Model 793.10 MultiPurpose TestWare®

User Information and Software Reference

100-068-915 E

Copyright information

© 2003 MTS Systems Corporation. All rights reserved.

Trademark information

MicroConsole, MTS, RPC, Temposonics, and TestWare are registered trademarks of MTS Systems Corporation.

FlexTest, MPT, Station Builder, Station Manager, and TestStar are trademarks of MTS Systems Corporation.

Adobe is a registered trademark of Adobe Systems Inc.

Acrobat is a trademark of Adobe Systems Inc.

Lotus 1-2-3 is a registered trademark of Lotus Development Corporation.

Microsoft, Windows, and Windows NT are registered trademarks of Microsoft Corporation.

Software verification and validation

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

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

Publication information

Manual Part Number	Publication Date
100-068-915 A	November 2001
100-068-915 B	October 2002
100-068-915 C	November 2002
100-068-915 D	February 2003
100-068-915 E	June 2003

Contents

Chapter 4 Working with Processes 97

Chapter 5 Windows and Controls 123

Chapter 6 Process Descriptions 195

Appendix A Default Templates 449

Appendix B Excel File Converter 455

Appendix C Profiles 459

Preface

Safety first!

Before you attempt to use your MTS equipment in your test system, read and understand the *Safety* manual. Like an automobile, your test system is very useful—but if misused, it is capable of deadly force. You should not be afraid of your test system, but you should always maintain a healthy respect for it.

Improper installation, operation, or maintenance of MTS equipment in your test system can result in hazardous conditions that can cause severe personal injury or death, and damage to your equipment and specimen. Again, read and understand the *Safety* manual before you continue. It is very important that you remain aware of hazards that apply to your test system.

Other MTS manuals

In addition to this manual, you may receive additional MTS manuals in paper or electronic form.

If you have purchased a test system, it may include an *MTS System Documentation CD.* This CD contains an electronic copy of all of the MTS manuals that pertain to your test system, including controller manuals, hydraulic and mechanical component manuals, assembly drawings and parts lists, and operation and preventive maintenance manuals.

Contents

Conventions

The following paragraphs describe some of the conventions that are used in your MTS manuals.

Hazard conventions

As necessary, hazard notices are embedded in this manual. These notices contain safety information that is specific to the task to be performed. Hazard notices immediately precede the step or procedure that may lead to an associated hazard. Read all hazard notices carefully and follow the directions that are given. Three different levels of hazard notices may appear in your manuals. Following are examples of all three levels.

Note *For general safety information, see the Safety manual included with your system.*

Danger notices

Danger notices indicate the presence of a hazard which *will* cause severe personal injury, death, or substantial property damage if the danger is ignored. For example:

DANGER

High intensity light and dangerous radiation are emitted by class 3B lasers.

Viewing a class 3b laser directly or viewing it using optical instruments will cause immediate and severe injury.

Avoid eye or skin exposure to the laser beam. Ensure that all power to the laser is off before attempting any maintenance, service, or adjustment procedure.

Warning notices

Warning notices indicate the presence of a hazard which *can* cause severe personal injury, death, or substantial property damage if the warning is ignored. For example:

WARNING

Hazardous fumes can accumulate in the test chamber.

Breathing hazardous fumes can cause nausea, fainting, or death.

Ensure the chamber is properly ventilated before you open the chamber door or put your head or hands into the chamber. To do this, ensure the temperature controller is off and allow sufficient time for the ventilation system to completely exchange the atmosphere within the chamber.

Caution notices

Caution notices indicate the presence of a hazard which *will* or *can* cause minor personal injury, cause minor equipment damage, or endanger test integrity if the caution is ignored. For example:

CAUTION

This specimen can develop sharp edges as a result of testing.

Handling the specimen with unprotected hands can result in cuts and slivers.

Always wear protective gloves when you handle the specimen.

Other conventions

Other conventions used in your manuals are described below:

Notes

Notes provide additional information about operating your system or highlight easily overlooked items. For example:

Note *Resources that are put back on the hardware lists show up at the end of the list.*

Special terms

The first occurrence of special terms is shown in *italics*.

Illustrations

Illustrations appear in this manual to clarify text. It is important for you to be aware that these illustrations are examples only and do not necessarily represent your actual system configuration, test application, or software.

Electronic manual conventions

This manual is available as an electronic document in the Portable Document File (PDF) format. It can be viewed on any computer that has Adobe Acrobat Reader installed.

Hypertext links

The electronic document has many hypertext links displayed in a blue font. All blue words in the body text, along with all contents entries and index page numbers are hypertext links. When you click a hypertext link, the application jumps to the corresponding topic.

Technical Support

Start with your manuals

The manuals supplied by MTS provide most of the information you will need to use and maintain your equipment. If your equipment includes MTS software, look for README files that contain additional product information.

If you cannot find answers to your technical questions from these sources, you can use the internet, telephone, or fax to contact MTS for assistance. You can also fill out the Problem Submittal Form that is available on the MTS web site and in the back of many MTS manuals that are distributed in paper form.

Technical support numbers

MTS provides a full range of support services after your system is installed. If you have any questions about a system or product, contact MTS in one of the following ways.

MTS web site www.mts.com

The MTS web site gives you access to our technical support staff by means of a Problem Submittal Form and a Technical Support link.

- Problem Submittal Form:
  www.mts.com > Contact MTS > Problem Submittal Form
- Technical Support:
  www.mts.com > Contact MTS > Technical Support

E-mail:

info@mts.com

Telephone

HELPLine 800-328-2255
Weekdays 7:00 A.M. to 6:00 P.M.,
Central Time

Fax

952-937-4515
Please include an MTS contact name if possible.

Before you contact MTS

MTS can help you more efficiently if you have the following information available when you contact us for support.

Know your site number and system number

The site number contains your company number and identifies your equipment type (material testing, simulation, and so forth). The number is usually written on a label on your MTS equipment before the system leaves MTS. If you do not have or do not know your MTS site number, contact your MTS sales engineer.

Example site number: 571167

When you have more than one MTS system, the system number identifies which system you are calling about. You can find your job number in the papers sent to you when you ordered your system.

Example system number: US1.42460

Know information from prior technical assistance

If you have contacted MTS about this problem before, we can recall your file. You will need to tell us the:

- MTS notification number
- Name of the person who helped you

Identify the problem

Describe the problem you are experiencing and know the answers to the following questions.

- How long has the problem been occurring?
- Can you reproduce the problem?
- Were any hardware or software changes made to the system before the problem started?
- What are the model and serial numbers of the suspect equipment?

Know relevant computer information

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

- Manufacturer's name and model number
- Operating software type and service patch information. Examples:
  - Windows XP Service Pack 1 (SP1)
  - Windows 2000 Service Pack 3 (SP3)
  - Windows NT 4.0 Service Pack 7 (SP7)
- Amount of system memory. Example: 640 MB of RAM.
- Amount of free space on the hard drive in which the application resides. Example: 11.2 GB free space, or 72% free space.
- Current status of hard-drive fragmentation. Example: 3% total fragmentation.

Know relevant software information

For MTS software application problems, have the following information available.

- The software application's name, version number, build number, and if available, software patch number. This information is displayed briefly when you launch the application, and can typically be found in the "About" selection in the "Help" menu.

  Example: Station Manager, Version 3.3A, Build 1190, Patch 4

- The same information for other MTS software included with your system
- Names of other non-MTS applications that are running on your computer, such as screen savers, keyboard enhancers, print spoolers, and so forth

If you contact MTS by phone

Your call will be registered by a HELPLine agent if you are calling within the United States or Canada. Before connecting you with a technical support specialist, your agent will ask you for your site number, name, company, company address, and the phone number where you can normally be reached.

Identify system type

To assist your HELPLine agent with connecting you to the most qualified technical support specialist available, identify your system as one of the following types:

- Electromechanical materials test system
- Hydromechanical materials test system
- Vehicles test system
- Vehicles component test system
- Aero test system

Be prepared to troubleshoot

Prepare yourself for troubleshooting while on the phone.

- Call from a telephone close to the system so that you can try implementing suggestions made over the phone.
- Have the original operating and application software media available.
- If you are not familiar with all aspects of the equipment operation, have an experienced user nearby to assist you.

Write down relevant information

Prepare yourself in case we need to call you back.

- Remember to ask for the notification number.
- Record the name of the person who helped you.
- Write down any specific instructions to be followed, such as data recording or performance monitoring.

After you call

MTS logs and tracks all calls to ensure that you receive assistance and that action is taken regarding your problem or request. If you have questions about the status of your problem or have additional information to report, please contact MTS again.

Problem Submittal Form in MTS manuals

In addition to the Problem Submittal Form on the MTS web site, there is also a paper version of this form (postage paid) in the back of many MTS manuals. Use this form to communicate problems you are experiencing with your MTS software, hardware, manuals, or service. This form includes check boxes that allow you to indicate the urgency of our problem and your expectation of an acceptable response time. We guarantee a timely response—your feedback is important to us.

Chapter 1

Introduction

Overview

The Model 793.10 MultiPurpose TestWare application is an advanced test designer available to MTS Series 793 Controllers.

With MultiPurpose TestWare, you can:

- Create complex test procedures that include command, data acquisition, event detection, and external control instructions.
- Generate programs based on profiles created with a text editor application, a spreadsheet application, or the Model 793.11 Profile Editor application.
- Acquire and monitor real-time trend or fatigue data.

Using MPT with Other MTS Series 793 Applications

The MPT application is not a stand-alone application. It must be used in conjunction with other applications in the Model 793.00 System Software bundle.

Before you can start the MPT application, you must:

1. Create a station configuration file with the Station Builder application.
2. Start the Station Manager application and open the station configuration file.
3. Select MPT from the **Applications** menu of Station Manager.

.

User Interface

MPT control panel

The MPT control panel appears in the application area of Station Manager, as shown.

MPT Procedure Editor button

Displays the **Procedure Editor** and **Process Palette**, as shown on the next page.

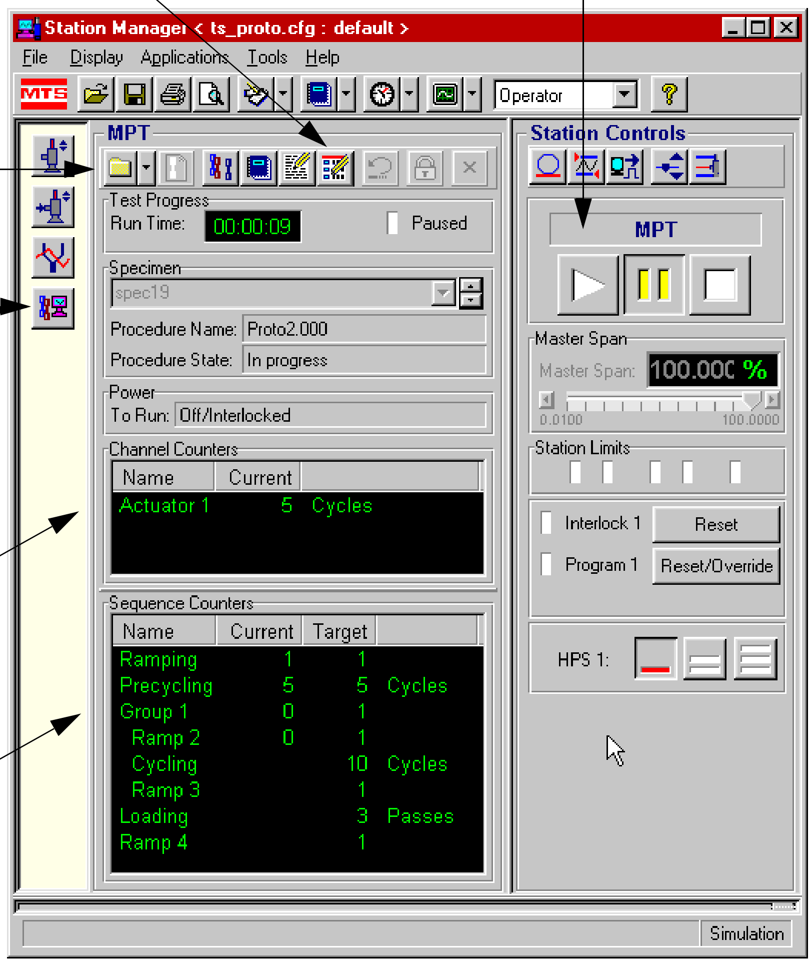
Station Controls

You execute MPT procedures with the run, stop, and hold controls.

MPT toolbar

MPT application button

When you've launched MPT this button will appear in the applications control panel. You can quickly select among launched applications with the buttons in this panel.



Channel Counters

Channel Counters display the cumulative count of segments applied to specific channels.

Sequence Counters

Sequence counters display current and target counts for processes. Individual process counters are arranged according to the process icons on the **Procedure Editor**. Group process counters are indented to show nesting.

Procedure Editor and Process Palette

The **Procedure Editor** contains a table on which you build MPT test procedures by selecting, assigning parameters to, and sequencing individual test processes. Test processes represent individual test activities. The processes available are displayed on the **Process Palette**.

Procedure Editor

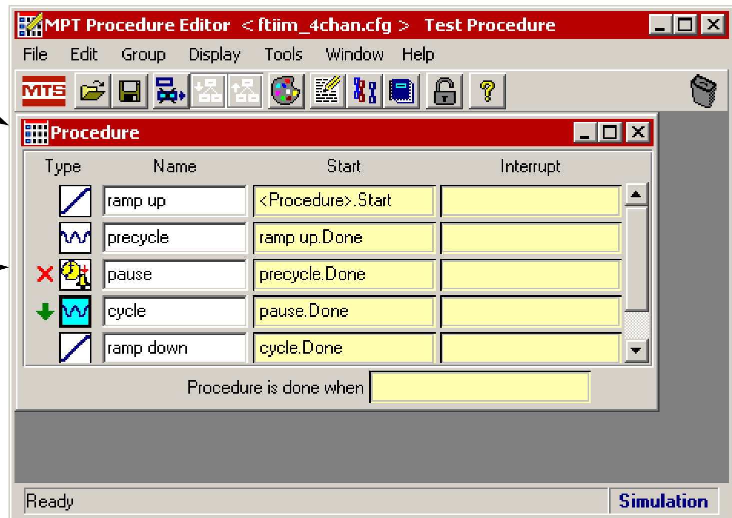
The **Procedure Editor** contains a table on which you build MPT test procedures.

Status and progress indicators for processes

A blue arrow pointing toward a process (not shown) indicates it is executing. This lets you watch the test's progress.

A red "X" next to a process indicates it has been disabled.

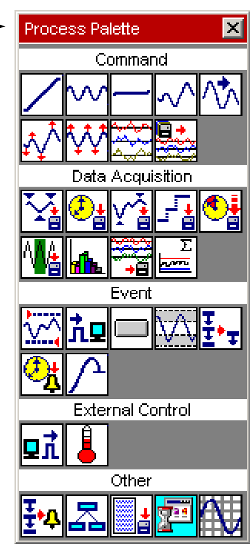
A green arrow pointing downward next to a process indicates it will start and then issue its done signal immediately, without performing its programmed activity.



Process Palette

Individual processes are arranged on the palette by type.

You can double-click a process on the **Process Palette** to copy an instance of it to the table on the **Procedure Editor**, or you can use the drag-and-drop method.



MPT Test Procedures

An MPT test procedure, or procedure for short, contains two types of information:

- Process information, which defines your processes (command, data acquisition, signal monitoring, and so forth) and the sequence in which each process executes.
- MPT application information, which includes test unit and dimension assignments, the behavior of hold and resume functions, the data file format, and so on.

Creating procedures

You create procedures by linking together processes that represent individual test activities, such as ramping the actuator or acquiring peak/valley data.

For information on how to create a procedure, see "Creating a Sample Procedure" on page 35. For information about the various MPT processes, see "Process Descriptions" on page 195.

Running procedures

You run your procedures with the run, stop, and hold controls on the **Station Controls** panel. For more information about how to run procedures, see "How to Run a Procedure" on page 25.

Introduction

Chapter 2

Working with Procedures

Common Tasks

How to Start MultiPurpose TestWare

Before you can start the MultiPurpose TestWare application, you must start the Station Manager application and open a station configuration file.

Note *If the Station Manager application is already started, proceed to Step 3.*

1. Start the Station Manager application:

   Path: Start > Programs > MTS FlexTest (or TestStar) > Station Manager

2. Select the desired station configuration (.cfg) file using the **Open Station** window.

3. Start the MultiPurpose TestWare application:

   Path: Station Manager> Applications > MultiPurpose TestWare

   When you select **MultiPurpose TestWare** from the Station Manager's **Applications** menu, the MPT control panel will be displayed. You can run existing procedures from the MPT control panel. If you want to edit or create new procedures, you need to access the **MPT Procedure Editor**. For more information, see "MPT Procedure Editor" on page 137.

How to Display the MPT Procedure Editor

On the MPT control panel toolbar, click .

How to Create a New Procedure

When you start MPT a new (undefined) procedure is automatically created. To create a new procedure any time MPT is active, from the **MPT Procedure Editor**, select the **File** menu, then click **New Procedure**.

Note *If you would like to review a sample test design, see* *"Creating a Sample Procedure" on page 35.*

How to Open a Procedure

1. From the MPT control panel (or from the **MPT Procedure Editor**), click the **Open Procedure** icon. Alternately, from the **MPT Procedure Editor**, click the **File** menu, then click **Open Procedure** to display the **Open Procedure** window.

2. Double-click the desired procedure, or click on the desired procedure once, then click the **Open** button.

How to Save a Procedure

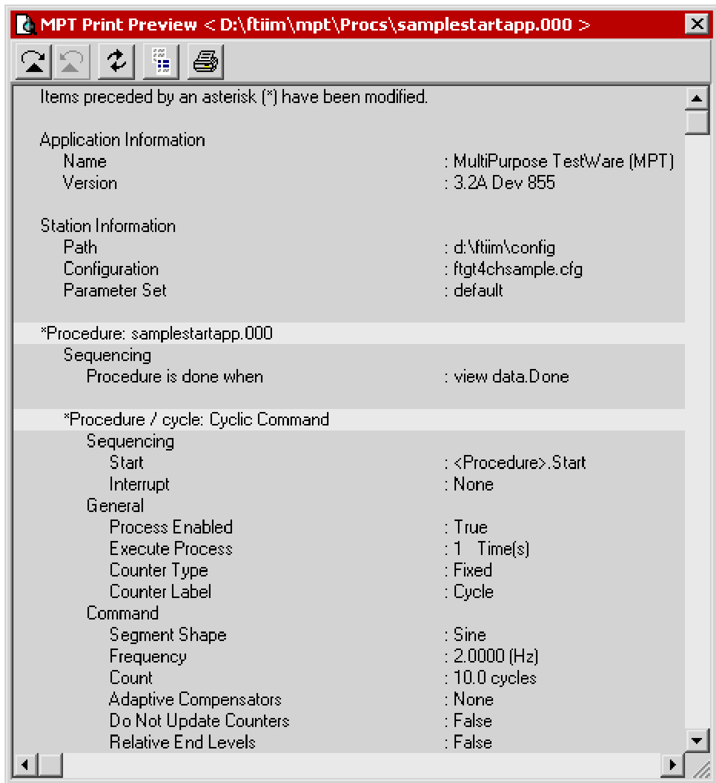
From the **MPT Procedure Editor**, click the **Save Procedure** icon, or click the **File** menu and select the desired save option.

Alternately, from the MPT control panel, on the toolbar, click the file icon's pull down menu, then select the desired save option.

Note *MultiPurpose TestWare automatically appends ".000" to the file name you enter. If you do not want to include the .000 extension, type a period (.) at the end of your file name.*

How to Preview a Procedure

From the **MPT Procedure Editor**, select the **File** menu, then click **Print Preview** to view your procedure on-screen. Click **Print to File** to save the procedure to a text file.



NOTE *Items preceded by an asterisk (*) have been modified since the procedure file was open.*

How to Print a Procedure

From the **MPT Procedure Editor**, select the **File** menu, then click **Print Procedure**.

How to Run a Procedure

Running a procedure begins with station preparation and physical specimen installation, both tasks that you accomplish with the Station Manager application.

The following steps assume you have already done the following:

- Created a configuration file with Station Builder, and tuned control channels and calibrated sensor/conditioner pairs
- Opened Station Manager and selected the configuration file
- Applied hydraulic pressure to the station
- Installed the physical specimen into the test fixturing
- Set station limits
- Set up readout devices to monitor station signals
- Opened MultiPurpose TestWare
- Selected a procedure

1. Create or select an MPT specimen

Before you can run a procedure, you have to either create a new or select an existing MPT specimen for the test.

The term “MPT specimen” refers to a directory of information associated with a specific execution of a procedure—not the physical specimen being tested.

Your test data, as well as the procedure you ran, messages generated during the test, recovery information, etc., will be located in the MPT specimen you create for this test.

A. To create a new MPT specimen:

On the MPT control panel toolbar, click  and a new MPT specimen labeled **spec***XX* (“spec01,” “spec02,” etc.) appears in the list box. Rename the new MPT specimen as desired.

B. To select an existing MPT specimen:

On the MPT control panel toolbar, select an existing MPT specimen from the **Specimen** list box.

2. Start your test.

On the **Station Controls** panel click the run button to start the test. Look at and listen to your test station after you start the test. If everything seems normal, you can let your test run to completion.

Note *After the completion of a test you have to press the reset button or create a new MPT specimen to run subsequent tests.*

Working with Counters

When you create a procedure with MPT, you define the parameters of each process in the procedure, including counter parameters.

In procedures, counters are linear tools used to indicate test progress and facilitate recovery. It is important to understand counters and to realize that how you define a counter can affect your test outcome and, as a result, the data that is generated from the test.

Note *Counters are not the only tools provided by the MPT application to indicate test progress. The* ***MPT Procedure Editor*** *displays special symbols when individual processes are active. Refer to "Procedure Table" on page 148 for more information about process indicators.*

About counters

The various types of counters available for procedures are displayed on the MPT control panel, provided the MPT control panel has been configured to display them.

The following paragraphs provide additional information about how you control whether they appear on the MPT control panel, and what effect they have on your test and your test data. Refer to "MPT Control Panel" on page 126 for more information.

Types of counters

The most prominent counters on the MPT control panel are the **Run Time** counter, the **Channel Counters**, and the **Sequence Counters**.

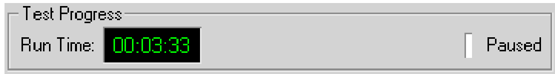
There are also specialized counters that display only in association with specific types of processes, such as:

- Profile counters, which display only with the Profile Command and Profile with ALC processes
- Status counters, which display only with the Time History Input, Time History Output, and Road Surface Output processes

Refer to "MPT Control Panel" on page 126 for more information about the **Run Time** display and the **Channel Counters** and **Sequence Counters** panels.

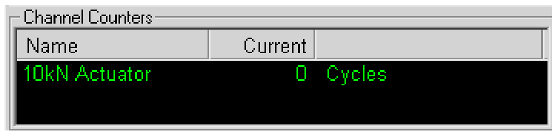
Run time counter

The **Run Time** counter displays the elapsed time of the current test since the last time it was reset.



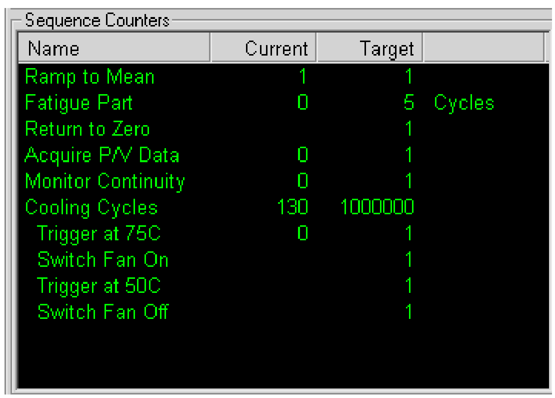
Channel counters

Channel Counters display a cumulative count of segments or cycles that have been applied to a specific channel since the start of the test.



Sequence counters

Sequence Counters display the progress of individual processes within the test. They can be especially useful for procedures that contain nested groups, because you can configure them to show test progress within the nested structure of the procedure.



This sample procedure includes a group process labeled “Cooling Cycles,” which contains four nested processes (with indented label names, “Trigger at 75C,” etc.)

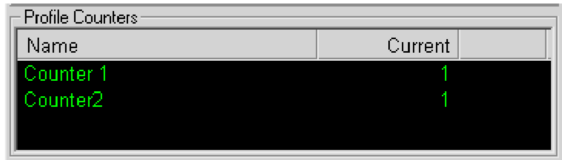
Name displays the assigned process label

Current displays the current value of the counter

Target displays the assigned number of times the process should execute (or repeat),

Profile counters (Profile and Profile with ALC processes only)

Profile Counters display counters that increment according to special markers that profile designers put in profiles.



Status counters (Time History Output, Time History Input, and Road Surface processes only)

Status counters display **Elapsed Time (or Distance)** and **Total Time** (or **Total Distance**) counters, and a **Percent Completed** status bar.

Elapsed Time displays the time since the process started. **Distance** displays the distance traveled since the process started, which is calculated from the **Vehicle Speed** entry on the process' **Status** tab.

Total Time displays the anticipated total time required to complete the process (this does not include any other processes that may be part of the test). **Total Distance** displays the anticipated total distance required to complete the process, calculated from the **Vehicle Speed** entry on the process' **Status** tab.

The **Percent Completed** status bar displays the percentage relationship of the elapsed time (or distance) versus the total time (or distance) for the process (the status bar updates according to the update rate of the test, typically every 10 seconds or so).

Displaying Counters and Status Panels

You can choose to show or hide the various types of counters and status panels on the MPT control panel when running tests.

Note *The Run Time Counter, Channel Counters, and Sequence Counters display by default. For information about changing default settings, see "Default Templates" on page 449.*

How to display Channel and Sequence Counters

To display **Channel Counters** and **Sequence Counters**, open the **MPT Options Editor,** select the **Control Panel** tab, and locate the **Counters** panel. Check the boxes as desired to show or hide the counters and to choose their display format. By default these counters are displayed.

To display the **Run Time** counter, open the **MPT Options Editor,** select the **Control Panel** tab, and locate the **Test Progress** panel. Check the box as desired to show or hide the **Run Time** counter.

How to display Status panels

To display **Status** panels, open the relevant process on the procedure table (Time History Input, Time History Output, and Road Surface Output) and select the **Status** tab. Check the **Show Status Panel at Run Time** box as desired. By default status panels are not displayed.

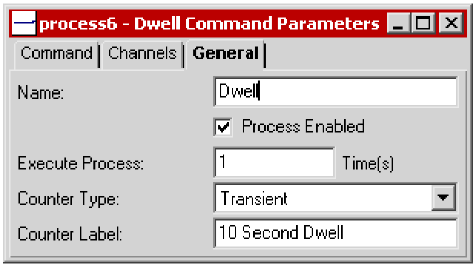
Note *If you enable the **Show Status Panel at Run Time** box, you will have the option of having the status panel display the distance traveled by the vehicle, for which you enter a nominal vehicle speed for the distance calculations.*

How to display Profile Counters

To display profile counters, you must include a Profile Command or Profile with ALC process in your procedure that uses a profile that includes the keyword "**Action**." For more information, see "Action and Counter Syntax" on page 465.

How to display individual processes on the Sequence Counters panel

The Sequence Counters panel displays by default, but each counter that populates the **Sequence Counters** panel must be enabled for display individually. To do this you must open each process on the procedure table to display its **Parameters** window, select the **General** tab, then select the desired **Counter Type** option.



The **Counter Type** options are:

None—The counter label and counter are not displayed on the **Sequence Counters** panel.

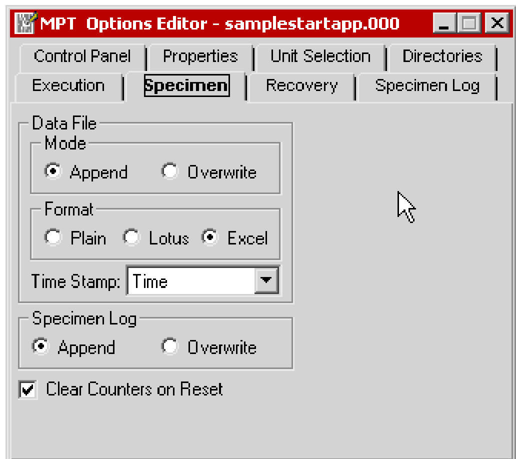
Transient—The counter label and counter are only displayed on the **Sequence Counters** panel when the process is active. When the process ends, transient counters disappear.

Fixed—The counter label and counter are displayed on the **Sequence Counters** panel for the duration of the test.

Using the Clear Counters on Reset Function

The **Clear Counters on Reset** check box (found on the **Specimen** tab of the **MPT Options Editor**) allows you to choose whether run time, channel counters, and profile counters:

- Clear when you reset the test (default setting), or
- Continue to increment after reset



Note ***Sequence Counters** are always cleared on reset as this counter is used in the recovery process.*

How to reset counters manually

If the **Clear Counters on Reset** control is disabled and you want to manually reset counters, you can:

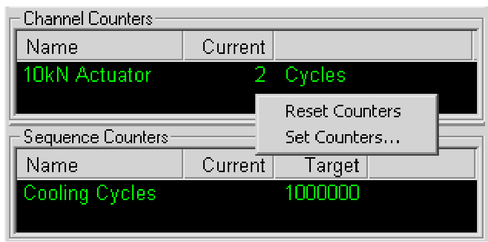
- Select a new specimen (before resuming or between tests), or
- Use the right-mouse click menu to reset each type of counter individually (applies to run time counters, channel counters, and profile counters)

How to use the right-mouse menu for run time counters

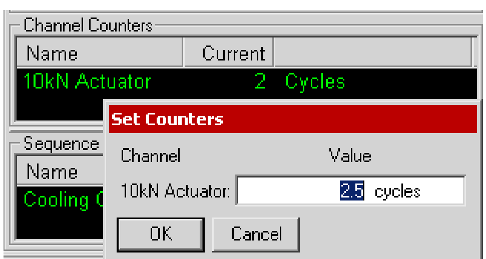
On the MPT control panel, position the cursor over the run time counter (ensuring the procedure is locked and reset), and click the right-mouse button. The **Reset** menu appears, which allows you to reset the counter.

How to use the right-mouse menu for channel counters

On the MPT control panel, position the cursor in the channel counter panel (ensuring the procedure is locked and reset) and click the right-mouse button. The **Reset** menu appears, which includes **Reset Counters** and **Set Counters...** selections.



The **Reset Counters** selection allows you to manually reset the channel counters to zero.



The **Set Counters...** selection displays a window that allows you to manually set counter values.

How to use the right-mouse menu for profile counters

On the MPT control panel, position the cursor over the profile counter panel at any time the counters are active and click the right-mouse button. The **Reset Counters** menu appears, which allows you to manually reset the counters to zero.

Note *The procedure does not have to be in the reset state to use the **Reset Counters** button for profile counters. You can reset profile counters on-the-fly, while the test is running.*

Creating a Sample Procedure

You create procedures by selecting the processes required to accomplish your test objectives from the **Process Palette** and placing them on the procedure table. Next you assign parameters and sequence information to each process. This section describes how to create a sample procedure, and will give you a good sense of how all the elements of a procedure fit together.

If you are new to MultiPurpose TestWare it is good practice to set your control software to the simulation mode (using the Demo System Loader) and follow along, using one of the default configuration files included with your system, or one that you have created yourself.

You may not be able to create the sample procedure precisely as shown because your system may be supplied with different hardware resources than used to create the sample test.

Even so, if you follow along and create a less complex version of the sample procedure, or even if you just review the pages without using your software, you will still benefit by learning the fundamentals of creating procedures.

Note *The hardware resources and label names provided in this section are for illustration purposes only. Your hardware resources and label names will vary with the .hwi file and station configuration file you use.*

The following tasks are described in this section:

Task 1 Plan Your Test

To plan your test, you should:

- Identify test objectives and significant test information.
- Review the test environment.

The following sample test objectives are used throughout this section to show how they affect the selection, triggering, and definition of individual processes in the sample test.

1. **Identify the test objectives.**

   For this sample test, we want to:

   A. Ramp to 3000 N in force control.

   B. Apply a sinusoidal force input of ±1000 N at 1 Hz, for 100 cycles to fatigue the physical specimen.

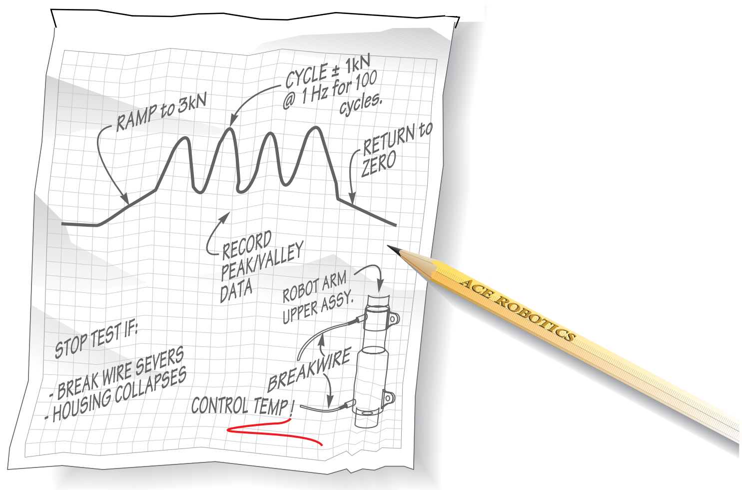
   C. Return to 0 N in force control.

   During the test, we also want to:

   A. Acquire peak/valley data while the physical specimen is being fatigued.

   B. Activate and deactivate an external fan to keep the physical specimen’s temperature between 50–75°C.

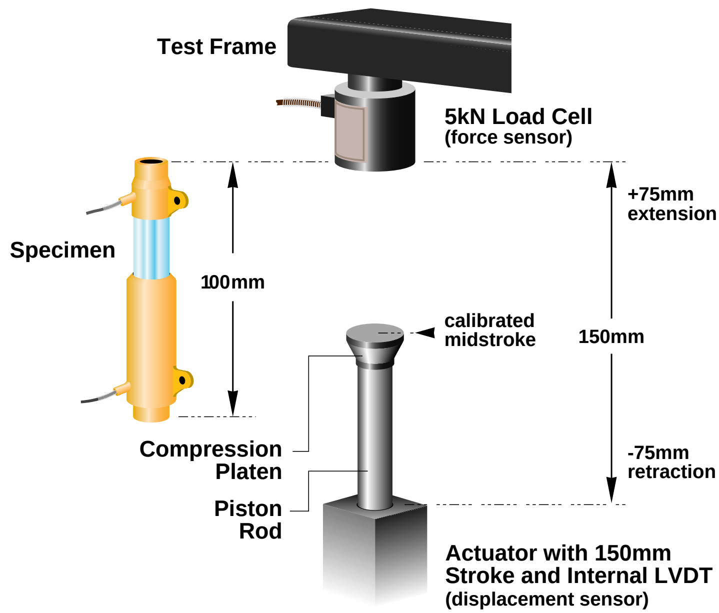
C. Monitor the physical specimen's internal breakwire for continuity, and stop the test if it severs, or if the detected error level indicates imminent failure of the specimen.



The sample test design as it may be sketched out.

2. Review the test environment.

The test environment includes our physical specimen, test frame, and sensors. Their dimensions and capacities will be important considerations in our test setup. The test environment for our sample test is as shown.



Task 2 Determine Which Processes are Needed

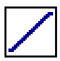
A process is a “mini-program” that performs a specific test activity. You use processes like building blocks to construct a procedure.

Note *For detailed information about MPT processes, see* *“Process Descriptions” on page 195.*

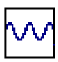
To complete this task, we must determine which processes on the **Process Palette** are needed to accomplish our test objectives.

1. **Determine which processes are needed to ramp and fatigue the physical specimen.**

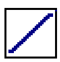
   This test requires three command processes:



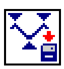
   - A **Segment Command** process to ramp the applied force to the mean level from which cyclic forces will be applied.



   - A **Cyclic Command** process to apply cyclic forces to the physical specimen to fatigue it.

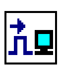


   - A **Segment Command** process to return the applied force to zero.



2. **Determine which processes are needed to acquire data.**

   This test requires a **Peak/Valley Data Acquisition** process to acquire peak/valley data as the physical specimen is being fatigued.



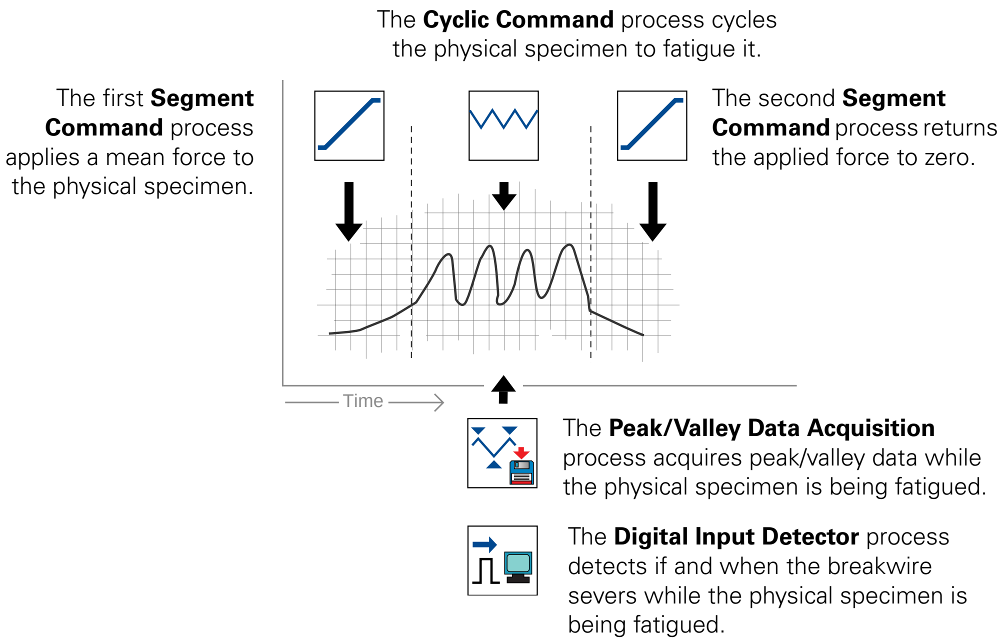
3. **Determine which processes are needed to monitor continuity.**

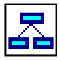
   This test requires a **Digital Input Detector** process to monitor the physical specimen continuity via a breakwire.

Note *In this example the Station Manager’s Error detectors, rather than an MPT process, are used to monitor structural failure of the physical specimen.*

How these processes will work

The following illustrates Steps 1 - 3.

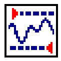




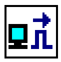
4. Determine which processes are needed to control the fan.

This test requires a Group process to provide the logic for switching the cooling fan on and off.

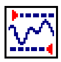
The Group process will contain four individual processes:



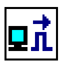
– A Data Limit Detector process to monitor the physical specimen's temperature and trigger the next process when the temperature reaches 75°C.



– A Digital Output process to turn the fan on to cool the physical specimen.



– Another Data Limit Detector process to monitor the physical specimen's temperature and trigger the next process when the temperature drops to 50°C.

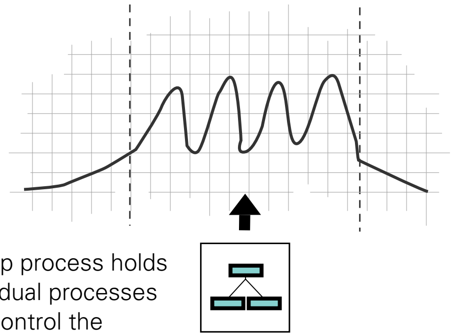


– Another Digital Output process to turn the fan off.

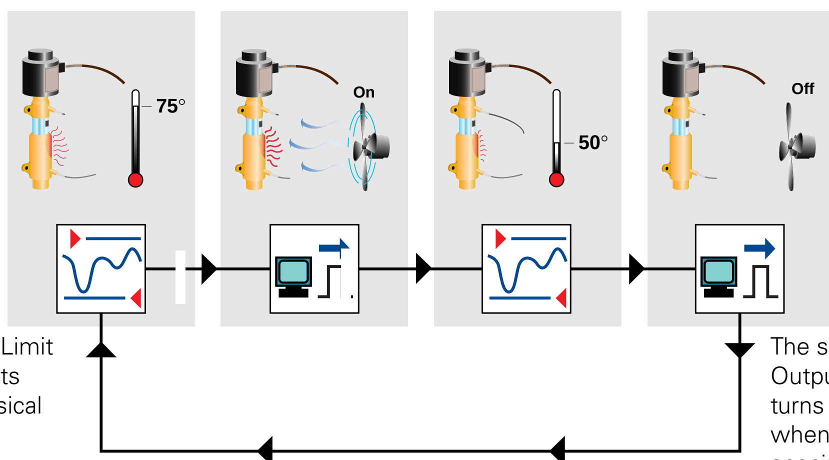
Finally, to create continuous fan control during the test, we should specify that when the fourth process in the group is done, the first process begins again.

Example of a group process at work

The following illustrates Step 4.



The Group process holds the individual processes used to control the cooling fan.



The first Data Limit process detects when the physical specimen's temperature reaches 75°C.

The first Digital Output process sends out a pulse that turns the fan on when the physical specimen's temperature reaches 75°C.

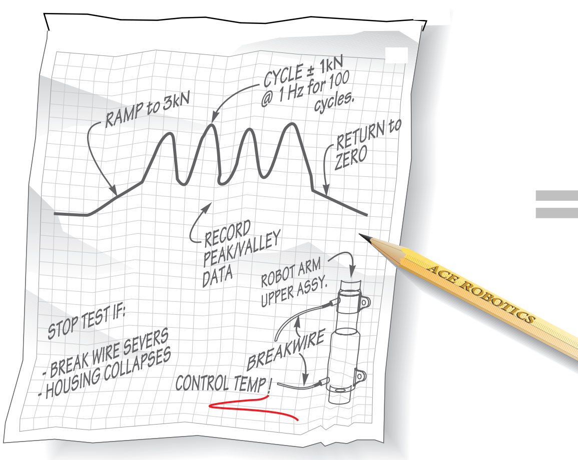
The second Data Limit process detects when the physical specimen's temperature drops to 50°C.

The second Digital Output process turns the fan off when the physical specimen's temperature drops to 50°C, then the Group process starts over with the first Data Limit process.

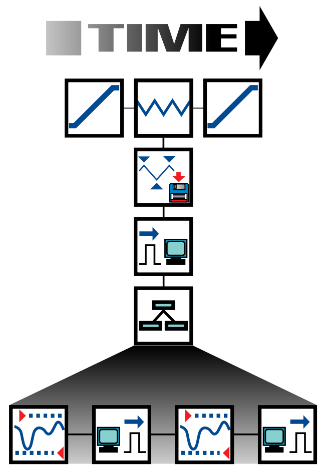
Put it all together

All of the processes required to perform the activities in our test sketch are shown below in the process diagram.

Note *The process diagram below shows the procedure as a function of time (from left to right), because it is easy to visualize that way. On the **MPT Procedure Editor** you assign the order in which processes run by assigning start and interrupt triggers to each process. Because of this, the order of the processes on the **MPT Procedure Editor**, which are arranged in a single column, may not determine the order in which they run.*



We converted the information in the sample test sketch into the process diagram illustrated to the right.



This diagram illustrates the sample test sketch in terms of MPT processes. This graphic is used in the margins of the following sections to illustrate adding and sequencing processes.

Choosing a method to build your procedure

Now that you have an understanding of which MPT processes are necessary to accomplish the objectives of the sample test scenario, you need to choose how to build the sample procedure.

Building a procedure involves the following basic activities:

- Adding the necessary processes to the procedure table
- Sequencing the processes with start and interrupt triggers
- Setting the parameters of the individual processes
- Selecting procedure options

MultiPurpose TestWare is versatile—there are many ways to accomplish these basic activities to build the same procedure.

The method used to build the sample procedure

For simplicity, the sample procedure created in the following pages uses this basic activity flow for all of its processes from beginning-to-end before going on to the next activity. For instance, the first basic activity in Task 3 is adding all of the processes required for the test to the procedure table, including all command, data acquisition, detection, and digital output processes.

Next, in Task 4, all processes are sequenced by assigning start and interrupt triggers to them. Then the parameters are set for each process, and finally, options are selected for the procedure.

The end result is a sample procedure that accomplishes the test objectives of the sample test scenario.

An alternate method—building and testing functional layers

An alternative method for building procedures involves separating the procedure into functional layers, and adding, sequencing, and setting the parameters for all of the processes within each layer before building the next layer.

As an exercise, you could build the sample procedure this way.

For instance, rather than adding all the processes required for the test to the procedure table, at first add only the command processes. This defines the command layer of the procedure. Next, sequence and set the parameters for the command processes, and run the procedure to ensure the command layer is functional.

Next, add the data acquisition layer to the command layer. To do this, add, sequence, and set the parameters for the data acquisition process. Then run the procedure to ensure both layers are functional. You can check the data stored in the MPT specimen (created when you run the test) to ensure you are getting the type of data you expect.

Finally, add, sequence, and set the parameters of the processes used to control the cooling fan.

Note *The advantage of building procedures in functional layers rise with the complexity of the procedure. Using this method in reverse is also an effective method for debugging procedures.*

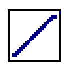
Task 3 Add Processes to the Procedure Table

To complete this task, we will:

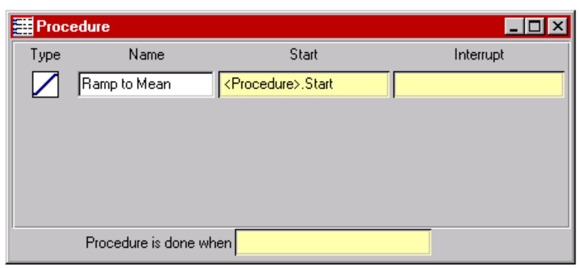
- Add processes to the **Procedure** table.
- Add processes to the **Group** table.

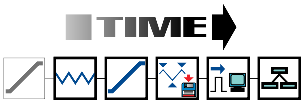
This step adds the first process to the **Procedure** table.

1. Add the first process to the Procedure table.

A. Drag the  icon from the **Process Types** palette to the **Procedure** table.

B. Click the **Segment Command** process name in the **Procedure** table and rename it “**Ramp to Mean**”, as shown.





This step adds the remaining processes to the **Procedure** table.

2. Add the remaining processes to the Procedure table.

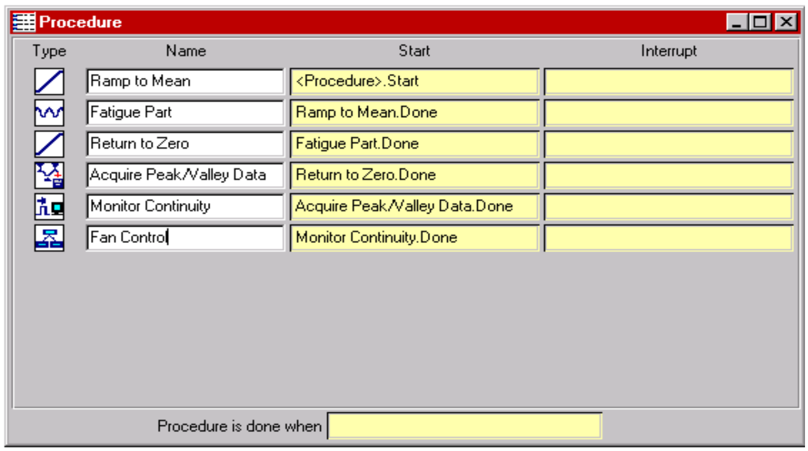
A. Add a Cyclic Command process to the **Procedure** table and name it **Fatigue Part**.

B. Add a Segment Command process to the **Procedure** table and name it **Return to Zero**.

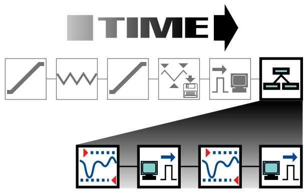
C. Add a Peak/Valley Acquisition process and name it **Acquire Peak/Valley Data**.

D. Add a Digital Input Detector process and name it **Monitor Continuity**.

E. Add a Group process and name it **Fan Control**.

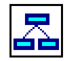


This is how our Procedure table should look when we have completed this step.



This step adds processes to the **Group** table (to control the fan).

3. Add the necessary processes to the Group process window.

A. Click the Group process icon  on the MPT Procedure Editor.

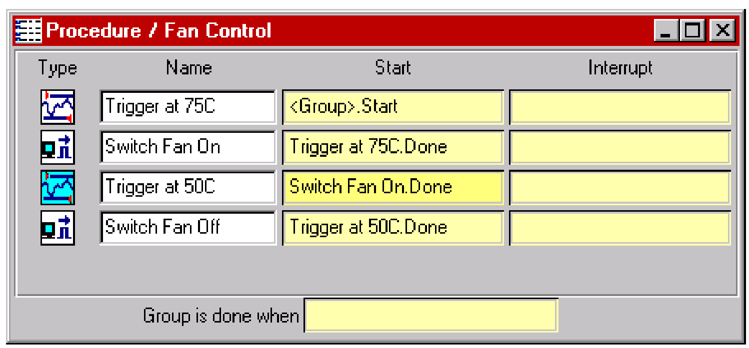
B. Click  on the MPT Procedure Editor toolbar. This is how you display the group table associated with the Group process icon.

