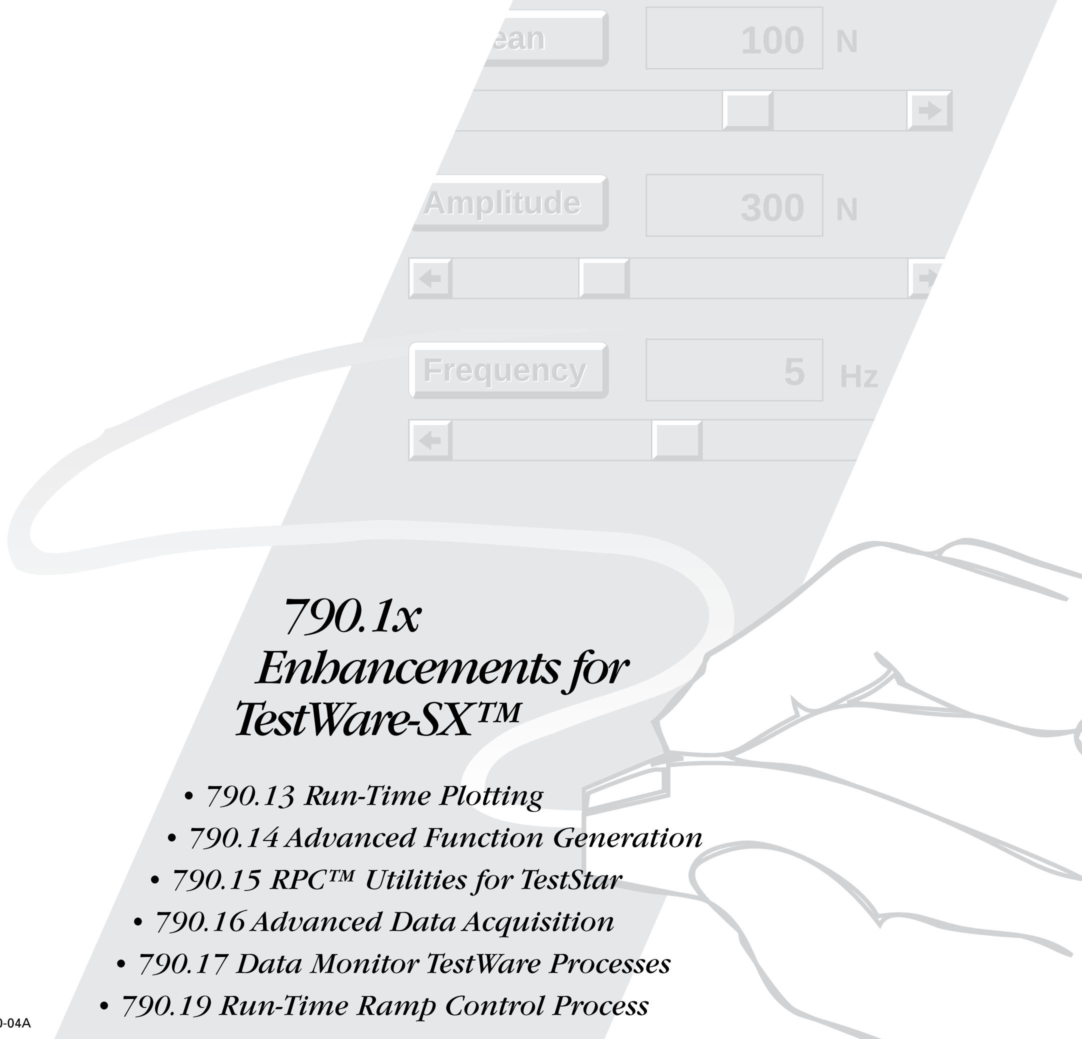
MTS®

TestStar™ II

Control System



790.1x Enhancements for TestWare-SX™

- *790.13 Run-Time Plotting*
- *790.14 Advanced Function Generation*
- *790.15 RPC™ Utilities for TestStar*
- *790.16 Advanced Data Acquisition*
- *790.17 Data Monitor TestWare Processes*
- *790.19 Run-Time Ramp Control Process*

150330-04A

Proprietary data

Copyright information

Trademark information

Copyright information

Manual Number	Publication Date
150330-01A	August 1993
150330-02A	April 1994
150330-03A	March 1995
150330-04A	March 1996

Table of Contents

Chapter 1 Introduction 7

Chapter 2 Run-Time Plotting 27

Chapter 3 Advanced Function Generation 35

Chapter 7 Data Monitor Processes 101

Chapter 8 Run-Time Ramp Control 127

Index 143

Chapter 1

Introduction

MTS 790.1X processes are optional processes you can use along with the standard processes included in your 790.10 TestWare-SX™ application to expand your test design possibilities.

Note *Your system may be equipped with some or all of these products, depending on what you purchased—so you need refer only to the parts that pertain to you.*

Contents

This chapter introduces these processes:

In addition, this chapter provides information regarding:

This topic describes how 790.1X processes interface with the standard processes in your 790.10 TestWare-SX™ application. It also discusses the phase shift that may occur between transducers and test events.

This topic describes how to install 790.1X processes on your system.

This topic lists information about contacting MTS Systems Corporation.

Prerequisites

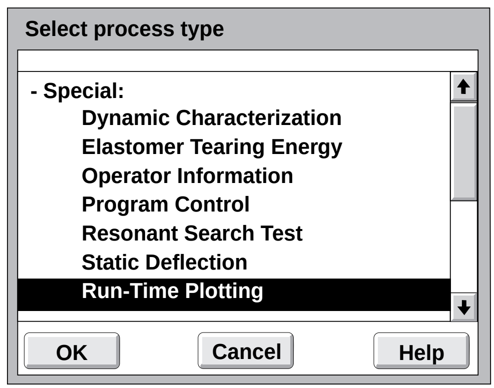
You must have 790.00 TestStar and 790.10 TestWare-SX to use the processes described in this manual.

Model 790.13 Run-Time Plotting Process

The Model 790.13 Run-Time Plotting process opens a window and plots data from one of the system sensors.

You can find the Run-Time Plotting process with the Select Process Type window when you are creating a test template.

The Run-Time Plotting process is added to the list of Special processes.



Model 790.14 Advanced Function Generation TestWare

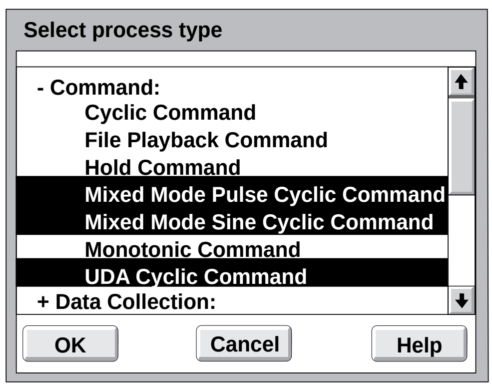
Model 790.14 Advanced Function Generation TestWare adds the following advanced command processes to TestWare-SX:

- Mixed Mode Sine Cyclic
- Mixed Mode Pulse Cyclic
- User Defined Arbitrary (UDA) Cyclic

These command processes offer unique waveshapes and produce vary accurate test results using special compensation techniques.

You can access the Advanced Function Generation processes with the Select Process Type window when you are creating a test template:

The Advanced Function Generation processes are added to the list of Command processes.

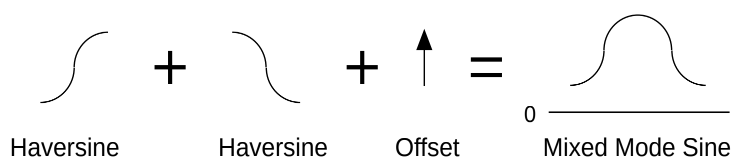


Introduction

Mixed mode sine cyclic

The mixed mode sine cyclic command is similar to the standard cyclic command, with the following differences:

- The segment shape is fixed with the haversine waveform.
- Two control parameters can be used for the waveform; one for the amplitude, the other for the mean level.
- This process uses phase and amplitude control (PAC compensation) to optimize the system's response to mixed mode sine commands.

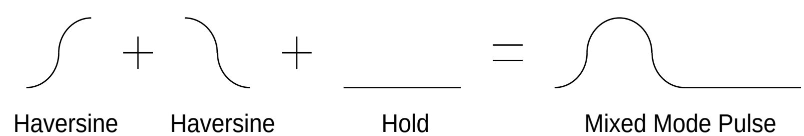


Mixed mode pulse cyclic

The mixed mode pulse cyclic command may be thought of as two haversine command segments and a hold time command segment combined into a single segment.

- The shape of the waveform is controlled by pulse amplitude and pulse width parameters, and the waveform can be offset from zero.
- This process uses a frequency-based iterative technique (FIT compensation) to optimize the system's response to mixed mode pulse commands.

Two control parameters can be used for the waveform—one for amplitude, the other for mean offset.

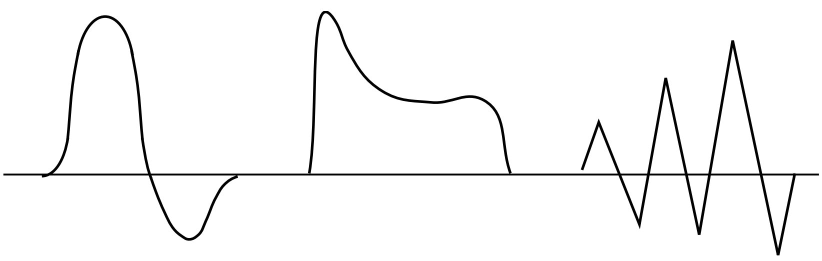


UDA cyclic

The UDA (user defined arbitrary) cyclic command lets you define a shape for a single segment that is repeated. You define the shape of the waveform with a file that specifies 1024 end level values and units.

- This process uses a frequency-based iterative technique (FIT compensation) to optimize the system's response to UDA cyclic commands.

Three arbitrary shape examples that could be defined with shape files.



Compensation methods

There are two methods used to optimize the performance of your system when using 790.14 Advanced Function Generation processes.

- FIT compensation is a frequency based iterative technique that dynamically optimizes the performance of your system. The mixed mode pulse cyclic command and the UDA cyclic command processes use FIT compensation.
- Patented PAC compensation provides phase and amplitude control. PAC is part of the TestStar software. See the TestStar Reference manual for information about the Adjust Compensators window (located in the Adjust menu). The mixed mode sine cyclic command process uses PAC compensation.

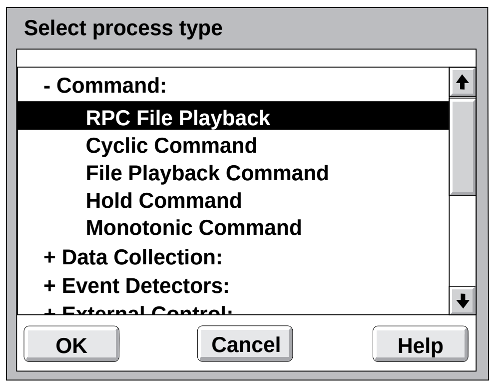
Model 790.15 RPC™ Utilities for TestStar

Model 790.15 RPC™ Utilities for TestStar includes a process for TestWare-SX that lets you use RPC files with TestStar.

This process is similar to the standard file playback process except it can only use RPC™ files. RPC (Remote Parameter Control) files are created on a RPC system.

You can find the RPC File Playback process with the Select process type window when you are creating a test template.

The RPC File Playback process is added to the list of Command processes.

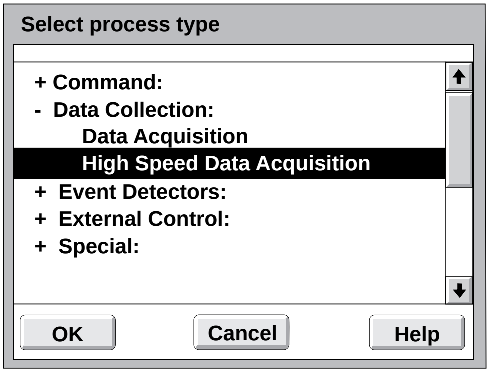


Model 790.16
Advanced Data Acquisition TestWare

Model 790.16 Advanced Data Acquisition TestWare adds the High Speed Data Acquisition process to TestWare-SX.

- The high speed data acquisition process is similar to the standard data acquisition process. It has a higher rate of data acquisition instead of the versatility of the standard data acquisition process.
- The High Speed Data Acquisition process can acquire data at a maximum rate of 70 kHz (for a single channel) vs. 5 kHz for the standard data acquisition process.
- This process cannot run at a rate less than 100 Hz. The number of data input channels reduces the maximum data acquisition rate.

You can find the high speed data acquisition process with the Select process type window when you are creating a test template.



The high speed data acquisition process is added to the list of Data Collection processes.

Model 790.17
Data Monitor TestWare Processes

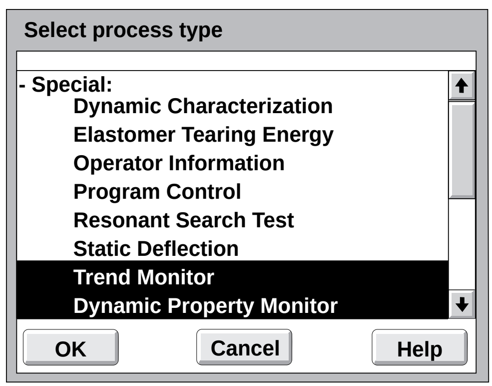
Model 790.17 adds the following special processes to TestWare-SX:

- Trend Monitoring Process
- Dynamic Property Monitoring Process

These processes are designed for very long fatigue tests. They let you monitor several parameters and dynamic properties. These processes help you identify tends that may indicate the beginning of specimen failure.

You can find the trend monitoring process or the dynamic property monitoring process with the Select Process Type window when you are creating a test template.

The Trend Monitoring and Dynamic Property Monitoring processes are added to the list of Special processes.

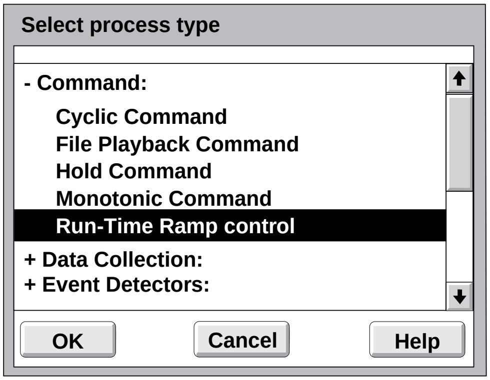


Model 790.19 Run-Time Ramp Control.

Model 790.19 Run-time ramp control process lets you to simultaneously ramp up to four control channels in an incremental fashion, stepping toward indeterminate end levels.

You can find the ramp control process with the Select Process Type window when you are creating a test template.

The Run-time ramp control process is added to the list of Special processes.



Test Design Considerations

Using 790.1X processes with TestWare-SX

MTS 790.1X processes are optional processes you can use with the standard processes included in your 790.10 TestWare-SX application to expand your test design possibilities. As you know, the TestWare-SX application is a collection of general-purpose test elements called processes.

Processes can be thought of as building blocks. Each building block performs a specific part of test, such as function generation, data acquisition, or data detection.

You can join these building blocks together to create test instruction files. Like standard processes, each 790.1X process is a building block that performs a specific part of a test.

You can join 790.1X processes together with standard processes to construct tests. You can also join 790.1X processes with more advanced processes, such as the790.3X processes (typically used for elastomeric testing) or, 790.6X processes (typically used for rock mechanics testing), to construct more sophisticated tests.

Phase shift

Phase shift refers to the lag between a physical event in a test and the measured response of a sensor. This phenomenon occurs because transducers and sensor conditioners, regardless of manufacture or application, do not have perfectly flat frequency response.

In general, the phase shift of a given sensor conditioner increases with frequency and the amount of filtering used on the signal.

Also, different types of transducer conditioners (AC, DC, Charge amplifiers, etc.) have different phase shift characteristics. The magnitude and relevance of the phase shift introduced depends on the frequencies at which you test, the type of transducer/conditioner pairs you use in the acquisition process, and the filters used in the conditioner.

Neither the standard TestWare-SX™ Data Acquisition process nor the High Speed Data Acquisition process compensates for phase shift. They behave similarly to any other data storage device (i.e., oscilloscope, pen plotter, strip chart recorder, etc.) in the measurement of analog signals. In some situations, this may adversely effect your data. Because of phase shift, measured phase angle relationships between input displacements and output loads may not be the same as actual relationships.

In multi-axial systems, requested phase between channels may be different from the actual inter-channel phase.

For systems used in the determination of dynamic properties using a DC load cell and an AC displacement transducer with minimal filtering of the analog signal, the phase shift will be approximately 0.7 degrees per 10 Hz.

The important thing is to be aware that it exists, and to anticipate its effect on your test results. As always, if you have questions regarding technical issues contact MTS through the HELPLine or contact your local MTS service representative.

Software Installation

Normally, the TestWare products you purchased with your test system will be installed on your system when your TestStar system software is installed. This is also true for software updates.

This section describes how to install these software products as an addition to an existing system or independent software updates.

Procedure

The following is a list of steps to install any or all of the optional TestWare software products described in this manual.

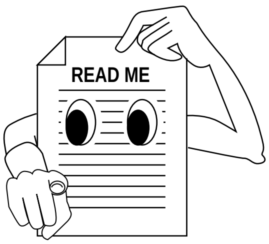
Note *Normally, the optional software products are installed when TestStar and TestWare-SX are installed. This procedure assumes you are adding the products to your system after the initial software installation or software update.*

The setup program has several operating modes. This procedure describes only the Additional Application Installation mode. The Software Installation chapter of the TestStar Installation Manual (Chapter 4) describes how to use the setup program.

Step 1 Backup your disks

Make a copy of the software. Use the copies to install the software onto the hard disk. Keep the original copy in a safe place.

Step 2 Read the readme file



MTS software typically includes a README.TXT file that contains late-breaking information not included in this manual. If the file is included, it should be opened and the information reviewed before installing the system software. The file is located on the disk.

Note *The software installation program will ask you if you have read the readme file. The installation program will automatically display the file if you want to see it.*

To read the file, insert the 790.xx application disk, double-click Drive A on the desktop then double-click the README.TXT file or open an OS/2 window and enter the following command:

```
TYPE A:README.TXT | MORE
```

Press any key to display the next page of the file. Press ^C (cntl + C) to exit the file.

Step 3 Start the software installation program

The TestStar setup program is called SETUP.EXE. This program is located in the TS2 directory of the hard disk after TestStar II is installed.

Note *The following assumes the setup program is in the default TestStar directory C:\TS2\SETUP.*

To start the setup program perform the following:

A Double-click the OS/2 System icon

B Double-click the Command Prompts folder

C Double-click the OS/2 Window icon or the OS/2 Full Screen icon.

D Type the command:

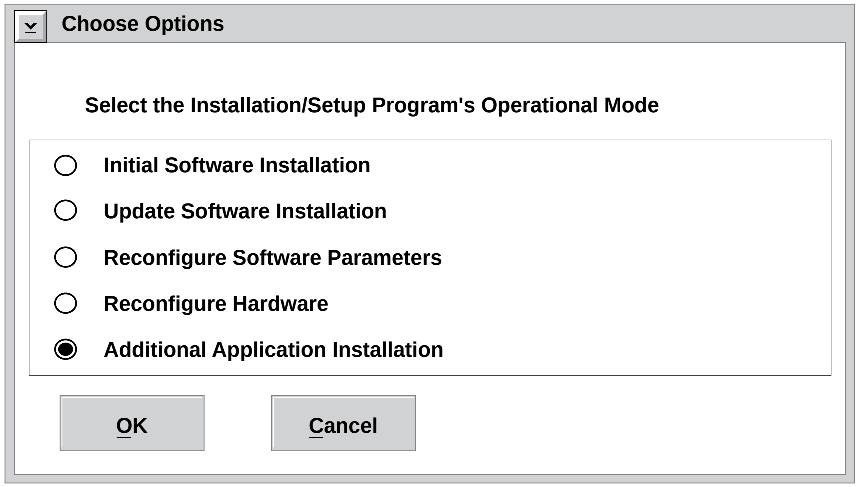
```
C:TS2\Setup
```

Step 4 Select the Setup Program's Operational Mode

The setup program displays the Ask Options prompt: Select **Additional Application Installation** to install additional applications. This mode selection is dedicated to installing optional application software.

Select **Additional Application Installation** from the list shown.

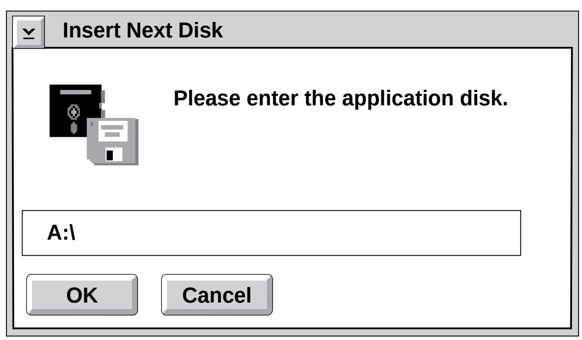
Press the OK button.



Step 5 Insert the application disk in the appropriate drive

Enter the drive location that you will use to install the software. Then install the application disk in that drive.

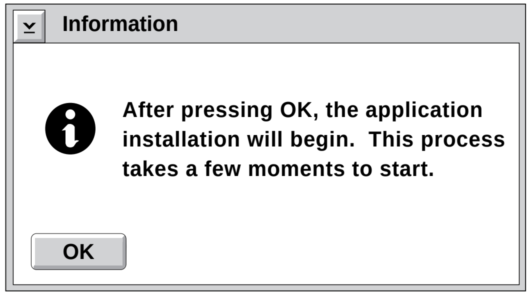
The default drive location is A.

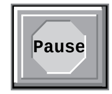


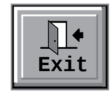
Step 6 Press OK to start the software installation

Press the **Pause** pushbutton to suspend the installation

If you do not want to continue the installation, you can press the **Exit** pushbutton.





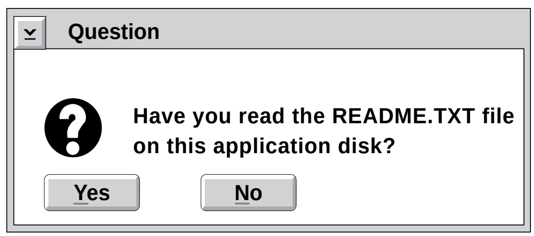


Step 7 Note about the readme file

If you select No, a message tells you that the readme file will be displayed. When you have finished reading the file, close the window to continue the setup program.

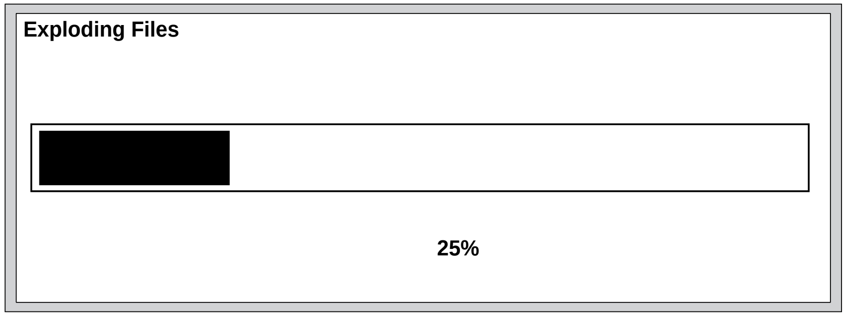
Select **Yes** to continue the installation procedure.

