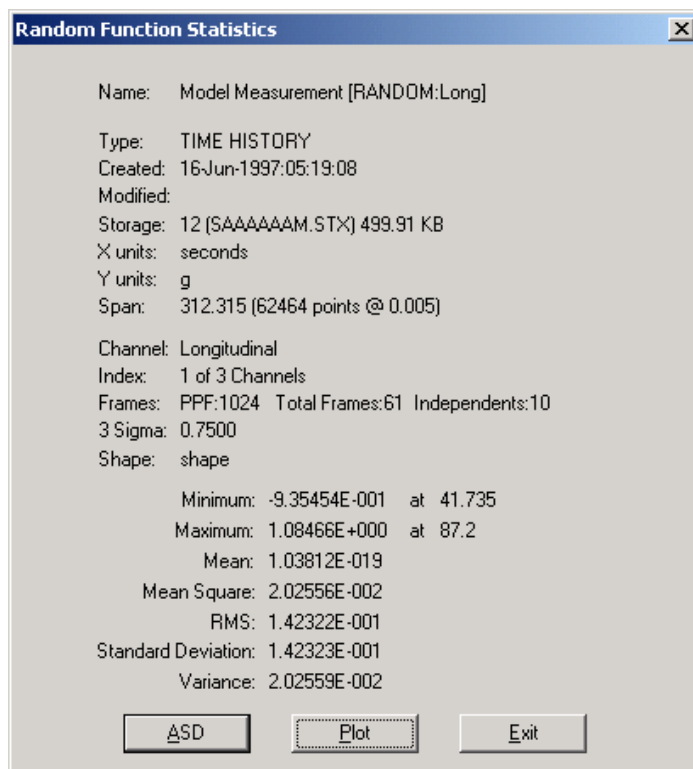
- Result name
- Result type
- Creation and modification dates and times
- Permanent storage information
- Units for both axes
- Span information with count and resolution
- Description - automatically created for USER results only
- Minimum and maximum result values
- Mean, mean square, RMS, standard deviation, and variance values

Pressing the “**Plot**” button on the statistic dialog will open the plotting dialog. Refer to [Using the Plotting Option](#) beginning on [page 41](#).

Pressing the “**ASD**” button results in a plotting dialog that displays the frequency content of the time history.

Random Time History Statistics Dialog

When you double click on a random time history in the result list on the main dialog, the **Random Function Statistics** dialog appears.



The dialog displays the following information:

- Result name
- Result type
- Creation date and time
- Permanent storage information
- Units for both axes
- Span information with count and resolution
- Degree of Freedom using this random time history
- Index and count of random signals in the vector
- Frame size and count (total and independents/DOF)
- Three sigma level
- ASD used to create the random time history
- Minimum and maximum result values
- Mean, mean square, RMS, standard deviation, and variance values

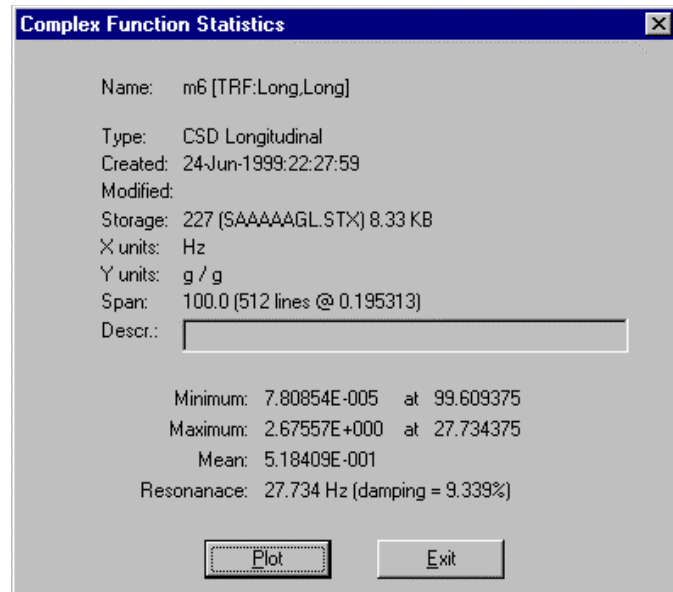
Refer to [“Random Signal Architecture”](#) on page 171 for random signal information.

Pressing the **“Plot”** button on the statistic dialog will open the plotting dialog. Refer to [Using the Plotting Option](#) beginning on [page 41](#).

Pressing the **“ASD”** button results in a plotting dialog that displays the frequency content of the time history.

Complex Spectrum Statistics Dialog

When you double click on a complex spectrum in the result list on the Main Dialog, the **Complex Function Statistics** dialog appears.



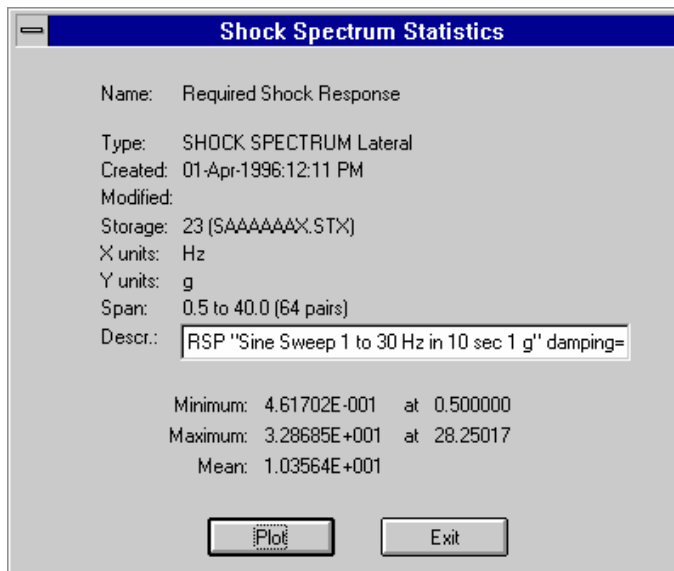
The dialog displays the following information:

- Result name
- Result type
- Creation date and time
- Permanent storage information
- Units for both axes
- Span information with count and resolution
- Minimum and maximum amplitude values
- Mean amplitude
- Description - automatically created for USER results only
- Resonance and damping (computed)

Pressing the “**Plot**” button on the statistic dialog will open the plotting dialog. Refer to [Using the Plotting Option](#) beginning on [page 41](#).

Response Spectrum Statistics Dialog

When you double click on a shock response spectrum in the result list on the main dialog, the **Shock Spectrum Statistics** dialog appears.



When you double click on a shock response spectrum in the result list on the main dialog, the **Shock Spectrum Statistics** dialog appears.

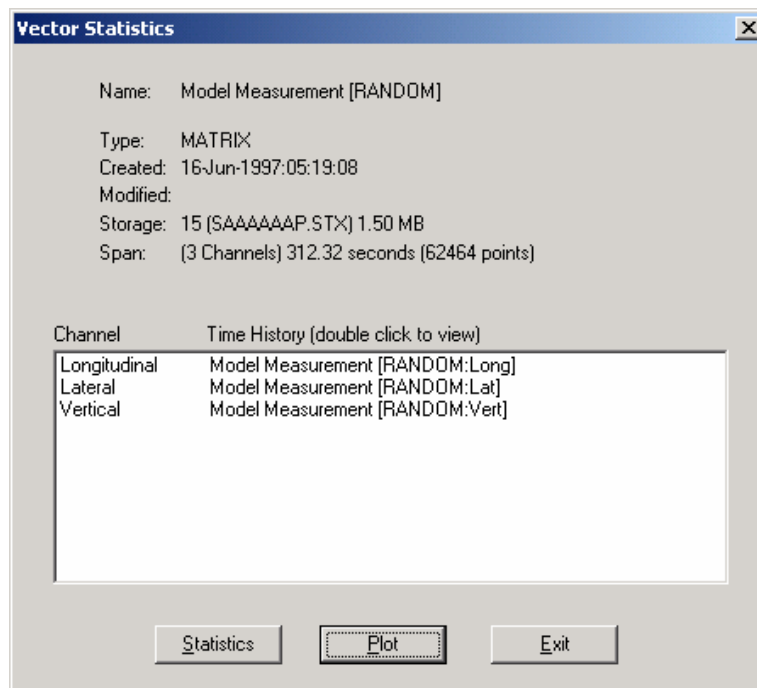
The dialog displays the following information:

- Result name
- Result type
- Creation and modification dates and times
- Permanent storage information
- Units for both axes
- Span information with count and resolution
- Description - automatically created for USER results only
- Minimum and maximum amplitude values
- Mean amplitude

Pressing the “**Plot**” button on the statistic dialog will open the plotting dialog. Refer to [Using the Plotting Option](#) beginning on [page 41](#).

Vector Statistics Dialogs

When you double click on a vector in the result list on the main dialog, the **Vector Statistics** dialog appears.



The dialog displays the following information:

- Result name
- Result type
- Creation and modification dates and times
- Permanent storage information
- Span, including channel count
- Channel list with time history element names

Pressing the **“Plot”** button on the statistic dialog will open the plotting dialog.

Pressing the “**Statistics**” button opens the **Channel Statistics** dialog for all the channels. Double clicking on a channel in the channel list will open a Statistics dialog for that channel only.

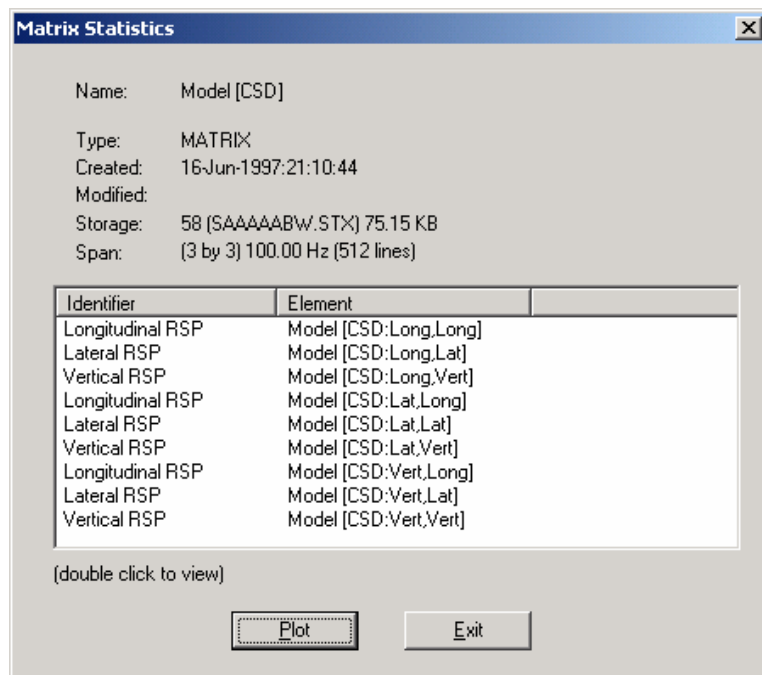
Identifier	Units	Minimum	Maximum	Mean	RMS
Longitudinal	g	-0.253739	0.40174	0.0	0.041062
Lateral	g	-0.179021	0.25059	0.0	0.034661
Vertical	g	-0.228294	0.228026	0.0	0.020437
Roll	g	-0.04372	0.045391	0.0	0.005275
Pitch	g	-0.044636	0.03641	0.0	0.005126
Yaw	g	-0.051541	0.057117	0.0	0.010294

This dialog displays the following information for each channel of the vector:

- Channel identifier
- Channel units
- Minimum, maximum, mean, and RMS values for the channel

Matrix Statistics Dialog

When you double click on a matrix in the result list on the main dialog, the **Matrix Statistics** dialog appears.



This dialog displays the following information:

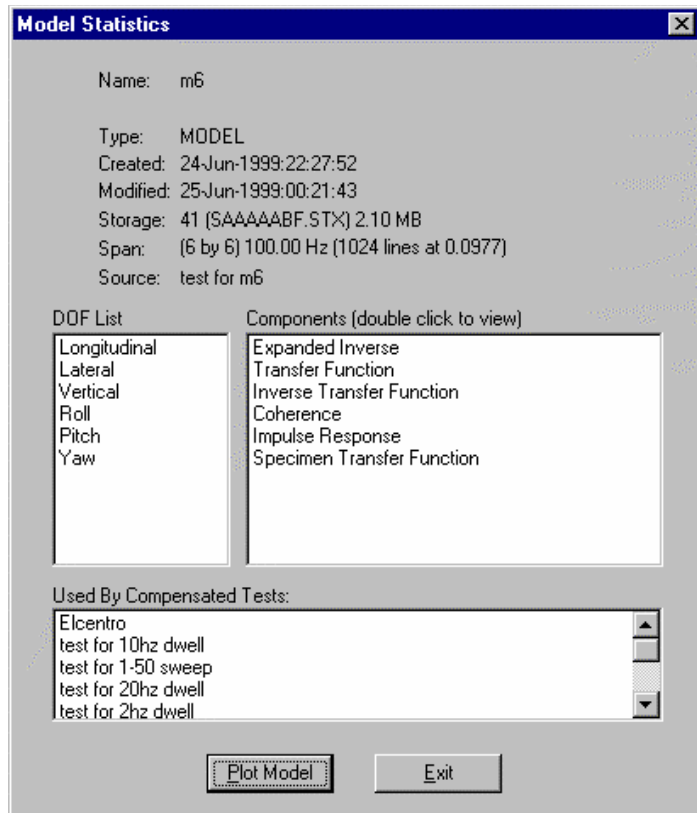
- Result name
- Result type
- Creation date and time
- Permanent storage information
- Span, including matrix dimensions
- List of element names

Pressing the “**Plot**” button on the statistic dialog will open the plotting dialog.

Double clicking on an element in the element list results in a statistic dialog for that element only.

Model Statistics Dialog

When you double click on a model in the result list on the main dialog, the **Model Statistics** dialog appears.



This dialog displays the following information:

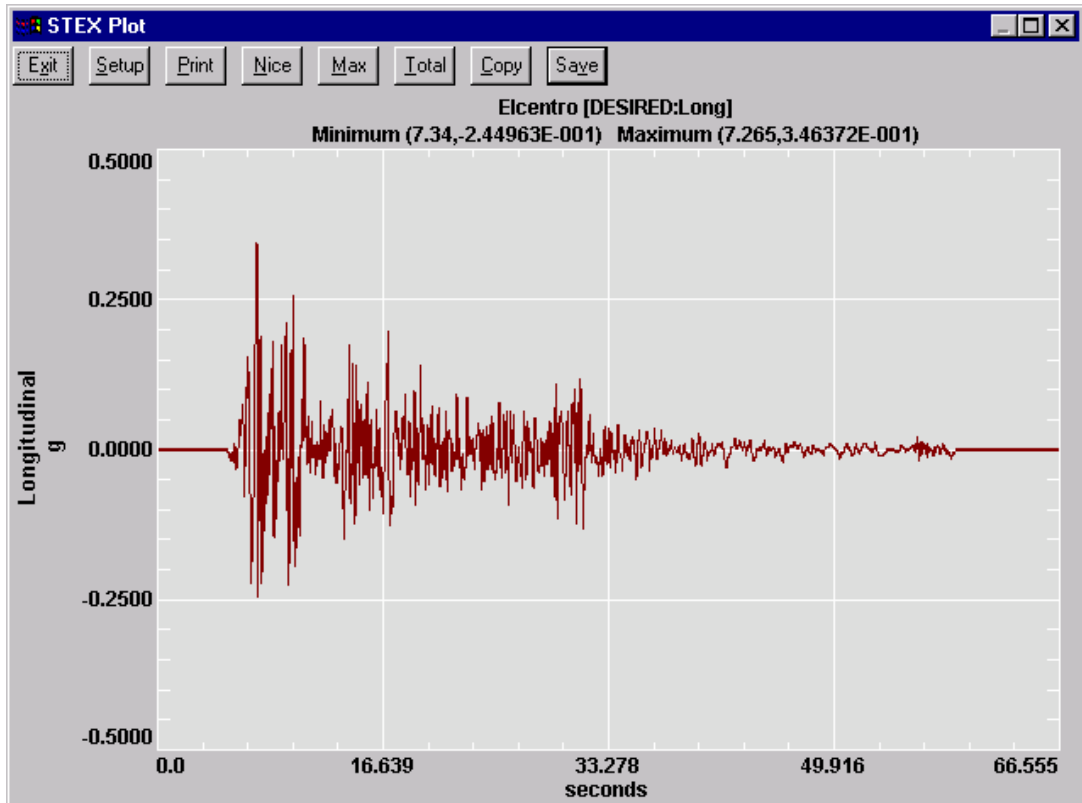
- Result name
- Result type
- Creation and modification dates and times
- Permanent storage information
- Span, including matrix dimensions
- Source Test
- List of DOFs
- List of model components available for viewing
- List of compensated tests using the model

Pressing the “**Plot**” button on the statistic dialog will open the plotting dialog.

Double clicking on a component in the component list results in statistics dialog for that component.

Using the Plotting Option

Selecting “**Plot**” on a dialog or double clicking on a component in a list on a statistics dialog will open the **STEX Plot** dialog.



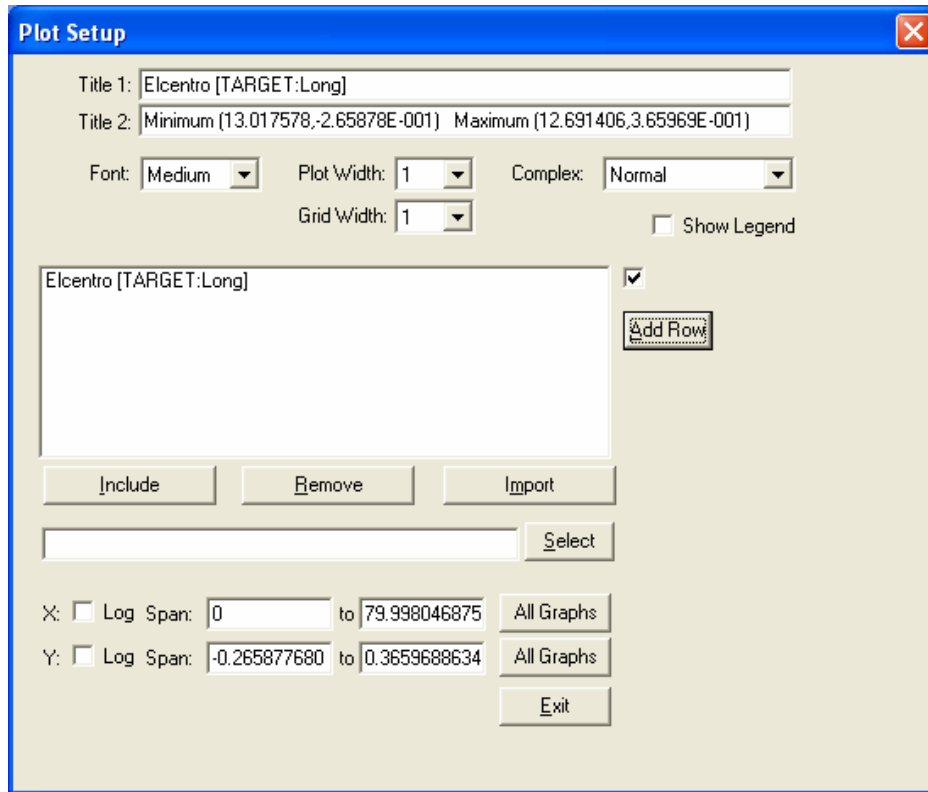
The STEX Plot dialog provides a default graphical depiction of the parameter selected for plotting.

The following buttons are provided on the top of the dialog:

- Exit (Exit the STEX Plot dialog and return to the statistic dialog)
- Setup (Opens a Setup dialog to manipulate the format of the graphic display. See, [“Plot Setup Dialog,”](#) on page 43)
- Print (Opens a Print dialog to select a printer for hardcopy)
- Nice (Sets the vertical span to “round” values)
- Max (Sets the vertical span to the extreme values)
- Total (Sets the horizontal span to the result maximum)
- Copy (Copies the plot image to the clipboard)
- Save (Saves the current plot settings to a file for later viewing)

Note *You can use the < and > keys to advance the plot range and the Shift < and > keys to enlarge the plot range.*

Plot Setup Dialog



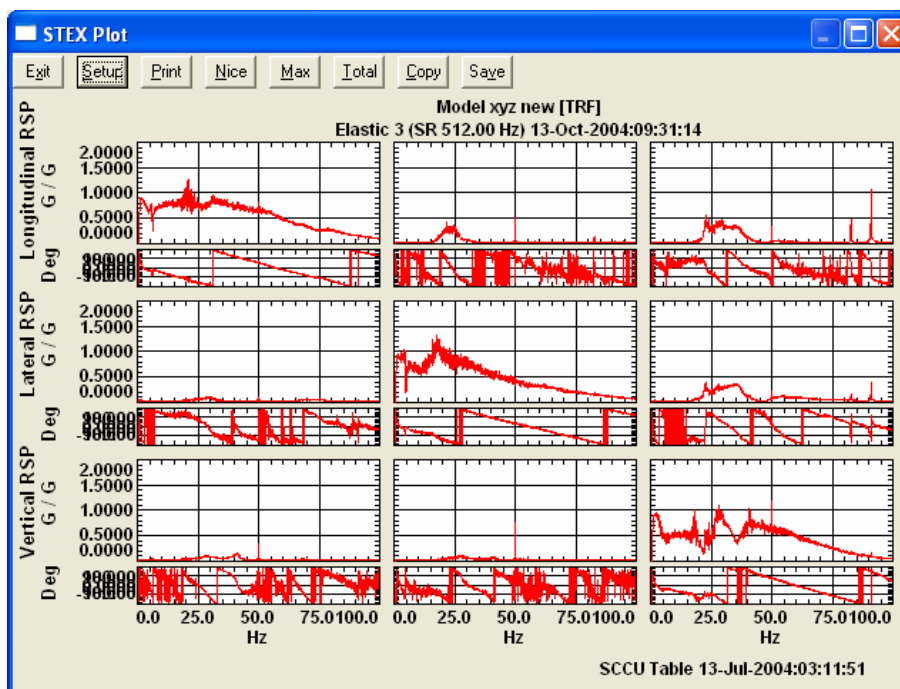
When you select “**Setup**” on the STEX Plot dialog, the **Plot Setup** dialog appears. This dialog provides the following items:

- Two editable title fields
- A font selection (small, medium, and large)
- Line widths (for plot trace *and* grid)
- Complex format (*normal*, *amplitude only*, and *phase only*)
- Graph configuration matrix (identifies the number of different graphs on the plotting dialog, and which of these graphs is currently detailed on the setup dialog); additional rows may be added
- Check boxes along the columns and rows of the graph matrix – cause a new plot dialog displaying the row or column selected.

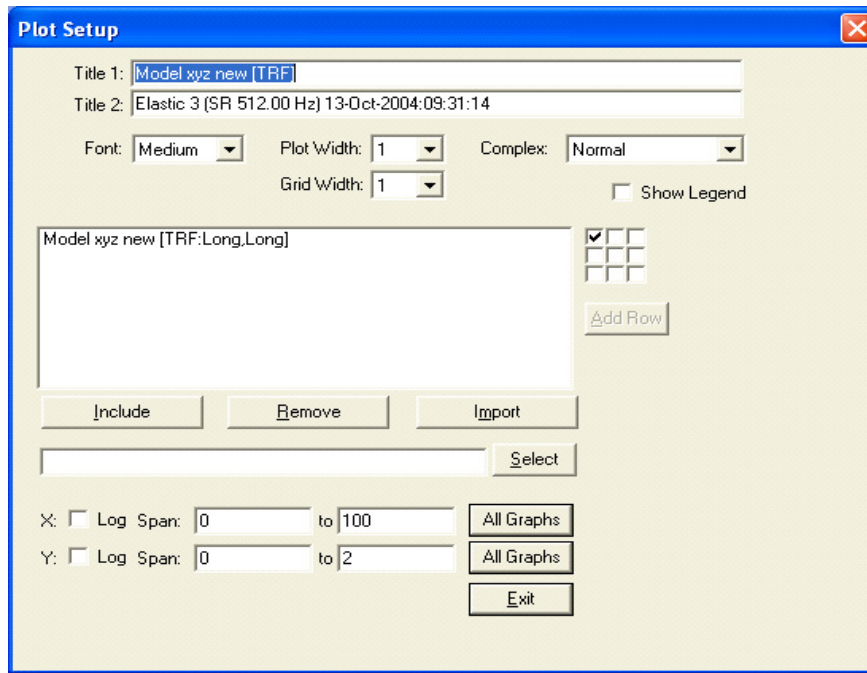
- Result list (lists the results to be displayed in the graph selected in the configuration matrix); results may be included or removed from the list
- X and Y axis parameters (linear/logarithmic, and spans)

Selecting “**All Graphs**” causes the associated axis parameters to be rendered to all of the graphs. This behavior is automatic *when the originally plotted result was a vector or matrix*. In this case, overlay plots are allowed but rows may not be added to the configuration matrix.

The following dialog demonstrates a plotting dialog resulting from a *matrix* statistic dialog, in this case a three DOF transfer function.



The plot setup dialog associated with this plot dialog is shown below.



Note The configuration matrix has a three by three section enabled and the upper left element is active. The result list at right, lists the name of the result displayed in the corresponding position on the plotting dialog.

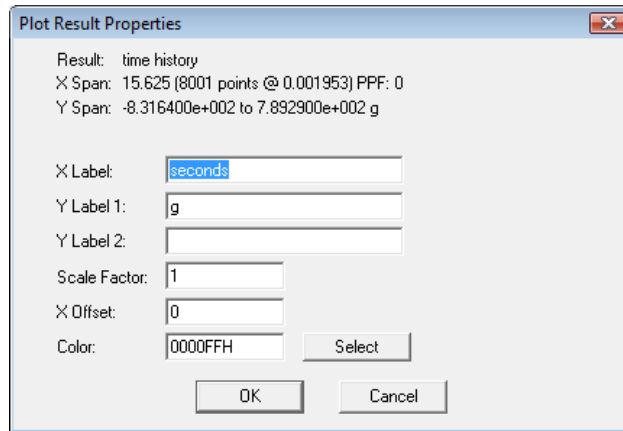
To add graphs to the plotting dialog, select the “**Add Row**” button. To define the contents of the new row, click “**Select**” and then “**Include**” or “**Import**” to identify the result to be included in the graph.

To edit the contents of the active element of the configuration matrix, use the “**Select**”, “**Include**”, “**Remove**”, and “**Import**” buttons as appropriate.

Note *Import allows you to plot results in a different test group.*

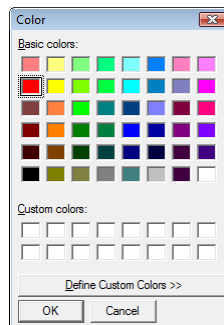
To create overlay plots, include multiple results for the same active element.

Double-click a result in the list to open the Plot Result Properties dialog and use this dialog to modify the display parameters.

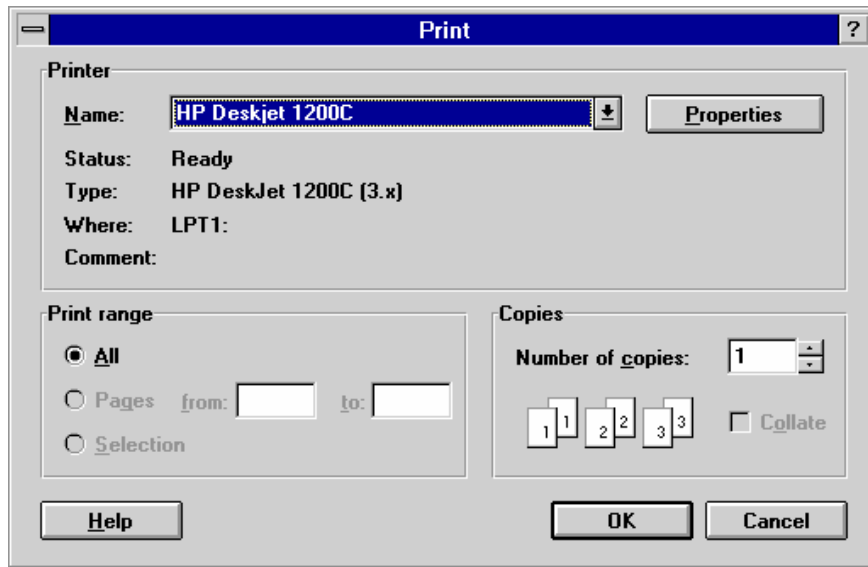


You can change the following display parameters

- **X Label** - Displays the string on the bottom of the graph along the X axis.
- **Y Labels 1 and 2** - Displays the strings at the left of the graph along the Y axis.
- **Scale factor** - Applies a scale factor to the amplitude of this result (for display only).
- **X Offset** - Applies an offset along the independent axis, shifting this result in relation to other results.
- **Color** - Changes the RGB value of the color used to display this result. To make a color selection from an interactive dialog, click the **Select** button. When you click the **Select** button, the Color dialog opens.