C. Add a Data Limit Detector process to the **Procedure/Fan Control** window and name it **Trigger at 75C**.

D. Add a Digital Output process and name it **Switch Fan On**.

E. Add a Data Limit Detector process and name it **Trigger at 50C**.

F. Add a Digital Output process and name it **Switch Fan Off**.



This is how our Group table should look when we have completed this step.

Task 4 Sequence Your Processes

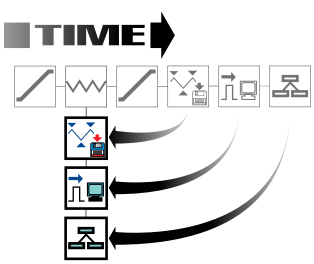
To complete this task, we will:

- Select **Start** and **Interrupt** triggers for each process on the **Procedure** and **Group** table.
- Select a **Procedure is done when** trigger for the **Procedure** table.
- Select a **Group is done when** trigger for the **Group** table.

This task determines the sequence in which the processes execute. With the exception of the first process in the procedure, the default **Start** trigger for each process is the done trigger generated by the previous process. The default **Interrupt** trigger is blank.

> ***Note*** *For information on how to choose specific **Start** and **Interrupt** triggers for individual processes, see "Sequencing Processes" on page 104.*

Note *There is a slight time delay between processes that start with a common trigger signal (for example, two processes using the <Procedure.Start> trigger signal). The process positioned the highest on the Procedure or Group table starts first.*



This step changes the default sequence of some of the processes in the procedure.

1. Specify Start triggers for each process in the Procedure table.

A. If you are still in the Group table, click  on the MPT toolbar to return to the main Procedure table.

B. Observe that the **Start** trigger for the **Ramp to Mean** process is **<Procedure>.Start**. This is the default start trigger, and will make this process start when the procedure starts.

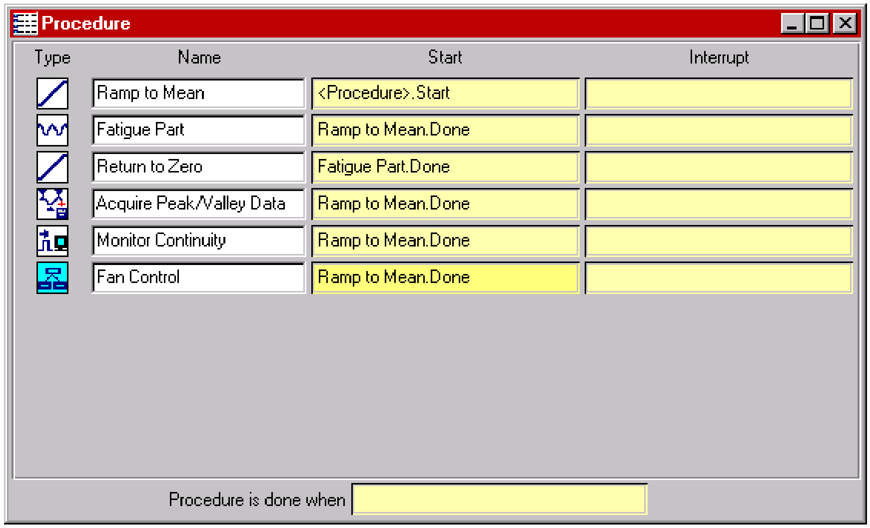
C. Set the **Start** trigger for the **Fatigue Part** process to **Ramp to Mean.Done**. This is the default.

D. Set the **Start** trigger for the **Return to Zero** process to **Fatigue Part.Done**. This is the default.

E. Change the **Start** trigger for the **Acquire Peak/Valley Data** process to **Ramp to Mean.Done**, which will make this process and the **Fatigue Part** process start almost simultaneously.

F. Change the **Start** trigger for the **Monitor Continuity** process to **Ramp to Mean.Done**, which will make this process and the **Fatigue Part** process start almost simultaneously.

G. Change the **Start** trigger for the **Fan Control** process to **Ramp to Mean.Done**, which will make this process and the **Fatigue Part** process start almost simultaneously.



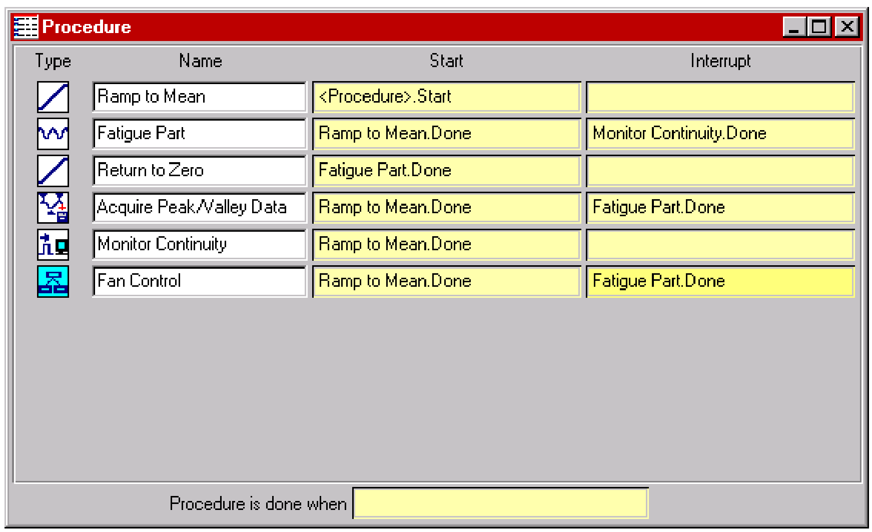
This is how our **Procedure** table should look when we have completed this step.

2. Specify the Interrupt triggers for processes in the Procedure table.

A. Change the **Interrupt** trigger for the **Fatigue Part** process to **Monitor Continuity.Done**. This ends the **Fatigue Part** process if the **Monitor Continuity** process triggers due to a severed breakwire.

B. Change the **Interrupt** trigger for the **Acquire Peak/Valley** process to **Fatigue Part.Done**. This ends the **Acquire Peak/ Valley Data** process when the **Fatigue Part** process finishes. In other words, data acquisition will stop when cycling stops.

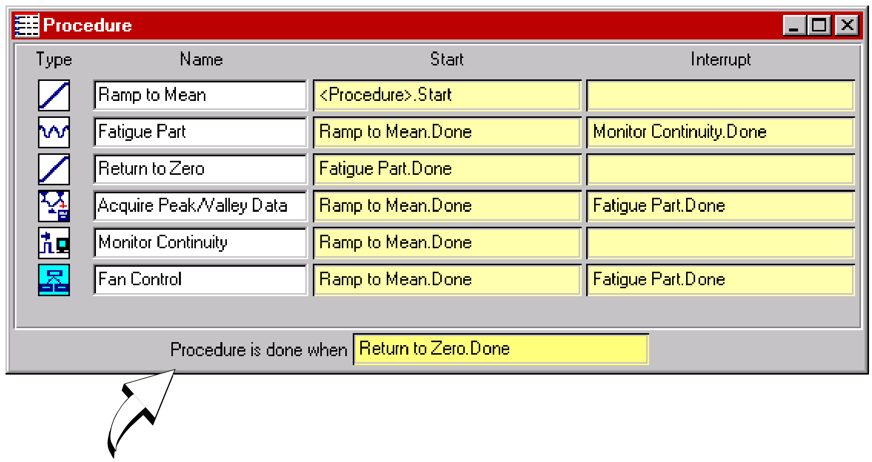
C. Change the **Interrupt** trigger for the **Fan Control** process to **Fatigue Part.Done**. This ends the **Fan Control** process when the **Fatigue Part** process finishes.



This is how the Procedure table should look when you have completed this step.

3. Specify a "Procedure is done when" trigger.

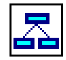
Set the **Procedure is done when** trigger to **Return to Zero.Done**. This ends the procedure when the **Return to Zero** process ends.



This is the **Procedure is done when** control. When the trigger we select for this control is received, the procedure terminates. In this case, we are choosing the **Return to Zero.Done** trigger.

This means the procedure will terminate when the **Return to Zero** process generates a done trigger.The procedure can terminate prematurely if one of the defined interrupts occur, i.e., if the **Fatigue Part** process receives a done trigger from the **Monitor Continuity** process.

4. Specify Start triggers for each process in the Group table.

A. Click the Group process icon  on the MPT Procedure Editor.

B. Click  on the MPT Procedure Editor toolbar.

C. Observe that the **Start** trigger for the **Trigger at 75** process is **Fan <Group>.Start**, which will make this process start when the **Group** process starts.

D. Observe that the **Start** trigger for the **Switch Fan On** process is **Trigger at 75C.Done**.

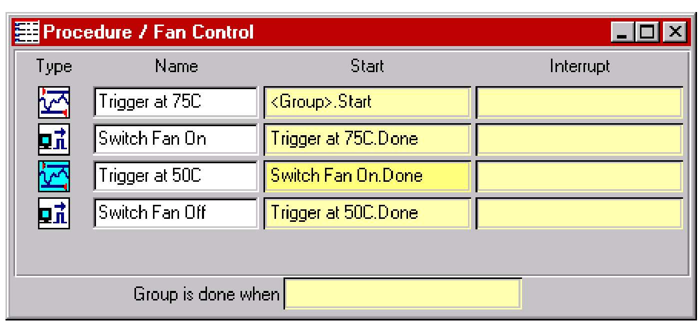
E. Observe that the **Start** trigger for the **Trigger at 50C** process is **Switch Fan On.Done**.

F. Observe that the **Start** trigger for the **Switch Fan Off** process is **Trigger at 50C.Done**.

5. Specify a “Group is done when” trigger.

Leave the **Group is done when** box blank. The **Group** process will automatically end when all the processes within it ends.

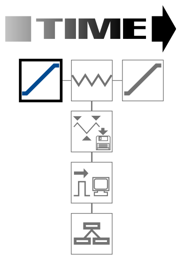
Note *This **Group is done when** trigger selection will end Group process activity, buy may leave the fan in a running condition at the end of test. There are a number of ways to design the procedure to ensure that the fan stops when the test stops, but to minimize complexity they are not included in this example.*



This is how our Group table should look when you have completed this task.

Task 5 Set Process Parameters in the Procedure Table

To complete this task, we will fill out the parameters windows for each process in the Procedure and Group tables.



Segment Command (Ramp to Mean)

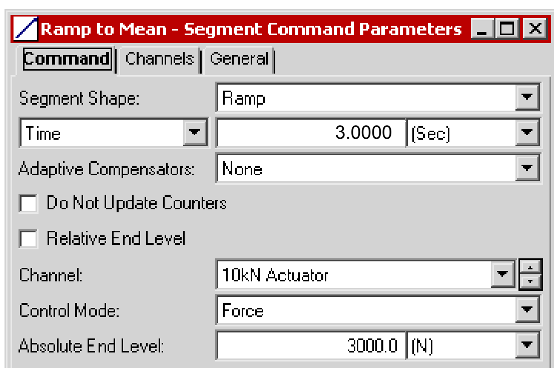
1. Define the Ramp to Mean (Segment Command) process.

Note *For information on the parameter window controls, see* *"Segment Command Parameter Descriptions" on page 200.*

A. In the **Procedure** table, double-click the **Segment Command** (**Ramp to Mean**) process icon.

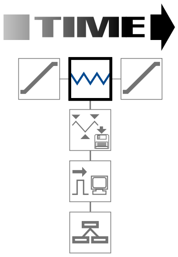
B. In the **Segment Command Parameters** window, click the **Channels** tab. Move all the channels you want to generate the command on to the **Included** list. Move all the others to the **Available** list.

C. Click the **Command** tab. Select and enter the parameters needed by the test, as shown:



D. Click the **General** tab and observe the default selections.

E. When you are finished, close the **Segment Command Parameters** window.



Cyclic Command (Fatigue Part)

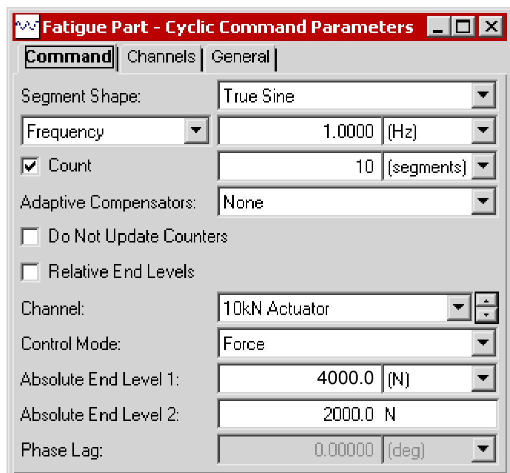
2. Define the Fatigue Part (Cyclic Command) process.

Note *For information on the parameter window controls, see "Cyclic Command Parameter Descriptions" on page 212.*

A. In the **Procedure** table, double-click the **Cyclic Command** (**Fatigue Part**) process icon.

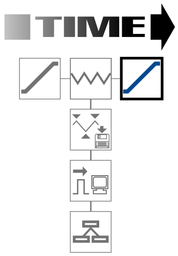
B. In the **Cyclic Command Parameters** window, click the **Channels** tab. Move all the channels you want to generate the command on to the **Included** list. Move all the others to the **Available** list.

C. Click the **Command** tab. Enter and select the parameters needed for the test, as shown:



D. Click the **General** tab and enter information as desired.

E. When you are finished, close the **Cyclic Command Parameters** window.



Segment Command (Return to Zero)

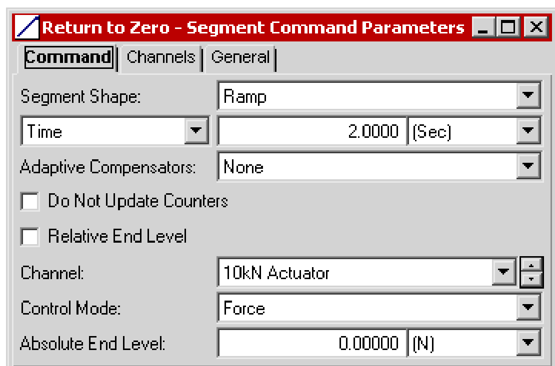
3. Define the Return to Zero (Segment Command) process.

Note *For information on the parameter window controls, see "Segment Command Parameter Descriptions" on page 200.*

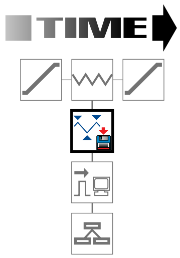
A. In the **Procedure** table, double-click the **Segment Command** (**Return to Zero**) process icon.

B. In the **Segment Command Parameters** window, click the **Channels** tab. Move all the channels you want to generate the command on to the **Included** list. Move all the others to the **Available** list.

C. Click the **Command** tab. Enter and select the parameters needed for the test, as shown:



D. Observe the settings on the **General** tab, then close the **Segment Command Parameters** window.



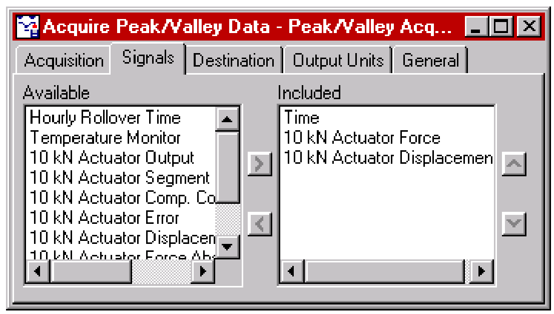
Peak/Valley Acquisition (Acquire Peak/Valley)

4. Define the Acquire Peak/Valley Data (Peak/Valley Data Acquisition) process.

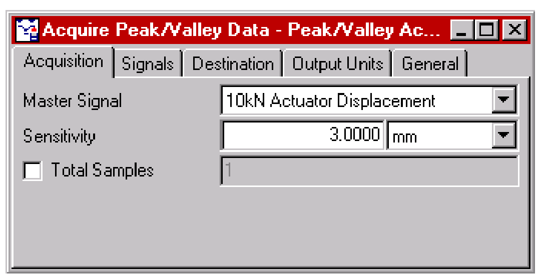
Note *For information on the parameter window controls, see* *"Peak/ Valley Acquisition Parameter Descriptions" on page 269.*

A. In the **Procedure** table, double-click the **Peak/Valley Acquisition** (**Acquire Peak/Valley Data**) process icon.

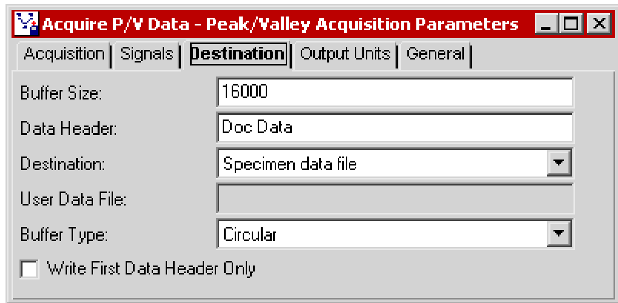
B. In the **Peak/Valley Acquisition Parameters** window, click the **Signals** tab. Move all the signals you want to take data on to the **Included** list. Move all the others to the **Available** list, as shown:



C. Click the **Acquisition** tab. Enter and select the parameters needed for the test, as shown:

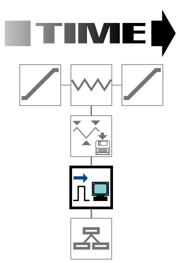


D. Click the **Destination** tab. Enter and select the parameters needed for the test, as shown:



E. Observe the default settings for the **Output Units** and **General** tabs, then close the **Peak/Valley Acquisition Parameters** window.

5. Define the Monitor Continuity process.



Digital Input Detector (Monitor Continuity)

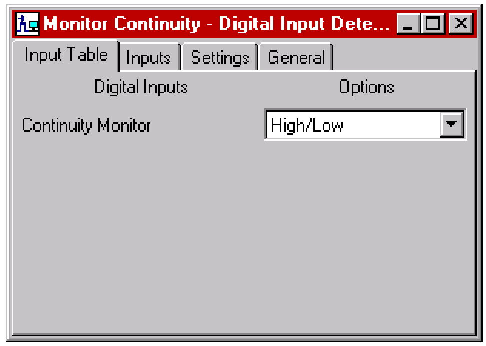
Note *For information on the parameter window controls, see "Digital Input Detector Parameter Descriptions" on page 342.*

A. In the **Procedure** table, double-click the **Digital Input Detector** (**Monitor Continuity**) process icon.

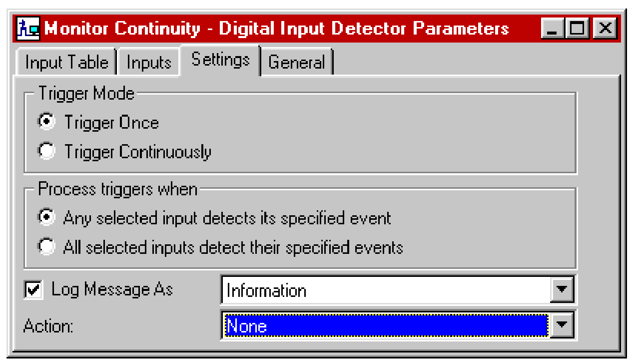
B. In the **Digital Input Detector Parameters** window, click the **Inputs** tab. This process monitors the digital input channel (**Temperature Monitor**) we set up with the Station Builder application.

C. Move the **Temperature Monitor** to the Included list and ensure all other resources are in the **Available** list.

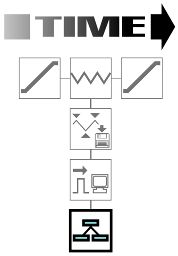
D. Click the **Input Table** tab. Enter and select the parameters needed for the test, as shown.



E. Click the **Settings** tab. Enter and select the parameters needed for the test, as shown.



F. Observe the default settings on the **General** tab, then close the **Digital Input Detector Parameters** window.

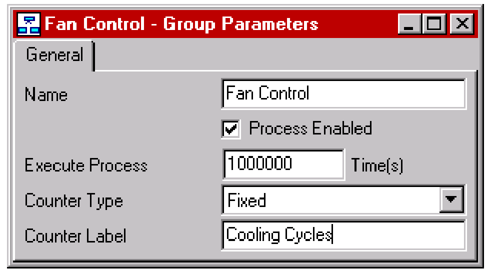


Group
(**Fan Control**)

6. Define the Group (Fan Control) process.

A. In the MPT Procedure Editor, double-click the **Group** (**Fan Control**) process icon.

B. Click the **General** tab. Instead of using the default settings, enter and select the parameters needed for the test, as shown

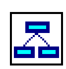


Note *By default, a* ***Group*** *process executes once. It completes when all of the processes in the group are done. By entering a large number in the* ***Execute Process*** *box, we are essentially instructing this* ***Group*** *process to loop continuously.*

C. Close the **Group** process window.

Task 6 Set Process Parameters in the Group Table

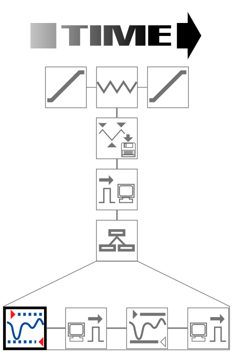
1. Display the procedure table that pertains to the group process.

A. Click the **Group** process icon  on the MPT Procedure Editor.

B. Click  on the MPT Procedure Editor toolbar.

2. Define the Trigger at 75C (Data Limit Detector) process.

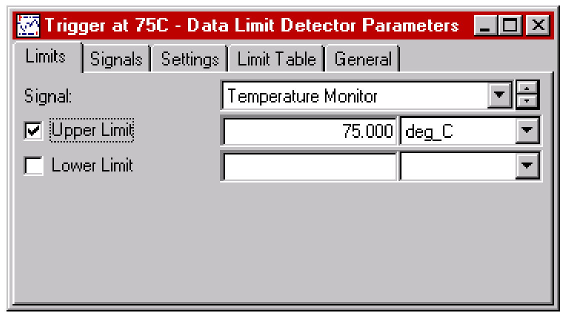
A. In the **Group** table, double-click the **Data Limit Detector** (**Trigger at 75C**) process icon.



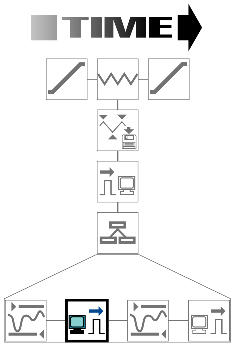
Data Limit Detect
(**Trigger at 75C**)

B. Click the **Signals** tab. This process monitors the auxiliary data channel (**Temperature Monitor**) we set up with the Station Builder application. Move the **Temperature Monitor** to the Included list and ensure all other signals are in the **Available** list.

C. Click the **Limits** tab. Enter and select the parameters needed for the test, as shown.



D. Observe the default settings on the **Settings**, **Limit Table**, and **General** tabs, then close the **Digital Limit Detectors Parameters** window.



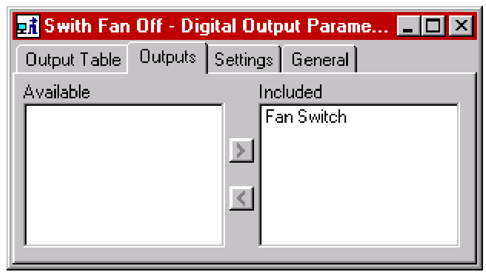
Digital Output (Switch Fan On)

3. Define the Switch Fan On (Digital Output) process.

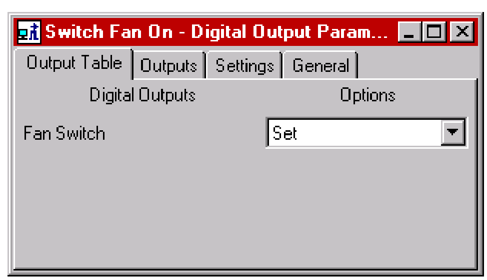
A. In the **Group** table, double-click the **Digital Output** (**Switch Fan On**) process icon.

B. Click the **Outputs** tab and move the **Fan Switch** resource to the **Included** list, as shown below.

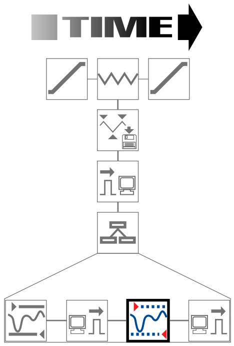
In this case the **Fan Switch** resource would appear only if you used the Station Builder application to assign a digital output resource to the station configuration file and gave it a label of **Fan Switch**.



C. Click the **Output Table** tab. In our sample test scenario, the cooling fan power supply is controlled by a microswitch. This microswitch applies 110 V to the fan when it is set high, and removes 110 V from the fan when it is set low (or cleared). So for this process, the appropriate setting is **Set**, which means set high. Select the parameters needed for the test, as shown.



D. Use the default settings on the **Settings** and the **General** tab, then close the **Digital Output Parameters** window.



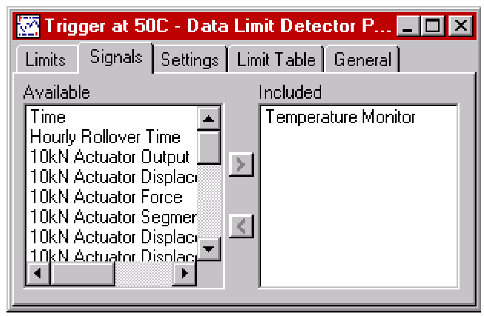
Data Limit Detector (Trigger at 50C)

4. Define the Trigger at 50C (Data Limit Detector) process.

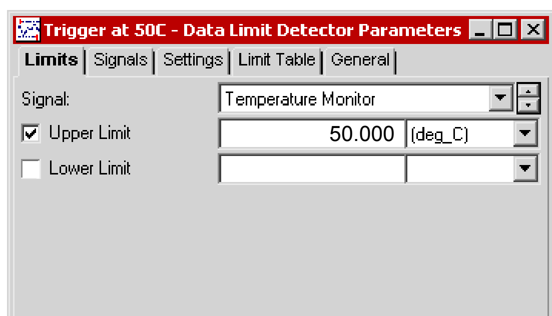
A. In the **Group** table, double-click the **Data Limit Detector** (**Trigger at 50C**) process icon.

B. Click the **Signals** tab and move the **Temperature Monitor** resource to the **Included** list, as shown below.

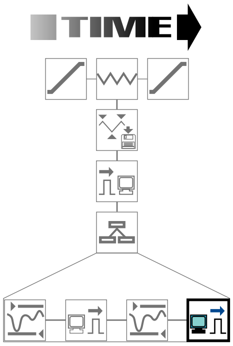
In this case the **Temperature Monitor** signal would appear only if you had used the Station Builder application to create an auxiliary data channel for the station configuration file and gave it a label of **Temperature Monitor**.



C. Click the **Limits** tab. Enter and select the parameters needed for the test, as shown.



D. Observe the default settings on the **Settings**, **Limit Tables**, and **General** tab, then close the **Data Limit Detector Parameters** window.



Digital Output (Switch Fan Off)

5. Define the Switch Fan Off process.

A. In the **Group** table, double-click the **Digital Output** (**Switch Fan Off**) process icon.

B. Click the **Outputs** tab. Move the **Fan Switch** resource to the **Included** list. In this case the **Fan Switch** resource would appear only if you used the Station Builder application to assign a digital output resource to the station configuration file and gave it a label of **Fan Switch**.

C. Click the **Output Table** tab. In our test, the cooling fan power supply is controlled by a microswitch. This microswitch applies 110 V to the fan when it is set high, and removes 110 V from the fan when it is set low (or cleared). So for this process, the appropriate setting is **Clear**, which turns off the fan.

D. Use the default settings on the **Settings** and **General** tabs, then close the **Digital Output Parameters** window.

6. Close the Group table and return to the main MPT Procedure Editor.

Task 7 Select Procedure Options

We can use the **MPT Options Editor** to set MPT preferences. The settings are saved with the procedure.

For our test, we need to specify the following application preferences:

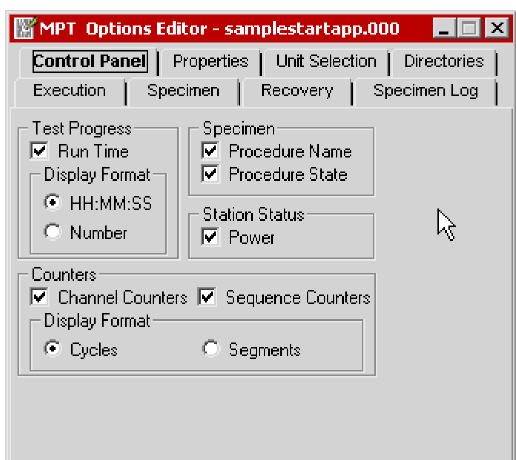
- The counter display on the MPT control panel
- The hydraulic state required to run tests
- The file format for data files generated by the MPT application
- A description of the procedure

1. **On the Tools menu, click Options Editor to display the MPT Options Editor.**
2. **Specify how the counters will be displayed on the MPT control panel.**

   A. Click the **Control Panel** tab.

   B. In the **Counters** box, click **Sequence Counters** and **Total Counters**.

   C. In the **Display Format** box, click **Cycles**.

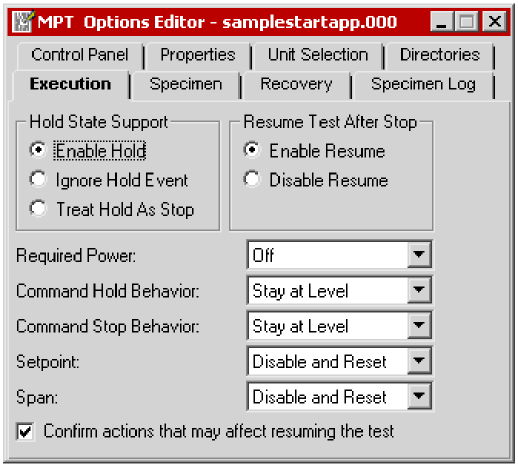


3. Specify the Hydraulic State required to run the test.

A. Click the **Execution** tab.

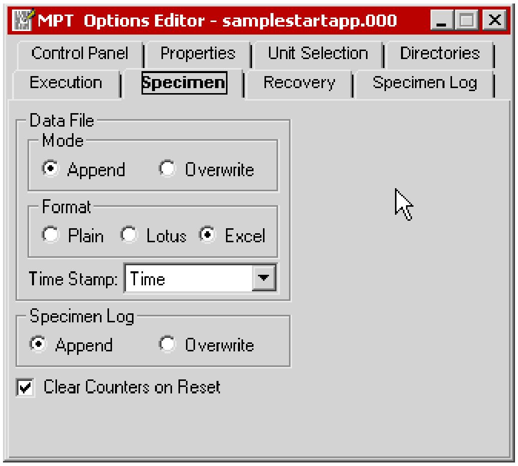
B. In the **Required Power** list, select **Off**.

Note *Since this program will only be used for simulated testing, click* ***Off****. This way, you can only run the test when hydraulic pressure is disabled.*



4. Set the file format for MPT data files.

Click the **Specimen** tab. Complete the tab as shown.

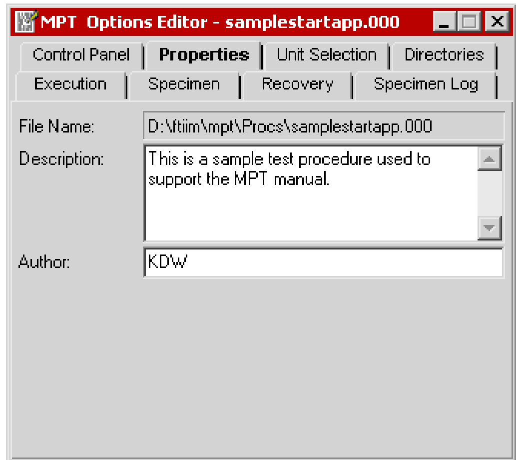


The **Data File Format** specifies the file format for test data.

5. Enter a description of the procedure.

A. Click the **Properties** tab.

B. Enter text as desired in the **Description** box to describe the procedure.



6. If desired, save your sample procedure.

Chapter 3

Working with MPT Specimens

About MPT Specimens

An *MPT specimen* contains a number of files associated with the execution of a specific procedure.

The default location for your MPT specimens is:

- c:\ftxxx (or tsxxx)\mpt\specimens, or
- c:\mts793\mpt\specimens

During the test, the MPT application writes test-generated data files to the selected MPT specimen. These files include log files, data acquisition files, test recovery files, and so forth. For detailed information on the files saved in the MPT specimen, see "MPT Specimen File Descriptions" on page 73.

Assigning MPT specimens

You must link a new or existing MPT specimen to a procedure before you can run it. Once an MPT specimen has been linked to a particular procedure, that procedure is available to the MPT specimen on the MPT control panel. For more information, see "Loading MPT Specimens" on page 81.

You can create new MPT specimens with the MPT control panel and the **MPT Specimen Editor**. You may want to use a new MPT specimen to collect data for a particular test run in a unique location. For more information on creating and linking MPT specimens, see "Creating and Linking MPT Specimens" on page 69.

Creating and Linking MPT Specimens

You can use the MPT control panel or the **MPT Specimen Editor** to create MPT specimens and link procedures to them.

Using the MPT control panel

The MPT control panel is designed to create new MPT specimens one at a time, or as needed, in less structured testing environments.

For example, suppose you want to modify a procedure (with an existing MPT Specimen) and run it against a new MPT specimen. On the MPT control panel, you could simply click the **New Specimen** button, name the MPT specimen, and then click **Run** on the **Station Controls** panel. The new data, test log entries, recovery information, and so on, will be stored in the new MPT specimen, instead of the previous MPT specimen.

MPT specimens are automatically linked to the current procedure when you click **Run**.

For a description of the MPT control panel controls and indicators, see "MPT Control Panel" on page 126.

Using the Specimen Editor

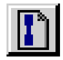
The **MPT Specimen Editor** is designed for preparing a batch of MPT specimens and linking them with one or more procedures for future testing. The **MPT Specimen Editor** is especially useful for test administrators who want to predefine a number of MPT specimens for a test operator so he or she can run a succession of tests.

For example, as a test administrator, you could create a number of new MPT specimens and link them all to your procedure. This way, the test operator could simply select the desired MPT specimen on the MPT control panel (which loads the linked procedure) and run the test. When the test completes, the operator could select the next MPT specimen, and could run the test again. The operator could perform this for each predefined MPT specimen.

For a description of the **MPT Specimen Editor** controls and indicators, see "MPT Specimen Editor" on page 159.

How to Create and Link a New MPT Specimen with the Control Panel

To create an MPT specimen with the MPT control panel:

1. Click  on the MPT control panel toolbar.
2. In the **Specimen** box, rename the new MPT specimen as desired.

To link the MPT specimen to a procedure file:

When you click **Run** on the **Station Controls** panel, the MPT specimen will automatically be linked to the loaded procedure.

How to Create and Link a New MPT Specimen with the Specimen Editor

To create an MPT specimen with the **MPT Specimen Editor**:

1. Click **New** on the **MPT Specimen Editor**. An MPT specimen will be added to the MPT specimen list.
2. In the box above the MPT specimen list, rename the new MPT specimen as desired and press the **Enter** key on the keyboard.

To link the MPT specimen to a procedure file:

1. Click  **Procedure Path/File Name**.
2. Click the desired procedure file in the **Select procedure file** window.

   Note *If the procedure is new, make sure you save it first; otherwise it will not appear in the list of procedures in the* ***Select procedure file*** *window.*

How to Link a Used MPT Specimen to a Different Procedure

Note *This procedure pertains to MPT specimens linked to procedures that have already been run.*

With the Control Panel

1. Open the procedure you want to link your MPT specimen to.
2. In the **Specimen** list on the MPT control panel, click the MPT specimen you want to link.
3. In the **Load Specimen** window that appears, click **Currently Loaded Procedure**.

For more information on loading procedures, see "Loading MPT Specimens" on page 81.

With the Specimen Editor

1. Open the **MPT Specimen Editor**.
2. In the MPT specimen list, click the MPT specimen you want to link.
3. Click  **Procedure Path/File Name**.
4. Click the desired procedure file in the **Select procedure file** window.

   Note *If the procedure is new, make sure you save it first; otherwise it will not appear in the list of procedures in the* ***Select procedure file*** *window.*

   Later, when you select the MPT specimen in the MPT control panel, MPT will display the **Load Specimen** window.
5. In the **Load Specimen** window, click **Procedure assigned to the Specimen**.

Running Multiple Tests Against the Same MPT Specimen

You may want to run a procedure against an MPT specimen more than once, or you may want to run different tests against the same MPT specimen. In either case, it is important to note how the data files in your MPT specimen are affected when you do this.

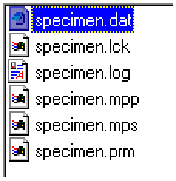
By default, if you run another test against your MPT specimen, the MPT application will:

- Append new test information to the existing data file and log file (see "Specimen.dat" on page 74 and "Specimen.log" on page 77).
- Write over the information in the procedure snapshot file (see "Specimen.mpp" on page 78) and dynamic state snapshot file (see "Specimen.mps" on page 79) with the new test information.

Note *You can configure whether new data overwrites or appends to the previous data with the **MPT Options Editor**. For more information, see page 165.*

MPT Specimen File Descriptions

The number of files (and the content of each file) in an MPT specimen depends on several things, including:



- The number of times a test was run against the MPT specimen
- The runtime state of the procedure when the test was exited
- Whether or not you have specified user-defined data files
- The processes in the procedure

In most cases, after you run a test, the MPT specimen will contain the following files:

- **Specimen.dat**—is an MPT specimen data file that contains data acquired by the data acquisition processes defined in your procedure (see "Specimen.dat" on page 74). If you configured a data acquisition process to write data to a user-specified data file, the file you specified will appear in the MPT specimen (see "User-Specified.dat" on page 76).
- **Specimen.log**—is a record of the test events that occurred while the procedure was running (see "Specimen.log" on page 77).
- **Specimen.mpp**—is a recovery file used to restore the procedure to the same state it was the last time the test was run (see "Specimen.mpp" on page 78).
- **Specimen.mps**—is a recovery file used to restore the runtime state of the procedure (see "Specimen.mps" on page 79).
- **Specimen.prm**—is a file that contains the name of the procedure currently linked to the MPT specimen (see "Specimen.prm" on page 80). This file also contains any comments entered with the **MPT Specimen Editor**.
- **Specimen.lck**—is a temporary control file used to prevent instances of MPT on different stations from simultaneously accessing the same MPT specimen.

Specimen.dat

The Specimen.dat file contains data acquired by the data acquisition processes in your procedure.

Each time you run a procedure against an MPT specimen (that is, each time you press run after pressing reset), newly acquired data overwrites or is appended to the current data in the Specimen.dat file. The specimen data file may contain data from more than one test if more than one test has been run against the same MPT specimen.

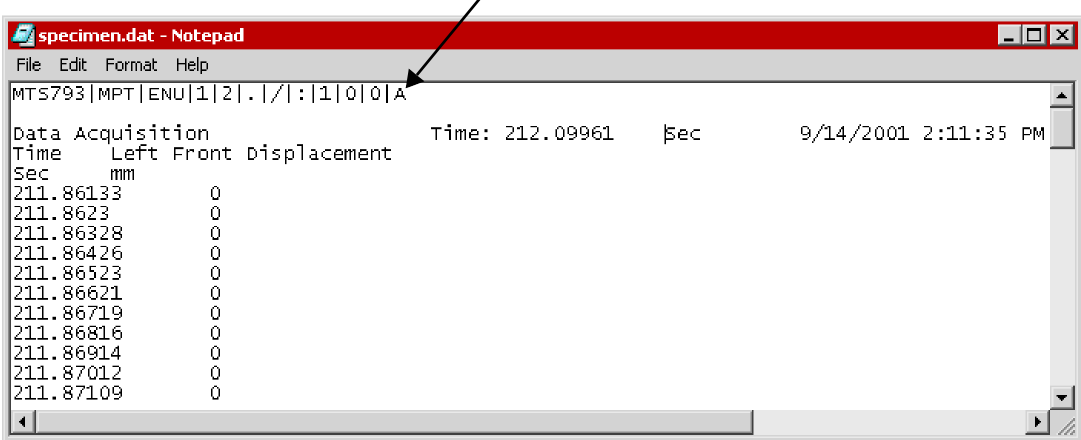
Note *You can configure whether new data overwrites or appends to the previous data with the **MPT Options Editor**. For more information, see page 165.*

Note *If you have configured a data acquisition process to write to a user-specified data file, it will write data to the user-specified file instead of the Specimen.dat file. For more information, see "User-Specified.dat" on page 76.*

Sample MPT specimen data file

The following data file was created by a **Peak/Valley Acquisition** process that monitored the displacement feedback and acquired data on the *time* and *displacement feedback* signals.

This text string specifies the file format. See "Parsing the MPT specimen data file" on page 75.



Formatting the data in the MPT specimen data file

You can configure the MPT application to format the data written to the Specimen.dat file in one of three ways:

- **Plain**—test data is space-delimited. This option provides the most useful output when viewing the test data in an application such as Notepad or Microsoft Word.
- **Lotus**—test data is comma-delimited, with text strings enclosed in quotation marks (this is sometimes referred to as the CSV format). This option formats the data for use with Lotus 1-2-3.
- **Excel**—test data is tab-delimited. This option formats the data for use with Microsoft Excel.

You specify the data file format on the **Specimen** tab of the **MPT Options Editor**. For more information, see "Specimen Panel" on page 130.

If you plan to use the **Excel File Converter**, the data must be tab-or comma-delimited. For more information, see Appendix B, "Excel File Converter".

Parsing the MPT specimen data file

The first line of the data file is a text string that includes information such as the decimal separator, column separator, time formats, date formats, and so forth. This header may be used if you plan to parse the data file with a custom application.

A typical header looks like this:

MTS793|MPT|ENU|1|0|.|/|:|1|0|0|A

The format of this header line is as follows:

MTS793|progName|sLanguage|version|iDelim|sDecimal|sDate|sTime|iCountry|iDate|iTime|sCode

Where:

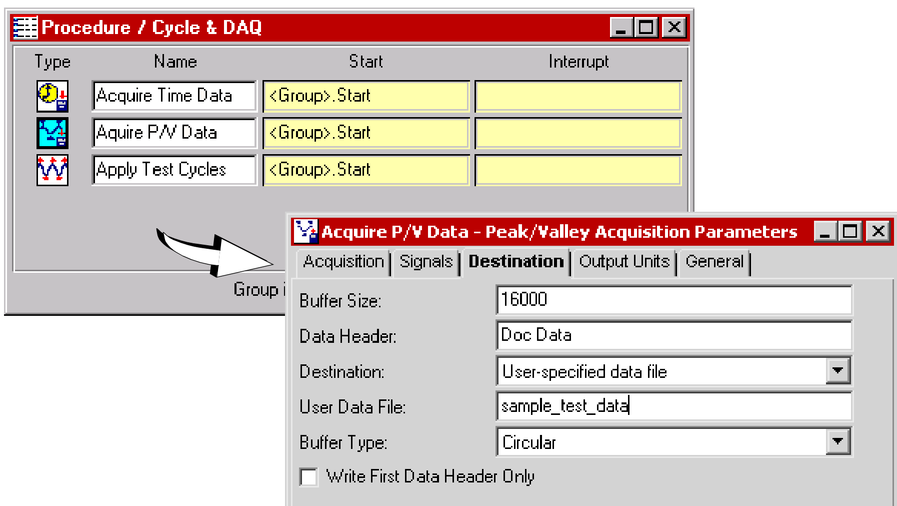
progName=	MPT or BTW
sLanguage=	Natural language (standard three-character abbreviation)
version=	Version of this format string
iDelim=	Data delimeter: 0 for space (plain), 1 for comma (Lotus), 2 for tab (Excel)
sDecimal=	International decimal separator
sDate=	International date separator
sTime=	International time separator
iCountry=	International country value
iDate=	International date format
iTime=	International time format
sCode=	Character Set Code: A for ANSI, O for OEM (currently always A)

User-Specified.dat

You can isolate the data acquired by any specific data acquisition process (or a set of processes) by configuring the process to write to the data file that you specify rather than the default Specimen.dat file.

To configure the process to write to a user-specified data file:

1. In the data acquisition process parameters window, click the **Destination** tab.
2. In the **Destination** list, click **User-specified data file**.
3. In the **User Data File** box, type a name for your data file. (A .dat extension will be added to the name you enter.)



Specimen.log

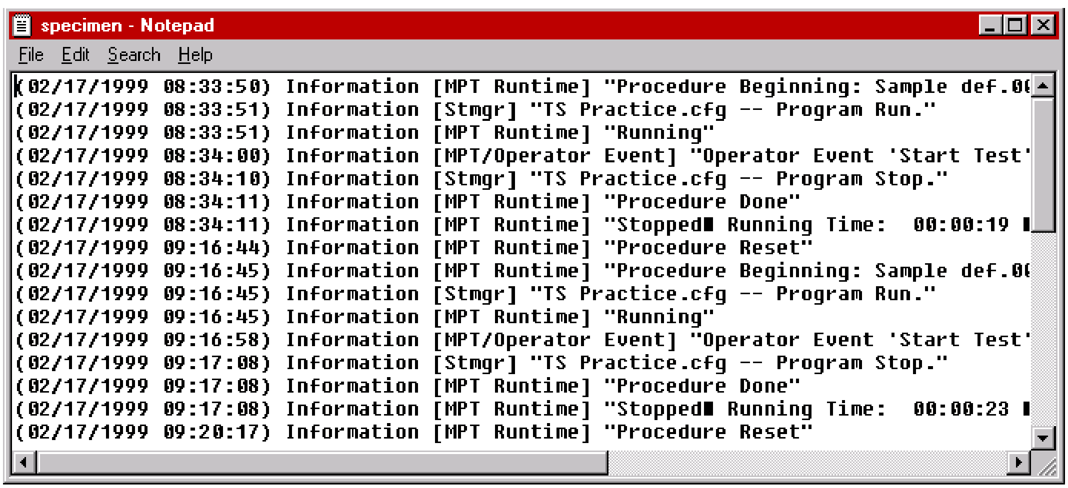
The Specimen.log file contains the contents of the **MPT Specimen Log**. This file is generated during the test, and it includes logged file events, station state changes (Run/Hold/Stop), program state changes (Starting, Running, and so on) and MPT counter values.

By default, this file is cumulative; that is, new information is added when you perform new tests against the same MPT specimen.

Changing the data acquired in the specimen.log file

You can change this behavior with the **MPT Options Editor** as follows:

- Use the **Specimen Log Overwrite/Append** controls on the **Specimen** tab
- Use the **Minimum Severity** and **Source** controls on the **Specimen Log** tab



Specimen.mpp

The Specimen.mpp file is a recovery file used to restore the procedure settings used the last time the test was run. Whenever you click **Run** on the **Station Controls** panel, the MPT application updates the .mpp file with the current test settings regardless of whether the settings are saved or not.

This way, if you modify and run your test, but you forget to save your changes when you close the test, you can restore the modifications the next time you run the test.

For information on restoring unsaved test settings, see "Loading MPT Specimens" on page 81.

Note *This file is created only when test recovery is enabled with the* ***MPT Options Editor****. For more information, see* *"Recovery Tab" on page 180*

Specimen.mps

The Specimen.mps file is a recovery file used to restore an interrupted test at (or close to) the point it was at when it was interrupted.

This way, if your test is interrupted before it completes (due to a power outage, interlock, or accidental stop) you can resume the test from where it was interrupted rather than restarting from the beginning of the test.

The MPT application updates this file according to the settings on the **MPT Options Editor** (see "Recovery Tab" on page 180). Each update includes the test program state, the current counter values, and the status of each active process in the procedure.

Note *By default, the MPT application updates the .mps file each time the program state changes, and at 60-second time intervals.*

For information on resuming an interrupted test from the point of interruption, see "Loading MPT Specimens" on page 81.

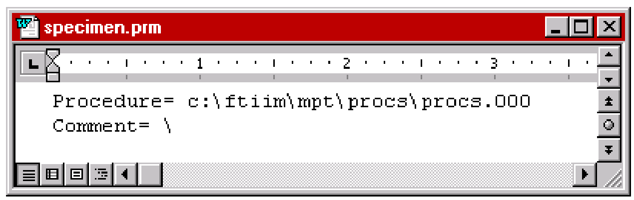
Specimen.prm

The Specimen.prm file contains the name of the procedure linked to the MPT specimen, along with any comments entered in the **Comment** box of the **MPT Specimen Editor**.

When you open an MPT specimen and proceed to load the **Procedure assigned to the Specimen**, the MPT application loads the procedure defined in this file.

The procedure named in this file refers to the last saved version of the procedure linked to the MPT specimen.

Note *If the **Comment** box was left blank on the **MPT Specimen Editor**, the entry "Comment=\" will still appear in this file.*

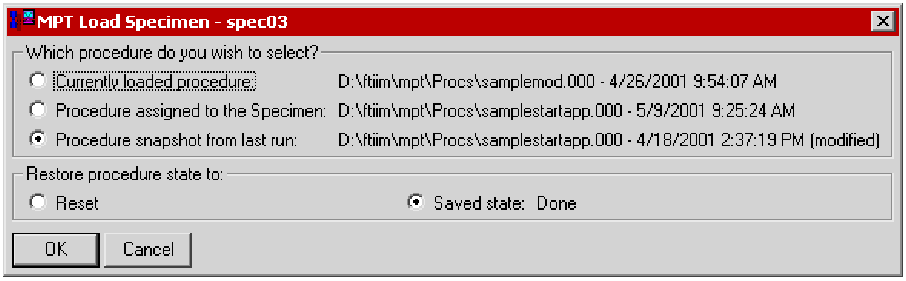


For information on loading MPT specimens and restoring procedures with the MPT control panel, see "Loading MPT Specimens" on page 81.