Select **No** to display the readme file.



Step 8 The setup program displays the installation progress

While the software is being installed, you will see the progress of the installation.

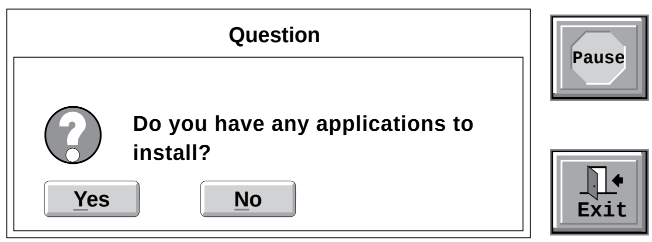


Step 9 Install any additional applications

Repeat steps 5 through 9 for each application you want to install.

Select **Yes** to install another application.

Select **No** if you do not want to install another application and end the setup program.



Technical Assistance

If you have any questions about an MTS system or product, contact the MTS corporate service center.

Note *Review the following pages for information about what to expect when you contact us.*

Address

MTS Systems Corporation
Service Support Group
Technical Support Department
14000 Technology Drive
Eden Prairie, Minnesota 55344-2290

Telephone

In the United States (all 50 states)	HELPLine (800) 328-2255
Outside U.S.	Contact your local service center

Telex

29-0521

Fax

Technical support questions	(612) 937-4766
General questions	(612) 937-4515

Internet

E-mail	Info@mts.com
Home Page	http://www.mts.com

Things We Might Ask You

Your call will be registered by a HELPLine agent. The agent will ask for you site number. If you do not have an MTS site number or do not know your site number, you should contact your MTS sales engineer.

The HELPLine agent may also ask to verify the following information:

- Your company's name
- Your company's address
- Your name and the telephone number where you can normally be reached.

If you have called before regarding this problem, we can recall your file. You'll need to tell us the following:

- The MTS work order number.
- The name of the person who helped you.
- Be prepared to respond to questions when interfacing with MTS technical support personnel. We may ask you to perform certain tasks so we can locate the source of the problem.

Before you call

Prepare the following information before you call HELPLine support to prepare for the troubleshooting process.

Know your site number and system number.

Describe the problem you are experiencing:

- How long has the problem been occurring?
- Can you reproduce the problem?
- Were any hardware changes made tot he application or system operating software before the problem started?

Have the following information available:

- If relevant, print-outs of configuration files, and test procedures.
- The type or model number of your test frame, load unit, etc.
- The type of model number of your controller
- Model number and size of your hydraulic service manifold
- Serial number of any suspect component

If you are experiencing a computer problem, please have the following information available:

- Manufacturer's name
- Manufacturer's model number
- Type of system memory
- Amount of system memory
- Floppy drive information (model number, size, and capacity)
- Hard drive information (model number, size, and capacity)
- Manufacturer of printer/plotter and model number
- Mouse information (bus, serial; connected to what port?)
- Graphics board information (manufacturer and model)
- What other boards are installed in the computer?
- Is the system part of a network?

If you are experiencing a software problem, please have the following information available:

- Operating software information
  - What type of operational software are you running?
  - What version level of operating system is running?
  - What window type is used?
- Application software information:
  - What applications are you running? (MultiPurpose TestWare, etc.)
  - Know the version of each software application involved.

Other software being used:

- What other software was running when the problem was encountered? This could include such things as screen savers, keyboard enhancers, print spoolers, etc.
- Know the name and version of each software program involved.

While on the phone

Prepare yourself for troubleshooting while on the phone:

- Try to call from a telephone close to the system so that you can conduct some active testing over the phone.
- Have the original operating and application software disks available.
- If you are not familiar with all aspects of the operation of the equipment, have the necessary people available to assist you.

Prepare yourself in case a call back is required:

- Remember to ask for the work order number.
- Record the name of the person who helped you.
- Make sure you are able to write down any specific instructions to be followed, such as data recording or performance monitoring.

Chapter 2

Run-Time Plotting

This chapter describes the 790.13 Run-Time Plotting process. This is a special process that provides on-screen plotting of selected data while the process is running.

Contents

How it works

The run-time plotting process is usually sequenced at the beginning of a step and runs for the duration of the step. Several plots can be displayed simultaneously by adding additional plot processes to the step.

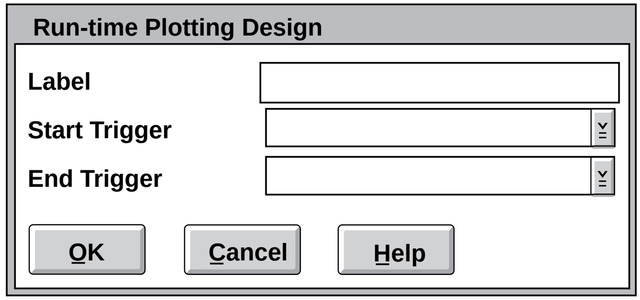
When you start a test that includes a run-time plotting process, a plot window opens. Data is continuously updated based on time or level crossing. The test step can run for hours, days or longer. You do not need to wait for the step to end before the plot is updated.

The plot windows close when the step is done. If the test is suspended, and the TestWare-SX test mode changes from execute to edit, all plot windows close.

Run-Time Plotting Design Window

This window names the process and specifies how the it starts and stops.

The information in this window is saved with the test template.



CONTROL	FUNCTION
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger

Start trigger

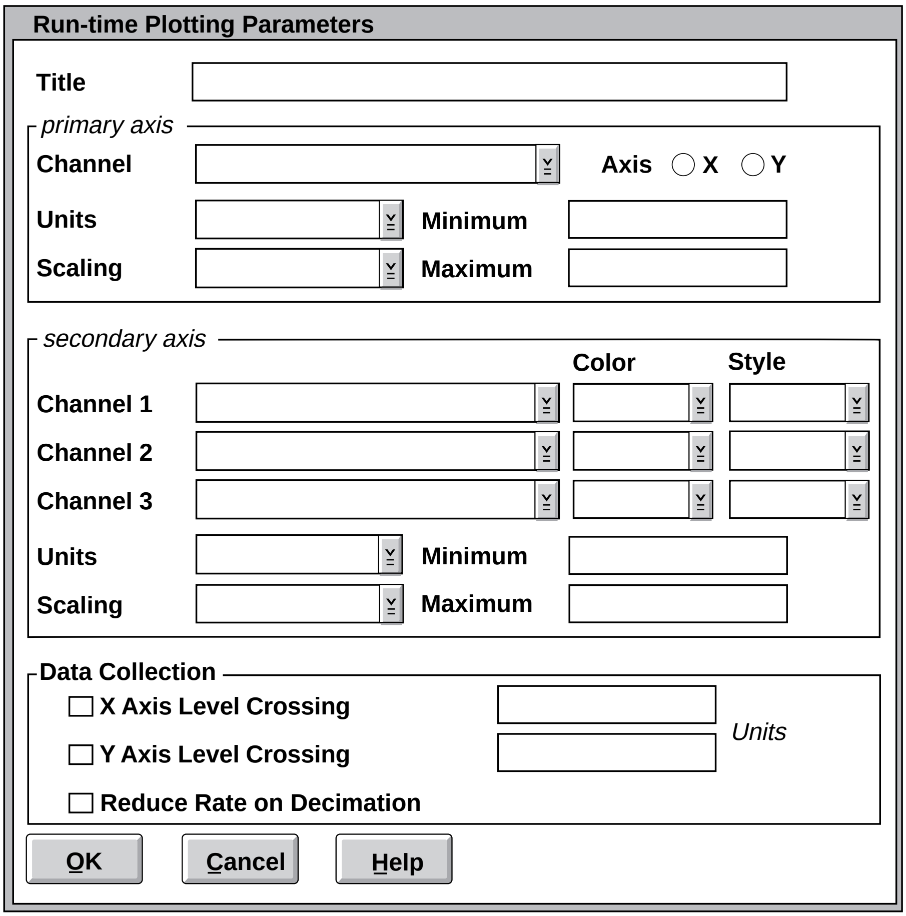
- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

End trigger

- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

Run-Time Plotting Parameters Window

This window assigns a title to the plot in the Run-Time Plot window and determines the primary axis and data collection method for that plot. You must create a run-time plotting process in the template with the Run-Time Plotting Design window before you can access this window. The information in this window is saved with the test procedure.



Control	Description
Title	Names the plot. This name appears in the title bar of the Run-Time plot window.
primary axis	Defines the primary axis. The primary axis can be the X axis or the Y axis.
Channel	Selects the input signal you want displayed on this axis.
Axis	Selects the X or Y axis is as the primary axis.
Units	Selects the units for the channel. The unit dimension is determined by the selected channel.
Scaling	Specifies if this axis of the plot is **linear** or **logarithmic**.
Minimum Maximum	Sets the initial minimum and maximum range of this axis in the run-time plotting window.
secondary axis	Defines the secondary axis. The axis title is determined by the axis button selected for the primary axis.
Channels 1, 2, 3	Up to three channels of data can be defined for this axis. Select an input signal (except the one selected for the primary channel) for each channel you want to display.
Color	Assigns a color to each channel. **Black** **Red** **Blue** **Green** **Cyan**
Style	Assigns a style of line for each channel plot. **Solid** **Dashed** **Dotted**
Units	Selects the units for the channel. The available units are determined by the dimension associated with the input selected for Channel 1.
Scaling	Specifies if this axis of the plot is **linear** or **logarithmic**.
Maximum Minimum	Sets the initial minimum and maximum range of this axis in the run-time plotting window.
Data Collection	Defines how data is collected. The X Axis and Y Axis Level Crossing cannot be used simultaneously with Reduce Rate on Decimation.
X Axis Level Crossing	Updates the plot data for each channel when the X axis channel changes by an amount you specify.
Y Axis Level Crossing	Updates the plot data for each channel when the any Y axis channel changes by an amount you specify.
Reduce Rate on Decimation	Enables the reduce rate on decimation feature. It reduces the rate of data collection each time the data buffer becomes full. This feature disables both level crossing features.

The X and Y axis selection

You can select the X or Y axis as the primary axis using the axis buttons. The axis title reflects the button selected. This selection determines the orientation of your data on the plot. The default is X as the primary axis. Data collected for the selected input channel is displayed along this axis. The remaining axis is the secondary axis. Up to three channels of data can be displayed on the secondary axis.

Data collection

Collected data is displayed in the run-time window plot. The plot buffer can store 4000 data points. When the plot buffer fills, half the data is removed so new data can be displayed. See *Decimation* on page 32 and *Decimation rate reduction* on page 33 for more information.

Data can be collected in the following ways:

Level crossing

You can specify an amount of change in any sensor feedback channel (X or Y axis) to trigger data collection for all channels. When the X axis or Y axis level crossing feature is enabled, you can enter the value desired for the level crossing increment.

For example, suppose you set the level crossing value to 1 mm. When any sensor feedback channel detects a 1 mm change, the run-time plot window updates the data from all of the channels.

Time base

If neither the X or Y axis level crossing feature is enabled, data is collected from all channels at a fixed time interval. You can specify a time interval by selecting Time as the X axis channel and enable the X axis level crossing.

Segment count

You can select a segment count or cycles as a channel for either axis. Then you can enable the X or Y axis level crossing and enter a segment count or number of cycles.

Note *2 segments = 1 cycle.*

Decimation

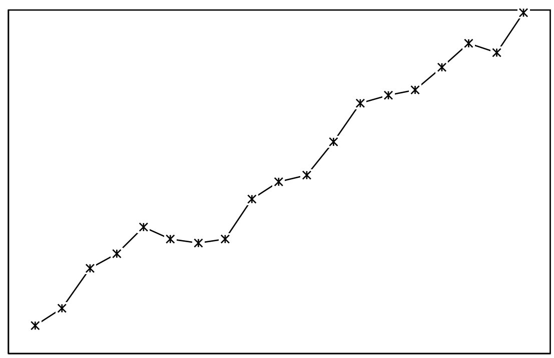
The plot buffer for each data series can hold 4000 data points. When a buffer becomes full, every other data point is removed to make room for more data. An additional 2000 data points of data is collected at the same rate as the first 4000 data points then decimation is repeated.

- Decimation is always enabled.
- Decimation reduces the resolution of the plot.

For example, suppose you collect data once per second. After 4000 data points are collected, every other data point is removed so 2000 data points remain. Data continues to be collected once per second until the buffer is full again. Then the data is decimated and new data continues to be collected once per second. At this point, the first 500 data points (1⁄4 of the buffer) are at 4 second intervals, the second 500 data points are of 2 second intervals, and the last 1000 data points are at 1 second intervals.

Before Decimation:

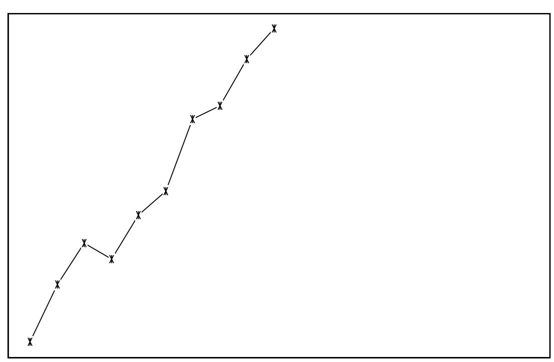
Data points are acquired until the data buffer is full (4000 data points).



After Decimation:

Half the data points are removed from the plot series, 2000 points remain.

Here, the remaining 2000 data points are compressed to half the display.



Decimation rate reduction

When the **Reduce Rate On Decimation** feature is enabled, data is then collected at half the previous rate following a decimation.

- When the data rate reduction feature is not enabled, the first data taken is at a different resolution than the most recent data taken.
- When the data rate reduction feature is enabled, the data resolution is uniform.

For example, suppose you collect data once per second. After 4000 data points are collected, every other data point is removed so points 2000 data points remain. Data continues to be collected every 2 seconds until the buffer is full. Then the data is decimated and new data continues to be collected every 4 seconds.

Number of Decimations	Data Acquisition Rate
0	every data point
1	every 2nd point
2	every 4th point
3	every 8th point

Note *You may not notice a change in the data plot on your computer monitor for the first several decimations. A high resolution monitor may show only 1024 data points.*

Chapter 3

Advanced Function Generation

This chapter describes MTS Model 790.14 Advanced Function Generation TestWare. The Advanced Function Generation TestWare contains the following command processes and compensation technique:

- Mixed Mode Sine Cyclic Command
- Mixed Mode Pulse Command
- UDA Cyclic Command
- FIT Compensation

Contents

Mixed Mode Sine Cyclic Command

A mixed mode sine cyclic process produces a command signal that controls a servovalve or servomotor. This process should be sequenced in series with other command processes. Use this process in testing applications such as engine mounts, rotating machinery, or automotive suspension components.

Note *This process uses PAC compensation (phase and amplitude compensation). PAC is part of TestStar. See the TestStar Reference manual for information about the Compensation window (located in the Adjust menu). The following are controls for PAC compensation:*

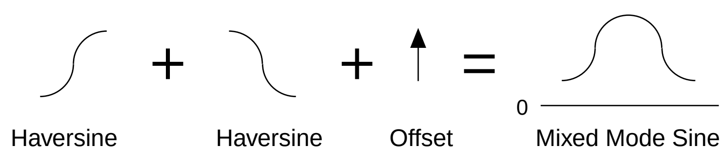
- *Phase and Amplitude Convergence Rate*
- *Amplitude Limit*
- *Mean Convergence Rate*

Contents

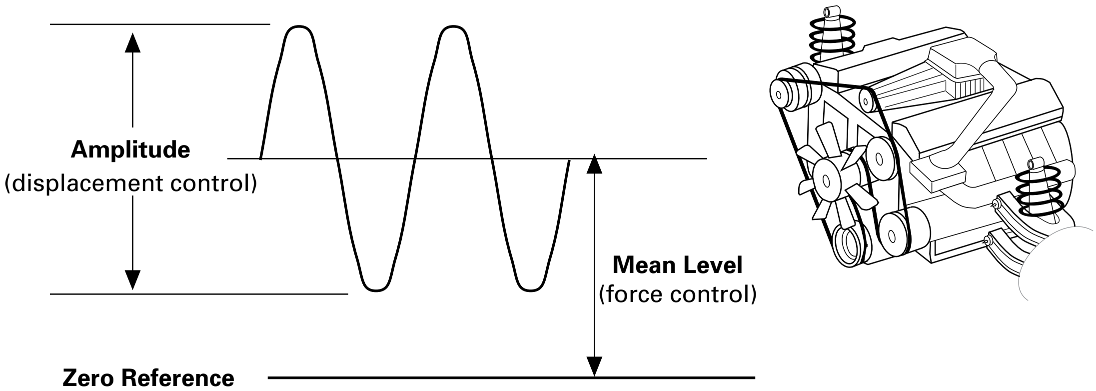
How it works

The mixed mode sine cyclic command is similar to the standard cyclic command. The mixed mode sine command has these differences:

- The segment shape is fixed with the haversine waveform.
- Two control parameters can be used; one for amplitude, the other for the mean level of the cyclic waveform.
- Uses phase, amplitude, and mean level compensation (PAC compensation) of the TestStar software.



For example, suppose you are testing an engine mount. In this case the mean level represents the weight of the engine and the amplitude represents the vibration caused by the engine.

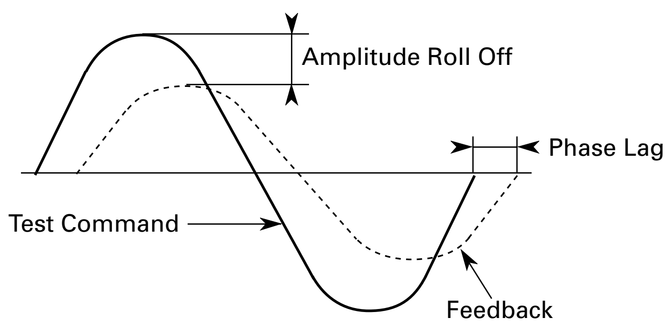


Phase and amplitude control

Phase and amplitude control compensates for amplitude roll-off and phase lag. Amplitude roll-off refers to the tendency of amplitudes measured by the sensors to be less than the desired amplitudes. Phase lag refers to the tendency of the feedback signal to trail the command signal at higher frequencies.

The phase lag and amplitude roll-off are determined when the feedback is compared to the test command.

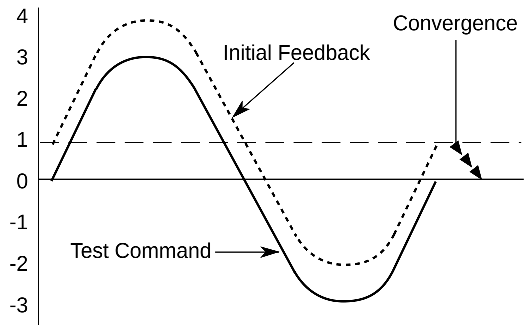
The PAC algorithm compensates for these characteristics.



Mean level control

The mean level of the test command and the feedback are compared to detect any difference.

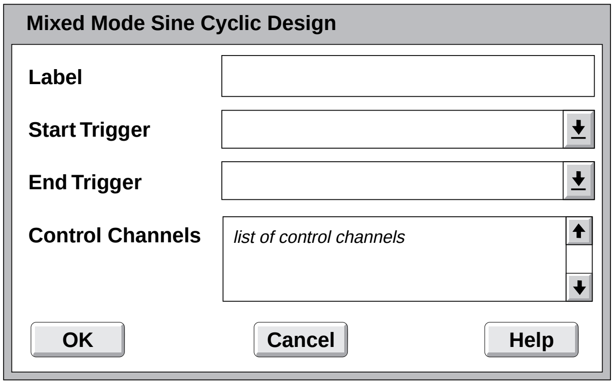
The TestStar PAC algorithm compensates for mean level differences.



Mixed Mode Sine Cyclic Design Window

This window names the process and specifies how it starts and stops.

The information in this window is saved with the test template.



CONTROL	DESCRIPTION
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger
Control Channels	Specifies which control channels the process is applied (more than one may be selected). The control channels are defined by the TestStar configuration file.

Start trigger

- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

Note *Don't try to start more than one command process at a time—if you do, you will cause an error. Sequence test command processes in series.*

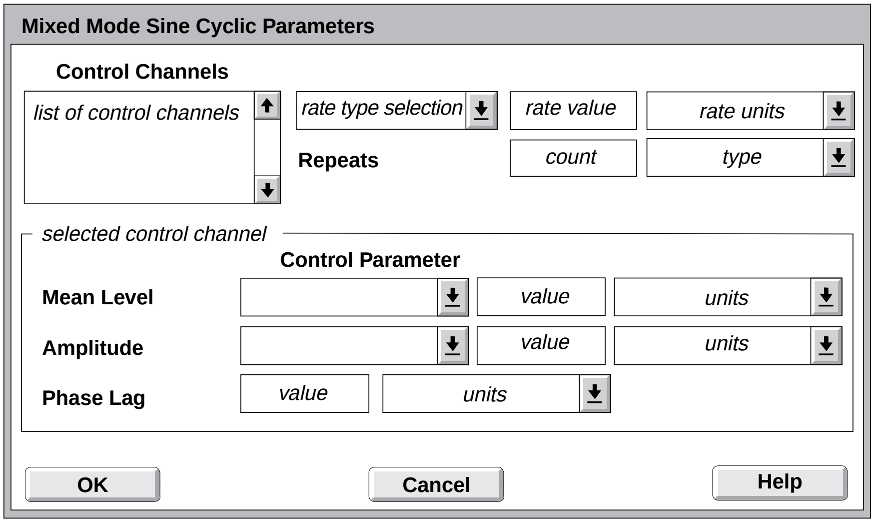
End trigger

- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

Mixed Mode Sine Cyclic Parameters Window

You must establish a cyclic process with the Mixed Mode Sine Cyclic Command Design window before you can use this window. The information in this window is saved with the test procedure.

Use this window to define the specific characteristics of the cyclic waveform.



CONTROL	DESCRIPTION
Control Channels	Displays a list of the control channels selected in the design window. Select the control channel to define. The control channel labels the control parameter area where you define the parameters for that channel.
rate type selection	Specifies the type of rate values for the process. The selections are: **Frequency** **Time** **Rate**
rate units	Selects the units for the rate type value. The type of units depends on the type of rate you select.
rate values	Specifies the value you want for the command. Type a value in the middle entry field.
Repeats	Selects **segments** or **cycles** as the type of count. Select type of repeats you want and enter the number of repeats you want. ***Note*** *2 segments = 1 cycle.* Selecting a repeat value of 0 causes the waveform to run continuously or until you press the Stop pushbutton in the Execute Procedure window or by an End Trigger in the process.

CONTROL	DESCRIPTION
Mean Level	Specifies the offset value of the defined amplitude. Select the control parameter first, then the units, and enter the value last. The control parameter determines which units are appropriate.
Amplitude	Specifies the peak-to-peak value of the defined cycle. Select the control parameter first, then the units, and enter the value last. The control parameter determines which units are appropriate.
Phase Lag	Specifies how much the mixed mode sine cyclic command of the selected control channel leads or lags another control channel. Select the phase units (**degrees** or **radians**) in the right entry field. Type the amount of phase into the left entry field. A positive value lags the reference control channel and a negative value leads the reference control channel.