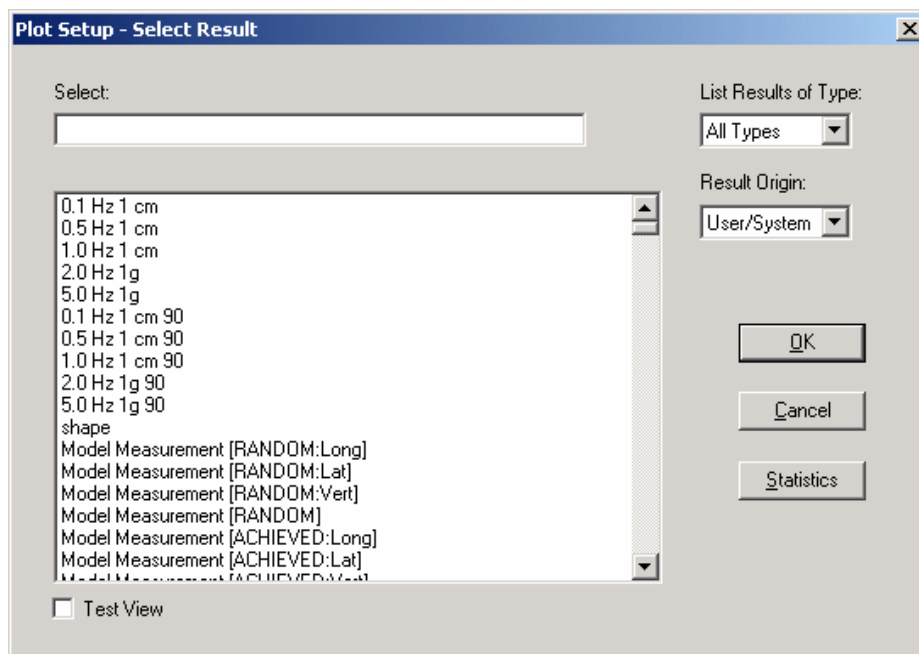
Using the Printing Option

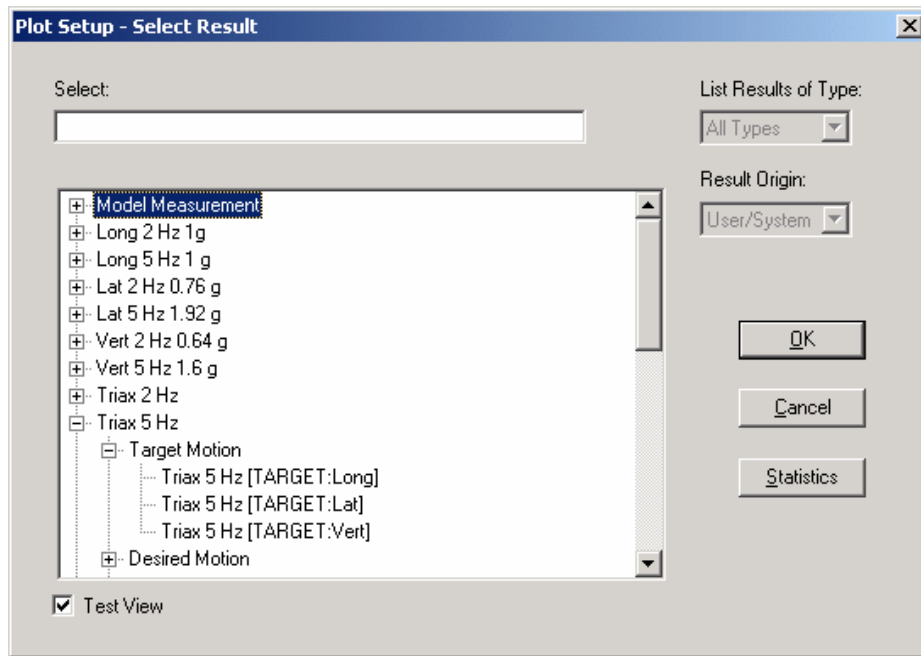


This standard Windows dialog allows you to select a printer and printer related parameters. You may create a hardcopy or cancel the operation.

Using the Select Option

When you press the “**Select**” button on any of the STEX dialogs, a **Result Selection** dialog will appear.





This dialog provides a list of all results in the active test group. When you highlight the desired result in the list and select “**OK**”, the selected name will be inserted into the edit field next to the “Select” button used to invoke the selection dialog. Alternatively, you may type the desired result name into the edit field. Selecting “**Statistics**” will result in a statistics dialog for the result highlighted in the listbox.

A result type parameter may be selected by the user (this parameter may be preset by the dialog that invoked the selection dialog). When the result type field is changed, the result list is updated to include results that match the selected type. In addition, you may instruct the dialog to list user results, system results, or both.

Test Groups

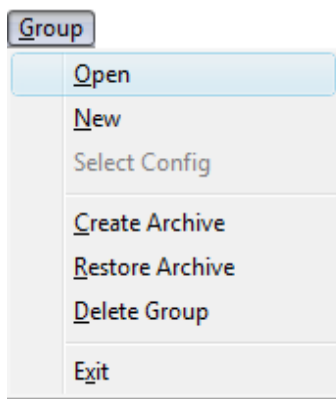
Overview

The test group is the highest level of data organization for a test database. The group contains test records as well as analysis results (e.g. time histories, spectra, and models). When a group is created, you must specify a group name and sample rate. All time histories within a group must share the same sample rate and all spectra must share the same Nyquist frequency. STEX will automatically modify the group name to add the group sample rate.

Only one test group can be active. The STEX main dialog displays the name of the currently active test group along with the group parameters (Nyquist frequency, sample rate, and resolution). Also, the names of all the tests and results contained in the test group are listed.

Test groups can be created, opened, deleted, and archived using the “Group” pulldown menu located on the STEX main dialog.

Group Menu Options



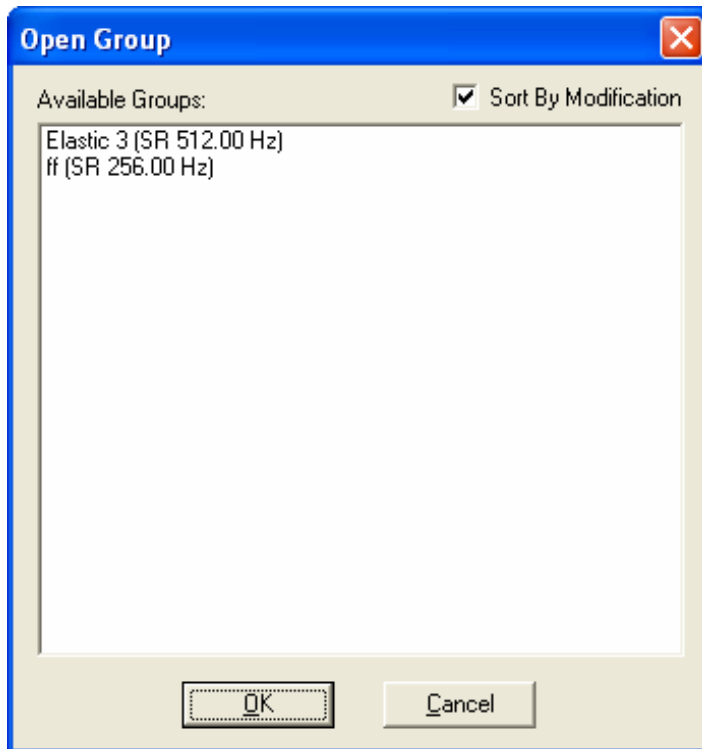
The “Group” pulldown menu contains the following options:

Open	(activates an existing test group)
New	(creates a new test group)
Select Config	(select from DOF configurations, if applicable)
Create Archive	(creates an archive file of the current group)
Restore Archive	(restores a group from an archive file)
Delete Group	(destroys the current group)
Exit	(exits the STEX application)

Note When there is no active group, only “Open”, “New”, “Exit”, and “Restore Archive” options are enabled in the “Group” pulldown menu.

Opening an Existing Group

When you select “**Open**” from the Group pulldown menu on the STEX main dialog, the **Open Group** dialog appears.



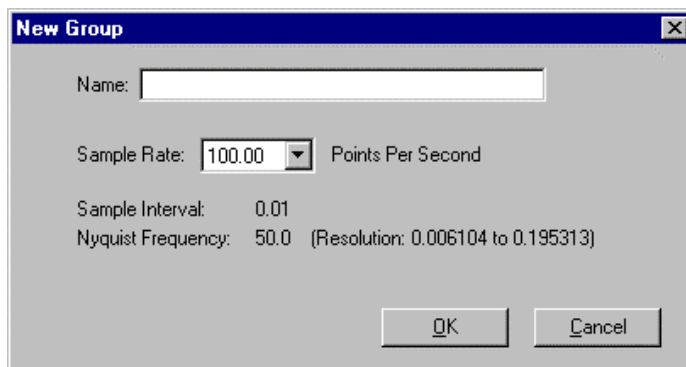
This dialog lists the currently defined test groups. To open a test group, either double click on the desired group name in the list or highlight the desired group name and press the “**OK**” button. The STEX main dialog will reappear, updated to reflect the contents of the selected group.

To sort the groups based on when each was modified, check the **Sort By Modification** check box.

Creating a New Group

When you select “New” from the Group pulldown menu on the STEX main dialog, one of the following **New Group** dialogs appears, depending on the type of data acquisition hardware used with the system.

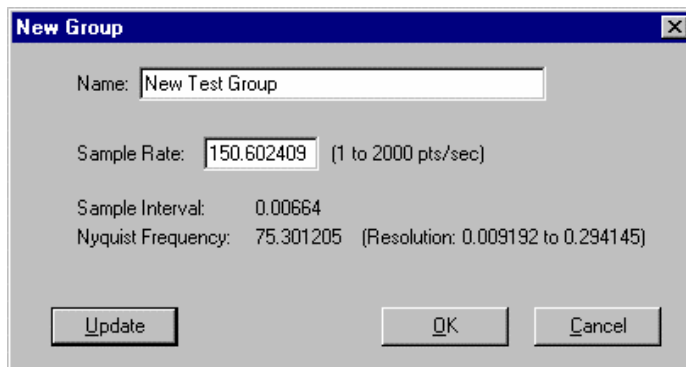
Discrete List Selection



The "New Group" dialog box for Discrete List Selection has a blue title bar with the text "New Group" and a close button. It contains the following fields and controls:

- Name:** A text input field.
- Sample Rate:** A dropdown menu showing "100.00" and a "Points Per Second" label.
- Sample Interval:** A text input field showing "0.01".
- Nyquist Frequency:** A text input field showing "50.0" with a resolution range "(Resolution: 0.006104 to 0.195313)".
- Buttons:** "OK" and "Cancel" buttons at the bottom right.

Edit Field/Update Selection



The "New Group" dialog box for Edit Field/Update Selection has a blue title bar with the text "New Group" and a close button. It contains the following fields and controls:

- Name:** A text input field containing "New Test Group".
- Sample Rate:** A text input field showing "150.602409" with a range "(1 to 2000 pts/sec)".
- Sample Interval:** A text input field showing "0.00664".
- Nyquist Frequency:** A text input field showing "75.301205" with a resolution range "(Resolution: 0.009192 to 0.294145)".
- Buttons:** "Update", "OK", and "Cancel" buttons at the bottom.

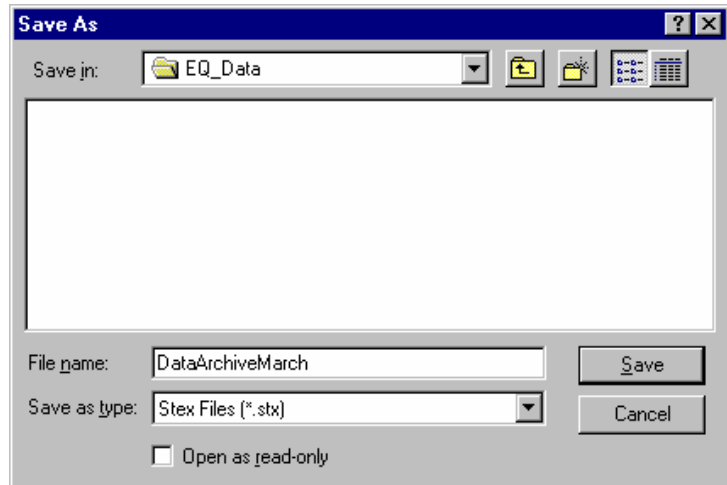
Input the desired **Group Name**. Select a **Sample Rate**. If necessary, press **Update** to Automatically update the sample rate, sample interval, and Nyquist frequency parameters.

Test Group for Specimen XYZ [SR 200.00 Hz]

When you are satisfied with the New Group parameters, press “**OK**”. The new group is then created and becomes active on the STEX main dialog.

Creating a Group Archive File

When you select “**Create Archive**” from the Group pulldown menu on the STEX main dialog, the Windows **Save As** dialog appears.

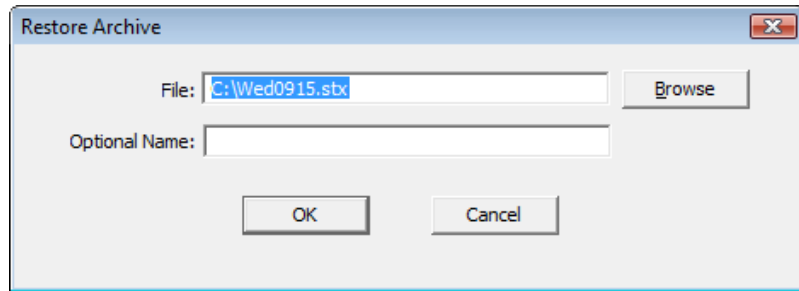


This dialog allows you to specify a directory and filename for archiving the current group.

When you press the “**OK**” button, the entire content of the currently active group is assembled and stored into a single file which may be restored at a later date using the “Restore Archive” option. Refer to, [“Restoring a Group Archive File,”](#) on page 56.

Restoring a Group Archive File

When you select “**Restore Archive**” from the Group pulldown menu on the STEX main dialog, the Restore Archive dialog appears.

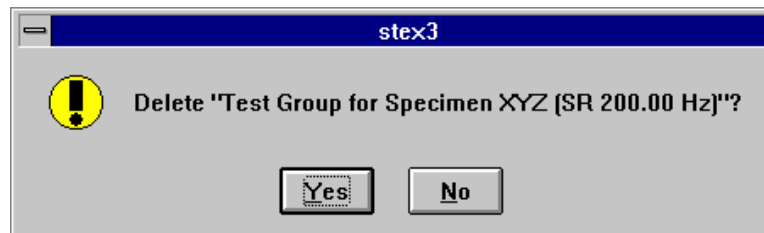
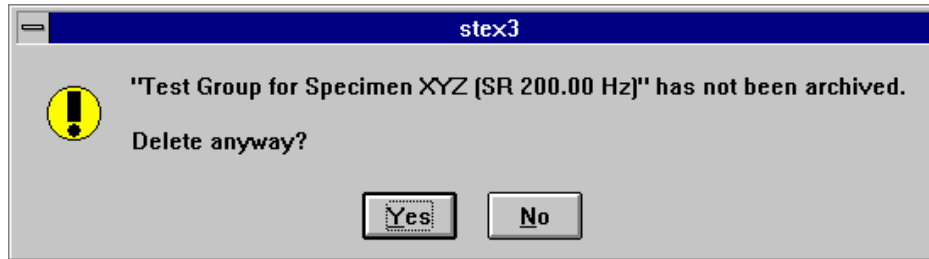


Enter the name of a file or use the **Browse** button to navigate to and select the directory and filename of an existing archived file that was previously created using the “Create Archive” option. If desired, you may specify a new name for the restored group or you can continue to use the group name specified within the archive file.

When you press the “**OK**” button, the archived file will be disassembled and a new group created. The restored group will become the currently active group.

Deleting an Existing Group

When you select “**Delete**” from the Group pulldown menu on the STEX main dialog, one of the following **Delete Verification** dialogs will appear.



If the group has not been archived since the last change to the database, the first dialog will appear informing the user. In either case, you must reaffirm or cancel the delete request.

Quitting the STEX Application

When you select “**Exit**” from the Group pulldown menu on the STEX main dialog, the STEX application will close.

Tests

Overview

Several test types are available. Test type and system parameters are defined during test configuration. Once configured, the test may be executed and viewed. Tests are created using the Test pulldown menu and are listed in the Test list box on the STEX main dialog. When a test is created, it must be given a unique name.

Since STEX is deployed in laboratories utilizing a wide variety of data acquisition and control system equipment, test options may vary. This chapter is divided into sections describing all of the possible configurations.

Test Types

Available test types are:

- Acquisition only (no drive motion)
- Model Measurement (for creating models)
- Time History (time history drive for each DOF)
- Compensated Time History (uses model for compensation)

An **Acquisition Only** test is defined by a set of DOFs and a set of specimen acquisition channels. The response channels of the DOFs are sampled along with the specimen acquisition channels for the duration of the test. There are no drive signals. (This test is only available on systems equipped with a Pacific Instruments data acquisition system.)

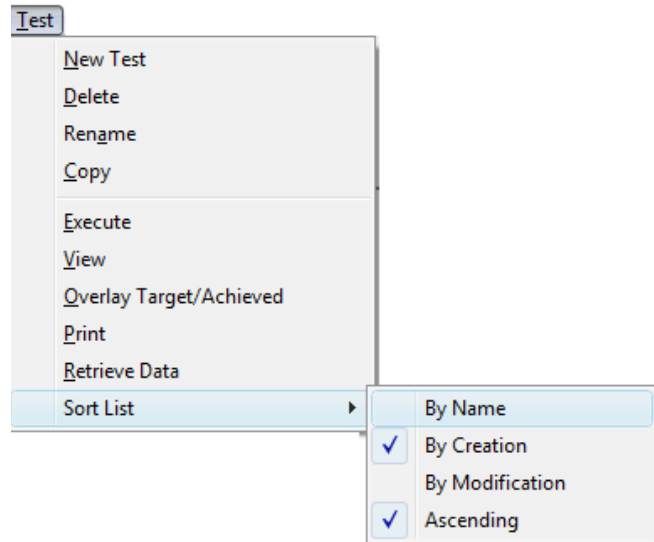
A **Model Measurement** test uses shaped random noise to drive each DOF independently (i.e. consecutively). The drive signals are specially designed to provide accurate model information so that compensation may be used to improve system response and drive correlation.

Time History and **Compensated Time History** tests have signals defined which drive the control system during a test, along with DOF and specimen acquisition parameters. A time history test includes drive signals for each DOF included for control. Adding compensation to a time history test involves using a model to attempt to remove the dynamic effects of the control system by adjusting the drive signal based on experience gained during a model measurement test.

When you double click on a test in the test list box or highlight the test and choose the “View” option from the Test pulldown menu, a test viewing dialog will open, allowing you to review the test parameters. Refer to, [“Viewing Test Parameters,”](#) on page 101.

Test Menu Options

Tests are created, executed, and deleted using the “Test” pull-down menu on the STEX main dialog.



The “Test” pulldown menu contains the following options:

- New Test (configure a new test)
- Delete (delete a test from the current group)
- Rename (rename a test)
- Copy (copy a test)
- Execute (execute a test)
- View (view the state of a test)
- Overlay Target/Achieved (create a plotting dialog with the target and achieved signals in an overlay plot)
- Print (print test signals)
- Retrieve Data (retrieve acquired test data from acquisition data)
- Sort List (Test list sorting options)

During test excitation, acquired data is written to a raw data file. After the test, STEX automatically reads this data file and creates the new ACHIEVED (and SPECIMEN) time histories. If this process is interrupted or cancelled, the time history records are not updated properly. If this should occur, the RETRIEVE DATA option allows the raw data file to be re-read, thereby updating the test signals. Note that since the raw data file is overwritten each time, Retrieve Data is limited to the last executed test.

Configuring New Tests

When you select “**New**” from the Test pulldown menu on the STEX main dialog, the **Configure Test** dialog appears.

Configure Test

Name:

Test Type:

Channel	Identifier	Full Scale
Long	Acceleration (ctrl)	15.000 g
	Acceleration (fdbk)	15.000 g

DOF: Mode (control): Mode (feedback):

☐ Specimen Control

Model:

☐ Filter Achieved

This dialog allows you to configure a new test by entering a test name, test type, and a set of controlled degrees of freedom (with control and feedback modes).

Enter the following parameters on this dialog:

1. **Name:** Enter a unique test name, up to 40 characters in length.

2. **Test Type:** Select one of the following test types:

Acquisition only - no drive motion (only available for PI data acquisition)

Model Measurement - use to create models for compensated tests

Time History - create a time history for each DOF

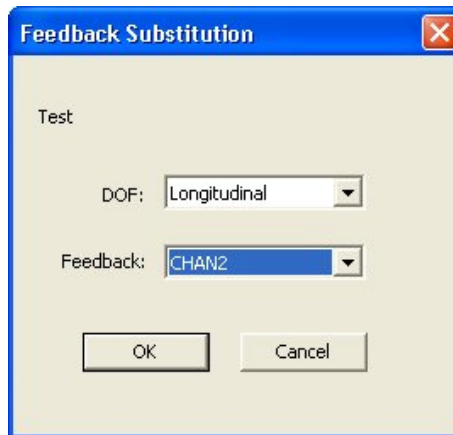
Compensated Time History - use a model for compensation

3. **DOF, MODE (control), MODE (feedback):** Select the control degrees of freedom and the control and feedback modes for each DOF.

Select a DOF, control mode, and feedback mode. When the **Add DOF** button is pressed, the selected **DOF, Mode (control)**, and **Mode (feedback)** is added to the DOF list box.

To delete a DOF from the list, highlight the DOF identifier in the list box and press the **Delete DOF** button.

To allow a customer's transducer feedback to be used instead of 469D DOF feedback for the test achieved signal, check the **Specimen Control** box. If selected, the following panel appears:

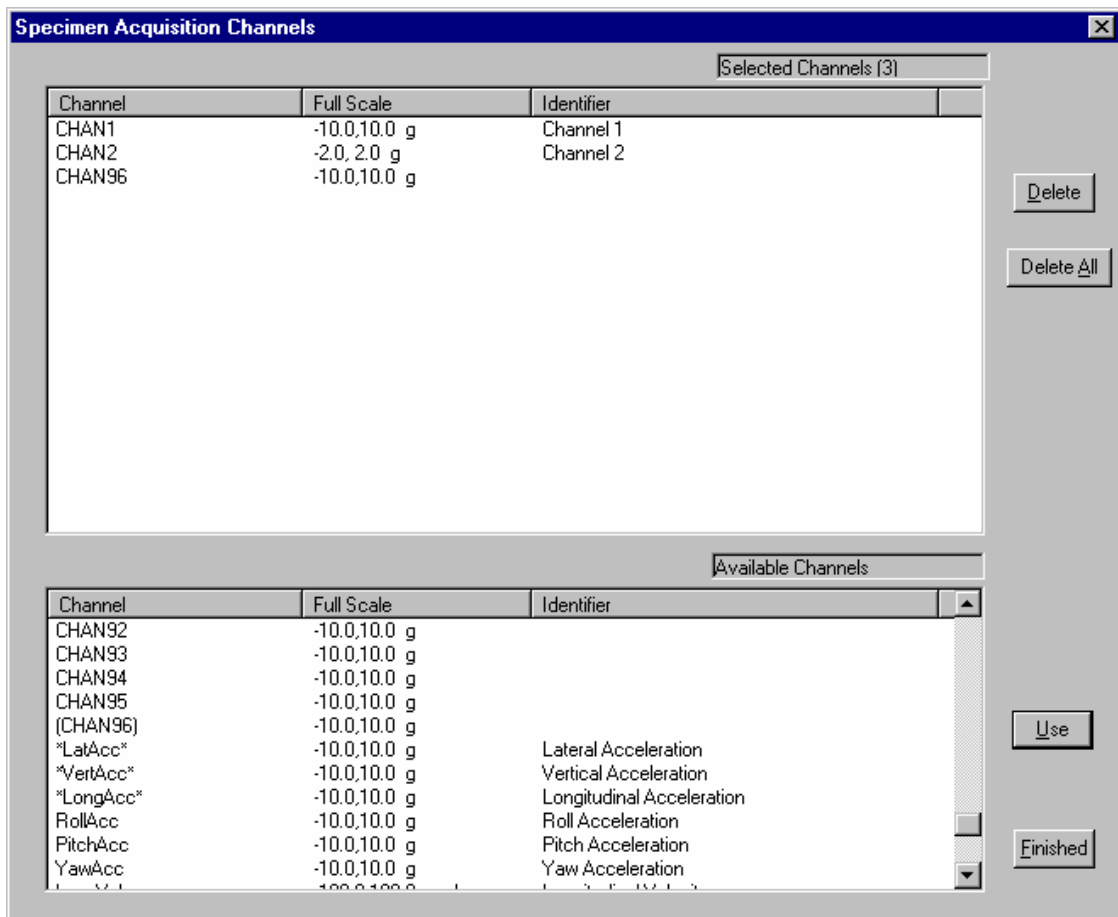


This panel provides two list boxes. The first contains an entry for each DOF selected for control in this test. The second box contains an entry for each specimen transducer included in this test. The operator must make a selection from each box.

After the test is completed, the previously recorded feedback for the selected **DOF** is discarded and overwritten with the data acquired from the specimen channel selected in **Feedback**.

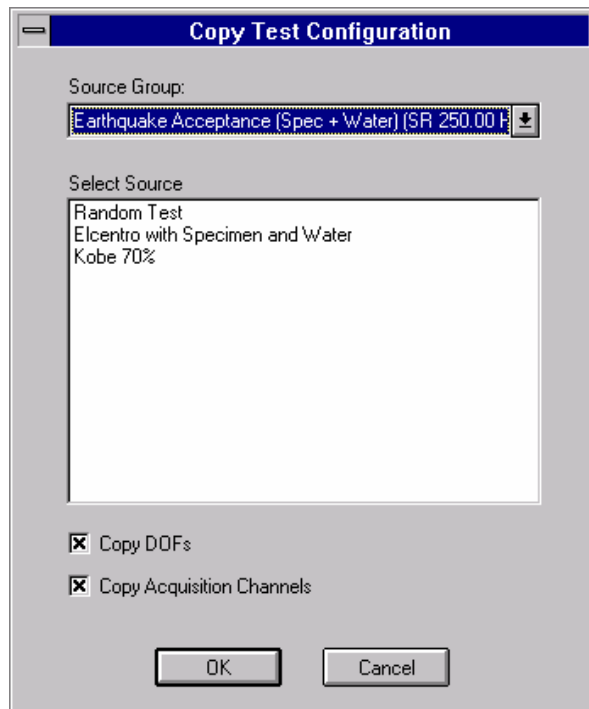
This panel requires that the units of the feedback channel match the control mode specified in the previous test configuration screen (g's or gal in most cases).

4. **Model:** Select a Model only if configuring a Compensated Time History. Enter the desired model name into the Model selection box. Pressing the Select button will open a selection box listing all available models. Refer to, [“Using the Select Option,”](#) on page 48 for information pertaining to using the Select button.
5. **Acquisition Channels:** Pressing this button will open up the **Specimen Acquisition Channels** dialog to allow you to select specimen acquisition channels.



The selected channels are listed in the upper box. To add other channels:

- A. Select a channel from the available channels selection box. Channels listed in parenthesis are already used and may not be selected. Channels preceded by an asterisk are active DOF feedback channels and are automatically acquired.
 - B. Select the “**Use**” button to add the channel to the selected channels list box.
 - C. To remove a channel from the selected channels list box, first select the channel in the selected channels list box and then select the “**Delete**” button. To delete all channels from the list box, select the “**Delete All**” button.
 - D. Select the “**Finish**” button to return to the Configure Test dialog.
6. **Copy:** Press this button to copy configuration parameters from an existing test. When this button is pressed, the **Copy Test Configuration** dialog will open.

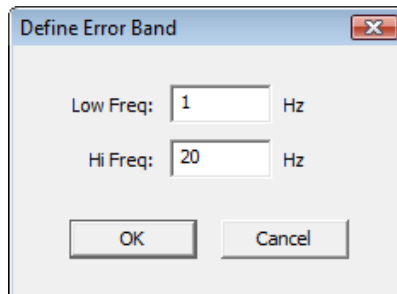


First select the group in which the source test exists and then select the source test. Check the desired boxes to copy DOFs and/or Acquisition channel information. Note that if any DOFs and/or acquisition channels have already been defined, you cannot copy this information from the source test.