Loading MPT Specimens

If you select an MPT specimen that MPT has associated with a procedure with more than one version (where the snapshot in the MPT specimen is different than the version saved by the user), MPT displays the **Load Specimen** window.

Note *If you load an MPT specimen that has not been run and you have not linked a procedure to it (via the* ***MPT Specimen Editor****), this window will not appear.*

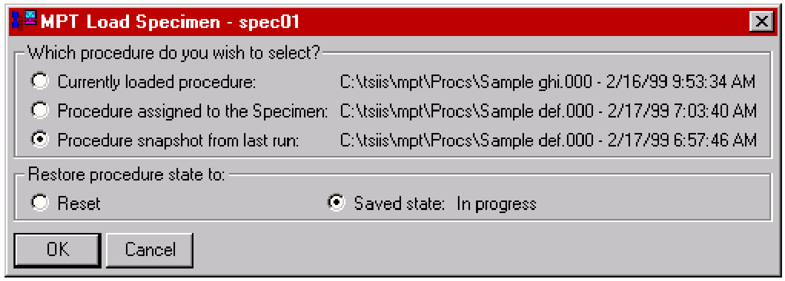


Sample MPT Load Specimen window

When MPT displays the **Load Specimen** window, you should make your selection *carefully*. If you select a procedure without understanding the consequences, you may limit your options later.

For example, assume you choose an MPT specimen on the MPT control panel and you load a test other than the one that is currently assigned to that MPT specimen. In this case, once you click **Run**, you would not be able to retrieve the recovery information created for the original procedure. For more information, see "About the Load Specimen Window" on page 82.

About the Load Specimen Window



Depending on the past usage of an MPT specimen, the **Load Specimen** window will include some or all of these options:

- Currently loaded procedure
- Procedure linked (or assigned) to the MPT specimen
- Procedure snapshot from last run
- Most recently saved version

Each option is followed by a path name and time stamp to help you determine which procedure to load. For more information, see "About the Load Procedure Selections" on page 83.

Note *The time stamp reflects when the procedure was last saved by the operator. It may be helpful to think of the time stamp as the test version number.*

If the procedure you select in the **Load Specimen** window has been run against the selected MPT specimen, you will be able to retrieve it in either its **Reset** state or its **Saved state**. For more information, see "About the Restore Procedure State Selections" on page 84.

To see how these selections are used in specific situations, see "MPT Specimen Loading Examples" on page 85.

About the Load Procedure Selections

The following sections describe the options available in the **Load Specimen** window:

Currently loaded procedure

This option uses the procedure currently open in the **MPT Procedure Editor**. When you click **Run** on the **Station Controls** panel, the selected MPT specimen is linked to the current procedure.

Note *If the currently loaded procedure has been modified but has not been saved, "**modified**" is displayed after the time stamp. If the procedure was never saved, it will not display a time stamp.*

Procedure linked (or assigned) to the MPT specimen

This option loads the procedure that is currently linked to the MPT specimen. This procedure is recorded in the .prm file in the MPT specimen. For more information on the recovery file, see "Specimen.prm" on page 80.

Note *If the referenced procedure file has been deleted or moved, this selection will be dimmed, and "**not found**" is displayed after the file name.*

Procedure snapshot from last run

This option loads a copy of the last procedure run against the MPT specimen. This procedure is recorded in the .mpp recovery file in the MPT specimen. For more information on the recovery file, see "Specimen.mpp" on page 78.

You should select this option if you want to load the procedure and restore the settings exactly as they were the last time you ran the test. This option can be used in conjunction with the saved state selection to recover an interrupted test (see "About the Restore Procedure State Selections" on page 84).

If the procedure was modified without being saved before it was run, "modified" is displayed after the time stamp. This means the snapshot version of the procedure run against the MPT specimen contains modifications that may not be present in the saved version of the procedure.

Most recently saved version

This option loads the most recently saved version of the procedure.

This option will appear only if the saved procedure has been modified since it was saved and is different from the linked procedure.

About the Restore Procedure State Selections

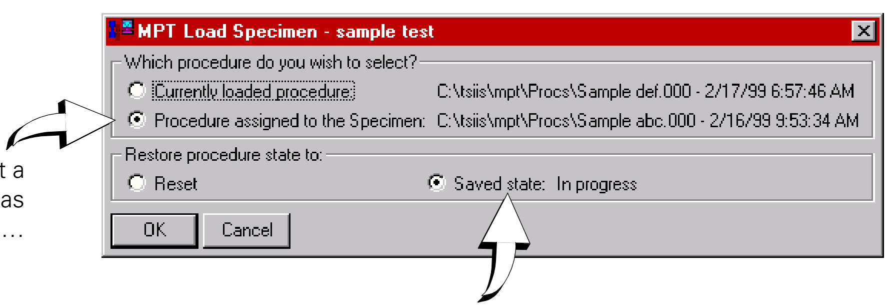
If test recovery is enabled, the MPT application records the state and run-time status of the test as specified on the **Recovery** tab of the **MPT Options Editor**. This data is recorded in the .mps recovery file in the MPT specimen. For more information on the recovery file, see "Specimen.mps" on page 79.

In the **Load Specimen** window, when you select a procedure that has run against the current MPT specimen, you can use the **Restore procedure state to** control to reset or restore the procedure state.

One or more of the following options are available:

- **Reset**—restores the selected procedure to its starting state, with counters reset to zero.
- **Saved state: Done**—restores the procedure in its done state, and includes a record of counter values as they were when the test was complete.
- **Saved state: In progress—**restores the procedure back to the point the test was stopped, and includes a record of the test's counter values, hydraulic status, and so on, at that time. This is the appropriate selection if you wish to recover a test; that is, you have selected the snapshot procedure and wish to resume the test at, or close to, the point it was interrupted.
- **Saved state: Error**—brings the procedure back to the point at which it was stopped by an error condition (there may be a corresponding entry in the .log file that reveals more information).

If you select a procedure that has been run…



…you will have the option of restoring it to its last available runtime state.

MPT Specimen Loading Examples

The following examples are provided to help you choose the right procedure when you load an existing MPT specimen on the MPT control panel.

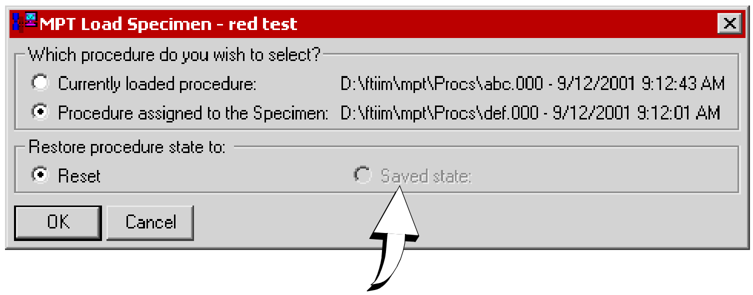
The following scenarios are shown:

- "Procedure Has Not Been Run Against the MPT Specimen" on page 86
- "Procedure Has Been Run Against the MPT Specimen" on page 87
- "Procedure Has Been Run Against the MPT Specimen, and the MPT Specimen Relinked to Another Procedure" on page 89
- "Procedure Has Been Run Against the MPT Specimen, Modified, and Saved" on page 91
- "Procedure Has Been Modified and Run Against the MPT Specimen Without Being Saved" on page 93
- "Procedure Has Been Modified, Run Against the MPT Specimen, and Saved" on page 95

The windows shown for these scenarios do not show all possible combinations; the windows that MPT displays for you will depend on your situation.

Procedure Has Not Been Run Against the MPT Specimen

Suppose you use the **MPT Specimen Editor** to create a new MPT specimen named **red test** and link it to procedure “def.000”. Then you open procedure “abc.000” from the **File** menu and select MPT specimen **red test** on the MPT control panel. When you do this, the following choices will display:



When the **Saved state** selection is disabled, it means the procedure linked to the MPT specimen has not been run against the MPT specimen (as in this scenario), or that the procedure has been run, but its recovery feature has been disabled (see “Recovery Tab” on page 180).

Currently loaded procedure

If you choose this selection and then run the test, MPT will unlink the MPT specimen from procedure “def.000” and link it to procedure “abc.000”. The MPT specimen will then contain information (test data, log entries, and so on) from procedure “abc.000”.

Procedure assigned to the MPT specimen

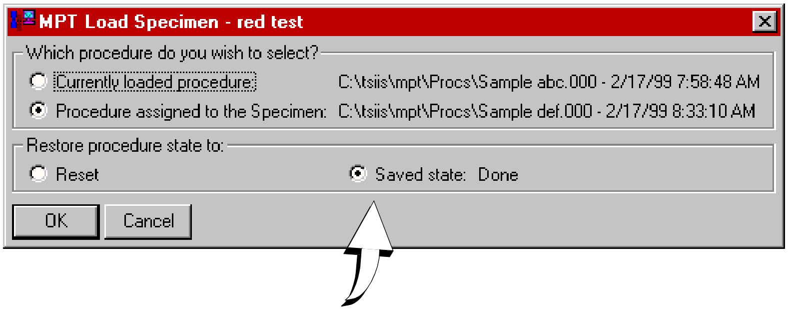
This is the default selection. If you choose it and then run the test, MPT will replace the procedure currently in memory, “abc.000”, with the procedure you linked to the MPT specimen with the MPT Specimen Editor, “def.000”.

Restore procedure state

In this case, procedure “def.000” has never been run against the MPT specimen, so MPT has not created an .mps file or an .mpp file in the MPT specimen yet. Therefore, the Saved state selection is disabled, and Reset is the only option available. Reset causes the procedure to begin at its starting point, with counters reset to zero.

Procedure Has Been Run Against the MPT Specimen

Suppose you load procedure “Sample def.000” into memory, and run it unmodified on MPT specimen **red test**. Then you load procedure “Sample abc.000” from the **File** menu, and select MPT specimen **red test** from the MPT control panel. When you do this, the following choices will display:



This shows the procedure was run against the MPT specimen to completion.

Currently loaded procedure

If you choose this selection and then run the test, MPT will unlink the MPT specimen from procedure “Sample def.000” and link it to procedure “Sample abc.000”. The MPT specimen will then contain information (test data, log entries, and so on) from different procedures (“Sample abc.000” and “Sample def.000”). Choosing this selection will also prevent you from being able to recover status information about when you ran “Sample def.000” in the future.

Procedure assigned to the MPT specimen

This is the default selection. If you choose it and then run the test, MPT will replace the procedure currently in memory, “Sample abc.000”, with the procedure linked to MPT specimen, “Sample def.000”. This is the saved version of the procedure. In this scenario, this version is the same as the version that MPT automatically saved to the .mpp file when the procedure was last run.

Restore procedure state to

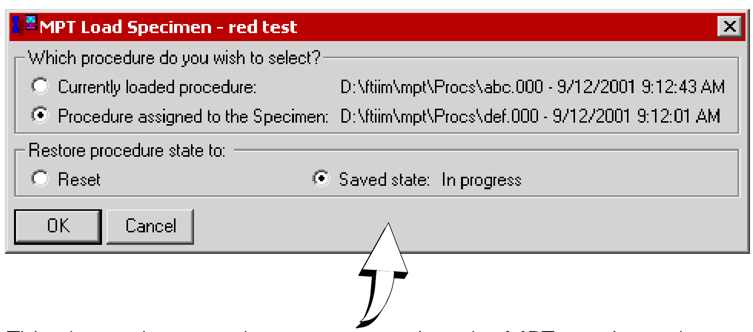
While the procedure “Sample def.000” was being run, MPT recorded the progress or state of the execution of the test at intervals defined in the **Recovery** tab of the **Options Editor**. MPT saves this information in the .mps file, where it is continually updated until the test is complete or in its Done state.

The **Saved state** selection in the **Restore procedure state to** box displays the last saved state stored in the .mps file. In this case the test ran to completion, so the last saved state is **Done**. **Done** brings the procedure back to its done state, and includes information such as counter values as they were when the test was complete; in contrast, **Reset** brings the procedure back to its starting point, with counters reset to zero.

Example

Suppose that in this scenario the test was interrupted while it was being run. In this case the window shown would appear, which is the same as the previous window shown for this scenario, except the **Saved state** selection is **In progress** instead of **Done**.

The **Saved state** option lets you bring the test back to the state it was in when it was stopped.

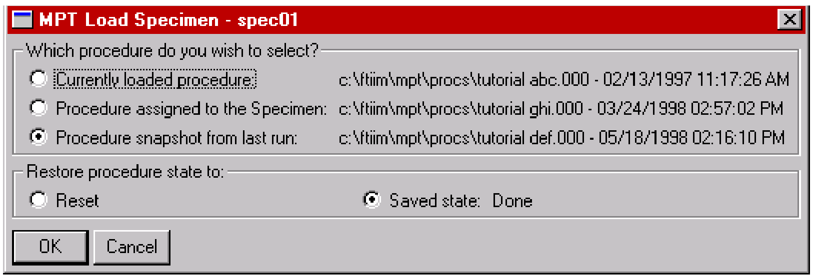


This shows the procedure was run against the MPT specimen, but was stopped before the test was complete. When a procedure is stopped in progress, MPT saves the state of the procedure when it was stopped in the .mps file (see “Specimen.mps” on page 79).

If the test had been inadvertently interrupted and you wished to resume the test at (or close to) the point of interruption, you would choose this **Saved state** selection.

Procedure Has Been Run Against the MPT Specimen, and the MPT Specimen Relinked to Another Procedure

Suppose you load procedure “tutorial def.000” into memory, and run it unmodified on MPT specimen **spec01**. Next, you use the **MPT Specimen Editor** to link procedure “tutorial ghi.000” to MPT Specimen **spec01**. Then you load procedure “tutorial abc.000” from the **File** menu, and select **spec01** in the MPT control panel. When you do this, the following choices will display:



Currently loaded procedure

If you select this and then run the test, MPT will unlink the MPT specimen from both the procedure it was last run against, “tutorial def.000”, and from the procedure you have linked to it with the **MPT Specimen Editor**, “tutorial ghi.000”. MPT will link the MPT specimen to the procedure currently in memory, “tutorial abc.000”.

The MPT specimen will contain information (test data, log entries, and so on) from two different procedures (“tutorial def.000” and “tutorial abc.000”). Choosing this selection will also prevent you from being able to recover status information when you run “tutorial def.000” in the future.

Procedure assigned to the MPT specimen

If you choose this and then run the test, MPT will unlink the MPT specimen from the procedure that it was last run against, “tutorial def.000”, and link it to procedure “tutorial ghi.000”. The MPT specimen will contain information (test data, log entries, and so on) from different procedures (“tutorial ghi.000” and “tutorial def.000”), and you will not be able to recover “tutorial def.000” in the future.

Procedure snapshot from last run

This is the default selection. If you choose it and then run the test, MPT will replace the procedure currently in memory, "tutorial abc.000", with procedure "tutorial def.000" as it was last run against the MPT specimen. This selection will dissociate the MPT specimen from the procedure you linked to it with the **MPT Specimen Editor**, "tutorial ghi.000".

When the operator clicked the **Run** button on the **Station Controls** panel to execute "tutorial def.000" (as it was last run on the MPT specimen), MPT recorded the current state of the procedure and stored it in the .mpp file (see "Specimen.mpp" on page 78). So even though you linked a new procedure to the MPT specimen with the **MPT Options Editor**, the .mpp file contains a copy of the procedure as it was last run against the MPT specimen, called a snapshot.

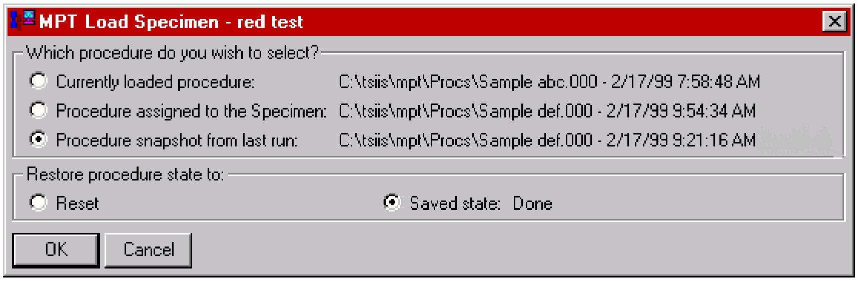
Restore procedure state to

You can choose the state to which the specimen will be restored. In this case the saved state is Done, so selecting the saved state will bring the procedure back to its done state, and will include information such as counter values as they were when the test was complete. Selecting **Reset** will bring the procedure back to its starting point, with counters set to zero.

The **Saved state** information is stored in the .mps file (see "Specimen.mps" on page 79). If the test had been inadvertently interrupted and you wished to resume the test at (or close to) the point of interruption, you would choose this selection.

Procedure Has Been Run Against the MPT Specimen, Modified, and Saved

Suppose you load procedure "Sample def.000" into memory, and run it unmodified on MPT specimen **red test**. Next, you modify this procedure and save your changes. Then you load procedure "Sample abc.000" from the **File** menu, and select **red test** in the MPT control panel. When you do this, the following choices will display.



Currently loaded procedure

MPT will unlink the MPT specimen from procedure "Sample def.000" and link it to procedure "Sample abc.000". If you do this and run the test, the MPT specimen will contain information (test data, log entries, and so on) from different procedures (sample abc.000 and sample def.000). Choosing this selection will also prevent you from being able to recover status information about when you ran "Sample def.000" in the future.

Procedure assigned to the MPT specimen

MPT will replace the procedure currently in memory, "Sample abc.000", with the procedure linked to MPT specimen, "Sample def.000". This version of the procedure includes modifications not present when the procedure was last run, because the operator modified and saved it after it was run.

Procedure snapshot from last run

This is the default selection. If you choose it and then run the test, MPT will replace the procedure currently in memory, "Sample abc.000", with procedure "Sample def.000" as it was last run against the MPT specimen. This is the version of the procedure that MPT automatically saved to the .mpp file when the test was run. In this case, the procedure was modified and saved after the test was run, so the current version of the procedure includes modifications not present in the .mpp file.

Restore procedure state to

Since the MPT specimen was last run against the modified version of "Sample def.000", you can choose the state to which it will be restored. In this case the saved state is **Done**, so selecting the saved state will bring the procedure back to its done state, and will include information such as counter values as they were when the test was complete. Selecting **Reset** will bring the procedure back to its starting point, with counters set to zero.

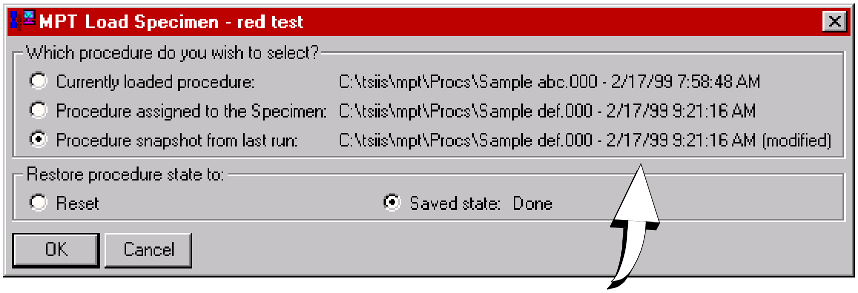
The **Saved state** information is stored in the .mps file (see "Specimen.mps" on page 79). If the test had been inadvertently interrupted and you wished to resume the test at (or close to) the point of interruption, you would choose this selection.

In this scenario, if you select the **Procedure assigned to the Specimen** and select the **Restore procedure state to: Saved State** options, then the runtime state will be restored to a different procedure than the one that was running when it was saved.

If changes to the procedure included adding, removing, or renaming processes, however, the runtime state may not restore completely, and a warning message will be displayed.

Procedure Has Been Modified and Run Against the MPT Specimen Without Being Saved

Suppose you load procedure “Sample def.000” into memory, modify it without saving your changes, and then run it against MPT specimen **red test**. Next, you load procedure “Sample abc.000” from the **File** menu, and select **red test** in the MPT control panel. When you do this, the following choices will display:



The "**modified**" flag means the procedure was modified but not saved by the operator when run against the MPT specimen. MPT automatically saves the modified version in the .mpp file when the operator presses the **Run** button on the **Station Controls** panel. This selection lets you retrieve the modified version of the procedure as it was run against the MPT specimen.

Note *The time stamp is the same for both "Sample def.000" selections because it reflects the most current version of the procedure saved by the operator. However, if you choose the snapshot version, MPT will retrieve the modified version of the procedure that MPT automatically saved to the .mpp file when the last test was run. This version is different from the version last saved by the operator.*

Currently loaded procedure

MPT will unlink the MPT specimen from procedure sample def.000 and link it to procedure “Sample abc.000”. If you do this and run the test, the MPT specimen will contain information (test data, log entries, and so on) from different procedures (“Sample abc.000” and “Sample def.000”). Choosing this selection will also prevent you from being able to recover status information about when you ran “Sample def.000” in the future.

Procedure assigned to the MPT specimen

MPT will replace the procedure currently in memory, “Sample abc.000”, with the procedure linked to MPT specimen, “Sample def.000”.This version of procedure “Sample def.000” will not include the modifications present when it was last run against the MPT specimen. It is the version last saved by the operator.

Procedure snapshot from last run

This is the default selection. If you choose it and then run the test, MPT will replace the procedure currently in memory, “Sample abc.000”, with the procedure “Sample def.000” as it was last run against the MPT specimen.

This version of procedure “Sample def.000” includes the modifications present when it was last run against the MPT specimen. MPT automatically saved a copy of the modified procedure to the .mpp file (see “Specimen.mpp” on page 78) when the test was last run.

The “**modified**” flag means MPT noted a difference between the version of the procedure saved by the operator and the version that MPT saved to the .mpp file when the test was run.

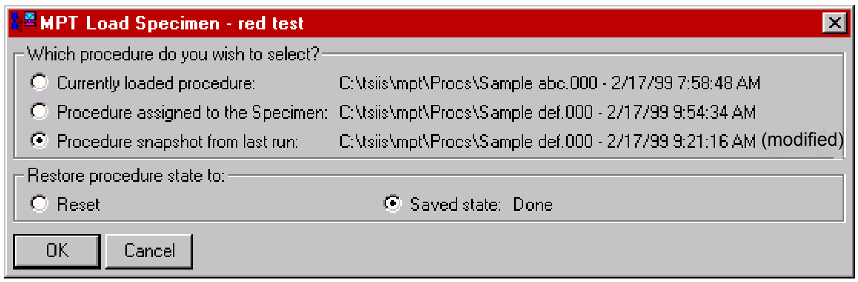
Restore procedure state to

Since the MPT specimen was last run against the modified version of “Sample def.000”, you can choose the state to which it will be restored. In this case the saved state is **Done**, so selecting the saved state will bring the procedure back to its done state, which will include information such as counter values as they were when the test was complete. Clicking **Reset** will bring the procedure back to its starting point, with counters set to zero.

The **Saved state** information is stored in the .mps file (see “Specimen.mps” on page 79). If the test had been inadvertently interrupted and you wanted to resume the test at (or close to) the point of interruption, you would choose this selection.

Procedure Has Been Modified, Run Against the MPT Specimen, and Saved

Suppose you load procedure “Sample def.000” into memory, modify it without saving your changes, then run it against MPT specimen **red test**. Then, after you have run, you save the modified procedure. Next, you load procedure “Sample abc.000” from the **File** menu, and select **red test** in the MPT control panel. When you do this, the following choices will display:



Currently loaded procedure

If you choose this and then run the test, MPT will unlink the MPT specimen from procedure “Sample def.000” and link it to procedure “Sample abc.000”. The MPT specimen will contain information (such as test data and log entries) from two different procedures (“Sample abc.000” and “Sample def.000”). This will also prevent you from being able to recover “Sample def.000” in the future.

Procedure assigned to the MPT specimen

If you choose this and then run the test, MPT will replace the procedure currently in memory, “Sample abc.000”, with the procedure linked to MPT specimen, “Sample def.000”.

In this scenario, this version of procedure “Sample def.000” will include the modifications present when it was last run against the MPT specimen, because these changes were saved after the test was run. So in this case, the saved version of procedure “Sample def.000” is identical to the version saved in the .mpp file. However, since MPT cannot tell if additional changes have been made before it was saved, it displays both options.

Procedure snapshot from last run

This is the default selection. MPT will replace the procedure currently in memory, “Sample abc.000”, with procedure “Sample def.000” as it was last run against the MPT specimen.

This version of procedure “Sample def.000” includes the modifications present when it was last run against the MPT specimen. A copy of the modified procedure was automatically saved to the .mpp file (see page 78) when the test was last run.

The “**modified**” flag means MPT noted a difference between the version of the procedure saved by the operator and the version MPT saved to the .mpp file when the test was run. In this case, even though the **modified** flag is present, both versions are identical because the modified version was saved by the operator after the test was run.

Restore procedure state to

Since the MPT specimen was last run against the snapshot version of “Sample def.000”, you can choose the state to which it will be restored. In this case the saved state is Done, so selecting the saved state will bring the procedure back to its done state, which will include information such as counter values as they were when the test was complete. Clicking **Reset** will bring the procedure back to its starting point, with counters set to zero.

The **Saved** state information is stored in the .mps file (see “Specimen.mps” on page 79). If the test had been inadvertently interrupted and you wished to resume the test at (or close to) the point of interruption, you would choose this selection.

Chapter 4

Working with Processes

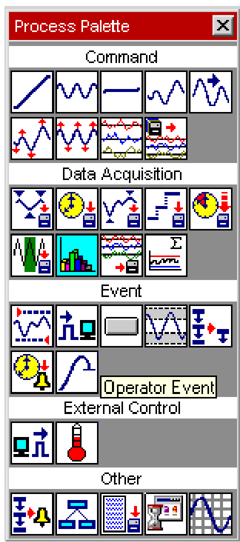
About MPT Processes

When you create tests with the MPT application, you use MPT *processes* like building blocks to define test actions such as sending command signals, performing data acquisition, and responding to test events.

A complete MPT test normally consists of a number of MPT processes linked together and saved as an MPT *procedure*.

Process Palette

Your MPT process icons are displayed on the **Process Palette**.



Process categories

Processes are grouped on the palette in the following categories:

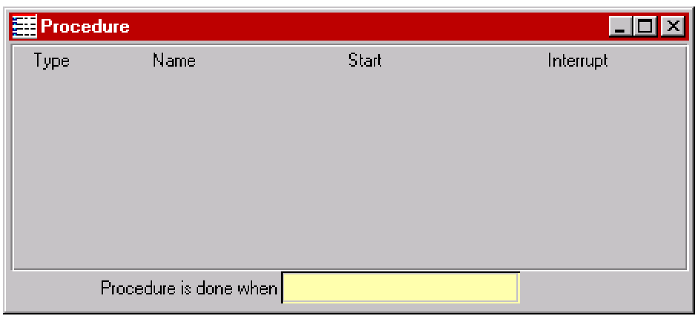
- Command Processes send commands to a servovalve or external controller.
- Data Acquisition Processes acquire sensor data from the physical specimen.
- Event Processes monitor test signals and perform actions when specified conditions are met.
- External Control Processes control external controllers or logic devices.
- Other Processes perform miscellaneous functions.

Adding Processes to the Procedure Table

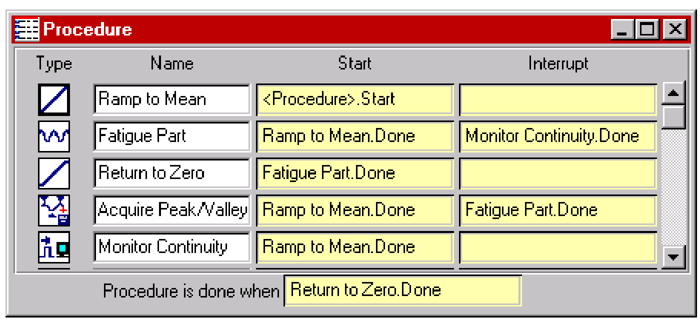
The first and most basic step you must perform to create a test procedure is to add processes to your procedure table.

There are two ways to add a process to the procedure table:

1. Click the process icon on the **Process Palette**, and then click **Add Process** on the **Edit** menu of the **MPT Procedure Editor**.
2. Drag the process icon from the **Process Palette** to your procedure and group tables.



Empty procedure table



Procedure table with processes

Working with Processes

How to Add Processes Using the Add Command

1. Click the desired process icon on the **Process Palette**.
2. On the **Edit** menu of the **MPT Procedure Editor**, click **Add Process**.

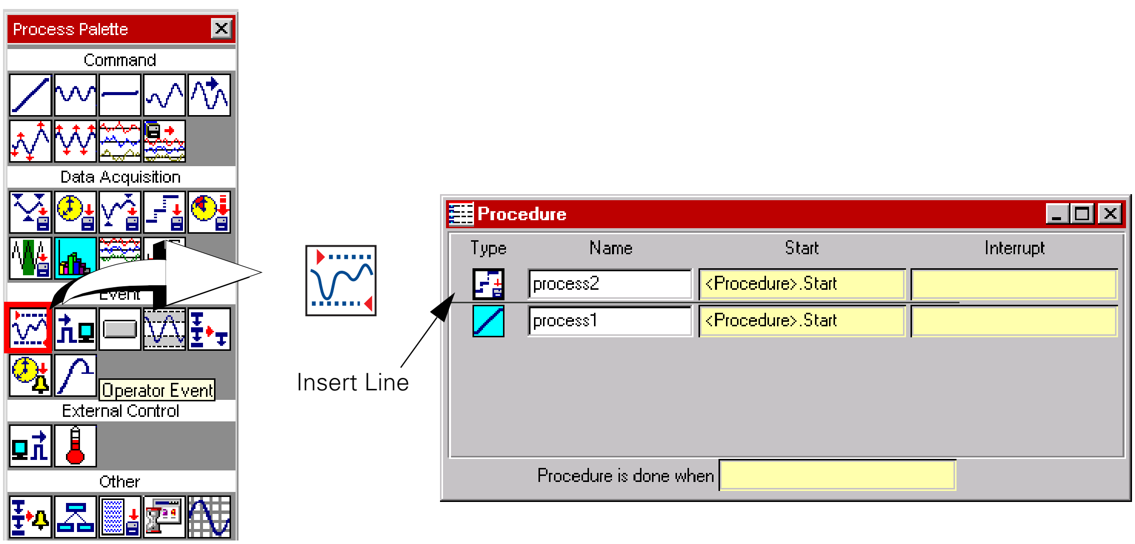
By default, the process icon is inserted below any existing process icons in the window. To insert a process in a particular position, in the procedure table, click the icon you want to insert the new process above, and then add the new process.

Note *You can also drag icons up and down in the procedure table to reposition them.*

How to Add Processes by Dragging-and-Dropping

Drag the process icon from the **Process Palette** to your group or procedure table.

(Use the horizontal insert line to position your process icon.)



Selecting Processes in Procedure Tables

You may want to select process icons in your procedure table to copy or delete them. You can select process icons the same way you would select files in Windows® operating system software.

How to Select Process Icons

To select a process icon in your procedure table, click it.

Note *The icon will appear highlighted when selected.*

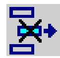
To select a number of individual process icons, hold down the **Ctrl** key, and then click each icon you want to select.

To select a group of process icons that are next to each other in the procedure table, click the first process icon, hold down the **Shift** key, then click the last process icon in the desired range.

Deleting a Process from the Procedure Window

You can delete processes from your procedure table several ways:

In the procedure table, click the process icon or icons you desire to delete to highlight them, and perform one of the following:

1. Click  on the **MPT Procedure Editor** toolbar.
2. On the **Edit** menu, click **Delete Process**.
3. Drag the process icon or icons to the trash can located on the upper right hand corner of the **MPT Procedure Editor**.
4. Click the right-mouse button and select **Delete** from the menu.

Important *You cannot undo a delete operation.*

Cutting, Copying, and Pasting Processes

You can cut, copy, and paste processes on the same procedure table, or from one procedure table to another.

Note *When you copy a process icon, or a group of process icons, the parameters of the original processes are retained, but the sequencing information may not be. You may have to reassign Start and Interrupt signals.*

How to Cut/Copy, and Paste a Process or Group of Processes

1. Select the process or processes you want to copy and paste.
2. On the **Edit** menu, click **Copy Process** or **Cut Process**.
3. Open the procedure table where you want to place the copied processes.
4. Select the process before which you want to paste the processes on the clipboard.
5. On the **Edit** menu, click **Paste Process**.

How to Quick Copy and Paste Selected Processes in the Same Window

1. Click the process or processes you want to copy.
2. Press and hold the **Ctrl** key, and then drag the selected processes. Use the insert line to paste the copied process in the desired position.

Note *To copy and paste processes between tables in separate MPT applications, you must use the **Copy Process** and **Paste Process** commands on the **Edit** menu. Dragging and dropping processes does not work in this situation.*

Sequencing Processes

Processes in your MPT procedure must be linked together so that they start and stop at the proper times.

In your procedure or group table:

- Use the **Start** control to determine when each process starts.
- Use the **Interrupt** control if you want to specify a test event that will interrupt the process.
- Use the **Procedure** (or **Group**) **is done when** control to determine when the procedure ends.

Procedure start and interrupt signals

All signal lists in your main procedure table include the **<Procedure>.Start** and **<Procedure>.Interrupt** signals.

If you set the **Start** or **Interrupt** control to **<Procedure>.Start**, the process will start or interrupt when the procedure starts.

Note *In group tables, the controls are labeled <Group>.Start and <Group>.Interrupt, and they pertain to the start and interrupt of that particular Group process. See “Sequencing Group Processes” on page 116.*

Done signals

When an individual process ends, it generates a “done” signal. These done signals are labeled *process name*.Done in your **Start**, **Interrupt**, and **Procedure is done when** lists.

For example, if you created a segment command process named **Ramp Up**, the done signal that appears in your signal lists would be labeled **Ramp Up.Done**.

Each process automatically generates a done signal when the process completes its programmed function, or when it is interrupted by another signal. Process done signals are typically used to specify the start of another process and/or the end of your procedure.

Special process signals

In addition to done signals, some data acquisition processes can also send out a “buffer full” signal (indicating when the data buffer is full), and some event processes can send out a “trigger” signal (indicating when the process is triggered).

A buffer full signal or trigger can also be used to specify the start of another process, the end of the test, or to interrupt other processes.

Fan-in and fan-out processes

You can select more than one signal in each **Start**, **Interrupt**, and **Procedure is done when** list to create a “fan-in” process. The process (or procedure) will respond if any of the selected signals is detected.

Conversely, you can use the <Procedure.Start>, or a single done, buffer full, or trigger signal to start, interrupt, or end several other processes. This is referred to as a “fan-out” process.

Note *There is a slight time delay between processes that start with a common trigger signal (for example, two processes using the <Procedure.Start> trigger signal). The process positioned the highest on the Procedure or Group table starts first. When using data acquisition and command processes in parallel, position the data acquisition process higher on the table than the associated command process.*

Ending a procedure

The procedure table has a **Procedure is done when** control located at the bottom of the window.

If you specify a signal in this list, when the specified signal is received, all active processes in this window will be interrupted and the procedure will end. You can use this control to ensure that your procedure ends properly.

Note *For information on ending a* ***Group*** *process, see* *“Ending a group process” on page 116.*

If you leave this control blank, the procedure will stop when all of its child processes are done.

More About Buffer Full Signals

Some acquisition processes can generate buffer full signals when the data buffer fills. These buffer full signals are labeled *process name*.**Buffer Full** in your **Start**, **Interrupt**, and **Procedure is done when** lists.

For example, if you created a timed acquisition process named **Acquire Time Data**, the buffer full signal that appears in your signal lists would be labeled **Acquire Time Data.Buffer Full**.

Note *You define the size of the buffer on the **Destination** tab of the acquisition process' **Parameters** window.*

Cyclic acquisition processes can generate buffer full signals when data has been acquired for the number of cycles or segments specified on the **Data Storage** tab in the **Cyclic Acquisition Parameters** window.

When buffer full signals are sent

If you do not specify the number of samples to acquire for the acquisition process, the process will continuously sample and generate a buffer full signal every time the buffer fills, until the process ends.

If you specify the number of samples to acquire, the process will send a buffer full signal every time the buffer fills until the requested number of samples are acquired.

Special uses

You can use a data acquisition buffer full signal to start and interrupt other processes in your test.

For example, if you configure your data acquisition process to discard data when the buffer fills (instead of saving it), the acquisition process can continuously send buffer full signals to other processes without using up disk space.

More About Trigger Signals

Some processes can generate trigger signals when they detect the specified event. These trigger signals are labeled *process name***.Trigger** in your **Start**, **Interrupt**, and **Procedure is done when** lists.

For example, if you created a digital input process named **Input Monitor**, the trigger signal that appears in your signal lists would be labeled **Input Monitor.Trigger**.

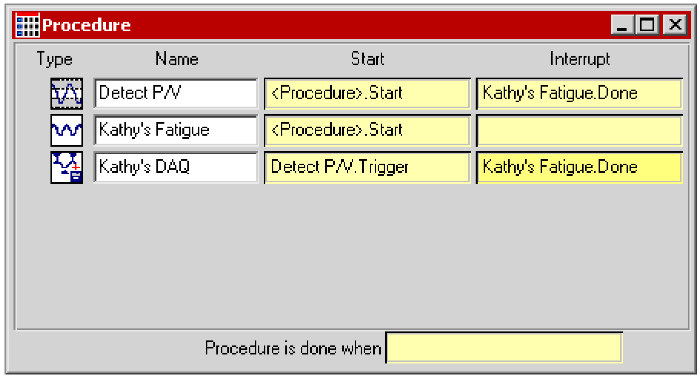
When trigger signals are sent

You can configure these processes to trigger once or to trigger continuously. When a process is set to trigger once, it will generate a trigger when the specified condition is met, and then it will generate a done signal. When a process is set to trigger continuously, it will continue to send triggers each time the specified condition is met, and it will not generate a done signal.

Note *If you set up a process to trigger continuously, the process will not generate a done signal until it is interrupted. You should specify an "Interrupt" signal or a "Procedure/Group is done when" signal to end a continuously triggered process.*

Trigger signal example

The following sample procedure makes use of a trigger signal.



In this example:

- The Peak/Valley Change Detector process (labeled Detect P/V) is set to trigger continuously. (It will generate a trigger signal whenever it detects a peak or a valley outside the specified tolerance band.)
- The **Detect P/V.Trigger** is used to start the peak/valley acquisition process (labeled **Kathy's DAQ**), which will acquire a specified amount of peak/valley data and then stop.

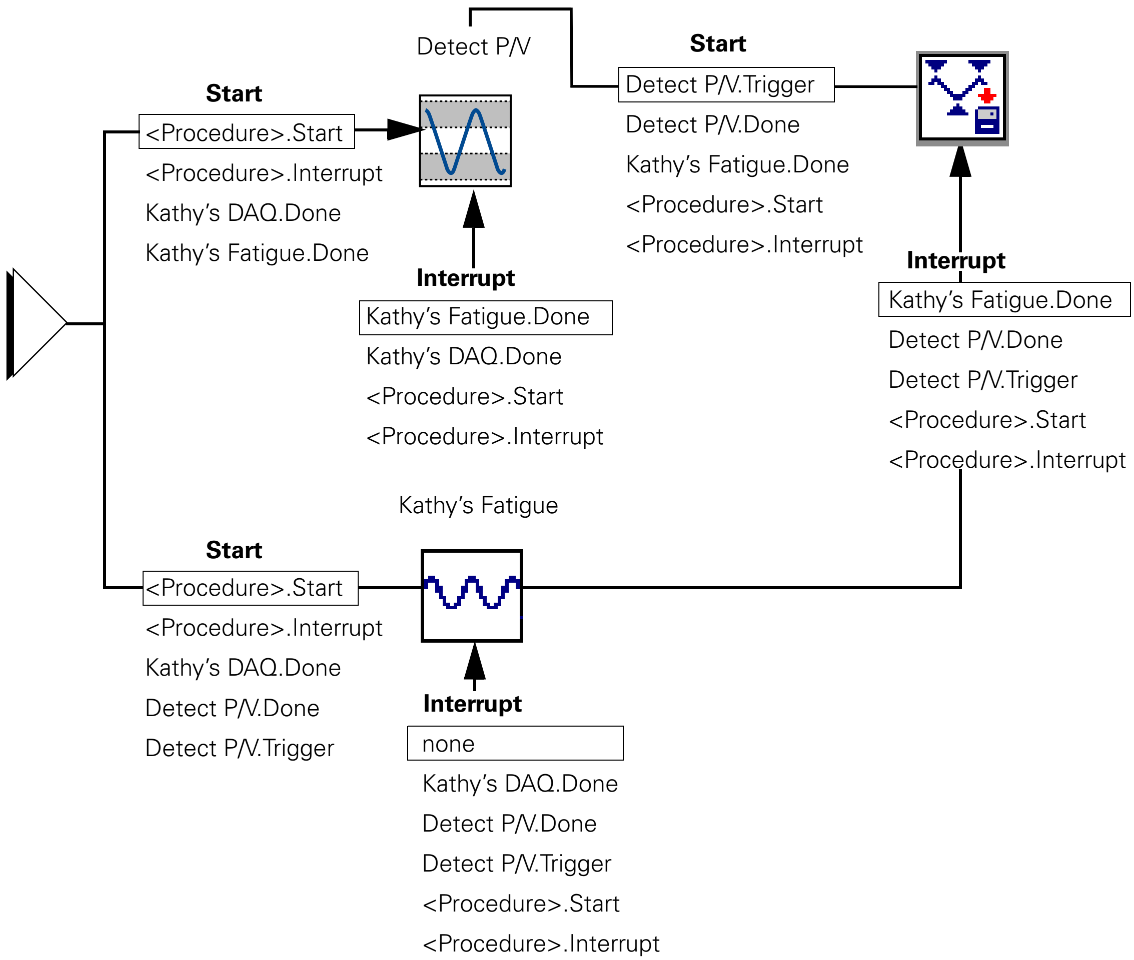
With this configuration, this procedure will acquire batches of peak/valley data as it detects specified changes in the feedback's peaks and valleys. Data will only be acquired when the Detect P/V process issues a trigger signal, rather than in a continuous stream.

Since its job is to continually monitor feedback, the Detect P/V process will never generate a done signal. Because of this, you typically will not use its **Detect P/V.Done** signal to start or interrupt any other process.

We use the **Kathy's Fatigue.Done** signal to interrupt the Detect P/V process (and generate a **Detect P/V.Done** signal). This prevents the test item running indefinitely.

Procedure diagram

The diagram below illustrates the connections between the processes used in the trigger signal example. The diagram illustrates how the trigger signal is used.



Note *To simplify this illustration, some of the signals normally available in this scenario are not shown.*

Working with Processes

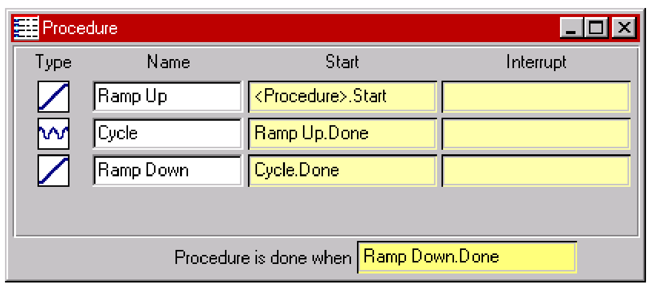
Basic Sequencing Example

Assume when you start your test you want the program to ramp up, cycle for some time, ramp down, and then end.

You would sequence your processes to run in the following order:

- The “Ramp Up” Segment process should start when the procedure starts.
- The “Cycle” Cyclic process should start when the “Ramp Up” process is done.
- The “Ramp Down” Segment process should start when the “Cycle” Cyclic process is done.
- The Procedure should end when the “Ramp Down” Segment process is done.

The following procedure table illustrates this:



How to Edit Sequencing Trigger Signals

Procedure and group tables that are populated with processes contain three areas where you can edit trigger signals:

- The **Start** box
- The **Interrupt** box
- The **Procedure** (or **Group**) **is done when** box.

MPT selects default trigger signals for the **Start** box. The **Interrupt** and **Procedure** (or **Group**) **is done when** boxes are blank by default.

To display a trigger list

Even if it is blank, each box contains a hidden trigger signal list. To display the list, double-click the box.

To select a trigger and hide the list

When the trigger signal list is displayed, you can select any individual trigger signal in the list by double-clicking it. This will replace whatever the box had in it before with the current selection, and then close the list.

Note *You can also select a trigger signal (and hide the list) by highlighting the trigger signal in the list and pressing **Enter**.*

To remove a trigger and hide the list

To remove the current trigger signal from a box, double-click the box to display the boxes' trigger signal list, then double-click the same trigger signal in the list. This will clear the box and hide the list.

To select multiple trigger signals in a list

To select multiple trigger signals in a list, you can hold down the **Shift** key to select a range of trigger signals, or hold down the **Ctrl** key to select a trigger signals arbitrarily. When you select multiple signals, the process will perform the action (start the process, interrupt the process, end the procedure or group) when any of the selected signals is received.

How to Set/Edit the Interrupt Signal

1. To specify the interrupt signal(s), double-click the **Interrupt** list box to display the signal list.
2. In the signal list, hold the **Ctrl** key and click each signal you want to interrupt the procedure. (If you select multiple signals, the procedure will interrupt when any of the selected signals is received.)
3. Press **Enter**.

If you want to clear your interrupt signal selections:

1. Double-click the signal name to display the signal list.
2. In the signal list, hold the **Ctrl** key and click each signal you want to remove from the list.
3. Press **Enter**.

Note *You can deselect all of the signals highlighted and exit the list by double-clicking any of the individual selections highlighted in the list.*

How to Specify/Change the Procedure Done Signal

1. To specify the procedure done signal(s), double-click the **Procedure is done when** box to display the signal list.
2. In the signal list, hold the **Ctrl** key and click each signal you want to end the procedure. (If you select multiple signals, the procedure will end when any of the selected signals are received.)
3. Press **Enter**.

If you want to clear your procedure done signal selections:

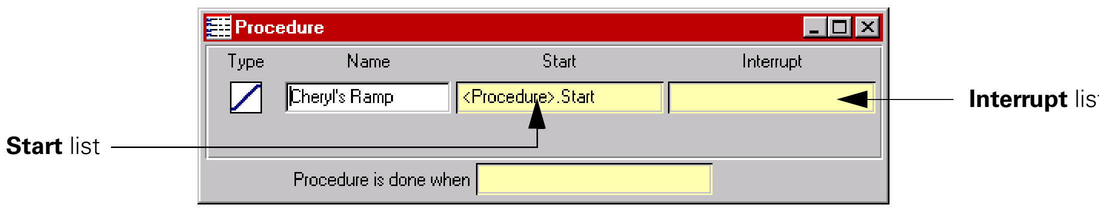
1. Double-click the signal name to display the signal list.
2. In the signal list, hold the **Ctrl** key and click each signal you want to remove from the list.
3. Press **Enter**.

Note *You can deselect all of the signals highlighted and exit the list by double-clicking any of the individual selections highlighted in the list.*

How to Sequence a Single Process (Example)

The procedure shown contains one **Segment Command** process named “Cheryl’s Ramp.” The diagram that follows illustrates the signals available to start and interrupt Cheryl’s Ramp.

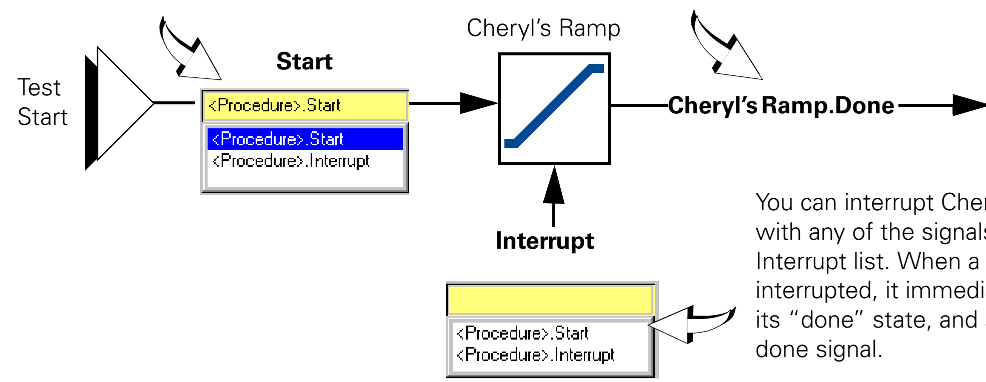
Window with Segment process named “Cheryl’s Ramp”



Signals received by and sent from “Cheryl’s Ramp”

To start Cheryl’s Ramp, you select a signal in the **Start** list. When you first create a test procedure, the application selects a signal for you. For instance, the <Procedure>.Start signal is automatically assigned to the first process on the procedure table.

When Cheryl’s Ramp has completed what it has been programmed to do, it sends a “Done” signal displayed as “Cheryl’s Ramp.Done.” You can use this signal to start or interrupt other processes.

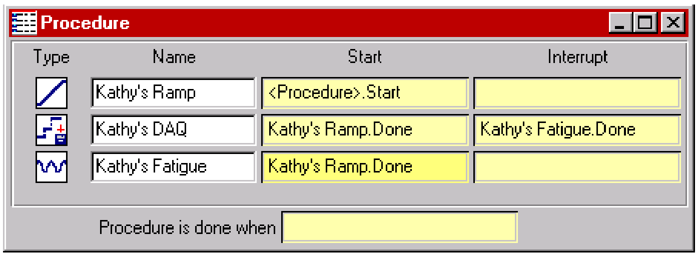


You can interrupt Cheryl’s Ramp with any of the signals listed in the Interrupt list. When a process is interrupted, it immediately attains its “done” state, and sends out a done signal.