Using the window

Some window selections affect other selections; therefore, complete the selections in the order given:

1. The upper portion of the window defines the mixed mode sine cycle parameters that are *common* for each control channel.
   - Select the following in the order given; a control channel, rate, units, values, and the number of repeats.
2. The lower portion of the window defines mixed mode sine cycle parameters that can be *unique* for each control channel.
   - For each control channel, enter control parameters, units, and values. If phasing of a control channel's signal is desired, enter phase lag units and phase values.

Time and frequency

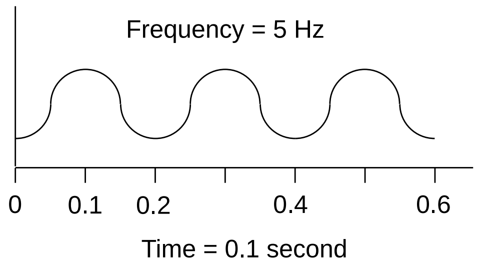
Selecting time allows you to specify the time for a single-segment cycle (half cycle). Selecting frequency allows you to specify the number of cycles per unit of time. A cycle consists of two single-segments.

Time Units
milliseconds
seconds
hours
days

Frequency Units
hertz (Hz)
cycles per second (cps)
cycles per minute (cpm)
cycles per day

Selecting time or frequency changes the rate type value label to Time or Frequency respectively.

- The units for Time are based on a single-segment.
- The units for Frequency are based on a cycle (two segments).
- Both rate types can specify the same waveform using different values



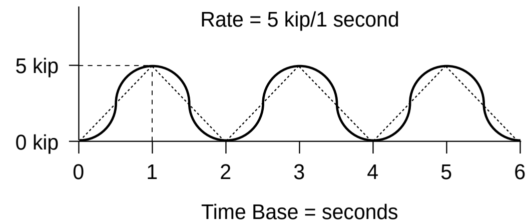
Rate

Selecting rate allows you to specify a segment with a constant rate between end level 1 and end level 2. A rate value represents the amount that the control mode changes in one time unit. Rate is usually associated with a ramp.

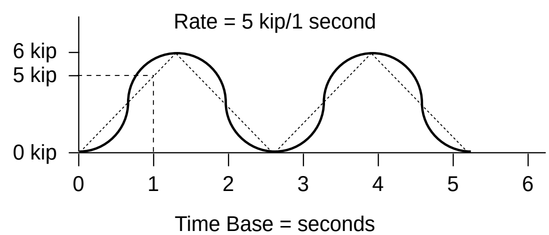
You specify the rate as if a ramp is being programmed. The segment shape of a haversine or step (square wave) is maintained when you use rate.

Selecting rate changes the rate type value label to Rate. Select the units of measurement to the right of the rate type value field.

A rate is expressed as the amount of change per time for a single segment.



The rate is maintained even though the amplitude has changed.

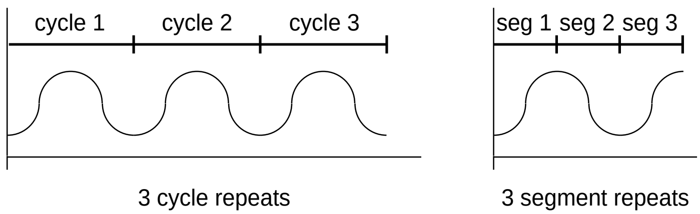


Repeats

Repeating a segment starts at one peak and ends at the other peak of the amplitude parameter. Repeating a full cycle starts and ends at the same peak.

Entering 0 repeats causes the waveform to cycle continuously.

You can repeat cycles or segments. Two segments create one cycle.

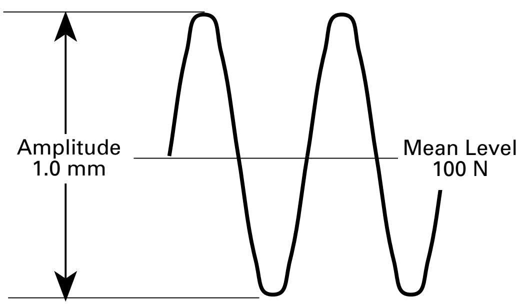


Mean level and amplitude

The mean level is an offset referenced to zero. A positive value (assuming standard phasing) represents tension and a negative value represents compression.

Amplitude is a peak-to-peak value centered on the mean level. The cycle always starts in tension.

Different control parameters (force, length, etc.) can control the amplitude and mean level.



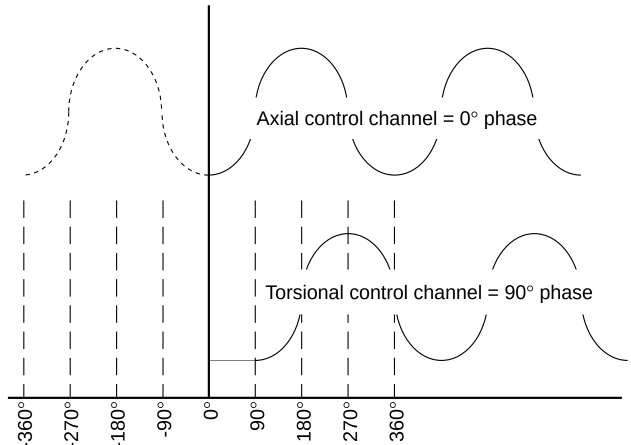
Advanced Function Generation

Phase Lag

The Phase Lag specification is available when more than one control channel is available.

For example, assume two control channels use the same cyclic waveform. The lead control channel starts, while the lagging control channel starts 90° after the lead waveform has started.

One control channel should have a 0° phase; this provides a reference for another channel's phasing.



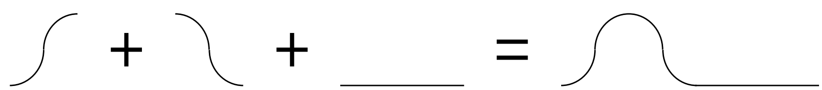
Mixed Mode Pulse Command

A mixed mode pulse cyclic process produces a command signal that controls a servovalve or servomotor. This process should be sequenced in series with other command processes. Use this process in applications where there is a periodic pulse and hold such as rolling wheels, paper processing, or engine valves.

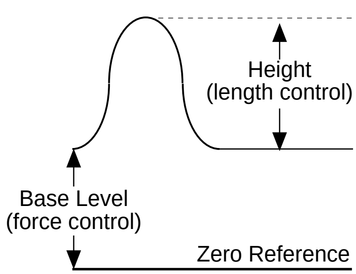
Contents

How it works

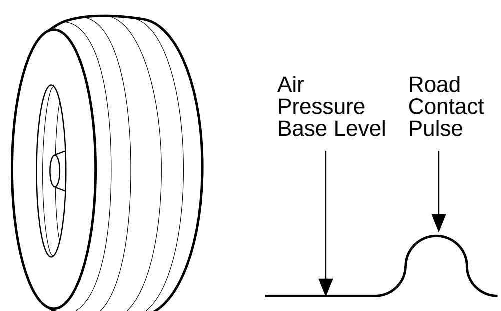
The mixed mode pulse cyclic command is like combining two haversine commands and a hold time command. The difference is that a haversine and hold command contain 3 segments of data where the mixed mode pulse is one segment. A single segment allows the use of FIT compensation (Frequency based iterative technique).



Haversine   Haversine   Hold   Mixed Mode Pulse



One control parameter can be applied to the height (amplitude) of the pulse while a different control parameter can be applied to the base (mean level) of the pulse



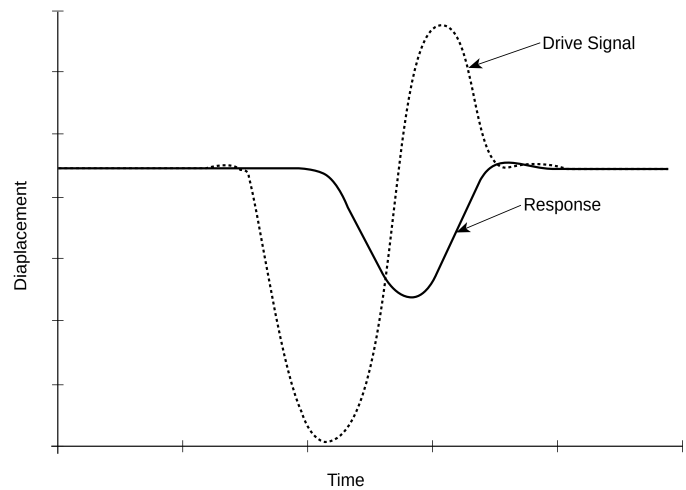
For example, suppose you are testing the sidewall of a tire. The base level parameter represents the air pressure in the tire and the height parameter represents the additional force when the sidewall is nearest to the road.

Using compensation files

The mixed mode pulse command can also use compensation files that allow you to optimize the command each time you run the test. Compensation files contain information about the differences between the command and the actual feedback. This produces a new command signal that achieves the desired system response.

The drive signal to the servovalve may look completely different from the actual response from the system.

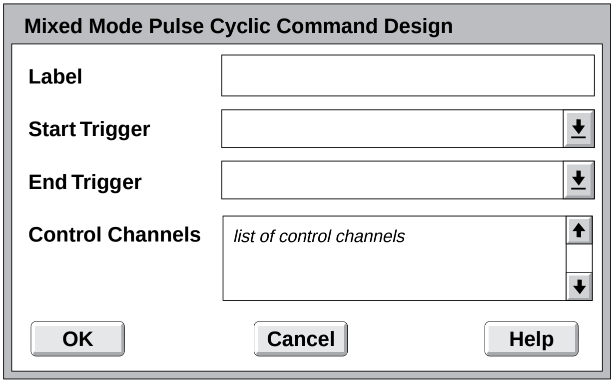
See *FIT Compensation* on page 73 for additional information.



Mixed Mode Pulse Design Window

The information in this window is saved with the test template

This window names the process and specifies how it starts and stops
.



CONTROL	DESCRIPTION
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger
Control Channels	Specifies which control channels the process is applied (more than one may be selected). The control channels are defined by the TestStar configuration file.

Start trigger

- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

Note *Don't try to start more than one command process at a time—if you do, you will cause an error. Sequence test command processes in series.*

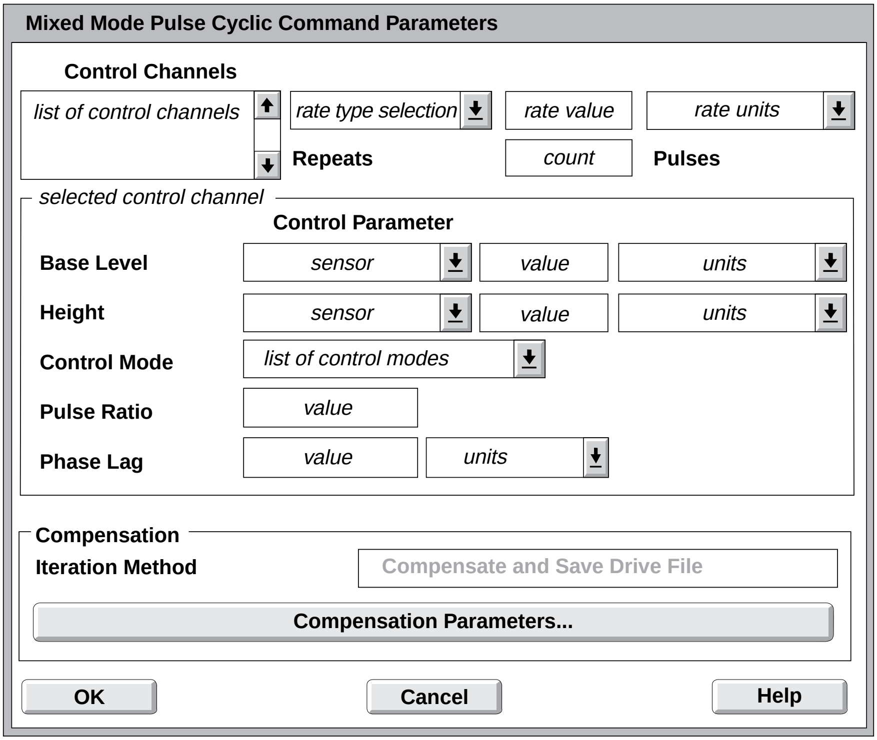
End trigger

- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

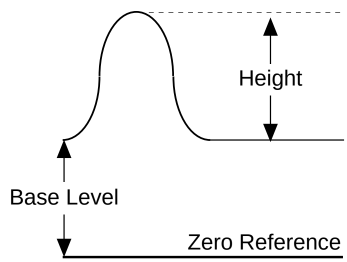
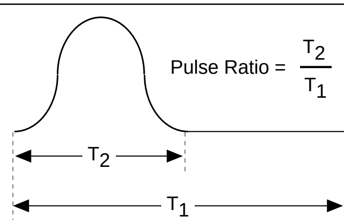
Mixed Mode Pulse Parameters Window

You must establish a mixed mode pulse cyclic process with the Mixed Mode Pulse Cyclic Command Design window before you can use this window.

Use this window to define the specific characteristics of the pulse waveform.



Control	Function
Control Channels	Displays a list of the control channels selected in the design window.
rate type selection	Specifies the type of rate values for the process. The selections are: **Frequency** **Time** **Rate**
rate units	Selects the units for the rate type value. The type of units depends on the type of rate you select.
rate values	Specifies the value you want for the command. Type a value in the middle entry field. *See Computing exact rates* on page 55.

Control	**Function**
Repeats/Pulses	Specifies the number of times the pulse is repeated. Enter the number of pulses you wish to execute. ***Note*** *Due to the FIT algorithm, there may be 1 to 4 extra pulses added to the count. See Computing exact rates on page 55.* Selecting a repeat value of 0 causes the waveform to run continuously.
Height	 Defines the amplitude and the control parameter for the pulse of the cycle.
Base Level	Defines an offset and the control parameter for the hold time of the cycle. The offset is the mean level of the control parameter during the hold time. Select the control parameter in the left entry field, then select the units in the right entry field, and type the units in the middle entry field.
Control Mode	Displays a list of the control modes available for the selected control channel.
Pulse Ratio	 Defines the ratio between the pulse width and the cycle time. The cycle time is established with the rate type selection above.
Phase Lag	Specifies how much the command of the selected control channel leads or lags another control channel. Select the phase units (**degrees** or **radians**) with the right entry field. Type the amount of phase into the left entry field. A positive value lags the reference control channel and a negative value leads the reference control channel.
Compensation	Displays the Compensation Parameters window where you select the iteration method. The default method is No Compensation. After you have made your selection, it appears in the Iteration Method field.

Using the window

The following is a guideline to complete this window: Some window selections affect other selections; therefore, complete the selections in the order given.

1. The upper portion of the window defines the cycle parameters that are *common* for each of the listed control channels.
   - Select the following in the order given; a control channel, rate, units, values, and the number of repeats.
2. The lower portion of the window defines cycle parameters that can be *unique* for each of the listed control channels.
   - For each control channel selected, enter control parameters, control mode, units, and values. If phasing of a control channel's signal is desired, enter phase lag units and phase values.
3. Finally, select the compensation/iteration method.

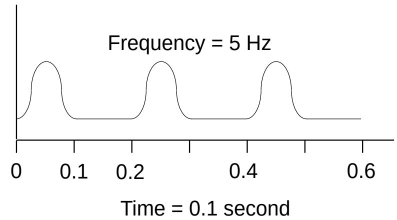
Time and frequency

Selecting time allows you to specify the time for a single-segment cycle. Selecting frequency allows you to specify the number of cycles per unit of time. Both rate types can specify the same waveform using different types of values.

Time Units
milliseconds
seconds
hours
days

Frequency Units
hertz (Hz)
cycles per second (cps)
cycles per minute (cpm)
cycles per day

The units for Time are based on a single segment. The units for Frequency are based on a cycle (two segments).



Advanced Function Generation

Repeats

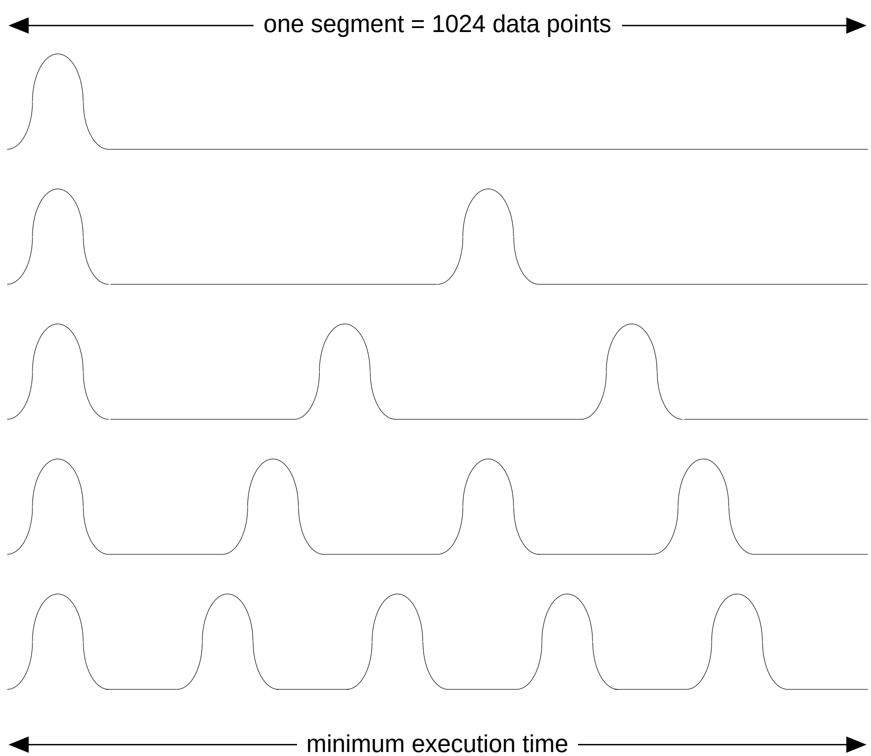
The test will perform *at least* the specified number of pulses. However, up to 4 pulses may be added to the number of repeats. This is caused by the way the segment is created, the frequency of the waveform, and the servo loop update rate.

The waveform is created with 1024 data points. The frequency of the waveform determines the amount of time to execute the segment. The servo loop update rate dictates a minimum execution time to execute a segment. The segment shape may include up to five pulses and hold periods to accomplish the required waveform frequency. This is accomplished by redefining the 1024 data points of the segment shape.

To produce higher frequencies the process changes the shape of the segment

For example, If two pulses define a segment, 1 extra pulse is added when an odd number of repeats is entered.

See *Computing exact rates* on page 55 for additional information.



Additional pulses are more likely to be added with higher pulse rates. This can be checked by clicking in another field the new number, if any, is then displayed.

Control modes

The availability of control mode selections depends on the preselected control parameter used to control pulse height.

If the preselected control parameter has displacement units (expressed in terms of length or angle), you won't be able to make alternate selections. Only the length control mode that the program will use will be displayed in the list box.

If the preselected control parameter has force units (expressed in terms of load or torque), you may select from any of the available force and displacement control modes displayed in the list box.

Note *If you select a force control mode, the system will control in force and seek force end-levels (this is 'true' force control, vs. controlling in displacement and seeking force end-levels).*

Computing exact rates

Your servo loop update rate dictates a discrete number of rate values. This is based on the minimum execution time of a segment and the number of pulses in a single segment.

Use the following formulas to determine the minimum execution time of a segment or to determine a new servo loop update rate.

For update rates >2500 Hz

$$T_{min} = \frac{numberofdatapoints}{\frac{f_{servoloop}}{2}}$$

For update rates ≤2500 Hz

$$T_{min} = \frac{numberofdatapoints}{f_{servoloop}}$$

Where:

- $f_{servoloop}$ is the frequency of the servo loop update rate.
- The number of data points that defines a segment is 1024.
- T_{min} is the minimum execution time in seconds

To check or change the servo loop update rate check the TestStar Performance Rate window. Activate the TestStar main window and type **cntl + u** or press the Hz button in the toolbar.

See the *Servo Loop Update Rate* topic in Appendix C of the TestStar Reference manual for more information

For example, suppose the servo loop update rate is 5 Hz and you want a repeat rate of 90 ms. When you enter a rate of 90 ms, the rate value changes to 81.92 ms (when you click on a different entry field). The chart shows that 5 pulses per segment are needed to obtain the rate. Use the following chart as a guide in setting up rate values.

N	Pulses per Segment	Repeat Rates (in sec) for Update Rates (in Hz) 5 kHz	4 kHz	3 kHz	2.5 kHz
1	**5**	0.4067	0.1024	0.1365	0.0819
	4	0.1024	0.1280	0.1707	0.1024
	3	0.1365	0.1707	0.2276	0.1365
	2	0.2048	0.2560	0.3413	0.2048
	1	0.4096	0.5120	0.6827	0.4096
2	**5**	0.1638	0.2048	0.2731	0.1638
	4	0.2048	0.2560	0.3413	0.2048
	3	0.2731	0.3414	0.4551	0.2731
	2	Identical to n=1 and PPS=1			
	1	0.8192	1.0240	1.3653	0.8192
3	**5**	0.2458	0.3072	0.4096	0.2458
	4	0.3072	0.3840	0.5120	0.3072
	3	Identical to n=1 and PPS=1			
	2	0.6144	0.7680	1.0240	0.6144
	1	1.2288	1.5360	2.0480	1.2288

$$T_1 = n\left(\frac{T_{min}}{P}\right)$$

T_1 is the execution time for each pulse.

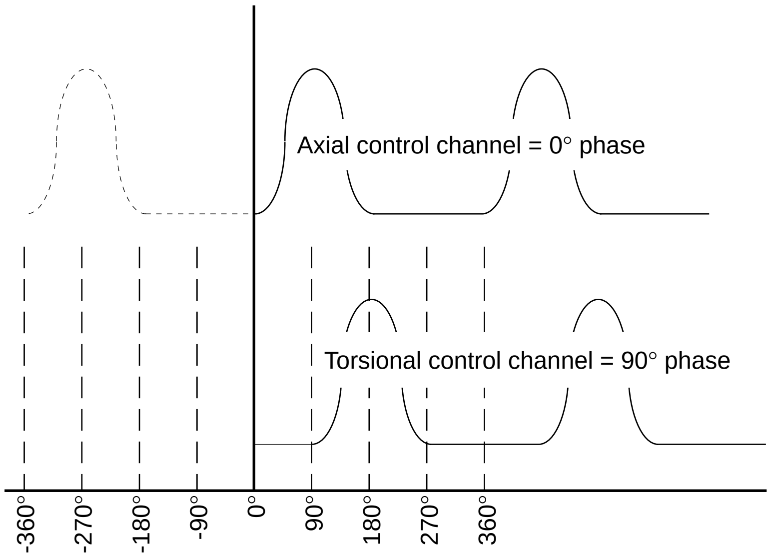
P can be 1 - 5

T_{min} is the minimum execution time in seconds.

Phase lag

The Phase Lag specification is available when the cyclic command is applied to more than one control channel.

For example, suppose two control channels use the same cyclic waveform. The axial control channel starts the waveform, then the torsional control channel starts the waveform 90° after the axial waveform has started. To get this effect, enter 0 in the axial Phase Lag field and 90 in the torsional Phase Lag field.



Compensation

Because system response is a function of the system's electromechanical response and of the specimen's response, you should consider creating the compensation files whenever there is a significant change in the specimen's characteristics.

See *FIT Compensation* on page 73 for information about setting up and using compensation files.

Changing parameters after running iterations

Once you have run and saved the ITF and Drive files (running iterations), changing any values in this window affects these files as shown below.