Select the “**OK**” button to return to the Configure Test Dialog.

Press the “**OK**” button on the Configure Test Dialog after entering all desired parameters. A Test Setup dialog will appear.

7. **Filter Achieved:** Optionally, click the **Filter Achieved** check box when you want to filter the achieved signals of a compensated test for an error calculation. If selected, the following Define Error Band dialog appears:



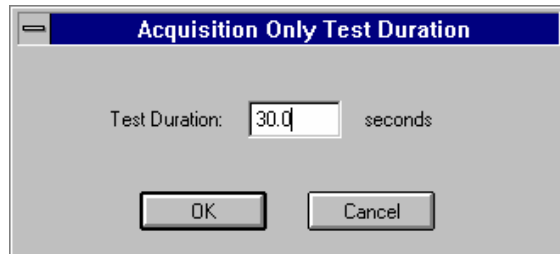
After each iteration, the error estimate is computed over the frequency band defined on the Define Error Band dialog.

Test Setup Dialogs

A test setup dialog is opened when the “**OK**” button is pressed on the Configure Test dialog. There is a different Test Setup dialog for each test type.

Acquisition Only Test Setup Dialog

When the test type is Acquisition Only, the **Duration** dialog is opened.



This dialog allows you to input the desired duration (in seconds) for the acquisition test. When you press the “**OK**” button, the test will be created. The STEX main dialog will reappear and the created test will appear in the Test list.

Model Measurement Test Setup Dialog

When the test type is Model Measurement, the **Define Random Motions** dialog is opened.

Test: Model Measurement Test

DOF	3 Sigma	Shape
Longitudinal	1.000000e+000 g	Random Shape
Lateral	1.000000e+000 g	Random Shape
Vertical	1.000000e+000 g	Random Shape

DOF:

Shape:

3 Sigma: g

Independent Frames:

Duration: 71.670 seconds (7168 points)

This dialog allows you to select an existing random shaping function (a real valued spectrum) along with a three-sigma level parameter for each degree of freedom.

Enter the following parameters on this dialog:

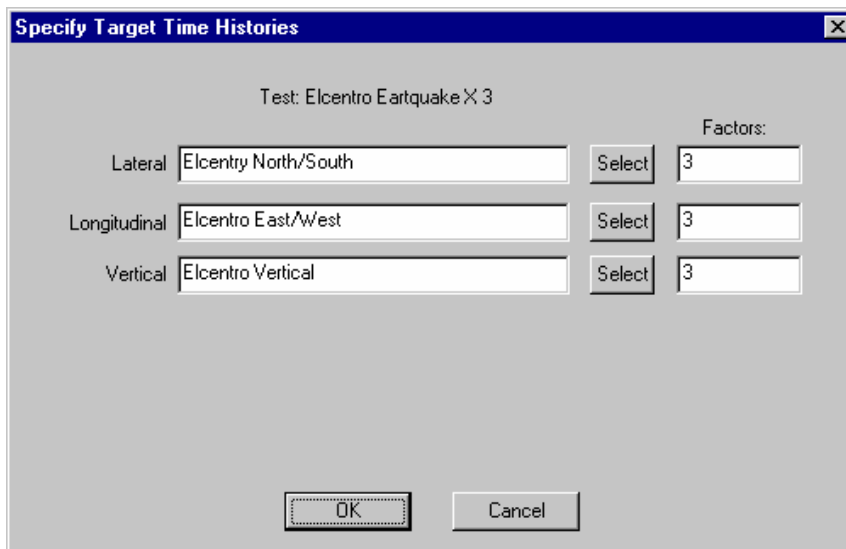
1. Choose the desired DOF in the **DOF** selection box.
2. Choose the desired shaping function for the selected DOF using the **shape** selection box. Shaping functions are created using the Shape option on the Analysis pulldown menu located on the STEX main dialog. Pressing the “**Select**” button will open a selection box listing all available shaping functions. Refer to , [“Using the Select Option,”](#) on page 48 for information pertaining to using the Select button.
3. Enter the desired 3 Sigma level into the **3 Sigma** text box.

4. Press the “**Use Shape**” button to enter the selected DOF and associated parameters into the DOF list.
5. Repeat steps 1-4 for each included DOF.
6. Enter the desired number of independent frames for the test into the **Independent Frames** text box. This value will default to one. Refer to Appendix B for information on random signals.
7. Use the “**Update**” button to calculate the duration of the test. The duration calculation is based on the number of independent frames and the frame size (derived from the shape function(s)).

When the “**OK**” button is pressed, the random signals will be created, one for each DOF. A vector will be created combining these time histories. The STEX main dialog will reappear, the created test will appear in the Test list and the created time histories and vector will appear in the Result list.

Time History and Compensated Time History Test Setup Dialog

When the test type is Time History or Compensated Time History, the **Specify Target Time Histories** dialog is opened.



The dialog box titled "Specify Target Time Histories" has a close button (X) in the top right corner. The main content area shows the test configuration: "Test: Elcentro Earthquake X 3". Below this, there are three rows of input fields for different DOFs: Lateral, Longitudinal, and Vertical. Each row has a text box for the time history name, a "Select" button, and a "Factors:" text box. The Lateral row shows "Elcentro North/South", "Select", and "3". The Longitudinal row shows "Elcentro East/West", "Select", and "3". The Vertical row shows "Elcentro Vertical", "Select", and "3". At the bottom of the dialog are "OK" and "Cancel" buttons.

DOF	Time History	Select	Factors
Lateral	Elcentro North/South	Select	3
Longitudinal	Elcentro East/West	Select	3
Vertical	Elcentro Vertical	Select	3

This dialog allows you to select an existing time history for each DOF included in the test configuration. Use the Factor fields to enter scaling factors to be applied to the specified time histories.

Enter the following parameters on this dialog:

1. Enter the desired time history for each DOF into the associated text box.
The referenced time history will be copied to create a target vector. You can either type in the exact name of the desired Time History or use the Select button to open a selection box listing all available time histories. Refer to , [“Using the Select Option,”](#) on page 48 for information pertaining to using the Select button.

When the “OK” button is pressed, the **Taper Target Motions** dialog will open.

The **Taper Target Motions** dialog allows you to add tapering to each target time history. You can specify the duration of the tapering at the beginning and the end of each time history. You may also choose no tapering.

Vector: [TARGET] Elcentro Earthquake Test

		Start Duration	End Duration	
Lateral	[TGT:Lat] Elcentro Earthquake Test	0	0	All
Longitudinal	[TGT:Lon] Elcentro Earthquake Test	0	0	
Vertical	[TGT:Ver] Elcentro Earthquake Test	0	0	
		0	0	
		0	0	
		0	0	

Plot Taper NO Taper

Enter the following parameters on this dialog:

1. Enter the desired duration, in seconds, of tapering at the start and end of each time history. The duration may be zero, if desired.
2. If all the time histories have the same tapering, you can use the “All” button to copy the Start and End duration's defined for the first time history to the duration fields for the remaining time histories.

3. Pressing the “**Plot**” button will display the time histories as they exist before tapering.
4. Press either “**Taper**” or “**No Taper**” button.

Choosing **No Taper** will create the test using untapered time histories. After the test is created, the STEX main dialog will reappear and the test name will appear in the Test list box. If the test is compensated, the loopback step will follow. Refer to, “[Compensated Test Loopback](#),” on page 72.

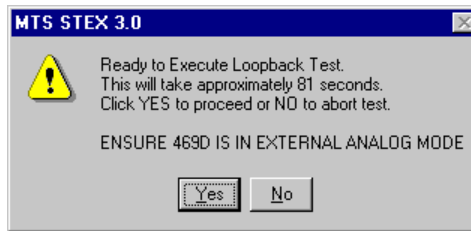
Choosing **Taper** will create tapered time histories along with a vector of these time histories. These time histories will be listed in the Results list box on the STEX main dialog. The test will be created using the tapered time histories. After the test is created, the STEX main dialog will reappear and the test name will appear in the Test list box.

Compensated Test Loopback

Note *This section applies only to systems equipped with a Pacific Instruments data acquisition system.*

During a test, DOF feedback signals are filtered using the anti-aliasing filters. This filtering can add errors into the compensation process unless the loopback is performed. The result of the loopback is the Desired vector which is created by passing the Target vector through the anti-aliasing filters. The Desired vector is used for compensation.

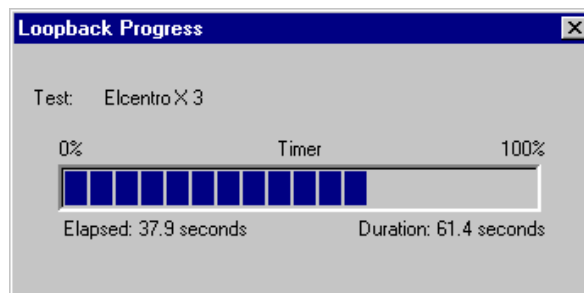
After the Taper Target Motions dialog and prior to the loopback the following message box appears (only on systems equipped with an MTS 469D digital control system):



This dialog displays the duration of the loopback test and allows you to continue with the loopback test or cancel. If you choose to cancel, the loopback test will automatically be run when you attempt to execute a compensated test.

Note *The 469D must be in external analog mode. This is not checked by STEX.*

When the “OK” button is pressed, the following dialog appears:



Executing Tests

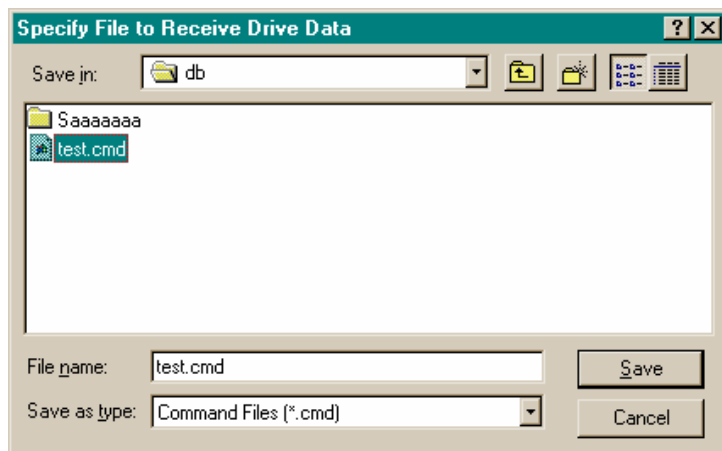
Test Execution begins when you choose the **“Execute”** option from the Test pulldown menu on the STEX main dialog. Test execution steps vary with test type and data acquisition hardware. [“Test Execution - No Data Acquisition Hardware”](#) on page 74 describes the steps that occur when there is no data acquisition hardware connected to the STEX system and [“Test Execution - Data Acquisition Hardware”](#) on page 76 describes the test execution steps that occur with data acquisition hardware.

Test Execution - No Data Acquisition Hardware

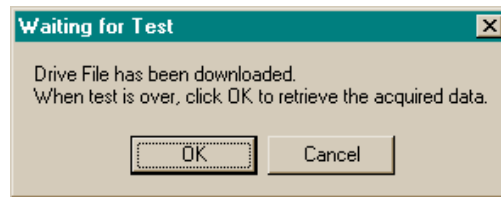
For systems without data acquisition hardware connected to the STEX system, a series of dialogs will appear to guide you through setting up and downloading drive files, performing test excitation using the real-time control system and importing acquired data into the STEX database.

All tests, with the exception of compensated time history tests, will begin with the Specify File to Receive Drive Data dialog. The compensated time history tests begin with compensated test dialogs which are used to effect the compensation process. These are followed by the Specify File to Receive Drive data dialog. Refer to , [“Compensated Test Dialogs,”](#) on page 88.

In order to execute a test, STEX prepares a file containing time histories which will be used to drive the control channels on the test system. The Specify File to Receive Drive Data dialog is used to specify the name of a file in which to save the drive data for use by the real-time control system. The dialog box which gathers this information is a standard Windows SAVE AS dialog box.



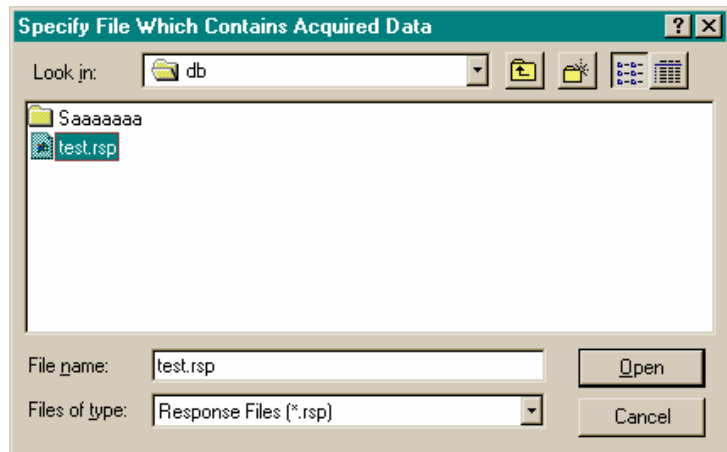
After you specify a drive file name, STEX will download the drive data into that file, and then notify you when the file has been downloaded.



At this point, use the 469D real-time control system to perform test excitation.

When the test has finished, click **“OK”** on the Waiting for Test Dialog to notify STEX that the real-time control system has finished executing the test.

After you click **“OK”**, STEX will prompt for the name of the file that contains the acquired data, using the following standard Windows Open dialog box.



Specify the location of the data that the real-time control system acquired. Once the file is specified, STEX will import the data, check it for consistency and then store it in the STEX database.

Test Execution - Data Acquisition Hardware

All tests, with the exception of compensated time history tests begin with a **Channels to Monitor** dialog which allows you to choose up to two channels to monitor during excitation. It is followed by a **Pretest Checks** dialog which displays information on the state of the test equipment and then a **Test Excitation** dialog which allows you to start and/or abort the test. Refer to the following discussions:

[“Channels to Monitor Dialog”](#) on page 81

[“Pretest Checks Dialog”](#) on page 82

[“Test Excitation Dialog”](#) on page 86

The compensated time history tests, begin with compensation test dialogs which are used to effect the compensation process. These are followed by the **Channel to Monitor**, **Pretest Checks**, and **Test Excitation** dialogs described in the preceding paragraph. Refer to, [“Compensated Test Dialogs,”](#) on page 88.

Before test execution begins, STEX checks the full-scale values, units and descriptors associated with each acquisition channel against the current contents of the transducer database. If discrepancies exist, you are given the opportunity to resolve them through a series of reconciliation dialogs. Refer to, [“Reconciliation Dialogs,”](#) on page 76. (Note that these dialogs apply to systems equipped with auxiliary data acquisition channels for recording specimen-mounted transducer signals.)

Reconciliation Dialogs

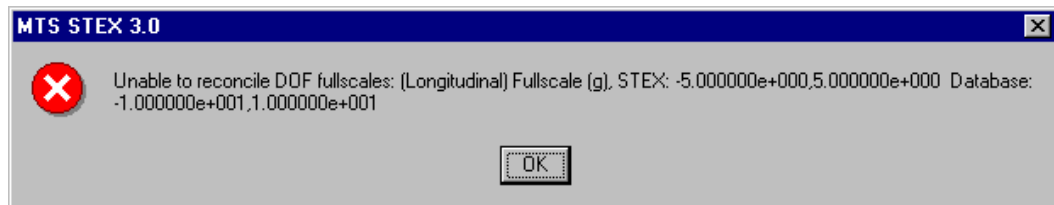
It is possible for a user to change information in the Transducer database between the time a test is defined and the time it is executed (e.g. transducers are recalibrated). This has potentially serious ramifications on the proper execution of the test, since the interpretation of control signals and acquired data signals are both scaled based on the calibrated full-scale value of the appropriate channel. STEX will attempt to identify such discrepancies when you execute a test, and will give you the opportunity to resolve the discrepancies using Reconciliation dialogs.

When the user chooses **“Execute”** option from the Test pulldown menu on the main dialog, STEX checks the values in the internal STEX database against the external database used to maintain transducer setup and calibration information. If a discrepancy exists one of the following dialogs will open.

Control DOF Channel Discrepancy

The most serious discrepancy occurs when a DOF channel is involved, since the scaled data directly affects the magnitude of the signal used to drive the system's actuators.

The following warning message will be displayed advising you of the seriousness of this discrepancy.

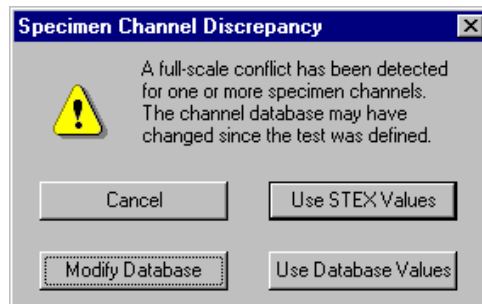


The dialog indicates that the feedback channel full-scale information is invalid for the Longitudinal DOF; the differing values are given. After pressing “OK” the test execution is canceled. The source of the discrepancy must be determined and fixed. The likely cause is a corrupt database file, however, the full-scale values for DOF feedbacks must agree with values defined for the control system. Great care is recommended in determining the proper full-scale values as these should have been defined during installation and rarely, if ever, change. If you are unsure as to how to proceed, contact MTS Service.

Specimen Channel Discrepancy

The Specimen Channel Discrepancy dialog is opened when STEX detects a discrepancy between the internal STEX database and the external Transducer database for channels other than control DOF. There will be different dialogs for different data acquisition systems as described in the following sections.

Specimen Channel Discrepancy Dialog – Pacific Instruments Data Acquisition



The following options exist for reconciling the channel database with the STEX test record:

- Cancel
- Use values from test record
- Use values from channel database
- Modify database manually, one channel at a time

Cancel

Choosing this option results in changes to neither the STEX test record nor the channel database. The discrepancy will still exist and will reappear the next time the test is executed.

Use Values from Test Records

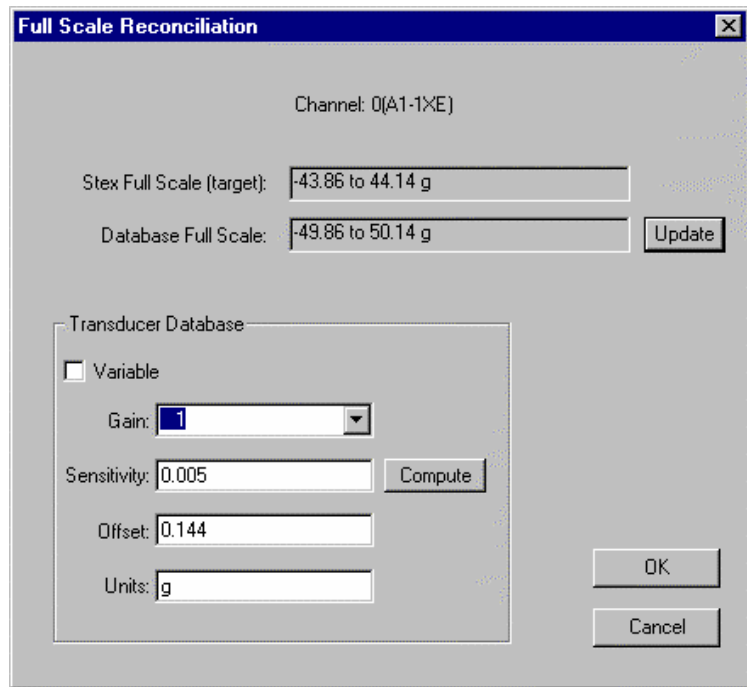
The channel information stored with the test record is copied to the channel database and execution proceeds. Note that the channel information stored in the test record originated when the test was created or the last time a reconciliation occurred.

Use Values from Channel Database

The channel information stored in the channel database is copied to the STEX test record.

Modify Database Manually, One Channel at a Time

For each channel for which a discrepancy was determined, the following Full-Scale Reconciliation dialog appears. You may adjust full-scale parameters for each channel until the discrepancies are eliminated. Note that this dialog may be slightly different for some data acquisition systems due to differences in hardware capabilities.



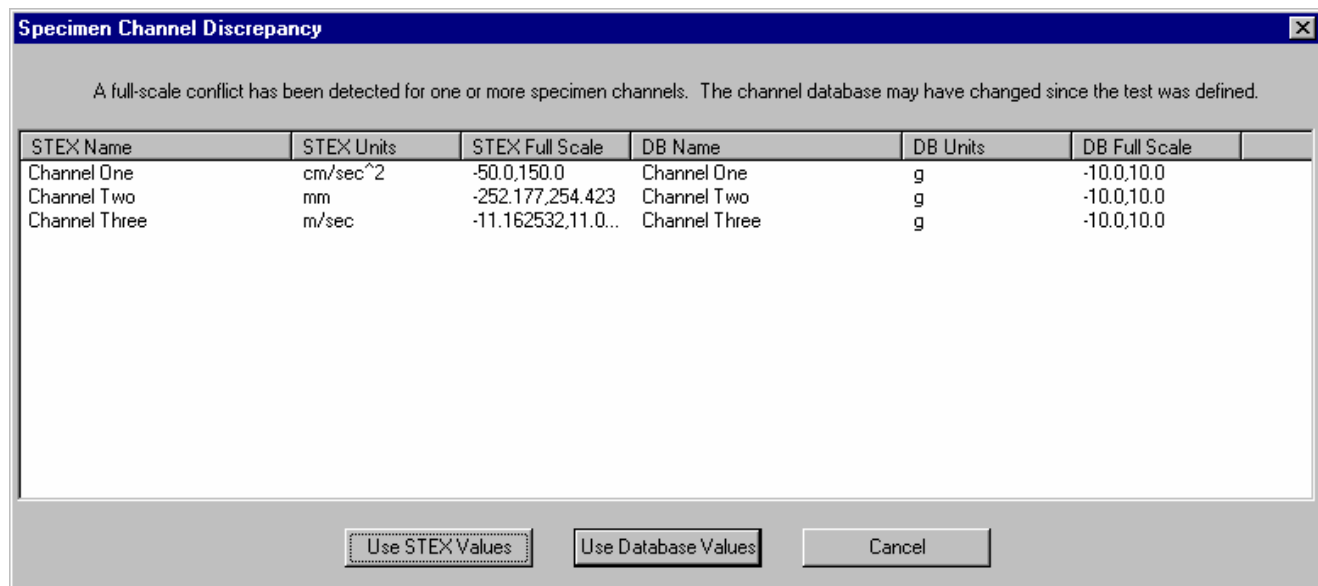
The dialog box is titled "Full Scale Reconciliation" and contains the following elements:

- Channel:** 0(A1-1XE)
- Stex Full Scale (target):** -43.86 to 44.14 g
- Database Full Scale:** -49.86 to 50.14 g
- Update** button
- Transducer Database** section:
 - ☐ Variable
 - Gain:** 1 (dropdown menu)
 - Sensitivity:** 0.005
 - Offset:** 0.144
 - Units:** g
 - Compute** button
- OK** button
- Cancel** button

Along with the channel identifier, the full-scale values are given. The purpose of this dialog is to adjust channel information in the channel database to agree with the STEX test record full-scale.

You may adjust the various parameters manually, or you may select “**Compute**” to have the parameters set automatically. Selecting “**Update**” causes the Database Full-Scale value to be recomputed. When this value agrees with the STEX Full-Scale, you may proceed by pressing “**OK**”. If other discrepancies exist, this dialog will reappear, updated with another channel. If no other discrepancies exist, test execution continues.

Specimen Channel Discrepancy Dialog – MTS 498 and 493 Data Acquisition



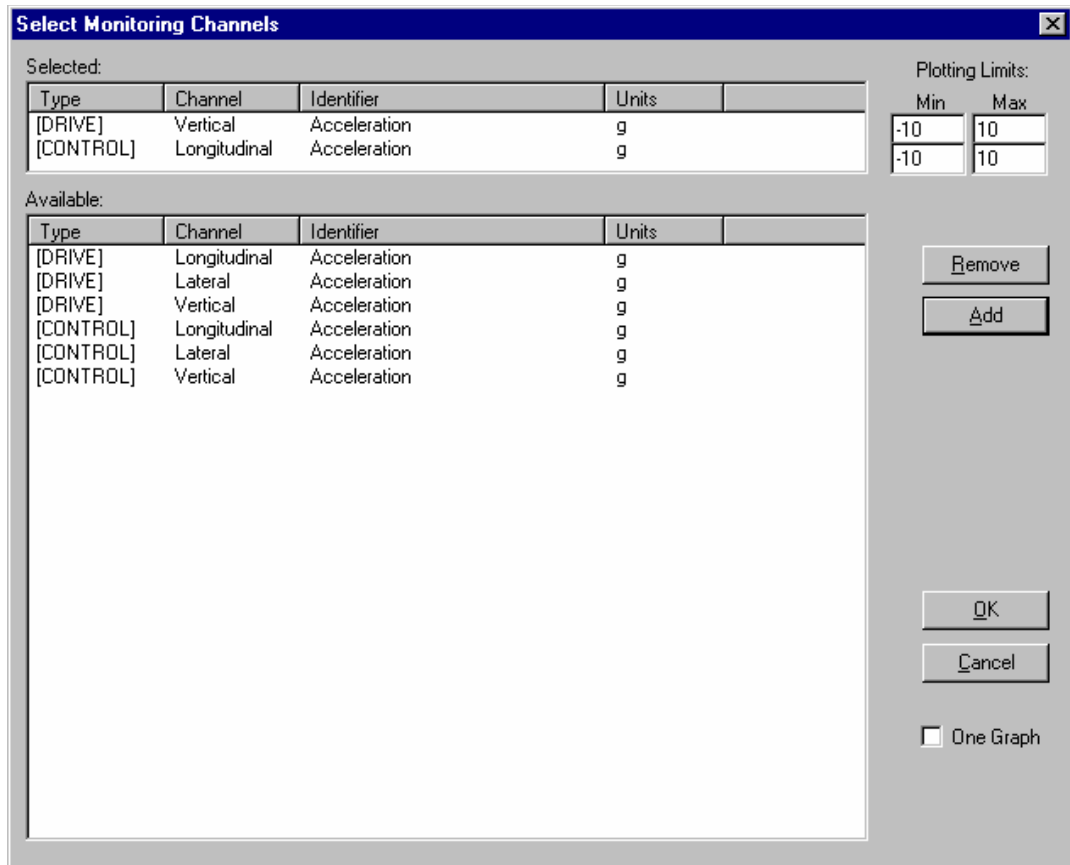
The Specimen Channel Discrepancy dialog will appear if, after you have selected a test for execution, STEX determines that the channel information in its test record does not match the channel database file. This dialog lists the details of the problem channels.

The following options exist for reconciling the channel database with the STEX test record:

1. Use STEX Values – the database will be changed to reflect the values in the STEX record.
2. Use Database Values – the test record will be changed to reflect the values in the database.
3. Cancel – no changes are made and the execution is cancelled.
4. Double-click any channel in the STEX Name column to open a Channel Settings dialog (described in, [“Channel Setup - MTS 498/493 Data Acquisition System,”](#) on page 95) and manually edit the full-scale parameters for that channel. The discrepancy list will be updated when the Finished button is pressed. If no discrepancies are found, the dialogs disappear and test execution proceeds.

Channels to Monitor Dialog

When you select “Execute” from the Test pulldown menu, the **Channels to Monitor** dialog will appear. If you are selecting a compensated test for execution the **Compensated Test** dialog will appear instead. Refer to, “[Compensated Test Dialogs](#),” on page 88.



This dialog allows you to select up to two channels to monitor during test excitation.

To select a channel, highlight a channel in the available list and press the “**Add**” button.

To delete a channel from the top Selected Channels list, first highlight the channel and then press the “**Remove**” button.

When channels are selected for monitoring, the Plot Limit fields are set to the channel full-scale values. You may change these limits to effect the way they are displayed during the test.

Press the “**OK**” button to continue with test excitation.

Pretest Checks Dialog

When you select “**OK**” on the Channels to Monitor dialog, the **Pretest Checks** dialog will appear.

Pretest Checks - Elcentro

Drive Channels:

DOF	Scaling	Full-scale	Violations
Longitudinal	100%	10.0 g	0
Lateral	100%	10.0 g	0
Vertical	100%	10.0 g	0

Warnings:

- Longitudinal axis is set to TVC displacement.
- Longitudinal axis has wrong mode.
- Longitudinal Amplitude Control is ON
- Lateral axis is set to TVC displacement.
- Lateral axis has wrong mode.
- Vertical axis is set to TVC displacement.
- Vertical axis has wrong mode.
- Vertical Computer Control NOT selected
- HSM is not on.
- Table not in normal position
- Program ramp is not at 10 Volts.
- Program ramp is at 0 Volts.
- System is not in Range 1.