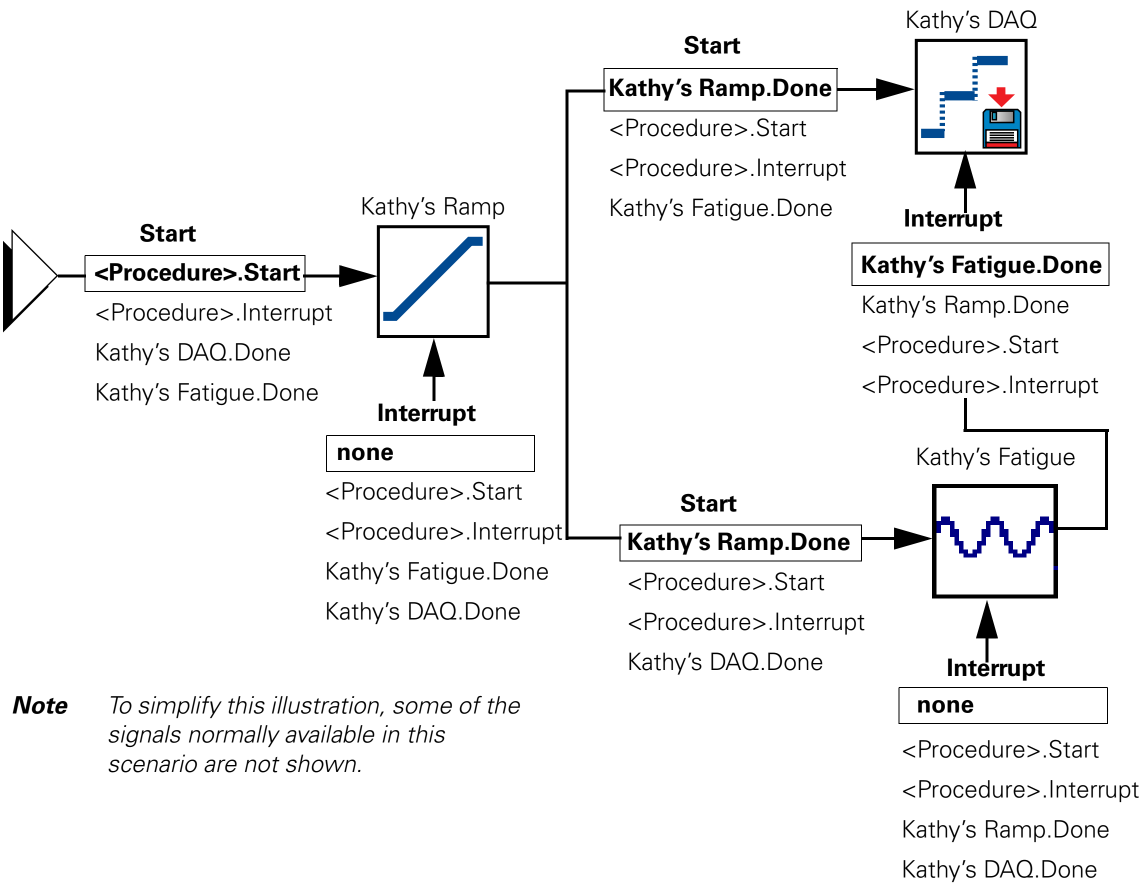
How to Sequence Multiple Processes (Example)

The window below contains three processes. The diagram that follows illustrates the signals selected and available for each process.

Window with Three Processes



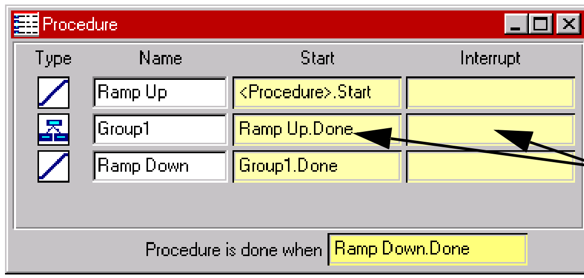
Signals received by and sent from processes



Note *To simplify this illustration, some of the signals normally available in this scenario are not shown.*

Sequencing Group Processes

A Group process is basically a sub-procedure that runs within your main MPT procedure (or within another Group process). You sequence a group process to run the same way you sequence other processes in your procedure table.



Set the Group process' **Start** and **Interrupt** signals in the procedure table.

Sequencing processes in the group

In addition to sequencing the Group process, you must also sequence processes defined in your Group process.

In your group table, you can set the Group process **Start** and **Interrupt** signals the same way do for other processes. When you open the group table, you will notice the **Start** and **Interrupt** signal lists include the **<Group>.Start** and **<Group>.Interrupt** signals.

If you set a process' **Start** or **Interrupt** control to **<Group>.Start**, the process will start or interrupt when the group process starts. If you set the **Start** or **Interrupt** control to **<Group>.Interrupt**, the process will start or interrupt when the group process is interrupted.

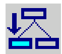
Ending a group process

The Group process procedure table has a **Group is done when** control located at the bottom of the window.

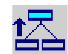
If you specify a signal in this list, when the specified signal is received, all active processes in this group will be interrupted and the Group process will end.

It is possible to configure your Group process so that the processes it defines will still execute normally even when the Group process is interrupted. For more information, see "About Group Processes Interrupts" on page 119.

How to Open the Group Table

Method 1: Click the Group process icon in your procedure table, then click  on the MPT toolbar.

Method 2: Right-click on the Group process icon and select **Open Table** from the menu.

To return to the previous procedure (or group) table, click 

About Group Process Naming Conventions

You can add a Group process to the main procedure table or to another Group processes (making a group within a group).

The title bar of the Group process window displays where the group is relative to the main procedure table.

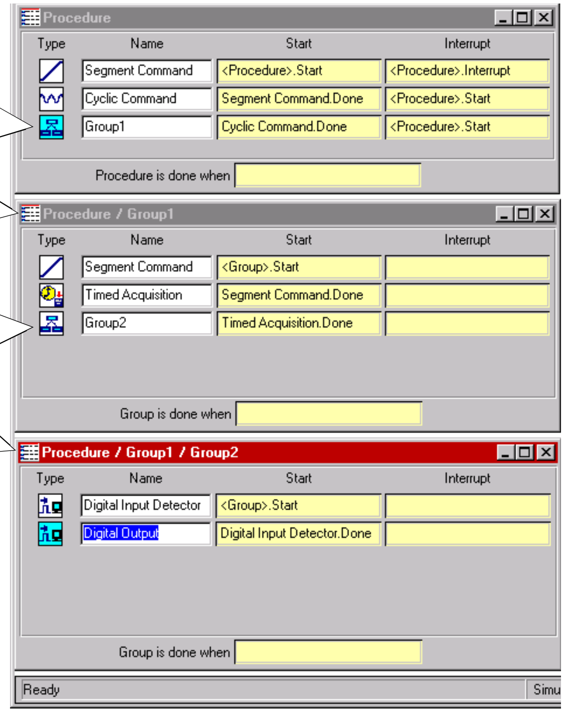
- For example, if your main procedure contains a group process, the group process title bar would read—Procedure/*Group Name*.
- If your group contained another group, the title bar for the second group process would read—Procedure/*Group Name*/*Group Name*.

This is an example of how the naming conventions work:

Two separate Group processes have been added to this procedure table.

We named the first Group process "Group1." That name then appears in the title bar of the first Group process window.

We named the second Group process "Group2." That name then appears in the title bar of the second Group process window.



About Group Processes Interrupts

Normal interrupt

Group processes have an interrupt terminal. By default, all of the processes inside a Group process are interrupted when the Group process is interrupted, and the Group process generates a done signal immediately. You can change this behavior by creating a user defined interrupt.

User defined interrupt

You can connect the interrupt terminal of a Group process to the start or interrupt terminals of any of the processes inside the Group process. When you do this, you are creating a user defined interrupt.

When you create a user defined interrupt, only the active processes inside of the Group process that are connected to the Group process' interrupt terminal are affected when an interrupt occurs. The other processes in the Group process are not affected, and execute normally.

Repeat counts in interrupted groups

In all instances of receiving an interrupt, a Group process will not perform a repeat count. When a Group process is interrupted, the processes inside the Group complete their activities according to how they are connected to the Group process' Interrupt terminal.

Normal interrupt versus user defined interrupt

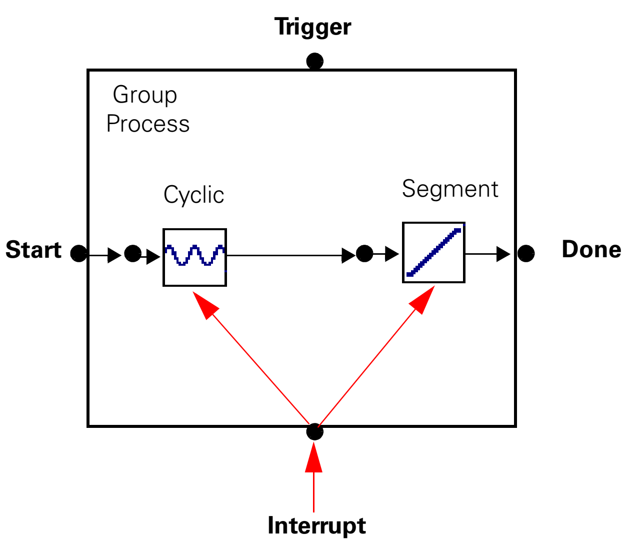
The following examples illustrate the differences between a normal interrupt and a user defined interrupt on a Group process.

Note *The end effect of user defined interrupts may be difficult to predict in complex procedures. It is good practice to perform a test run of your procedure in the simulation mode before actual testing.*

Suppose we have a Group process that contains a Cyclic process and a Segment process, and that the Group receives an interrupt while the Cyclic process is active.

If neither the Cyclic or Segment processes have an interrupt selected, the following things occur when the Group is interrupted:

- The Cyclic process ends immediately.
- The Segment process ends immediately.
- The Group process ends immediately.



Normal Interrupt to a Group Process

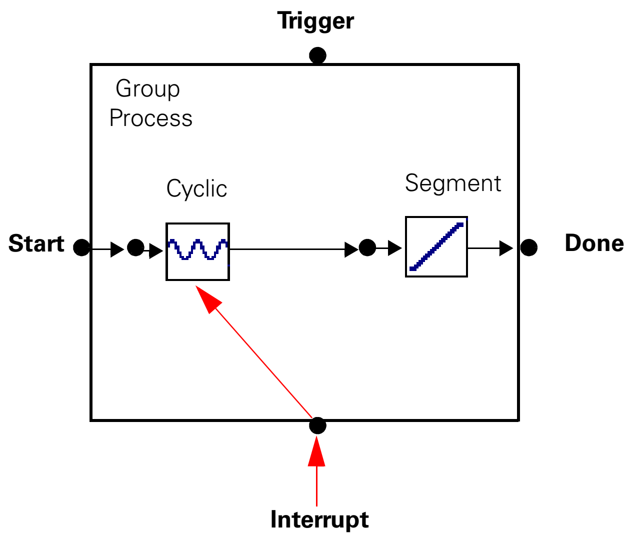
When the Group process is interrupted, the the Group ends immediately.

User-defined interrupt example

Suppose we start with the same scenario, but in this case, the Cyclic process' interrupt is set to **<Group>.Interrupt**.

In this case, when the Group is interrupted, the following things happen:

- The Cyclic process ends immediately.
- The Segment process executes normally.
- When the Segment process is done, the Group process ends.



User Defined Interrupt to a Group Process

When the Group process is interrupted, the Cyclic process ends immediately, but the Segment process executes normally.

Disabling Processes

You can disable a process in the following ways:

1. On the **General** tab in the process **Parameters** window, clear **Process Enabled**.

   This prevents the process from starting and generating any signals used to start or interrupt another process. If you disable a command process this way, all subsequent command processes will be disabled. When you do this, the process will appear with a red “X” in the left margin of the Procedure (or Group) table.

2. On the **General** tab of the process’ **Parameters** window, enter zero.

   This method starts the process, but immediately sends out the process “done” signal to other processes. This method works best if you want to disable a command process without disabling all subsequent command processes. When you do this, the process will appear with a downward pointing green arrow in the left margin of the Procedure (or Group) table.

3. On the Procedure table, select the process, right-click the mouse, then click the **Enable** selection to uncheck it. When you do this, the process will appear with a red “X” in the left margin of the Procedure (or Group) table.

Chapter 5

Windows and Controls

Initial MPT Display

When you have an existing station configuration open with the Station Manager application, then select **MultiPurpose TestWare** from the **Applications** menu, the display changes and appears as shown.

- The MPT control panel provides access to all the functions needed to create, edit, and run procedures for the current station configuration.
- The run, stop, and hold controls pertain to procedures.
- Also, if the current procedure includes an Operator Event process, a special panel will appear in the initial display.

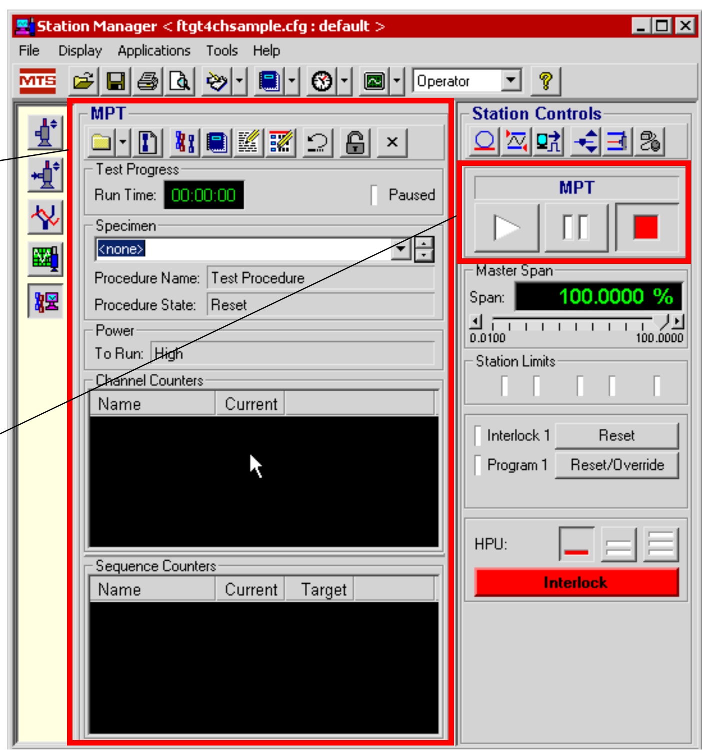
Note *When you select MPT this way, you can create, edit, and run procedures. You can also select an 'edit only' instance of MPT (**Applications** > **MultiPurpose TestWare (Edit Only)**) which allows you to run one procedure while editing other procedures.*

Initial MPT display

MPT control panel

Run, hold, and stop controls pertain to procedures when MPT is selected as the controlling application.

.



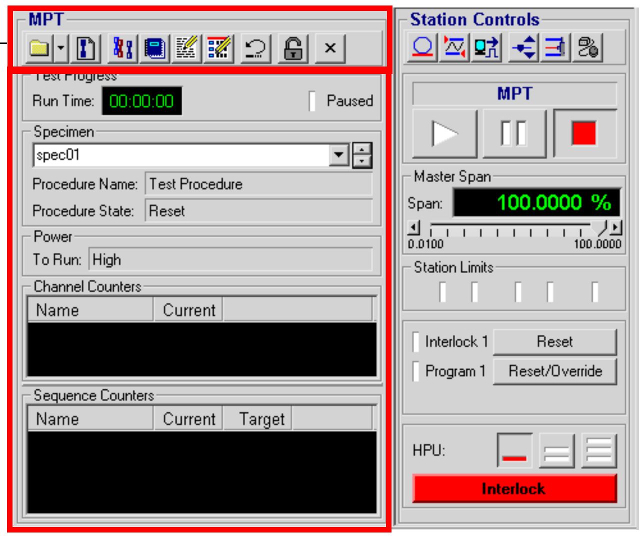
MPT Control Panel

Purpose

With the MPT control panel you can:

- Access common controls from the toolbar (including the **MPT Procedure Editor**, the place where you build procedures)
- Monitor the test's status and counter information (channel, sequence, and run time counters)
- Create and rename MPT specimens

The MPT control panel includes a toolbar that performs actions (like unlocking procedures) and provides access to editors (like the **MPT Procedure Editor** and the **MPT Specimen Editor**) and the **MPT Specimen Log**.

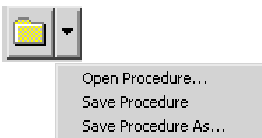
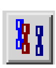


For information about setting MPT control panel preferences, see "Control Panel Tab" on page 186.

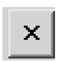
MPT Control Panel Toolbar

The buttons that comprise the MPT control panel toolbar provide quick access to common commands and windows.

MPT Control Panel Toolbar (part 1 of 2)

ITEM	DESCRIPTION
	Displays the **Open Procedure** window, saves the current procedure, or displays the **Save Procedure As** window. You can use the **Open Procedure** window functions to navigate to, select, and open an existing procedure. You can use the **Save Procedure As** window functions to save the current procedure as a new file with a specified name and location. ***Note*** *If you are using **Save Procedure** for a previously unsaved file, the **Save Procedure As** window will open to allow you to place and name the procedure according to your wishes. However, after a procedure has been named and saved the first time, the system simply saves the procedure as is and does not open a window in the process.*
	Creates a new MPT specimen for the procedure that is currently loaded. The new folder is immediately visible in the **Specimen** panel.
	Displays the **MPT Specimen Editor** (see page 159). You can use the **MPT Specimen Editor** to create and edit multiple MPT specimens.
	Displays the **MPT Specimen Log** (see page 155), which contains messages that pertain to the current MPT specimen.
	Displays the **MPT Options Editor** (see page 165). With the **MPT Options Editor**, you can customize the MPT interface to suit your needs by defining control options and preferences.
	Displays the **MPT Procedure Editor** (see page 137), the place where you create and edit procedures consisting of MPT test processes. ***Note*** *To edit a selected procedure on the **MPT Procedure Editor**, you must first be in the **Edit** mode (see "Edit mode" on page 128).*
	Resets the current procedure to the beginning. Pressing run after pressing reset will restart the procedure from the beginning. ***Note*** *After the procedure completes, you must click **Reset** before you can run the procedure again on the same MPT specimen.*

MPT Control Panel Toolbar (part 2 of 2)

ITEM	DESCRIPTION
 **Edit** mode	Changes between edit and execute test modes. To change (toggle between) modes, click the edit /execute mode button. You must be in edit mode to create or make changes to your procedure. ***Note*** *You cannot switch to edit mode if the procedure is running or holding.*
 **Execute** mode	You must be in the execute mode to run your test. The procedure automatically switches to execute mode if you click run on the **Station Controls** panel. ***Note*** *When you change to execute mode, the MPT application becomes the controlling application of the control channels used in the procedure. You cannot use the **Station Manager** manual command controls on the control channels used in the procedure.*
	Exits the MultiPurpose TestWare application.

Test Progress Panel

Purpose

The **Test Progress** panel displays the elapsed run time of the test that is in progress and indicates if the test is currently paused.

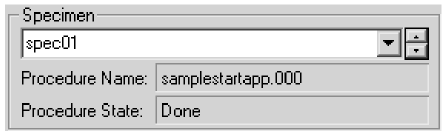
MPT Test Progress Panel

ITEM	DESCRIPTION
Run Time	Displays the elapsed time of the current test (the elapsed run time does not include time spent in hold or stop states). You can turn the **Run Time** display on-and-off and select a display format using the **Control Panel** tab of the **MPT Options Editor**. See “Control Panel Tab” on page 186. When the procedure is locked and reset, you can access the **Run Time** menu by performing a right-mouse click with the cursor positioned over the **Run Time** counter. When you right-mouse click, the **Reset** menu item appears. Select the **Reset** menu item to reset the **Run Time** counter to zero.
Paused	Indicates when the procedure has been paused by a Program Control process. The paused state is not the same as the hold state. • When the test is holding, you must click the run button to resume the test. • When the test is paused, the test must be resumed by another Program Control process (set to **Program Resume**). For more information about pausing a test with the Program Control process, see “Program Control” on page 391.

Specimen Panel

Purpose

The **Specimen** panel displays the current MPT specimen selection and the **Procedure Name** and **Procedure State** associated with the selected specimen. In addition, this panel allows you to navigate to other MPT specimens that are displayed in the **Specimen** list.



MPT Specimen Panel (part 1 of 2)

ITEM	DESCRIPTION
Specimen	Displays the MPT specimen that will be used for the procedure. You can select the desired MPT specimen from the **Specimen** list or click  to create a new MPT specimen. Use the up and down scroll arrows to select an available MPT specimen from the list. You can rename the selected MPT specimen (if you are in edit mode) by typing over the folder name in the **Specimen** box. ***Note*** *New MPT specimens that are created using the*  *icon are assigned names that are based on a default root name (in the example shown above, "spec" is the default root) with an incremental number suffix. The default root can be changed during installation or by using the name template in the* ***MPT Specimen Editor****.* ***Note*** *For more information about MPT specimens, see "Working with MPT Specimens" on page 67.* ***Note*** *To cut, copy, paste, or delete the displayed MPT specimen name, right-mouse click on the current MPT specimen name and select the desired right-mouse menu item.*

MPT Specimen Panel (part 2 of 2)

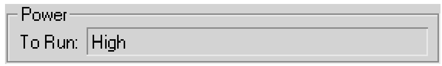
ITEM	DESCRIPTION
Procedure Name	Displays the name of the current procedure selection. The procedure name is established when you save the procedure.
Procedure State	Displays the current state of the procedure, which may be one of the following: • **Reset**—The test is at the beginning but has not been started. • **In Progress**—The test has started but is not yet completed. • **Done**—The test has been completed. ***Note*** *To run the procedure on the same MPT specimen again, press **Reset**. Otherwise, create a new MPT specimen for the next instance of running the same procedure.* • **Error—**An error has occurred (you must click **Reset** to run the test again).

Windows and Controls

Power Panel

Purpose

The **Power** panel displays the hydraulic supply pressure required to run the selected test.



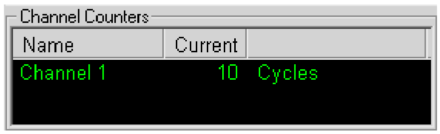
MPT Power Panel

ITEM	DESCRIPTION
To Run	Displays the hydraulic pressure required to run the control channels in the current procedure (hydraulic pressure options are **High**, **High/Low**, **Off/Interlocked**, or **Off**). ***Note*** *Power status applies only to control channels. If a procedure does not contain control channels, the power status will have no effect on the procedure. For instance, if hydraulic power is removed from a procedure without control channels, the procedure will still run.* You can specify the required hydraulic pressure for the control channels of the current procedure using the **Execution** tab of the **MPT Options Editor** (see “Execution Tab” on page 166). ***Note*** *You must be in edit mode to change settings on the* ***Execution*** *tab.* The **High** selection applies to most testing situations. **High** requires you to operate MPT with high hydraulic pressure applied to the station’s Hydraulic Service Manifolds (HSM). The **Off/Interlocked** and **Off** selections are typically used while you are learning the application software or running test simulations. The **Off/Interlocked** selection disregards active interlocks for the sake of allowing you to run the test unhindered.

Channel Counters Panel

Purpose

The **Channel Counters** panel displays the running total of completed command channel cycles or segments during test execution.



MPT Channel Counters Panel

ITEM	DESCRIPTION
Channel Counters	Displays a running total of the cycles or segments that have been executed during the test for each command channel specified in the procedure. The **Channel Counters** panel will display all channels that are selected for use during a test. A channel does not have to be used in every process to be displayed. You can specify the **Channel Counters** display format using the **Control Panel** tab of the **MPT Options Editor**. See “Control Panel Tab” on page 186. You must be in edit mode to change the **Control Panel** settings. To access menu items that pertain to **Channel Counters** position the cursor over the **Channel Counters** display and perform a right-mouse click. When you do this, the **Reset Counters** and **Set Counters...** menu items appear (provided that the procedure is locked and in the reset state). The **Reset Counters** selection resets all active channel counters to zero. The **Set Counters...** selection allows you to manually set each channel counter to a user specified value.

Sequence Counters Panel

Purpose

The **Sequence Counters** panel displays process counters for each specified counter using the selected process counter format.

MPT Sequence Counters Panel

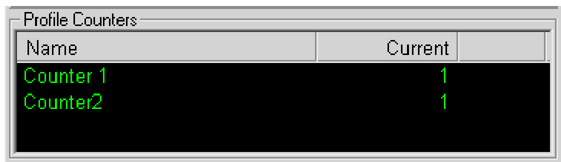
ITEM	DESCRIPTION
Sequence Counters	Displays individual process counters. You can specify the counter type using the **General** tab of each process parameters window. • If you specified a **Fixed** counter, the counter label and counter will be displayed during the entire procedure. • If you specified a **Transient** counter, the counter label and counter are displayed only while the process is active. For more information about using counters, see “Working with Counters” on page 27.

Process Specific Panels

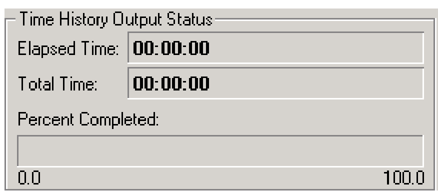
Purpose

Process specific panels display only when certain processes are selected from the **Process Palette** and are configured specifically to display at or during run time. Examples of some of these displays are shown below. For more information about these panels, refer to “Process Descriptions” on page 195.

Profile Counter Panel

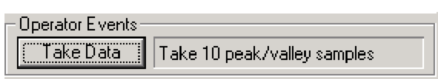


Process Status Panel



Note *Applies to Time History Output, Time History Input, and Road Surface Output processes only.*

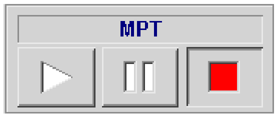
Operator Events Panel



Run/Stop/Hold Panel

Purpose

The run/stop/hold panel controls run, stop, and hold test activities for the current application. When you select MultiPurpose TestWare from the **Applications** menu, it becomes the controlling application and “MPT” appears on this panel.



MPT Run/Stop/Hold Panel

ITEM	DESCRIPTION
	Starts the procedure. When you click this button, the run indicator blinks while the procedure transitions from the current state to the “run” state. Once the run state is achieved the indicator remains constant unless additional transitions or program induced pauses occur. If the duration of the transitions or pauses are long enough, the indicator will blink again.
	Suspends all active command and data acquisition processes. ***Note*** *Active limit detector and digital input processes will remain active.* When you click hold, the hold indicator blinks while the procedure transitions from the current state to the “hold” state. Once the hold state is achieved, the indicator remains constant. To resume the run state from the hold state, click run. ***Note*** *The* hold *button may be disabled if* ***Ignore Hold Event*** *is selected on the* ***MPT Options Editor*** *window* ***Execution*** *tab. Refer to “MPT Options Editor” on page 165 for more information.*
	Stops the procedure. When you click stop, the stop indicator blinks while the procedure transitions from the current state to the “stop” state. Once the stop state is achieved, the indicator remains constant.

MPT Procedure Editor

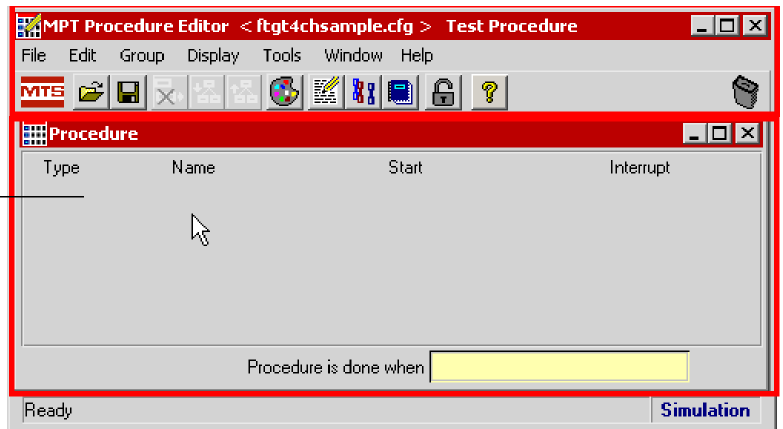
Purpose

The **MPT Procedure Editor** is where you create procedures by combining and defining the parameters of individual test processes available on the **Process Palette**.

To display the **MPT Procedure Editor**, click on this icon .

The **MPT Procedure Editor** includes the menu and toolbar, and the **Procedure** table.

You create procedures by populating the **Procedure** table with test processes from the **Process Palette**.



"**Simulation**" appears in the status bar when you run the "Demo" (demonstration) System Loader program.

MPT Procedure Editor Menu

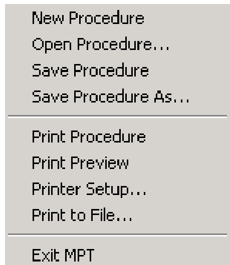
The following menus are available on the **MPT Procedure Editor** window:

- "File Menu" on page 139
- "Edit Menu" on page 140
- "Group Menu" on page 141
- "Display Menu" on page 142
- "Tools Menu" on page 143
- "Window Menu" on page 144
- "Help Menu" on page 145

File Menu

Purpose

Use the **File** menu commands to open, create, save, and print procedures.



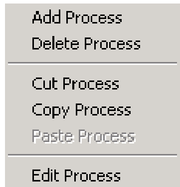
Note *When viewing the output files generated by the **Print Procedure**, **Print Preview**, and **Print to File** selections, items preceded by an asterisk (*) have been changed since the procedure was last opened or saved.*

File Menu

ITEM	DESCRIPTION
New Procedure	Opens a new, untitled procedure.
Open Procedure...	Displays the **Open Procedure** window.
Save Procedure	Saves the current procedure. If saving a procedure for the first time, the **Save Procedure As** window will open.
Save Procedure As...	Displays the **Save Procedure As** window.
Print Procedure	Prints the current procedure.
Print Preview	Displays the procedure parameters on-screen.
Printer Setup...	Displays the printer setup window. ***Note*** *This window varies with the printer driver you are using.*
Print to File...	Displays the **Print Procedure to File** window. This window allows you to name the output file, select the file type, and navigate to a desired print location.
Exit MPT	Quits the MultiPurpose TestWare application.

Edit Menu

Purpose Use the **Edit** menu commands to add, delete, copy, paste, and edit processes that are selected in the **Procedure** table.



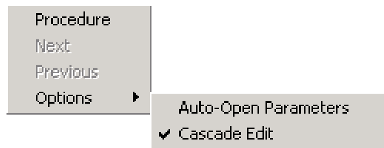
Edit Menu

ITEM	DESCRIPTION
Add Process	Adds the process selected on the **Process Palette** to the active **Procedure** table.
Delete Process	Deletes the process (or processes) selected in the active **Procedure** table.
Cut Process	Removes the process (or processes) selected in the active **Procedure** table and copies it to the clipboard.
Copy Process	Copies the process (or processes) selected in the active **Procedure** table.
Paste Process	Pastes the process (or processes) to the active **Procedure** table.
Edit Process	Displays the **Parameters** window for the process selected in the active **Procedure** table.

Group Menu

Purpose

Use the **Group** menu commands to restore, navigate through, and control specific **Group** table functions.



Group Menu

Item	Description
Procedure	Restores the main **Procedure** table if it has been minimized or hidden.
Next	Opens the **Group** table for the selected group process. ***Note*** *This command is only available when you select a **Group** process icon.*
Previous	Displays the parent **Procedure** table for the current **Group** table. ***Note*** *This command is only available when you are in a Group table.*
Options	**Auto-Open Parameters**—Automatically opens the process parameters window for any process you add to the **Group** table. **Cascade Edit**—Displays your procedure and group process windows in a cascading window style. If **Cascade Edit** is disabled, only one procedure table will be displayed at a time.

Display Menu

Purpose

Use the **Display** menu commands to:

- Change between the test edit mode and test execution mode.
- Display the **Process Palette**.
- Display the **MPT Specimen Log**.

Display Menu

ITEM	DESCRIPTION
Change to edit mode/ Change to execute mode	Changes the current test mode.
Process Palette	Displays the **Process Palette**.
Specimen Log	Displays the **MPT Specimen Log** for the current MPT specimen.

Tools Menu

Purpose

Use the **Tools** menu commands to display various editors used to customize the MPT environment and/or the current procedure.

Tools Menu

ITEM	DESCRIPTION
Specimen Editor	Displays the **"MPT Specimen Editor"** on page 159.
Unit Set Editor	Displays the **"Unit Assignment Set Editor"** on page 162.
Options Editor	Displays the **"MPT Options Editor"** on page 165.

Window Menu

Purpose

Use the **Window** menu commands to configure MPT client windows within the main MPT window.

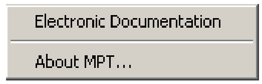
Window Menu

ITEM	DESCRIPTION
Cascade	Cascades all of the open **Procedure**, **Group**, and process **Parameters** windows.
Tile	Tiles all of the open **Procedure**, **Group**, and process **Parameters** windows.
Arrange Icons	Arranges all minimized **Procedure**, **Group**, and process **Parameters** windows to the lower left corner of the main **MPT** display.
Toolbar	Hides or displays the **MPT** toolbar.
Status Bar	Hides or displays the **MPT** status bar.
Window List	Lists all the open **Procedure**, **Group**, and process **Parameters** windows, in the order they were opened. A check mark is displayed next to the active window. You can click the window name to make it the active window.

Help Menu

Purpose

Use the **Help** menu to access electronic documentation and to view general application information.



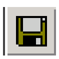
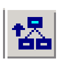
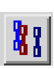
Help Menu

Item	Description
Electronic Documentation	Opens the MultiPurpose TestWare manual in the portable document file (.pdf) format.
About MPT...	Displays the **About MPT** window.

MPT Procedure Editor Toolbar

The buttons that comprise the **MPT Procedure Editor** toolbar provide quick access to common commands and windows.

MPT Procedure Editor Toolbar (part 1 of 2)

ITEM	DESCRIPTION
	Displays the **Open Procedure** window.
	Saves the current procedure.
	Deletes the process (or processes) selected in the active **Procedure** table.
	Opens the next group table. This button is only active if there are multiple group tables available.
	Opens the previous group table. This button is only active if there are multiple group tables available.
	Displays the **Process Palette** (see page 154). You can select processes from the palette to copy to the **MPT Procedure Editor**, or you can drag and drop processes from the palette to the **MPT Procedure Editor**.
	Displays the **MPT Options Editor** (see page 165). With the **MPT Options Editor**, you can select control options and preferences for the current procedure.
	Displays the **MPT Specimen Editor** (see page 159). You can use the **MPT Specimen Editor** to create and edit multiple MPT specimens.
	Displays the **MPT Specimen Log** (see page 155) for the current MPT specimen.

MPT Procedure Editor Toolbar (part 2 of 2)

ITEM	DESCRIPTION
 **Edit mode**  **Execute mode**	Changes between edit and execute test modes. To change (toggle between) modes, click the edit /execute mode button. You must be in edit mode to create or make changes to your procedure. ***Note*** *You cannot switch to edit mode if the procedure is running or holding.* You must be in execute mode to run your test. The procedure automatically switches to execute mode if you click **Run** on the **Station Controls** panel. ***Note*** *When you change to execute mode, MPT takes control of the various resources used in the procedure, such as control channels, digitial output channels, and so on. You cannot use these resources with Station Manager until you unlock the procedure and attain the edit mode.*
	Displays the MultiPurpose TestWare manual in the portable document file (.pdf) format.
	Trash can. Processes are not stored in the trash can as they are in the Windows Recycling Bin. They are deleted immediately after you drag them to the trash can icon.

Procedure Table

Purpose

You use this window to organize and synchronize the processes that make up your test.

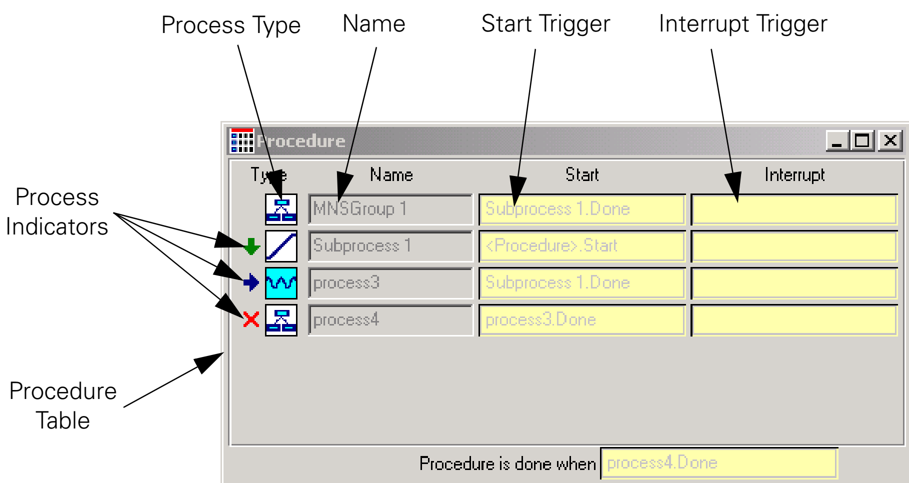
Access

From the **MPT Procedure Editor** window, the access paths are:

File > Open (or New) Procedure

or

Group > Procedure...



Procedure Table

ITEM	DESCRIPTION
Process indicators	Indicate the status of the adjacent process. Indicators have the following meanings:  The process is enabled and active.  The process has been disabled.  The process is enabled, but configured to execute zero times.
Type	Displays the individual process icons that you have selected. You can double-click an icon to open the associated process parameters window. In addition, if you right-click on an icon you can open the parameters window associated with the process, open a new process (or group) table, enable or disable the process, and delete the process.
Name	Specifies the name for this process. This field is directly editable, simply highlight the displayed name and type your new name. You can also double click on the process icon and use the **General** tab to specify the process name.
Start	Specifies the triggers or events that cause this process to start.
Interrupt	Specifies the triggers or events that interrupt this process.
Procedure is done when	Specifies the triggers or events that cause the end of the procedure.

Other Windows and Control Descriptions

For descriptions of the controls available in a specific window, refer to the following table.

Note *For descriptions of the MPT toolbar, see "MPT Procedure Editor Toolbar" on page 146.*

MPT Windows

WINDOW	MPT PROCEDURE EDITOR MENU
"Open Procedure" on page 151	File
"Save Procedure As" on page 152	File
"Print Procedure to File" on page 153	File
"Process Palette" on page 154	Display
"MPT Specimen Log" on page 155	Display
"MPT Specimen Editor" on page 159	Tools
"Unit Assignment Set Editor" on page 162	Tools
"MPT Options Editor" on page 165	Tools
"About MPT" on page 193	Help

Windows and Controls

Open Procedure

Purpose Use the **Open Procedure** window to navigate to and to open existing procedure files.

Access From the **MPT Procedure Editor** window, the access paths are:

File > Open Procedure...

or

Open Procedure Window

ITEM	DESCRIPTION
Look in:	Lists the available drives and directories.
File name:	Displays the file name of the selected file.
Files of type:	Lists the types of files available. For example, the extension for a procedure file is (.000).

Save Procedure As

Purpose

Use the **Save Procedure** or **Save Procedure As...** command to save your current procedure.

Access

From the **MPT Procedure Editor** window, the access paths are:

File > Save Procedure or Save Procedure As...

or

If the current procedure is unnamed, selecting **Save Procedure** or **Save Procedure As...** will open the **Save Procedure As** window. This window prompts you for a procedure name and a folder location. When you click **Save** on the **Save Procedure As** window, the application creates a new procedure using the specified file name and closes the original procedure.

If the current procedure is already named, selecting **Save Procedure** simply initiates the save function without opening any window.

Save Procedure As Window

ITEM	DESCRIPTION
Save in:	Lists available drives and directories where you can save your procedures.
File name:	Sets the name of your procedures.
Save as type:	Displays the procedure default file type and automatically appends a .000 extension to the name you enter. If you do not want the .000 extension, put a period (.) at the end of your name.

Print Procedure to File

Purpose

Use the **Print Procedure to File** window to create a text (.txt) file that contains the current procedure information.

Access

From the **MPT Procedure Editor** window, the access path is:

File > Print to File...

Note *We recommend that you use the default directory so your files are stored in a common directory. If desired, use the controls to change the directory.*

Print Procedure to File Window

ITEM	DESCRIPTION
Save in:	Lists the available drives and directories where you can save your text file.
File name:	Sets the name of your text file.
Save as type:	This selection automatically appends the .txt extension to the file name you enter. If you do not want the .txt extension, put a period (.) at the end of your file name.

Process Palette

Purpose

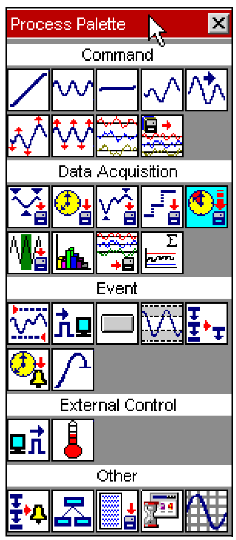
Use the **Process Palette** to select test processes by using their representative icons.

Access

From the **MPT Procedure Editor** window, the access paths are:

Display > Process Palette

or



Note *Some processes are optional and will appear on the Process Palette only when purchased separately.*

The **Process Palette** contains the icons of processes that you can add to the procedure table to build a procedure (For more information, see “Adding Processes to the Procedure Table” on page 99).

MPT Specimen Log

Purpose

Use the **MPT Specimen Log** to display a record of test events in the **Message Logs** window.

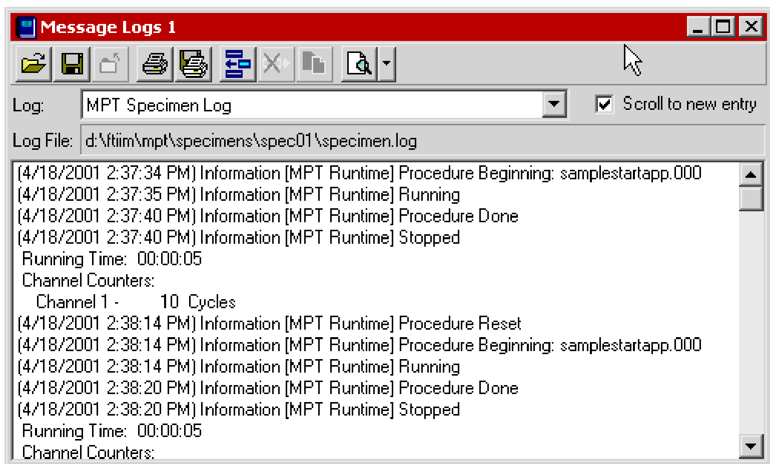
Access

From the **MPT Procedure Editor** window, the access paths are:

Display > Specimen Log

or

MPT control panel > 



The **MPT Specimen Log** records selected station and test events as they occur.

Note *The **MPT Specimen Log** is displayed on the **Message Logs** window, which can also display **Station Log** information (by selecting **Station Log** in the **Log** list).*

Events that can be logged include file events, resource conflicts, hydraulic status changes, station state changes, detector activity, hardware over temperature conditions, and full-scale changes.

Counter data is saved to the **MPT Specimen Log** automatically and manually (for more information, see "Working with Counters" on page 27).

Windows and Controls

About auto-archiving

The MPT application automatically saves your **MPT Specimen Log** when it accumulates at least 1000 log entries. When log entries are saved to the log file, they are cleared from the **MPT Specimen Log** display.

Specimen log file naming

The default name for the current **MPT Specimen Log** file is *specimen .log*.

Archived log files are named *specimen000.log*, *specimen001.log*, *specimen002.log*, and so forth, up to *specimen500.log*.

Note *The maximum number of log files that can be saved for a particular test is 500.*

Once you reach 500 log files, additional log files overwrite the oldest log files in order (typically archiving restarts at *specimen 000.log*).

When a log file is saved (either manually or automatically), a log entry is made noting the archival. This entry includes a time/date stamp and information about where the file was saved. This entry becomes the first entry in the current log.

⚠ CAUTION

The MPT application can unexpectedly quit if you run out of disk space when saving log files.

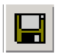
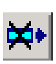
Quitting while a test is in progress can result in a loss of data.

If you think your test may generate an excessive number of MPT Specimen Log entries, you should filter the messages written to the MPT Specimen Log with the **MPT Options Editor**. For information about filtering MPT Specimen Log entries, see “Specimen Log Tab” on page 183.

If the MPT application shuts down as a result of log files exhausting disk space, you must remove some of the MPT Specimen Log files from your disk before you restart the system software.

Windows and Controls

Message Logs Window (part 1 of 2)

ITEM	DESCRIPTION
	Displays the **Open Message Log File** window where you can open any archived or saved message log file (.log) you wish to view.
	Saves message log entries to a log file. If you click the Save icon and no log entries are selected, all of the messages in the **MPT Specimen Log** will be saved to file and the entire log will be cleared. If you click the Save icon and one or more log entries are selected, all of the messages up to and including the last selected message are saved to file and cleared from the log.
	Closes the selected **MPT Specimen Log** and removes the log from the **Log** list.
	Opens the **Message Log Print** window. The **Message Log Print** window allows you to set certain print parameters (such as **Print Range** and **Print Filters**), access the **Print Setup** window, and initiate printing of the selected message log.
	Displays the **Message Log Print To File** window. The **Message Log Print To File** window allows you to save your log as a text (.txt) file.
	Adds your message entry to all active logs. ***Note*** *User entries are sent to all active logs (Station log, Basic TestWare log, MPT Specimen log); user entries are not affected by filtering attributes.*
	Deletes user-entered messages from the message log. ***Note*** *You can only delete user-entered messages.*
	Copies the selected **Message Log** entries to the clipboard.

Message Logs Window (part 2 of 2)

ITEM	DESCRIPTION
	Enables all the display options or allows individual display option enable/disable functions. If you click on the enable icon, all display functions (Date/Time, Severity, Source) are enabled. If you click on the down arrow, the individual display functions appear. If you click on a function, the function state changes between enabled and disabled. If a check mark appears next to the function, that function is enabled.
Log	Displays the current log file. Use the down arrow to display other log files that are available.
Log File	Displays the path to the current log file.
Scroll to new entry	Disables or enables the **MPT Specimen Log** scrolling function. This function allows the message log to automatically shift (scroll) to display new entries as they occur. ***Note*** *By default, the **Scroll to new entry** box is enabled.* If **Scroll to new entry** is selected, new log entries will appear at the bottom of the log as they occur. The log will scroll down automatically to ensure the latest entry is displayed. If **Scroll to new entry** is cleared, new log entries will be added to the bottom of the log as they occur, but the log will not scroll down. New entries may not be visible unless you manually scroll to the bottom of the log. This selection allows you to review the contents of the log without the window ‘jumping’ to the bottom of the display each time a new log entry arrives.

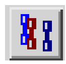
MPT Specimen Editor

Purpose

Use the **MPT Specimen Editor** to create, edit, and delete MPT specimens.

Access

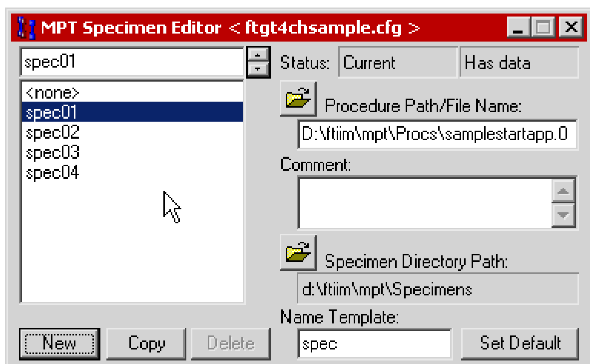
From the MPT control panel and the **MPT Procedure Editor**, the access path is:



or

Also, from the **MPT Procedure Editor** window, an alternate path is:

Tools > Specimen Editor



MPT specimens store a number of files associated with running a procedure. Before you can run your procedure, you must specify a new or existing MPT specimen for the test.

For information about working with MPT specimens, see "Working with MPT Specimens" on page 67.

MPT Specimen Editor (part 1 of 2)

ITEM	DESCRIPTION
MPT specimen list	Displays the MPT specimens saved in the selected **MPT Specimen Directory**. ***Note*** *While in the execute mode, if you click an MPT specimen that is currently loaded in the MPT control panel or in another MPT application, the entire list of MPT specimens will be dimmed and the controls will be unavailable. To make the controls available, you must use the up and down arrows to switch to an available MPT specimen.*
Status	Displays access and data status of the MPT specimen. In the first box, the MPT specimen access status is displayed: • **Available**—You can edit the MPT specimen. • **Unavailable**—You cannot edit the MPT specimen because it is currently assigned to another instance of the MPT application. • **Current**—The MPT specimen is currently assigned on the MPT control panel. You can edit but not delete this specimen. In the second box, the MPT specimen data status is displayed: • **Empty**—The MPT specimen does not contain test information. • **Has Data**—The MPT specimen contains test information.
 **Procedure Path/ File Name**	Displays the procedure associated with the selected MPT specimen. To choose a new procedure, click the **Procedure Path/File Name** button, and either enter the desired procedure or use the **Select Procedure File** window to link a different procedure to your current MPT specimen.
Comment	Enters comment information that is saved in the MPT specimen.
New	Creates a new MPT specimen. ***Note*** *The new MPT specimen is empty and is not linked to a procedure.*
Copy	Creates a new MPT specimen based on the selected MPT specimen. The new copy includes all attributes of the original MPT specimen, including procedure association, comments, and MPT specimen directory. ***Note*** *Test data and status are not copied.*
Delete	Deletes the selected MPT specimen.