Parameter	ITF File	Drive File
Rate type/rate values/ rate units	No longer valid	No longer valid
Base Level		
Control Parameter (example: force to length)	No longer valid	No longer valid
Values and Units	Still valid	See Note below
Height		
Control Parameter	No longer valid	No longer valid
Values and Units	Still valid	See Note below
Pulse Ratio	Still valid	No longer valid
Phase Lag	Still valid	No longer valid
Control Mode	No longer valid	No longer valid
*PIDF Values (tuning)	No longer valid	No longer valid

* An error is displayed when incompatible files are detected except for PIDF tuning changes. Also, tuning values not equal to those present when the file was created may cause your system to be unstable or sluggish.

Note *Values and Units can be changed to equivalent dimensions.*

Examples: 1000 N = 1 kN (still valid)

1000 N ≠ 1000 kN (not valid)

1000 N ≠ 2000 N (not valid)

UDA Cyclic Command

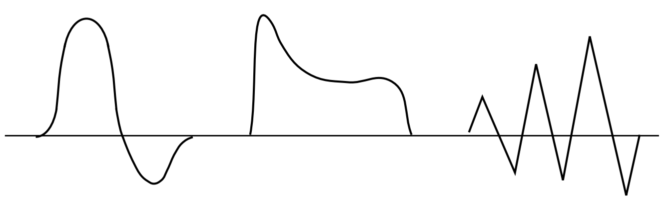
A UDA (user defined arbitrary) cyclic process produces a command signal that controls a servovalve or servomotor. This process should be sequenced in series with other command processes.

Use this process in testing applications where a non-symmetrical waveform is needed such as landing gear, shoes, or a tire sidewall.

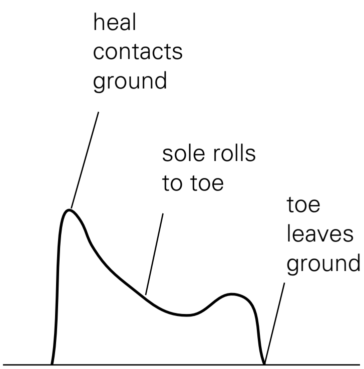
Contents

How it works

The UDA cyclic command is like a file playback process except that it defines a single segment with an arbitrary shape. The user defined segment can be repeated. A haversine segment is similar to the UDA process, it defines a shape using a series of data points.



Here are three arbitrary shapes that can be defined with shape files for the UDA Cyclic Command

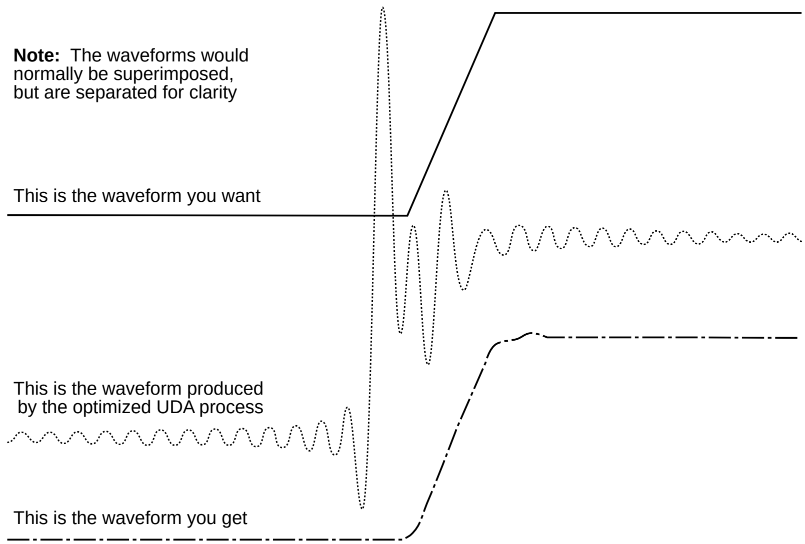


For example, suppose you are testing a running shoe. You could create a segment shape that simulates the force applied to the shoe as the heel contacts the ground, the sole rolls toward the toe, and the toe leaves the ground. The segment shape can be repeated to simulate a marathon.

Using compensation files

The user defined shape can also use compensation files that allow you to optimize the command each time you run the process. Compensation files contain information about the differences between the command and the actual feedback. This produces a new command file (drive file) that achieves the desired response.

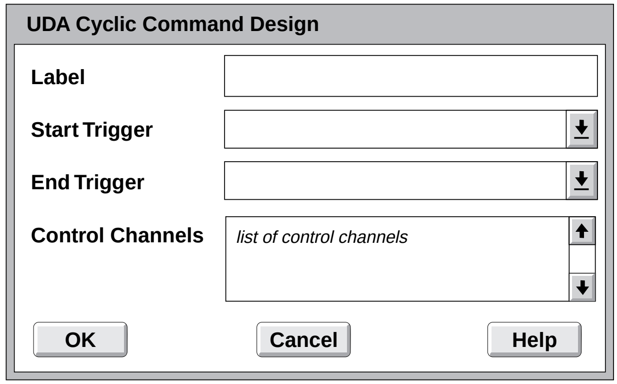
The drive signal to the servovalve may look completely different from the actual response from the system.



UDA Cyclic Command Design Window

The information in this window is saved with the test template

This window names the process and specifies how it starts and stops.
.



CONTROL	DESCRIPTION
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger
Control Channels	Specifies which control channels the process is applied (more than one may be selected). The control channels are defined by the TestStar configuration file.

Start trigger

- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

Note *Don't try to start more than one command process at a time—if you do, you will cause an error. Sequence test command processes in series.*

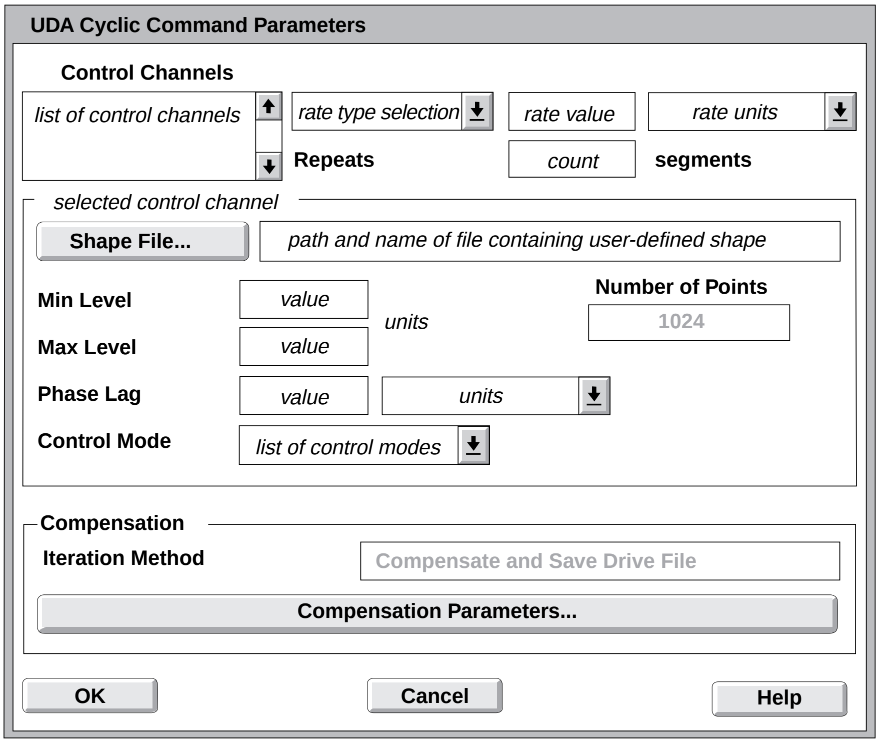
End trigger

- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

UDA Cyclic Command Parameters Window

You must establish a UDA cyclic process with the UDA Cyclic Command Design window before you can use this window. The information in this window is saved with the test procedure.

Use this window to define the specific characteristics of the cyclic waveform.



Control	Function
Control Channels	Displays a list of the control channels selected in the design window.
rate type selection	Specifies the type of rate values for the process. The selections are: **Frequency** **Time** **Rate**
rate units	Selects the units for the rate type value. The type of units depends on the type of rate you select.
rate values	Specifies the value you want for the command. Type a value in the middle entry field. *See Computing exact rates* on page 67.
Repeats/ Segment	Specifies the number of times the segment is repeated. Enter the number of segments you wish to execute. Selecting a repeat value of 0 causes the waveform to run continuously or until you press the Stop pushbutton in the Execute Procedure window or by an End Trigger in the process.
Shape File	Opens the Select or Enter a UDA Shape-File Name window. After you have selected the desired file, the file's path and name are displayed next to the pushbutton. Before you can use this function, you must have created a file that defines the segment shape. See *Creating Shape Files* on page 70 to define your segment shape.
Min Level **Max Level**	Displays the predefined waveshape's minimum and maximum levels as read in from the shape file. The units specified for the selected rate are shown to the right of the values.
Number of Points	Displays the number of points that define the segment shape. Any shape file that contains less than 1024 points disables compensation function. (No file iterations can be made).
Phase Lag	Specifies how much the command of the selected control channel leads or lags another control channel. Select the phase units (**degrees** or **radians**) with the right entry field. Type the amount of phase into the left entry field. A positive value lags the reference control channel and a negative value leads the reference control channel.
Control Mode	Displays a list of the control modes available for the selected control channel.
Compensation	Displays the Compensation Parameters window where you select the iteration method. The default method is No Compensation. After you have made your selection, it appears in the Iteration Method field.

Using the window

The following is a guideline to complete this window: Some window selections affect other selections; therefore, complete the selections in the order given.

1. The upper portion of the window defines the cycle parameters that are *common* for each of the listed control channels.

   - Select the following in the order given; a control channel, rate, units, values, and the number of repeats.

Note *When you enter a rate value, the process may modify the value slightly. This is caused by the relationship of the servo loop update rate and the number of data points that define the segment shape.*

2. The lower portion of the window defines cycle parameters that can be unique for each of the listed control channels.

   - For each control channel, select a shape file and control mode. If phasing of a control channel's signal is desired, enter phase lag units and phase values.

3. If needed, select the compensation/iteration method.

Time and frequency

Selecting time allows you to specify the time for the single-segment cycle. Selecting frequency allows you to specify the number of cycles per unit of time. Both rate types can specify the same waveform using different types of values.

Time Units
milliseconds
seconds
hours
days

Frequency Units
hertz (Hz)
cycles per second (cps)
cycles per minute (cpm)
cycles per day

The UDA process has a minimum cycle time (maximum frequency). This restriction is caused by the servo loop update rate. If you enter a time below the minimum cycle time, the process will change the value to the minimum cycle time.

Shape file

The shape file defines the segment shape with a series of end levels. The series of end levels work like a series of monotonic ramp commands. The shape file also specifies a dimension and units. The dimension determines which control mode is assigned to the process.

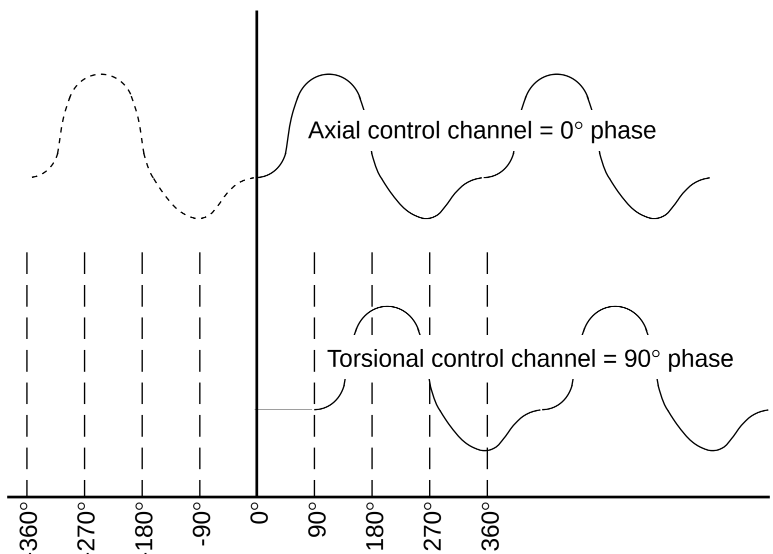
For example, suppose that the dimension length is specified in the shape file for the axial channel. The process compares the dimension *length* to the control modes of the axial channel. If more than one length control mode exists, the process selects the first one it detects.

Note *See Creating Shape Files on page 70 for additional information.*

Phase lag

The Phase Lag specification is available when the cyclic command is to be applied to more than one control channel.

For example, suppose two control channels use the same cyclic waveform. The axial control channel starts the waveform, then the torsional control channel starts the waveform 90° after the axial waveform has started. To get this effect, enter 0 in the axial Phase Lag field and 90 in the torsional Phase Lag field.



Computing exact rates

Your servo loop update rate dictates a discrete number of rate values. This is based on the minimum execution time of a segment and the number of pulses in a single segment.

Use the following formulas to determine the minimum execution time of a segment or to determine a new servo loop update rate.

For update rates >2500 Hz

$$T_{min} = \frac{numberofdatapoints}{\frac{f_{servoloop}}{2}}$$

For update rates ≤2500 Hz

$$T_{min} = \frac{numberofdatapoints}{f_{servoloop}}$$

Where:

- $f_{servoloop}$ is the frequency of the servo loop update rate.
- The number of data points that defines a segment is 1024.
- T_{min} is the minimum execution time in seconds

To check or change the servo loop update rate check the TestStar Performance Rate window. Activate the TestStar main window and type **cntl + u** or press the Hz button in the toolbar. See the *Servo Loop Update Rate* topic in Appendix C of the TestStar Reference manual for more information

For example, assume the servo loop update rate is 5 Hz and you want a repeat rate of 500 ms. When you enter a rate of 500 ms, the rate value changes to 0.4096 (when you click on a different entry field). Use the following chart as a guide in determining discrete rate values.

N	Pulses per	Repeat Rates (in sec) for Update Rates (in Hz)			
	Segment	5 kHz	4 kHz	3 kHz	2.5 kHz
1	**5**	0.4096	0.4096	0.6827	0.4096
	4	0.8192	0.8192	1.3650	0.8192
	3	1.2288	1.2288	2.0480	1.2288
	2	1.6384	1.6384	2.7307	1.6384
	1	0.4096	0.4096	0.6827	0.4096

$$T_1 = n\left(\frac{T_{min}}{P}\right)$$

T_1 is the execution time for each pulse.

P can be 1 - 5 pulses per segment

T_{min} is the minimum execution time in seconds.

Compensation

Because system response is a function of the system's electromechanical response *and* of the specimen's response, you should consider creating or modifying these files whenever there is a significant change in the specimen's characteristics.

See *FIT Compensation* on page 73 for additional information.

Changing parameters after running iterations

Once you have run and saved the ITF and Drive files (running iterations), changing any values in this window affects these files as shown below.

PARAMETER	ITF FILE	DRIVE FILE
Rate type/rate values/ rate units	No longer valid	No longer valid
Shape File	No longer valid	No longer valid
Phase Lag	Still valid	No longer valid
Control Mode	No longer valid	No longer valid
*PIDF Values (tuning)	No longer valid	No longer valid

* An error is displayed when incompatible files are detected except for PIDF tuning changes. Also, tuning values not equal to those present when the file was created may cause your system to be unstable or sluggish.

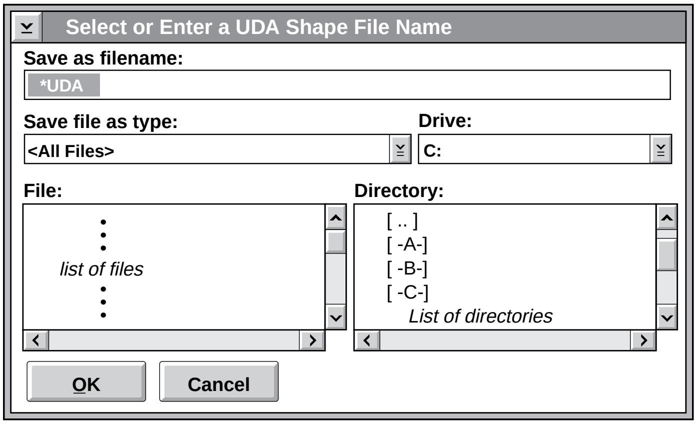
Note *Values and Units can be changed to equivalent dimensions.*

Examples: 1000 N = 1 kN (still valid)

1000 N ≠ 2000 N (not valid)

Select or Enter a UDA Shape-File Name

Use this window to select waveform files to be used with UDA cyclic commands.



CONTROL	FUNCTION
Save as filename:	Displays ***.UDA** in the entry field when the window appears. The file name ***.UDA** causes all files with the **.UDA** extension to be shown in the Files list. To save a UDA file, type the name you want in the entry field.
Save file as type:	This is not used.
Drive:	Lists the drives available to your computer.
Files:	Lists the UDA files in the current directory. To open a file, click on its name to select it and press OK. This file name and its path will appear in the UDA Cyclic Command Parameters window.
Directory:	Lists the available directories. Double-clicking a different directory displays the new path shown next to the current directory label.

Creating Shape Files

The UDA process uses an ASCII file to function like a series of monotonic command processes. An ASCII file for the UDA shape requires the following attributes:

- a file name with a .UDA extension
- a dimension and unit specification
- end level data

Creating the file

A shape file is an ASCII file that contains data in a column-oriented syntax. You may want to rename the shape file to include the extension .UDA so the file is easier to find when using the Shape File button in the parameters window.

The shape file can be created using one of the following tools:

- a traditional text editor
- a spreadsheet application
- extracted from a data base
- a custom program written to expressly generate the file

Each data entry must be separated by a line feed or cartridge return. If you use a spreadsheet or data base application, be sure to set it up so the cell information is delimited by a space or tabs when you save it as a text file.

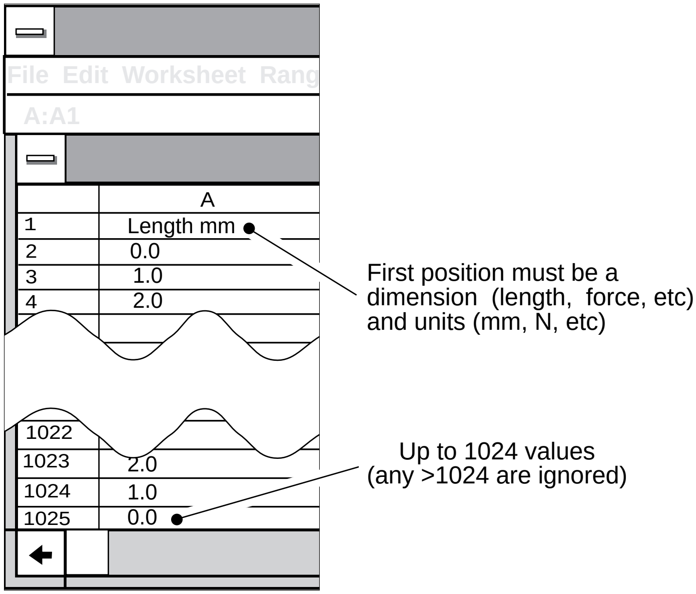
File format

All information in a shape file is in a column-oriented format. The first line must define a dimension and appropriate units (separated by a space). The remaining lines contain a single value that represents an end level.

Note *The ideal number of data points is 1024. Shape files with less than 1024 data points cannot be optimized using file iterations. The process will accept the first 1024 data points from shape files with more than 1024 data points.*

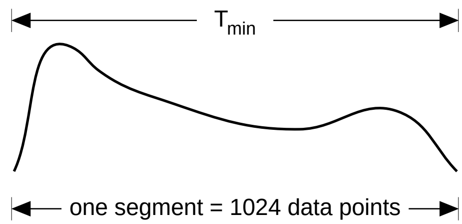
Use a spreadsheet, text editor, or any ASCII file to define a segment shape.

Use exactly 1024 data points to define a segment shape so you can utilize FIT compensation.



Design tips

- A UDA shape file should have 1024 data points to define the segment shape.
- The first end level value and the last end level value should be different to avoid a momentary hold when the segment is repeated (unless a hold time is desired).



- The UDA process has a minimum cycle time (maximum frequency). Determine the minimum cycle time so you know the limits of the segment.

Use the following formulas to determine the minimum execution time of a segment or to determine a new servo loop update rate.

For update rates >2500 Hz

$$T_{min} = \frac{numberofdatapoints}{\frac{f_{servoloop}}{2}}$$

For update rates ≤2500 Hz

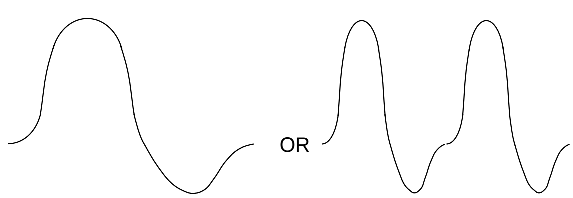
$$T_{min} = \frac{numberofdatapoints}{f_{servoloop}}$$

Where:

- $f_{servoloop}$ is the frequency of the servo loop update rate.
- The number of data points that defines a segment is 1024.
- T_{min} is the minimum execution time in seconds

- If your test requires a faster cycle time, design the segment with two identical shapes.

Both segments contain 1024 data points.



FIT Compensation

FIT is a frequency based iterative technique that dynamically optimizes the performance of your system. The mixed mode pulse cyclic command and the UDA cyclic command processes use FIT compensation.

Contents

How it works

Two compensation files work together to dynamically control system response, an ITF (Inverse Transfer Function) file and a Drive file.

- The ITF file contains information about the response of the system.
- The Drive file contains information to produce the desired waveshape.

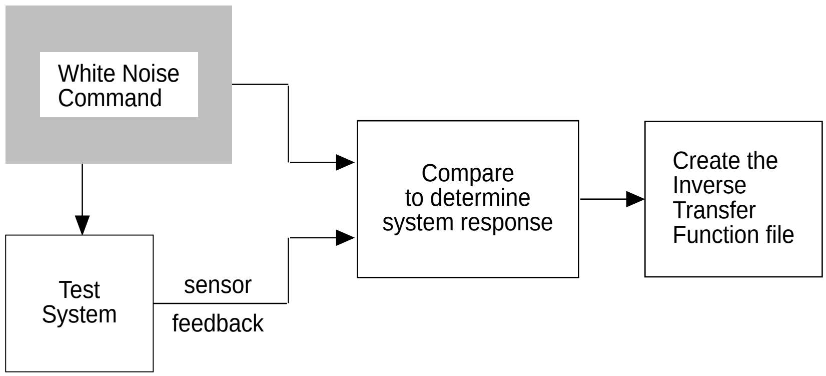
The process is controlled by how you set up the Iteration Method and Compensation Parameters window. The process works like this:

1. The program sends a command signal to the system.
2. The sensor response to this command indicates how the system as a whole (the electronics, the hydraulics, and the specimen itself) reacts to the command.
3. The program analyzes the difference between the response and the command signal. The program attempts to close the difference by modifying the command signal.
4. This process continues, with each iteration, modifying the command again. The iterations compensate the response so that it approaches the desired pulse or arbitrary shape.

ITF files

When you create a new ITF file, the program sends a white noise command to the test system and analyzes the feedback to characterize the system response. This analysis detects roll-off with increasing frequency and phase shifting between the command and feedback. The ITF file contains correction factors for amplitude and phase at discrete frequencies.