Scale Plot Kinematics

Acquisition Channels:

Channel	Identifier	Full-scale	Filter
Longitudinal	Acceleration	10.0 g	40.00 Hz
Lateral	Acceleration	10.0 g	40.00 Hz
Vertical	Acceleration	10.0 g	40.00 Hz

Recheck Continue Abort

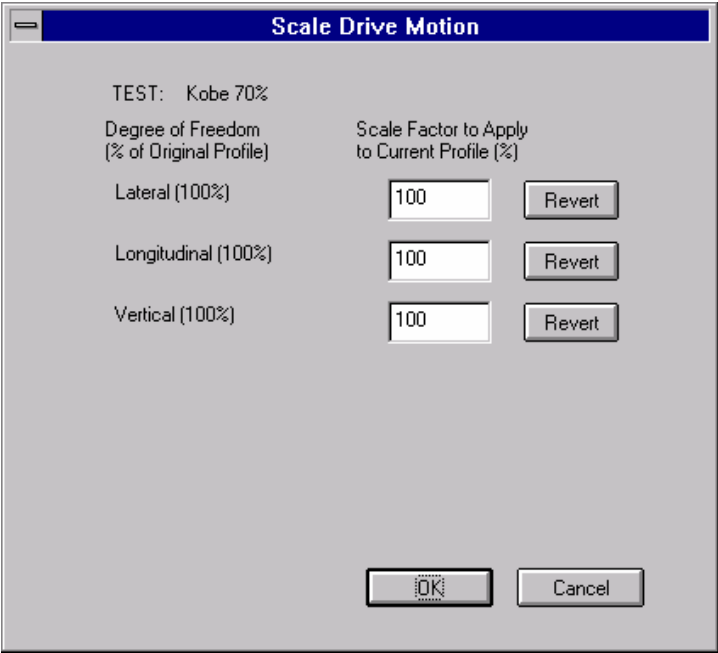
This dialog provides the following information:

- The set of controlled DOFs, current scaling of each DOF, in percent, along with full-scale value
- Full-scale violations per DOF; a full-scale violation occurs when the DOF time history exceeds the full-scale value recognized by the system control hardware. When this occurs, the DOF time history should be scaled using the “Scale” button. See below.
- The set of specimen acquisition channels along with the full-scale value and filter setting for each channel
- Test equipment status warnings. These warning situations should be eliminated before executing the test. The warning list can be updated by pressing the “**Recheck**” button.

The “**Recheck**” button will update the warnings list and the full-scale violations list.

The “**Plot**” button will plot the time histories using the Plot dialog. Refer to, “[Using the Plotting Option,](#)” on page 41.

The “**Scale**” button will open the Scale Drive Motion dialog, allowing you to apply a scale factor to each time history.



The dialog box is titled "Scale Drive Motion" in a blue header bar. It contains the following elements:

- TEST:** Kobe 70%
- Degree of Freedom (% of Original Profile)** (left column)
- Scale Factor to Apply to Current Profile (%)** (right column)
- Three rows of input fields for Lateral, Longitudinal, and Vertical degrees of freedom, each with a "Revert" button.
- At the bottom, "OK" and "Cancel" buttons.

Degree of Freedom (% of Original Profile)	Scale Factor to Apply to Current Profile (%)	Action
Lateral (100%)	100	Revert
Longitudinal (100%)	100	Revert
Vertical (100%)	100	Revert

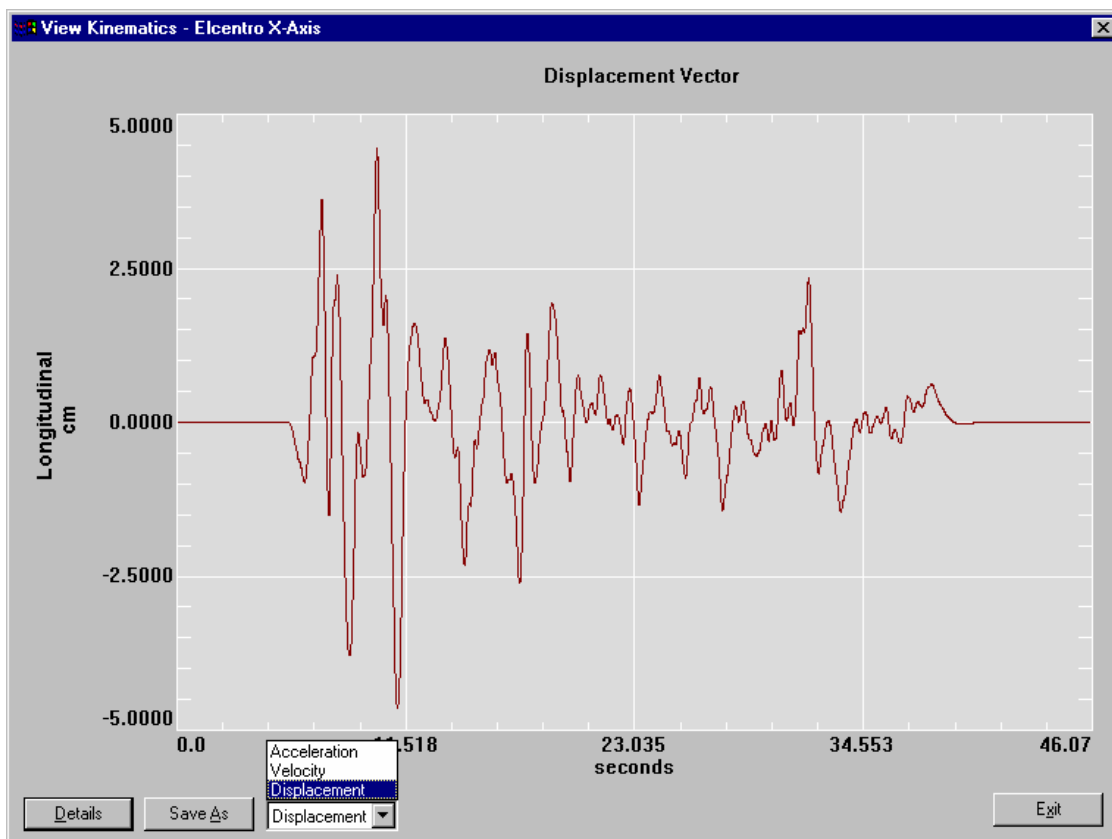
Enter the desired scale factors (in %) to be applied to the current DOF time histories. The current scale factor (% of the original time history) is displayed beside each DOF.

Pressing the “**Revert**” button will cause the DOF time history to revert back to its original level (apply original scale factor).

Press “**OK**” to apply the new scale factors to each DOF time history and return to the Pretest Checks dialog. Then press the “**Recheck**” button on the Pretest Checks dialog to update the Full-scale violations list.

When the system warnings and violations are eliminated, press the “**Continue**” button on the Pretest Checks dialog to continue with test excitation. Pressing “**Abort**” will cancel the execution and return to the STEX main dialog.

The “**Kinematics**” button will open the **View Kinematics – Elcentro X-Axis** dialog, displaying the estimated velocity, acceleration and displacement. Note that one of these three selections was given by the user at test definition and is used to calculate the other two parameters.



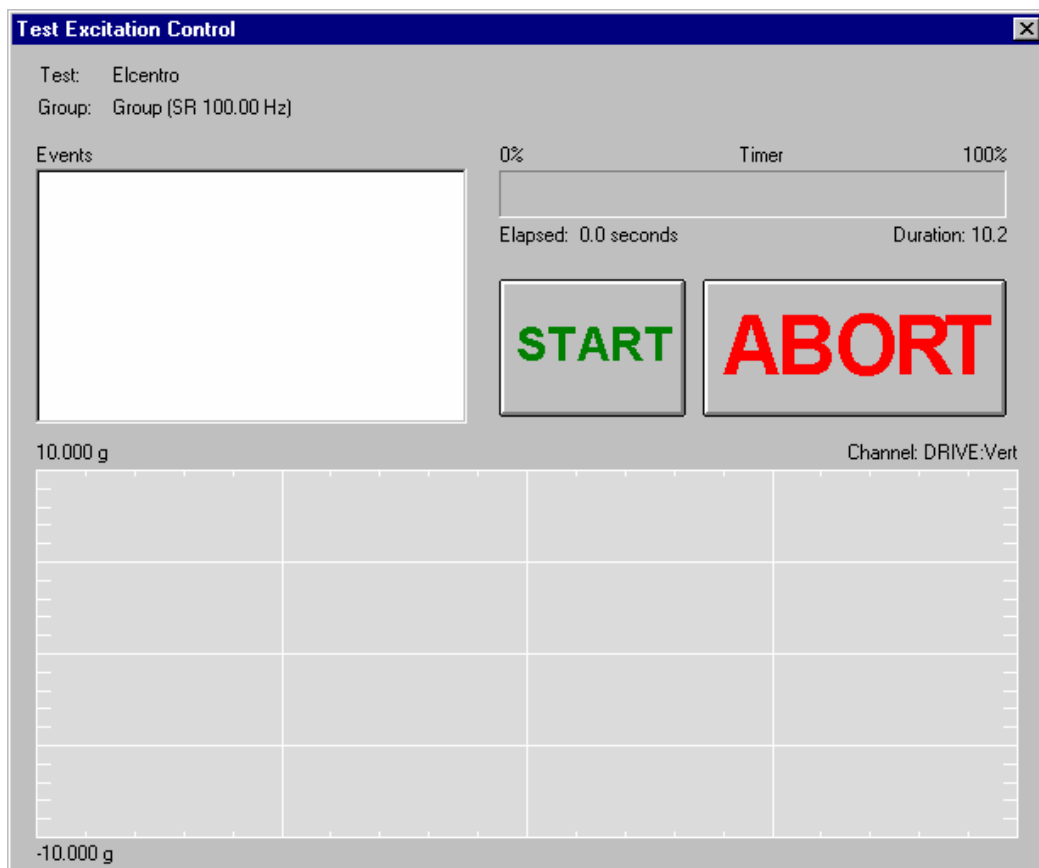
Use the list box to select the signal to be displayed.

Pressing the “**Details**” button will open a statistics dialog for the signal currently displayed.

Pressing the “**Save As**” button will export the displayed signal to an external file.

Test Excitation Dialog

When you press “Continue” on the Pretest Checks dialog, the **Test Excitation Control** dialog will open.

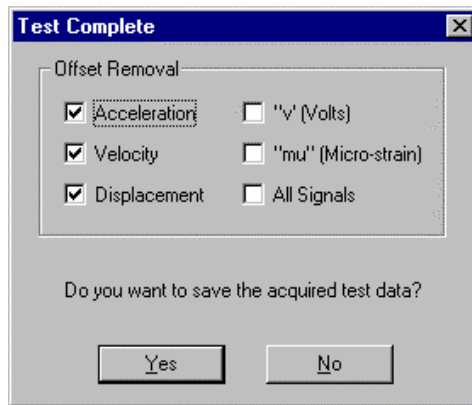


This dialog displays test events, a test progress meter, and the channels that were selected on the Channels to Monitor dialog.

Press the “**Abort**” button to cancel the test execution and return to the STEX main dialog.

Press the “**Start**” button to begin the test execution. You can monitor test execution using the progress meter and displays on this panel.

When the test has completed, the following Test Completion dialog will appear.



You have the option to specify how post-processing proceeds by indicating which types of specimen data have their means removed.

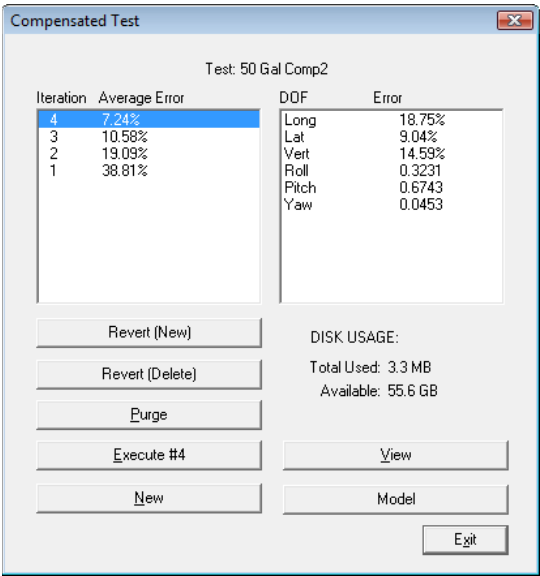
Pressing “**Yes**” button will save the data acquired during the test. This data will overwrite any pre-existing data.

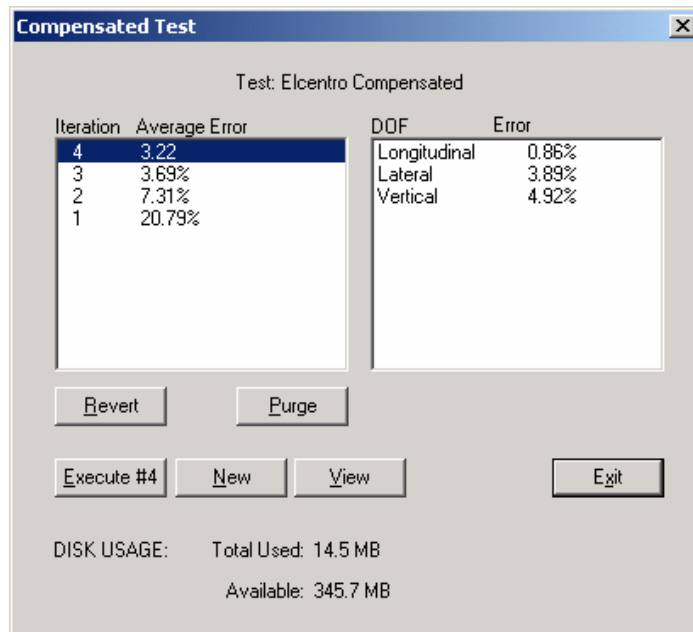
Pressing “**No**” button will discard the data acquired during the test.

Pressing either the “**Yes**” or “**No**” button will also close the test window and return either to the main dialog or the compensated test dialog (if running a compensated test).

Compensated Test Dialogs

Whenever you select the compensation test for execution, the **Compensated Test** dialog appears.





This dialog provides the following information:

- the left list box displays the iterations that have been performed along with the average error RMS for each iteration
- the right list box displays the error RMS for each degree of freedom for the iteration selected in the left list box
- the amount of disk space used by the test and the available disk space is displayed

Selecting a non-current iteration in the left list box and then selecting the **“Revert”** button will delete all iterations (drive and achieved signals) that have occurred after the selected one.

Revert (New) button

Selecting a non-current iteration in the left list box and then selecting the **“Revert (New)”** button will create a new iteration with the same drive as an existing previous iteration. Intervening iterations are not destroyed.

Revert (Delete) button

Selecting a non-current iteration in the left list box and then selecting the **“Revert (Delete)”** button will make the selected iteration current by deleting all intervening iterations.

Purge button

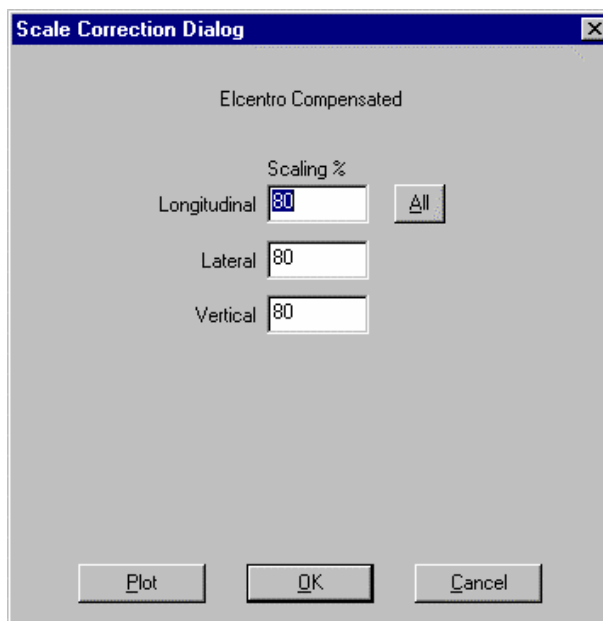
Selecting **“Purge”** will delete all iterations except the current one. delete all iterations (drive and achieved signals) that have occurred after the selected one

Execute button

Selecting “**Execute**” will execute the test using the most current iteration and drive. For systems with data acquisition hardware, the **Reconciliation** dialogs will appear, if necessary, followed by the **Channels to Monitor** dialog, the **Pretests Checks** dialog and then the **Test Excitation** dialog. Refer to “[Channels to Monitor Dialog](#)” on page 81 through “[Test Excitation Dialog](#)” on page 86. For systems without data acquisition hardware, special dialogs will appear to guide the user through the test execution process. Refer to “[Test Execution - No Data Acquisition Hardware](#)” on page 74.

New button

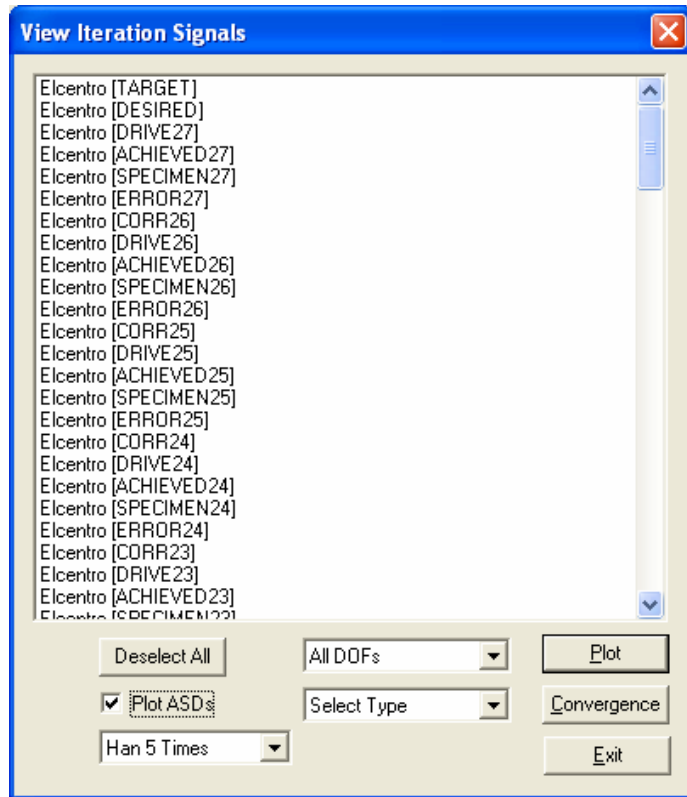
Selecting “**New**” will create a new iteration with a new drive. When you select “**New**”, the **Scale Correction** dialog appears.



This dialog displays a correction scale factor for each DOF. The correction is computed based on the error signal of the previous execution and the model associated with the test. Typical scaling is between 20% and 80%, with difficult tests having a scaling factor of 20%.

You may plot and scale the correction before applying the correction to the drive signal.

View button Selecting “**View**” will display the **View Iteration** dialog.



This dialog lists all of the vectors that exist for the test including old drive and achieved signals. Select one or more of these signals and then press the “**Plot**” button and a plotting dialog will appear for whichever signal(s) you selected.

Check the **Plot ASD** box to convert the plot to the frequency domain.

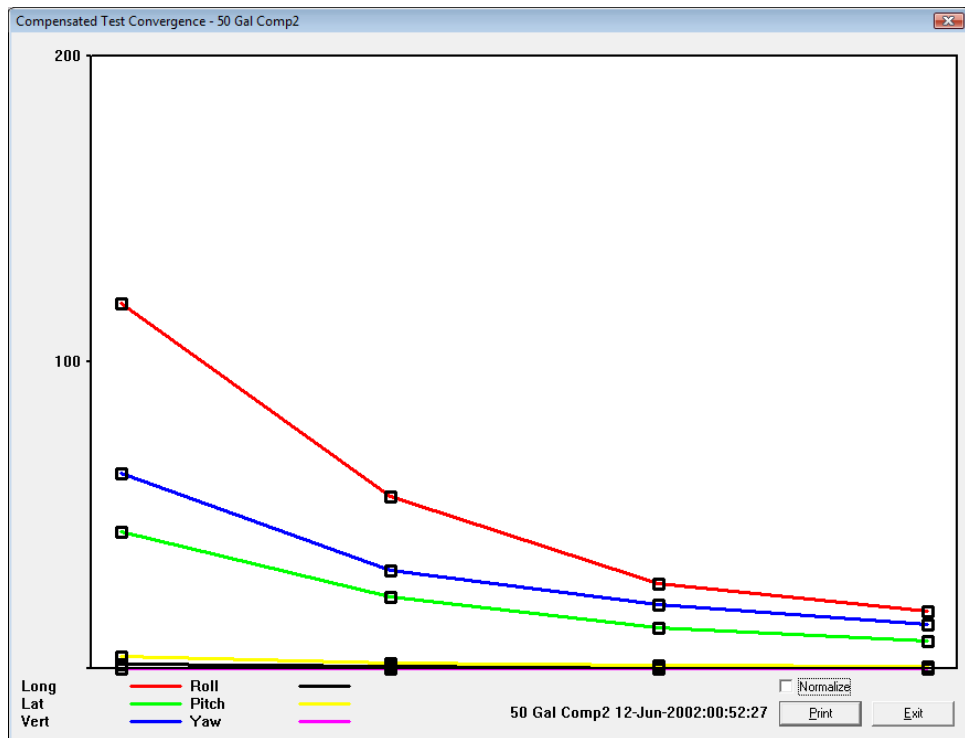
For ASD plots, the number of hannings (smoothing) can be selected: none, 1, 5, 10, 50, 100 times.

“All DOFs” may be plotted or any one of the DOFs defined in the test.

The **Select Type** option has: Achieved, Error, Correction, or Drive. When selected, this will highlight all appropriate signals in the list above.

Deselect All will simply un-select any signals in the list.

When you select the “**Convergence**” button and two or more iterations have been completed, the error history plotting dialog appears.



The error history plot shows the history of the error RMS values for each DOF for all iterations completed. The error RMS for each DOF is plotted against iteration number.

Check the **Normalize** box to normalize each DOF by the error of the first iteration.

Click the **Exit** button to exit the error history plot dialog or select the **Print** button to print the image.

Model button

Selecting “**Model**” allows you to specify a new model record for this compensation.

Channel Setup Dialogs

Channel Setup – Pacific Instruments Data Acquisition

You can modify the Transducer database directly by choosing the **Channel Setup** selection on the **Hardware** pull-down menu. The Transducer Database Channel Parameter dialog will open allowing you to view and manipulate the database.

The screenshot shows the 'Transducer Database Channel Parameters' dialog box. At the top, there's a title bar with the text 'Transducer Database Channel Parameters' and a close button. Below the title bar, there are two main sections. The first section contains a 'Channel' dropdown menu with '1' selected and a 'Full Scale' text box with '10000.00 mV'. The second section is titled 'Transducer Database' and contains several fields: 'Name' (CHAN0), 'Description' (Unknown), 'Location' (Unknown), 'Variable' (checkbox), 'Gain' (dropdown menu with '1'), 'Sensitivity' (text box with '1'), 'Offset' (text box with '0'), 'Units' (text box with 'mV'), and 'Filter' (dropdown menu with 'Wideband'). At the bottom right of the dialog, there are 'OK' and 'Cancel' buttons.

Choose the channel to be viewed or edited in the **Channel** drop down list. When this field is highlighted, the up and down arrow keys can be used to scroll through the channels. DOF channels are identified by their DOF designation (e.g. Lateral Acceleration). No distinction is made between DOF channels and specimen acquisition channels.

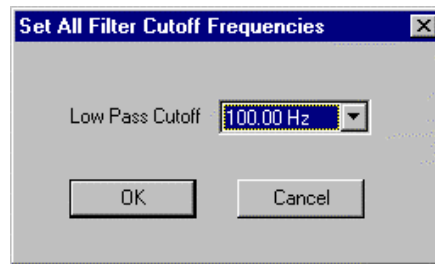
The **Full-Scale** indicator is a read only text box that is updated whenever any of the parameters affecting full-scale are changed. These include the Gain, Sensitivity, Offset, and Unit parameters.

The **Variable Gain** check box is used to indicate when a gain is entered that does not correspond to one of the predetermined gain levels in the list box. The box is checked (or unchecked) automatically depending on whether the user selects a gain level from the list box or types in a level that is not in the list.

The Name, Location and Description are the three separate components of descriptive information that are stored in the Transducer Database. When viewed in STEX, these components are concatenated together, separated by colons.

The Filter list box contains a list of available filter settings for the channel. The user must choose a setting from the list. A filter setting would typically be chosen which is less than the Nyquist frequency. In addition, the list typically contains a Wideband setting, which is equivalent to no filtering.

You can change the filter setting for all channels by choosing the **Set Filters** selection on the **Hardware** pull-down menu. The **Set All Filter Cutoff Frequencies** dialog will open allowing you to select a low-pass cutoff frequency value.



When you select OK, the selected low-pass cutoff frequency value will be applied to all channels in the channel database.

Channel Setup - MTS 498/493 Data Acquisition System

If your system is equipped with an MTS 498 Data Acquisition system, the Conditioner Calibration panel will open when you choose the **Channel Setup** selection on the **Hardware** pull-down menu.

Conditioner - Calibration Panel

Select:

- CHAN1
- CHAN2
- CHAN3
- CHAN4
- CHAN5
- CHAN6
- CHAN7
- CHAN8
- CHAN9
- CHAN10
- CHAN11
- CHAN12

Description:

☒ DC Conditioner

Internal Connections:

Readout:

Input:

Output:

Filter: ☒

Calibration Adjustments:

Excitation: $\pm$ V

Gain: V/V

AC Conditioner Parameters:

Demod. Phase: Deg

Coarse Zero: V

Fine Zero: V

Delta K (+): V/V

DC Conditioner Parameters:

Bridge Balance: V

Offset Zero: V

Units:

Scale Factor:

Offset:

Fullscale: -10.0,10.0 g

Conditioner Output:

Volts

g

Use this panel to calibrate the MTS 497 Series AC and DC Conditioner boards used in the 498 Data Acquisition system. Refer to the 497 Product Manual for information regarding the settings on this panel. Note that the Filter checkbox should always be enabled.

If your system is equipped with an MTS 493 Data Acquisition system, the following Conditioner Calibration panel will open when you choose the **Channel Setup** selection on the **Hardware** pull-down menu.

Conditioner - Calibration Panel

Select Channel:

- 12Z+.76Y-
- CHAN2
- CHAN3
- CHAN4
- CHAN5
- CHAN6
- CHAN7
- CHAN8
- CHAN9
- CHAN10
- CHAN11
- CHAN12

Filter: 100 Hz

Excitation: ± 0.0000 V

Gain: 0.0000 V/V

Delta K (+): 0.9500 V/V

Zero: 0.0000 V

Name: 12Z+.76Y- Set

Description: Channel One

Type: DC Conditioner

Units: V

Scale Factor: 10.000000

Offset: 0.000000

Fullscale: -10.0,10.0 V

☐ Shunt ☐ DIO ☐ DAC ☐ ADC ☐ Invert Polarity

Output: 3.000000 Volts 3.000000 V

Finished

This panel is quite similar to that of the 498 version on the previous page. There are slight differences owing to the different features of a 493 DUC versus the 497 AC/DC conditioners. Note that the filter should be selected appropriately.

The desired channel is chosen from the list located in the upper left corner. You can change the identifier for a channel by double-clicking on the selected channel. The following dialog will appear:



You may define a new identifier for the channel. After selecting OK, the calibration panel will become active, and the list of channels will be updated.

Selecting the “Digital I/O” button on the 498 Conditioner Calibration panel or the “DIO” button on the 493 Conditioner Calibration panel results in the Digital I/O dialog.