MPT Specimen Editor (part 2 of 2)

ITEM	DESCRIPTION
 **Specimen Directory Path**	Specifies the directory where your MPT specimens are saved for the current MPT session. To choose a new directory, click the **Specimen Directory Path** button and use the **MPT Specimen Directory Path** window to select a new directory. Changes made to the MPT **Specimen Directory Path** window apply only to the current MPT session. The directory path will revert to the default directory path when MPT is reloaded. To change the default MPT specimen directory path, use the **Directories Tab** on the **MPT Options Editor**.
Name Template	Specifies the default label applied to MPT specimens you create. ***Note*** *The default MPT specimen name is "spec" (set during installation). When you create new MPT specimens, their default labels are "spec01," "spec02," "spec03," and so forth.* To change the default MPT specimen name, type a name in the **Name Template** box, and the new name will apply to the current MPT session. If you want to make this new name the new default name, click **Set Default**. Changes are effective immediately. The name template may also be changed during software installation. ***Note*** *It is good practice to add a space after your name entry. This will make additional instances of the name easier to read; for example, "dogbone 01, dogbone 02, dogbone 03," versus "dogbone01, dogbone02, dogbone03."*

Unit Assignment Set Editor

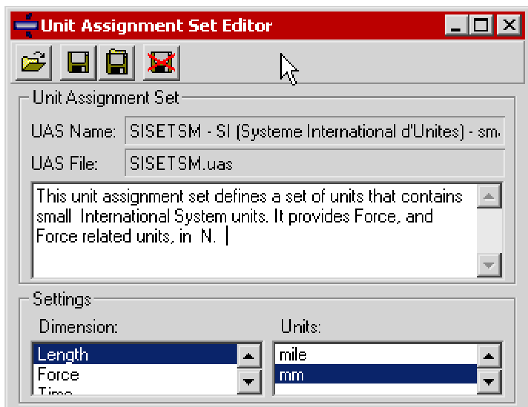
Purpose

Use the **Unit Assignment Set Editor** to create and modify unit assignment sets (UAS). Unit assignment sets define the units of measurement that are available for each dimension used in procedures. For descriptions of the unit assignment sets that are installed with your system software, see “Standard unit sets” on page 164.

Access

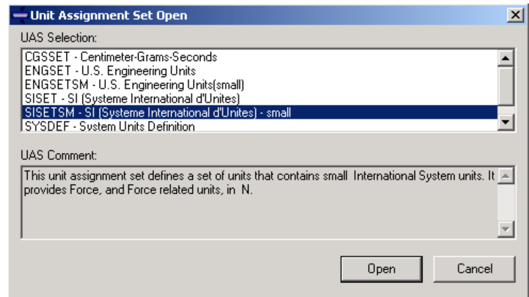
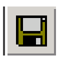
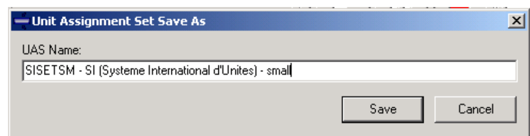
From the **MPT Procedure Editor** windows the access path is:

Tools > Unit Set Editor



Note *Initially, you specify the default UAS (unit assignment set) when you install the system software. To change the default UAS, use the Unit Selection Tab (see page 189) of the* ***MPT Options Editor.***

Unit Set Editor

Item	Description
 **Open**	Displays the **Unit Assignment Set Open** window. This window allows you to select the appropriate unit assignment set and to complete the open function by selecting the **Open** button. 
 **Save**	Saves any changes to the selected unit assignment set.
 **Save As**	Displays the **Unit Assignment Set Save As** window. This window allows you to view and edit the selected UAS Name and to complete the save function by selecting the **Save** button. 
 **Delete**	Deletes the selected unit assignment set.
UAS Name	Displays the name of the unit set currently selected.
UAS File	Displays the file name of the unit set currently selected.
Comment	Displays a comment about the selected UAS. We recommend you describe the characteristics of the unit set you create.
Dimension and **Units**	Displays the dimensions and corresponding units used by the system software. When you click a dimension name, the units assigned to the current set will be highlighted. To change a setting, click the dimension, and then click the desired units. ***Note*** *You can quickly find an item in the* ***Dimensions*** *or* ***Units*** *list boxes with the keyboard by typing the first letter of the desired item until it is displayed. For example: To select the engineering force unit kN, type the letter 'k' repeatedly until kN is highlighted in the list box.*

Standard unit sets

CGSSET - **Centimeters-Grams-Seconds**—Defines a set of units that is based upon centimeters, grams, and seconds.

ENGSET - **U.S. Engineering Units**—Defines a set of units that contains customary U.S. engineering units. It provides force-related units in “kip,” and length-related units in “in.”

ENGSETSM - **U.S. Engineering Units (small)**—Defines a set of units that contains customary U.S. engineering units. It provides force-related units, in “lbf,” and length-related units in “in.”

SISET - **SI (Systeme International d’Unites)**—Defines a set of units that contains customary international (metric) units. It provides force-related units in “kN,” and length-related units in “mm.”

SISETSM - **SI (Systeme International d’Unites) - small**—Defines a set of units that contains customary international (metric) units. It provides force, and force-related units in “N,” and length-related units in “mm.”

SYSDEF - **System Units Definition**—Contains a copy of the units that are used in the hardware and software to represent test values of interest.

MPT Options Editor

Use the **MPT Options Editor** to set preferences that are saved with the procedure.

The following tabs are available:

- "Execution Tab" on page 166
- "About Transition States" on page 173
- "Recovery Tab" on page 180
- "Specimen Log Tab" on page 183
- "Control Panel Tab" on page 186
- "Properties Tab" on page 188
- "Unit Selection Tab" on page 189
- "Directories Tab" on page 191

Saving option preferences

When you create a new procedure, it will contain default settings for all of the controls in the **Options Editor**, such as preferences for the data file format, command hold behavior, unit assignment set, etc. You can change these default settings by modifying the "default.000" file. For instructions, see "Default Templates" on page 449.

Windows and Controls

Execution Tab

Purpose

Use the **Execution** tab on the **MPT Options Editor** to specify options related to the execution of your test.

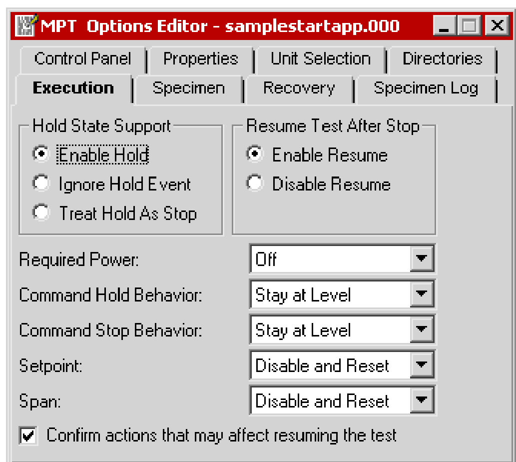
Access

From the MPT control panel or **MPT Procedure Editor** window, the access path is:

 > Execution tab

From the **MPT Procedure Editor** window, an alternate path is:

Tools > Options Editor > Execution tab



Options that are controlled from the **Execution** tab include:

- How the system responds when **Hold** is selected on the run, stop, hold panel.
- Whether or not a test can be resumed if it has been stopped, but it has not completed.
- Whether or not the Station Manager **Span** and **Setpoint** controls can affect your procedure.

- What hydraulic state is required for the control channels in the procedure to run.
- What happens to the command when the test is put in hold or stopped.
- Whether the application displays confirmation messages when actions that affect test resumption occur.

Execution Tab (part 1 of 3)

ITEM	DESCRIPTION
Hold State Support	Specifies how the **Hold** button on the **Run/Stop/Hold** panel operates. **Enable Hold**—Clicking **Hold** puts the test in a hold state. **Ignore Hold Event**—**Hold** is disabled and, if selected, has no effect. **Treat Hold As Stop**—**Hold** operates identically to **Stop**. For more information, refer to "Run/Stop/Hold Panel" on page 136.
Resume Test After Stop	Specifies whether or not you can resume a test after it is stopped. ***Note*** *This applies whether the test is stopped by an interlock, a program control process, or the **Stop** button on the **Run/Stop/Hold** panel.* If the enable option is not selected and a test is stopped, you will have to click the **Reset** button or select a new MPT specimen before you can run the procedure again.

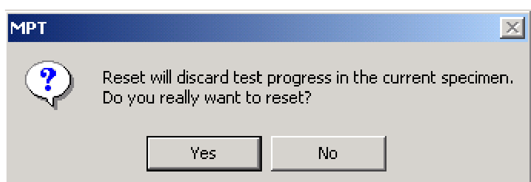
Windows and Controls

Execution Tab (part 2 of 3)

ITEM	DESCRIPTION
Required Power:	Specifies the hydraulic pressure setting required by the control channels in a procedure to start and continue running. ***Note*** *Power status applies only to control channels. If a procedure does not contain control channels, the power status will have no effect on the procedure. For instance, if a procedure without control channels has a power status of* ***High*** *and hydraulic power is removed from the station, the procedure will still run.* The selections are **High**, **High/Low**, **Off**, and **Off/Interlocked**. The **High** selection applies to most testing situations; it requires you to operate MPT with high hydraulic pressure applied to the station's Hydraulic Service Manifolds (HSM). The **Off** and **Off/Interlocked** selections are typically used while you are learning the application software or running test simulations. **Off/Interlocked** disregards active interlocks for the sake of allowing you to run the test unhindered.
Command Hold Behavior:	Specifies how the command process will behave when the procedure transitions to the hold state. For additional information about this behavior, See "About Command Hold and Stop Behaviors" on page 170.
Command Stop Behavior:	Specifies how the command process will behave when the procedure transitions to the stop state. For additional information about this behavior, See "About Command Hold and Stop Behaviors" on page 170.
Setpoint:	Enables or disables the **Setpoint** control on the **Setpoint and Span** window. **Enable**—The procedure is affected by the **Setpoint** slider on the **Setpoint and Span** window. **Disable and Reset—**The procedure is not affected by the **Setpoint** slider on the Station Manager's **Setpoint and Span** window. The procedure plays out at 100% span and at the procedure-specified setpoint. This control does not apply to **External Command** processes. For more information, see "Overriding system setpoint and span" on page 231.

Windows and Controls

Execution Tab (part 3 of 3)

ITEM	DESCRIPTION
Span:	Enables or disables the Station Manager's **Span** slider on the **Setpoint and Span** window and the **Master Span** slider on the **Station Controls** panel. **Enable**—The procedure is affected by the **Span** and **Master Span** sliders. **Disable and Reset—**The procedure is not affected by the **Span** and **Master Span** sliders The procedure plays out at 100% span. These controls do not apply to **External Command** processes. For more information, see "Overriding system setpoint and span" on page 231.
Confirm actions that may affect resuming the test	Enables or disables this confirmation function. When enabled (as indicated by a check mark in the box), this function causes query windows to pop up. The intent of these windows is to point out the potential loss of data and stimulate additional thought about the possible effects of your actions. A query window example is shown below: 

About Command Hold and Stop Behaviors

The **Command Hold Behavior** and **Command Stop Behavior** functions allow you to specify how an active MPT command process will behave when a procedure transitions to the hold or stop states.

The following selections are the same for each control:

- **Stay at Level**—The waveform will hold/stop at the current level.
- **Ramp to Mean**—The waveform will ramp to mean.
- **Ramp to Zero**—The waveform will ramp to zero.
- **Taper to Mean**—The waveform will taper to mean.
- **Taper to Zero**—The waveform will taper to zero.

The transition times for all ramp and taper functions are specified using the **Command Options** tab on the Station Manager's **Channel Options** window. The default transition time is two seconds. For more information about specifying command options, see the ***Model 793.00 System Software*** manual.

Special consideration

If the procedure ends normally, and the **Command Stop Behavior** is set to **Ramp to Mean** or **Taper to Mean**, the signal may not respond as you expect. When a procedure ends normally (without being manually held or stopped), all of its processes end normally, and then the **Command Stop Behavior** is executed. In these situations, **Ramp to Mean** and **Taper to Mean** settings behave like a "stay at level" setting, and the **Taper to Zero** setting behaves like a "ramp to zero" setting. The **Ramp to Zero** setting works as expected.

About Data Acquisition Resulting from Command Stop Behavior

When a test is stopped by an operator, MPT continues to acquire data through all subsequent transition periods.

Example

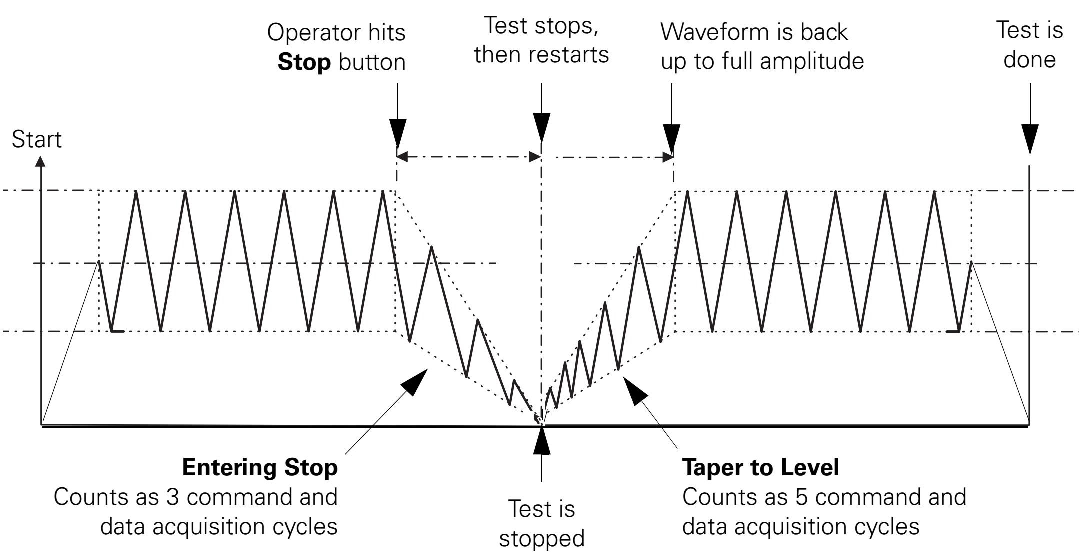
Suppose you are running a procedure with a cyclic waveform, and wish to acquire data on all the command cycles. The procedure requires 100 command cycles and 100 data acquisition cycles, where 1 cycle=1 second.

You push the **Stop** button during the test, and it takes 3 seconds to taper to zero. Those 3 cycles count as command and data acquisition cycles.

Then you push the **Start** button. It takes 5 seconds for the system to taper back up to full amplitude. Those 5 cycles also count as command and data acquisition cycles.

When the test reaches its "natural" done state (after 100 cycles,) it will have accomplished 100 command cycles and 100 data acquisition cycles, 8 of which are tapered.

Note *It is important to realize that, because of the 8 tapered cycles, some data points (those that are smaller than the specified sensitivity) might be missed if you are taking peak/valley data.*



About Command Stop Behaviors and Automatic Setpoint Adjustments

Note *When you enable **Setpoint**, the **Return to Zero** and **Taper to Zero** functions operate relative to the **Setpoint**, not to absolute zero.*

If you enable **Setpoint** and **Span** and set **Command Stop Behavior** to **Stay at Level**, you may find that **Setpoint** increases unexpectedly if you stop, unlock, reset, and restart the test. The reason for this is that, using the previously described conditions, **Setpoint** automatically shifts by the amount of offset present when the procedure was stopped.

Example

Suppose you run a procedure in which you execute a ±10 mm cyclic command from a mean level of 20 mm. You enable **Span** and **Setpoint** and select a **Command Stop Behavior** of **Stay at Level**. You set **Span** to 100% and **Setpoint** to 0 mm. When the actuator is at 20 mm, you stop the test. This causes the actuator to stop and hold at 20 mm. Then you unlock the procedure and resume the test. In this situation, the **Setpoint** value automatically increases to 20 mm, causing the actuator to travel to mean level of 40 mm before resuming the ±10 mm cyclic waveform.

Avoiding automatic Setpoint adjustments

To avoid this behavior and run the test relative to the original **Setpoint**, select a **Command Stop Behavior** of **Ramp to Zero** or **Taper to Zero**. This way, when you unlock MPT, the commanded level will be equal to the current **Setpoint** value.

Example

Suppose you run a procedure similar to the one in the previous example, except you select a **Command Stop Behavior** of **Ramp to Zero**. When the actuator is at 20 mm, you stop the test. This causes the actuator to ramp to 0 mm at the system ramp rate. Next, you unlock the procedure and resume the test. In this situation, the **Setpoint** value remains 0 mm, and the actuator ramps at the same rate back to 20 mm before executing the ±10 mm cyclic waveform.

Making manual Setpoint adjustments

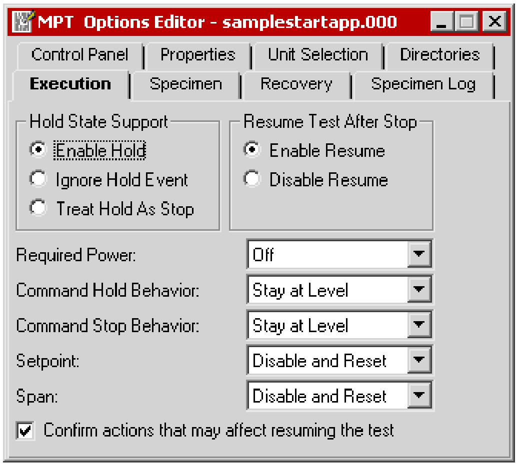
If the desired combination of **Setpoint** and **Command Stop Behavior** for your situation results in a **Setpoint** shift, you will have to manually adjust the **Setpoint** to the desired position before you restart a test.

About Transition States

Of the seven test states in MPT, there are four Entering and Exiting states: Entering Stopped, Exiting Stopped, Entering Hold, and Exiting Hold. These states could be described as “transition states”; that is, when MPT is leaving one state and entering another, such as changing from a Running state to a Holding state. Sometimes these transitions are instantaneous, but they still exist.

How long the test remains in a transition state depends on the test parameters you have chosen, using two separate windows: the **Execution** tab on the **MPT Options Editor** and the **Command Options** tab on the Station Manager’s **Channel Options** window.

MPT Execution tab

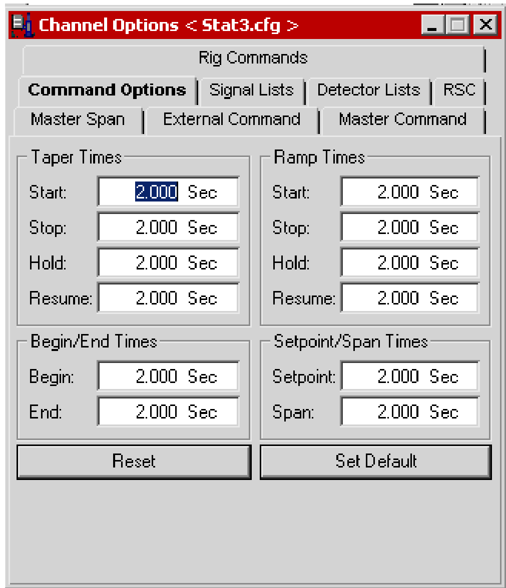


Use the MPT Execution tab to select the following Command Hold and Command Stop Behaviors:

- Stay at Level
- Ramp to Mean
- Ramp to Zero
- Taper to Mean
- Taper to Zero

(See “About Command Hold and Stop Behaviors” on page 170 for more information.)

Station Manager Command Options tab



Use the Station Manager Command Options tab to set the times for these behaviors:

- Taper Times
- Ramp Times
- Begin/End Times
- Setpoint/Span Times

(see the *System Software* manual for more information.)

Windows and Controls

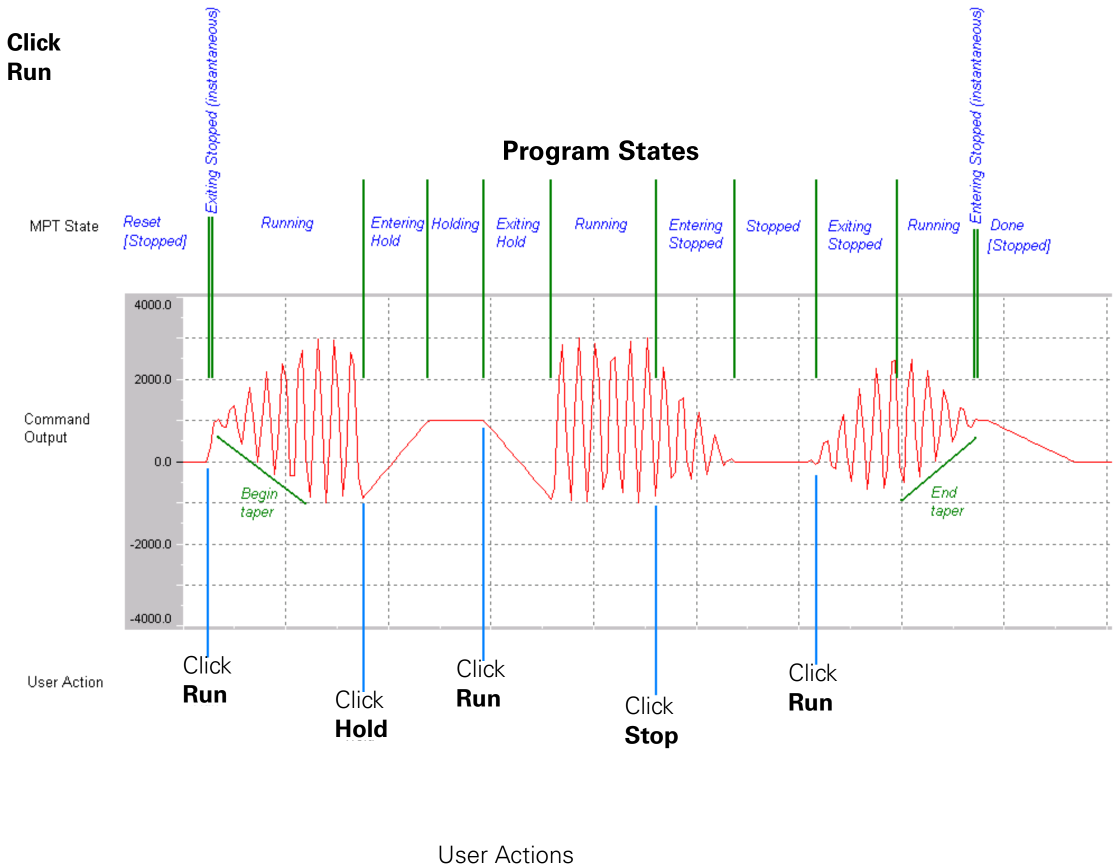
Example

The Entering Hold state lasts as long as it takes the system to move from the Running state to the Holding state. If you chose **Stay at Level**, the Entering Hold state would be instantaneous. If you chose **Ramp to Mean** or **Taper to Mean,** then this state would exist for a longer time because time is needed to transition from the current level to the level state.

The amount of time the test remains in a state also results from the time you have specified on the Station Manager **Command Options** tab:

- If you select a **Ramp** option for **Command Hold Behavior** or **Command Stop Behavior** in MPT, the **Ramp Times** you specify in Station Manager **Command Options** will be used. When you click **Hold** or **Stop** while a test is running, the test will be in a transition state during the ramp. If you click **Run** to resume the test, the test will again be in a transition state during the ramp.
- If you select a **Taper** option for the **Command Hold** or **Stop Behavior**, the **Taper Times** you specify in Station Manager **Command Options** are used. When you click **Hold** or **Stop** while a test is running, the test will be in a transition state during the taper. If you click **Run** to resume the test, the test will again be in a transition state during the taper.
- If you select a tapered waveform in a Cyclic Command, Profile, or Profile with ALC process in MPT, the **Begin/End Times** you enter in Station Manager are used. Those tapers will only appear at the very beginning and the very end of the waveform generated by these processes. These tapers do not occur in a transition state because they are considered part of the test (the tapered cycles are counted).

This diagram shows a test with a Sine Tapered waveform, Ramp to Mean on hold, and Taper to Zero on stop. It also shows each of the program states within the test:



Specimen Tab

Purpose

Use the **Specimen** tab to determine the storage mode and format of the data that is written to MPT specimens when you run procedures.

Access

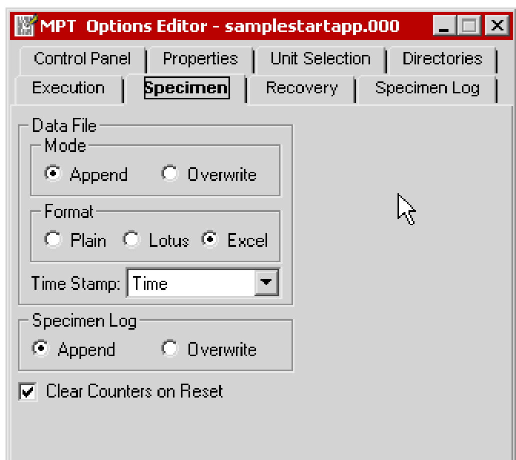
From the MPT control panel or **MPT Procedure Editor** window, the access path is:

 > Specimen tab

or

From the **MPT Procedure Editor** window, an alternate path is:

Tools > Options Editor > Specimen tab



Specimen Tab (part 1 of 2)

ITEM	DESCRIPTION
Data File	Data File functions control how data is added to the data file and data format. In addition, it controls the type of time stamp that is used in association with the data. **Mode Append**—Causes new data to be added to the end of data files. **Mode Overwrite**—Clears the MPS specimen data file (and user-specified files) when you start a new test run (that is, when you click **Run** after the test has been reset), which causes new data to write over existing data in data files. **Format Plain**—Writes space-delimited data. This is the most useful format for direct printouts. **Format Lotus**—Writes comma-delimited data with text strings in double quotes (sometimes referred to as the CSV format). Use this format when you plan to import data files into Lotus 1-2-3™. **Format Excel**—Writes tab-delimited data. Use this format when you plan to import data files into Microsoft® Excel. **Time Stamp**—Selects which signal is used to timestamp data records in the data files. **Time** is the elapsed time from run to reset. **Running Time** is the time from run to reset minus the time attributed to the **Hold** or **Stop** functions.

Specimen Tab (part 2 of 2)

Item	Description
Specimen Log	**Append**—Causes new messages to be added to the end of the **MPT Specimen Log** file. **Overwrite**—Clears the **MPT Specimen Log** when you start a new test (that is, when you click **Run** after the test has been reset), which causes new messages to write over existing messages in the **MPT Specimen Log** file.
Clear Counters on Reset	Enables and disables the clear function for the Run Time, Channel Counters, and Profile Counters (it does not affect the Sequence Counters). When enabled, the affected counters are cleared (reset to zero) whenever you click the **Reset** button. When disabled, the affected counters are not cleared (reset to zero) when you click the **Reset** button. Not resetting the counters can affect various test related functions. For example, it can affect counter related limits (not resetting may cause you to exceed a segment or cycle related limit). Counter resets are sometimes used to indicate the start of a new test in your data file. Without the resets, multiple iterations of a test may be viewed as one long test.

Recovery Tab

Purpose

Use the **Recovery** tab to determine how often the application saves a "snapshot" of the test state and status.

Access

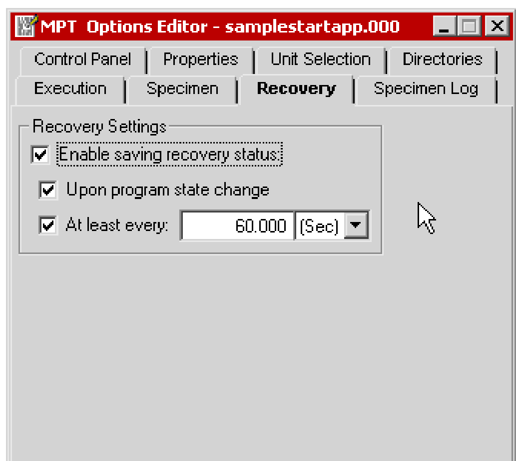
From the MPT control panel or **MPT Procedure Editor** windows, the access path is:

 > Recovery tab

or

From the **MPT Procedure Editor** window, an alternate path is:

Tools > Options Editor > Recovery tab



The snapshots of the test state and status can be used to restart a test that was interrupted by an event (for example, a power failure or interlock).

Recovery Tab

ITEM	DESCRIPTION
Recovery Settings	**Enable saving recovery status**—Enables the recovery feature and specifies how the progression of the test state and status is saved. ***Note*** *MTS recommends you leave this feature enabled unless you have strong reasons to disable it.* **Upon program state change**—Saves the test status whenever the run, stop, or hold buttons are clicked on the **Station Controls** panel (including interlocks). **At least every**—Saves the test status at a regular time interval (user specified).

More About Test Recovery

When you enable the recovery feature, the application updates a recovery file (saved in the MPT specimen) at the intervals you specify.

Snapshot intervals can be set using the program state changes (includes **Run** to **Stop** or **Stop** to **Run** state changes, including interlocks) and time intervals that you specify.

You can use a recovery file to restart a test even if the test was running during the last snapshot. You cannot restart tests if they have completed or have been reset.

What is saved

Information saved to the recovery file includes:

- A copy of the procedure. (It is possible to edit a procedure, then execute it without saving it. In this situation, the test status contains the information used in the execution, not the original procedure.)
- The active group and individual processes. It also identifies how much of each process is complete.
- The current program state (**Run**, **Hold**, **Stop**). When recovering a test, if the program was running or holding, the test is restored in the **Stop** state.
- The state of the sequence counters and the channel counters.

What is not saved

When you recover a test, the test status is restored to the point where the last snapshot was taken.

- If the test is stopped by an uncontrolled event (such as power failure or application failure), any data not saved to disk at the point of the shutdown is lost.
- If data was being transferred from the data buffer to the computer, data that had not been transferred is lost.

DIO settings are not restored during the MPT recovery process. For information about DIO settings, refer to the Station Manager chapter in your Model 793.00 System Software manual.

Recovery accuracy

There are two measures of “accuracy” associated with a saved test status:

- **How old is the snapshot?** If the snapshot is taken one minute before the test stops, many additional cycles may be applied to the MPT specimen that the test status would not know about.
- **How consistent is the snapshot?** If the snapshot is taken while the test is running, each active process is checked in sequence. This results in test status data that is taken at slightly different times.

Recovery considerations

Review the following characteristics of the test recovery feature to understand the advantages and disadvantages of recovering a test under different circumstances.

- Recovering a test that was saved periodically provides test status data that is relatively accurate in relation to the point where the test stopped. The shorter the period, the closer the test status is to the point of failure. However, saving a test does require additional processor time. If the autosave rate is too fast, the accuracy of the data acquisition processes (data output and communication between processes) can be adversely affected.
- Each autosave action overwrites the test status. If the system crashes (power fails) while the system is saving the test status, you probably will not be able to recover the test status.
- The test status maintains a copy of the procedure when entering the **Execute** mode in MPT.

Specimen Log Tab

Purpose

Use the **Specimen Log** tab on the **MPT Options Editor** to filter the messages written to the **MPT Specimen Log**. You can also specify what items are displayed in the **Message Logs** window.

Access

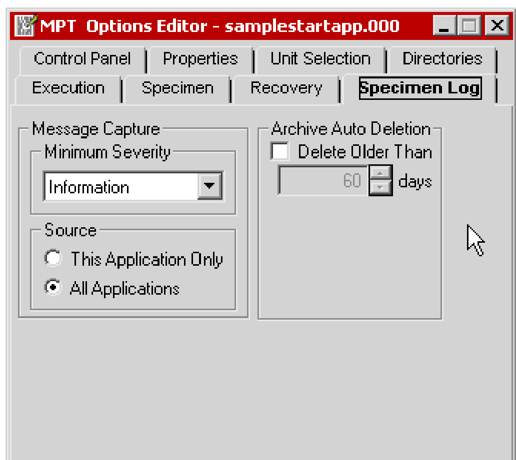
From the MPT control panel or **MPT Procedure Editor** window, the access path is:

 > Specimen Log tab

or

From the **MPT Procedure Editor** window, an alternate path is:

Tools > Options Editor > Specimen Log tab



If your test generates an excessive number of messages, you should filter what messages are written to the log to avoid exhausting disk space.

For instance, you can change the **Source** filter from **All Applications** to **This Application Only** to keep Station Manager messages out of the **MPT Specimen Log**.

Specimen Log Tab (part 1 of 2)

ITEM	DESCRIPTION
Message Capture	Controls the severity level and source of the information that is written to the **MPT Specimen Log**. **Minimum Severity** sets the severity level that must be reached before a MultiPurpose TestWare writes a message to the Specimen Log file. There are five levels: • **Diagnostic**—Writes **Diagnostic**, **Information**, **Warning**, **Error**, and **Fatal Error** messages. • **Information**—Writes **Information**, **Warning**, **Error**, and **Fatal Error** messages; does not write **Diagnostic** messages. • **Warning**—Writes **Warning**, **Error**, and **Fatal Error** messages; does not write **Diagnostic** and **Information** messages. • **Error**—Writes **Error** and **Fatal Error** messages; does not write **Diagnostic**, **Information**, and **Warning** messages. • **Fatal Error**—Writes only **Fatal Error** messages; does not write Diagnostic, Information, Warning, and Error messages. **Source**—Selects the information source from which the log is built. • **This Application Only**—Records messages from the MPT application only. • **All Applications**—Records messages from other sources, such as the Station Manager application.

Specimen Log Tab (part 2 of 2)

ITEM	DESCRIPTION
Archive Auto Deletion	***Important*** *Enabling this control may result in the inadvertent deletion of specimen log entries. The Specimen Log is unlikely to exceed the internal limit of 500 Specimen Log files for a given MPT specimen (unless your procedure is generating an excessive number of messages). Therefore, MTS recommends that you disable this control if you routinely use new MPT specimens from test-to-test. Disabling this control will ensure that you will be able to recover relevant log data for your MPT specimens in the future.* This control enables or disables the auto deletion function and sets the deletion interval in days. The **Specimen Log** archive has an internal limit of 500 **Specimen Log** files, with each file containing approximately 1000 messages. Once 500 archived files are accumulated and a new archive file is generated, the oldest archive file is automatically deleted to make room for the newest archive file. The **Archive Auto Deletion** control lets you program when archived files are deleted in terms of days (within the internal limit of 500 archived files). To enable the **Archive Auto Deletion** function, click in the **Delete Older Than** checkbox such that a check mark appears in the box. If the check mark appears in the box, the function is enabled. Set the deletion interval using the scroll arrows to change the number of days displayed. The scroll arrows are only active if the **Archive Auto Deletion** function is enabled.

Control Panel Tab

Purpose

Use the **Control Panel** tab to specify what information is displayed on the MPT control panel.

Access

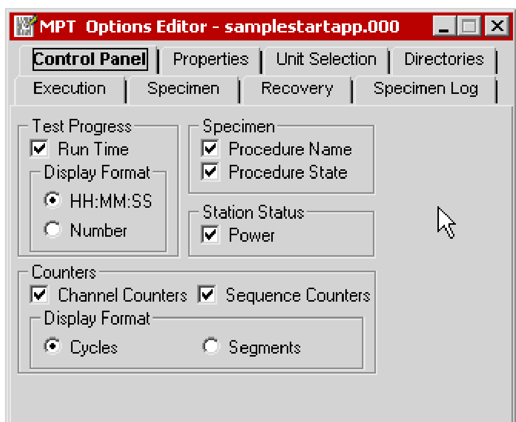
From the MPT control panel or **MPT Procedure Editor** window, the access path is:

 > Control Panel tab

or

From the **MPT Procedure Editor** window, an alternate path is:

Tools > Options Editor > Control Panel tab



Control Panel Tab

Item	Description
Test Progress	Enables and disables the **Run Time** portion of the **Test Progress** panel on the MPT control panel. In addition, it allows you to change the **Run Time** panel display format. **Run Time**—Enables/disables the **Run Time** display. If **Run Time** is disabled, the **Run Time** display does not appear on the **Test Progress** panel (see “Test Progress Panel” on page 129). **Display Format**—Changes the **Run Time** display format. The format options are HH:MM:SS (hours:minutes:seconds) and cumulative time units.
Specimen	Enables and disables the **Procedure Name** and **Procedure State** portion of the **Specimen** panel. If the **Procedure Name** or **Procedure State** is disabled, the respective display does not appear on the **Test Progress** panel. **Procedure Name**—Displays the name of the current procedure. **Procedure State**—Displays the current procedure state.
Station Status	Enables and disables the **Power** panel on the MPT control panel. If **Power** is disabled, the **Power** panel does not appear on the MPT control panel.
Counters	Enables and disables the **Channel Counters** and **Sequence Counters**. In addition, it controls the counter display format. **Channel Counters**—Displays the **Channel Counters** when enabled (see “Channel Counters Panel” on page 133). **Sequence Counters**—Displays the **Sequence Counters** when enabled (see “Sequence Counters Panel” on page 134). **Display Format**—Controls the **Channel Counters** and **Sequence Counters** display format. The format options are: • **Cycles -** Displays the total number of cycles that have been executed. • **Segments** - Displays the total number of segments that have been executed.

Properties Tab

Purpose

Use the **Properties** tab to enter information that is saved with the procedure.

Access

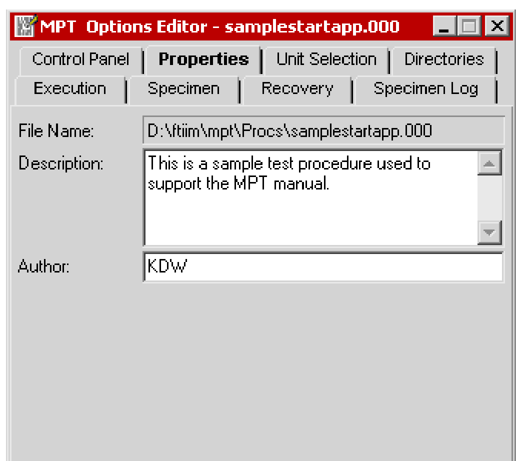
From the MPT control panel or **MPT Procedure Editor** window, the access path is:

 > Properties tab

or

From the **MPT Procedure Editor** window, an alternate path is:

Tools > Options Editor > Properties tab



You can enter a process summary, identify the procedure, describe the test, or write instructions to the test operator. This information will appear in the **Print Preview** window.

Properties Tab

ITEM	DESCRIPTION
File Name	Displays the name and path of the current procedure selection.
Description	Enter a description or other information about the procedure here.
Author	Enter the author's name here.

Unit Selection Tab

Purpose

Use the **Unit Selection** tab on the **MPT Options Editor** to assign a different unit assignment set to the current procedure.

Access

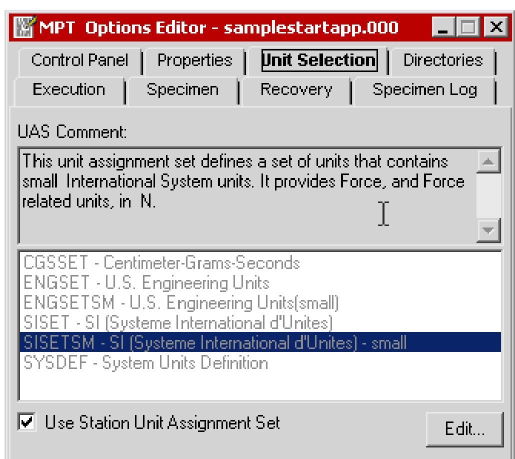
From the MPT control panel or **MPT Procedure Editor** window, the access path is:

 > Unit Selection tab

or

From the **MPT Procedure Editor** window, an alternate path is:

Tools > Options Editor > Unit Selection tab



Unit Selection Tab

ITEM	DESCRIPTION
UAS Comment	Displays a description of the selected unit assignment set.
Selection list	Displays a list of the defined unit assignment sets. The currently selected set is shown.
Use Station Unit Assignment Set	***Note*** *By default, this control is enabled in the default template. To disable this control for new procedures, you must disable it in the default template (see "Default Templates" on page 449).* **When enabled for existing procedures:** Causes the procedure to acquire the current default UAS selection, which is set in the registry during the installation of Station Manager. If this box was disabled when you opened a procedure and you enable it while editing the procedure, you will see the units in your procedure change from the UAS assigned when you opened the procedure to the UAS set in the registry. The new UAS assignment will not be permanent until you save the procedure. If you close without saving, the UAS assignment will revert to the set assigned when you opened the procedure. **When enabled for new procedures:** Causes new procedures to open with the UAS selection set in the registry. **When disabled for existing procedures:** Allows you to select and edit any UAS and assign it to the current procedure. When you save your procedure, your specific UAS selection will be saved with your procedure. If keeping a specific UAS assignment linked to your procedure is important to you, this is a good choice. **When disabled for new procedures:** Causes new procedures to open with the UAS selection set in the registry. If you want all new procedures to open with a specific UAS that is independent of the station configuration UAS, change the UAS selection in the default template.
Edit...	Opens the Unit Assignment Set Editor. Using this editor, you can create new or edit existing unit assignment sets.

Directories Tab

Purpose Use the **Directories** tab to determine the default directory path of procedures, profiles, default templates, and MPT specimens. Selections you make in this tab are not saved as part of any specific procedure.

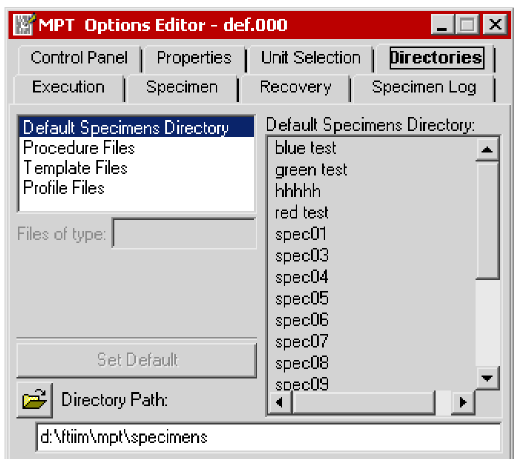
Access From the MPT control panel or **MPT Procedure Editor** window, the access path is:

 > Directories tab

or

From the **MPT Procedure Editor** window, an alternate path is:

Tools > Options Editor > Directories tab



Directories Tab (part 1 of 2)

ITEM	DESCRIPTION
directories list	Lists available MPT directories that you may browse. When you select a directory from the list, the selected directory name appears at the top of the display panel to the right and the selected directory files are displayed according to the extension shown.
Files of type	Displays the file type for the current file selection.

Directories Tab (part 2 of 2)

ITEM	DESCRIPTION
Set Default	Updates the directory path in the registry file to match the current directory path. ***Note*** *If you click **Set Default**, the registry is changed immediately, regardless of whether or not you save your procedure, and the change will become the default value that will be used by subsequent MPT sessions. If you do not click **Set Default**, modified directory paths will apply only until you exit MPT.*
Directory Path	Displays the directory path for the selected file type. If you click on the **Directory Path** button, the **Browse for Folder** window opens and allows you to navigate to a new folder.

About MPT

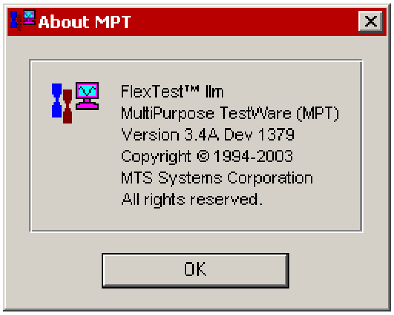
Purpose

The **About MPT** window displays the MultiPurpose TestWare application version number and copyright information.

Access

From the **MPT Procedure Editor** window, the access path is:

Help > About



Chapter 6

Process Descriptions

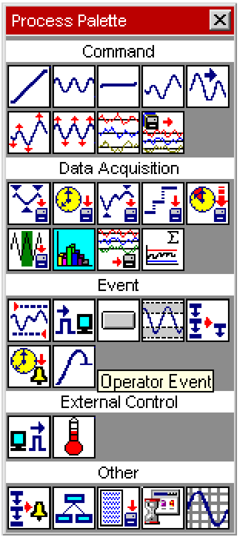
About MPT Processes

When you create tests with the MPT application, you use MPT *processes* like building blocks to define test actions such as sending command signals, performing data acquisition, and responding to test events.

A complete MPT test normally consists of a number of MPT processes linked together and saved as an MPT *procedure*.

Process Palette

Your MPT process icons are displayed on the **Process Palette**.



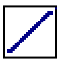
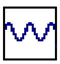
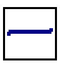
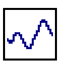
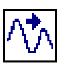
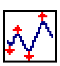
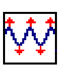
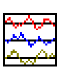
Process categories

Processes are grouped on the palette in the following categories:

- Command Processes send commands to a servovalve or external controller to apply forces to the physical specimen.
- Data Acquisition Processes acquire sensor data from the physical specimen.
- Event Processes monitor test signals and perform actions when specified conditions are met.
- External Control Processes control external controllers or logic devices.
- Other Processes perform miscellaneous functions.

Command Processes

Note *If a command process includes both normal (high-rate) and low rate channels, the channels will start and stop together. However, they may not stop at the same level. They have be off in phase as much as one low-rate tick.*

Icon	Process Name	Description
	**Segment Command**	Produces a monotonic command signal. (See page 200.)
	**Cyclic Command**	Produces a cyclic command signal. (See page 205.)
	**Dwell Command**	Produces a dwell command signal. (See page 216.)
	**Profile Command**	Produces a command signal based on the contents of a “profile” created with a text editor or the **Profile Editor** application. (See page 219.)
	**External Command**	Enables a command signal from an external program source. (See page 228.)
	**Profile with ALC**	Produces an ALC compensated command signal based on the contents of a “profile” created with a text editor or the **Profile Editor** application. (See page 236.) **Note** *This process is not included with all systems.*
	**Cyclic with ALC**	Produces an ALC compensated cyclic command signal. (See page 242.) **Note** *This process is not included with all systems.*
	**Time History Output**	Produces a command signal based on an MTS RPC II or RPC III time history file. This process can also perform time history data acquisition synchronized with the command. (See page 245.) **Note** *This process is not included with all systems.*
	**Road Surface Output**	Produces a command signal that controls multiple station channels with an MTS RPC II, RPC III, or RPC Pro *Road Surface file*. (See page 257.) **Note** *This process is not included with all systems.*

About Command Compensation

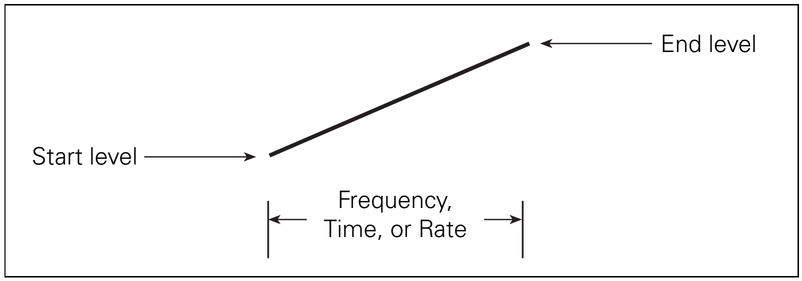
The **Adaptive Compensators** list includes all compensators available on at least one channel selected in the **Channels** tab. It is important to know which compensator is available for each channel in your station configuration because when this process is executed, compensation will occur only on the channels for which it was enabled.

Note *A special annotation appears if the compensator is not available on all channels. For example, "**AIC (1 of 3)**" implies that the AIC compensator is only supported on one of the three included channels.*

Segment Command

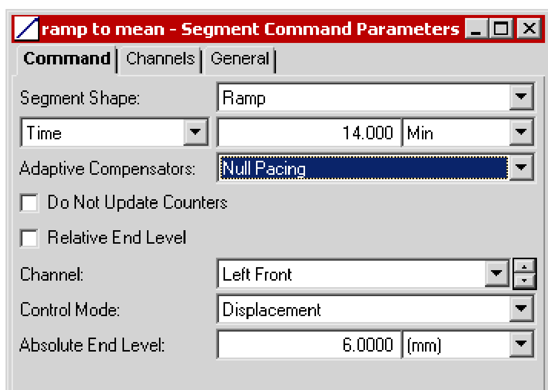
The **Segment Command** process defines a monotonic function that begins at the current level and moves to the end level you specify.

Note *This process supports relative end levels and absolute end levels.*



Segment Command Parameter Descriptions

Command tab

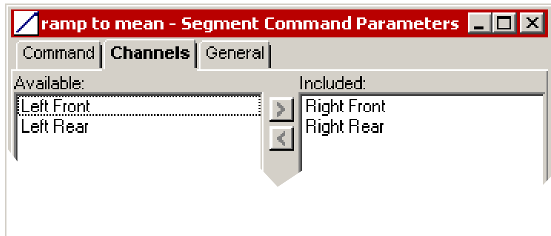


Command Tab

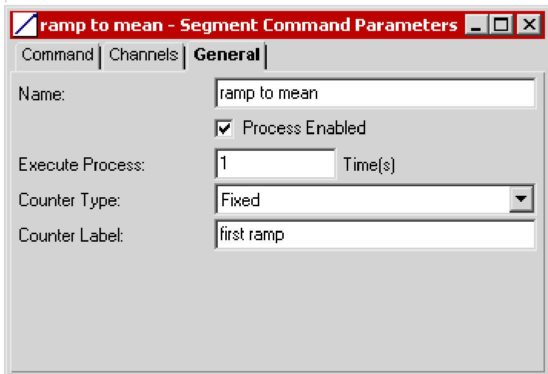
Item	Description
Segment Shape	Specifies the segment shape. See "About Segment Shapes" on page 209.
Time/Frequency/Rate	Specifies how fast the segment executes. The following methods are supported: • **Time** specifies the time to execute one segment. • **Frequency** specifies the time to execute a two-segment cycle (even though a single segment executes). • **Rate** specifies a constant rate between the starting level and the end level. Rate is typically used with a ramp segment. ***Note*** *You cannot use **Rate** for a command process applied to multiple channels. If you do, an error message will appear when the procedure is locked.*
Adaptive Compensators	Enables command compensation during the command process. See "About Command Compensation" on page 199.
Do Not Update Counters	Disables the counters. When enabled, the **Channel Counters** do not increment for this process and the counter totals are unaffected.
Relative End Level	Makes the end level relative.
Channel	Specifies the channel.
Control Mode	Specifies the control mode.
Absolute End Level/ Relative End Level	Defines the segment end level: • Absolute End Level is relative to setpoint. • Relative End Level is relative to the starting command value (the value when the process begins).