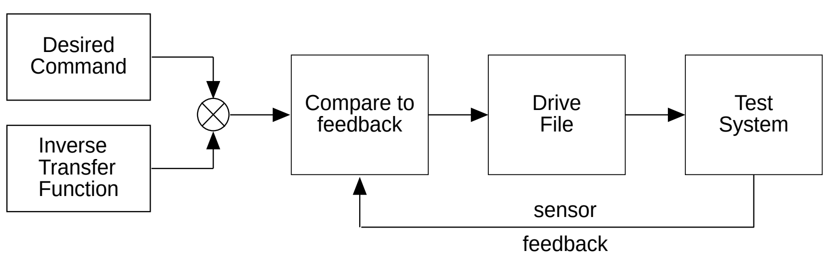
The sensor feedback and white noise command are compared and analyzed (by the FIT algorithm) to find the system inverse transfer function.



Drive files

After the white noise creates the inverse transfer function, it is applied to the desired command. This produces a drive file that contains 1024 control values that correspond to the 1024 segment data points.

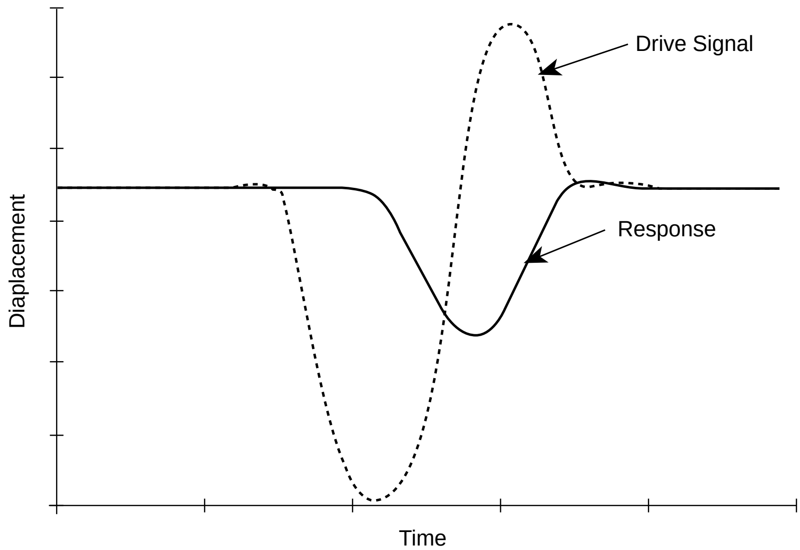
Each time the segment command is run, the system feedback is analyzed and the drive file is fine tuned. The more iterations you permit, the more accurately the system responds to the cyclic command.



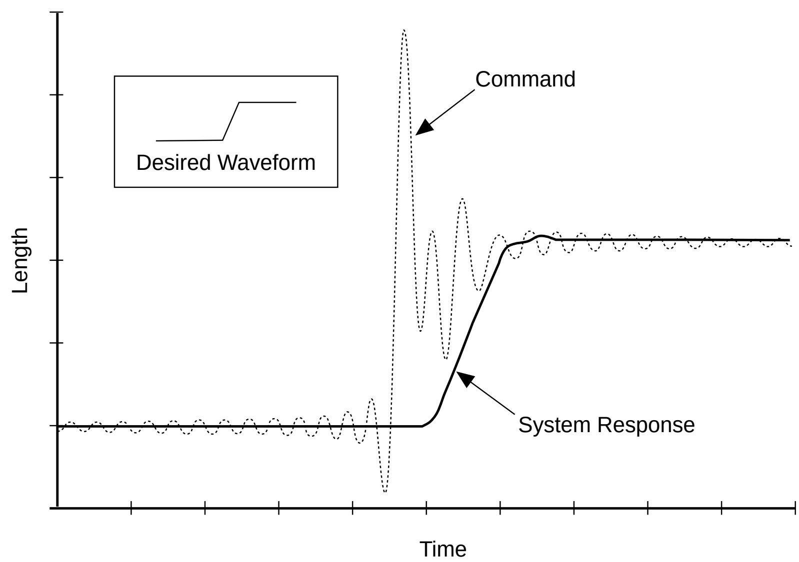
Drive files

The following examples show how an optimized drive file creates a command that achieves the desired system response.

In this mixed mode pulse example, the drive signal to the servovalve looks completely different from the actual system response.



In this UDA command example, you can see the difference between the command, the drive signal, and the feedback.

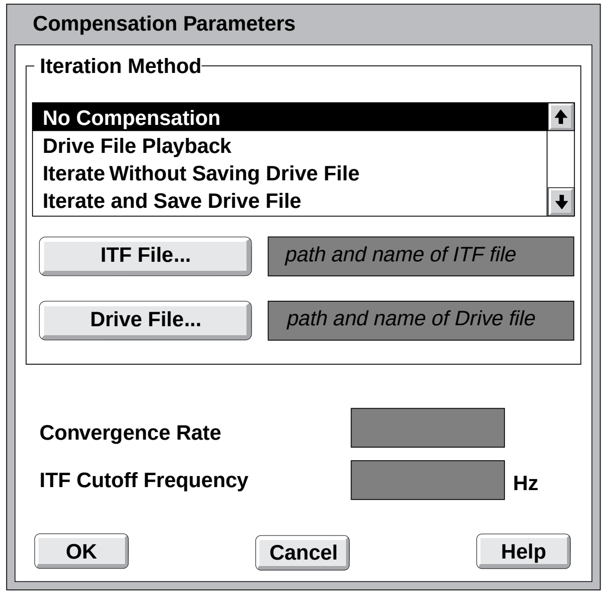


Compensation Parameters Window

This window accesses the ITF and drive files that you have created. To create a new file, simply use a new file name.

Use the window to select ITF and Drive files for your test.

You can also set up how the iterations will be run.

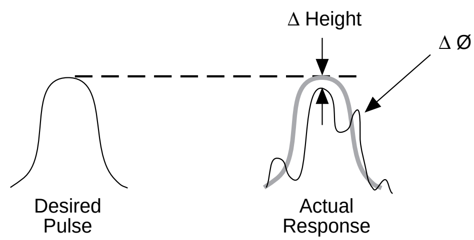


CONTROL	FUNCTION
Iteration Method	Selects the iteration method to be used for the selected file:
No Compensation	There are no iterations. The pulse is controlled solely by the PIDF tuning already set up in the system.
Drive File Playback	A previously created drive file is used without any additional iterations.
Iterate Without Saving Drive File	Iterations continue for the number of repeats specified. The iteration computations from this run are not saved.
Iterate and Save Drive File	Iterations continue for the number of repeats specified. The iteration computations from this run are saved. The Drive file is updated to contain the new values.
ITF File... **Drive File...**	If the pushbutton is enabled, pressing it displays the Select or Enter an ITF (or Drive) File window. The files enabled depend upon the iteration method you have selected The choices are as follows: Selection / ITF File (enabled) / Drive File (enabled) No Compensation / No / No Drive File Playback / No / Yes Iterate w/o Save / Yes / No Iterate with Save / Yes / Yes After the file(s) have been selected, the field next to the pushbutton displays the file's path.
Convergence Rate	Specifies the ratio of the error between the feedback and the desired pulse that is used to compensate the drive. Default value is 0.4.
ITF Cutoff Frequency	Defines the ITF frequency range used to compensate the drive during the iterations. Default value is 60 Hz.

Convergence rate

This value determines how much of the error between the desired pulse and the feedback will be used to modify the command signal during the iterations. The lower this number, the more iterations it will take to achieve the desired response.

This illustration shows that the application has found a height error (ΔHeight) and a phase error (ΔØ). In correcting the error, the application computes a new drive file.

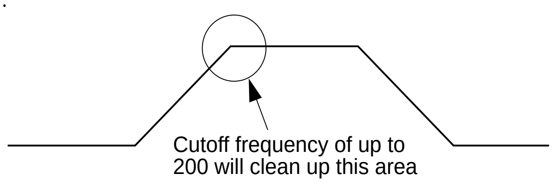


Start the test using the default value. Increase it (Convergence›1) if convergence is too slow. A convergence that is too high may cause the system to become unstable.

ITF cutoff frequency

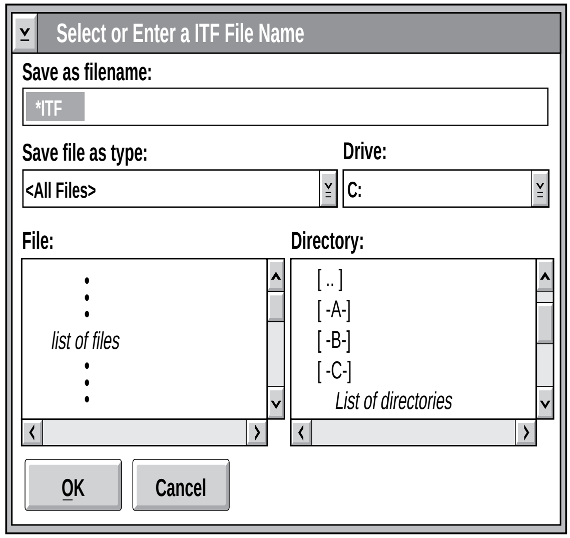
Use the default value of 60 Hz for normal testing. For best results, keep the following considerations in mind:

- If you have a cutoff frequency that is too high, the application may amplify any high-frequency noise. This can cause the response to contain erroneous high-frequency components, which can lead to instability.
- For normal pulse wave forms, one rule-of-thumb is to select a value that is four times the effective frequency or 60 Hz, whichever is greater. *For example:* Consider a 1 Hz pulse with a ratio of 0.05. The *effective* frequency of this pulse itself is 20 Hz. Use a cutoff frequency of 80 Hz.
- When you are dealing with shapes that are more like square waves than haversines, setting a high cutoff frequency will help get sharp edges. Make sure to lower the number if the system becomes unstable.



Select or Enter an ITF-File Name Window

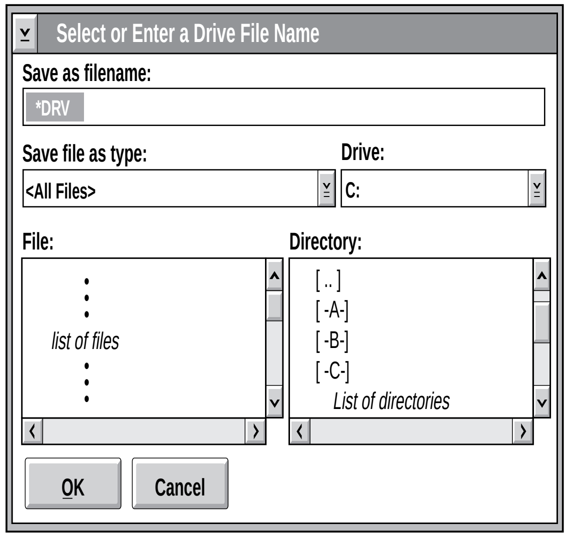
Use this window to save or select ITF files to be used with mixed mode pulse cyclic commands.



CONTROL	FUNCTION
Save as filename:	Displays ***.ITF** in the entry field when the window appears. The file name ***.ITF** causes all files with the **.ITF** extension to be shown in the Files list. To save a UDA file, type the name you want in the entry field.
Save file as type:	This is not used.
Drive:	Lists the drives available to your computer.
Files:	Lists the UDA files in the current directory. To open a file, click on its name to select it and press OK. This file name and its path will appear in the UDA Cyclic Command Parameters window.
Directory:	Lists the available directories. Double-clicking a different directory displays the new path shown next to the current directory label.

Select or Enter a Drive-File Name Window

Use this window to save or select Drive files to be used with mixed mode pulse cyclic commands.



CONTROL	FUNCTION
Save as filename:	Displays ***.DRV** in the entry field when the window appears. The file name ***.DRV** causes all files with the **.DRV** extension to be shown in the Files list. To save a UDA file, type the name you want in the entry field.
Save file as type:	This is not used.
Drive:	Lists the drives available to your computer.
Files:	Lists the UDA files in the current directory. To open a file, click on its name to select it and press OK. This file name and its path will appear in the UDA Cyclic Command Parameters window.
Directory:	Lists the available directories. Double-clicking a different directory displays the new path shown next to the current directory label.

Chapter 4

RPC™ Utilities for TestStar

This chapter describes the Model 790.15 RPC™ Utilities for TestStar. The first of these utilities is the RPC File Playback process for TestWare-SX™.

This process is similar to the standard File Playback process except that it is designed to play only RPC™ files. This process supports the RPC II™, RPC III™, and Component RPC™ formats, which are specific file formats. It also supports RPC III response data acquisition when an RPC III drive file is played.

Prerequisites

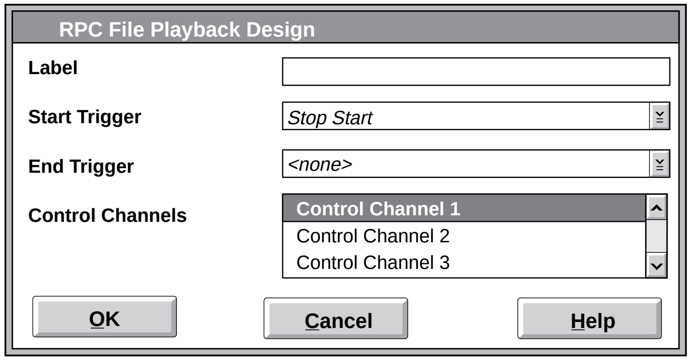
This process requires that you have a RPC package so appropriate files can be created for playback. You must be familiar with the RPC file you intend to use. You must know how each RPC channel is intended to be used.

Contents

RPC File Playback Design Window

The information in this window is saved with the test template

This window names the process and specifies how it starts and stops.



Control	Description
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger
Control Channels	Specifies which control channels the process is applied (more than one may be selected). The control channels are defined by the TestStar configuration file.

Start trigger

- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

Note *Don't try to start more than one command process at a time—if you do, you will cause an error. Sequence test command processes in series.*

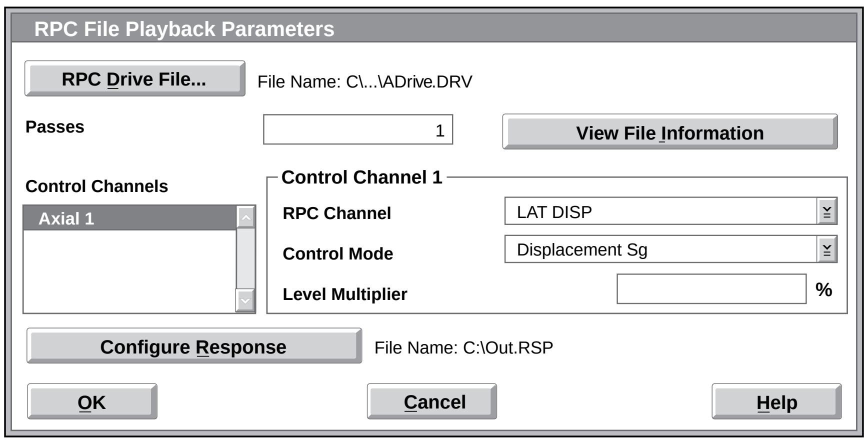
End trigger

- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

RPC File Playback Parameters Window

You must create an RPC file playback process with the RPC File Playback Design window before you can access this window. The information in this window is saved with the test procedure.

Use this window to select a playback file and map the RPC channels and control modes. This window defines the RPC drive file and control channel mapping of the RPC channels to the TestStar channels. When the drive file is a RPC III file, you may also specify response file parameters.



When an RPC III drive file is selected, the Configure Response button is enabled; otherwise, it is disabled. If a response file has not been defined, the response file path is labeled <none>.

Control	Function
RPC drive file...	Displays the RPC Drive File window to select a RPC drive file. The file may be a Component RPC, RPC II or RPC III format. When a file is selected its file name and complete path are shown next to the RPC drive file button.
Passes	Type the number of times the file is repeated. Enter 0 for continuous repeats. The passes field is disabled and set to 1 when a response file is defined. See the *Configure Response* button bellow.
View/Hide File Information	Lists the header information from the Selected Drive file (e.g., type, number of points, sample rate, channel information, full scale, units, etc.)
Control Channels	Lists the TestStar control channels that will be mapped to RPC channels. You must assign an RPC channel to each TestStar control channel.
RPC Channel	Assigns one of the RPC channels of data contained in the selected drive file to the selected TestStar control channel. For each TestStar control channel, you must select a RPC channel. There may be more RPC channels than TestStar control channels—you need to know how each RPC channel should be used.
Control Mode	Selects a control mode for the RPC channel. The selected RPC channel determines which control modes are available. Modes using force feedback are listed when the RPC channel data is force data. Length modes are listed when the RPC channel data is length, etc.
Level Multiplier	Multiplies the RPC time history data prior to playing out the end levels. The value must be greater than 0% and less than 100%.
Configure Response	Displays the Configure Response window where TestStar input channels are mapped to RPC inputs, and the response file name is specified. This button is disabled when the selected drive file is not an RPC III file format.

Using the window

Perform the following to complete RPC file playback parameters window:

1. Press the RPC drive file pushbutton to display the RPC Drive File selection window.
2. Locate the RPC file and press OK.
3. Press the View File Information pushbutton view information about the RPC file.
4. Enter the number of times you want the file run in the Passes entry field.
5. Select a TestStar control channel.
6. Select an RPC channel and a control mode.
7. Repeat steps 5 and 6 for each control channel.

File information

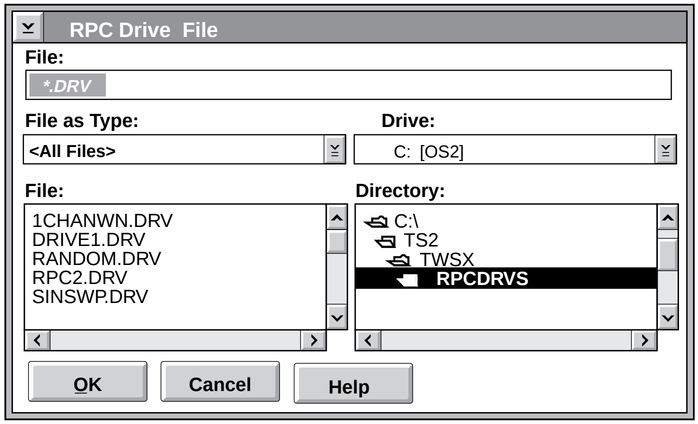
Pressing the View File Information pushbutton displays a window with the RPC file header information shown. It shows the RPC file type, the number of points, the time per point, and information about each channel.

You may wish to view the Channel Information to help decide which RPC channel should assigned to each TestStar control channel.

RPC Drive File Window

This window selects the RPC File for the parameters window.

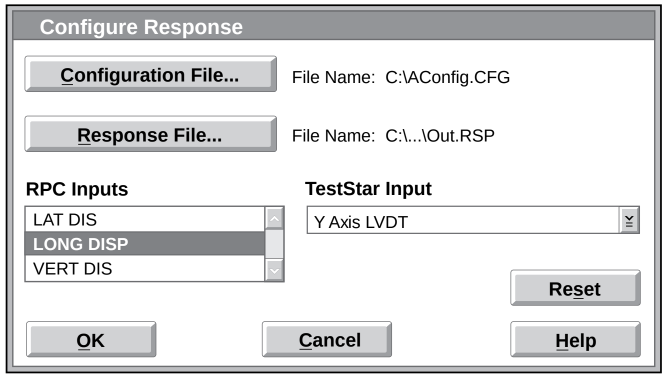
Use this window to select an RPC drive file to play.



Control	Function
File:	Displays the currently selected drive file. The entry "***.DRV**" causes all files with the.DRV extension to be shown in the Files list.
File as Type:	This control is not used.
Drive	Displays the current drive. When you select a different drive, the directory list is updated.
File	Lists the playback files in the current directory.
Directory	Lists the available directories. Double-clicking a different directory displays the new path next to the current directory label.

Configure Response Window

This window specifies the RPC III configuration file, RPC III response file and defines the mapping of the RPC input channels to TestStar input channels. You use this window when you need to sample data channels for the purpose of doing iteration.



Prerequisite

You must select an RPC III file in the parameters window to enable the Configure Response button. Pressing the Configure Response button in the parameters window opens this window.

Control	Function
Configuration File…	Opens the RPC Configuration File window where you select a RPC III configuration file. The file must have been created using RPC application software running on a VAX (then transferred to the PC). The configuration file contains the RPC inputs definition.
Response File…	Opens the RPC Response File window where you name a RPC III response file. This file is created by this process when the process executes and it contains the input channel data, as defined in the configuration file.
RPC Inputs	Lists the RPC input channel names contained in the Configuration file. One TestStar input must be mapped to each RPC input in the list.
TestStar Input	Lists the possible TestStar inputs that can be mapped to the selected RPC input. Inputs of the same dimension as the RPC input channel are listed. The same TestStar input cannot be more than one RPC input.
Reset	Resets the currently specified configuration (CFG) file and response (RSP) file. The filenames beside each button is set to <none>. This in effect turns off Response data acquisition.

Using the window

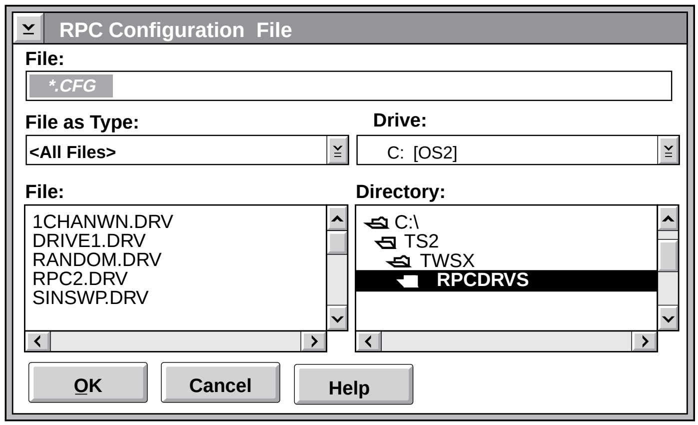
This window opens an RPC configuration file and creates a response file. It also maps the RPC input channels to the TestStar channels.

1. Press the Configuration File… button to open the RPC Configuration File window. Select a configuration file and press OK.
2. Press the Response File… button to open the RPC Response File window. Type a name for the file in the File entry field and press OK.
3. Select an RPC input. Then select a TestStar channel. This assigns an RPC channel to a specific TestStar channel.
4. Repeat Step 3 for each RPC input listed.
5. Press OK when you are done.

Configuration File Window

This Configuration File window specifies the RPC III configuration file and defines the mapping of the RPC channels to TestStar input channels.

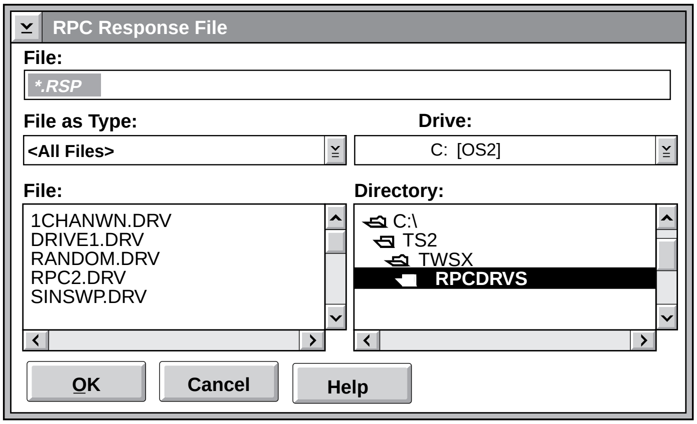
Use this window to open up a current configuration file.