Digital I/O

STEX Digital Outputs

0 ☐ Run/Stop	4 ☐ Source Inhibit (Vert)
1 ☐ Source Inhibit (Long)	5 ☐ Source Inhibit (Pitch)
2 ☐ Source Inhibit (Yaw)	6 ☐ Source Inhibit (Roll)
3 ☐ Source Inhibit (Lat)	7 ☐ Loopback Enable

STEX Digital Inputs

0 ☒ HSM is not on.	16 ☒ Ampl. Control (Lat)
1 ☐ Table not in normal position	17 ☐ Computer Selected (Vert)
2 ☐ Program ramp is not at 10	18 ☒ Mode Bit 1 (Vert)
3 ☒ Program ramp is at 0 Volts.	19 ☒ Mode Bit 2 (Vert)
4 ☐ System is not in Range 1.	20 ☐ Ampl. Control (Vert)
5 ☒ Computer Selected (Long)	21 ☒ Computer Selected (Pitch)
6 ☒ Mode Bit 1 (Long)	22 ☐ Mode Bit 1 (Pitch)
7 ☐ Mode Bit 2 (Long)	23 ☒ Mode Bit 2 (Pitch)
8 ☒ Ampl. Control (Long)	24 ☐ Ampl. Control (Pitch)
9 ☒ Computer Selected (Yaw)	25 ☒ Computer Selected (Roll)
10 ☐ Mode Bit 1 (Yaw)	26 ☒ Mode Bit 1 (Roll)
11 ☒ Mode Bit 2 (Yaw)	27 ☐ Mode Bit 2 (Roll)
12 ☒ Ampl. Control (Yaw)	28 ☒ Ampl. Control (Roll)
13 ☒ Computer Selected (Lat)	29 ☐
14 ☒ Mode Bit 1 (Lat)	30 ☒
15 ☐ Mode Bit 2 (Lat)	31 ☒

Set All

Reset All

Sample DIO

Exit

Digital Outputs are shown in the upper area and the Digital Inputs are shown in the lower area. You may change a Digital Output by setting the appropriate check-box. All Digital Outputs may be set or reset using the buttons to the right.

Selecting “Sample DIO” causes all of the Digital Inputs to be read and the display updated.

Selecting the “Set DACs” button on the 498 Conditioner Calibration panel or the “DAC” button on the 493 Conditioner Calibration panel results in the Set D/As dialog.

The dialog box titled "Set D/As" contains a table with columns for "Voltage (+/- 10)", "Accel", "Vel", and "Displ". Each row represents a degree of freedom (DOF) with a "Set" button next to the voltage input field. Below the table, there are "Set All" and "Exit" buttons. The unit "(volts)" is indicated at the bottom right of the table.

	Voltage (+/- 10)	Accel	Vel	Displ
Longitudinal	1	0.083	0.080	0.010
Lateral	2	0.049	0.080	0.054
Vertical	3	0.033	0.042	0.039
Roll	4	0.032	0.056	0.013
Pitch	5	0.066	0.056	0.040
Yaw	6	0.003	0.003	0.050

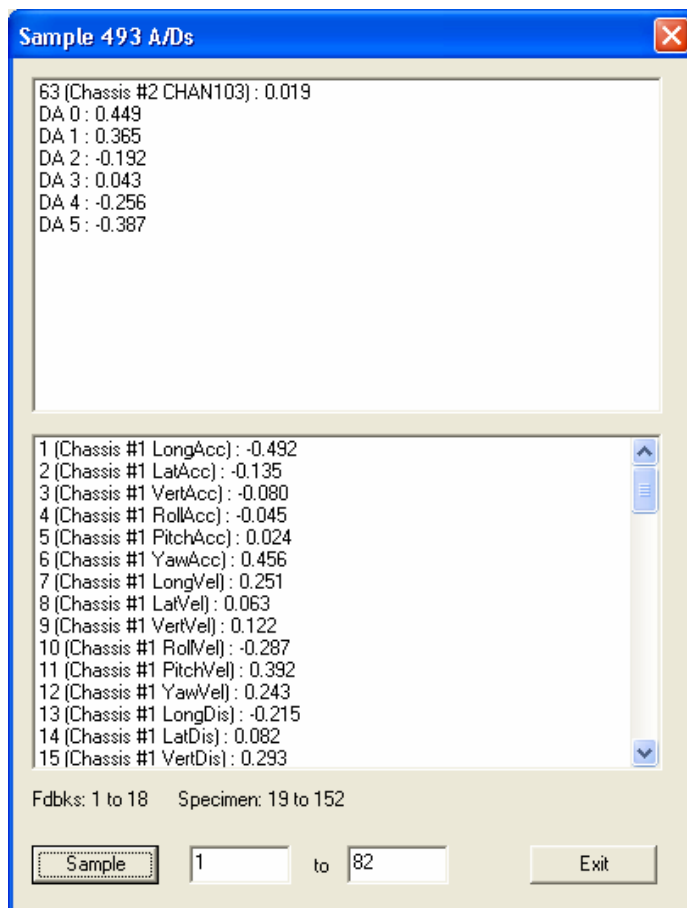
(volts)

When this dialog first appears, STEX measures the current feedback level for all channels and displays the voltage value for each. This measurement is repeated whenever a D/A output is set.

This dialog allows you to set the current voltage output of the D/A converters.

Initially, the output levels are not known, so you must use “Set All”. After this, the Set button for each DOF will be enabled, and individual channels may be modified.

Selecting the “ADC” button on the 493 Conditioner Calibration panel results in the Sample 493 A/Ds dialog.



The operator may select a range of A/D and then Sample. Then, all of the selected channels are sampled and the results are broken into two lists. (Units are in volts.)

The top list contains all of the D/As, AND A/D channels whose voltages are near zero. The lower list contains any A/D channels whose voltages are not near zero. (Note that the D/As are not selectable; they are always sampled.)

Viewing Test Parameters

Before and after test execution, you may view the state of a test by highlighting the test in the Test list box on the STEX main dialog and selecting “**View**” from the Test pulldown menu. This will open the View Test dialog.

View Test

Name: Elcentro Uncompensated
 Test Type: Time History
 Created: 17-Jun-1997:00:39:42
 Modified: 17-Jun-1997:00:44:03
 Duration: 66.555 seconds
 Storage: 2142 (SAAAABXS.STX) 2.99 MB

☐ Drive: Elcentro Uncompensated [DRIVE]
☐ Achieved: Elcentro Uncompensated [ACHIEVED]
☐ Specimen: Elcentro Uncompensated [SPECIMEN]

DOF	Control Mode	Full Scale	Feedback Mode	Full Scale
Longitudinal	Acceleration	10.0 g	Acceleration	10.0 g
Lateral	Acceleration	10.0 g	Acceleration	10.0 g
Vertical	Acceleration	10.0 g	Acceleration	10.0 g

Channel	Full Scale	Identifier
18(18)	-110.6,110.6 g	Cond0902
19(19)	-101.9,101.9 g	Cond1001
20(20)	-42.4,42.4 g	Cond1002
21(21)	-43.6,43.6 g	Cond1101
22(22)	-105.0,105.0 g	Cond1102
23(23)	-44.5,44.5 g	Cond1201
24(24)	-81.6,81.6 g	Cond1202

Report **View Test Log** **Exit**

This dialog contains the following information:

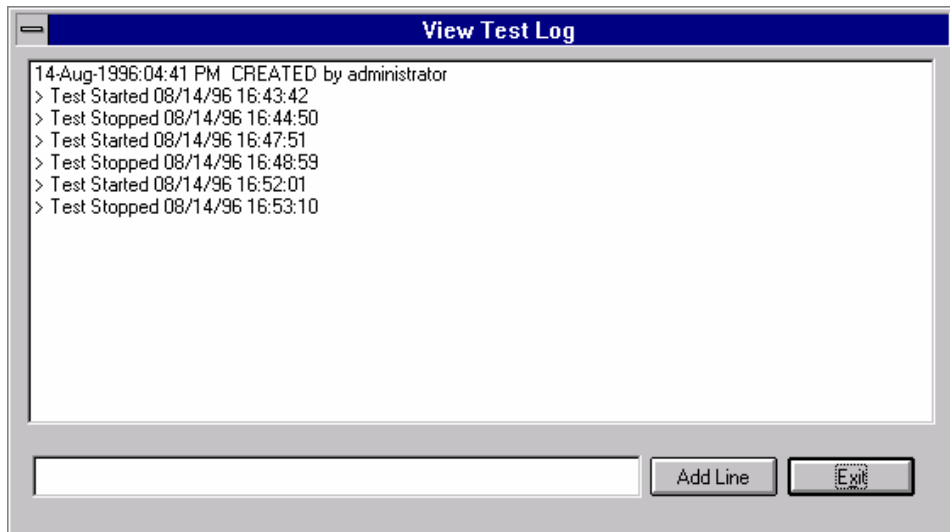
- Test name
- Test type
- Test creation and modification dates
- Permanent storage information
- Test duration (in seconds)
- Model name (for compensated tests)
- Target motion vector name
- Desired motion vector name
- Drive motion vector name
- Specimen motion vector name (if any)
- Achieved motion vector name
- A list detailing the DOFs controlled by the test, along with mode and full-scale information
- A list of specimen acquisition channels along with identifier and full-scale information

Selecting a button next to one of the vector names (target, achieved, drive, or specimen) will open an appropriate statistics dialog for that vector. Refer to [“Viewing Result Parameters”](#) on page 28.

You can view time histories associated with the test by locating the time history (or vector) in the Result list box on the STEX main dialog and double clicking on it. This will open a statistic dialog. Refer to [“Viewing Result Parameters”](#) on page 28.

Selecting the **“Report”** button will generate a test report in a text file. The filename is specified using the Save As dialog. The test report describes the test, including channel information as well as the test log.

Selecting the “**View Test Log**” button will open a View Test Log dialog, allowing you to scroll through the test log.



You may add one or more comment lines to the log by entering text in the edit field and selecting the “**Add Line**” button.

Deleting a Test

To delete a test from the currently active test group, highlight the test in the Test list box and select “**Delete**” from the Test pulldown menu on the main dialog. You will be prompted for verification of the desire to delete the selected test. If so instructed, the test and all associated time histories will be deleted from the test group.

Analysis Results

Overview

Results are elements created during testing activities and are contained by a test group. Results have a name and a type. A result is either a simple function (usually of time or frequency) or a collection of simple functions or a model. Results are categorized as user or system, and are either created using the Analysis pulldown menu (user) or by STEX (system). Results are listed in the Results list box on the STEX main dialog.

When you double click on a result in the result list box, an appropriate statistic dialog will open, allowing you to view and plot result parameters. Each result type provides a uniquely formatted statistic dialog presenting information derived from the result content. Refer to [“Viewing Result Parameters”](#) on page 28.

User vs. System Results

Results are categorized as System or User indicating the result's origin. Each result has a unique name.

User results (e.g. a time history, spectrum, model) are created by the user using options on the Analysis pulldown menu. User result names cannot include the brackets [and].

System results are created by STEX (i.e. time histories of a test including vectors, or elements of a model). You cannot delete or edit system results. When the system creates a result, it appends a special description to the beginning, enclosed in brackets:

Model Measurement Test [RANDOM:Lat]

The button “Hide (Show) System Results” located on the STEX main dialog will remove (or include) the system results from the list box, depending on the current state. This can be helpful when there is a large number of system results.

Result Types

Results include functions of time (time histories) and functions of frequency (spectra) but also include vectors, matrices, and models. Analysis results for a test group are arranged according to type and listed in the Results list box on the STEX main dialog.

The types of results are:

- Time Histories (including Vectors)
- Spectra
- Response Spectra
- Models (including matrices)

Time Histories and Vectors

A **time history** consists of a sequence of values defined along the time axis in equal increments (the group sample interval). The time history has units, duration, and point count.

A **vector** can only be created by the system and is generally a set of time histories associated with a list of channels (either drive or acquisition). For example, when a test is defined with a list of time histories (one for each degree of freedom controlled during the test), a vector is created by STEX. The vector *refers* to individual time histories defined in the test group; these time histories are listed separately in the result list box on the main dialog. The vector concept provides a meaningful construct for organizing the multitude of time histories created during test activities. The vector has a unique name and can be plotted.

Spectra

A **spectrum** consists of sequence of values defined along the frequency axis in equal increments (the frequency resolution). The values may be real or complex. A real valued spectrum is sometimes referred to as an ASD. A complex valued spectrum is sometimes referred to as a CSD and has both amplitude and phase information for each frequency included in the spectrum.

The span of the spectrum is from zero to the Nyquist frequency associated with the test group. The spectrum has units, resolution, and line count. Spectra may differ in resolution but not in span. While all spectra are bounded by the group Nyquist value, resolution is based on Nyquist *and* line count. When a spectrum is created, the user has the option of setting the Points Per Frame parameter (PPF) used whenever Fourier methods are used. Typically a set of possible PPF values are presented to the user; for example, frame sizes of 512, 1024, 2048, 4096, and 8192 are common. The *count* of the spectrum is exactly half the PPF and is known as Lines Per Frame (LPF). The resolution of the spectrum is the Nyquist frequency divided by LPF. Increasing the PPF parameter provides a finer resolution in the frequency domain.

Response Spectra

A **response spectrum** consists of a series of real values defined logarithmically along the frequency axis. The span of the spectrum must begin at a non-zero frequency. The highest frequency represented in the response spectrum must not exceed 80% of the test group Nyquist frequency. Values of the response spectra represent the motion characteristics of a specimen's response to some arbitrary excitation assuming a given damping. Response spectra may be computed from a time history and, conversely, a time history may be created based on a desired response spectra. Response spectra have units associated to describe the kind of data represented.

Matrices and Models

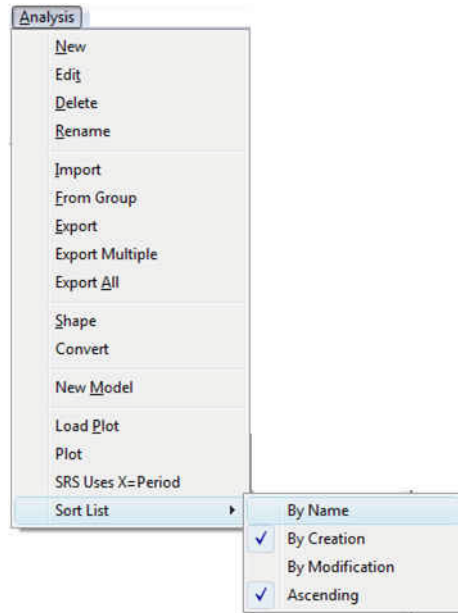
A result **matrix** is a set of similar results (time histories or spectra) that are arranged in rows and columns that correspond to channels of test control or acquisition. Most commonly the matrix is square. Further, matrices are created exclusively during system model calculations.

A **model** result consists of several matrices. The primary element of the system model is the transfer function. The transfer function is a matrix of complex results representing an approximation to the linear, dynamic characteristics of the excitation equipment. The model element called the expanded inverse is used during a compensated time history test to improve system fidelity.

The model is derived from the results of a model measurement test. The drive and achieved signals of the test are processed to derive the transfer function. The transfer function is *inverted* to create the inverse transfer function. Finally, the inverse transfer function is *expanded*. Prior to expansion, the user may edit the inverse transfer function to control aspects of the compensation process.

Analysis Menu Options

Results are created, edited, and deleted using the “Analysis” pulldown menu on the STEX main dialog.

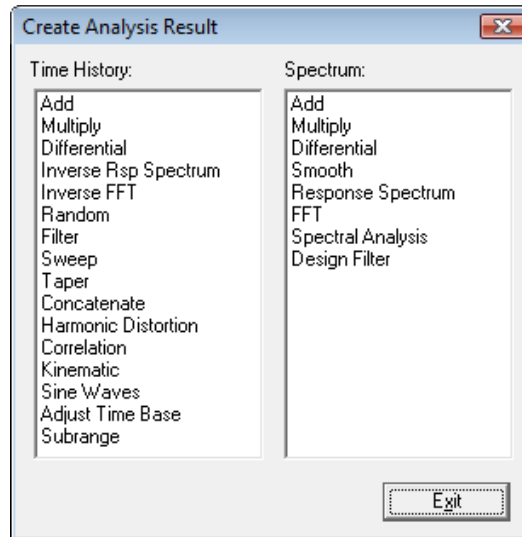


The “Analysis” pulldown menu contains the following options:

- New Create new time histories and spectra
- Edit Edit a result
- Delete Delete a result
- Rename Rename a result
- Import Import a [simple] result from a text file
- From Group Import a [simple] result from another test group
- Export Export a [simple] result to a text file
- Export Multiple Export one or more results to an external file.
- Export All Export all [simple] results to individual text files
- Shape Create a shape function - time history or spectra
- Convert Convert an acceleration signal to a displacement estimate
- New Model Create a model from a test
- Load Plot Loads a previously saved plot setup for display
- Plot Plot a result
- SRS Uses X=Period The response spectra are displayed with Period as the X axis rather than frequency
- Sort List Result list sorting options

Creating a New Result

When you select “New” on the Analysis pulldown menu on the STEX main dialog, the **Create Analysis Result** dialog appears.



This dialog is divided into two columns representing the operations that can be performed to create time histories and spectra, respectively. Once created, the result will appear in the Result list located on the main dialog. Press the “**Exit**” button to return to the main dialog.

Creating Time Histories

The create analysis result dialog allows the user to create time histories using the following operations:

- Add Add (and scale) existing time histories
- Multiply Multiply existing time histories
- Differential Differentiate and integrate an existing time history
- Inverse Rsp Spectrum Synthesis of inverse shock response
- Inverse FFT Create the inverse FFT of an existing complex spectrum
- Random Create a random signal with specified level and frequency content
- Filter Filter an existing time history
- Sweep Create a sine sweep (single or varying frequency - linear, logarithmic, octave)
- Taper Taper an existing time history
- Concatenate Concatenate an existing time histories
- Harmonic Distortion Create a harmonic Distortion of an existing time history
- Correlation Correlation coefficient between 2 time histories
- Kinematic Kinematic (i.e. units) conversion of an existing time history. Accel Conversion corrects previously recorded acceleration signals
- Sine Waves Create multiple sine wave results
- Adjust Time Base Adjust the duration and frequency content of a signal
- Subrange Select a portion of an existing time history result

Once created, the new time history will appear in the Results list box on the STEX main dialog. The following subsections further describe each of these options.

Addition and Scaling of Time Histories

When you select “**Add**” on the time history column of the Create Analysis Result dialog, the **Add Time History** dialog appears.

The screenshot shows the "Add Time Histories" dialog box. It features a title bar with the text "Add Time Histories". Below the title bar, there is a "Result:" label followed by a text box containing "Time History ABC + DEF - GHI". Below this is a "Units:" label followed by an empty text box. The main area contains four rows, each representing a time history to be added. Each row has a label "Time History #1:" through "Time History #4:", a text box for the history name, a "Select" button, and a "Factors:" text box. The factors are 1, 1, -1, and 0 respectively. Below these rows is an "Offset:" label followed by a text box containing "0" and a "Remove Offset" checkbox. At the bottom of the dialog are "OK" and "Cancel" buttons.

This dialog allows you to add together up to four existing time histories with scale factors and offset parameters. The duration of the resultant is equal to the longest time history referenced.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Enter the result units. The units of the first time history will be used if no units are specified.
3. Select up to four time histories. The names of the time histories may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48.

All but the first source time history field may be blank.

4. Enter a scale factor for each time history, if desired.

5. If desired, input an offset value to be added to the result. Alternatively, you may elect to have the mean subtracted from the result by clicking the “**Remove Offset**” button; this disables the Offset edit field.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Multiplying Time Histories

When you select “**Multiply**” on the time history column of the Create Analysis Result dialog, the **Multiply Time History** dialog appears.

The screenshot shows the "Multiply Time Histories" dialog box. It features a title bar with the text "Multiply Time Histories" and a close button (X). The main area contains the following elements:

- Result:** A text box containing "LOG X / Y".
- Units:** An empty text box.
- Time History #1:** A text box containing "X", followed by a "Select" button and a dropdown menu set to "Log".
- Time History #2:** A text box containing "Y", followed by a "Select" button and a dropdown menu set to "Inverted".
- Time History #3:** An empty text box, followed by a "Select" button.
- Time History #4:** An empty text box, followed by a "Select" button.
- Factor:** A text box containing "1".
- Offset:** A text box containing "0", followed by an unchecked checkbox labeled "Remove Offset".
- Buttons:** "OK" and "Cancel" buttons at the bottom.

This dialog allows you to multiply together up to four existing time histories with scale factors and offset parameters. The duration of the resultant is equal to the shortest time history referenced.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Enter the result units. The units of the first time history will be used if no units are specified.

3. Select up to four time histories. The names of the time histories may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to “[Using the Select Option](#)” on page 48. All but the first source time history field may be blank.
4. Select either **Log**, **Exp** or **Normal** to be applied to Time History #1. If neither is selected, each point in the selected time history is used in the multiplication. If **Log** is selected, the multiplication is performed using the log of each point. If **Exp** is selected, the exponent of each point in the selected time history is computed and used in the multiplication. Note that this applies only to the Time History #1.
5. Select either **Inverted**, or **Normal** to be applied to Time History #2. If **Inverted** is selected, the reciprocal of each time history #2 point value is computed prior to applying it as a factor to the result.
6. Enter a scale factor for each time history, if desired.
7. If desired, input an offset value to be added to the result. Alternatively, you may elect to have the mean subtracted from the result by clicking the “**Remove Offset**” button; this disables the Offset edit field.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Time History Differential Operations

When you select “**Differential**” on the time history column of the Create Analysis Result dialog, the **Differential Time History** dialog appears.

The screenshot shows a dialog box titled "Time History Differential". It contains the following elements:

- Result:** A text field containing "Derivative of ABC".
- Source:** A text field containing "Time History ABC" with a "Select" button to its right.
- Units:** An empty text field.
- Scale:** A text field containing "1".
- Use Integration:** An unchecked checkbox.
- High Pass Filter:** An unchecked checkbox.
- OK** and **Cancel** buttons at the bottom.

This dialog allows you to differentiate or integrate a time history with scaling and filtering options.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select a source time history. The name of the source time history may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to “[Using the Select Option](#)” on page 48.
3. Enter the result units. The units of the source time history will be used if no units are specified.
4. Enter a scale factor for the result time history, if desired.
5. Select the “**Use Integration**” box if integration is desired. If this box is not selected, the derivative will be computed.
6. Select the “**High Pass Filter**” box if you desire the result of the differential operation to be high pass filtered, with a cutoff frequency of 0.7 Hz, nominal.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Inverse Shock Time History Synthesis

When you select “**Inverse Shock**” on the time history column of the Create Analysis Result dialog, the **Inverse Shock Response** dialog appears.

The screenshot shows the "Inverse Shock Response" dialog box. It has a title bar with the text "Inverse Shock Response". Inside the dialog, there are several input fields and buttons. The "Result:" label is followed by a text box containing "New Earthquake". The "Spectrum:" label is followed by a text box containing "Required Shock Response" and a "Select" button to its right. The "Shape:" label is followed by a text box containing "Time History Shape 30 sec" and a "Select" button to its right. Below these, the "Damping:" label is followed by a text box containing "0.05" and the text "0 to 1" to its right. The "Result Units:" label is followed by a text box containing "g". The "Lines Per Octave:" label is followed by a text box containing "6" and the text "1 to 100" to its right. At the bottom of the dialog are two buttons: "OK" and "Cancel".

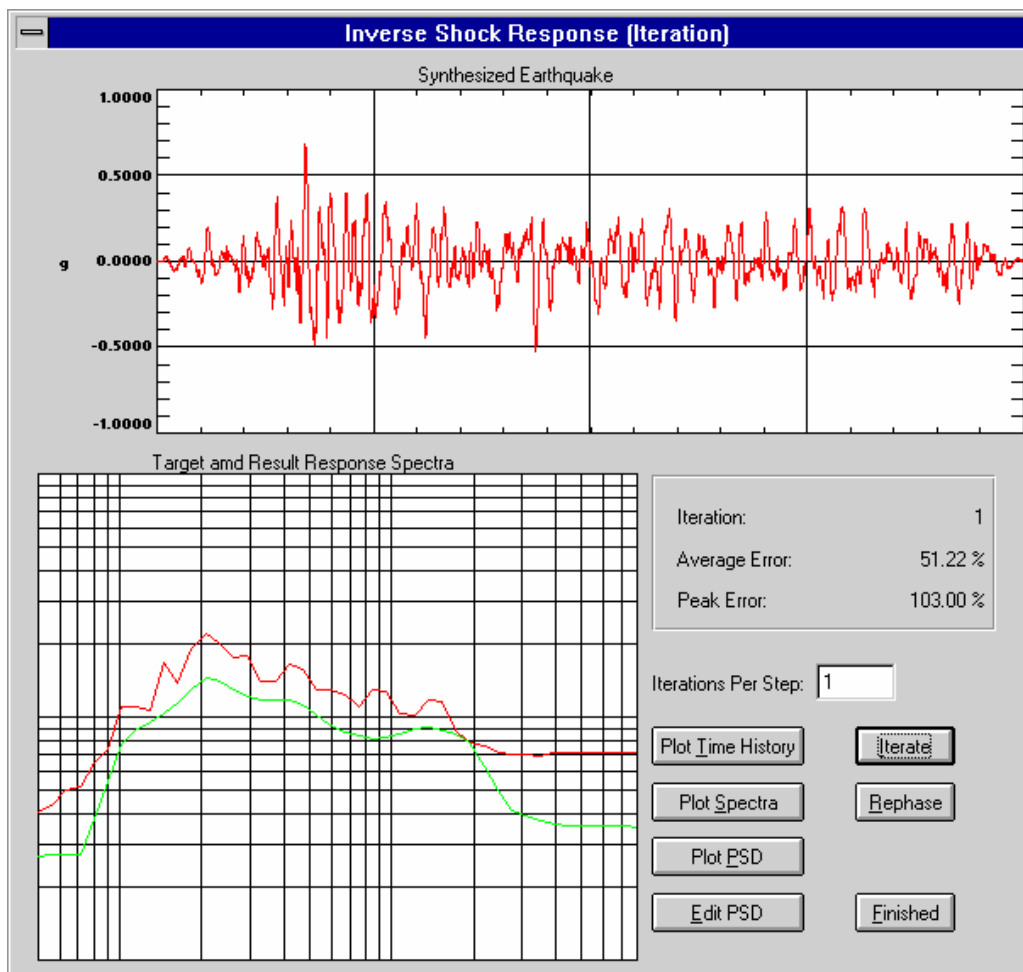
This dialog allows you to create an artificial shock profile from a desired shock response spectrum and a shaping function.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select a desired shock response spectrum. The name of the spectrum may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48.
3. Select the time history shape. The name of the shape may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48.
4. Enter the desired damping ratio. This value must be between 0 and 1.
5. Enter the result units. Must be acceleration units. Refer to the Appendix E for units that STEX will recognize.

- Input the lines per octave. This value must be between 1 and 100. Lines per octave indicates the density of the analysis. The frequency range is determined from the desired shock spectrum.

Pressing the “OK” button causes the first iteration of time history synthesis to be performed. When this iteration is completed, the **Inverse Shock Response (Iteration)** dialog will appear.



The iteration dialog displays the current earthquake time history, an overlay plot of the desired and achieved shock response, the number of iterations completed, average error, peak error, and the number of iterations to be performed per step.

Selecting the “**Iterate**” button causes an iteration step to be performed; for each iteration, the time history is adjusted to bring the result shock response closer to the desired.

The “**Iterations Per Step**” field reflects the number of iterations done consecutively per step. After iteration, the plots are updated and the error statistics are recomputed. The previous error statistics are encoded in parenthesis along with the new values. For example 23.45% (65.35) indicates that the error of 65.35% was superseded with 23.45%.

Select “**Plot Time History**” to plot the time history. (This option provides a more functional plotting environment than that of the iteration dialog.)

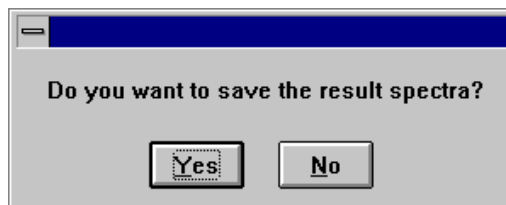
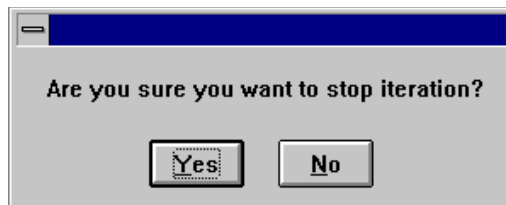
Select “**Plot Spectra**” to plot the achieved shock response spectra.

Select “**Plot PSD**” to plot the current PSD function used by the synthesis algorithm to iterate.

Select “**Edit PSD**” to edit the values of the PSD function to be edited. Refer to “[Editing Existing User Results](#)” on page 144.

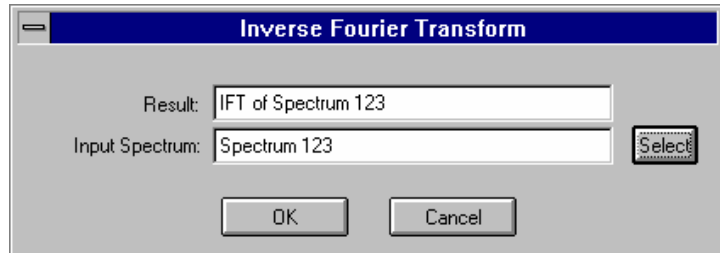
Select “**Rephase**” to randomize the phase function; this phase function is combined with the PSD by the iteration algorithm. Randomization may be useful for situations of non-convergence.