Channels tab

Displays the channels available to this process. Use the arrow keys to move desired channels from the **Available** list to the **Included** list.



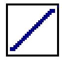
General tab



General Tab

ITEM	DESCRIPTION
Name	Specifies the process name displayed on the **Procedure** or **Group** process window.
Process Enabled	Enables the process. (Clear this box to disable the process.)
Execute Process	Specifies the number of times the process will be executed.
Counter Type	Enables a process counter. Options are **None**, **Transient**, or **Fixed**. Transient counters are displayed only when the process is active. Fixed counters are displayed throughout the test.
Counter Label	Names the counter. When you run the test, this name appears under **Sequence Counters** on the MPT control panel. ***Note*** *If this box is blank, the process name is used as the counter label.*

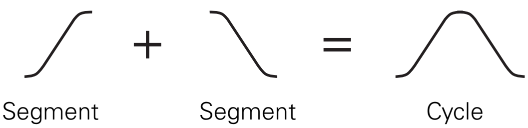
How to Define a Segment Command Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Channels** tab.
4. Move the channels that you want to use to generate the command onto the **Included** list. Move all other channels to the **Available** list.
5. Click the **Command** tab.
6. In the **Segment Shape** list, click the segment shape.
7. Set the **Time**, **Rate**, or **Frequency** dimension; set the units of measurement; and then type a value.
8. Optional—In the **Adaptive Compensators** list, click a compensator.
9. Optional—If you do not want channel counters to increment when this process runs, click **Do Not Update Counters**.
10. Optional—If you want to define a relative end level, select the **Relative End Levels** check box.
11. In the **Channel** list, click your desired channel.
12. In the **Control Mode** list, click your desired control mode.
13. In the **End Level** box, set the units of measurement, and then type an end level value.
14. Repeat steps 11 - 13 for each channel you want to command with this segment process.

15. Click the **General** tab.
16. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
17. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
18. In the **Execute Process** box, enter the number of times you want the process to execute.
19. Optional—In the **Counter Type** list, click a counter type.
20. Optional—If you enabled a counter, type a **Counter Label**.
21. Close the parameters window, and set the desired **Start** and **Interrupt** triggers for the process in the Group or Procedure table.

Cyclic Command

The **Cyclic Command** process defines a cyclic command by assembling two single segments and repeating them continuously for a specified number of cycles.



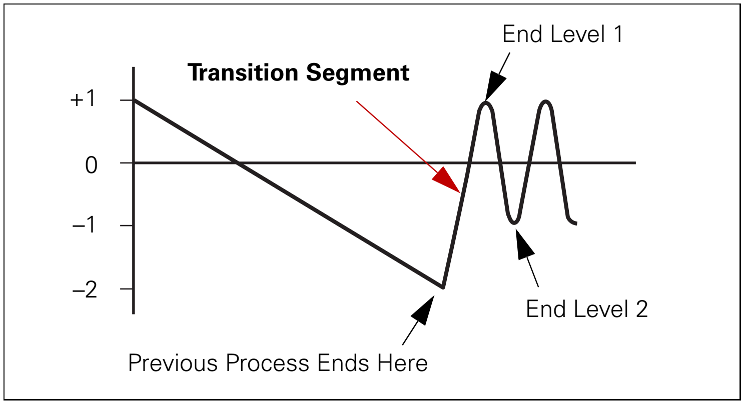
One cycle command works like two segment commands.

More About the Cyclic Command Process

When a cyclic command starts, it always moves first to **End Level 1**, and then to **End Level 2**.

Sine, ramp, and square start levels

Sine, ramp, and square commands use the first segment of the cyclic command to bridge any difference between the current level and the cyclic command **End Level 1** that you specify. This *transition segment* is counted as the first segment.



In this sine command example, the first segment of the process is a transition segment that eases the transition between the end of the previous process and the first end level of the cyclic process.

CAUTION

A high-speed transition segment can result in an unwanted spike (or bump) in your feedback signal.

Unwanted feedback signal spikes can damage the physical test specimen.

To avoid spikes in the feedback signal, use a segment command process to bridge processes where end levels and start levels differ greatly. If a process end level is not near the start level of your cyclic command process, the process uses the first segment to move all the way from the current level to the start level at the transition speed that you specify.

True sine start levels

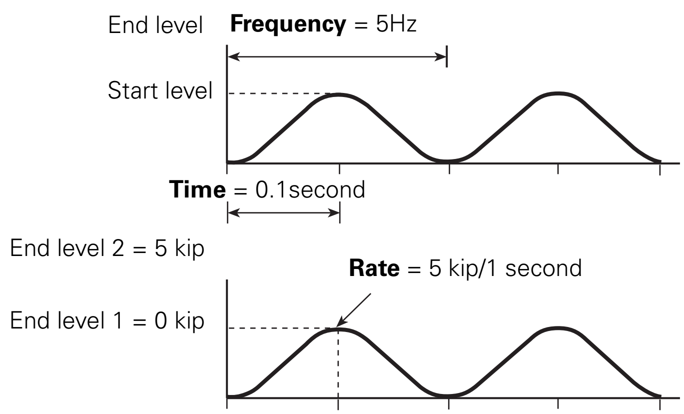
True sine commands start at the mean level, which is the midpoint between the **End Level 1** and **End Level 2** you specify. The system begins a true sine cyclic process by stepping from the current level to the mean level. (This step does not count as a segment.)

Once at the mean, the command travels through **End Level 1** at the specified rate. When using a true sine command, you should design your procedure so that the system is at or very near its mean level before the true sine process begins. You can include a segment command process that ramps from the current level to the mean level. If you do not, your system may experience a step (or jolt) between processes. The more difference there is between the current level and the true sine command's mean level, the greater the step.

Cycles and segments

You can repeat cycles or segments. Two segments create one cycle. Repeating a full cycle first goes to end level 1 and ends at end level 2. Repeating a segment can end at end level 1 or end level 2 (depending on whether the count value is odd or even).

The conventional way to define the duration of a cyclic command is by frequency or time.



Note *The end levels selected in processes such as Segment Command or Cyclic Command can be set to values greater than the full scale preset in Station Manager. The command will attempt to reach the specified end level; however, the feedback will become saturated before reaching the specified end level and will cause a hydraulic interlock.*

End levels

The two end levels specify the amplitude of the cyclic waveform. When two test command processes with different end levels are sequenced, the transition between them is accomplished at the rate and segment shape of the following process. This provides a smooth transition between test commands.

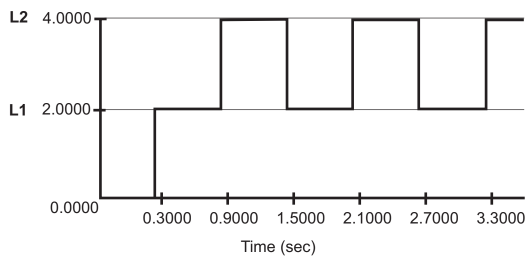
Example

Assume a cyclic command completes its process at **End Level 2** (in this case, +2 cm). The next Cyclic Command process is defined as **End Level 1** = –1 cm and **End Level 2** = +1 cm. The transition segment bridges the two commands from the +2 cm level to the –1 cm level at the time, rate, or frequency of the second process.

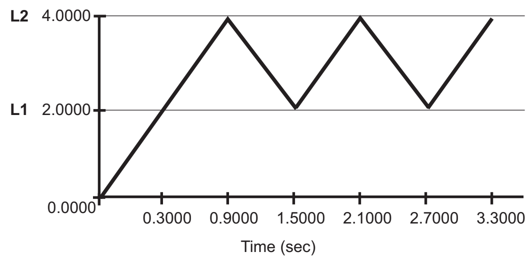
About Segment Shapes

Each of the following examples uses the following parameters: Frequency (Hz) 1, Count (segments) 6, Level 1 (mm) 2, Level 2 (mm) 4, with a starting level of 0.0.

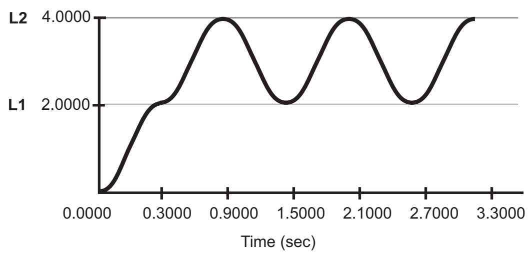
Square



Ramp



Sine



Process Descriptions

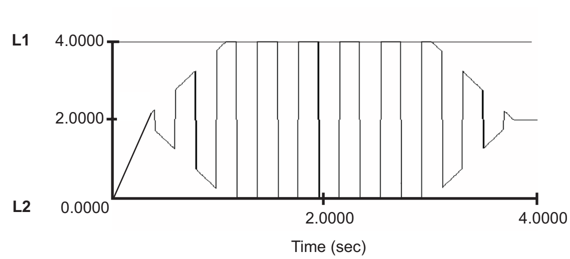
Tapered wave shapes

You specify the beginning and ending taper times on the **Command Options** tab in the Station Manager's **Channel Options** window before you execute the profile.

The **Begin** time formats the leading edge and the **End** time formats the trailing edge.

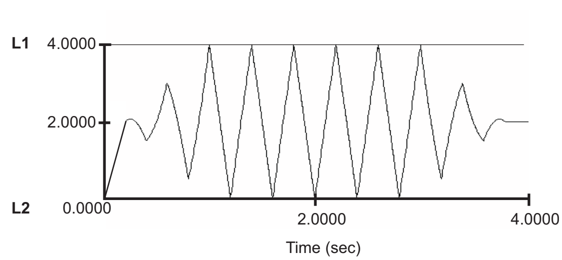
Square Tapered

This square wave tapers from 0% to 100% amplitude at the beginning of its execution, and from 100% to 0% at the ending of its execution.



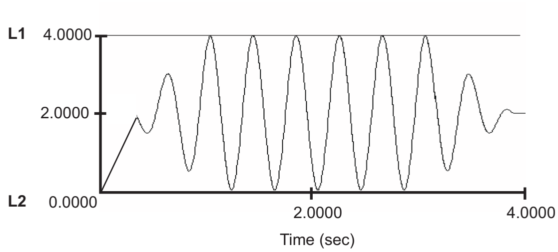
Ramp Tapered

This ramp wave tapers from 0% to 100% amplitude at the beginning of its execution, and from 100% to 0% at the ending of its execution.



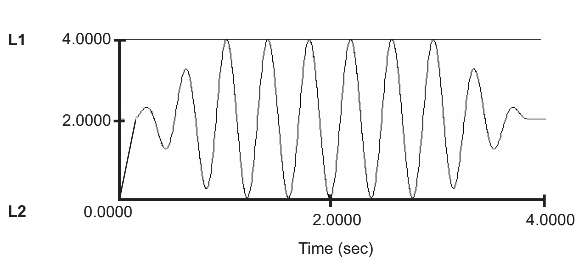
Sine Tapered

This sine wave tapers from 0% to 100% amplitude at the beginning of its execution, and from 100% to 0% at the ending of its execution.



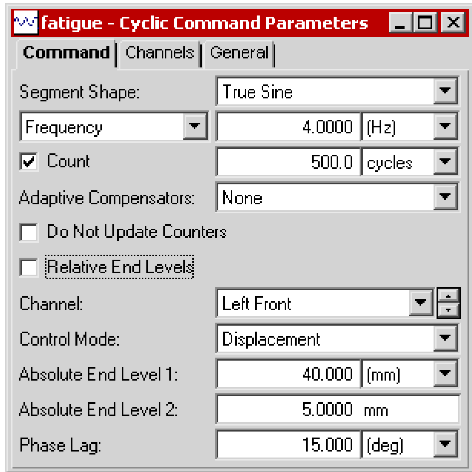
True Sine Tapered

This true sine wave tapers from 0% to 100% amplitude at the beginning of its execution, and from 100% to 0% at the ending of its execution.



Cyclic Command Parameter Descriptions

Command tab



Command Tab (part 1 of 2)

ITEM	DESCRIPTION
Segment Shape	Specifies the segment shape. See “About Segment Shapes” on page 209.
Time, Frequency, Rate	Specifies how fast each segment executes. The following methods are supported: • **Time** specifies the time to execute one segment. • **Frequency** specifies the time to execute a two-segment cycle. • **Rate** specifies a constant rate between the starting level and the end levels. Rate is typically used with a ramp segment. ***Note*** *You cannot use **Rate** for a command process applied to multiple channels. If you do, an error message will appear when the procedure is locked.* ***Note*** *The rate of the first segment can be faster than the other segments if it must transition from a distant end level.*
Count	Specifies how many times the waveform will be repeated. ***Note*** *If you clear the **Count** box, the waveform will repeat continuously.*
Adaptive Compensators	Enables command compensation during the command process. See “About Command Compensation” on page 199.

Command Tab (part 2 of 2)

ITEM	DESCRIPTION
Do Not Update Counters	Disables the counters. When enabled, the **Channel Counters** do not increment for this process and the counter totals are unaffected.
Relative End Levels	Defines the segment end level: • Absolute End Level is relative to setpoint. • Relative End Level is relative to the starting command value (the value when the process begins).
Channel	Specifies the channel.
Control Mode	Specifies the control mode.
Absolute End Level 1/ Relative End Level 1	Defines the end level value for the first segment.
Absolute End Level 2/ Relative End Level 2	Defines the second end level value for the second segment.
Phase Lag	Specifies the phase relationship of the waveform generated by this process from channel to channel. **Phase Lag** is disabled if **Rate** is specified for the process. ***Note*** *Phase Lag is relative to the channel that has the smallest degree of phase. The Phase Lag of each channel is referenced to the lowest Phase Lag defined in the channels.*

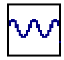
Channels tab

See "Channels tab" on page 202.

General tab

See "General tab" on page 202.

How to Define a Cyclic Command Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Channels** tab.
4. Move all the channels you want to generate the command on to the **Included** list. Move all the others to the **Available** list.
5. Click the **Command** tab.
6. In the **Segment Shape** list, click the segment shape.
7. Select **Time**, **Rate**, or **Frequency**, set the units of measurement, and then type a value.
8. Select the **Count** check box. Set the units (cycles or segments), and then type the number of times the cyclic command is to repeat. For continuous cycling, clear the **Count** check box.

Note *The **Execute Process** value multiplies the **Count** in the **Command** parameters. If the **Execute Process** parameter is set to 10 and the **Command Count** is set to 5, a total of 50 cycles will occur.*

9. Optional—In the **Adaptive Compensators** list, click a compensator.
10. Optional—If you do not want channel counters to increment when this process runs, click **Do Not Update Counters**.
11. Optional—If you want to define a relative end level, select the **Relative End Levels** check box.
12. In the **Channel** list, click your desired channel.
13. In the **Control Mode** list, click your desired control mode.
14. Set the units of measurement for the end levels, then enter values for **End Level 1** and **End Level 2**.
15. Optional—In the **Phase Lag** box, type a value.
16. Repeat steps 12 through 15 for each channel you want to command with this process.
17. Click the **General** tab.

18. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
19. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
20. In the **Execute Process** box, type number of times you want the process to execute.
21. Optional—In the **Counter Type** list, click a counter type.
22. Optional—If you enabled a counter, type a **Counter Label**.
23. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

Dwell Command

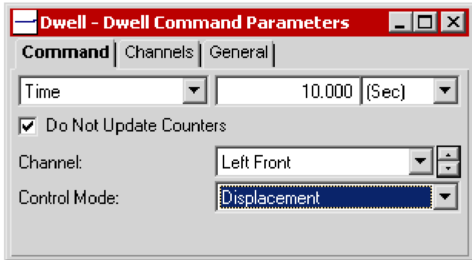
The **Dwell Command** process is a single segment designed to hold your command at a static level for a specified amount of time.

When you define a **Dwell Command** process you must specify the duration of the command as a time or a frequency:

- **Time** specifies the amount of time to execute the single dwell command segment.
- **Frequency** specifies the frequency of a command cycle (two-segments). Since a dwell command is a single segment, the actual dwell command hold time is half of the frequency setting. (For example, a frequency setting of 2 Hz holds the command for 0.25 seconds.)

Dwell Command Parameter Descriptions

Command tab



Command Tab

ITEM	DESCRIPTION
Time/Frequency	Specifies how fast the segment executes. Options are: • **Time**—specifies the time to execute one segment. • **Frequency**—specifies the time to execute a two-segment cycle.
Do Not Update Counters	Disables the counters. When enabled, the **Channel Counters** do not increment for this process and the counter totals are unaffected.
Channel	Specifies the channel.
Control Mode	Specifies the control mode.

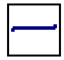
Channels tab

See “Channels tab” on page 202.

General tab

See “General tab” on page 202.

How to Define a Dwell Command Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Channels** tab.
4. Move the channels that you want to use to generate the command onto the **Included** list. Move all the others to the **Available** list.
5. Click the **Command** tab.
6. Select **Time** or **Frequency**, set the units of measurement, then type a value.
7. Optional—If you do not want channel counters to increment when this process runs, click **Do Not Update Counters**.
8. In the **Channel** list, click your desired channel.
9. In the **Control Mode** list, click your desired control mode.
10. Repeat steps 8 and 9 for each channel you wish to control.
11. Click the **General** tab.
12. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
13. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
14. In the **Execute Process** box, enter the number of times you want the process to execute.
15. Optional—In the **Counter Type** list, click a counter type.
16. Optional—If you enabled a counter, type a **Counter Label**.
17. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

Profile Command

The **Profile Command** process generates a command based on a profile file (or just profile for short) created with a text editor, spreadsheet application, or the **MTS Profile Editor** application. The MPT application supports *Block-Arbitrary* and *Phase* profile formats.

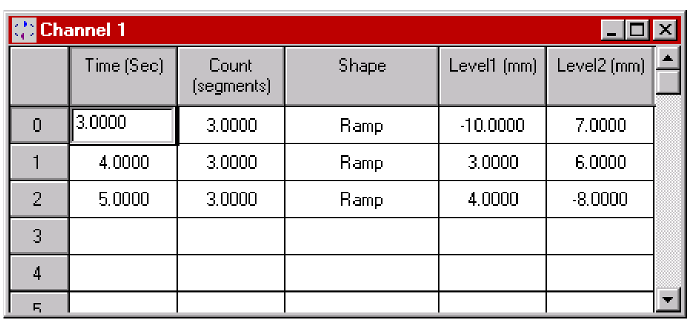
For additional information on profiles in general and syntax requirements for profiles, see “General Profile Syntax Requirements” on page 463.

More About Profiles

A profile is basically a spreadsheet that defines a particular command segment (or a series of segments) in each row. As the controller plays out the profile, it starts by playing the segment or cycle defined in the first row, then the second row, then the third row, and so forth.

Note *A single profile often contains the command content of an entire procedure.*

When you create a profile, you must define the segment shape, end levels values, time between end levels, and count for the command in each row. For information on creating profiles with the **Profile Editor** application, see the *Model 793.11 Profile Editor* manual.



	Time (Sec)	Count (segments)	Shape	Level1 (mm)	Level2 (mm)
0	3.0000	3.0000	Ramp	-10.0000	7.0000
1	4.0000	3.0000	Ramp	3.0000	6.0000
2	5.0000	3.0000	Ramp	4.0000	-8.0000
3					
4					
5					

Sample Profile

User-defined actions and counter names

The profile process supports counters and user-defined actions. Counters appear on the MPT control panel when the process is running and are incremented after the corresponding row in the profile has played out.

User-defined actions defined with the Station Manager application are triggered after the corresponding row in the profile has played out.

Counters and actions are specified on specific rows (with specific syntax) in profiles.

Special consideration when running phase type profiles

While running a procedure containing a phase profile, if you stop and unlock the procedure, the resulting cycle counts on the slave channels may not be exact when you resume the procedure. The profile process determines where to restart based on the master channel. If the other channels are on a different segment because of phase differences, the Channel Counters on the MPT control panel will be different by that amount.

Profiles with normalized dimensions

If you create a profile channel using a *normalized* dimension and map it to a station channel (on the **Mapping** tab), then you can select any of the control modes available to the station channel when defining the control mode of the profile channel.

Normalized dimensions include:

- Ratio
- Percentage
- Unitless
- Volts

When you use normalized dimensions you define the full-scale value of the normalized dimension by selecting one of the control modes available in the **Control Mode** list on the **Mapping** tab, along with a unit type and a **Level Multiplier** value. The MPT application converts all end-level values in the profile to command levels accordingly.

Example

Suppose you create a station configuration that uses the force dimension for a force control mode, and the length dimension for a displacement control mode. In addition, suppose you create a profile in which you select the volts dimension to define end levels.

Then, you launch the MPT application against the station configuration, and with the Profile Command process on the **Command** tab, you select the profile you have created. On the **Mapping** tab, you select **Profile Channel**. In this scenario, displacement and force will be available in the **Control Mode** list.

Suppose you select displacement in the control mode list, and enter a value of 10 cm in the **Level Multiplier** box. This will cause all of the end-level values in the profile to be converted to displacement values. For volts, the full-scale value equals 10 V, which the MPT application will play out as 10 cm. So in this case, if the first end-level values in the profile are 2 V and 9 V, they will be played out as 2 cm and 9 cm.

Using the same profile, suppose you change the **Control Mode** selection to force, and enter a **Level Multiplier** value of 100 N. This will cause all of the end levels in the profile to be converted to force values. In this case, the MPT application will play out 10 V as 100 N. So if the first end-level values in the profile are 2 V and 9 V, they will be played out as 20 N and 90 N.

Full-scale values of normalized dimensions

Ratio = 1:1

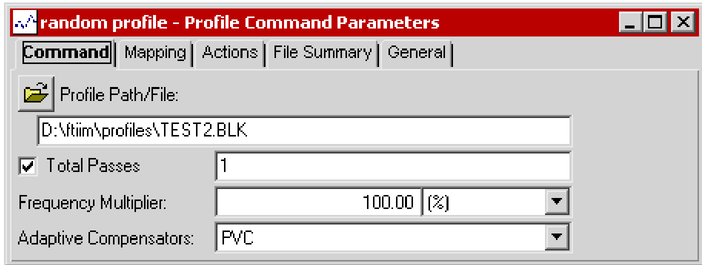
Percentage = 100%

Unitless = 1.0

Volts = 10 V

Profile Command Parameter Descriptions

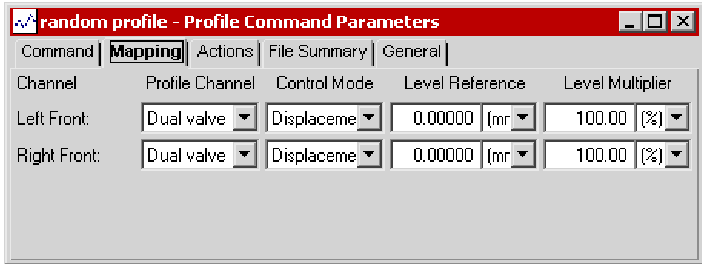
Command tab



Command Tab

ITEM	DESCRIPTION
Profile Path/File	Specifies the profile to be used.
Total Passes	Specifies how many times the profile will be played out. ***Note*** *If you clear the **Total Passes** box, the waveform will repeat indefinitely.*
Frequency Multiplier	Scales the segment rate type (time/rate/frequency) specified in the profile. If set to 100%, the profile will play out as defined in the profile. If set to 50%, the profile will play out at half speed. If it is set to 200%, the profile will play out at twice the speed.
Adaptive Compensators	Enables command compensation during the command process. See "About Command Compensation" on page 199. ***Note*** *To use the ALC compensator with your profile, you must define a Profile with ALC process instead.*

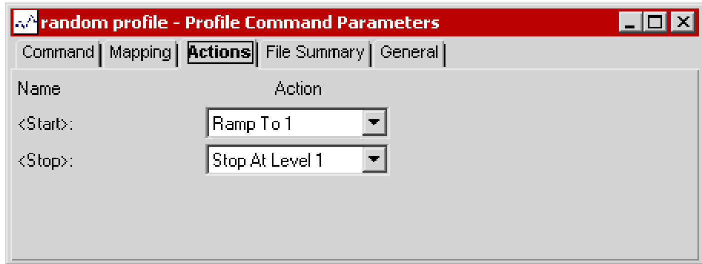
Mapping tab



Mapping Tab

ITEM	DESCRIPTION
Channel	Displays all station control channels.
Profile Channel	Selects the profile channel that will be played out on this station channel. You must map the logical channels in your profile to the control channels in your station configuration. You can arbitrarily map any profile channel to any station channel, or you can map a single profile channel to multiple station channels. If you leave any station channel unmapped, no profile will be played out on that channel. If you gave the profile channels the same name(s) as the station channels, they will automatically be mapped to the station channel with the matching name. A profile channel will appear in the **Profile Channel** list box next to its corresponding (station) **Channel**. Each **Profile Channel** list box will contain all the profile channels with units that can be played out by any control mode for the station channel. For example, a profile channel that measures length will not map to a station channel with a temperature control mode. If the profile was created with normalized dimensions then you can select any of the control modes available to the station channel when defining the control mode of the profile channel (see page 220).
Control Mode	Specifies the control mode for the selected station channel.
Level Reference	Introduces a mean level offset to the selected station channel.
Level Multiplier	Scales the end level values of the profile. The **Level Multiplier** will function for both engineering units and “normalized units.” ***Note*** *For normalized units, the **Profile Command** process uses the **Level Multiplier** value as the full scale.*

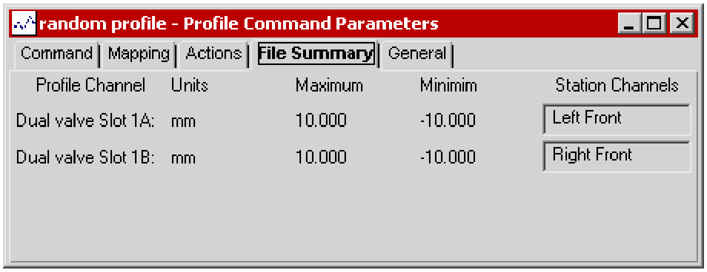
Actions tab



Action Tab

ITEM	DESCRIPTION
Name	Displays the name of the action in the profile.
Action	Displays the action, as defined by the Station Manager’s Event-Action editor, to be triggered when the associated action name (for example, <Start> and <Stop>) is encountered by the profile.

File Summary tab



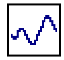
File Summary Tab

ITEM	DESCRIPTION
Profile Channel	Displays all available profile channels.
Units	Displays the unit of measurement for the end level dimension.
Maximum	Displays the maximum value for the channel. ***Note*** *This may be zero if the profile was not created using the MTS* ***Profile Editor*** *application.*
Minimum	Displays the minimum value for the channel. ***Note*** *This may be zero if the profile was not created using the MTS* ***Profile Editor*** *application.*
Station Channels	Displays the station control channels where the profile will be played out. ***Note*** *This box is blank if you have not mapped the profile channel to a station channel on the* ***Mapping*** *tab.*

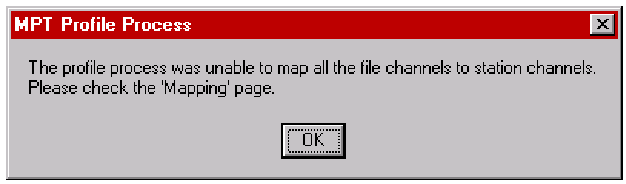
General tab

See “General tab” on page 202.

How to Define a Profile Command Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Command** tab.
4. Click the **Profile Path/File** button to open the **Select profile file** window. Click the desired profile, and then click **Open**.

   Note *If the channel names in the profile do not match the channel names in your station configuration, MPT will display the following:*

5. Select the **Total Passes** check box, and then type a value.
6. Optional—In the **Frequency Multiplier** box, set the units of measurement, and then type a value.
7. Optional—In the **Adaptive Compensators** list, click a compensator.
8. Click the **Mapping** tab.
9. Use the **Profile Channel** lists to map the desired profile channel to each station channel. (Station channel names are listed in the left column.)
10. For each station channel, select a **Control Mode** and optionally, a **Level Multiplier**, and **Level Reference**.
11. Click the **File Summary** tab and verify your channel settings.
12. Optional—Click the **Actions** tab to map station actions to file actions.

13. Click the **General** tab.

14. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)

15. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)

16. In the **Execute Process** box, enter the number of times you want the process to execute.

Note *The **Execute Process** value multiplies the **Total Passes** in the **Command** parameters. If the **Execute Process** parameter is set to 10 and the **Total Passes** is set to 5, a total of 50 passes will occur.*

17. Optional—In the **Counter Type** list, click a counter type.

18. Optional—If you enabled a counter, type a **Counter Label**.

19. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

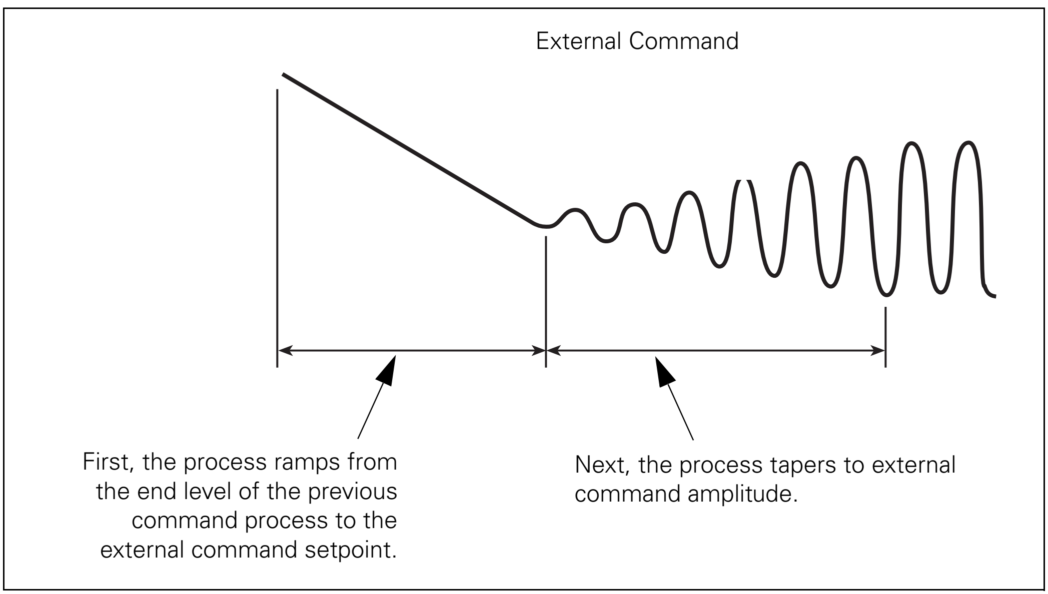
External Command

The External Command process accommodates program inputs from an external programmer or controller.

Before you can use an External Command process, you must:

- Connect the external command source to your controller.
- Use Station Builder to allocate an analog input resource (for the command input signal).
- Adjust the command input signals with the Station Manager application.

When you define an External Command process, you must define the command ramp time and which channel(s) the process will control.



More About the External Command Process

Starting external commands

If the external command is not running when MPT starts the **External Command** process, your test will stop responding.

To ensure your external program is running when it is needed, do one of the following:

- Start your external program source before you start your MPT procedure. This ensures that the external command is running when the **External Command** process starts and limits unwanted bumps in your command signal.
- If your external programmer supports digital input and output signals, configure a Digital Output to start the external program, and a Digital Input Detector to start the **External Command** process.

Stopping external command processes

Typically, you will use an MPT Data Limit Detector or Digital Input Detector to end the **External Command** process.

- You can use a Data Limit Detector to interrupt the **External Command** process after a specified time period or when specified signal limits are reached.
- If your external programmer can issue a signal when the program is complete, you can use a Digital Input Detector to interrupt the **External Command** process.
- You can stop the **External Command** process manually by pressing the stop control on the **Station Controls** panel. This stops the process and the procedure.
- You can resume the **External Command** process by pressing the run control on the **Station Controls** panel.

When the **External Command** process stops or is interrupted, the signal tapers to the setpoint specified on the **Scaling** tab. If a setpoint is not specified on the **Scaling** tab, the signal tapers to the system setpoint.

If you click **Run** after holding the command, the command will ramp back to setpoint, and then it will taper up to the defined level. (The initial ramp will be skipped if all the channels are still at the setpoint.)

Setpoint/span settings

You can also define a different setpoint and span to be used while the external command process plays out.

You should understand how setpoint and span settings on the different Station Manager and MPT windows affect one another.

- **Scaling** tab **External Command Parameters** window, MPT application.

  The **Change Setpoints** and **Change Spans** check boxes let you override the system setpoint and span values while the process is being executed.

- **Setpoint and Span** window, Station Manager application.

  This window contains the system setpoint (**Setpoint** slider) and span (**Span** slider) values that the External Command process overrides while the process is being executed.

- **Command Options** tab, **Channel Options** window, Station Manager application.

  The value in the **Setpoint** entry box determines the default rate at which the controller ramps to the setpoint value entered in the **External Command Parameters** window. Likewise, the **Begin** control determines the rate at which the controller ramps the external program to the span value entered in the **External Command Parameters** window.

- **Execution** tab, **MPT Options Editor**, **Span** and **Setpoint** controls, MPT application.

  The **Span** and **Setpoint** controls determine whether you can edit the setpoint and span values entered in the Station Manager's **Span and Setpoint** window while the External Command process is being executed.

Overriding system setpoint and span

With the **External Command** process, you can temporarily override the default system span and setpoint values, even if you select **Disable and Reset** for the **Span** and **Setpoint** controls on the **Execution** tab of the **MPT Options Editor**.

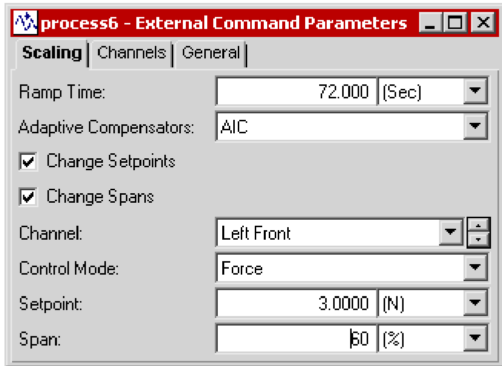
To do this, enable the **Change Setpoints** and/or **Change Spans** check boxes in the **Scaling** tab of the **External Command Parameters** window, and enter span and setpoint values in the associated entry boxes. This causes the span and setpoint values you enter to overwrite system span and setpoint values when MPT executes the External Command process.

The precise way MPT performs the override depends on whether you select **Disable and Reset** or **Enable**:

- If you select **Disable and Reset**, the controller will set span at 100% and setpoint at zero as it plays out the procedure until it begins to play out the External Command process. At that point the controller switches to the External Command process setpoint and span values. When the process is done, the setpoint reverts to zero, and the span reverts to 100% for the remainder of the procedure. With this selection, you cannot make adjustments to the **Setpoint** and **Span** sliders while the test is running.
- If you select **Enable**, the controller will use the system setpoint and span values as it plays out the procedure until it begins to play out the External Command process. At that point, the controller switches to the External Command process setpoint and span values and uses them for the remainder of the procedure. With this selection, you can make adjustments to the **Setpoint** and **Span** sliders while the procedure is running.

External Command Parameter Descriptions

Scaling tab



Scaling Tab (part 1 of 2)

ITEM	DESCRIPTION
Ramp Time	Specifies the amount of time the command ramps from the starting level of the process to the specified setpoint. If you set the **Ramp Time** to zero, the controller will use the **Setpoint/ Span** time specified on the Station Manager **Channel Options Command Options** tab.
Adaptive Compensators	Enables command compensation during the command process. See "About Command Compensation" on page 199.
Change Setpoints	Enables the **Setpoint** control. If disabled, the **Setpoint** will remain unchanged.
Change Spans	If enabled, the process will play out whatever percentage of full scale amplitude is entered in the **Span** box of this window. If disabled, the process will play out whatever percentage of full scale amplitude that is specified for the system span value. The system span value is set in Station Manager. (Pressing the **Span Setpoint** button on the **Station Controls** panel displays the **Setpoint and Span** window.)
Channel	Specifies the channel for which the controls on this tab apply.

Scaling Tab (part 2 of 2)

ITEM	DESCRIPTION
Control mode	Specifies the control mode. Only control modes that have the same dimension as the channel's external input signal are available.
Setpoint	Defines the setpoint value for the selected channel. The value entered in the **setpoint** box will overwrite the system setpoint value when the process becomes active.
Span	Defines the span value for the selected channel. The value entered in the **Span** box will overwrite the system span value when the process becomes active.

Channels tab

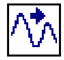
See "Channels tab" on page 202.

Note *Only channels in station configurations that have an external command signal defined in Station Builder are available to this process.*

General tab

See "General tab" on page 202.

How to Define an External Command Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Channels** tab.
4. Move the channels that you want to use to generate the command onto the **Included** list. Move all other channels to the **Available** list.
5. Click the **Scaling** tab.
6. In the **Ramp Time** box, set the units of measurement, and then type a value.
7. Optional—In the **Adaptive Compensators** list, click a compensator.
8. Optional—If you want the external command to use a different setpoint than the one specified in Station Manager, select the **Change Setpoints** check box.
9. Optional—If you want the external command to use a different span than the one specified in Station Manager, select the **Change Spans** check box.
10. In the **Channel** list, click your desired channel.
11. In the **Control Mode** list, click your desired control mode.
12. If you selected **Change Setpoints**, define the new setpoint value and units in the **Setpoint** box.
13. If you selected **Change Spans**, enter a span value for the external command in the **Span** box.
14. Repeat steps 10 - 13 for each channel you wish command with the **External Command** process.

15. Click the **General** tab.
16. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
17. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
18. In the **Execute Process** box, enter the number of times you want the process to execute.
19. Optional—In the **Counter Type** list, click a counter type.
20. Optional—If you enabled a counter, type a **Counter Label**.
21. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

Profile with ALC

The **Profile with ALC** process works the same as the Profile process (see page 219). You must use the **Profile with ALC** process if you want to play out a profile and use ALC (arbitrary end-level compensation).

Note *The Profile with ALC process is optional. You must enter a special key code during installation to enable this process.*

For additional information on creating and playing out profiles, see Appendix C, "Profiles".

About ALC compensation

Arbitrary end-level control compensation is an adaptive compensation technique that improves the tracking accuracy of MPT.

ALC compensates for peak and valley errors by building (and continually updating) a matrix of amplitude compensation factors.

For detailed information about the ALC compensator, see the *Model 793.00 System Software* manual.

Profile with ALC Parameter Descriptions

Command tab

See "Command tab" on page 222.

Mapping tab

See "Mapping tab" on page 223.

Actions tab

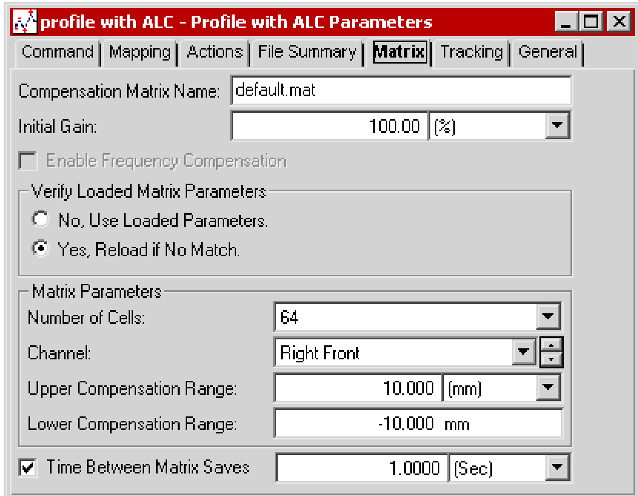
See "Actions tab" on page 224.

File Summary tab

See "File Summary tab" on page 225.

Matrix tab

Note *This tab is enabled only if ALC has been selected as the* ***Adaptive Compensator*** *on the* ***Command*** *tab.*



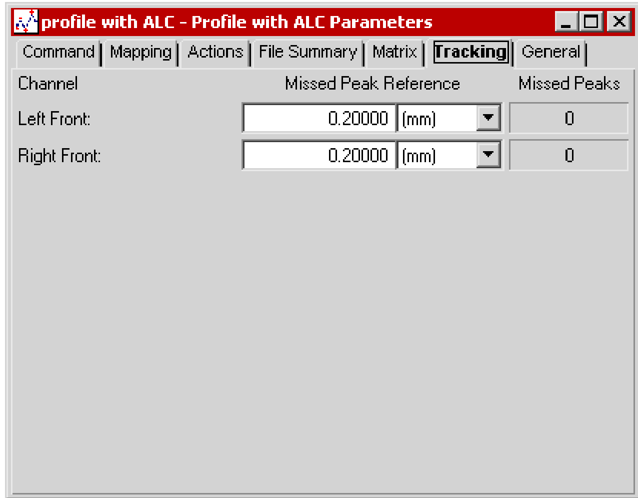
Matrix Tab (part 1 of 2)

ITEM	DESCRIPTION
Compensation Matrix Name	Names the matrix used by the ALC compensator. ALC matrix files are saved in the Specimen directory. The default ALC matrix file is **default.mat**. If you want to use an existing matrix, copy the .mat file into the Specimen directory, and then enter the matrix file name in the **Compensation Matrix Name** box.
Initial Gain	Specifies the amount of initial gain applied to the signal. The default is 100%, or no magnification.
Enable Frequency Compensation	Attempts to run the test at a faster rate than specified in the selected profile. For example, if the profile specifies that the end levels be achieved using a frequency of 0.2 Hz, the application may actually achieve these end levels at 0.5 Hz. The maximum rate increase is specified on the **ALC** tab in the **Compensators** tab in the **Station Setup** window in Station Manager. ***Note*** *This option is available only if you have specified a single-channel profile.*

Matrix Tab (part 2 of 2)

ITEM	DESCRIPTION
Verify Loaded Matrix Parameters	Specifies whether you want the application to compare the matrix parameters entered on this Matrix window with the parameters currently loaded in the controller's real-time. If you choose **No, Use Loaded Parameters** the application will disregard any file saved on disk that matches the entered name and use the matrix parameters currently loaded in the controller's real-time. If you choose **Yes, Reload if No Match**. the application will compare the matrix parameters entered in this Matrix window with the parameters currently loaded in the controller's real-time. If the parameters do not match, the application will redefine the real-time matrix using the parameters currently entered in this Matrix window. In this case the application will also compare the entered parameters with those stored in any file in the current Specimen directory that uses the entered name. If they match, the matrix values from the disk file will be down-loaded into the real-time matrix. If not, the real-time matrix values will be reset to indicate no correction values are present. To use a saved compensation matrix for the current test, you must copy (using Windows operating software) the desired matrix file into the current Specimen directory before you enter the desired matrix name (as it appears in the Specimen directory) and definition parameters in this window. If the name or other parameters do not match precisely, or if the desired matrix does not exist in the current Specimen directory, the application will create a new real-time matrix with the entered name.
Number of Cells	Specifies the number of cells in each axis of the matrix (16, 32, or 64), which determines the resolution of the correction (the higher the number, the greater the resolution).
Channel	Selects the channel to which the compensation ranges apply. Compensation ranges must be entered for each channel used by the profile.
Upper Compensation Range	Specifies the upper range value and associated units of the compensation matrix for the selected channel.
Lower Compensation Range	Specifies the lower range value and associated units of the compensation matrix for the selected channel.
Time Between Matrix Saves	Selects the time interval between matrix saves. If you Hold the test, the timer for saving the matrix also holds. ***Note*** *If you want the matrix to be saved only at the end of the test, enter a time longer than the profile.*

Tracking tab



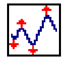
Tracking Tab

ITEM	DESCRIPTION
Channel	Lists the channel(s) to which the ALC compensation will apply.
Missed Peak Reference	Specifies the difference between the commanded peak and the actual peak achieved that you want the application to detect.
Missed Peaks	Displays the total number of missed peaks (also stored in the .log file). This value is saved as part of the run-time state of the process (in the .mps file), and can be restored accordingly. For more information, see “Specimen.mps” on page 79. ***Note*** *When you reset the procedure, the ALC missed peaks values displayed on the **Tracking** tab are not immediately reset to zero. However, when you run the test again the process will start counting (and displaying) from zero.*

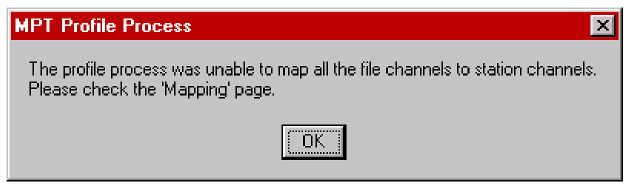
General tab

See “General tab” on page 202.

How to Define a Profile with ALC Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Command** tab.
4. Click the **Profile Path/File** button to open the **Select profile file** window. Click the desired profile, and then click **Open**.

   Note *If the channel names in the profile do not match the channel names in your station configuration, MPT will display the following:*



5. Select the **Total Passes** check box, and then type a value.
6. In the **Frequency Multiplier** box, set the units of measurement, and then type a value.
7. Click the **Mapping** tab.
8. Use the **Profile Channel** lists to map the desired profile channel to each station channel. (Station channel names are listed in the left column.)
9. For each station channel, select a **Control Mode** and optionally, a **Level Multiplier**, and **Level Reference**.
10. Click the **Command** tab.
11. In the **Adaptive Compensators** list, select the **ALC** compensator.
12. Optional—click the **Actions** tab, and map station actions to profile actions.
13. Click the **File Summary** tab and verify your channel settings.

14. Click the **Matrix** tab and enter a **Compensation Matrix Name**.
15. Optional—Select the **Enable Frequency Compensation** check box.
16. Under **Verify Loaded Matrix Parameters**, select whether or not you want the application to verify the loaded matrix parameters.
17. Under **Matrix Parameters**, configure your ALC matrix.
18. Optional—If you want the matrix to be saved, select the **Time Between Matrix Saves** check box, and then enter a save interval.
19. Click the **Tracking** tab.
20. Set the units and enter a value for the **Missed Peak Reference** for each channel.
21. Click the **General** tab.
22. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
23. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
24. In the **Execute Process** box, enter the number of times you want the process to execute.
25. Optional—In the **Counter Type** list, click a counter type.
26. Optional—If you enabled a counter, type a **Counter Label**.
27. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

Cyclic with ALC

The **Cyclic with ALC** process works the same as the Cyclic process (see page 205). You must use the **Cyclic with ALC** process if you want to program a cyclic command and use ALC (arbitrary end-level compensation).

Note *The Cyclic with ALC process is optional. You must enter a special key code during installation to enable this process.*

About ALC compensation

Arbitrary end-level control compensation is an adaptive compensation technique that improves the tracking accuracy of MPT.

ALC compensates for peak and valley errors by building (and continually updating) a matrix of amplitude compensation factors.

For detailed information on the ALC compensator, see the *Model 793.00 System Software* manual.

Cyclic with ALC Parameter Descriptions

Command tab

See “Command tab” on page 212.

Channels tab

See “Channels tab” on page 202.

Matrix tab

See “Matrix tab” on page 237.

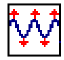
Tracking tab

See “Tracking tab” on page 239.

General tab

See “General tab” on page 202.

How to Define a Cyclic with ALC Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Channels** tab.
4. Move the channels that you want to use to generate the cyclic command onto the **Included** list. Move the other channels to the **Available** list.
5. Click the **Command** tab.
6. In the **Segment Shape** list, click the segment shape.
7. Select **Time**, **Rate**, or **Frequency**, and set the units of measurement, and then and type a value.
8. Select the **Count** check box. Set the units (cycles or segments), and then type the number of times the cyclic command is to repeat. For continuous cycling, clear the **Count** check box.
9. In the **Adaptive Compensators** list, click ALC.
10. Optional—If desired, select the **Do Not Update Counters** check box.
11. Optional—If you want to define a relative end level, select the **Relative End Levels** check box.
12. Select a **Channel**.
13. Select a **Control Mode**.
14. Select the units for the end levels, then enter values for **End Level 1** and **End Level 2**.
15. Enter a **Phase Lag**.
16. Repeat steps 12 - 15 for each channel you want to command with this process.
17. Click the **Matrix** tab and enter a **Compensation Matrix Name**.
18. Optional—Select **Enable Frequency Compensation**.
19. Choose whether you want the application to verify the loaded matrix parameters.

20. Under **Matrix Parameters**, configure your ALC matrix.
21. Optional—If you want the matrix to be saved, select the **Time Between Matrix Saves** check box, and then enter a save interval.
22. Click the **Tracking** tab and enter a **Missed Peak Reference** amount and corresponding units.
23. Click the **General** tab.
24. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
25. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
26. In the **Execute Process** box, enter the number of times you want the process to execute.
27. Optional—In the **Counter Type** list, click a counter type.
28. Optional—If you enabled a counter, type a **Counter Label**.
29. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

Time History Output

The Time History Output process drives one or more channels in your station with an RPC II or RPC III Time History file (with extension .drv).

It can also simultaneously digitize and store one or more channels of analog test response data in the RPC III file format. The digitized data will have the same sampling rate as the drive file, and the number of points sampled for the channel will match the drive file. This digitized file can be used to measure the system's transfer function or iterate to a desired response signal. It can also be used with the RPC suite of software applications.