CONTROL	FUNCTION
Open Filename	Displays the file name in the entry field when the window appears. The entry "***.CFG**" causes all files with the same CFG to be shown in the Files list. Click OK to select the current configuration file (e.g., AConfig.CFG).
Type of File	This control is not used.
Drive	Displays the current drive. When you select a different drive, the directory list is updated.
File	Lists the configuration files in the current directory.
Directory	Lists the available directories. Double-clicking a different directory displays the new path next to the current directory label.

Response File Window

This Response File window specifies the RPC III response file and defines the mapping of the RPC channels to TestStar input channels. This window is used to save the file to a filename, file type, and current directory.



CONTROL	FUNCTION
Open Filename	Displays the file name in the entry field when the window appears. The entry "***.RSP**" causes all files with the same RSP to be shown in the Files list. Click OK to select the current response file (e.g., Out.RSP).
Type of File	This control is not used.
Drive	Displays the current drive. When you select a different drive, the directory list is updated.
File	Lists the response files in the current directory.
Directory	Lists the available directories. Double-clicking a different directory displays the new path next to the current directory label.

Chapter 5

Advanced Data Acquisition

This chapter describes Model 790.16 Advanced Data Acquisition TestWare. Advanced Data Acquisition TestWare contains the High Speed Data Acquisition process. This is an optional process that is added to the Data Collection process in TestWare-SX.

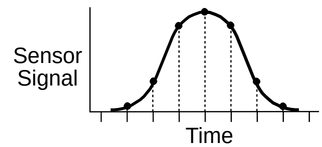
Contents

How it works

The High Speed Data Acquisition process is similar to the standard TestWare-SX data acquisition process. The difference is that the High Speed Data Acquisition process acquires data at a high speed according to a time increment using a single buffer (only the single buffer is available).

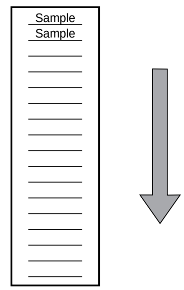
Timed data records data at specified time intervals.

For example, data could be acquired once each second on all selected data channels.



Data is recorded to fill a single buffer once, then stops the process and saves the data to disk.

The size of the data buffer determines how much data to collect.



The process acquires data for up to eight input channels (sensors). A time increment determines the rate of data acquisition. Each channel of data can have different units assigned to it.

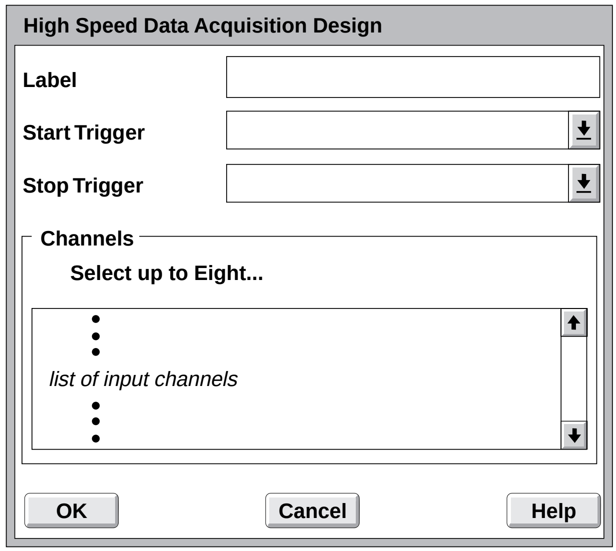
The High Speed Data Acquisition process can acquire data at a maximum rate of 70 kHz (for a single channel) vs. 5 kHz for the standard data acquisition process. This process cannot run at rate less than 100 Hz. The number of data input channels reduces the maximum data acquisition rate.

Input Channels	Rate
1	50 kHz
2	35 kHz
3	25 kHz
4	20 kHz
5	15 kHz
6	13 kHz
7	11 kHz
8	10 kHz

High Speed Data Acquisition Design Window

The information in this window is saved with the test template. This type of process should be sequenced in parallel with a command process. Meaningful data cannot be acquired unless the control channel is doing something.

This window names the process and specifies when the process starts and stops, and how it acquires data.



CONTROL	FUNCTION
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger.
Channels	Specifies additional input channels for data acquisition. Highlight each input channel to select it. Up to eight input channels can be selected.

Start trigger

- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

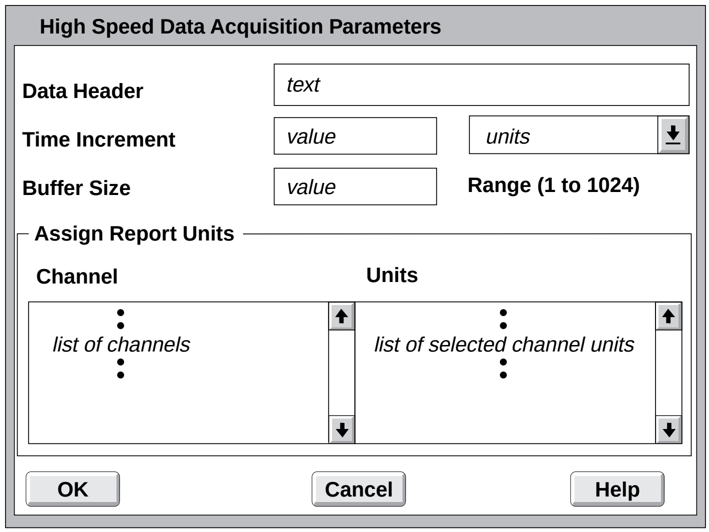
End trigger

- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

High Speed Data Acquisition Parameters

You must establish a data acquisition process with the High Speed Data Acquisition Design window before you can use this window. The information in this window is saved with the test procedure.

Use this window to define the parameters of the process.



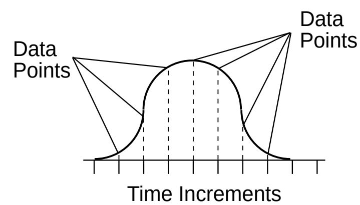
CONTROL	FUNCTION
Data Header	Labels the data in the data file.
Time Increment	Specifies how long between data acquisitions. Select the units with the right entry field then type the value you want to specify for the data acquisition requirement in the left entry field.
Buffer Size	Specifies the number of data elements the buffer can store. A data element represents one data sample of each channel.
Assign Report Units	This area of the window assigns the units for each channel. Select a channel, then select the units for that channel. Repeat this for each channel listed.

Buffer size

The default buffer size is 1024. You can set the data buffer to record 1 to 1024 data elements. A data element includes the data from all channels along with a relative time stamp.

Time increment

A Time increment value specifies how long between data acquisitions. The smallest time increment is 1/100 second.



For example, suppose you want to acquire data once each 1/100 second. Enter 0.01 second in the Time Increment entry field and select Seconds from the units list icon. Select the input channels you wish to monitor. Data is acquired from all channels each second.

Assigning units

Use this procedure to assign units to each data channel.

1. Highlight a channel in the Channel column. This causes the Units column to display the appropriate units for the dimension of the selected channel.
2. Highlight the units you want assigned to the channel data.
3. Repeat this procedure for each channel.

Data Files

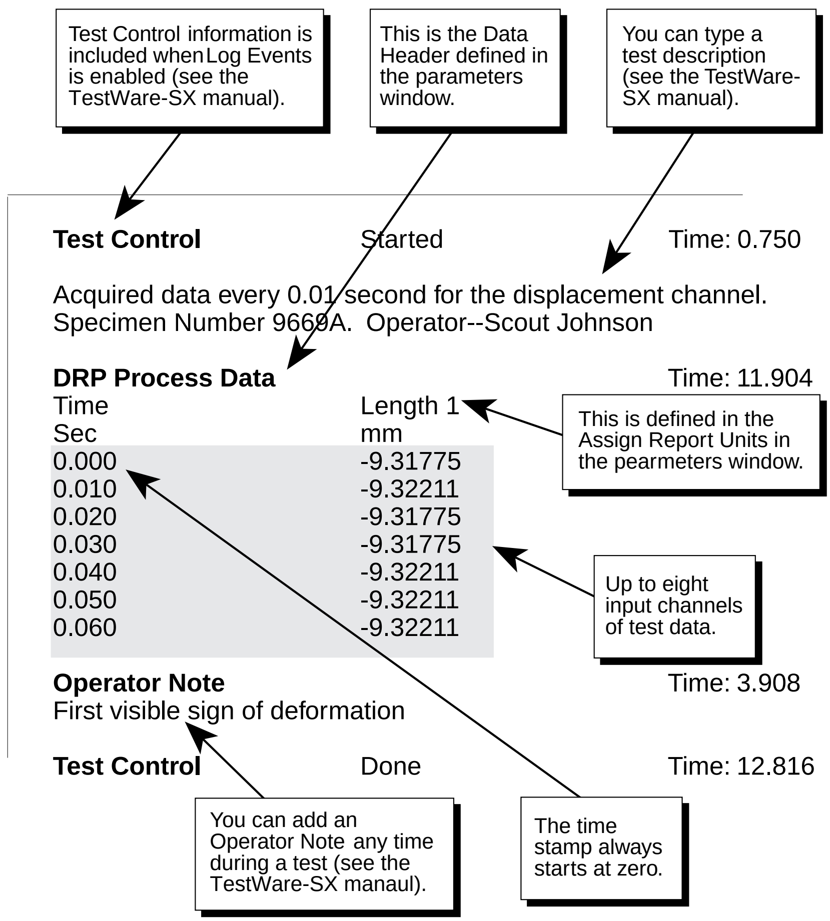
A data file contains the data acquired from a test along with a label indicating the type of data and the units of the data. Data is acquired for all available input channels. The data file format can be selected for use with popular spreadsheet programs or plain text.

The data acquired from the high speed data acquisition process is put into the standard TestWare-SX data file. The following is an example of the output from a data file.

This shows the types of information that can be saved in a data file.

The data format is in ASCII.

The file contains separators so that the columns of data automatically appear in separate columns in a spreadsheet program.



Chapter 6

Data Monitor Processes

This chapter describes the Model 790.17 Data Monitor TestWare Processes. These processes are designed to monitor TestStar sensors or dynamic properties during long cyclic tests. These are optional processes that are added to the special processes in TestWare-SX.

- The **Trend Monitor Process** displays minimum, maximum, mean, and span levels of TestStar input signals at regular intervals. Any configured TestStar input channel may be monitored.
- The **Dynamic Property Monitor Process** displays up to (7) seven dynamic properties in one process. The dynamic properties are K*, K', K", Phase, Tan Delta, Damping (C), and Total Energy.

Note *The abbreviation for trend monitor is TM. The abbreviation for dynamic property monitor is DPM.*

Contents

Trend Monitor Process

The trend monitor process monitors up to three statistics (minimum, maximum, and mean) calculated from input signals of the same dimension. The input signal data is displayed on a run time window. Use this process to monitor long fatigue tests.

How it works

The trend monitor process plots a sensor's maximum output, minimum output, or mean level over a period of time. The trend monitor must be defined to run within a loop, in a step, of a TestWare-SX test Procedure. The sensor data is plotted each time the loop is repeated.

Each trend monitor process is displayed in its own run-time plot window. Each plot window provides options to adjust the axis scaling, saving to a data file, and printing the current plot.

For example, assume the test cycles between ±3 mm displacement 1000 times within a loop, and repeats the loop 10,000 times. Also assume the trend monitor is set to plot the maximum force signal. Each time the loop is repeated, the run time window plots the maximum force recorded while the last loop was running.

- Up to three input signals of the same dimension can be plotted.
- A maximum of three data series from four statistics (Minimum, Maximum, Mean, and Span with the same dimension) can be monitored under one process.
- Each process displays a plot window. Each data series in the plot represents one statistic defined in the process.
- One input signal can be assigned to all three channels so you can monitor the minimum, maximum, and mean levels of one sensor.
- During the test, the run-time window can automatically change its scaling. You can also manually change the run time window scaling.

Print preview

The following is an example of a trend monitor process as printed from the print preview feature in TestWare-SX.

```
Procedure Name = TREND1 Default Procedure
File Specification = D:\TS2\TWSX\TREND1.000
Software Version = 3.1A
Printout Date = 10-19-1995 09:40:45 AM

Data File Options
   File Format = Excel Text File
   Log Events = Yes
   Include Procedure Description = No
Recovery Options
    Yugoslav disabled
Loop: Begin Loop
   Counter Name = Loop
   Total Count = 0

Trend: Step
   Step Done Trigger 1 = Cyclic Command

   Cyclic Command : Cyclic Command
      Start Trigger = Step Start
      End Trigger = <none>
      Segment Shape = Haversine
      Frequency = 10 ( Hz )
      Repeats = 1000 cycles
      Compensation = Phase/Amplitude (PAC)
      AXIAL
         Control Mode = DISP PID SG
         End level 1 = 3 ( mm )
         End level 2 = -3 ( mm )

     Trend Monitor Process : Trend Monitor
      Start Trigger = Step Start
      End Trigger = <none>
      Reduce Plot Rate When Decimation Occurs = No
      Auto Scaling = Off
      Input #1
         Sensor = LOAD CELL
         Statistic = Minimum
         Line Style = Solid
         Line Color = Red
      Input #2
         Sensor = LOAD CELL
         Statistic = Maximum
         Line Style = Solid
         Line Color = Blue
```

Continued…

Print preview (continued)

```
            Input #3
            Sensor              = LOAD CELL
            Statistic = Mean
            Line Style = Solid
            Line Color = Green
         Y Axis
            Scaling
                              Maximum = 1000 ( N )
                              Minimum = -1000 ( N )
            Limit Detector = Absolute
                              Maximum = 800 ( N )
                              Minimum = -950 ( N )
         X Axis
            Operator Defined Text = One pass is 100 seconds
            Scaling = Linear
Loop : End Loop
```

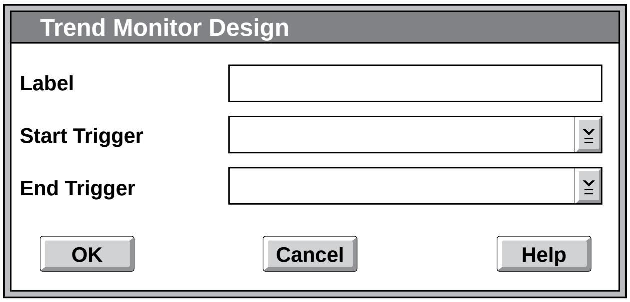
Data Monitor Processes

Trend Monitor Design Window

The information in this window is saved with the test template. This process must be run in parallel with a command process, see the Print preview for an example of how the process could be used in a test.

This window names the process and specifies when it starts and stops.

The trend monitor and command processes must be run from within a loop.



Control	Description
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger

Start trigger

- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

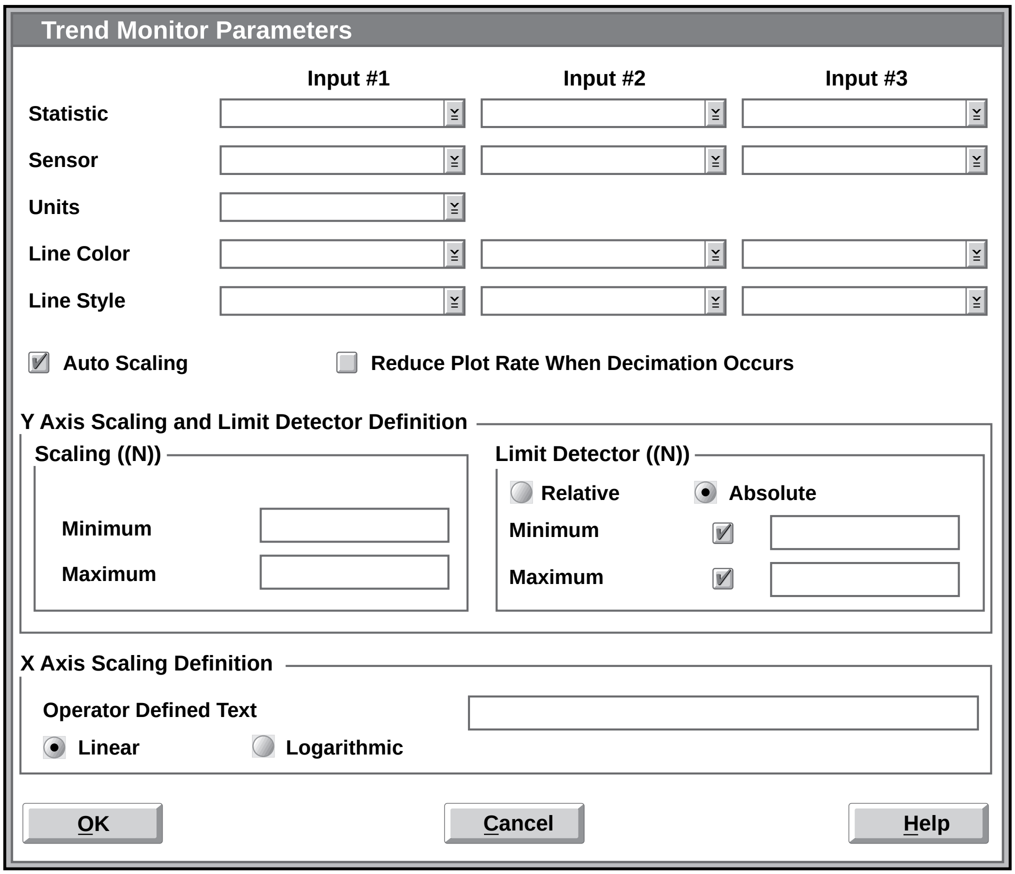
End trigger

- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

Trend Monitor Parameters Window

This window defines the specific characteristics of the trend monitor process.

You must create the trend monitor process in a template with the design window before you can access this window. The information in this window is saved with the test procedure.



Control	Function
Statistic	Specifies the type of sensor information being monitored You can specify the following types of data: • **Maximum** • **Minimum** • **Mean** (max-min) • **Span** ((max+min)/2) Undefined channels are automatically set to **inactive**.
Sensor	Defines up to three sensors. Each sensor specifies an input signal for which the statistic is plotted. Sensors 2 and 3 must be of the same dimension as Sensor 1.
Units	Specifies the units for all three channels. The dimension for the units is specified by the type of input signal selected for channel 1.
Line Color	Selects the color for the respective plot line: **Red** **Green** **Blue**
Line Style	Selects the style of line for the respective plot line: **Solid** **Dashed** **Dotted**
Auto Scaling	Enables the Auto Scale feature to automatically change the axis values so the plot data can always be seen.
Reduce Plot Rate When Decimation Occurs	Enables the reduce plot rate feature. After each decimation occurs, new data is collected at 1/2 the previous rate. This provides uniform resolution between old and new data.
Y Axis Scaling	Sets the initial minimum and maximum range of the Y axis in the run-time plot window. If Auto Scaling is enabled, the Y axis automatically changes when the plot exceeds the initial range. Otherwise, data exceeding the initial range is not shown.
Y Axis Limit Detector	Defines a limit detector that can end the process. You can specify the detector values as **absolute** values or as a **relative** value based on the percentage of the initial data.
X Axis Scaling Definition	You can enter a label for the x axis in the **Operator Defined Text** entry field. You can also specify if the X axis of the plot is **linear** or **logarithmic**.

Decimation

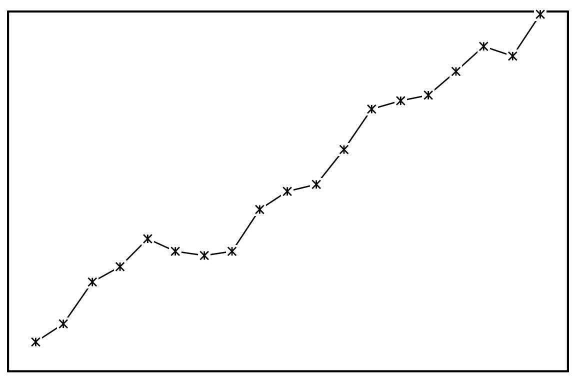
The plot buffer for each data series can hold 4000 data points. When a buffer becomes full, every other data point is removed to make room for more data. An additional 2000 data points of data is collected at the same rate as the first 4000 data points then decimation is repeated.

- Decimation is always enabled.
- Decimation reduces the resolution of the plot.

For example, suppose you collect data once per second. After 4000 data points are collected, every other data point is removed so 2000 data points remain. Data continues to be collected once per second until the buffer is full again. Then the data is decimated and new data continues to be collected once per second. At this point, the first 500 data points (1⁄4 of the buffer) are at 4 second intervals, the second 500 data points are of 2 second intervals, and the last 1000 data points are at 1 second intervals.

Before Decimation:

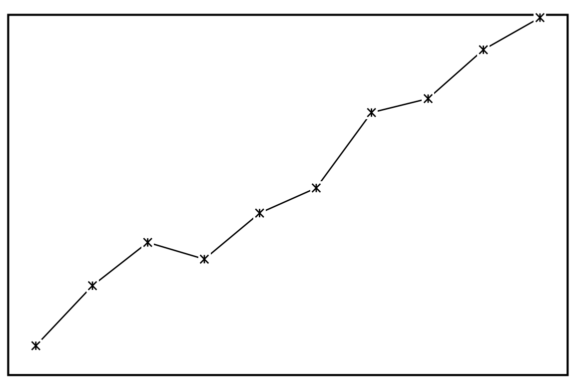
Data points are acquired until the data buffer is full (4000 data points).



After Decimation:

Half the data points are removed from the plot series, 2000 data points remain.

Here, the remaining 2000 data points are streached accross the full display.



Decimation rate reduction

When the **Reduce Plot Rate When Decimation Occurs** feature is enabled, data is then collected at half the previous rate following a decimation.

- When the data rate reduction feature is not enabled, the first data taken is at a different resolution than the most recent data taken.
- When the data rate reduction feature is enabled, the data resolution is uniform.

For example, suppose you collect data once per second. After 4000 data points are collected, every other data point is removed so points 2000 data points remain. Data continues to be collected every 2 seconds until the buffer is full. Then the data is decimated and new data continues to be collected every 4 seconds.

Number of Decimations	Data Acquisition Rate
0	every data point
1	every 2nd point
2	every 4th point
3	every 8th point

Note *You may not notice a change in the data plot on your computer monitor for the first several decimations. A high resolution monitor may show only 1024 points.*

Limit detector

You can define a limit detector to end the trend monitor process if the acquired data exceeds one of the limits. You can define a minimum detector, a maximum detector, or both detectors. You can also define the detector absolute or relative values.

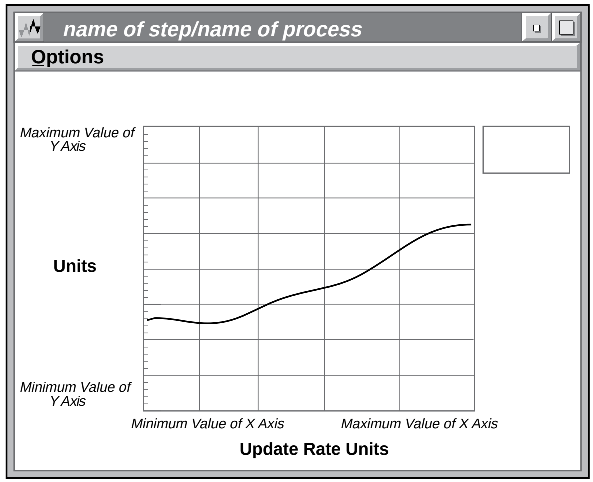
- If a data point goes below the minimum limit or higher than the maximum detector setting, the trend monitor process ends.
- **Absolute** values allow you to enter exact minimum and/or maximum limits in the units of the sensor.
- **Relative** values allows you to enter minimum and/or maximum limits as a percentage of a reference value. The reference value is the first data collected. The minimum value is the percentage below the reference value. The maximum value is the percentage above the reference value.