Select “**Finished**” to terminate the iteration process and store the result time history. You will be prompted to verify that iteration should cease. Then, you will be asked if the result shock spectrum should be saved.



Inverse FFT of Spectrum

When you select “**Inverse FFT**” on the time history column of the Create Analysis Result dialog, the **Inverse Fourier Transform** dialog appears.



This dialog allows you to create an inverse fast fourier transform.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select an input spectrum. The name of the input spectrum may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Random Time History

When you select “**Random**” on the time history column of the Create Analysis Result dialog, the **Generate Random Time History** dialog appears.

This dialog allows you to create a random time history.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select a spectral content shaping function. The name of the shaping function may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48.

Frame parameters are updated whenever the Select button is used to select an ASD shaping function; if a shaping function is typed in the name field, the **Update** button may be selected to update the frame parameters. Decay time is one half of a frame length.

3. Enter the result units.
4. Enter the 3-Sigma value. This value represents the amplitude maximum for 99.7% of the time history peak values.

- Specify duration in seconds or by number of frames. If you choose seconds, the duration will be adjusted to encompass an even number of frames. The frame count refers to the number of Independent/Repeat frame pairs. Refer to Appendix B for information on random signals.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Filtering a Time History

When you select “**Filter**” on the time history column of the Create Analysis Result dialog, the **Apply Filter** dialog appears.

This dialog allows you to apply a digital filter to an existing time history.

Enter the following parameters on this dialog:

- Enter the desired result name.
- Select a source time history. The name of the source time history may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48.

3. Select a filter (a real or complex function of frequency). The name of the filter may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to “[Using the Select Option](#)” on page 48.

If “Use Filter Profile” is not selected, select the filter settings desired rather than selecting a pre-existing filter profile.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Creating a Sine Sweep

When you select “**Sweep**” on the time history column of the Create Analysis Result dialog, the **Generate Sweep** dialog appears.

The screenshot shows the "Generate Sweep" dialog box. It contains the following fields and controls:

- Title Bar:** Generate Sweep
- Result:** Sine Sweep 1 to 30 Hz 30 seconds 1 G
- Initial Frequency:** 1 Hz
- Amplitude:** 1
- Final Frequency:** 30 Hz
- Units:** g
- Duration:** 30 seconds
- Sweep Type:** Linear (with a dropdown arrow)
- Phase Offset:** 0 degrees
- Frequency Rate:** 1 Octaves/min
- Buttons:** OK, Cancel

This dialog allows you to create sine sweep functions. Tapering may be performed using the tapering analysis function defined in the following section.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Enter the initial frequency and the final frequency. A single frequency “sweep” may be created by setting the initial and final frequencies to the same value (for linear and logarithmic sweeps only).

3. Enter the amplitude and units.

4. Select the sweep type:

Linear

Logarithmic

Octave

5. Enter the phase offset (defined from 0 to 360 degrees). This value determines the starting point of the sweep.

6. Enter the duration parameter:

For linear and logarithmic sweeps, input the duration in seconds. For octave sweeps, the duration is computed using the inputted frequency rate along with the initial and final frequencies.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Tapering Time Histories

When you select “**Taper**” on the time history column of the Create Analysis Result dialog, the **Taper Time History** dialog appears.

This dialog allows you to add tapering to a time history.

Enter the following parameters on this dialog:

1. Enter the desired result name. Note that if a result name is not specified, the source will automatically be tapered.

2. Select a source time history. The name of the source time history may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to “[Using the Select Option](#)” on page 48.
3. Select tapering at the start and/or end of time history.
4. Enter the taper duration. The duration of either taper is specified in seconds or as a percentage of the entire time history. The text on the button beside the duration edit field describes how the value shall be interpreted. Selecting the button toggles the text between “Seconds” and “Percent”.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Concatenation of Time Histories

When you select “**Concatenate**” on the time history column of the Create Analysis Result dialog, the **Concatenate Time History** dialog appears.

The screenshot shows the "Concatenate Time Histories" dialog box. It features a title bar with the text "Concatenate Time Histories". Inside the dialog, there is a "Result:" label followed by a text box containing "Concatenation of ABC and DEF and GHI". Below this is a "Units:" label followed by an empty text box. The main area contains four rows, each labeled "Time History #1:" through "Time History #4:". Each row has a text box for the time history name (containing "Time History ABC", "Time History DEF", "Time History GHI", and an empty box respectively) and a "Select" button to its right. To the right of these rows is a "Taper:" label, followed by a text box containing "0" and the word "seconds". At the bottom of the dialog are two buttons: "OK" and "Cancel".

This dialog allows you to link together up to four existing time histories with tapering.

Enter the following parameters on this dialog:

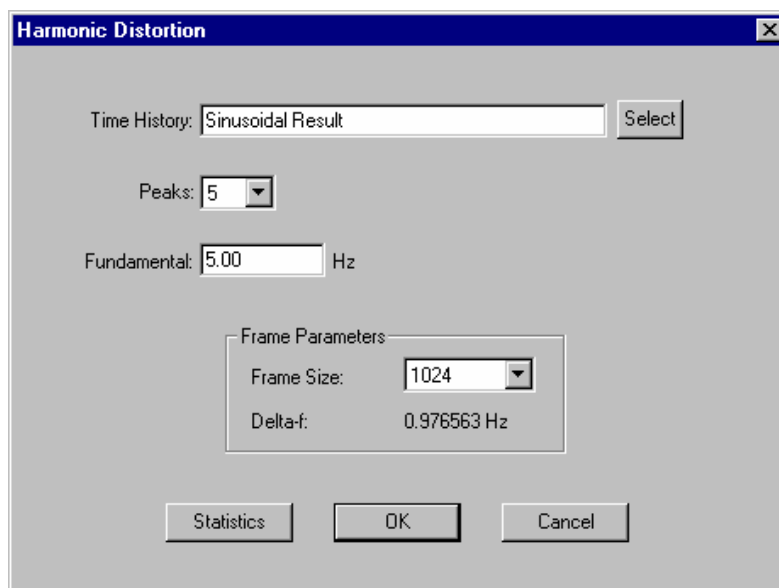
1. Enter the desired result name.
2. Enter the result units. The units of the first time history will be used if no units are specified.

3. Select up to four time histories. The names of the time histories may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48. All but the first source time history field may be blank.
4. Enter a tapering duration to be applied over the junction of the joined time histories. The tapering is divided equally between the two time histories.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Harmonic Distortion

When you select “**Harmonic Distortion**” on the time history column of the Create Analysis Result dialog, the **Harmonic Distortion** dialog appears.



The screenshot shows the "Harmonic Distortion" dialog box. It has a title bar with the text "Harmonic Distortion" and a close button (X). The dialog contains the following fields and controls:

- Time History:** A text field containing "Sinusoidal Result" and a "Select" button to its right.
- Peaks:** A dropdown menu showing the value "5".
- Fundamental:** A text field containing "5.00" followed by "Hz".
- Frame Parameters:** A sub-dialog box containing:
 - Frame Size:** A dropdown menu showing the value "1024".
 - Delta-f:** A text field showing the value "0.976563 Hz".
- Buttons:** "Statistics", "OK", and "Cancel" buttons at the bottom.

This dialog allows you to compute the harmonic distortion of a signal.

Enter the following parameters on this dialog:

1. Select an existing time history.
2. Select the number of harmonic peaks to use for calculation.
3. Select the fundamental frequency.

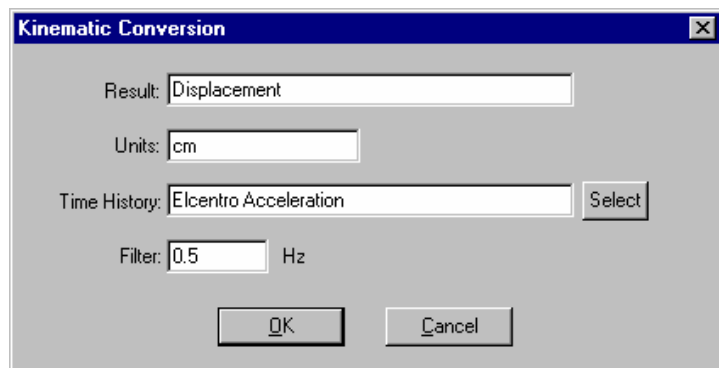
4. Adjust the frame size, if necessary.

Pressing “**Statistics**” will result in a statistics dialog for the result selected. Pressing the “**OK**” button will result in the distortion calculation as shown below.



Kinematic Conversion

When you select “**Kinematic**” on the time history column of the Create Analysis Result dialog, the **Kinematic Conversion** dialog appears.



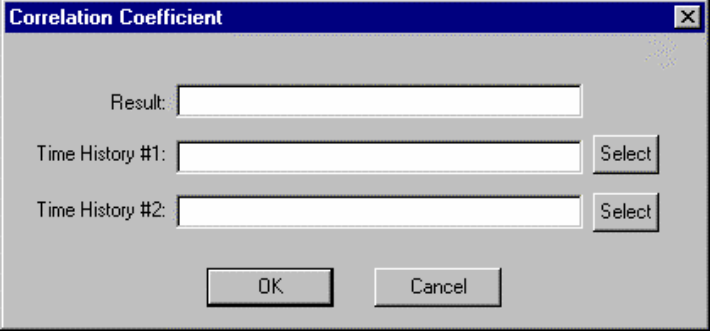
This dialog allows you to create a new time history result based on the conversion of units for an existing time history. Both units must be one of acceleration, velocity, or displacement and the units must be recognized by STEX. Refer to Appendix E for a list of available units.

You may select the cut-off frequency for a high-pass filter that is applied before conversion.

Pressing the “**OK**” button will create the time history and return to the Create Analysis dialog.

Correlation

When you select “**Correlation**” on the time history column of the Create Analysis Result dialog, the **Correlation Coefficient** dialog appears.

The image shows a dialog box titled "Correlation Coefficient" with a standard Windows-style title bar (blue with a close button). The dialog has a light gray background. It contains three input fields: "Result:" followed by a text box, "Time History #1:" followed by a text box and a "Select" button, and "Time History #2:" followed by a text box and a "Select" button. At the bottom, there are two buttons: "OK" and "Cancel".

This dialog allows you to specify two time histories and compute the correlation coefficient between them.

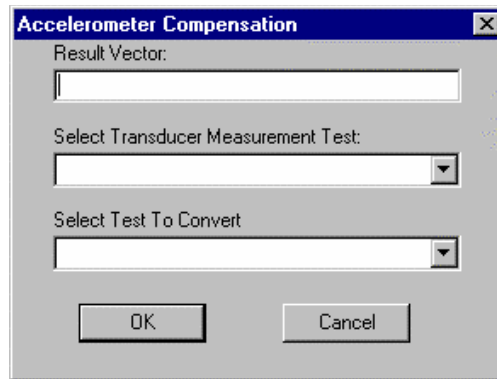
Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Enter Time History #1 or use the “**Select**” button to choose from the available time histories in the group.
3. Enter Time History #2 or use the “**Select**” button to choose from the available time histories in the group.

Pressing the “**OK**” button will create the new time history and return to the Create Analysis dialog.

Accel Conversion

When you select “**Accel Conversion**” on the time history column of the Create Analysis Result dialog, the **Accelerometer Compensation** dialog appears.



This dialog allows you to correct previously recorded acceleration signals by compensating for the dynamic response of the transducers. By creating and executing a special model measurement test with a single DOF of drive and one or more accelerometer specimen channels recorded, STEX can compute a transfer function for each channel. Each transfer function is then applied to the appropriate time history in the Conversion test, creating a new vector of corrected signals.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select the appropriate model measurement test from the list.
3. Select the appropriate conversion target test from the list.

Pressing the “**OK**” button will create the new vector. The appropriate acceleration signals in the target test achieved vector are recomputed based on the transfer function. Other signals (i.e. displacements) are simply copied to the new vector.

Sine Waves

When you select “**Sine waves**” on the time history column of the Create Analysis Result dialog, the **Sine Waves** dialog appears.

Enter the following parameters on this dialog:

1. Enter the name template used to automatically generate result names for all sine waves created
2. Set parameters defining the sine wave and click the **Add** button to add the definition to the list.
3. Repeat step 2 as needed
4. You may define a range of sine waves using the **Add Range** button. A sine wave will be created using the **Frequency** parameter; the increment will be added to this frequency and a new definition is created if that frequency does not exceed the **End Frequency**. This repeats until the **End Frequency** is reached.
5. Click the Create button to generate the sine waves.

Generate Sine Waves

Name Template:

Frequency	Amplitude	Duration	Init. Taper	End Ta...	Init. Zero	End Zero	Total Dur	Phase
1 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
1 Hz	2 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
2 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
3 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
3 Hz	2 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
4 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
5 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
5 Hz	2 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
6 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
7 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
7 Hz	2 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
8 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
9 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
9 Hz	2 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
10 Hz	1 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	
11 Hz	2 g	10 Sec	1 Sec	1 Sec	1 Sec	1 Sec	12 Sec	

Duration:

Initial Taper:

End Taper:

Initial Zero:

End Zero:

Amplitude:

Frequency:

Units:

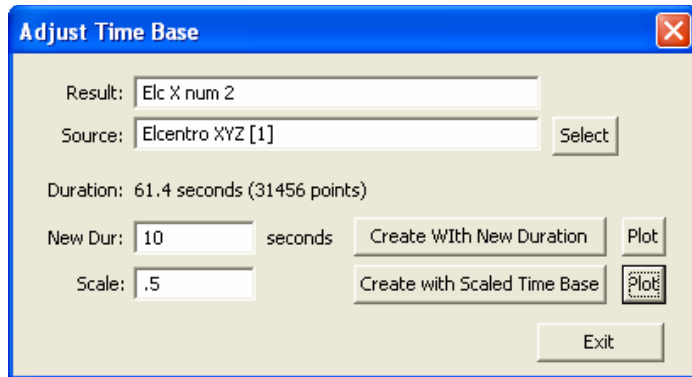
Phase:

End Frequency:

Frequency Increment:

Adjust Time Base

When you select “**Adjust time base**” on the time history column of the Create Analysis Result dialog, the **Adjust Time Base** dialog appears.



Use the Adjust Time Base dialog to change the duration and frequency content of a signal as is necessary for scaled test specimens. For example, a ½ scale structure may require that the frequency content of El Centro be reduced from 0-20 hz to 0-10 hz.

This panel allows an existing signal to the time base to be adjusted by either of the following methods:

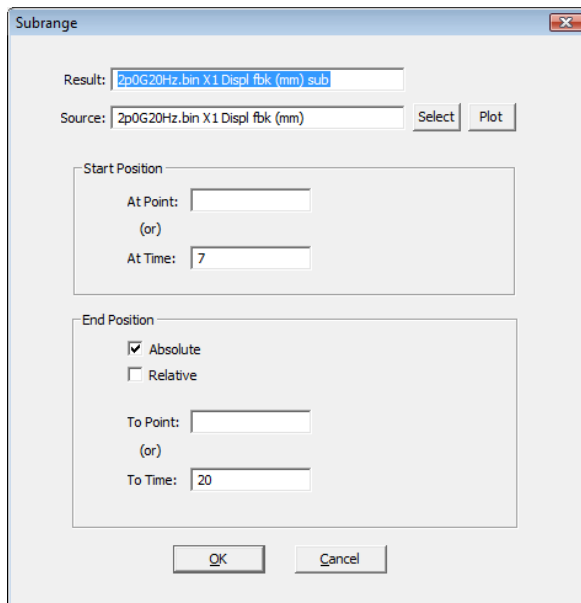
- Select and apply a new duration explicitly.
- Select and apply a scale factor to the duration of the existing selection.

Plot buttons allow you to see the result before you create the result.

Using the Adjust Time Base dialog example shown above, clicking the **Create With New Duration** (with an entered **New Dur** value of 10 seconds) results in the Elcentro signal being reduced from its initial value to 10 (a reduction of $10/IV$, where IV is the initial value). Clicking the **Create with Scaled Time Base** button (with an entered value of 0.5) results in the initial value reduction by a scale of 0.5 (or $1/2$). For example, if the initial value was 61 seconds, the value is reduced to 30.5 seconds.

Create Subrange

When you select “**Subrange**” from the time history column of the Create Analysis Result dialog, the **Subrange** dialog appears.



The Subrange dialog box is used to define a new time history as a sub-range of an existing one. It features a title bar with a close button. The main area contains several input fields and buttons. At the top, there is a 'Result' field with the text '2p0G20Hz.bin X1 Displ fbk (mm) sub' and a 'Source' field with the text '2p0G20Hz.bin X1 Displ fbk (mm)'. To the right of the 'Source' field are 'Select' and 'Plot' buttons. Below these are two sections: 'Start Position' and 'End Position'. The 'Start Position' section has 'At Point:' and 'At Time:' fields, with 'At Time' containing the value '7'. The 'End Position' section has a radio button for 'Absolute' (which is selected) and a radio button for 'Relative'. Below these are 'To Point:' and 'To Time:' fields, with 'To Time' containing the value '20'. At the bottom of the dialog are 'OK' and 'Cancel' buttons.

Subrange

Result: 2p0G20Hz.bin X1 Displ fbk (mm) sub

Source: 2p0G20Hz.bin X1 Displ fbk (mm) Select Plot

Start Position

At Point:

(or)

At Time: 7

End Position

☒ Absolute

☐ Relative

To Point:

(or)

To Time: 20

OK Cancel

This dialog allows you to create a new time history as a sub range of another (existing) time history. The start and end points are defined independently as either a time value or as a point count. The end position may be defined with an absolute value, or as a position relative to the start.

Creating Spectra

The create analysis result dialog allows the user to create spectra using the following operations:

- Addition (and scaling) of existing spectra
- Multiplication of existing spectra
- Differentiation and integration of an existing spectra
- Smoothing (Hanning) an existing spectra
- Shock response calculation
- FFT of an existing time history
- Spectral Analysis - ASD, CSD, Transfer Function, Coherence
- Digital filter amplitude profile (Butterworth or frequency elimination)

Once created, the new spectrum will appear in the Results list box on the STEX main dialog. The following subsections further describe each of these options.

Addition and Scaling of Existing Spectra

When you select “**Add**” on the spectrum column of the Create Analysis Result dialog, the **Add Spectra** dialog appears.

The **Add Spectra** dialog box is shown with the following fields and values:

- Result:** Spectrum ABC + DEF
- Units:** (empty)
- Spectra #1:** Spectrum ABC (with a **Select** button)
- Spectra #2:** Spectrum DEF (with a **Select** button)
- Spectra #3:** (empty) (with a **Select** button)
- Spectra #4:** (empty) (with a **Select** button)
- Factors:** 1, 1, -1, 0 (corresponding to Spectra #1 through #4)
- Offset:** 0
- Buttons:** OK, Cancel

This dialog allows you to add together up to four existing spectra with scale factors and an offset parameter.

In the case where the resolutions of the specified spectra do not match (due to disparate PPF values), the result is defined with the finest resolution present. The add operation does not entail any interpolation upon spectra with the more coarse resolution; simple value repetition is employed.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Enter the result units. The units of the first spectrum will be used if no units are specified.
3. Select up to four spectra. The names of the spectra may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48. All but the first spectrum field may be blank.
4. Enter a scale factor for each spectrum, if desired.

5. Input a real-valued offset value to be added to the result, if desired.

Pressing the “**OK**” button will create the new spectrum and return to the Create Analysis dialog.

Multiplying Spectra

When you select “**Multiply**” on the spectrum column of the Create Analysis Result dialog, the **Multiply Spectra** dialog appears.

This dialog allows you to multiply together up to four existing spectra with a real factor and offset parameter.

In the case where the resolutions of the specified spectra do not match (due to disparate PPF values), the result is defined with the finest resolution present. The multiplication operation does not entail any interpolation upon spectra with the more coarse resolution; simple value repetition is employed.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Enter the result units. The units of the first spectrum will be used if no units are specified.

3. Select up to four spectra. The names of the spectra may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to “[Using the Select Option](#)” on page 48. All but the first spectrum field may be blank. The first spectra may be inverted (i.e. the 1/x is used for each value of the spectra).
4. Enter a real factor and offset value to be added to the result, if desired.

Pressing the “**OK**” button will create the new spectrum and return to the Create Analysis dialog.

Spectra Differential Operations

When you select “**Differential**” on the spectrum column of the Create Analysis Result dialog, the **Spectrum Differential** dialog appears.

This dialog allows you to differentiate or integrate a spectrum with a scaling option.

Enter the following parameters on this dialog:

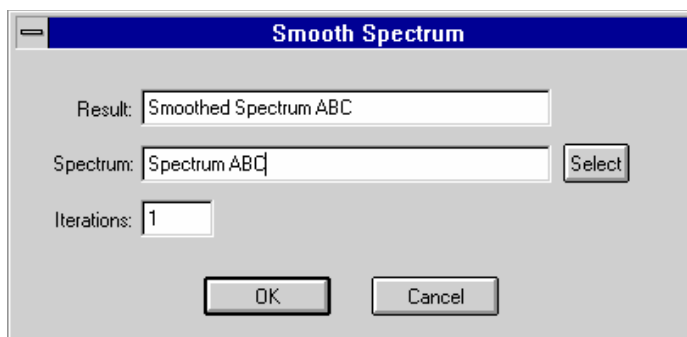
1. Enter the desired result name.
2. Select a source spectrum. The source spectrum may be real- or complex-valued. The name of the source spectrum may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to “[Using the Select Option](#)” on page 48.
3. Enter the result units. The units of the source spectrum will be used if no units are specified.
4. Enter a scale factor for the source spectra, if desired.

5. Select the “**Use Integration**” box if integration is desired. If this box is not selected, the derivative will be computed.

Pressing the “**OK**” button will create the new spectrum and return to the Create Analysis dialog.

Smoothing Spectra

When you select “**Smooth**” on the spectrum column of the Create Analysis Result dialog, the **Smooth Spectrum** dialog appears.



This dialog allows you to perform a smoothing operation (Hanning) on a spectrum.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select a source spectrum. The name of the source spectrum may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48.
3. Enter the iteration count. This is the number of times the smoothing operation will be performed on the source spectrum.

Pressing the “**OK**” button will create the new spectrum and return to the Create Analysis dialog.

Shock Response Calculation

When you select “**Shock Response**” on the spectrum column of the Create Analysis Result dialog, the **Shock Response** dialog appears.

Shock Response

Result: Shock Response of a Time History

Source: Acceleration Time History

Initial Frequency: 0.05 Hz 0.0500 .. 80.000

Final Frequency: 80 Hz 0.0500 .. 80.000

Lines Per Octave: 20 1 to 100

Damping: .05 0 to 1

ResultType: Absolute Acceleration ▼

Result Units: g

Plot Ref: Reference SRS to be Plotted with Result

This dialog allows you to calculate a shock response of a source time history.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select a source time history. The name of the source time history may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to [“Using the Select Option”](#) on page 48.
3. Enter the initial and final frequency values.
4. Enter the lines per octave. This value defines the “coarseness” of analysis over the frequency range.
5. Enter the damping ratio.

6. Select one of the following result types:

- Absolute Acceleration
- Equivalent Static Acceleration
- Pseudo Velocity
- Relative Displacement

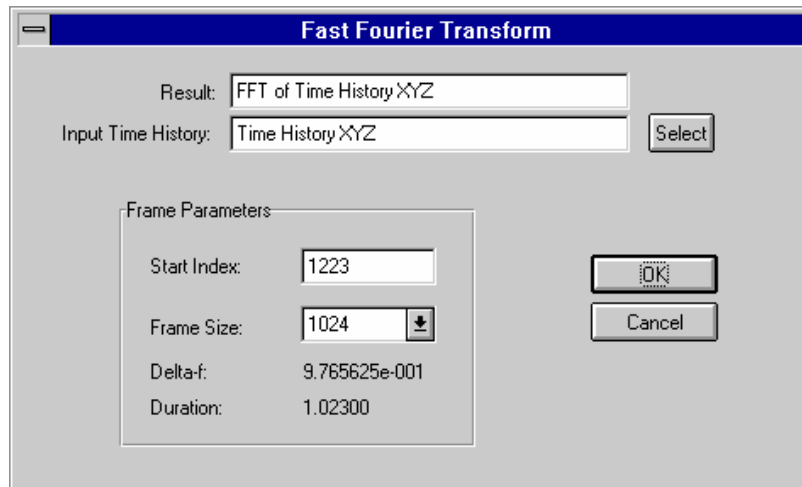
7. Enter the result units. The specified result units must match the result type. (Refer to Appendix E for a list of units strings recognized by STEX).

Pressing the “OK” button will create the new shock response and return to the Create Analysis dialog.

If a Plot Ref Shock response is selected, a plot dialog will appear after the new Shock Response is created, both results will be plotted, allowing you to compare the two results.

FFT of an Existing Time History

When you select “FFT” on the spectrum column of the Create Analysis Result dialog, the **Fast Fourier Transform** dialog appears.



The image shows a software dialog box titled "Fast Fourier Transform". It has a blue header bar with the title. Below the header, there are two text input fields: "Result:" containing "FFT of Time History XYZ" and "Input Time History:" containing "Time History XYZ". To the right of the "Input Time History:" field is a "Select" button. Below these fields is a section titled "Frame Parameters" which contains four rows of input fields: "Start Index:" with "1223", "Frame Size:" with "1024" and a dropdown arrow, "Delta-f:" with "9.765625e-001", and "Duration:" with "1.02300". To the right of the "Frame Parameters" section are two buttons: "OK" and "Cancel".

This dialog allows you to create a fast fourier transform of an existing time history.

Enter the following parameters on this dialog:

1. Enter the desired result name.

2. Select an input time history. The name of the time history may be entered manually or selected using the Result Selection dialog by pressing the “**Select**” button beside the name field. Refer to “[Using the Select Option](#)” on page 48.
3. Enter the frame parameters: start index and frame size.

The start index is the number of values from the start of the time history to skip over; the FFT is performed on the number of values specified in the frame size beginning at the start index.

The dialog displays the frequency resolution and frame duration based on the frame size selection.

Pressing the “**OK**” button will create the new FFT and return to the Create Analysis dialog.

Spectral Analysis of Existing Time Histories

When you select “**Spectral Analysis**” on the spectrum column of the Create Analysis Result dialog, the **Spectral Analysis** dialog appears.

The screenshot shows the 'Spectral Analysis' dialog box. It features a title bar with the text 'Spectral Analysis'. Inside the dialog, there are several input fields and buttons. The 'Result:' field contains the text 'CSD of Time History ABC & DEF'. Below it, the 'Input Time History:' field contains 'Time History ABC' and has a 'Select' button to its right. The 'Output Time History:' field contains 'Time History DEF' and also has a 'Select' button to its right. A section titled 'Frame Parameters' is located below the input fields, containing 'Frame Size:' with a value of '1024' and a dropdown arrow, and 'Delta-f:' with a value of '0.244141 Hz'. To the right of this section is a 'Result Type:' dropdown menu currently set to 'CSD'. At the bottom of the dialog are two buttons: 'OK' and 'Cancel'.

This dialog allows you to perform a spectral analysis on one or two time histories (input and output, respectively).

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select an input time history. The name of the time history may be entered manually or selected using the Result Selection dialog by pressing the **“Select”** button beside the name field. Refer to [“Using the Select Option”](#) on page 48.
3. Select an output time history, if needed. The name of the time history may be entered manually or selected using the Result Selection dialog by pressing the **“Select”** button beside the name field. Refer to [“Using the Select Option”](#) on page 48.
4. Enter the frame size.

As you change the frame size, the frequency resolution is updated.

5. Select one of the following result types:
 - ASD (auto spectral density of a single time history)
 - CSD (cross spectral density of two time histories)
 - Transfer Function (of two time histories)
 - Coherence (of two time histories)

Pressing the **“OK”** button will create the new FFT and return to the Create Analysis dialog.

Designing a Digital Filter

When you select “**Design Filter**” on the spectrum column of the Create Analysis Result dialog, the **Create Filter** dialog appears.

Create Filter

Result: Low Pass at 30 Hz Butterworth Order 5

Filter Shape: Low Pass

Resolution: 0.195313

Low Cutoff: 30 Hz

High Cutoff: 100 Hz

Filter Type

☒ Butterworth

☐ Frequency Elimination

Butterworth Order: 5

OK Cancel

This dialog allows you to create a digital Butterworth or Frequency Elimination filter.

Enter the following parameters on this dialog:

1. Enter the desired result name.
2. Select one of the following filter shapes:
 - Low Pass
 - High Pass
 - Notch
 - Bandpass
3. Select the spectral resolution value.