Note *The Time History Output process is optional. You must enter a special key code during installation to enable this process. The purpose of this process is to drive station channels with time history files, and includes **Command**, **Mapping**, **Actions**, **Status**, and **General** tabs. This process offers an optional data acquisition feature, which also requires a special key code during installation. The data acquisition feature adds **Acquisition** and **Trend** tabs to this process. All tabs are documented in this section.*

About Time History Output and Controller Rates

Time History rates supported by the controller are dependent on two parameters specified in your controller's hardware interface file:

- System rate
- High-clock rate

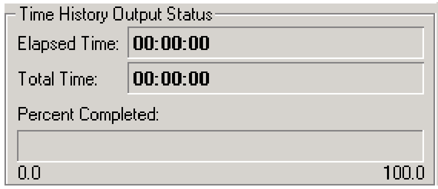
Allowable command generation (drive) rates are sub-multiples of the system rate; that is, they equal the system rate divided by certain values (1, 2, 2.5, 5, and so on). This is because the time histories are up-sampled from their given sampling rates to the system rate, and then digitally filtered to remove the effects of up-sampling. For standard settings of the system rate, time histories with the following sampling rates are supported:

- 204.8 Hz
- 409.6 Hz
- 512.0 Hz
- 1024 Hz

The up-sampling filters are designed to begin cutting off the signal at 35% of the sampling rate and to be 60 dB down at 65% of that rate. When a time history is entered into this process, the software will check the signal's sampling rate and determine whether it can be supported. If not, an error message will be generated.

Data acquisition rates are sub-multiples of the high-clock rate. If the RPC data acquisition option is enabled on your system, analog test data will be filtered using a 5-pole analog Bessel filter with a cutoff of 300 Hz. After being digitized, the signals are run through a digital filter whose cutoff is 40% of the requested sampling frequency, and then down-sampled to the requested sampling rate. These filters matched the ones used in RPC III 498 DSPAD boards, so data can be used interchangeably with that system. If RPC data acquisition is enabled, the software will also check the signal's sampling rate to ensure that acquisition can also be supported.

Time History Output Status Window

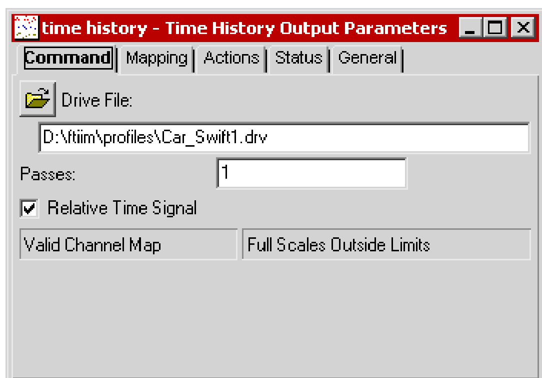


Note *This panel is displayed only if the user enables the* ***Show Status Panel at Runtime*** *option on the* ***Status*** *tab.*

During a test, the **Time History Output (or Input) Status** panel displays on the MPT control panel. The status panel appears when you lock the procedure.

Time History Output Parameter Descriptions

Command tab



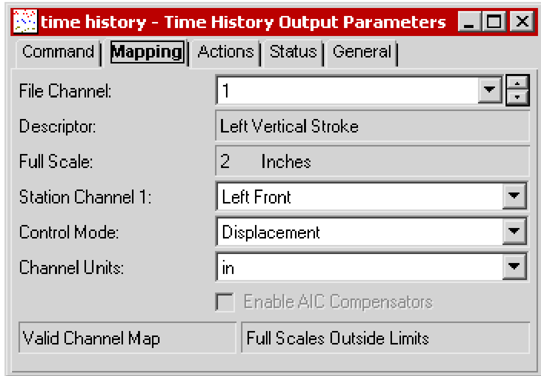
Command Tab (part 1 of 2)

ITEM	DESCRIPTION
Drive File	Selects an RPC II or RPC III drive file (.drv) to play out to the station. When you have selected a file, the application: • Verifies that it is a file it can read (either RPC II or RPC III format). • Reads in the file header. • Ensures that the number of channels in the file is not greater than the number of channels in the station. • Ensures that the sampling rate is supported. If any of these steps fail, a message will display the problem. If no problem exists, the application will attempt to map the file's channels to the station's channels. Next, the application will check for channel conflicts. You may make limited changes to the contents of an RPC time history file between the time its name is entered here and when it is actually used during the execution of a procedure. MPT will compare the file header at execution time to its contents during the definition phase. If it detects any differences in any of the following parameters, an error status will be returned and the test will be terminated: • RPC File Type (RPC II vs. RPC III) • Number of channels • Points per group • Sampling rate • Channel full scales • Channel descriptors • Channel units • Channel mapping

Command Tab (part 2 of 2)

ITEM	DESCRIPTION
Passes	Specifies the number of passes the drive file will play out each time the process executes, as specified on the **General** tab. This function differs from the **Execute Process** value on the **General** tab. The difference is that the **Time History Output** process will come to a complete halt and then re-initialize and restart for each **Execute Process** count specified in the **General** tab. This creates a delay between the end of one count and the beginning of another. Passes entered here do not result in a delay because the application simply reads data from the beginning of the already opened file and queues it in real time, which eliminates any delay.
Relative Time Signal	If checked, the process will read the current command value on each channel when the process begins executing. These commands will be added to each point in the time history as it is read from its file and the modified command will be played out. This allows you to preload a channel and then play out the time history relative to that preloaded value. If not checked, the time history will be played out as stored in the disk file.
Status	Displays messages about channel mapping and full-scale status after the application checks the mapping and unit selections on the **Mapping** tab. These messages are updated each time you modify the channel, mode, or unit selection on the **Mapping** tab. The application will ensure that no two file channels are mapped to the same station channel. When the check is complete, the application will display either “Invalid Channel Map” or “Valid Channel Map.” The process cannot be executed with an invalid channel map. If the mapping is valid, the application checks full scales of each channel in the file against the range of the selected control mode for each channel. The message “Full Scales Outside Limits” or “Full Scales In Limits” will display the results. If the mapping is bad, the outside limits text will display. Since an RPC time file does not necessarily have data near its full scale, the command may not actually exceed the channel’s limits when the drive file is played out. However, the message is intended to warn you the potential exists for driving a channel outside its limits.

Mapping tab



Mapping Tab (part 1 of 2)

ITEM	DESCRIPTION
File Channel/ Descriptor/Full Scale	**File Channel** selects the file channel's number. The **Descriptor** and **Full Scale** values are updated with the file header values for the selected file channel. The values are updated as you change the selected file channel. When you change the file channel selection, the application shows the currently selected station channel, control mode, and units to be used so that the data can be converted into the station channel's system units.
Station Channel	Selects the station channel associated with the selected file channel. When a drive file is read, the application looks to see if the file channel's descriptor matches the name of a station channel. If it does, the application maps that file channel to the matching station channel.
Control Mode	Selects the control mode associated with the selected station channel. By default, for each file channel the application selects the first control mode available for the associated station channel. This defines the dimension of the signal. By default, the units for the selected control mode's signal will be the calibrated units defined in Station Manager. However, when the application parses the requested drive file's header, if the application finds that the channel's unit descriptor matches one of the units available for that dimension, the application will select those units.

Mapping Tab (part 2 of 2)

ITEM	DESCRIPTION
Channel Units	Selects the units associated with the selected file channel.
Enable AIC Compensators	Enables the Adaptive Inverse Control (AIC) compensation method for all control channels that will support it. This box is available when the file has been entered, a valid channel map exists, and at least one of the selected channels supports AIC Compensation (which is done in Station Builder—see the **Edit Control Channels** section for Station Builder in the *Model 793.00 System Software* manual). ***Note*** *This compensation method will not optimize signals with frequency content above 128Hz. Therefore, this process will not allow AIC to be enabled if the sampling rate of the time histories exceed 256Hz.*
Status	Displays messages about channel mapping and full-scale status after the application checks the mapping and unit selections on the **Mapping** tab. These messages are updated each time the user modifies the channel, mode, or unit selection in the **Mapping** tab. The application will ensure that no two file channels are mapped to the same station channel. When the check is complete, the application will display either “Invalid Channel Map” or “Valid Channel Map.” The process cannot be executed with an invalid channel map. If the mapping is valid, the application checks full scales of each channel in the file against the range of the selected control mode for each channel. The message “Full Scales Outside Limits” or “Full Scales Inside Limits” will display the results. If the mapping is bad, the outside limits text will display. The valid/invalid channel map message and full scale in/outside limits messages reflect the validity of the channel mapping and the comparison of each file channel’s full scale versus the station channel’s full scale.

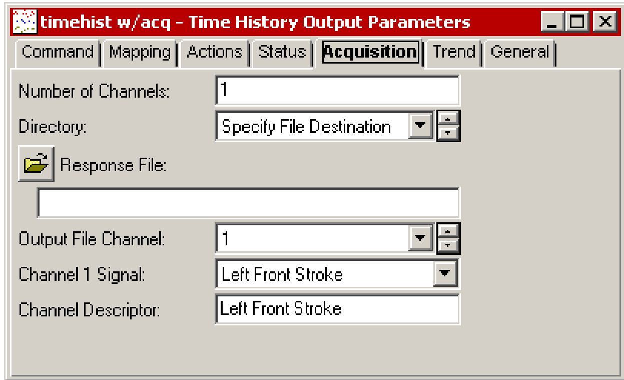
Actions tab

See “Actions tab” on page 236.

Note *When referring to the **Actions** tab, keep in mind for Time History processes you will be selecting “sequences” for a drive file, instead of <actions> for a profile.*

Acquisition tab

Note *This tab appears only if you have purchased the data acquisition option of the Time History Output process.*



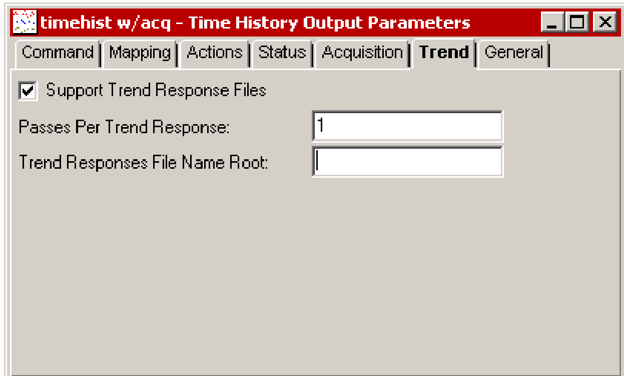
Acquisition Tab

ITEM	DESCRIPTION
Number of Channels	Specifies the number of channels on which data is acquired and formatted. The default number is zero.
Directory	Specifies the directory in which the response file (.rsp) will be saved. The **Specify File Destination** selection allows you to specify a path for the response file. The **Use Specimen Directory** selection saves the response file in the specimen directory. The default name of the response file is “new.rsp”—enter the desired file name in place of “new.”
Response File *(file icon)*	The process stores digitized response test data in a response file. If the **Directory** selection is **Specify File Destination**, the **Response File** file icon will be active. Click the file icon to open the **Select response file** window. Use the window to select a current response file, save a new response file, and specify the path.
Output File Channel	Specifies the output file channel.
***Channel 1* Signal**	Specifies the station signal associated with the selected output file channel.
Channel Descriptor	Specifies a descriptor for the selected output file channel that will be stored in the files header. The default entry is the station signal name.

Note *If you are executing a **Time History Output** process that includes data acquisition, and you hold and resume a test, there will be slight differences between the digitized response and the response you would have seen if the test had run without holding. These differences should not be statistically significant.*

Trend tab

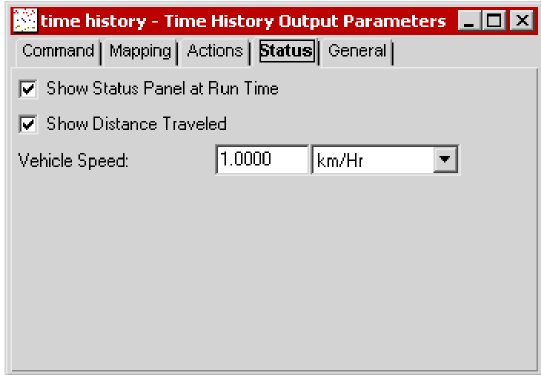
Note *This tab appears only if you have purchased the data acquisition option of the Time History Output process.*



Trend Tab

ITEM	DESCRIPTION
Support Trend Response Files	Enables MPT to acquire and store trend response data. This data can be used by the RPC III application to show changes that occurred in the specimen over the course of the test.
Passes Per Trend Response	Specifies the number of passes the drive file will play out before a trend response file is stored.
Trend Response File Name Root	Specifies the root of trend response file names. Trend response files are stored in the same folder as the response file defined on the **Acquisition** tab. When the first response file is saved, it is named *your root name*000.rsp. Subsequent response files are named *your root name*001.rsp, *your root name*002.rsp, *your root name*003.rsp, and so on.

Status tab



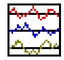
Status Tab

ITEM	DESCRIPTION
Show Status Panel at Run Time	Check this box to make a status panel appear in the MPT control panel when this process runs. The status panel displays the progress of the process execution. If you do not enable the **Show Distance Traveled** checkbox, the status panel will display process progress in terms of **Elapsed Time**, **Total Time** (target time), and **Percent Completed**.
Show Distance Traveled/Vehicle Speed	Check this box to make the status panel display process progress in terms of **Distance**, **Total Distance** (target distance), and **Percent Completed**. Checking this box enables the **Vehicle Speed** control.
Vehicle Speed	Lets you enter a velocity for the vehicle. The distance displayed on the status panel is calculated from the process' running time.

General tab

See “General tab” on page 202.

How to Define a Time History Output Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. In the parameters window, click the **Command** tab.
4. Click the **Drive File** button to open the **Select drive file** window. Click the desired drive file, and then click **Open**.
5. In the **Passes** box, enter of number of passes you want the drive file to make.
6. Optional—Select **Relative Time Signal** if you want to enable a relative (instead of absolute) time signal.
7. Click the **Mapping** tab.
8. Use the **File Channel** and **Station Channel** lists to map the drive file channels to the appropriate station channels. Set the **Control Mode** and **Channel Units** as needed.
9. Optional—Select **Enable AIC Compensators**.

Note *The Acquisition and Trend tabs apply only if you have the data acquisition option for the Time History Output process.*

10. Click the **Acquisition** tab.
11. In the **Number of Channels** list, type the number of channels you want to take data on and press **Enter**.
12. Use the **Directory** control to specify the directory in which the response file (.rsp) will be saved. The **Specify File Destination** selection allows you to specify a path for the response file. The **Use Specimen Directory** selection saves the response file in the specimen directory.
13. If you have selected **Specify File Destination** in the previous step, you can click the **Response File** file icon to open the **Select response file** window. Use this window to select the file and directory in which you wish to save response data.
14. In the **Output File Channel** list, click the first channel.
15. In the **Channel 1 Signal** list, click the signal to be monitored. (You can enter a descriptor in the **Channel Descriptor** box.)

16. Perform steps 14 and 15 for any additional channels.
17. Click the **Trend** tab.
18. Optional—Select **Support Trend Response Files** if you want to store trend data for use with the RPC application.

    If you select **Support Trend Response Files**:

    A. In the **Passes Per Trend Response** box, type the number of passes you want to play out before you save trend data.

    B. In the **Trend Responses File Name Root** box, type the root name for your trend response files.
19. Click the **Actions** tab.
20. Optional—Map station actions to drive file sequences.
21. Click the **Status** tab.
22. Optional—Select **Show Status Panel at Run Time** if you want the Time History Output Status Window to display when this process is active.
23. Click the **General** tab.
24. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
25. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
26. In the **Execute Process** box, enter the number of times you want the process to execute.
27. Optional—In the **Counter Type** list, click a counter type.
28. Optional—If you enabled a counter, type a **Counter Label**.
29. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

Road Surface Output

The **Road Surface Output** process lets you drive a station with RPC III and RPC Pro Road Surface files (with extensions .drvrs and .seq)

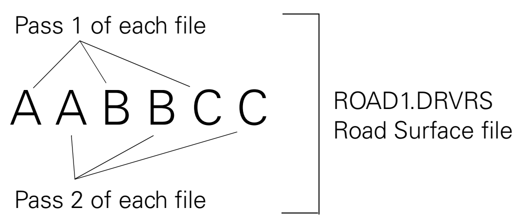
Note *The **Road Surface Output** process is optional. You must enter a special key code during installation to enable this process.*

About Road Surface Files

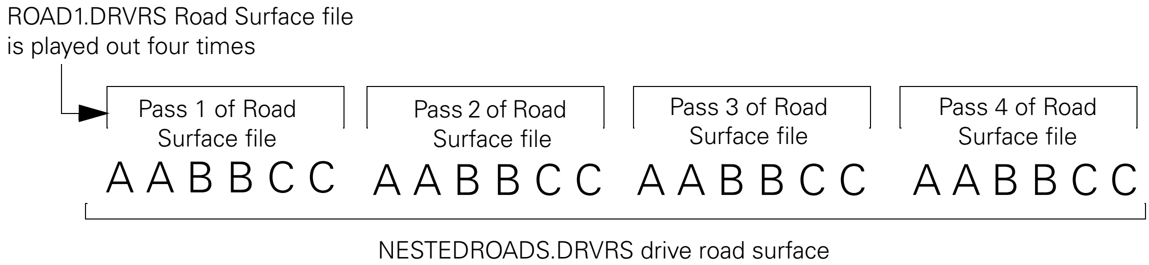
A Road Surface file is a file that you create with RPC software. It consists of a list of time history files and other Road Surface files. Each file can have a different number of channels.

Creating road surface files

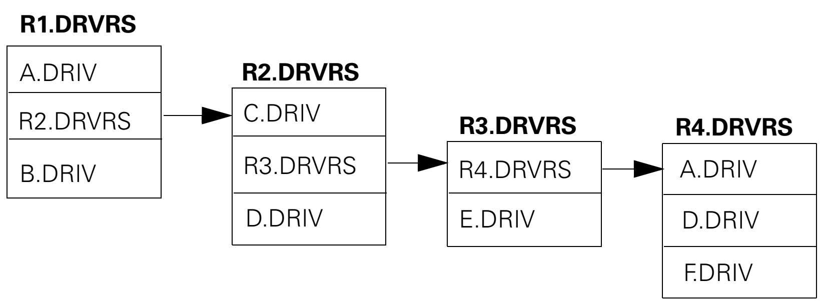
A Road Surface file is created by the RPC RSURF program or RPC Pro. It lets you enter up to 30 file names in one Road Surface file. The order in which files are listed determines the order in which they are played out. For example, if files A, B, and C are entered in the list in alphabetical order and each file has two passes, the files are played out as follows:



RSURF allows you to nest files to four levels. The following diagram shows two levels of nesting. Files A, B, and C make up a Road Surface file named “ROAD1.DRVRS.” The level 2 drive Road Surface file, “NESTEDROADS.DRVRS”, contains four passes of ROAD1.DRVRS.



You can nest a Road Surface file inside three other Road Surface files to produce a complex Road Surface file of drive files. For example, in the following diagram, R4.DRVRS is nested in Road Surface files R3.DRVRS, R2.DRVRS, and R1.DRVRS:

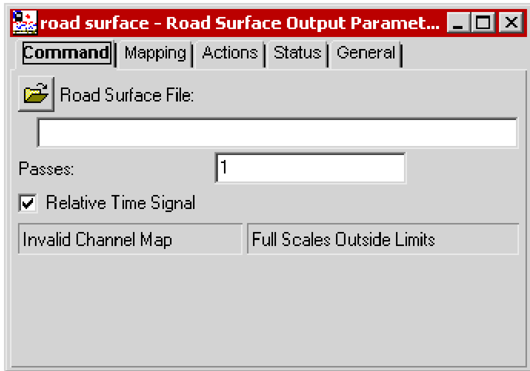


Road Surface Output Status Panel

During a test, the **Road Surface Output Status** panel displays on the MPT control panel. The status panel appears when you lock the procedure. Also, the sequence counter on the MPT control panel displays all files within the .DRVRS file, and indicates the active file.

Road Surface Output Parameter Descriptions

Command tab



Command Tab (part 1 of 2)

ITEM	DESCRIPTION
Road Surface File	Selects an RPC III or RPC Pro Road Surface file to play out. When you have selected a file, the application: • Verifies that it is a file it can read. • Reads in the file. • Reads in the headers of all time history files specified in the Road Surface file and any nested road surface file. • Ensures that the number of channels in each drive file is not greater than the number of channels in the station. • Ensures that the sampling rate is supported. • Ensures that the sampling rate and the channel calibrations (values and units) are consistent between all drive files. If any of these steps fail, a message will display the problem. If no problem exists, the application will attempt to map the file's channels to the station's channels. Next, the application will check for channel conflicts.
Passes	Selects the number of passes the Road Surface file will play out each time the process executes, as specified on the **General** tab.

Command Tab (part 2 of 2)

ITEM	DESCRIPTION
Relative Time Signal	When enabled, the process will read the current command value on each channel when the process begins executing. These commands will be added to each point in the time history as it is read from its file and the modified command will be played out. This allows you to preload a channel and then play out the time history relative to that preloaded value. If not checked, the time history will be played out as stored in the disk file.
Status	Displays messages about channel mapping and full-scale status after the application checks the mapping and unit selections on the **Mapping** tab. These messages are updated each time the user modifies the channel, mode, or unit selection on the **Mapping** tab. The application will ensure that no two file channels are mapped to the same station channel. When the check is complete, the application will display either “Invalid Channel Map” or “Valid Channel Map.” The process cannot be executed with an invalid channel map. If the mapping is valid, the application checks full scales of each channel in the file against the range of the selected control mode for each channel. The message “Full Scales Outside Limits” or “Full Scales in Limits” will display the results. If the mapping is bad, the outside limits text will display. The valid/invalid channel map message and full scale in/outside limits messages reflect the validity of the channel mapping and the comparison of each file channel’s full scale versus the station channel’s full scale.

Mapping tab

See “Mapping tab” on page 250.

Actions tab

See “Actions tab” on page 236.

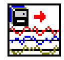
Status tab

See “Status tab” on page 254.

General tab

See “General tab” on page 202.

How to Define a Road Surface Output Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Command** tab.
4. Click the **Road Surface File** button to open the **Select road surface file** window. Click the desired drive file, and then click **Open**.
5. In the **Passes** box, enter the number of passes you want the drive file to make.
6. Optional—Select **Relative Time Signal** if you want to enable a relative (instead of absolute) time signal.
7. Click the **Mapping** tab.
8. Use the **File Channel** and **Station Channel** lists to map the drive file channels to the appropriate station channels. Set the **Control Mode** and **Channel Units** as needed.
9. Optional—Select **Enable AIC Compensators**.
10. Click the **Actions** tab.
11. Optional—Map station actions to drive file sequences.
12. Click on the **Status** tab.
13. Optional—Select **Show Status Panel at Run Time** if you want the Road Surface Output Status Panel to display when this process is active.

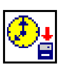
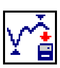
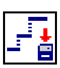
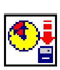
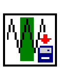
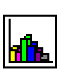
14. Click the **General** tab.
15. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
16. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
17. In the **Execute Process** box, enter the number of times you want the process to execute.
18. Optional—In the **Counter Type** list, click a counter type.
19. Optional—If you enabled a counter, type a **Counter Label**.
20. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process.

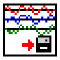
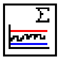
Data Acquisition Processes

Use data acquisition processes to acquire sensor data from the physical specimen during testing.

You must sequence this process in parallel with command processes. Meaningful data cannot be acquired unless the command process is performing an action that can be measured.

Note *There is a slight time delay between processes that start with a common trigger (for example, two processes using the <Procedure.Start> trigger). The process positioned the highest on the Procedure or Group table starts first. When using data acquisition and command processes in parallel, position the data acquisition process higher on the table than the associated command process..*

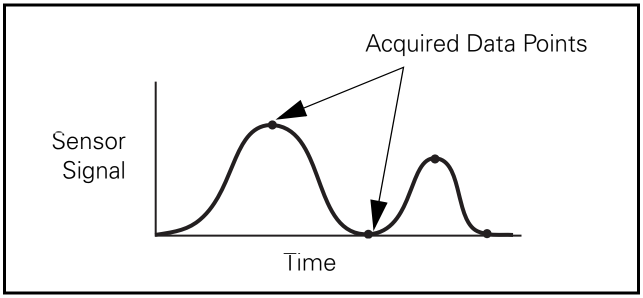
Icon	Process Name	Description
	**Peak/Valley Acquisition**	Records the peak/valley data for each cycle execution. (See page 265.)
	**Timed Acquisition**	Records data at user-specified time intervals. (See page 277.)
	**Max/Min Acquisition**	Records data at the highest peak and lowest valley during the life of the process. (See page 281.)
	**Level Crossing Acquisition**	Records data each time the selected signal changes by a specified amount. (See page 286.)
	**High Speed Timed Acquisition**	Records bursts of data at a high rate. (See page 290.)
	**Cyclic Acquisition**	Records timed, level crossing, interleaved timed and level crossing, or peak valley data from cyclic feedback. (See page 295.)
	**Fatigue**	Accumulates cycles in a histogram, then calculates damage based on the data on that histogram. The damage number is also sent to the Fatigue Monitoring application. (See page 303.)

Icon	Process Name	Description
	**Time History Input**	Acquires and digitizes multiple channels of analog test response data in the RPC III file format. (See page 316.)
	**Trend**	Calculates statistical values on time series data. The values are then stored to disk. It is also sent to the Trend Monitoring application. (See page 321.)

Peak/Valley Acquisition

The **Peak/Valley Acquisition** process records the output of all selected signals when the software detects a peak or valley in the master signal you specify.

When you define a **Peak/Valley Acquisition** process, you must define the master signal, the sensitivity, the signals you will record data on, the data destination, the buffer type, and the output units.



More About Data Acquisition Processes

When you set up data acquisition, you specify the signals for which data is acquired and the method of buffering.

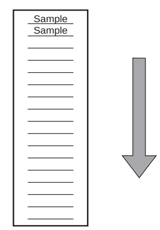
Buffers

Each type of buffer offers different operational characteristics:

- The buffer size specifies the maximum number of data elements the buffer will store before data is written to disk. (A data element includes the data from each selected signal.) You can set the buffer size between 1 and 16,000 data elements (default is 1024).
- Acquiring and saving data at fast rates can cause the computer to become sluggish (slow to respond to selections). If the acquisition rates are too fast, data over-run can occur. If this happens, the test stops and a message is logged.
- Linear buffers can generate a Process.Buffer Full signal when the controller acquires enough data elements to fill the specified buffer size. You can use this buffer full signal to start or interrupt other processes in the procedure.

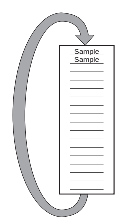
Linear buffer

A linear buffer records data until it is full, then saves the data to disk. The size of the buffer determines how much data is acquired before it is written to disk. Data is continuously saved to disk until the process ends or is stopped. When the process ends, any data in the buffer is sent to disk. The only limit is the size of your hard drive.



Circular buffer

A circular buffer records data continuously. When the buffer is full, new data overwrites the oldest data. This type of buffering saves data to disk when the test is stopped, when the process reaches its preset count, or when the process ends. The circular buffer is useful for acquiring data just before some crucial event (such as specimen failure), where data is not required for the whole test.



Data file header

The first line of the data file includes information like the decimal separator, column separator, time formats, date formats, and so forth.

A typical header looks like this:

MTS793|MPT|ENU|1|0|.|/|:|1|0|0|A

The format of this header line is as follows:

MTS793|progName|sLanguage|version|iDelim|sDecimal|sDate|sTime|iCountry|iDate|iTime|sCode

Where:

progName=	MPT or BTW
sLanguage=	Natural language (standard three-character abbreviation)
version=	Version of this format string
iDelim=	Data delimeter: 0 for space (plain), 1 for comma (Lotus), 2 for tab (Excel)
sDecimal=	International decimal separator
sDate=	International date separator
sTime=	International time separator
iCountry=	International country value
iDate=	International date format
iTime=	International time format
sCode=	Character Set Code: A for ANSI, O for OEM (currently always A)

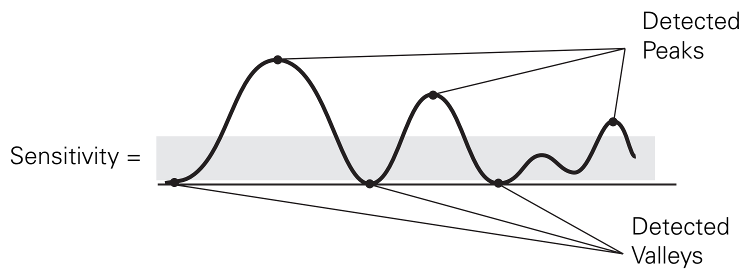
MPT generates a header for each data file buffer when the **Write First Data Header Only** check box is disabled (unchecked). When the check box is enabled, MPT suppresses all but the initial header.

Important *You can use the **Write First Data Header Only** check box to specify whether or not headers are written to the buffer subsequent to the initial header. This option is provided as an aid to data sorting and analysis and is only provided for the Peak/Valley, Timed, Level Crossing, Cyclic Acquisition, and High Speed Acquisition processes.*

Sensitivity

A sensitivity value specifies how much the master signal must change to detect a peak or valley:

- Setting the sensitivity too low may cause signal noise to be recognized as peaks and valleys.
- Setting the sensitivity too high may cause low amplitude signals to be missed.



Data destination

You set how your acquired data is saved on the **Destination** tab. The options are:

- **Specimen data file**

  Acquired data is written to the **specimen.dat** file saved in your MPT specimen.

  MPT specimens are located in your **MPT Specimens** folder (MPT > Specimens > spec01, spec02, and so forth).

- **User-specified data file**

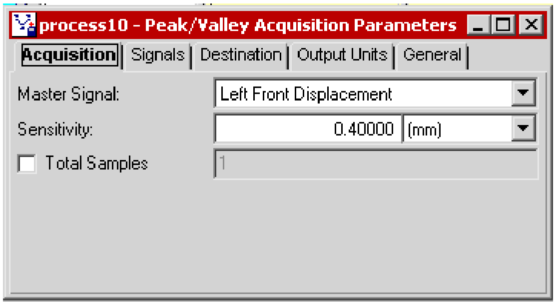
  Acquired data is written to the data file you specify (*filename*.dat) in the MPT specimen.

- **Discard data**

  Acquired data is written to the buffer, but is not saved to disk. Typically, this option is used to trigger another process when the current acquisition process fills its buffer. See "How to Trigger a Process When the Buffer is Full" on page 275.

Peak/Valley Acquisition Parameter Descriptions

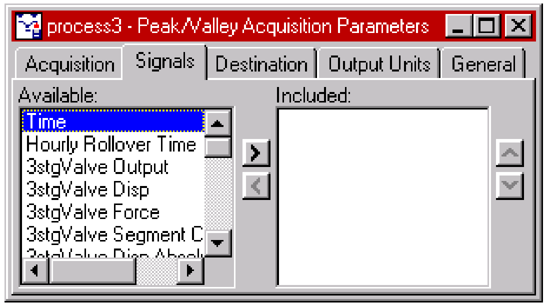
Acquisition tab



Acquisition Tab

ITEM	DESCRIPTION
Master Signal	Specifies the signal that is monitored by the peak/valley detector.
Sensitivity	Defines how much the signal must change before a level is considered a peak or valley.
Total Samples	Specifies the number of samples the process will acquire before it completes. ***Note*** *If disabled, samples will be taken as long as the process is active.*

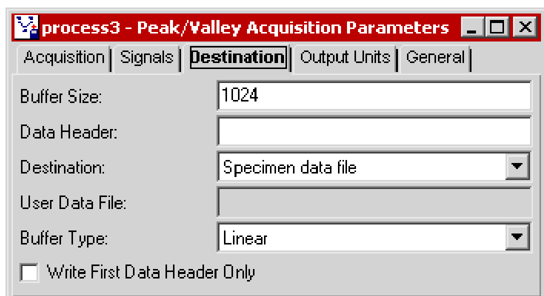
Signals tab



Selects signals from which this process acquires data. Use the left and right arrows to move desired signals from the **Available** list to the **Included** list.

Note *The order of the signals in the* ***Included*** *list determines the order of the columns of data in your data file. Use the up and down arrows to arrange signals in the desired order.*

Destination tab



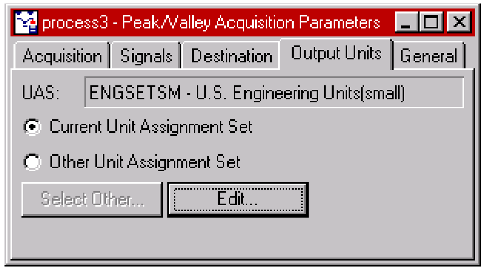
Destination Tab (part 1 of 2)

ITEM	DESCRIPTION
Buffer Size	Specifies the maximum number of data elements the buffer will store before data is written to the disk.
Data Header	Includes a label that describes the data on each buffer that is written to a file.

Destination Tab (part 2 of 2)

ITEM	DESCRIPTION
Destination	Specifies where data is saved: **Specimen data file**—the process writes buffers of data to the MPT specimen.dat file in the Specimen. **User-specified data file**—the process writes buffers of data to a separate data file in the Specimen (*yourname*.dat). **Discard data**—the process will not write data to the Specimen. For more information, see “Data destination” on page 268.
User Data File	Specifies the name of the file where data is saved. This lets you create separate data files in the MPT specimen. This control is enabled only when you have selected User-specified data file in the Destination list above.
Buffer Type	Specifies the buffer type. See “Buffers” on page 266.
Write First Data Header Only	Specifies if the data header is written for each buffer or suppressed after the first header is written. If disabled (unchecked), a data header is written for each buffer. ***Note*** *This feature is not recommended for procedures with multiple data acquisition processes writing data to the same data file.*

Output Units tab



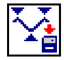
Output Units Tab

ITEM	DESCRIPTION
UAS	Specifies the unit assignment set that will be applied to acquired data.
Select Other	Displays other existing unit assignment sets from which you can choose a UAS. This button is enabled only when you select the **Other Unit Assignment Set** option.
Edit	Opens the **Unit Assignment Set Editor** so you can modify a unit assignment set.

General tab

See "General tab" on page 202.

How to Define a Peak/Valley Acquisition Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in the Procedure table.
3. In the parameters window, click the **Acquisition** tab.
4. In the **Master Signal** list, click the signal you want to monitor with this process.
5. In the **Sensitivity** box, set the units, and then type a value.
6. Optional—If you want to specify the number of samples to be acquired, select the **Total Samples** box. Otherwise it will acquire data as long as the process is active.
7. Click the **Signals** tab.
8. Move all the signals you want to monitor to the **Included** list. Move all the others to the **Available** list.

Note *The order of the signals in the **Included** list determines the order of the columns of data in your data file.*

9. Click the **Destination** tab.
10. Type the **Buffer Size** and type a description of the data file in the **Data Header** box.
11. In the **Destination** list, click the destination for the acquired data. (If you click **User-specified data file**, enter a file name in the **User Data File** box.) See "Data destination" on page 268.
12. In the **Buffer Type** list, click the buffer type.
13. Optional—If desired, enable **Write First Data Header Only** checkbox. This feature aids data sorting and analysis.
14. Optional—Click the **Output Units** tab.

    Change the unit assignment set if you want. Each data acquisition process can use a different unit assignment set.

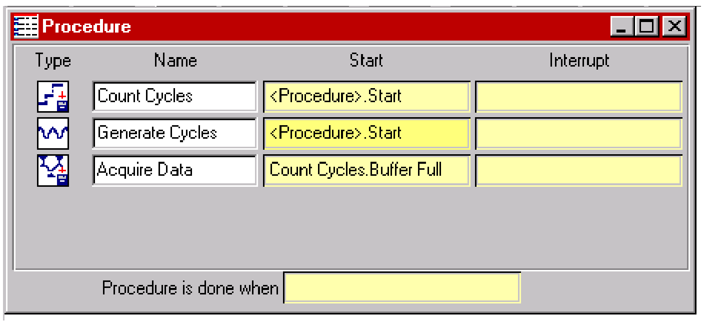
15. Click the **General** tab.
16. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
17. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
18. In the **Execute Process** box, enter the number of times you want the process to execute.
19. Optional—In the **Counter Type** list, click a counter type.
20. Optional—If you enabled a counter, type a **Counter Label**.
21. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

How to Trigger a Process When the Buffer is Full

By using the trigger or buffer full signal from one data acquisition process to start another data acquisition process, you can acquire test data in very selective ways.

Example

Suppose you want to create a procedure that will acquire 10 points of peak/valley data every 100 cycles and store the data in a separate file in the Specimen. To do this, you could create a procedure with a Level Crossing Data Acquisition process, a Cyclic Command process, and a Peak/Valley Data Acquisition process, like this:



In this scenario, to ensure the test ends properly, click **Generate Cycles.Done** in the **Procedure is done when** box, or **Generate Cycles.Done** in the **Count Cycles** process' **Interrupt** box.

Here is how you would set up the processes:

1. Level Crossing Acquisition process (named "Count Cycles"):
   - **Master Signal**: Ch 1 Count
   - **Level Increment**: 1 cycle
   - **Total Samples**: Continuous
   - **Buffer size**: 100
   - **Destination**: Discard data
2. Set up the Cyclic Command process (named "**Generate Cycles**") to generate 1000 cycles of the desired shape and at the desired amplitude and frequency.

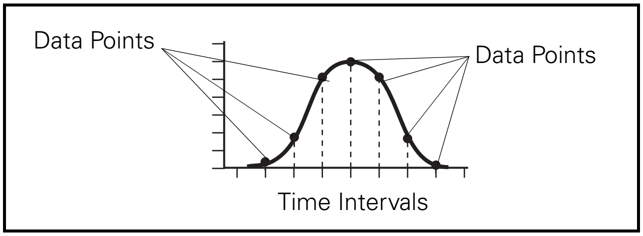
3. Peak/Valley Data Acquisition process (named “**Acquire Data**”):
   - **Master Signal**: As desired
   - **Sensitivity**: Appropriate for Master Signal (for example, 3mm)
   - **Total Samples**: 10
   - **Signals**: As desired
   - **Buffer size**: 10
   - **Destination**: User-specified data file
   - **User Data File**: As desired

When you run this test, the Peak/Valley Data Acquisition process will be triggered by the Count Cycles.Buffer Full signal generated by the Level Crossing Data Acquisition process every 100 cycles, and will acquire 10 points of peak/valley data. The data will be stored to the file (in the Specimen) entered in the **User Data File** box on the **Destination** tab of the **Peak/Valley Data Acquisition Parameters** window.

Timed Acquisition

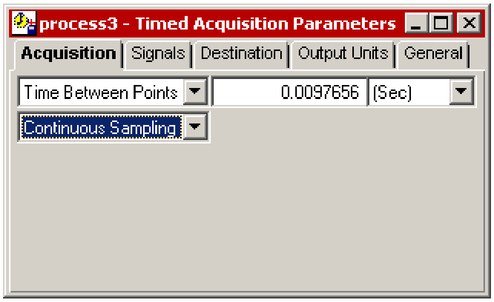
The **Timed Acquisition** process records the output of all selected signals at the specified time interval or sample rate.

When you define a **Timed Acquisition** process, you can define the time between samples (shown below) or enter a sample rate frequency.



Timed Acquisition Parameter Descriptions

Acquisition tab



Acquisition Tab (part 1 of 2)

ITEM	DESCRIPTION
Time Between Points	Specifies the time interval between data acquisitions. ***Note*** *When you enter a time increment, the MPT application may change your entry slightly to reflect your controller’s clock rate.*
Sample Rate	Specifies the sample rate frequency for data acquisition. ***Note*** *MPT may change the entered frequency slightly to reflect the resolution of your controller’s system clock rate. For instance, if you type in “1000 Hz” and press the enter key, MPT may change the number to 1024 Hz.*

Acquisition Tab (part 2 of 2)

ITEM	DESCRIPTION
Total Samples	Specifies the number of samples the process will acquire before it completes.
Sampling Duration	Specifies the length of time the process will attempt to acquire data.
Continuous Sampling	Causes the process to acquire samples continuously until it is interrupted.

Signals tab

See "Signals tab" on page 270.

Destination tab

See "Destination tab" on page 270.

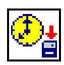
Output Units tab

See "Output Units tab" on page 272.

General tab

See "General tab" on page 202.

How to Define a Timed Acquisition Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in the Procedure table.
3. In the parameters window, click the **Acquisition** tab.
4. Set the timed data acquisition interval and units using **Time Between Points** or enter a **Sample Rate** frequency.

Note *When you enter a time increment, the MPT application may change your entry slightly to reflect your controller's clock rate.*

Note *MPT may change the entered sample rate frequency slightly to reflect the resolution of your controller's system clock rate. For instance, if you type in "1000 Hz" and press the enter key, MPT may change the number to 1024 Hz.*

5. If you want to:

   - Cause the process to acquire samples continuously until it is interrupted, select **Continuous Sampling** (default selection).
   - Specify the number of samples to be acquired, select the **Total Samples**, then enter the number of samples desired.
   - Specify the length of time the process will attempt to acquire data, select **Sampling Duration**, then enter the time over which you wish to acquire data.

6. Click the **Signals** tab.
7. Move all the signals you want to monitor to the **Included** list. Move all the others to the **Available** list.

Note *The order of the signals in the **Included** list determines the order of the columns of data in your data file.*

8. Click the **Destination** tab.
9. Type the **Buffer Size** and type a description of the data file in the **Data Header** box.
10. In the **Destination** list, click the destination for the acquired data. (If you click **User-specified data file**, enter a file name in the **User Data File** box.) See "Data destination" on page 268.
11. In the **Buffer Type** list, click the buffer type.
12. Optional—If desired, enable **Write First Data Header Only** checkbox. This feature aids data sorting and analysis.
13. Optional—Click the **Output Units** tab.

    Change the unit assignment set if you want. Each data acquisition process can use a different unit assignment set.

14. Click the **General** tab.
15. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)

16. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
17. In the **Execute Process** box, enter the number of times you want the process to execute.
18. Optional—In the **Counter Type** list, click a counter type.
19. Optional—If you enabled a counter, type a **Counter Label**.
20. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Max/Min Acquisition

The Max/Min Acquisition process records the maximum and minimum values of selected signals during a test.

You can configure this process to monitor selected signals for their maximum and minimum values. When the maximum and minimum values are detected, they are compared to the maximum and minimum values currently stored for the associated signals. If the new values exceed the stored values, they replace the stored values.

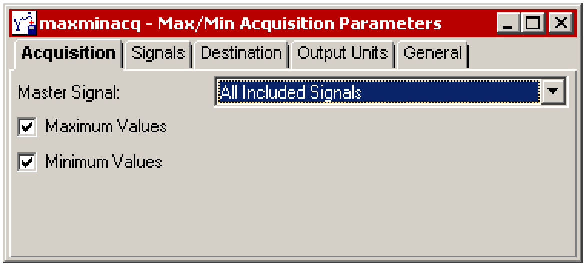
You can also configure this process to monitor a master signal for maximum or minimum values. When the maximum and minimum values are detected, the process samples the values of the other selected signals. When data is written to disk, the master signal values are absolute maximum and minimum values, while the other selected signals reflect their values at the time the master signal's maximum and minimum values were detected.

Keep in mind that this process writes data to disk when:

- The procedure ends normally.
- The procedure is stopped.
- The process is interrupted.

Max/Min Acquisition Parameter Descriptions

Acquisition tab



Acquisition Tab

ITEM	DESCRIPTION
Master Signal	Specifies the signals that are monitored by the max/min detector. When you select **All Included Signals** (default), the process monitors all included signals for their maximum and minimum values (as selected). When detected, the process compares them to the maximum and minimum values currently stored. If the new values exceed the stored values, they take their place. When you select an individual master signal, the process monitors the signal for its maximum and minimum values (as selected). When detected, the process samples the values of the other included signals. When data is written to disk, the master signal values are absolute maximum and minimum values, while the other included signals reflect their values at the time the master signal's maximum and minimum values were detected.
Maximum Values	Causes the process to monitor the master signal selection for maximum values.
Minimum Values	Causes the process to monitor the master signal selection for minimum values.

Signals tab

See "Signals tab" on page 270.

Destination tab

See "Destination tab" on page 270.

Note *Buffer settings, Discard Data options, and Write First Data Header Only options are not available for the Max/Min Acquisition process.*

Output Units tab

See "Output Units tab" on page 272.

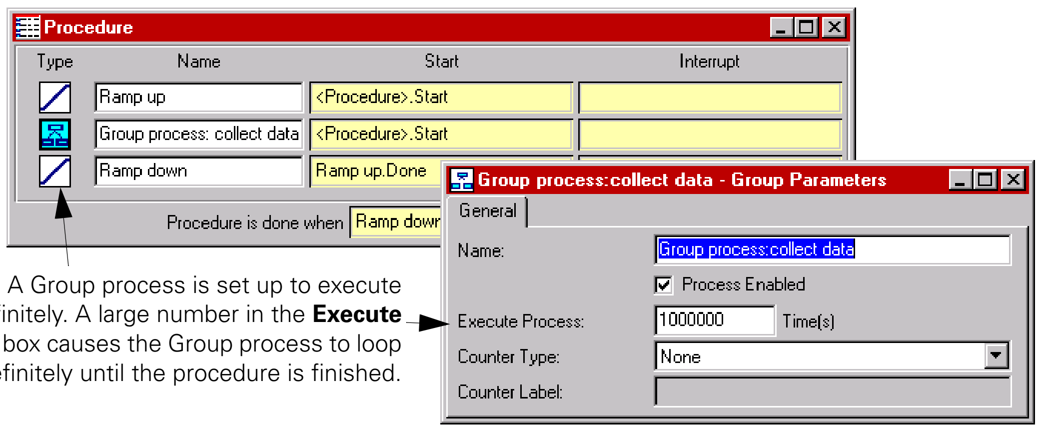
General tab

See "General tab" on page 202.

More About the Max/Min Acquisition Process

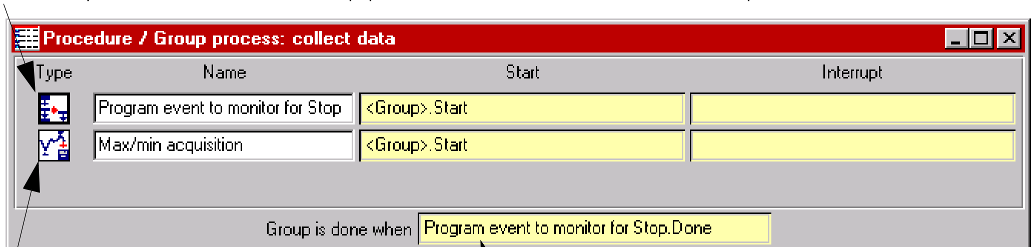
The data recorded by the Max/Min Acquisition process is the minimum and maximum values detected since the process was first activated, no matter how many times the test is stopped. If the test stops (for any reason) before it has run to completion, the data recorded will still be the highest and lowest values detected throughout the entire run portion of the test.

If you want the max/min values for a phase of the test, use a different Max/Min Acquisition process for each phase. Or, you could put a Max/Min Acquisition process in a repeated Group process. Design it so the Group process ends every time a stop occurs, using a Program Event process.



A Group process is set up to execute indefinitely. A large number in the **Execute Process** box causes the Group process to loop indefinitely until the procedure is finished.

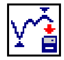
A Program Event process (within the Group process) monitors the test for a Stop event.



A Max/Min Acquisition process (within the Group process) will record max/min values detected from the time you click **Run** until you press **Stop**, or until the test is completed.

The Group process will be done when the Program Event process signals that it has detected a Stop. The Group process will execute indefinitely, recording max/min values whenever a Stop is detected, until the procedure is completed.

How to Define a Max/Min Acquisition Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in the Procedure table.
3. Click the **Acquisition** tab.
4. Optional—Select a **Master Signal**, and then select whether you want to monitor the master signal for **Maximum** or **Minimum** values.
5. Click the **Signals** tab.
6. Move all the signals you want to monitor to the **Included** list. Move all the others to the **Available** list.

Note *The order of the signals in the* ***Included*** *list determines the order of the columns of data in your data file.*

7. Click the **Destination** tab.
8. Type a description of the data file in the **Data Header** box.
9. In the **Destination** list, click the destination for the acquired data. (If you click **User-specified data file**, enter a file name in the **User Data File** box.) See "Data destination" on page 268.
10. Optional—Click the **Output Units** tab.

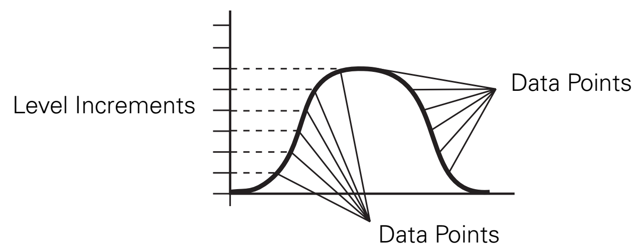
    Change the unit assignment set if you want. Each data acquisition process can use a different unit assignment set.
11. Click the **General** tab.
12. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)

13. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
14. In the **Execute Process** box, enter the number of times you want the process to execute.
15. Optional—In the **Counter Type** list, click a counter type.
16. Optional—If you enabled a counter, type a **Counter Label**.
17. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Level Crossing Acquisition

The **Level Crossing Acquisition** process records data each time the selected master signal changes by a specific amount.