Run-Time Plot Window

This window plots the data while the test is running. It can show up to three data series (plot lines). Each data series represents a channel's statistic defined in the Parameters window. Each data series plot can be a different color.

The title bar of this window includes the names of the step and process specified in the trend monitor test procedure.

.



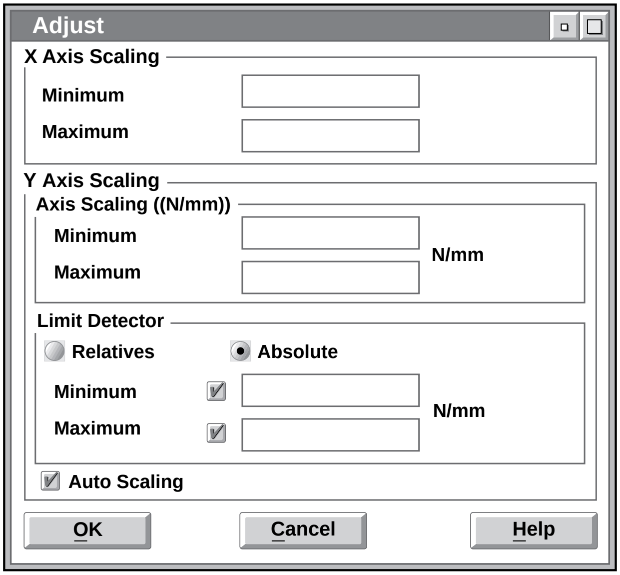
The Controls listed below are found in the **Options** menu.

CONTROL	FUNCTION
Reset	Clears the current display, resets the pass count to zero, and continues plotting new data. Reset doesn't affect the reference value if you are using a relative limit detector.
Adjust	Displays the Adjust window where you can manually change the scale of each axis or enable/disable auto scaling.
Save	Displays the Save Plot window where you can save the plot data in the same format defined by TestWare-SX.
Print	Prints the data as it appears in the current Run Time Plot window.

Adjust Window

Use the Options menu in the Trend Monitor Run-Time Plot window to display this window.

This window lets you manually change the scale of the plot and change the limit detector settings.



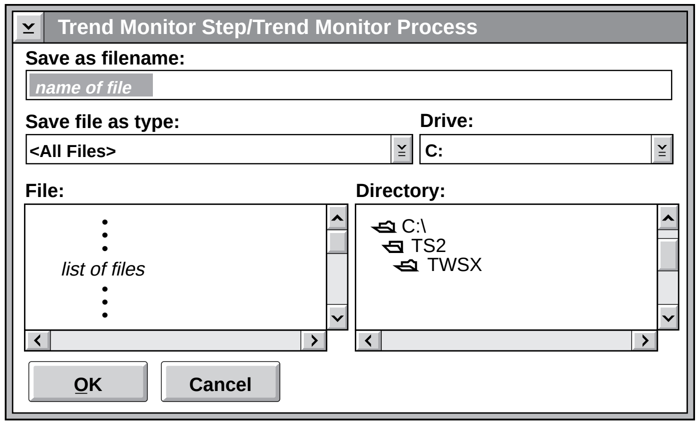
CONTROL	FUNCTION
X Axis Scaling	Specifies the minimum and maximum values for X Axis of the plot unless Auto Scaling is enabled.
Y Axis Scaling	Sets the initial minimum and maximum range of the Y axis in the run-time plot window.
Y Axis Limit Detectors	Defines a limit detector that can end the trend monitor process. You can specify the detector values as **absolute** values or as a **relative** value based on the percentage of the initial data.
Auto Scaling	Enables the Auto Scale feature to automatically change the axis values so the plot data can always be seen. If Auto Scaling is enabled, the Y axis automatically changes when the plot exceeds the initial range. Otherwise, data exceeding the initial range is not shown.

Save Plot Window

Use the Options Menu in the Trend Monitor Run Time Plot window to display this window.

Use this window to save plot data to an electronic file.

The title bar of this window includes the names of the step and process specified in the trend monitor test procedure.



CONTROL	FUNCTION
Save as filename	Type the name you want to call the plot data file in the File entry field.
Save file as type	This control is not used.
Drive	Displays the current drive. When you select a different drive, the directory list is updated.
File	Lists the data files in the current directory.
Directory	Lists the available directories. Double-clicking on a different directory displays the new path shown next to the current directory label.

Using the window

By default, this window should open to the directory where you keep your data files. If not:

1. Select the appropriate drive (C:\).
2. Find the proper directory (C:\TS\TWSX).
3. Enter a filename in the top entry field.
4. Press the OK pushbutton.

Data file example

The following is an actual data file. The data was acquired from a displacement channel.

Passes	Max (mm)	Min (mm)	Mean (mm)
1	0.73990631	-1.9866726	-0.68113506
2	-0.29411116	-1.9980507	-1.1460810
3	-0.30479959	-2.0090837	-1.1569417
4	-0.31583279	-2.0190828	-1.1674578
5	-0.32548684	-2.0297711	-1.1776290
6	-0.3341065	-2.0397699	-1.1869382
7	-0.34376055	-2.0501137	-1.1969371
8	-0.35341462	-2.0590780	-1.2062463
9	-0.36203429	-2.0676978	-1.2148660
10	-0.37203312	-2.0773518	-1.2246925
11	-0.3809976	-2.0866611	-1.2338294
12	-0.38823813	-2.0939016	-1.2410699
13	-0.39616826	-2.1018317	-1.2490000

Dynamic Property Monitor Process

The dynamic property monitor process monitors up to seven dynamic properties calculated from load/displacement data. You can monitor the dynamic properties on a run-time window. Use this process on the long durability tests.

Note *The abbreviation for dynamic property monitor is DPM.*

How it works

The dynamic property monitor process acquires force and displacement data at specified periodic intervals. From this data the process can calculate and plot up to seven dynamic properties: K*, K', K", Phase, Tan Delta, Damping (C), and Total Energy.

Each property is displayed as a data series in its own plot window. Each plot window provides options to scale the display, save the data, or print the plot window.

Each dynamic plot monitor can save its results to a data file. Each dynamic property monitor process can create a data file with all of the defined property values written to the same file. DPM process will have its own associated data file for output (using the first four characters of the TWSX file name plus a four digit serial number). Multiple properties under the same process will be written to the same data file.

Print preview

The following is an example of a dynamic property monitor process as printed from the print preview feature in TestWare-SX.

```
TestWare-SX

Procedure Name = DPM1 Default Procedure
File Specification  = D:\TS2\TWSX\DPM1.000
Software Version  = 3.1A
Printout Date... = 10-19-1995 09:40:10 AM
```

Continued...

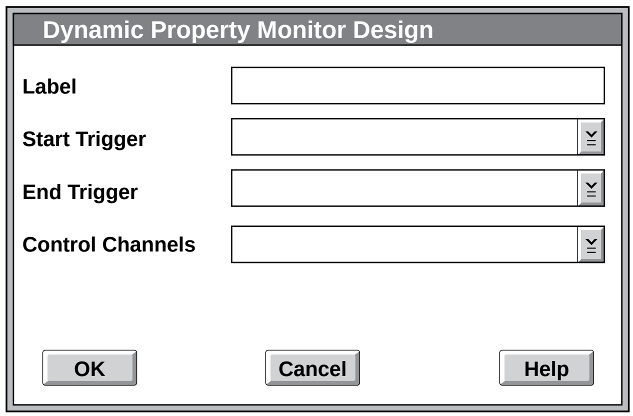
How it works
(continued)

```
Data File Options
   File Format = Excel Text File
   Log Events = Yes
   Include Procedure Description = No

Recovery Options
   Autosave disabled.

DPM Step : Step
Step Done Trigger 1 = DPM Process
   Cyclic Command : Cyclic Command
      Start Trigger = Step Start
      End Trigger = <none>
      Segment Shape = Haversine
      Frequency = 10 (Hz)
      Repeats = 0 (segments)
      Compensation = Phase/Amplitude (PAC)
      AXIAL
         Control Mode = DISP PID SG
         End level 1 = -0.2 ( mm )
         End level 2 = -0.6 ( mm )
   Delay : Data Limit Detector
      Start Trigger = Step Start
      End Trigger = <none>
      Data Channel = Time
      Limit Value = 5 ( Sec )
      Limit Value is = Relative
      Trigger Options   = Greater than Limit Value

   DPM Process : Dynamic Property Monitor
      Start Trigger = Delay
      End Trigger = <none>
      Control Channel   = AXIAL
      Force Sensor = LOAD CELL
      Length Sensor = LVDT
      Plot Update Rate  = 10 ( Sec )
      Reduce Plot Rate When Decimation Occurs = No
      Save Data = Yes
      X Axis Scaling... = Linear
      K*
         Axis Scaling
                         Minimum = 0 ( N/mm )
                         Maximum = 500 ( N/mm )
                         Limit Detector = Absolute
                         Minimum = 100 ( N/mm )
                         Maximum = 400 ( N/mm )
                         Auto Scaling = Yes
```

Dynamic Property Monitor Design Window

The information in this window is saved with the test template. This process must be run in parallel with a sinusoidal command process (cyclic or mixed mode sine). It is recommended to start this process after the cyclic process has had adequate time to achieve the desired amplitude. This is especially true if a compensation circuit is being used.

This window names the process and specifies when it starts and stops.



CONTROL	FUNCTION
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger.
Control Channel	Specifies which control channel the process is applied to. Only one channel can be selected. .The monitor process uses it to query frequency information from TestStar.

Start trigger

- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

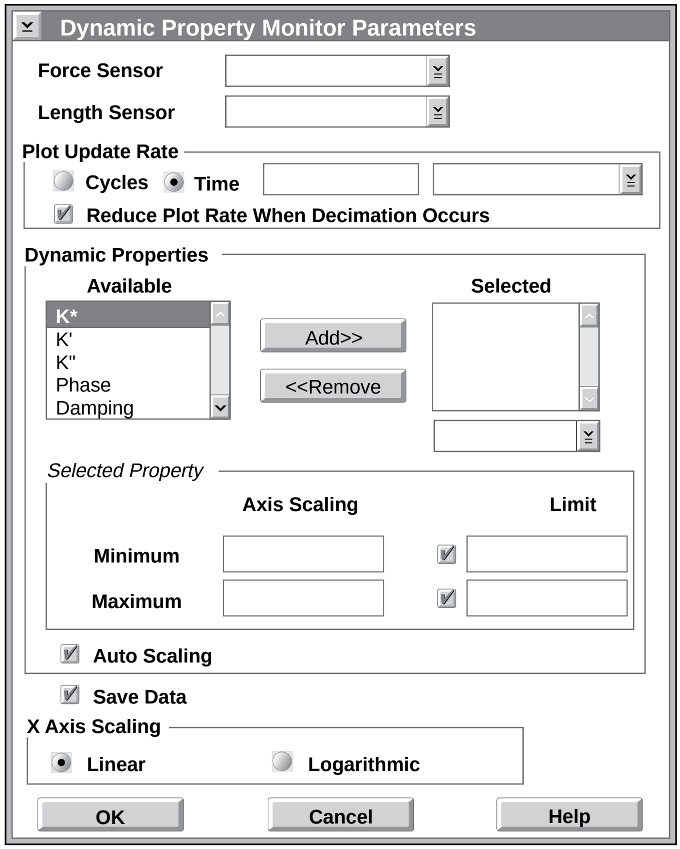
End trigger

- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

Dynamic Property Monitor Parameters Window

You must create a DPM process in a template with the design window before you can access this window. The information in this window is saved with the test procedure.

Use this window to define the specific characteristics of the dynamic property monitor process.



Control	Function
Force Sensor	Selects a force or load input signal.
Length Sensor	Selects a length or displacement input signal.
Plot Update Rate	Specifies how the plot is updated. Plot data can be taken at a specified **time** interval or after a specified **cycle** count. Use the entry fields to specify the value and units of the update rate.
Reduce Plot Rate When Decimation Occurs	Enables the reduce plot rate feature. After each decimation occurs, new data is collected at 1/2 the previous rate. This provides uniform spacing between old and new data.
Dynamic Properties	Specifies what data is to be plotted. The **Available** list shows what properties can be plotted. The **Selected** list shows what properties will be plotted. Use the **Add** and **Remove** pushbuttons to move a selected property from one list to the other. Below the Selected list you can specify the type of unit for the acquired data.
Y Axis Scaling	Sets the initial minimum and maximum range of the Y axis in the run-time plot window. If Auto Scaling is enabled, the Y axis automatically changes when the plot exceeds the initial range. Otherwise, data exceeding the initial range is not shown.
Limit Detectors	Defines a limit detector that can end the process. You can specifiy the detector values as **absolute** values or as a **relative** value based on the percentage of the initial data.
Auto Scaling	Enables the Auto Scale feature to automatically change the axis values so the plot data can always be seen.
Save Data	Saves all of the data to a data file (with the extension .TIM). This is different than the Save function in the Options menu in the run-time plotting window.
X Axis Scaling	Specifies if the X axis of the plot is **linear** or **logarithmic**.

Decimation

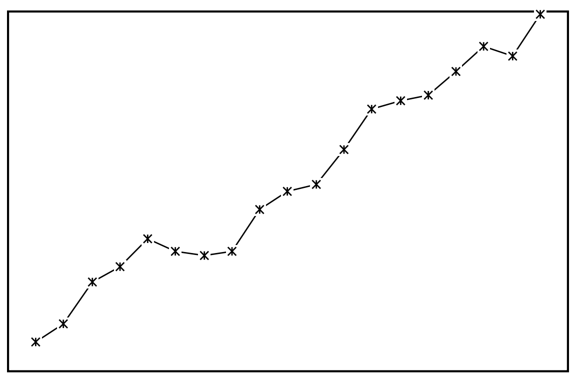
The plot buffer for each data series can hold 4000 data points. When a buffer becomes full, every other data point is removed to make room for more data. An additional 2000 data points of data is collected at the same rate as the first 4000 data points then decimation is repeated.

- Decimation is always enabled.
- Decimation reduces the resolution of the plot.

For example, suppose you collect data once per second. After 4000 data points are collected, every other data point is removed so 2000 data points remain. Data continues to be collected once per second until the buffer is full again. Then the data is decimated and new data continues to be collected once per second. At this point, the first 500 data points (1⁄4 of the buffer) are at 4 second intervals, the second 500 dta points are of 2 second intervals, and the last 1000 data points are at 1 second intervals.

Before Decimation:

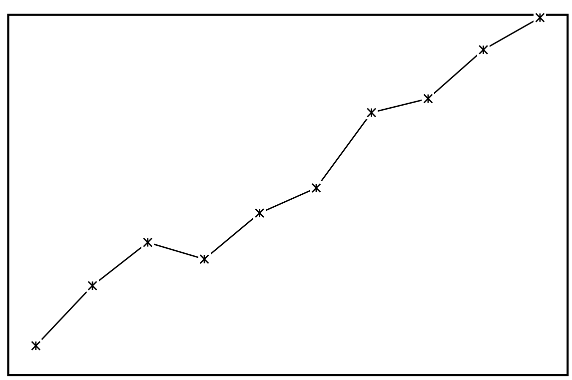
Data points are acquired until the data buffer is full (4000 data points).



After Decimation:

Half the data points are removed from the plot series, 2000 data points remain.

Here, the remaining 2000 data points are streached accross the full display.



Decimation rate reduction

When the **Reduce Plot Rate When Decimation Occurs** feature is enabled, data is then collected at half the previous rate following a decimation.

- When the data rate reduction feature is not enabled, the first data taken is at a different resolution than the most recent data taken.
- When the data rate reduction feature is enabled, the data resolution is uniform.

For example, suppose you collect data once per second. After 4000 data points are collected, every other data point is removed so points 2000 data points remain. Data continues to be collected every 2 seconds until the buffer is full. Then the data is decimated and new data continues to be collected every 4 seconds.

.

Number of Decimations	Data Acquisition Rate
0	every data point
1	every 2nd point
2	every 4th point
3	every 8th point

Note *You may not notice a change in the data plot on your computer monitor for the first several decimations. A high resolution monitor may show only 1024 points.*

Limit detector

You can define a limit detector to end the trend monitor process if the acquired data exceeds one of the limits. You can define a minimum detector, a maximum detector, or both detectors. You can also define the detector absolute or relative values.

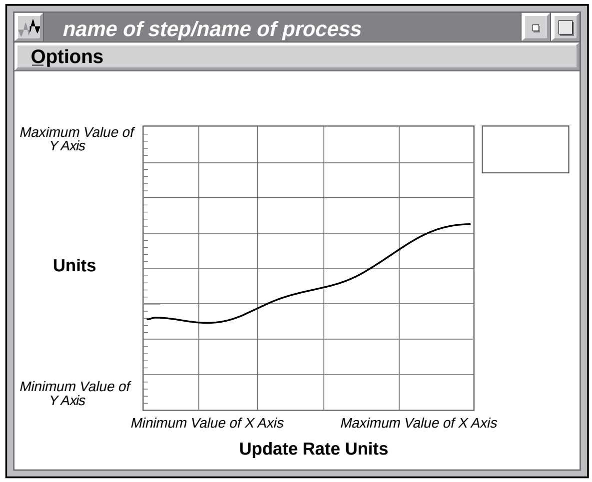
- If a data point goes below the minimum limit or higher than the maximum detector setting, the trend monitor process ends.
- **Absolute** values allow you to enter exact minimum and/or maximum limits in the units of the sensor.
- **Relative** values allows you to enter minimum and/or maximum limits as a percentage of a reference value. The reference value is the first data collected. The minimum value is the percentage below the reference value. The maximum value is the percentage above the reference value.

Data Monitor Processes

Run-Time Plot Window

A run-time plot window is displayed for each dynamic property selected in the Dynamic Property Monitor Parameters window. Each window show the data series for one of the dynamic properties.

The title bar of this window includes the names of the step and process specified in the trend monitor test procedure.



The Controls listed below are found in the **Options** menu.

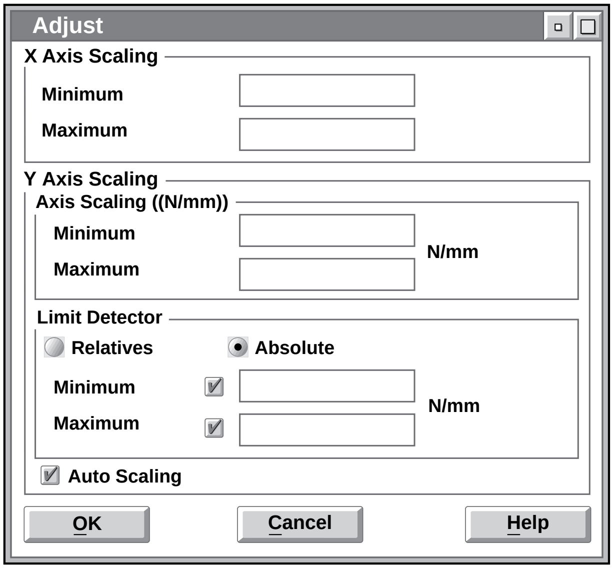
CONTROL	FUNCTION
Adjust	Displays the Adjust window where you can manually change the scale of each axis or enable/disable auto scaling.
Save	Displays the Save Plot window where you can save the plot data in the same format defined by TestWare-SX.
Print	Prints the data as it appears in the current Run Time Plot window.

Adjust Window

Use the Options menu in the Trend Monitor Run-Time Plot window to display this window.

This window lets you manually change the scale of the plot.

The maximum and minimum plot axes are labeled in this window



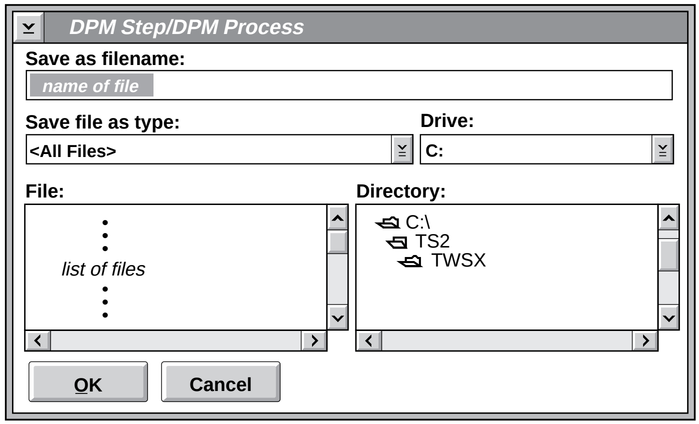
CONTROL	FUNCTION
X Axis Scaling	Specifies the minimum and maximum values for X Axis of the plot unless Auto Scaling is enabled.
Y Axis Scaling	Sets the initial minimum and maximum range of the Y axis in the run-time plot window.
Y Axis Limit Detectors	Defines a limit detector that can end the trend monitor process. You can specifiy the detector values as **absolute** values or as a **relative** value based on the percentage of the initial data.
Auto Scaling	Enables the Auto Scale feature to automatically change the axis values so the plot data can always be seen. If Auto Scaling is enabled, the Y axis automatically changes when the plot exceeds the initial range. Otherwise, data exceeding the initial range is not shown.

Save Plot Window

Use the Options Menu in the Trend Monitor Run Time Plot window to display this window.

Use this window to save plot data to an electronic file.

The title bar of this window includes the names of the step and process specified in the trend monitor test procedure.



CONTENTS	FUNCTION
Save as filename	Type the name you want to call the plot data file in the File entry field (before you save it).
Save file as type	This control is not used.
Drive	Displays the current drive. When you select a different drive, the directory list is updated.
File	Lists the data files in the current directory.
Directory	Lists the available directories. Double-clicking on a different directory displays the new path shown next to the current directory label.

Using the window

By default, this window should open to the directory where you keep your data files. If not:

1. Select the appropriate drive (C:\).
2. Find the proper directory (C:\TS\TWSX).
3. Enter a filename in the top entry field.
4. Press the OK pushbutton.

Data file example

An actual data file is provided as an example of the file format.

TIME (SECONDS)	PHASE (DEGREES)
5.031	4.638443
13.094	4.63176
21.062	4.629223
29.062	4.645523
37.062	4.567383
45.062	4.560802
53.094	4.570061
61.062	4.619307
69.062	4.608908

Chapter 7

Run-Time Ramp Control

The Run-time Ramp Control process allows you to simultaneously control up to four control channels in an incremental fashion, stepping toward indeterminate end levels. The process also allows you to define an intervening pause during which another process is triggered to send a signal to a remote data collection device.

This combination of ramp and pause allows you to ramp the actuators, wait while an external device collects data, adjust your step size, if desired, and select a button to start the next ramp. This process is open-ended, displaying a run-time window during the test which provides access to the various parameters and user controls. The window, along with the process, remains active and accessible until you elect to end the process or until a standard TestWare-SX event occurs (e.g., stop trigger, reset etc.).