4. Enter the low and high cutoff frequencies. The high cutoff frequency field is disabled when the filter shape is low-pass. The low cutoff frequency field is disabled when the filter shape is high-pass. Both fields are enabled when the filter shape is notch or bandpass.
5. Select the filter type; Butterworth or Frequency Elimination. If you select Butterworth, you must also select the order of the filter.

Pressing the “**OK**” button will create the new filter and return to the Create Analysis dialog.

Editing Existing User Results

You can edit an existing user result by highlighting the element in the result list (by left-clicking the mouse over the element name) and selecting “Edit” from the Analysis pull-down menu on the main dialog. The following result types may be edited:

- Time Histories
- Real and Complex Spectra
- Shock Response Spectra
- Models

Note *Vectors and matrices may not be edited. A model can only be edited by making changes to the inverse transfer function (and therefore the resulting expanded inverse).*

Editing Time History and Real Spectrum

When you highlight a time history or real valued spectrum in the result list box and select “**Edit**” from the Analysis pulldown menu on the main dialog, the time history-editing dialog appears.

Sine Sweep 1 to 30 Hz in 10 sec 1 g

Index	seconds	g
000000	0.000000e+000	0.000000e+000
000001	2.000000e-003	1.260248e-002
000002	4.000000e-003	2.527582e-002
000003	6.000000e-003	3.801793e-002
000004	8.000000e-003	5.082665e-002
000005	1.000000e-002	6.369976e-002
000006	1.200000e-002	7.663494e-002
000007	1.400000e-002	8.962983e-002
000008	1.600000e-002	1.026820e-001
000009	1.800000e-002	1.157889e-001
000010	2.000000e-002	1.289479e-001
000011	2.200000e-002	1.421564e-001
000012	2.400000e-002	1.554117e-001
000013	2.600000e-002	1.687108e-001

Update

Channel Name:

Units:

OK Cancel

A scrolling list is presented with three columns, one line per point in the time history. The three columns of the value list are index, independent axis value at that index, and result value at that index.

A Channel edit field allows you to modify the channel name; this name usually denotes the control DOF associated with the signal and is displayed on the left of the result plot. Note that the name of the result being edited is displayed in the caption of the editing dialog.

You may scroll through the list of values and select a line of the list box using the mouse. The value in the update edit box reflects the value of the result at the selected point. You may modify the value in the edit field, and select “**Update**” to modify the result. The next line is automatically highlighted for editing.

You may change the units defined for the result. Note that changing the units merely changes the units text field. No units conversion is performed on the actual result values.

Pressing “**OK**” will save the changes and return to the Main dialog. Pressing “**Cancel**” will cancel the operation.

Editing Complex Spectrum

When you highlight a complex spectrum in the result list box and select “**Edit**” from the Analysis pulldown menu on the main dialog, the complex spectrum-editing dialog appears.

Index	Hz	(g / g)	degrees
000000	0.000000e+000	6.261811e-003	149.3582
000001	9.765625e-001	6.261811e-003	149.3582
000002	1.953125e+000	2.638127e-002	-93.4188
000003	2.929688e+000	1.126541e-003	69.0026
000004	3.906250e+000	5.132474e-003	-70.4298
000005	4.882813e+000	6.294564e-003	106.5347
000006	5.859375e+000	8.418976e-003	-162.7084
000007	6.835938e+000	8.117245e-003	27.5121
000008	7.812500e+000	1.232262e-003	22.6186
000009	8.789063e+000	6.673987e-003	-146.2429
000010	9.765625e+000	1.162098e-002	97.4732
000011	1.074219e+001	1.176599e-002	76.3444
000012	1.171875e+001	6.898082e-003	-161.2236
000013	1.269531e+001	8.697809e-003	99.9695

Update: 0.001232261944562 22.6185869580873 OK Cancel

See Real/Imaginary

The name of the result being edited is displayed in the caption of the editing dialog.

A scrolling list is presented with four columns, one line per value in the spectrum. The four columns of the value list are index, independent axis value at that index, and result values, amplitude and phase, at that index.

You may scroll through the list of values and select a line of the list box using the mouse. The values in the update edit boxes reflect the complex value of the result at the selected frequency. You may modify the value in the edit fields, and select **“Update”** to modify the result. The next line is automatically highlighted for editing.

By selecting the format button in the lower left corner of the dialog, you may toggle the format of the complex value display in the list box. Initially, the list box is formatted with amplitude and phase for each frequency; the format button reads **“See Real/Imaginary”**. Selecting the button in this state causes the list box to be reformatted with real and imaginary components of the complex function values; the button text is changed to **“See Amplitude/Phase”**. Note that the *effect* of the Update button matches the current format of the list box.

Pressing **“OK”** will save the changes and return to the Main dialog. Pressing **“Cancel”** will cancel the operation.

Editing a Shock Response

When you highlight a shock response spectrum in the result list box and select **“Edit”** from the Analysis pulldown menu on the main dialog, the shock response spectrum-editing dialog appears.

The dialog box titled "Required Shock Response" contains a table with three columns: Index, Hz, and g. The first row (Index 000000) is highlighted. Below the table, there is an "Update" button, a text field containing "0.461702", an "OK" button, a "Cancel" button, and a "Channel Name:" label followed by an empty text field.

Index	Hz	g
000000	5.000000e-001	4.617020e-001
000001	5.360160e-001	5.128957e-001
000002	5.746264e-001	5.681499e-001
000003	6.160179e-001	6.274075e-001
000004	6.603910e-001	6.905270e-001
000005	7.079603e-001	7.572506e-001
000006	7.589561e-001	8.272180e-001
000007	8.136253e-001	8.999289e-001
000008	8.722324e-001	9.747529e-001
000009	9.350611e-001	1.050928e+000
000010	1.002416e+000	1.127559e+000
000011	1.074622e+000	1.203646e+000
000012	1.152029e+000	1.324471e+000
000013	1.235012e+000	1.597417e+000

A scrolling list is presented with three columns, one line per value in the spectrum. The three columns of the value list are index, independent axis value at that index, and result value at that index.

You may scroll through the list of values and select a line of the list box using the mouse. The values in the update edit boxes reflect the value of the result at the selected frequency. You may modify the value in the edit field and select **“Update”** to modify the result. The next line is automatically highlighted for editing.

Pressing **“OK”** will save the changes and return to the Main dialog. Pressing **“Cancel”** will cancel the operation.

Editing a Model

When you highlight a model in the result list box and select “**Edit**” from the Analysis pulldown menu on the main dialog, the inverse transfer function-editing dialog will appear. Refer to “[Editing the Inverse Transfer Function](#)” on page 163. The original (unmodified) inverse transfer function is presented for editing.

If an expanded inverse exists, a confirmation message box will appear with a warning that continuing will destroy the expanded inverse currently stored in the group. You may continue with the model editing by pressing the “**Continue**” button or cancel the operation.

After the inverse has been edited and you select “**Exit**”, the new expanded inverse is created, and the main dialog appears.

Deleting Results

When you highlight a result in the result list and select “**Delete**” from the Analysis pulldown menu on the main dialog, you will be prompted for verification of the desire to delete the selected result. If so instructed, the result is deleted from the test group.

Note that only user simple results may be deleted; vectors and matrices may not be deleted. Models may be deleted provided no compensated tests are referencing the model; when a model is deleted, all matrices and functions associated with the model are deleted as well.

Time histories and vector associated with a test are deleted whenever the test is deleted.

Data Import and Export

Simple STEX results may be stored externally to the group database in text files (one result per file). The external file is unknown to STEX and may be deleted or otherwise manipulated. Refer to Appendix D for file formats.

Select “**Import**” from the Analysis pulldown menu on the main dialog to read a text file containing a result definition. Refer to “[Data Import](#)” on page 152.

Select “**From Group**” from the Analysis pulldown menu on the main dialog to copy a result that is stored in another group. Refer to “[Copying Results From Another Group](#)” on page 154.

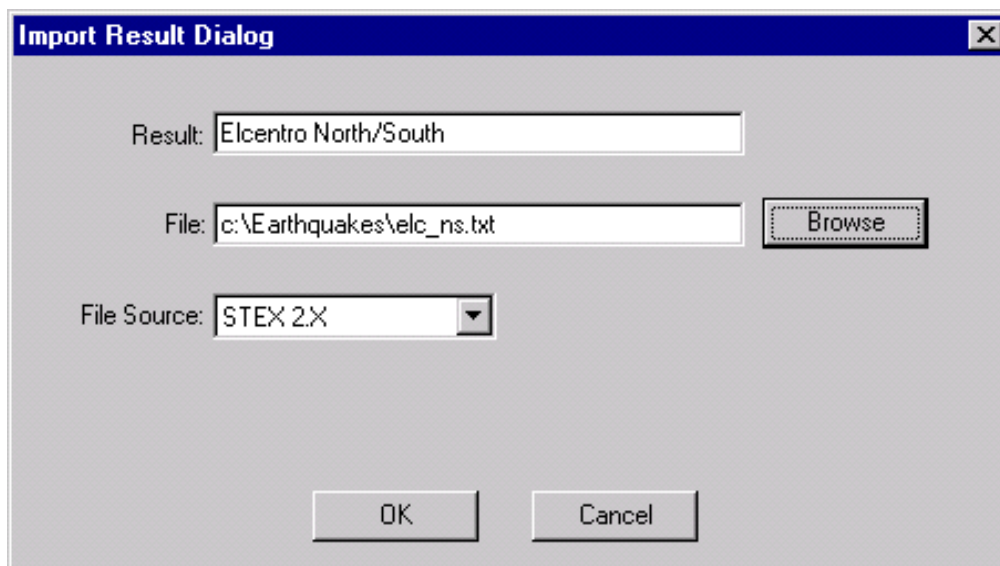
Select “**Export Multiple**” from the Analysis pulldown menu on the main dialog to select tests and/or results for export to external files. Refer to “[Exporting Multiple Results](#)” on page 157.

Select “**Export**” from the Analysis pulldown menu on the main dialog to write the currently highlighted result in the current test group to an external text file. (Only time histories, real and complex spectra, and shock response spectra may be written to an external file; matrices, vectors, and models may not be manipulated with import/export except through their individual components). Refer to “[Data Export](#)” on page 155.

Select “**Export All**” from the Analysis pulldown menu on the main dialog to write all the results in the current test group to external text files. (Only time histories, real and complex spectra, and shock response spectra may be written to an external file; matrices, vectors, and models may not be manipulated with import/export except through their individual components). Refer to “[Data Export](#)” on page 155.

Data Import

When you select “**Import**” from the Analysis pulldown menu on the main dialog, the file import dialog appears.

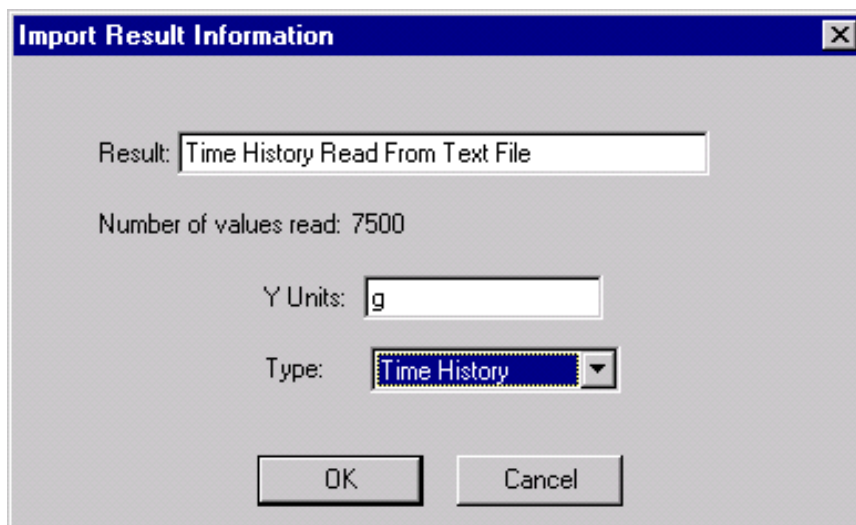


You may manipulate the filename and directory to select a file containing a text instance of a result. (Refer to Appendix D for file formats recognized by STEX). You may define the name of the result to be displayed in the result list on the main dialog; if not defined, the complete filename is used as the result name.

Selecting “**Browse**” results in an Open dialog.

The result to be imported is given a name. A file source is also required describing the file format. Refer to the Appendix D for file formats.

If the file source is ASCII No Header, you will be prompted for more information with the following dialog.



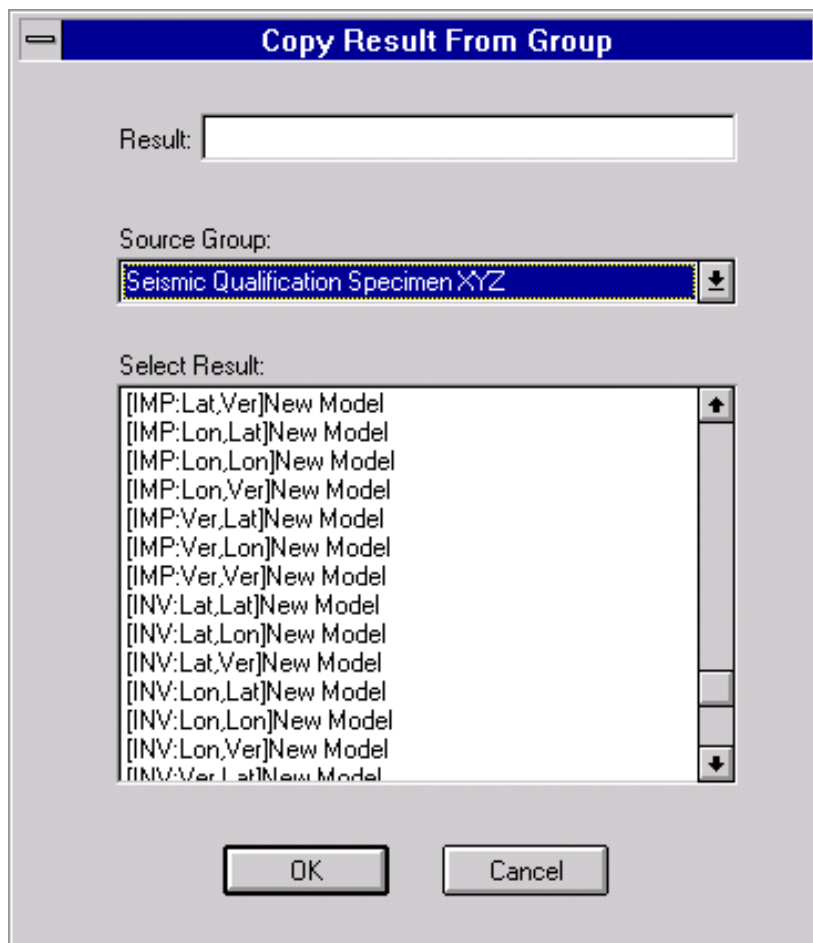
The dialog box has a title bar with the text "Import Result Information" and a close button (X). The main area contains the following fields and controls:

- Result:** A text box containing "Time History Read From Text File".
- Number of values read:** A label followed by the value "7500".
- Y Units:** A text box containing "g".
- Type:** A dropdown menu with "Time History" selected.
- Buttons:** "OK" and "Cancel" buttons at the bottom.

The result name may be changed. You must specify the result type and the Y Units. If the imported data does not meet the requirements for the selected result type, a message box appears.

Copying Results From Another Group

When you select “**From Group**” from the Analysis pulldown menu on the main dialog, the **Copy Results From Group** dialog appears.



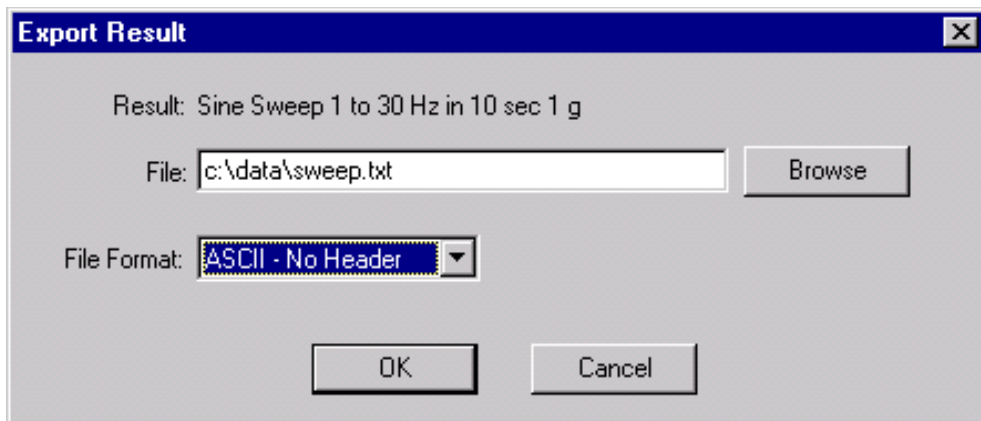
You may select the result name, the source group, and the result to copy. When the group selection is changed, the result list is updated. Only time histories and spectra (real, complex, and shock response) results may be copied. If the result name field is left blank, the original result name is used.

If the spectrum has a Nyquist frequency that disagrees with the current group, you are notified and asked whether or not to continue; if you choose to continue, the spectrum is copied but the Nyquist frequency is modified to agree with the current group.

If the time history has a sample interval that disagrees with the current group, you are notified. You may cancel the file copy or choose to have the signal interpolated; a linear interpolation scheme is employed. Note that this type of interpolation can result in signal distortion, especially if the target group has a lower Nyquist frequency.

Data Export

When you highlight a result in the result list and select “**Export**” from the Analysis pulldown menu on the main dialog, the **Export Result** dialog appears.

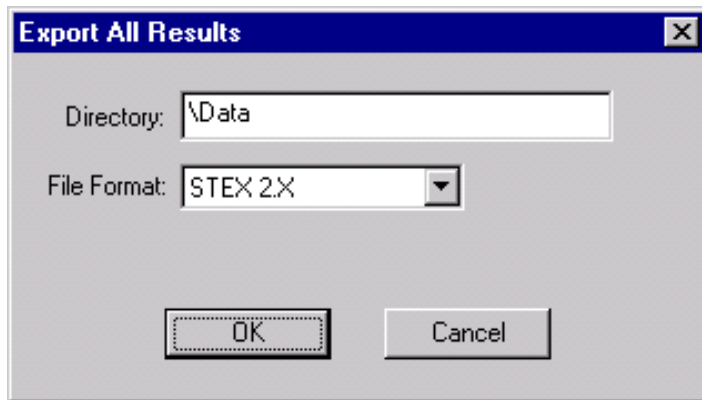


This dialog allows you to specify a directory and filename wherein the contents of the result is written. Selecting “Browse” results in an Open dialog.

A target file location is provided. The file format must also be specified. Refer to the Appendix D for file formats.

Exporting All Results

When you select “**Export All**” from the Analysis pulldown menu on the main dialog, the **Export All Results** dialog appears.



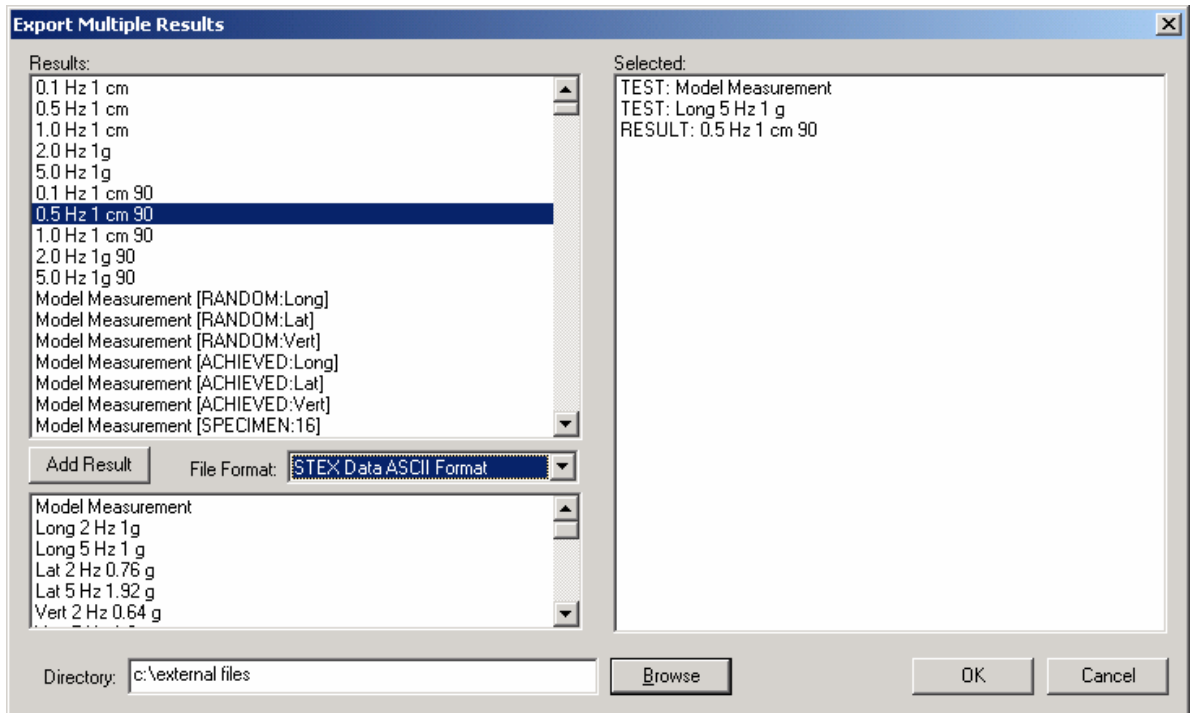
This dialog allows you to specify a directory into which all simple results will be written to external files.

One external file is created for *each* time history, real spectrum, complex spectrum, and shock response spectrum in the current test group.

A directory must be provided as well as the file format. Refer to Appendix D for file formats.

Exporting Multiple Results

When you select “**Export Multiple**” from the Analysis pulldown menu on the main dialog, the **Export Multiple Results** dialog appears.



You may select one or more results and select Add Result to have the result(s) added to the Selected list on the right.

You may double-click any tests listed in the bottom left list, to have that test added to the Selected list.

When all the necessary results and tests are listed, select a file format. Then enter a path to the directory to receive the exported files.

To perform the export, select OK.

Creating Shape Results

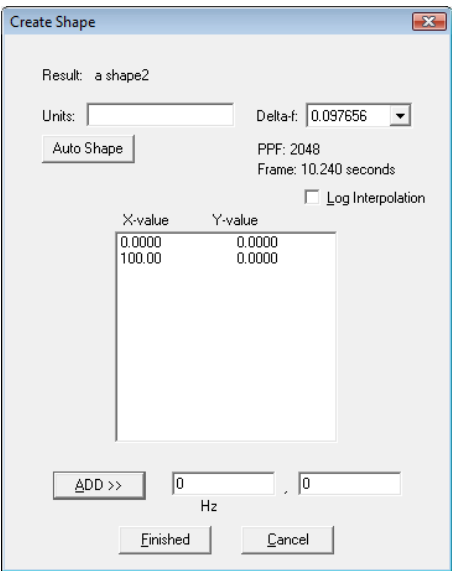
When you select “**Shape**” on the Analysis pulldown menu on the main dialog, the initial **Create Shape** dialog appears.



Use this dialog to enter the Result name and to select the data Type. The data Type options are:

- **Time History** used with inverse shock response
- **Spectrum (Real)** used when creating random signals
- **Response Spectrum** used with inverse shock response

When you have entered the result name and selected the data Type, click **OK** to continue or **Cancel** to exit (without saving). When you click **OK**, the full Create Shape dialog opens. Click **Cancel**, to return to the main STEX dialog.



The full Create Shape dialog allows you to create a shaping function that can be used to create random signals, inverse shock responses, and when you run model measurement tests. Use the Create Shape dialog to specify the result units and frame size (for ASD shapes only). In addition, the Create Shape dialog allows you to enter individual point values into the X-Y list or to click the **Autoshape** button and use the Autoshape dialog to create the result based on the shape type (rather than entering X-Y pairs). Note that Autoshape button is only available for Real Spectrum results.

To enter parameters using the Create shape dialog:

Result: a shape2

Units: Delta-f: 0.097656

Auto Shape

PPF: 2048
Frame: 10.240 seconds

☐ Log Interpolation

X-value	Y-value
0.0000	0.0000
100.00	0.0000

ADD >> 0 0

Hz

Finished Cancel

1. Enter the desired result units.
2. Select the desired frequency resolution.

- Enter values into the X-Y list by using the text boxes and **Add>>** button. Input the desired X value into the first text box and the corresponding Y value into the second box. Press the **ADD>>** button to insert this X-Y pair into the list.

Note For ASD shapes, the X-values may be from 0 to Nyquist only; these values are inserted automatically into the list when presented.

Log interpolation is provided for ASD shapes only. Y-values must be non-zero for log interpolation.

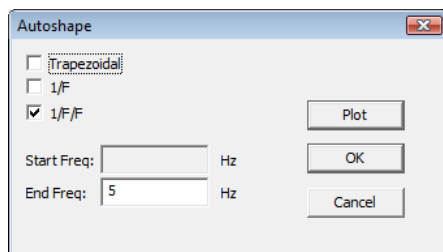
To change the Y-value for an X-value already included in the list, you may simply add the value in the normal fashion: the Y-value will be updated.

Also, double-clicking an element of the list will delete the pair.

Once the proper values are listed, select “**OK**” and the new result will be created by *interpolating* the values of the list.

To enter parameters using the Autoshape dialog:

- Use the Create Shape dialog to enter the desired result units.
- Use the Create Shape dialog to select the desired frequency resolution.
- On the Create Shape dialog, click the **Autoshape** button to open the Autoshape dialog.



- On the Autoshape dialog, click the check box that represents the shape that you want to create (**Trapezoid**, **1/F**, or **1/F/F**).

5. As appropriate, enter values for the desired **Start Freq** and **End Freq**.

For example, if **Trapezoid** is the selected shape, two values are required. For purposes of this example, assume those values are 1 and 100. When you click the **OK** button, a trapezoid is created with constant amplitude from 1 to 100 and trending to/from zero on the left/right (in this case, from 0.1 to 1, and 100.0 to Nyquist). Note that the 1/F and 1/F/F shapes only require a single frequency because the shape will be unity from 0 to the specified frequency, then fall off at 1/F or 1/F/F until Nyquist.

Note *To view a graphic representation of your selected and entered settings, click the **Plot** button. When you do, a temporary file is created so that you can view the effects of your selections and settings. To create a permanent file with your specified shape and values, use the **OK** button. However, clicking the **OK** button does not create a plot for you to view.*

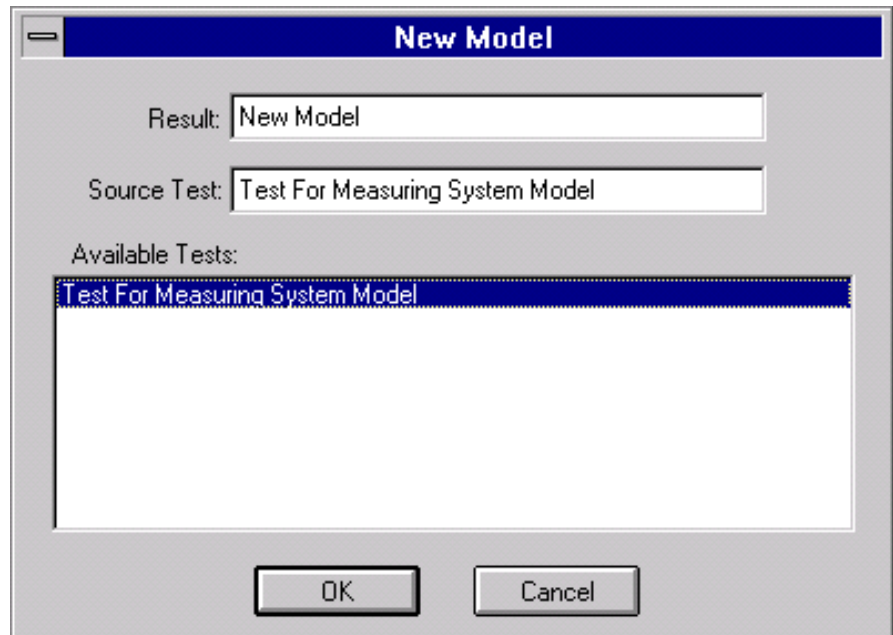
6. Click **OK** to create the result. Click **Cancel** to close the Autoshape dialog and return to the Create Shape dialog.

Creating a Model

A model is derived from the results of a model measurement test and is used during a compensated time history test.