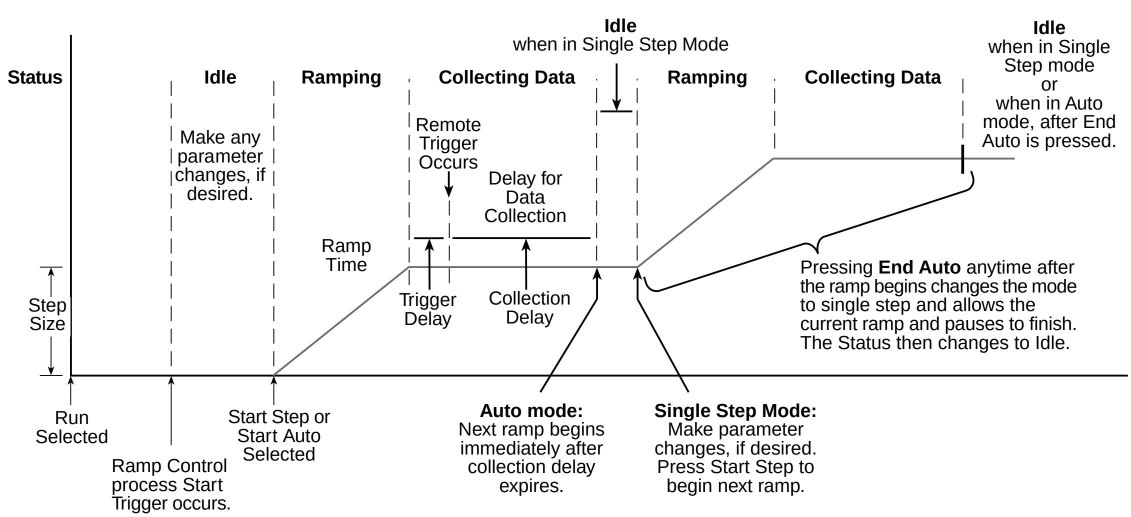
Contents

How it works

When Run is selected, the test sequences through the procedure in the normal manner until the Run-time Ramp Control process starts. The process starts by displaying the Run-time Ramp Control window. The status field in this window will indicate Idle. The process will remain in Idle until you decide to start a ramp. This Idle time gives you an opportunity to make any final corrections to the parameter fields displayed in the run-time window. When the Start Step/Start Auto button is pressed the first ramp begins. The command signal for each channel ramps the specified step size in the specified ramp time. If Remote Triggering was selected, the process ramp will pause for the specified delay times and trigger the external process to collect data at the end of the ramp

If Single Step mode was selected the command signal continues to hold at the current level. The process status returns to Idle allowing you to change parameters, if desired, before selecting Step Start again.

If Auto mode was selected the ramp continues automatically from the current level (after the delay times expire) with the same ramp time and step size. This cycle repeats itself until you decide to stop it by selecting End Auto (which puts the process in the Single Step mode where you can edit the parameters). The figure below illustrates the main features of the Run-time Ramp Control Process.

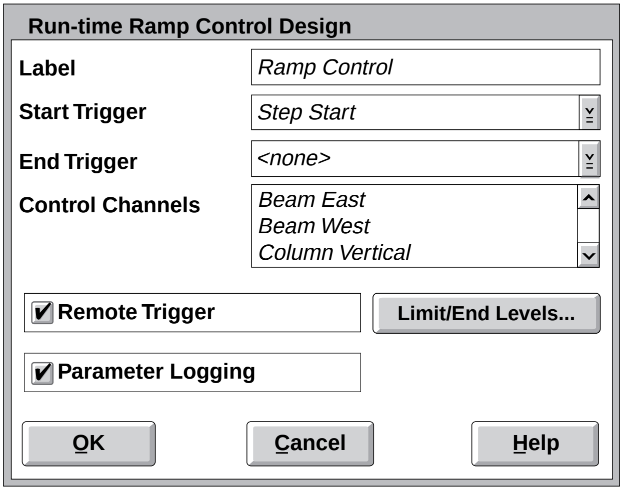


Run-time Ramp Control Design Window

The information in this window is saved with the test template.

This window names the process and specifies how the it starts and stops.

.



CONTROL	FUNCTION
Label	Names the process. Type the name you want to call the process in the entry field.
Start Trigger	Specifies the beginning of the process. Press the list icon and select a trigger.
End Trigger	Specifies the end of the process. Press the list icon and select a trigger
Control Channels	Select up to 4 channels for control. You must select at least one channel. Highlight each channel you want for this process.
Remote Trigger Active	Enables the process to issue an end trigger at the completion of every ramp. For example, use the remote trigger to start a data acquisition process.
Parameter Logging	Enables all process parameters to be written to the data file each time the Start Step/Start Auto button on the Run-time Ramp Control window is pressed. It also enables the Comments entry field.
Limit/End Levels...	Opens the Select Limit/End Level Channels window where you select the input signals for the process limits.

Start trigger

- Use the Step Start trigger (or the name of the step) to start the process at the beginning of the step.
- You can select another process (by name) to start this process after the selected process ends.

Note *Don't try to start more than one command process at a time—if you do, you will cause an error. Sequence test command processes in series.*

End trigger

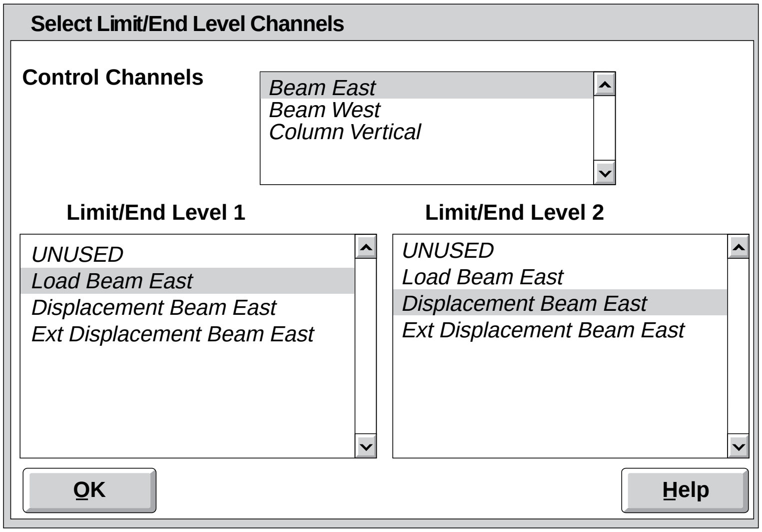
- The default end trigger is <none>. The none trigger means the process will complete its task unless some other process causes the Step Done process to be encountered.
- You can select another process (by name) end this process prematurely. When the selected process ends, it also stops this process.

Select Limit/End Level Channels Window

This window allows you to select up to two input signals for each selected control channel. Each input signal is monitored during the test. If the level specified by you is crossed, all ramps will stop. The limit levels are set in the parameters window.

You must have at least one control channel selected from the Run-time Ramp Control Design window in order to access this window.

Clicking OK returns you to the main Run-time Ramp Control Design window.



'Limits' as used in this process should not be confused with the TestStar upper and lower limit detectors.

'Limits' as used in this process simply define a level for the selected input. When the signal exceeds this level, all ramps stop and are put in a hold condition. No further actions are taken.

Control	Function
Control Channel	Shows the control channels selected in the Run-time Ramp Control Design window. Select each control channel to view the associated input channels.
Limit/End Level 1 and 2	Both lists are identical and show only those inputs associated with the selected control channel. If desired, select an input channel from one or both lists.

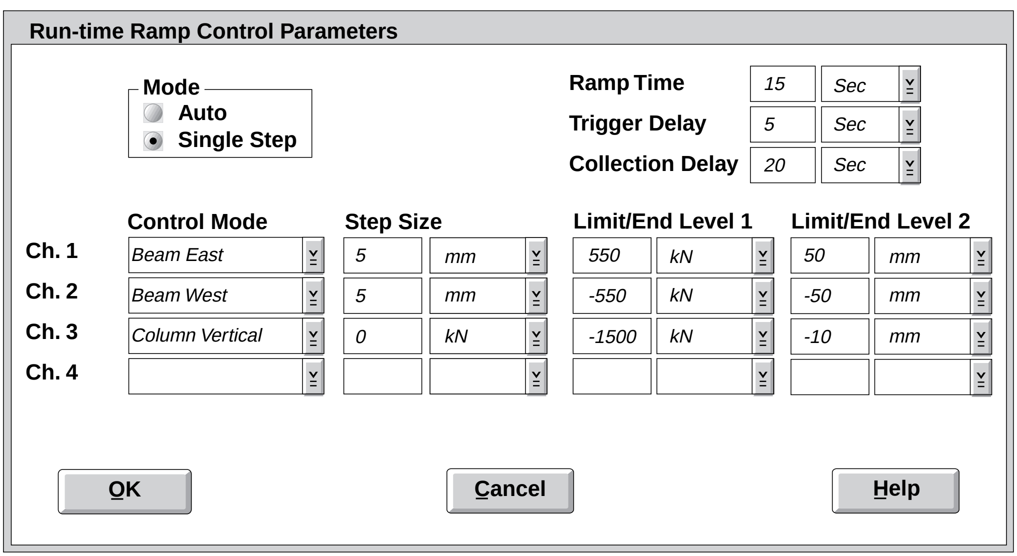
Run-time Ramp Control Parameters Window

You must create a Run-time Ramp Control process in a template in order to access this window. All the information in this window is saved with the template.

The Run-time Ramp Control Parameters window defines the ramp mode (single step or auto), a ramp time which applies to all channels and a ramp step size which is specific to each channel.

- If Remote Trigger was selected in the design window you can set a trigger delay which results in a pause at the end of a ramp before triggering other processes and a collection delay that follows the trigger delay.
- In the Auto Mode, the sum of the trigger delay and the collection delay represents the total delay from the end of one ramp to the beginning of the next.

All active parameters in this window can be edited from the Run-time Ramp Control window. The window below shows example parameters.



Control		Function
Mode	Auto	Performs ramps according to the Step Size, Ramp Time, and Collection Delay parameters. Ramps continue until a limit is detected or the End Auto button is pressed in the run-time window.
	Single Step	Performs a single ramp according to the Step Size, Ramp Time, and Collection Delay parameters.
Ramp Time		Specifies the length of time for the ramps on all channels. Select your preference in units and enter a value greater than 0.
Trigger Delay		Specifies the length of delay following ramp completion before the process trigger is issued. Select your preference in units and enter a value greater than 0.
Collection Delay		Specifies the length of the delay to accommodate another process and perhaps a remote device to acquire data. Select your preference in units and enter a value greater than 0.
Control Mode		Displays the available control modes for each control channel selected during the step design. Select a control mode for each control channel.
Step Size		Specifies the size of the ramp. The units correspond with the selected control mode. Select the units then enter a value. The value must be within the sensor limits. Define a step size for each listed control channel.
Limit/End Level 1 & 2		When active, specifies a level for each input selected during the step design. If any selected signal crosses the specified level, all ramps stop and data collection is triggered. The units correspond to the selected input. Select the units then enter a value. The value must be within the sensor limits. Define limits for each listed control channel.

Modes

This process can be run manually step-by-step or automatically.

Single step

The Single Step mode allows you to set your parameters and start the ramp. Once the ramp completes, the process holds the command and triggers a data acquisition process or a remote device. The process continues to pause for a collection delay interval so the data acquisition process or remote device has adequate time to acquire data. Then the process returns to an idle state. If you want, you can change the ramp parameters and run another single step or switch to the auto mode.

Auto

After you've set your parameters you can start the auto mode. Once the ramp completes, the process holds the command and triggers a data acquisition process or a remote device. The process continues to pause for a collection delay interval so the data acquisition process or remote device has adequate time to acquire data. The auto mode continues the ramp and data collection routine until a limit is exceeded or if the End Auto button is pressed. The End Auto button idles the process after the current ramp gives you the ability to return to the Single Step mode

Delays

Both the Trigger Delay and Collection Delay are enabled by the Remote Trigger option in the Design window. While these delays are being executed, the valve command is held at its current level. Both delays are primarily used for external data collection methods.

- The Trigger Delay is designed to allow the input sensor signals to settle.
- The Collection Delay is designed for external data collection devices. The collection of data from a large number of sensors may require a specified amount of time.

Limits

The Limit/End Level feature is enabled when input signals are assigned in the Select Limit/End Level window. You to specify an end level to act as a limit that stops all ramps. This capability is particularly useful in conjunction with use of the auto mode feature. For each control channel, you may select two input signals that can be monitored for a limit value.

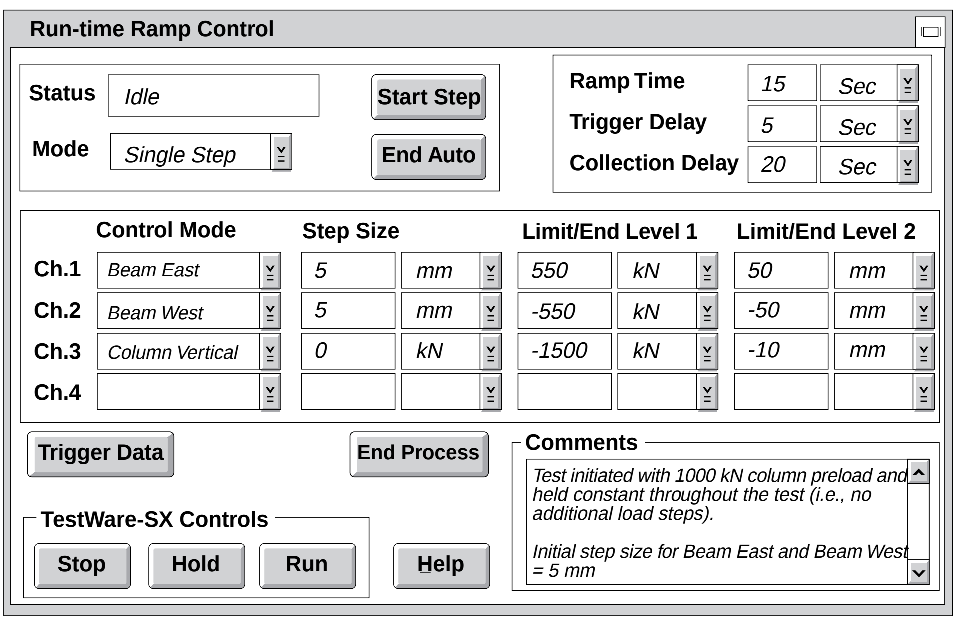
These channels are monitored during each ramp step. If any limit/end level is reached, all ramps are discontinued and the process returns to the idle state to await your next set of parameters and subsequent start signal.

Run-Time Ramp Control Window

This is the actual run-time window that is displayed after you select Run from the TestWare-SX Execute Procedure window. The ramp does not begin until you select Start Step/Start Auto. This allows you to make any final adjustments to the parameters before the ramp begins. All parameters in this window, including Mode, can be changed while the process Status indicates Idle or SX Stopped.

If Parameter Logging was selected during step design, the parameters in this window will be written to the data file whenever Start Step/Start Auto is pressed. The default template for the test you are running does not change unless you do a Save when you close the file.

The button in the upper right corner of this window allows you to toggle the frame size of the window. The full size view is shown below. The reduced view reduces the frame to display the left half of the window. This frees up space on your monitor while still allowing you access to control buttons.



Control		Function
Status		Displays the current status of the Run-Time Ramp Control process or the status of TestWare-SX. • **Idle** • **SX Holding** • **Ramping** • **SX Stopped** • **Collecting Data** • **SX Error**
Mode*	Single Step Auto	Executes a single ramp, collects data and sets the process to idle. Repeats the ramp and data collection cycle until a limit is reached.
Start Step **Start Auto**		Starts the ramp control process. The name of the pushbutton is determined by the mode selection.
End Auto		Changes the mode to single step while the automatic mode is operating. This pushbutton is active only when in the Auto mode and with status Ramping, Collecting Data or SX Holding.
Ramp Time*		Specifies the length of time for the ramp of each control channel.
Trigger Delay*		Specifies the length of time following ramp completion before the process trigger is issued.
Collection Delay*		Specifies the length of the time to collect data.
Control Mode*		Specifies the control mode for each channel.
Step Size*		Specifies the size of the ramp
Limit/End Level*1 and 2		Specifies a limit level for each input selected during the step design.
Trigger Data		Manually triggers the external process that handles data collection. A message is displayed asking you to confirm this selection.
End Process		Ends the process. Pressing this pushbutton displays a message asking you to confirm this selection. The process then issues a hold command to all ramping control channels, collects data and ends the process.
TestWare-SX Controls		These buttons duplicate the functions of the buttons on the TestWare-SX Execute Procedure window.
	Stop	Stops the test.
	Hold	Suspends the test until the Run or Stop pushbutton is pressed.
	Run	Starts the test or restarts this process after a hold.
Comments		This is enabled when Parameter Logging is selected in the design window. This is an entry field where you can record information about the test. Comments may be entered when status is Idle or SX Stopped.

***** These controls are described in more detail in the Run-time Parameters window.

Using the window

The Run-time Ramp Control window is displayed after you select Run from the TestWare-SX Execute Procedure window and the Run-time Ramp Control Process begins. The window continues to be displayed until you press End Process or select Reset from the TestWare-SX Execute Procedure window. See chapter 3 of the TestWare-SX Application manual for more information about running a test with TestWare-SX.

1. From the TestWare-SX Execute Procedure window select Run. This displays the Run-time Ramp Control window.

2. Review each of the parameters in the Run-time Ramp Control window. If changes are required make them now. If all entries are correct simply select Start Step or Start Auto (depending on the selected Mode)

Note *Any of the parameters are disabled can be enabled in the Run-time Ramp Control Design window.*

Single step mode

3. When the ramp ends and data collection is complete (if Remote Trigger was selected), the process status returns to Idle.

   You can select Start Step to begin the next ramp using the same parameters, or you can now make changes to any active parameter fields including comments (if Parameter Logging was selected). You can also change the mode from Single Step to Auto.

   Use the End Auto pushbutton to idle the process when the current ramp/pause pair is complete. This also puts you in the Single Step mode.

   You can enter comments now (if Parameter Logging was selected) and make changes to the parameter. You can select Start Step to begin a single ramp or switch the mode back to Auto and select Start Auto to begin repetitive ramps

Auto mode:

4. Select End Process to stop the process and close the Run-time Ramp Control window. To ensure all data has been collected before ending the process:

- **In Single Step Mode**—wait for status to return to Idle.
- **In Automatic Mode**—select End Auto and wait for the status to return to Idle.

Status

The following describe what can be displayed in the Status field:

- **Idle** — Is displayed whenever you are expected to do something to continue or end the test. Any active parameter fields can be changed while in Idle.
- **Ramping** — Indicates the command signal is changing as determined by the step size and ramp time. No changes to the parameter fields are allowed while in this state.
- **Collecting Data** — If Remote Trigger was selected in the design window, this indicates the process is pausing for the trigger delay and collection delay intervals. No changes to the parameter fields are allowed while in this state.
- **SX Holding** — Is displayed when the TestWare-SX Hold button is pressed which immediately suspends the command signal where it is. The Run-time Ramp Control process can be continued from where it stopped by pressing any TestWare-SX Run button. No changes to the parameter fields are allowed while in this state.
- **SX Stopped** — Is displayed when any TestWare-SX Stop button is pressed which immediately ends all ramps and delays. Any active parameter fields can be changed while in SX Stopped, including the Mode and Comments fields.
- **SX Error** — Is displayed when a TestWare-SX Error occurs. A reset is needed once the error has been corrected in order to restart the test. No changes to the parameter fields are allowed while in this state.

Trigger Data

When the Trigger Data button is used for an immediate trigger while the status is Ramping, the process will continue uninterrupted. Thus, despite an inactive Trigger Data button during the Collection Delay interval, an additional trigger will occur at the normal time following the completion of the ramp.

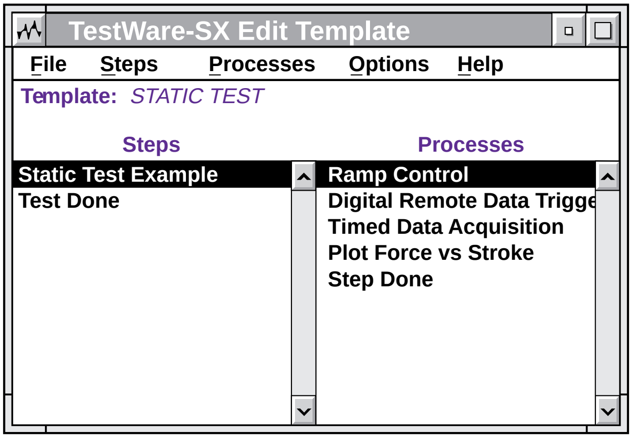
- If status is Ramping or Idle, selecting this button immediately sends a start trigger to the data collection process.
- If status is SX Stopped or SX Holding, the start trigger is issued and the message is displayed as soon as the test resumes after selecting Run.
- The button is inactive while the status is Collecting Data or SX Error, or if Hold was pressed while the status was Collecting Data.

The effects of multiple triggers on the data process and the remote device should be considered when using this button.

Example Static Test Template

The window below illustrates a simple example of a template for a static deflection test consisting of one step and four processes. The Run-time Ramp Control process (labeled Ramp Control) was described earlier in the manual. The Run-time Plotting process (labeled Plot Force vs. Stroke) is described later in this manual. The remaining processes in this template (Digital Output, labeled Digital Remote Data Trigger and Data Collection, labeled Timed Data Acquisition) and all other standard processes are described in the TestWare-SX Application manual.

An example template similar to this one is provided with your system software. You can review many of the process settings by clicking on File and selecting Print Preview.



Note *If the Ramp Control process is a trigger for the Step Done process when Remote Trigger is enabled, the test step will end prematurely when the first Remote Trigger occurs.*

If this happens, you will have to return to the Edit Template mode to deselect the Ramp Control process as a Step Done trigger and restart the test.

When designing your test, if you intend to use Remote Triggering, do not select Ramp Control as a Step Done trigger.

To use this example template, you must review and edit each design and parameter window to ensure they conform to your particular system configuration and test requirements.

You may also want to delete processes or add additional processes (e.g., another data acquisition process that is based on level crossing or additional run-time plot processes for each selected control channel).

Note *:The descriptions below of the Ramp Control and Digital Remote Data Trigger processes assume Remote Trigger was selected in the Ramp Control Design window and the Ramp Control process is the start trigger for the Digital Remote Data Trigger process.*

If Remote Trigger was not selected, no data collection from your data collection equipment can occur.

Processes Overview

- The Ramp Control process allows interactive ramp control for each selected control channel. This process begins at the start of the step and continues for the duration of the step.
- The Digital Remote Data Trigger Process begins each time it receives a signal from the Ramp Control process (during the pauses between ramps). The Digital Remote Data Trigger process then sends a timed pulse to start your data collection equipment (e.g. a data logger). This allows you to capture data from many channels of strain gages and external displacement transducers during the pauses in the test generated by the Ramp Control process.
- The Timed Data Acquisition process collects data from assigned feedback sensors (e.g. force transducers and displacement transducers) on each control channel. As the name implies, data acquisition will occur at a specific time interval determined by you. This process begins at the start of the step and continues (even during ramp pauses) for the duration of the step.
- The Plot Force vs. Stroke process provides a real time graphical display of the selected sensors feedback (in this case force and stroke) from one control channel. (Additional plot processes would have to be added to display other sensor inputs or inputs from other control channels.) This process begins at the start of the step and continues for the duration of the step.
- The Step Done process ends the step when a selected trigger occurs. When selecting a trigger for Step Done, do not use Ramp Control when Remote Trigger is enabled. Doing so will stop the test the first time Remote Trigger occurs. (Refer to the note on page 17.) Typically, the End Process button will be used to stop the step.

Index

Numerics

A

B

C

D

F

H

I

L

M

O

P

R

S

T

U

W