The drive and achieved signals of the model measurement test are processed to derive the transfer function. Intermediate results of this calculation are saved initially but may be deleted later in the model processing. The transfer function is *inverted* to create the inverse transfer function. Finally, the inverse transfer function is *expanded*. Prior to expansion, you may edit the inverse transfer function to control aspects of the compensation process.

When you select “**New Model**” on the Analysis pulldown menu on the main dialog, the **New Model** dialog appears.

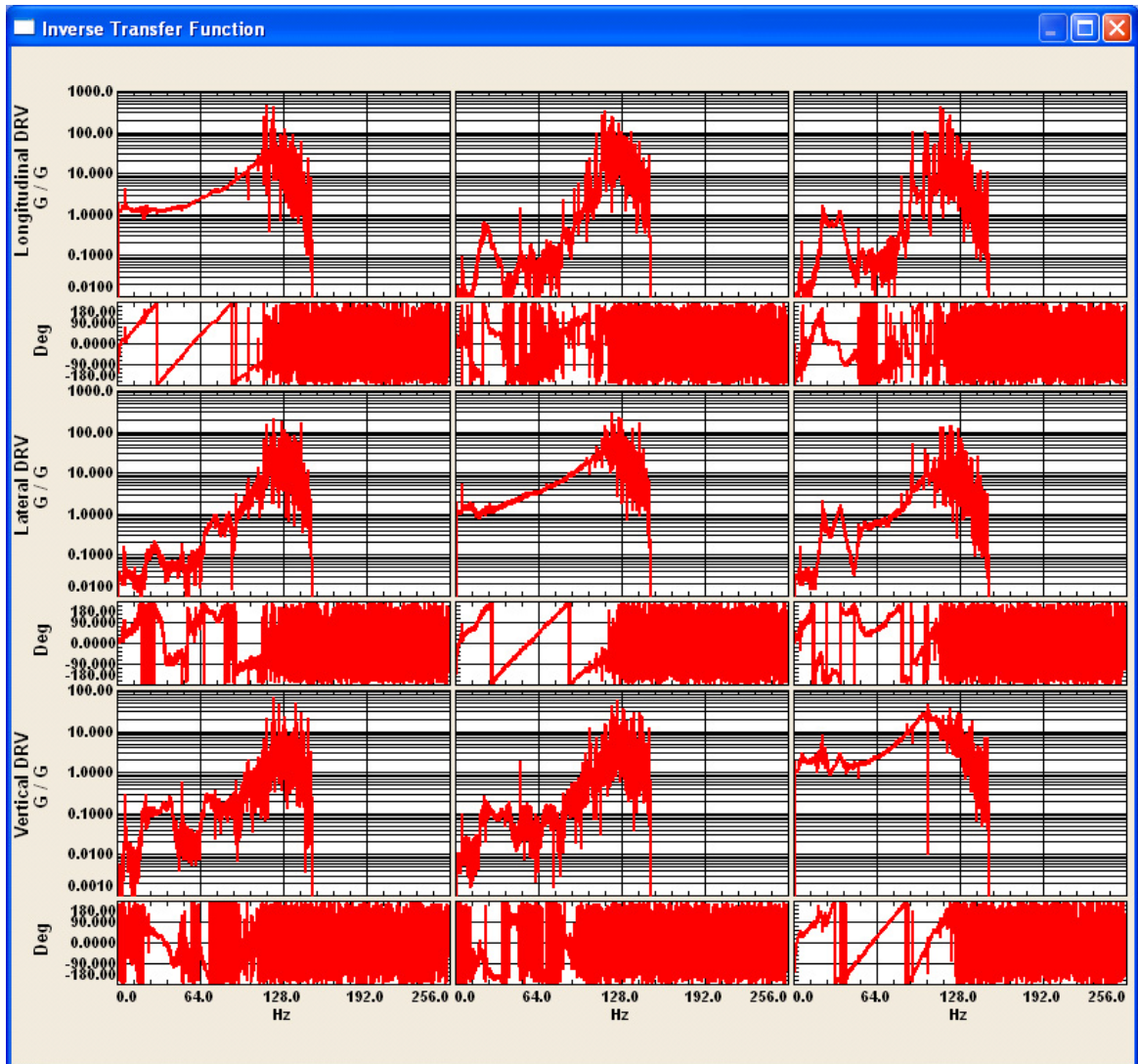


This dialog allows you to specify a model result name and a model measurement source test. You may use the mouse to select the source test from a list of model measurement tests.

When you press the “**OK**” button, the transfer function and inverse transfer function are computed. Then the **Edit Model** dialog will appear, allowing you to edit the inverse transfer function before it is expanded.

Editing the Inverse Transfer Function

The **Inverse Transfer Function** dialog will appear when you press “OK” on the New Model dialog.

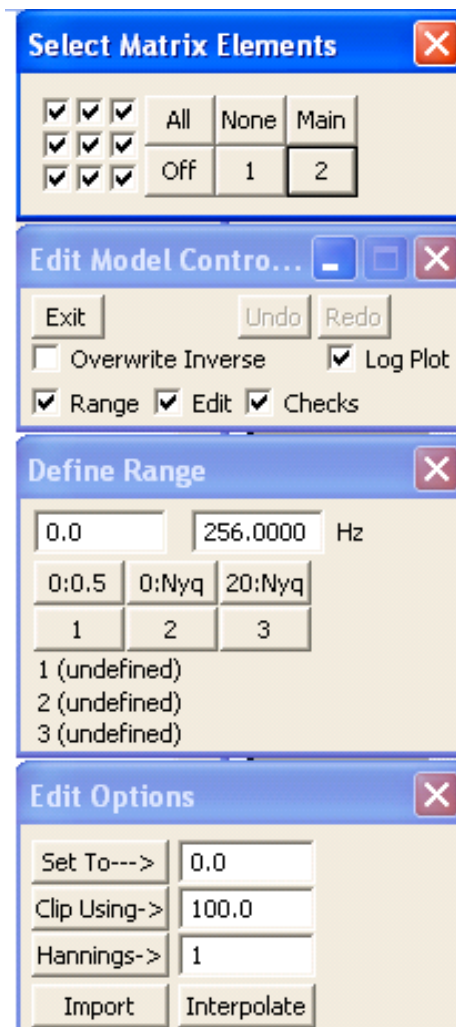


The inverse is displayed in this dialog.

Left clicking on any element of the inverse transfer function graphical display will bring up a plot dialog of only that element. That element may be analyzed and then dismissed to continue editing.

The following windows are movable and consist of four primary controls:

- An element selection matrix
- A plot range control
- An editing range
- Editing buttons with associated value fields



Element Selection Matrix

The element selection matrix consists of a matrix of check boxes; the dimensions of this matrix reflect the dimensions of the model (and expanded inverse) currently under construction.

If an element of the selection matrix is checked, the corresponding element of the matrix is effected by the Set, Clip, Hanning, Interpolate, and Import editing operations (see below).

The Select matrix Elements buttons:

All – selects all checkboxes, **None** - deselects all checkboxes.

Main – selects all main diagonal checkboxes and deselects non-main diagonal checkboxes.

Off – opposite of main, selects all non-main diagonal checkboxes and deselects main diagonal checkboxes.

1 and 2 – similar to range buttons. Shift key defines 1 and 2 to current checkbox configuration; pressing 1 or 2 without shift button sets current checkboxes to the current definition for that button.

Plot Range Control

There are undo/redo buttons. The Checkboxes labeled Range, Edit, Checks, allow the little dialogs to be hidden or re-displayed to clear room on the screen. Log Plot toggles the graphical display setting of linear or log plot of the inverse transfer function.

If the Overwrite Inverse box is checked the editing changes to be permanently applied to the Inverse Transfer matrix. Otherwise, the changes are only used to compute the Expanded Inverse and are lost if re-edited.

Define Range Control

The editing range control specifies a range over span of the complex functions within the inverse transfer function.

Preset ranges may be defined using “Define Ranges”. Holding the shift key and selecting buttons 1 or 2 or 3 will define whatever the current range is to correspond with that button. Then, next time 1 or 2 or 3 is selected (without shift) that range is used to define the edit boxes (where 0 and 256 are shown here).

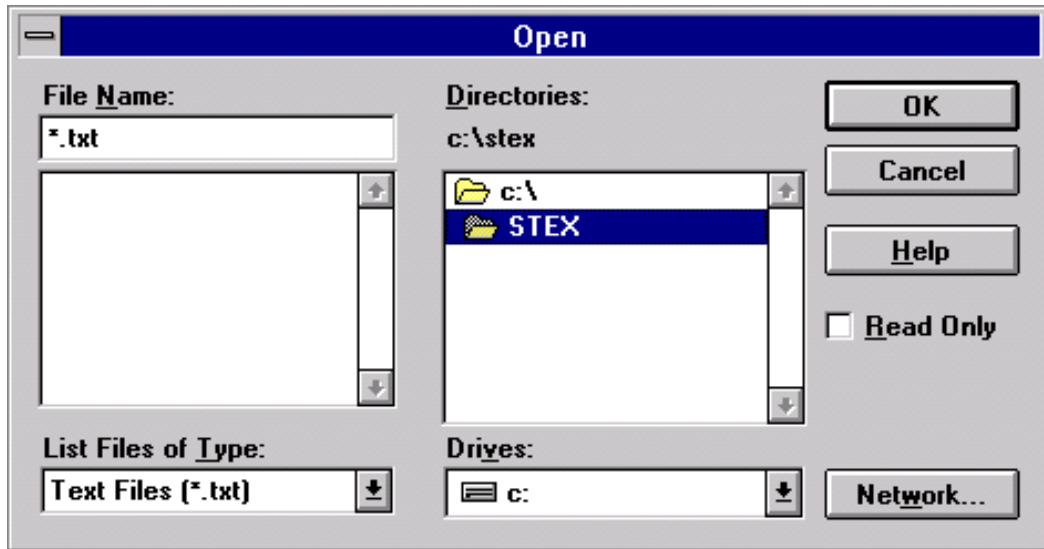
Editing Buttons with Associated Value Fields

The editing buttons provide the following operations:

- Set To (Set the amplitudes over a range to a single value)
- Clip Using (Limit the maximum amplitude over a range)
- Hannings (Apply a hanning operation over each matrix element)
- Import (Import a complex function from an external file)
- Interpolate (Perform linear interpolation over a range)

Using the Import option, a Selection dialog appears (page 2-20) that allows the operator to select an existing spectrum that will replace the elements selected in the checkboxes (Select Matrix Elements). The Import function allows the elements to be defined using a spectrum defined in the test group (not an external file).

When you select “Select” on the model-editing dialog, the **Open File** dialog appears.



Use this dialog to select and open the file that contains the spectrum you want to use to replace selected elements of the inverse transfer function. The spectrum must be complex and match the model in terms of spectral resolution and value count.

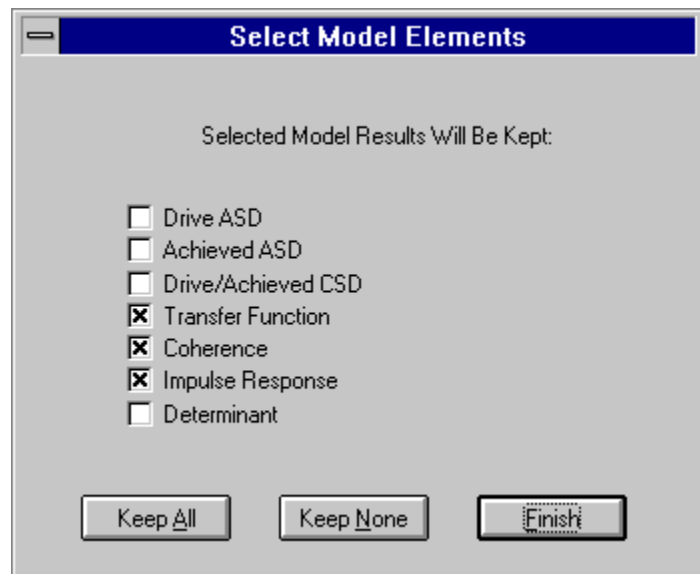
Plot Range Control

You may modify the limits of the inverse transfer function plots on the Edit Inverse dialog. By setting the “**From**” and “**To**” fields to the desired sub-span (from zero to Nyquist) and selecting “**Update**”, the graphs are redisplayed with the desired span.

When you select “**Exit**” on the Edit Inverse dialog, the expanded inverse is created from the inverse transfer function. (Refer to Appendix C for details of this process.) The following results are created:

- Achieved ASD and Drive ASD
- Achieved/Drive CSD
- Transfer Function
- Inverse Transfer Function
- Coherence Matrix
- Determinant Spectrum
- Impulse Response
- Expanded Inverse
- Specimen Transfer Function (if specimen channels are present)

The **Select Model Elements** dialog will appear to allow you to decide which model elements to keep or delete.



The expanded inverse is required for use with compensated tests; the inverse transfer function is required in case the expanded inverse needs to be recreated. The remaining model elements may be kept or deleted at the user's discretion. Check the boxes of the elements you wish to keep. You can press the "**Keep All**" button to select all the elements or the "**Keep None**" button to deselect all the elements. When you are finished selecting the elements, press the "**Finish**" button. Model elements not checked on the dialog are deleted before the main dialog reappears.

Naming Conventions

All data stored in the STEX database are given unique names that you choose when you create the data.

Examples:

Test Group: Test Group for XYZ (S.R. 100 Hz)

Result: Sweep 1 to 10 Hz at 1 g for 10 sec

When you create a test group, test or result you specify a name that describes the item.

A name must be unique so that no like data objects have the same name.

A name may be up to 40 characters in length.

A name that you select may not use the bracket characters [or]. When STEX creates a system result (i.e. time histories of a test or elements of the model), a special description is appended to the end, enclosed in brackets.

The possible types are:

DSD	Drive Spectral Density
ASD	Feedback Spectral Density
CSD	Cross Spectral Density (of Drive and Feedback)
TRF	Transfer Function
COH	Coherence
SPCTRF	Specimen Transfer Function
IMPULSE	Impulse Response
INVERSE	Inverse Transfer Function
EXPAND	Expanded Inverse Transfer Function
RANDOM	Random Time History or Vector
TARGET	Target Time History or Vector
ACHIEVED	Achieved Time History or Vector
DESIRED	Desired Time History or Vector
ERROR	Error Time History or Vector
DRIVE	Drive Time History or Vector

CORR	Correction Time History or Vector
SPECIMEN	Specimen Time History or Vector
DETERMINANT	Determinant

The channel (if present) is constructed based on the channel associated with the result.

Example:

My Model [CSD:Ch1,Ch2]

This system result is the CSD between channel 1 and channel 2.

The description is provided by the user.

Random Signal Architecture

STEX allows iterative correction of drive signals with its compensated time history test. The process involves adding a correction based on an error signal and a system model. A model is computed based on a model measurement test.

A model measurement test uses a random signal for each DOF included in the test. The following describes the way a random is assembled.

A random is a specially constructed time history designed to result in an accurate and useful characterization of the excitation equipment. The characterization is recorded in a transfer function matrix that describes the response of the system according to a given input.

The random signal is broken down into frames, all with the same point count. All randoms in a vector associated with the test use the same frame size which is derived from the number of lines in the shape ASD.

Frames are characterized as independent and repeat frames. Independent and repeat frames come in pairs; these frames are identical except that tapering is performed on the independent frames to avoid transient motions in the random response. These frames are derived from a shaping ASD (a real-valued spectrum) that defines the frequency content of the random.

The following example demonstrates the way frames are assembled in a two DOF, “three” frame random. Note that the three refers to three I/R pairs.

DOF1	T	R1		I2	R2	I3	R3	T		Z	Z	Z		Z	Z	Z
DOF2	Z	Z		Z	Z	Z	Z	T		R1	I2	R2		I3	R3	T

T = Tapered independent frames

I# = #th independent frame

R# = #th repeat frame

z = zero frame

Note *I2 and R2 are identical except that the beginning of I2 has been tapered to match the end of R1.*

The repeat frames are used in the spectral calculation used to derive the model. The response taken during the independent frame is assumed to contain a transient component that is not desirable and can cause inaccuracy in the model.

The number of frames in a test of N DOFs and M I/R pairs is:

$$NF = 2 * M * N + 1$$

Increasing the number of I/R pairs in a test increases both the accuracy of the model and the potential damage done to the specimen during the model measurement test.

With low damped systems, a larger frame size should be employed so that the transient component may be damped out during the independent frame.

Model Creation

A model represents a statistical characterization of the control system testing activities. The model consists of several elements including the expanded inverse transfer function. The remaining model elements are intermediate results of the expanded inverse computation.

The expanded inverse transfer function is used directly during the compensation process by which STEX iteratively attempts to improve the response characteristics of a test to match the target motions defined by the operator.

The model is defined for all control DOFs included in the model measurement (random) test from which the model is derived. The model matrices are n by n in size, where n is the number of DOFs, except for the specimen transfer function which has 7 DOF columns, and the number of rows corresponds to the number of specimen channels in the model measurement test.

The model elements are:

- Drive Auto Spectral Density (ASD)
- Achieved Auto Spectral Density (ASD)
- Drive/Achieved Cross Spectral Density (CSD)
- Transfer Function
- Inverse Transfer Function
- Coherence Function
- Impulse Response
- Specimen Transfer Function
- Expanded Inverse
- Determinant

These elements are described below.

Drive Auto Spectral Density (ASD)

The Drive Auto Spectral Density (ASD) is the auto spectral density of the drive vector of the model measurement test used to compute the model. This matrix is square (n by n , where n is the number of DOFs in the test).

Achieved Cross Spectral Density (CSD)

The Achieved Auto Spectral Density (ASD) is the auto spectral density of the achieved vector of the model measurement test used to compute the model. This matrix is square (n by n , where n is the number of DOFs in the test). Due to the vector nature of the data, ASD's may have non-zero phase.

Drive/Achieved Cross Spectral Density (CSD)

The Drive/Achieved Cross Spectral Density (CSD) is the cross spectral density of the drive and achieved vectors of the model measurement test used to compute the model. This matrix is square (n by n , where n is the number of DOFs in the test).

Transfer Function

The Transfer Function represents the response of the control system to a generic drive signal. Since the transfer function is a matrix, the relationship between DOFs is the model.

The transfer function is derived from the Drive Auto Spectral Density (ASD) and the Drive/Achieved Cross Spectral Density (CSD) of the model to which it belongs.

The transfer function is inverted to derive the Inverse Transfer Function.

Inverse Transfer Function

The Inverse Transfer Function is the inverse of the Transfer Function and is used to derive the Expanded Inverse Transfer Function. Using the Model editing dialog, you can effect the compensation process by manipulating the inverse transfer function. These changes are not permanently stored in the inverse but are used during the expansion to derive the expanded inverse.

Coherence Function

The Coherence Function measures the linearity of the system represented by the model.

A coherence value of 1 is ideal and represents linearity. A linear system is comparatively easier to iterate than a non-linear system.

A coherence value of 0 implies non-linearity. Coherence values close to zero usually prove more difficult to control using compensation.

Specimen Transfer Function

Since the DOF feedback vector is used as the “drive” parameter of the specimen transfer function calculation, the results may be less accurate. Unlike normal drive vectors, the feedback vector is effected by various control system characteristics (for example, cross coupling). These effects can compromise the transfer function calculation algorithm. Users should be aware of this when evaluating this model element.

The Specimen Transfer Function is the transfer function between the driven DOFs and the specimen acquisition channels in the model measurement test.

This matrix is not necessarily square. The number of rows correspond to the number of acquisition channels, while the number of columns correspond to the number of DOFs.

Each element of the specimen transfer function approximates the response of the specimen to the motion of the corresponding DOF.

This matrix is only computed if the model measurement test contains specimen channels. If more than six such channels are defined, only the first six are used.

Impulse Response

Better results may be obtained by increasing the frame size of the random shape spectra and creating a new model measurement test; adjusting the sampling interval (in a new test group) will also effect the frame duration.

The Impulse Response function is the transfer function represented in the time domain (via FFTs). The impulse response function is composed of time histories arranged in a matrix format and is a result of the model computation.

Problems may arise during a compensated test if the impulse response time histories do not damp out completely by the half frame point. In such a case, the transient component of the system response will not have been damped by the time the random repeat frame occurs during the model measurement test. This can result in inaccuracy in the transfer function.

Expanded Inverse

The Expanded Inverse is derived from the Inverse Transfer Function of the model to which it belongs.

Before expansion, you can edit the inverse using the Model Editing dialog.

Expansion is a process that is employed to make the inverse easier (i.e. faster) to use when using convolution during compensated test iteration. The steps of expansion are:

1. The user edit's the inverse transfer function matrix

For each spectrum of the matrix, STEX performs:

2. IFT into the time domain
3. Perform a half-frame circular shift
4. Append a frame of zeros
5. FFT the frame pair back to the frequency domain

From this you can see that the spectra of the expanded inverse have a resolution equal to half that of the original inverse. Moreover, the phase has been affected by this process, a fact that was evident on older versions of STEX because the expanded inverse plot showed a distorted phase component. This phase effect is necessary for the convolution process (and is a direct result of the expansion process); however, newer versions of STEX adjust the expanded inverse plot to make the phase look more normal.

The expanded inverse is the data actually used for compensation. The inverse is only used as a source of editing to create a new expanded inverse.

Determinant

The determinant is a complex spectrum that is derived during the process of inverting the transfer function of a model. Inversion is performed at each frequency along the span of the model. The inversion is performed on a n by n matrix of complex values. In this process, a complex determinant is computed.

The determinant can be used to identify frequencies that may have difficulty being corrected during compensation test iteration.

A zero in the determinant denotes a singularity in the transfer function that cannot be inverted.

File Formats

STEX FILE FORMATS (Import/Export)

Import formats	The following formats may be used for import:
STEX 2.X (Text)	This ASCII format conforms to the standard recognized by MTS STEX version 2. A header precedes the list of numerical values. The header includes value count and units information. Result types: Time History, Real and Complex Spectra and Shock Spectra.
ASCII No Header	This ASCII format consists of a list of values without the usual header information. The result must be simple (I.e. a time history, real or complex spectrum, or a response spectrum). A dialog will appear asking for header information (e.g. units); the type of the result is also specified. Result types: Time History, Real and Complex Spectra, and Shock Spectra.
ASCII Columns	The file represents one or more time histories. Each time history occupies a column in the file. If more than one time history is detected, a vector is also created. Result types: Time History and Vector
469D	This format may be read and written by MTS' digital controller software (MTS 469D). Result types: Time History and Vector
RPC ASCII	This format is read and written by MTS RPC version 3. Result types: Time History and Vector
RPC Binary	This format is read and written by MTS RPC version 3. Result types: Time History and Vector

Export Formats

The following formats may be selected when exporting:

STEX 2.X (Text)

This ASCII format conforms to the standard recognized by MTS STEX Version 2. A header precedes the list of numerical values. The header includes value count and units information.

Results types: Time History, Real and Complex Spectra, and Shock Spectra

ASCII No Header

This ASCII format consists of a list of values without the usual header information. If the result is a vector, each component occupies a single column in the output file.

Result types: Time History, Real and Complex Spectra, Shock Spectra, and Vectors

ASCII Delta-t

This format is identical to the preceding “ASCII No Header” format except that the first line of the output file identifies the sampling interval associated with the data that follows.

Result types: Time History and Vectors

STEX Data ASCII Format

This format may be used to export all result types to an external file. The format is ASCII so the resulting files may be edited. STEX can import these files.

469D

This format may be read and written by MTS’ digital controller software (MTS 469D).

Result types: Time History and Vectors

The following types of results may be imported and exported from STEX. Examples for each follow:

- Time History - real valued
- Time History - word valued
- Real Spectrum
- Complex Spectrum
- Response Spectrum

Example of real valued time history

”seconds	“	Independent Units
0.00000000e+000		Minimum
1.00000000e+001		Maximum
EVEN_TABULATED		Representation
1.00000000e-003		Resolution
10001		Values count
“g	“	Dependent Units
REAL_TYPE		Data type
1.00000000e+000		Scale
0.00000000e+000		Offset
1024		Points per frame
0.00000000e+000		
6.43990627e-003		
1.31930498e-002		
2.02590591e-002		
(9997 values follow)		

Note $Maximum = (Count-1) * Resolution$

Example of word valued time history:

“seconds	“	Independent Units
0.00000E+00		Minimum
2.86430E+01		Maximum
EVEN_TABULATED		Representation
3.33000E-04		Resolution
86016		Values count
“g	“	Dependent Units
WORD_TYPE		Data type
1.22070E-03		Scale
2.85341E-03		Offset
4096		Points per frame
-16 0 0 0 0 0 0 0		-16 0 0

(86006 values follow)

Note Each value of the time history is computed as $WORD/2048 \times Scale + Offset$. Also, this format may be imported to STEX only. The format is used by older versions of STEX to record sampled data.

Example of real valued spectrum:

“Hz	“	Independent Units
0.00000000e+000		Minimum
5.00000000e+002		Maximum
EVEN_TABULATED		Representation
9.76562500e-001		Resolution
512		Values count
“g	“	Dependent Units
REAL_TYPE		Data type
1.00000000e+000		Scale
0.00000000e+000		Offset
1024		Points per frame
0.00000000e+000		
9.76562500e-001		
(510 values follow)		

Example of complex valued spectrum:

“Hz	“	Independent Units
0.00000000e+000		Minimum
5.00000000e+002		Maximum
EVEN_TABULATED		Representation
9.76562500e-001		Resolution
512		Values count
“g/g	“	Dependent Units
COMPLEX_TYPE		Data type
1.00000000e+000		Complex real scale
1.00000000e+000		Complex imaginary scale
0.00000000e+000		Complex real offset
0.00000000e+000		Complex imaginary offset
1024		Points per frame
7.31775392e-001	6.51248109e-001	
9.76517668e-001	1.84542005e-001	
(510 complex values follow)		

Note *Adjacent floating point values represent the real and imaginary components of the complex value.*

Example of complex valued spectrum:

“Hz	“	Independent Units
0.00000000e+000		Minimum
5.00000000e+001		Maximum
PAIRED		Representation
153		Number of pairs
“g’s	“	Dependent Units
REAL_TYPE		Data type
1.00000000e+000		Scale
0.00000000e+000		Offset
1024		Points per frame
2.50000000e-001	7.38699804e-002	
2.58867997e-001	7.64992831e-002	
2.68050560e-001	7.92136816e-002	
(151 value pairs follow)		

Note *The first column represents the X axis and the second represents the Y axis. The X values must be in order and may not be zero or negative.*

Result Units

Simple results (i.e. time histories, spectra, and response spectra) are functions of either time or frequency. The values of these functions are measured in terms of specific units. Typical units are:

Acceleration Units	
g	g's
m/sec ²	M/second ²
m/s ²	m/s/s
m/sec/sec	m/second/second
meters/sec ²	meters/second ²
meters/s ²	meters/s/s
meters/sec/sec	meters/second/second
gal	in/sec ²
in/second ²	in/s ²
in/s/s	in/sec/sec
in/second/second	inches/sec ²
inches/second ²	inches/s ²
inches s/s	inches/sec/sec
inches/second/second	ft/sec ²
ft/second ²	ft/s ²
ft/s/s	ft/sec/sec
ft/second/second	feet/sec ²
feet/second ²	feet/s ²
feet/s/s	feet/sec/sec
feet/second/second	cm/sec ²

Acceleration Units (Continued)

cm/sec/sec	cm/s ²
deg/s ²	deg/sec ²
deg/sec/sec	deg/second/second
radians/s ²	radians/sec ²
radians/sec/sec	radians/second/second
rad/s ²	rad/sec ²
rad/sec/sec	rad/second/second
degrees/s ²	degrees/sec ²
degrees/sec/sec	degrees/second/second
deg/second	degrees/second
deg/sec	degrees/sec

Velocity Units

cm/s	cm/sec
cm/second	centimeters/s
centimeters/sec	centimeters/second
mm/s	mm/sec
mm/second	millimeters/s
millimeters/sec	millimeters/second
in/s	in/sec
in/second	inches/s
inches/sec	inches/second
feet/s	feet/sec
feet/second	ft/s
ft/sec	ft/second
miles/hour	m/s

Velocity Units (Continued)

m/sec	meters/second
Displacement	mm
millimeters	cm
centimeters	m
meters	in
inches	ft
feet	yards
miles	

Units are defined for both axes of a result. The independent axis units are almost always either “seconds” or “Hz”; the dependent axis units vary.



MTS Systems Corporation

14000 Technology Drive

Eden Prairie, Minnesota 55344-2290 USA

Toll Free Phone: 800-328-2255

(within the U.S. or Canada)

Phone: 952-937-4000

(outside the U.S. or Canada)

Fax: 952-937-4515

E-mail: info@mts.com

Internet: www.mts.com

ISO 9001 Certified QMS