When you define a **Level Crossing Acquisition** process, you must define the level increment at which data is collected.



Data is recorded each time the signal moves by the specified level increment

Level crossing acquisition example

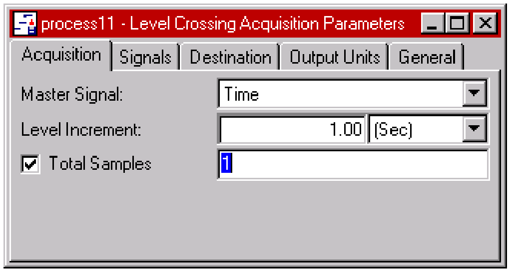
Assume you want to collect data every time the displacement changes 2 millimeters. You would select the displacement feedback signal as your master signal, and set the **Level Increment** to 2 mm.

Now, data will be recorded when the master signal reaches 2 mm, 4 mm, 6 mm, and so on.

Note *You should not set the level increment below the level of any signal noise.*

Level Crossing Acquisition Parameter Descriptions

Acquisition tab



Acquisition Tab

ITEM	DESCRIPTION
Master Signal	Selects a signal that determines when data is acquired.
Level Increment	Defines how much the signal must change before acquiring data.
Total Samples	Specifies the number of samples the process will acquire before it completes. ***Note*** *If disabled, samples will be taken as long as the process is active.*

Signals tab

See “Signals tab” on page 270.

Destination tab

See “Destination tab” on page 270.

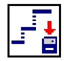
Output Units tab

See “Output Units tab” on page 272.

General tab

See “General tab” on page 202.

How to Define a Level Crossing Acquisition Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in the Procedure table.
3. In the parameters window, click the **Acquisition** tab.
4. In the **Master Signal** list, click the signal you want to monitor with this process.
5. In the **Level Increment** box, set the units of measurement, and then type a value.
6. Optional—If you want to specify the number of samples to be acquired, select the **Total Samples** box. Otherwise it will acquire data as long as the process is active.
7. Click the **Signals** tab.
8. Move all the signals you want to monitor to the **Included** list. Move all the others to the **Available** list.

Note *The order of the signals in the* ***Included*** *list determines the order of the columns of data in your data file.*

9. Click the **Destination** tab.
10. Type the **Buffer Size** and type a description of the data file in the **Data Header** box.
11. In the **Destination** list, click the destination for the acquired data. (If you click **User-specified data file**, enter a file name in the **User Data File** box.) See “Data destination” on page 268.
12. In the **Buffer Type** list, click the buffer type.
13. Optional—If desired, enable **Write First Data Header Only** checkbox. This feature aids data sorting and analysis.
14. Optional—Click the **Output Units** tab.

    Change the unit assignment set if you want. Each data acquisition process can use a different unit assignment set.

15. Click the **General** tab.

16. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)

17. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)

18. In the **Execute Process** box, enter the number of times you want the process to execute.

19. Optional—In the **Counter Type** list, click a counter type.

20. Optional—If you enabled a counter, type a **Counter Label**.

21. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

High Speed Timed Acquisition

The **High Speed Timed Acquisition** process records short bursts of sensor data conditioned by Model 493.21B and Model 493.25 Conditioners.

Unlike other data acquisition processes, this process is not designed to acquire data for the duration of a test, or for even long periods of time within a test. This process is typically used in tests in which it is triggered and then quickly interrupted by other processes, so that it is active only for short periods of time.

When you define a **High Speed Timed Acquisition** process, you must define the time between samples, or the sample rate. This determines how the process acquires data when it is active.

- You can acquire data at periodic intervals.
- Data is recorded in the units specified by the selected unit assignment set.
- You can specify how many samples are taken, and when the samples are acquired.
- If a number of samples is not specified, samples are taken as long as the process is active.

Sample rate and sampling duration limitations

The following tables show the relationship between the number of signals you can select and the maximum potential sample rate at which the process can acquire data.

Keep in mind that the actual sample duration (and number of points acquired) is limited by various elements of the test environment, including processor speed, number of signals being sampled, the number and type of other processes running concurrently, etc. If MPT cannot store data as fast as it is acquired, an acquisition overrun will occur, and the test will stop (data acquired up to that point will be saved).

Note *If an acquisition overrun occurs, the test can be resumed without being reset.*

System Rate: 4096

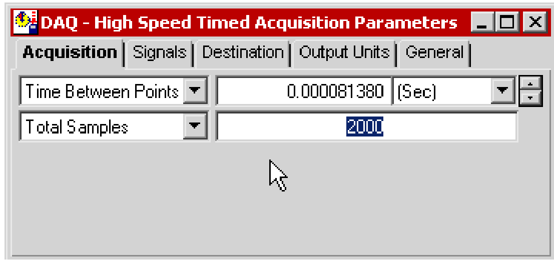
Signal Number	Maximum Rate (Hz)
1 - 2	49152
3 - 5	24576
6 - 8	16384
9 - 11	12288
12 - 17	8192
> 17	4096

System Rate: 6144

Signal Number	Maximum Rate (Hz)
1 - 2	49152
3 - 5	24576
6 - 11	12288
> 11	6144

High Speed Timed Acquisition Parameter Descriptions

Acquisition tab



Acquisition Tab

ITEM	DESCRIPTION
Time Between Points	Specifies the time interval between which the process acquires data.
Sample Rate	Specifies the rate at which the process acquires data. Systems running at a clock rate of 4096 can sample data at rates of 4096, 8192, 12228, 16384, 24576, and 49152 Hz. Systems running at a clock rate of 6144 can sample data at rates of 6144, 12288, 24576, and 49152 Hz. ***Note*** *MPT will change the number you enter to reflect the resolution of your controller's system clock rate. For instance, if you type in "12000" (Hz) and press the* ***Enter*** *key, MPT will change the number to "12228" (Hz).*
Total Samples	Specifies the number of samples the process will acquire before it completes.
Sampling Duration	Specifies the length of time the process will attempt to acquire data.
Continuous Sampling	Causes the process to acquire samples continuously until it is interrupted.

Signals tab

See "Signals tab" on page 270.

Note *Only the "Time" signal and signals conditioned by Models 493.21B and 493.25 Conditioners are available. If Time is selected data is generated, not sampled. The Time signal value always starts at zero.*

Destination tab

See "Destination tab" on page 270.

Note *Discard Data options are not available for the High Speed Acquisition process.*

Note *The default buffer size is 16384, the range is from 1024 - 16384.*

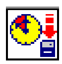
Output Units tab

See "Output Units tab" on page 272.

General tab

See "General tab" on page 202.

How to Define a High Speed Timed Acquisition Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in the Procedure table.
3. In the parameters window, click the **Acquisition** tab.
4. In the **Time Between Points** box, set the units of measurement and enter a value. Or, select **Sample Rate,** and specify the rate at which the system will acquire data.
5. In the **Sample Duration** box, specify the length of time you wish the system to acquire data. Or, select **Total Samples**, and specify the number of samples you wish the system to acquire.
6. Click the **Signals** tab.
7. Move all the signals you want to monitor to the **Included** list. Move all the others to the **Available** list.

Note *The order of the signals in the **Included** list determines the order of the columns of data in your data file.*

8. Click the **Destination** tab.
9. Type the **Buffer Size** and type a description of the data file in the **Data Header** box.

10. In the **Destination** list, click the destination for the acquired data. (If you click **User-specified data file**, enter a file name in the **User Data File** box.) See "Data destination" on page 268.

11. In the **Buffer Type** list, click the buffer type.

12. Optional—If desired, enable **Write First Data Header Only** checkbox. This feature aids data sorting and analysis.

13. Optional—Click the **Output Units** tab.

    Change the unit assignment set if you want. Each data acquisition process can use a different unit assignment set.

14. Click the **General** tab.

15. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)

16. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)

17. In the **Execute Process** box, enter the number of times you want the process to execute.

18. Optional—In the **Counter Type** list, click a counter type.

19. Optional—If you enabled a counter, type a **Counter Label**.

20. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Cyclic Acquisition

The **Cyclic Acquisition** process acquires timed, level crossing, interleaved timed and level crossing, or peak/valley data from cyclic feedback. You can specify the segments or cycles from which the process acquires data arbitrarily, or in linear or logarithmic progressions.

The **Cyclic Acquisition** process monitors sensor feedback that results from cyclic command. This process acquires data from feedback according to one of the following:

- Equal changes in time that you specify (time data)
- Equal changes in amplitude that you specify (level crossing data)
- Both changes in time and amplitude (interleaved timed and level crossing data)
- Signal reversals which occur outside of a sensitivity level that you specify (peak/valley data)

Sample uses

You could use this process to sample the load feedback resulting from a long cyclic fatigue test according to a logarithmic progression, and simultaneously acquire level crossing and timed data from each sampled cycle.

The process saves all data pertaining to an individual cycle in the buffer, and then writes the entire cycle's data to the data file in one block. By default, the process writes data to the specimen.dat file in the MPT specimen. If desired, the process will write data to a user-defined data file within the MPT specimen.

More About the Cyclic Acquisition Process

Disabling the relative count feature

If you clear the **Relative Cycle or Segment Counts** check box, the process will not acquire data on specified cycles that have occurred before the process was active.

For instance, suppose you begin a procedure with a Cyclic Command process that cycles the specimen one hundred times at a small amplitude to precondition the specimen before acquiring data.

Next, you use another Cyclic Command process to apply one million cycles to the specimen at a much larger amplitude, and simultaneously acquire data from the feedback with a cyclic acquisition process.

In this case, the cyclic acquisition process will not trigger until after cycle 100. So if you specified a logarithmic progression of 1, 2, 5, 10, and so on, the cyclic acquisition process would skip all of the cycles performed during preconditioning (1, 2, 5, 10, 20, 50, and 100), and begin acquiring data at cycle 200.

If you had enabled the **Relative Cycle (or Segment) Counts**, the process would trigger at cycle 101 of the test. The process would count cycle 101 of the test as cycle 1 of its logarithmic progression. This is because cycle 101 of the test is the first cycle performed relative to when the cyclic acquisition process is triggered.

Considerations when acquiring cycle clusters

By default, the process stores the data it acquires from each cluster in its buffer, and then writes the content of its buffer to the data file when it is finished acquiring data on the cluster.

On a slow test, the process may take a long time to fill its buffer for a given cluster. As the buffer fills, the data in the buffer will be lost if something interrupts the progress of the test (for example, power failure, crash, and so on) before the buffer fills completely.

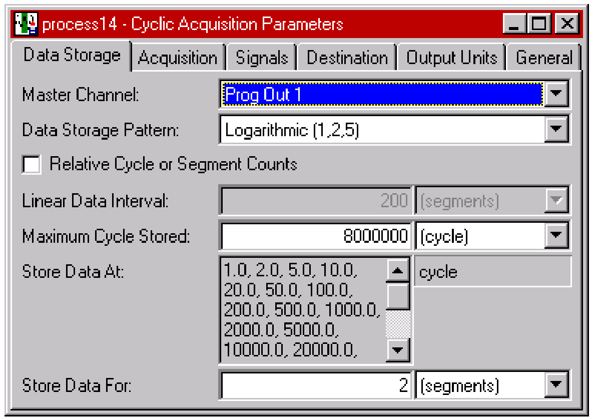
The number of points the process acquires for a given cluster is open-ended and may become quite large, depending on the test frequency and acquisition intervals. For optimal results, some systems may require memory upgrades to accommodate this situation.

When acquiring data, the process fills its buffer by receiving small buffered chunks of data from the controller. Before the process can write this buffered data to its data file, the process buffer must fill completely. Because of timing and size differences between the controller buffer and process buffer, the process may take up to thirty seconds before it writes data to its data file once its buffer is full.

Note *The process will not issue its Process.Buffer Full signal until it has written data to its data file.*

Cyclic Acquisition Parameter Descriptions

Data Storage tab



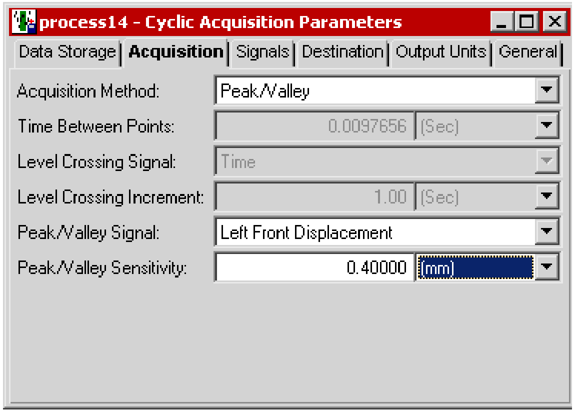
Data Storage Tab (part 1 of 2)

ITEM	DESCRIPTION
Master Channel	Specifies the channel whose segment/cycle counter determines when data is acquired. ***Note*** ***Master Channel*** *is not related to the signal used for level crossing or peak/valley acquisition within the cycle.*
Data Storage Pattern	Specifies the pattern at which data is collected. The selections are: • **Logarithmic (1,2,5)**—Pattern is: 1, 2, 5, 10, 20, 50, 100, 200, 500, 1000, 2000, 5000, and so forth, up to and including the maximum cycle stored. • **Logarithmic (1,2,3,4,5,6,7,8,9)**—Pattern is 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 20, 30, 40, 50, 60, 70, 80, 90, 100, 200, 300, 400, 500, 600, 700, 800, 900, 1000, and so forth, up to and including the maximum cycle stored. • **Linear**—Pattern is: n, 2n, 3n, and so on. You set the n value with the **Linear Data Interval** control. Example: If you choose 200 as the interval, the pattern is 200, 400, 600, and so forth, up to and including the maximum cycle stored. • **User Defined**—Use the cycles you specify in the **Store Data At** list.

Data Storage Tab (part 2 of 2)

ITEM	DESCRIPTION
Relative Cycle or Segment Counts	Determines whether the cycle/segment count starts when this process starts or when the procedure starts. If you want the cycle/segment count to start when this process starts, select **Relative Cycle or Segment Counts**. If you want the cycle/segment count to start when the procedure starts, click to clear **Relative Cycle or Segment Counts**. For more information, see “Disabling the relative count feature” on page 296.
Linear Data Interval	Specifies the multiplier used to generate the data acquisition cycle or segment sequence. ***Note*** *This control is available when you set the **Data Storage Pattern** to **Linear**.* Example: If you choose 100 as the interval, the cycle sequence is 100, 200, 300, and so forth, up to and including the maximum cycle stored.
Maximum Cycle Stored	Specifies the maximum cycle/segment count that data is collected on.
Store Data At	Displays which cycles or segments the process collects data on. ***Note*** *The entire list of cycle numbers may not be displayed.* If the **Data Storage Pattern** is **User Defined**, you must type the cycles or segments you want to collect data on. Segments must be defined as a list of positive integers. Cycles can be defined as a list of positive integers or integers and half cycles (1.5).
Store Data For	Specifies how many cycles or segments the process acquires data for when it detects a cycle in the **Store Data At** list. You can select from 0.5 (one segment) to 1000 cycles. The default value is 1.0 cycle.

Acquisition tab



Acquisition Tab

ITEM	DESCRIPTION
Acquisition Method	Specifies how data is acquired. **Timed**—acquires data at a user-specified time interval. **Level Crossing**—acquires data when the selected Level Crossing signal changes a specific amount. **Interleaved Timed and Level Crossing**—acquires both Timed and Level Crossing data. **Peak/Valley**—acquires data when a peak or valley is detected in the specified Peak/Valley signal.
Time Between Points	Specifies the time interval at which the process acquires data. ***Note*** *MPT may change the number you enter slightly to reflect the resolution of your system clock rate. For instance, if you type in "0.003" (3 milliseconds) and press the* ***Enter*** *key, the number may change to "0.0029."*
Level Crossing Signal	Selects the signal the process monitors for level changes.
Level Crossing Increment	Specifies how much the signal must change before the process acquires data.
Peak/Valley Signal	Selects the signal the process monitors for peak and valley levels.
Peak/Valley Sensitivity	Defines how much the signal must change before a level is considered a peak or valley.

Signals tab

See "Signals tab" on page 270.

Destination tab

See "Destination tab" on page 270.

Note *Buffer settings are not available for the Cyclic Data Acquisition process.*

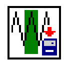
Output Units tab

See "Output Units tab" on page 272.

General tab

See "General tab" on page 202.

How to Define a Cyclic Acquisition Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in the Procedure table.
3. In the parameters window, click the **Data Storage** tab.
4. In the **Master Channel** list, click the channel you want to monitor with this process.
5. Choose a **Data Storage Pattern**, and then type a value in the **Maximum Cycle Stored** box.

   For a linear pattern, type a value in the **Linear Data Interval** box. For a user-defined pattern, enter arbitrary values in the **Store Data At** box.
6. Optional—Select **Relative Cycle or Segment Counts**.

   If you want the process to start acquiring data from cycles relative to when the process starts, select **Relative Cycle or Segment Counts**.

   If you want the process to start acquiring data from cycles relative to the beginning of the procedure, click to clear **Relative Cycle or Segment Counts**.
7. Optional—In the **Store Data For** box, enter the number of cycles you want to collect data for each **Store Data At** entry.
8. Click the **Acquisition** tab.

9. Choose an **Acquisition Method**.

   - For **Timed** data, enter a value in the **Time Between Points** box.
   - For **Level Crossing** data, click a signal in the **Level Crossing Signal** list, and enter a value in the **Level Crossing Increment** box.
   - For **Interleaved Timed and Level Crossing** data, enter a value in the **Timed Between Points** box, click a signal in the **Level Crossing Signal** box, and enter a value in the **Level Crossing Increment** box.
   - For **Peak/Valley** data, click a signal in the **Peak/Valley Signal** list and enter a value in the **Peak/Valley Sensitivity** box.

10. Click the **Signals** tab.

11. Move all the signals you want to record data on to the **Included** list. Move all the others to the **Available** list.

Note *The order of the signals in the **Included** list determines the order of the columns of data in your data file.*

12. Click the **Destination** tab.

13. Type a description of the data file in the **Data Header** box.

14. In the **Destination** list, click the destination for the acquired data. (If you click **User-specified data file**, enter a file name in the **User Data File** box.) See “Data destination” on page 268.

15. Optional—If desired, enable **Write First Data Header Only** checkbox. This feature aids data sorting and analysis.

16. Optional—Click the **Output Units** tab.

    Change the unit assignment set if you want. Each data acquisition process can use a different unit assignment set.

17. Click the **General** tab.
18. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
19. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
20. In the **Execute Process** box, type number of times you want the process to execute.
21. Optional—In the **Counter Type** list, click a counter type.
22. Optional—If you enabled a counter, type a **Counter Label**.
23. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Fatigue

The **Fatigue** process accumulates cycles in a histogram and calculates a damage number (in the form of a statistical value) based on the histogram content.

Note *The MPT **Fatigue** process calculates damage numbers using the same algorithms as the Advanced Editing and Analysis feature of the MTS RPC III™ product.*

Periodically, the histogram and damage numbers are saved to disk, and sent to the Fatigue Monitoring Application via a proprietary data exchange protocol.

With the Fatigue Monitoring application, you can monitor the fatigue data in a real-time, graphical format. For more information, see "Fatigue Monitoring Application" on page 431.

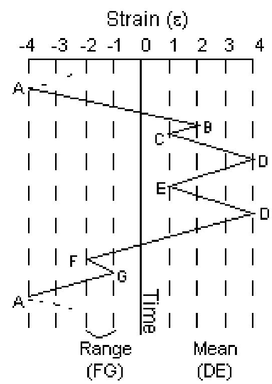
About Rainflow Counting and Histograms

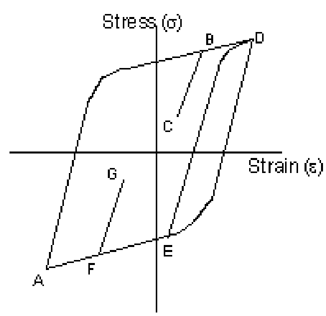
Rainflow counting

Rainflow counting is a method of concisely representing the cyclic content of a signal. It is used to reduce large amounts of data to a more usable form by creating one-pass histograms from drive or response signals. Strain ranges for closed stress-strain hysteresis loops are recorded, and then fatigue life predictions can be made using the strain range and the associated number of closed cycles.

The following example illustrates rainflow counting of a simple strain time history using the Range histogram method. The associated stress-strain response is also shown.

	Cycle Count	
1	2	BC FG
2		
3	1	DE
4		
5		
6		
7		
8	1	AD
9		
10		





Stress-Strain Curve

Histogram types

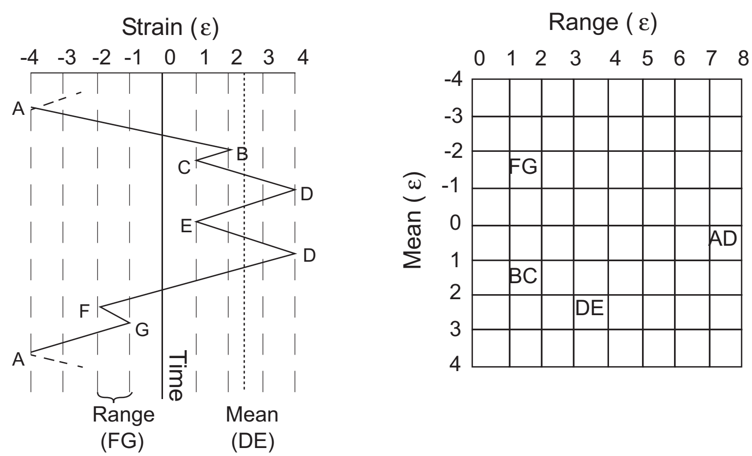
Rainflow-counted histograms provide a means for summarizing the cycles that are identified by rainflow counting (see "Rainflow counting" on page 304). A histogram is created by using data compression to reduce large amounts of data into blocks of information (bins). Each histogram bin contains a cumulative count for data occurrences within a specified bin range.

The **Fatigue** process provides three histogram types: RangeMean, Range, and MaxMin.

A description of each type appears below. For purposes of illustration, the local stress/strain hysteresis response is used to show the methods for creating histograms.

RangeMean histogram

The RangeMean histogram identifies the range and the mean of each cycle identified by the rainflow-counting method. The following example illustrates the method for creating a RangeMean histogram.

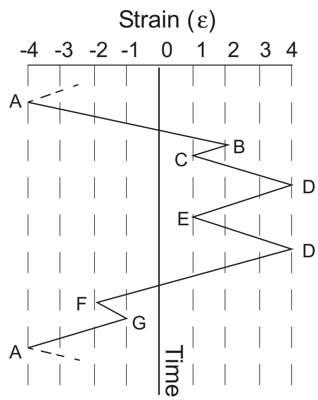


The closed cycles and their associated range and mean are:

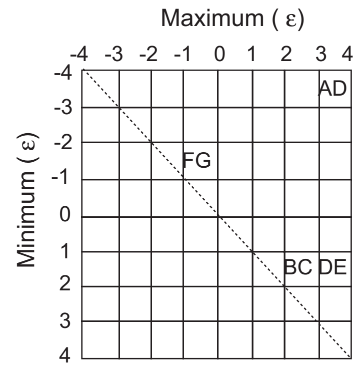
- BC—Range of 1; Mean of 1.5
- DE—Range of 3; Mean of 2.5
- FG—Range of 1; Mean of –1.5
- AD—Range of 8; Mean of 0.0

MinMax histogram

The MinMax histogram identifies the rainflow-counted cycles by the identified maximum/minimum value. The following example illustrates the method for creating a MinMax histogram.



Strain Time History

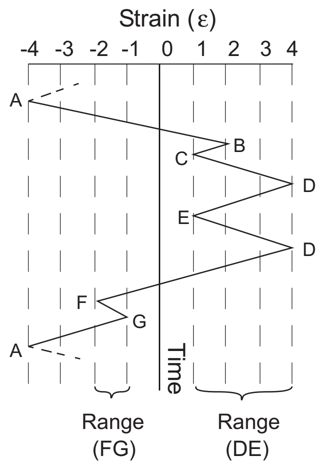


When using the MinMax method, all the histogram counts must fall in the upper-right triangle of the histogram because the maximum value must be greater than the minimum value. The closed cycles and their associated minimum and maximum are:

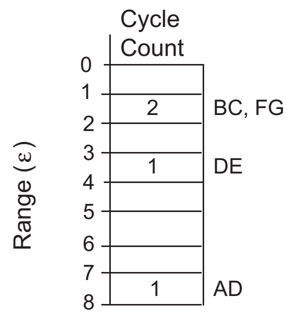
- BC—Minimum of 1; Maximum of 2
- DE—Minimum of 1; Maximum of 4
- FG—Minimum of –2; Maximum of –1
- AD—Minimum of –4; Maximum of 4

Range histogram

This method identifies cycles only by the cycle range. The following example illustrates the method for creating a Range histogram:



Strain Time History



The closed cycles and their associated range values are:

- BC—Range of 1
- DE—Range of 3
- FG—Range of 1
- AD—Range of 8

Bins

Changing the bin size, lower range, upper range, or number will cause the other numbers to change in order to maintain the following relationship:

$$\text{Bin Size} = \frac{(\text{Upper Range} - \text{Lower Range})}{\text{Number of Bins}}$$

Bin size

The bin size specifies the distance between each histogram bin boundary for the selected channel.

Lower range

The lower range specifies the lower boundary of the histogram.

Upper range

The upper range specifies the upper boundary of the histogram.

Number of bins

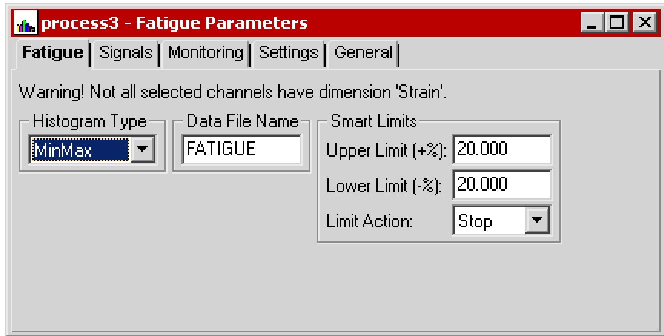
The number of bins specifies the number of bins between the upper and lower range.

For more information

For more information about histograms, rainflow-counting, and bins, see the *RPC III Reference Manual*.

Fatigue Parameter Descriptions

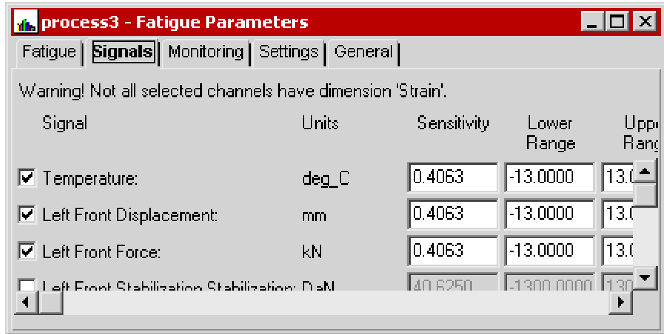
Fatigue tab



Fatigue Tab

ITEM	DESCRIPTION
Histogram Type	Specifies the type of histogram data desired. For a information on each type, see "Histogram types" on page 305.
Data File Name	Names the file where the fatigue data will be stored. The data file is saved in the MPT specimen, with a ".dat" extension.
Smart Limits	Automatically calculates limits based on the first pass of acquired data.
Upper Limit (+%) **Lower Limit (–%)**	Defines the upper and lower limit for the damage number in terms of percentage.
Limit Action	Specifies the action performed when either of the specified limits are exceeded. Options include: **None**—Limit is disabled. **Warning**—A warning message is issued and a message is written in the **MPT Specimen Log**. **Stop**—The process stops and a message is written in the **MPT Specimen Log.** **Trigger**—The process sends a trigger that can be used to start another process. ***Note*** *Since smart limits are established during the first pass of data, limit checking is not in effect until after the first pass of data.*

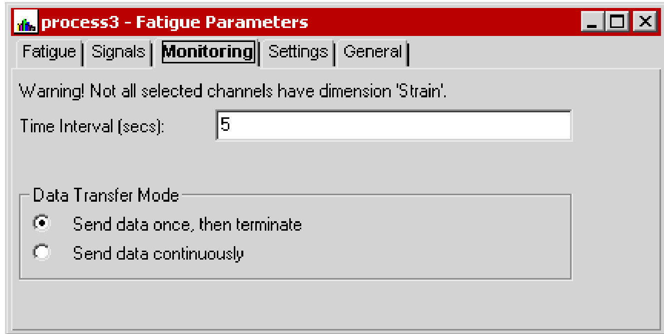
Signals tab



Signals Tab

ITEM	DESCRIPTION
Signals	Displays the signals which can monitored by the Fatigue process.
Units	Displays the signal units.
Sensitivity	Defines the **Sensitivity** for the peak/valley detector. ***Note*** *The units are the same as the full scale units.*
Lower Range	Specifies the lower boundary of the histogram.
Upper Range	Specifies the upper boundary of the histogram.
Bin Size	Specifies the distance between each histogram bin boundary for the selected channel. ***Note*** *The units are the same as the full scale units.*
Number of Bins	Specifies the number of bins between the upper and lower range.
Material	Assigns a material to the current signal.
Exp. Life	Specifies a value for the specimen life expectancy (the number of tests the specimen will survive before breaking). ***Note*** *If you don't know it, leave it at 1.0. This value is used by the Fatigue Monitoring application to display the **Percentage of Lifetime** instead of the damage.*

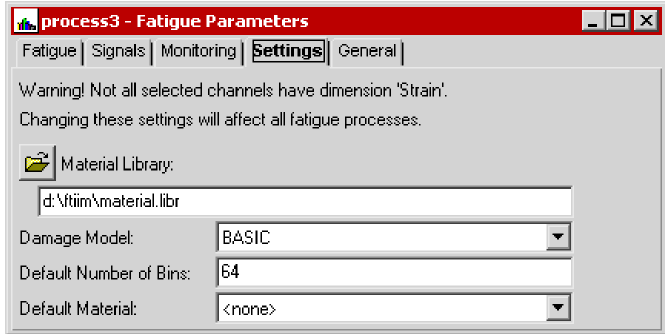
Monitoring tab



Monitoring Tab

ITEM	DESCRIPTION
Time Interval (secs)	Specifies the time interval between data transfers (which occur once every pass). Data is acquired from each included signal (the signals checked in the “Signals tab” on page 310). The acquired data is *peak-picked* and sent to the Rainflow counter (see “Rainflow counting” on page 304). At the end of a pass, the acquired damage numbers are stored to the data file and transmitted to the Fatigue Monitoring application. If the Fatigue Monitoring application is not running, a warning message is written to the **MPT Specimen Log**. ***Note*** *The Fatigue Monitoring application is started automatically if you click **Reset** on the MPT control panel after a new Specimen has been selected.*
Data Transfer Mode	Specifies whether the data is processed once, or processed periodically when a set number of seconds has elapsed. **Send data once, then terminate**—The data will be processed once (useful if you want the statistics monitored only during part of your command signal), then the Fatigue process will terminate. **Send data continuously**—The data will be processed and sent to the Fatigue Monitoring application every *x* seconds until the Fatigue process terminates.

Settings tab



Settings Tab (part 1 of 2)

ITEM	DESCRIPTION
Material Library	Specifies a materials library file, which defines the materials available to each channel. The default is the RPC III materials library supplied with the **Fatigue** process.
Damage Model	Selects the damage model you will use for all fatigue processes. RPC III provides both strain life and generic stress life damage models. The Local Strain-life Approach (LSA) damage model selections include the following: • **BASIC**—strain-life approach to fatigue analysis (zero mean stress). • **MORROW**—strain-life approach to fatigue analysis (mean stress correction applied). • **SWT** (Smith-Watson-Topper)—strain-life approach to fatigue analysis (mean stress correction applied). The Generic Stress Life (GSL) models include: • **GSL**—generic stress life approach, no mean correction • **GSL_MEAN**—generic stress life approach, mean correction ***Note*** *The selected damage model may influence the available materials. Only materials that contain valid data for the selected damage model will appear in the list shown in the* ***Materials*** *tab.*
Default Number of Bins	Specifies the number of bins you will use for all new fatigue processes (the default is 64). ***Note*** *If the default number of bins is changed, the value is also changed on the* ***Signals*** *tab for all corresponding signals.*

Settings Tab (part 2 of 2)

ITEM	DESCRIPTION
Default Material	Specifies the material that will be the default for all new fatigue processes that are created. ***Note*** *If the default material is changed, the value is also changed on the* ***Signals*** *tab for all corresponding signals.*

General tab

See "General tab" on page 202.

Setting Up a Test to Acquire Fatigue Data

To acquire fatigue data during your test, you should define a **Group** process that includes the **Fatigue** process and your command process.

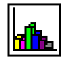
You should sequence your test so that:

- In the group process, the **Fatigue** process starts when the command process starts.
- When the command process ends, the **Fatigue** process is terminated.
- When the **Fatigue** process ends, the **Group** process ends.

In this setup, each execution of the **Group** process yields one point of data (which is not enough to create a fatigue plot). To acquire enough data points for your plot, you can set the number of times the **Group** process executes equal to the number of passes you desire (typically greater than one hundred).

Note *You can use the Trend process in parallel with the Fatigue process to acquire trend data.*

How to Define a Fatigue Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the **Fatigue** process icon.
3. In the parameters window, click the **Fatigue** tab.
4. Choose a **Histogram Type**, and enter a **Data File Name**.

   Note *If you have multiple **Fatigue** processes within one procedure in sequence, be sure to enter a unique file name for each process. Otherwise, all data will be written to the same data file.*

5. Under **Smart Limits**, enter **Upper Limit** and **Lower Limit** values, and then set the **Limit Action**.
6. Click the **Settings** tab and specify the **Materials Library**, **Damage Model**, **Default Number of Bins**, and **Default Material**.
7. Click the **Signals** tab, and select the box next to each signal you want to acquire fatigue data from.
8. For the selected signals, enter the desired values for the **Sensitivity**, **Lower Range**, **Upper Range**, **Bin Size**, **Number of Bins**, **Material**, and **Exp. Life** (life expectancy).
9. Click the **Monitoring** tab, type a **Time Interval**, and then select the **Data Transfer Mode**.
10. Click the **General** tab.
11. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
12. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
13. In the **Execute Process** box, type the number of times you want the process to execute.
14. Optional—In the **Counter Type** list, click a counter type.
15. Optional—If you enabled a counter, type a **Counter Label**.
16. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Important tips

If your procedure will run for a long time, make sure the Windows virtual memory paging file is large enough (MTS recommends 250 MB). To access the Windows paging file: Start > Settings > Control Panels > System.

In the **Fatigue Process Parameters** window, make sure the **Fatigue** process is enabled and the **Execute Process** value is set to one (on the **General** tab).

Because the **Fatigue** process parallels a command process, it will run as long as the command process is executing. However, it is possible to specify the Fatigue process' run time within the Fatigue process:

1. Click the **Monitoring** tab.
2. Specify a **Time Interval**.
3. Select **Send data once, then terminate**.

Note *If you choose **Send data continuously** on the **Monitoring** tab, the **Fatigue** process will not stop on its own, and therefore, will not repeat according to the **Execute Process** value you enter on the **General** tab.*

Time History Input

The **Time History Input** process acquires and digitizes multiple channels of analog test response data in the RPC III file format.

About Standard Data Acquisition Rates

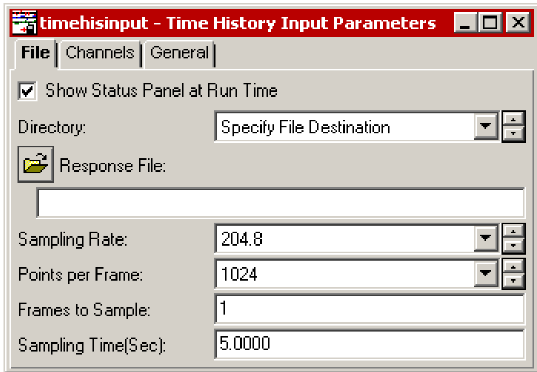
Data acquisition rates are sub-multiples of the high-clock rate; that is, they equal the high-clock rate divided by certain integers (4, 8, 10, 16, 20, 32, and 40) as follows:

- 1024 Hz
- 512 Hz
- 409.6 Hz
- 256 Hz
- 204.8 Hz
- 128 Hz
- 102.4 Hz

The **Time History Input** process filters the analog test data you acquire through an analog 5-pole Bessel filter with a cutoff of 300 Hz. After being digitized, the signals are run through a digital filter whose cutoff is set to 40% of the requested sampling frequency and then down-sampled to the requested sampling rate. These digital filters match the ones used in RPC III 498 DSPAD boards so data can be used interchangeably with that system.

Time History Input Parameter Descriptions

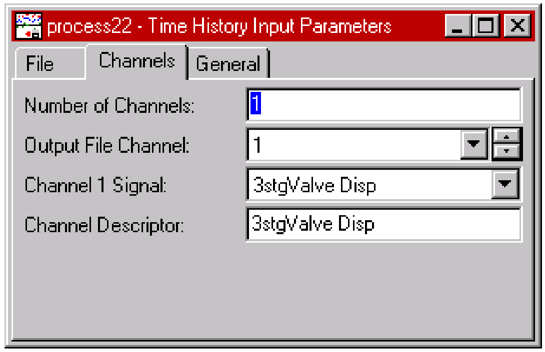
File tab



File Tab

Item	Description
Show Status Panel at Run Time	Displays the **Time History Input Status** panel during the test. For more information, see page 247.
Directory	Specifies the directory in which the response file (.rsp) will be saved. The **Specify File Destination** selection allows you to specify a path for the response file. The **Use Specimen Directory** selection saves the response file in the specimen directory.
Response File *(file icon)*	The process stores digitized response test data in a response file (.rsp). If the **Directory** selection is **Specify File Destination**, the **Response File** file icon will be active. Click the file icon to open the **Select response file** window. Use the window to select a current response file, save a new response file, and specify the path.
Sampling Rate	Specifies the rate at which the controller samples response feedback.
Points per Frame	Specifies the number of data points the controller samples per subdivision of the digitized data.
Frames to Sample	Specifies the number of subdivisions of the digitized data in which the controller samples.
Sampling Time (Sec)	Specifies the total time (in seconds) during which the controller samples data from response feedback.

Channels tab



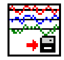
Channels Tab

ITEM	DESCRIPTION
Number of Channels	Selects the number of channels on which data is acquired and formatted.
Output File Channel	Switches between **Output File Channels**.
Station Signal	Defines the station signal associated with the selected output file channel.
Channel Descriptor	Specifies the **Output File Channel** descriptor used in the response file header. ***Note*** *The default name is the **Station Signal** name.*

General tab

See "General tab" on page 202.

How to Define a Time History Input Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in the Procedure table.
3. In the parameters window, click the **Channels** tab.
4. In the **Number of Channels** list, type the number of channels you want to take data on and press **Enter**.
5. In the **Output File** list, click the first channel.
6. In the **Channel 1 Signal** list, click the signal to be monitored. (You can enter a descriptor in the **Channel Descriptor** box.)
7. Perform steps 5 - 6 for any additional channels.
8. Click the **File** tab.
9. Optional—If desired, enable the **Show Status Panel at Run Time** control.
10. Use the **Directory** control to specify the directory in which the response file (.rsp) will be saved. The **Specify File Destination** selection allows you to specify a path for the response file. The **Use Specimen Directory** selection saves the response file in the specimen directory.
11. If you have selected **Specify File Destination** in the previous step, you can click the **Response File** file icon to open the **Select response file** window. Use this window to select the file and directory in which you wish to save response data.
12. In the **Sampling Rate** list, click the desired sampling rate.
13. In the **Points per Frame** list, click the desired number of data points the controller samples per subdivision of the digitized data.
14. In the **Frames to Sample** box, type the desired number of subdivisions of the digitized data in which the controller samples.
15. In the **Sampling Time (Sec)** list, type the total time (in seconds) during which the controller samples data from response feedback.

16. Click the **General** tab.
17. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
18. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
19. In the **Execute Process** box, type number of times you want the process to execute.
20. Optional—In the **Counter Type** list, click a counter type.
21. Optional—If you enabled a counter, type a **Counter Label**.
22. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Trend

The **Trend** process collects signal data and calculates five statistical trend values: Minimum, Maximum, Mean, RMS, and Standard Deviation.

Periodically, the trend values are saved to disk, and sent to the Trend Monitoring Application via a proprietary data exchange protocol.

With the Trend Monitoring application, you can monitor the trend data in a real-time, graphical format. For more information about trend monitoring, see “Trend Monitoring Application” on page 440.

About Data Handling

Data is acquired for each included signal (the channels with a check mark next to their names). At the end of a pass, the acquired statistical values are stored to the data file and transmitted to the Trend Monitoring application. If the Trend Monitoring application is not running, a warning message will be written to the **MPT Specimen Log**.

Limits that are changed by the Trend Monitoring application during the procedure will be transferred back to the Trend process and used for the limit check.

Note *The Trend Monitoring application is started automatically if you click* ***Reset*** *on the MPT control panel after a new Specimen has been selected.*

Smart Limits are calculated after the first complete pass. This means that limit checking is effectively disabled until after the first pass has run to completion. If you need limit checking during the first pass as well, use the **Trend** tab to enter absolute limit values and adjust the limits as soon as the Trend Monitoring application shows the data for the first pass.

If a smart limit is based on the data value, and the data value is too small (<0.01% of range), limits based on full scale will be used instead.

Example

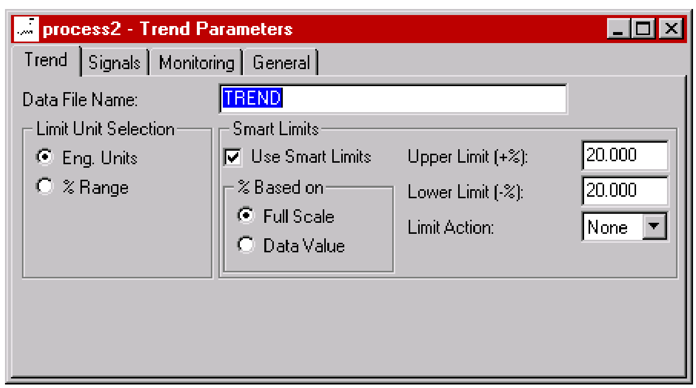
You could monitor strain levels in a critical component of the physical specimen. You could also monitor shock absorber temperatures to ensure that the shock absorbers do not overheat during a drive test. You can monitor statistical trends to determine if the response signal matches your expectations.

The data on the selected signal channels are acquired independently on each channel, and five statistical values (Minimum, Maximum, Mean, RMS, Standard Deviation) are calculated. Depending on the settings in the Trend process, the data is stored and processed every x seconds, or when the process' end terminal triggers; that is, when a Group process is repeated. The data is then stored to disk in ASCII text format as tab-separated values and can be read directly by spreadsheet programs such as Microsoft™ Excel®. The data file is stored in the Specimen directory.

The data is also sent to the Trend Monitoring application (via a proprietary data exchange protocol) which displays the statistics on-line and allows for limit setting during the procedure; limit changes are transmitted back to MPT which does the actual limits checking.

Trend Parameter Descriptions

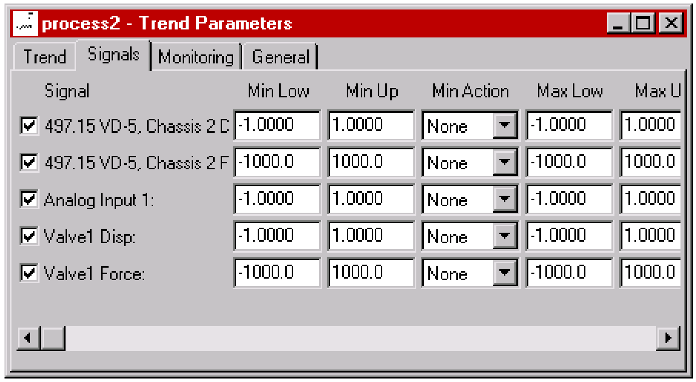
Trend tab



Trend Tab

Item	Description
Data File Name	Names the file where the trend data is stored. ***Note*** *Trend files are stored in your MPT specimen with a ".dat" extension.*
Limit Unit Selection	Specifies the units for the trend data limits.
Use Smart Limits	Automatically calculates limits during the procedure based on the first pass of acquired data. ***Note*** *If you select **Use Smart Limits**, the **Signals Limits** boxes (on the **Signals** tab) are disabled. Limits that are changed during the test by the Trend Monitoring application are transferred back to the MPT Trend process.*
% Based on	Defines the reference of the data value on which the limit % is based. **Full Scale**—Uses the full scale value as the reference value. **Data Value**—Uses the first-pass value as the reference value.
Upper Limit (+%) **Lower Limit (-%)**	Defines the lower and upper limit for each statistical value.
Limit Action	Specifies the action performed when either of the specified limits are exceeded. Options include: **None**—Limit is disabled. **Warning**—A warning message is issued and a message is written in the MPT message log. **Stop**—The process stops and a message is written in the **MPT Specimen Log**. **Trigger**—The process sends a trigger that can be used to start another process.

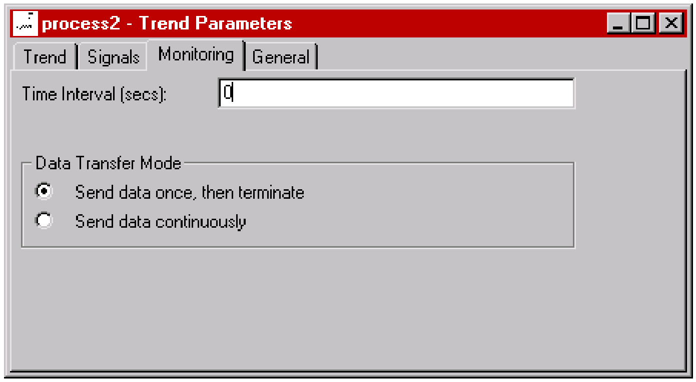
Signals tab



Signals Tab

ITEM	DESCRIPTION
Signal Names	Displays the available channels which can be monitored by the Trend process. Select the box next to the signal name if you want the Trend process to monitor it, and then enter the upper and lower limits for the five statistical values. ***Note*** *The upper limit, lower limit, and limit action boxes are disabled if you selected **Use Smart Limits** on the Trend tab.*
Min Low, Min Up, Min Action **Max Low, Max Up, Max Action**	For each statistical value, this table defines: • The minimum lower and upper limits, and associated actions • The maximum lower and upper limits, and associated actions ***Note*** *Note that the values are interpreted according to the **Limit Unit Selection** on the **Trend** tab.*

Monitoring tab



Monitoring Tab

ITEM	DESCRIPTION
Time Interval (secs)	Specifies the time interval between data transfers (which happens once every pass). First, choose a suitable time interval (for example, 120 seconds; thus, a pass would be two minutes in length).
Data Transfer Mode	Specifies whether the data is saved and sent once, or saved and sent each time the specified **Time Interval** elapses. **Send data once, then terminate**—The data will be saved and sent once at the end of the first **Time Interval**. (This setting is useful if you want the monitor statistics only for a part of the test.) **Send data continuously**—The data will be saved and sent continuously in the **Time Interval** specified until the Trend process is done or interrupted.

General tab

See “General tab” on page 202.

Setting Up a Test to Acquire Trend Data

To acquire trend data during your test, you should define a **Group** process that includes the **Trend** process and your command process.

Note *Your command process is typically a* ***Profile*** *or* ***Time History Output*** *process.*

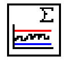
Typical trend tests are sequenced so that:

- When the command process starts, the **Trend** process starts.
- When the command process ends, the **Trend** process is terminated.
- When the **Trend** process ends, the **Group** process ends.

One pass of this **Group** process will yield only one point of data for each statistical value, which is not enough to create a trend plot. To acquire more points, you must set the number of times the **Group** process executes equal to the number of command process passes you desire (this is typically greater than one hundred).

Note *If desired, you can use the Fatigue process in parallel with the Trend process to acquire fatigue data.*

How to Define a Trend Process

1. Drag the  icon from the **Process Palette** to your Procedure or Group table.
2. Double-click the process icon in the Procedure or Group table.
3. Click the **Trend** tab. Enter a **Data File Name**, select the **Limits Units** and, if you want, enable the **Smart Limits**.
4. Click the **Signals** tab.
5. Select all the signals that you want to acquire data from by selecting the check box to the left of the signal name.
6. For each selected signal, and for each statistical value, enter the desired values for the upper and lower limit and the limit action. (This step applies only if **Smart Limits** are disabled.)
7. Click the **Monitoring** tab and select a **Time Interval** (typically set to **0**) and **Data Transfer Mode**.
8. Click the **General** tab.
9. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
10. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
11. In the **Execute Process** box, type number of times you want the process to execute.
12. Optional—In the **Counter Type** list, click a counter type.
13. Optional—If you enabled a counter, type a **Counter Label**.
14. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Important hints

- If your procedure will run for a long time, make sure the Windows virtual memory paging file is large enough (MTS recommends 250 MB). To access the Windows paging file controls: Start > Settings > Control Panels > System
- On the **General** tab, make sure the **Trend** process is enabled and the **Execute Process** value is set to one. Because the **Trend** process parallels a command process, it will run as long as the command process is executing. However, it is possible to specify the Trend process' run time within the Trend process:

  Click the **Monitoring** tab.

  Specify a **Time Interval**.

  Select **Send data once, then terminate**.

  Note *If you choose **Send data continuously** on the **Monitoring** tab, the **Trend** process will not stop on its own, and therefore, will not repeat according to the **Execute Process** value you enter on the **General** tab.*
- If you change limits during the procedure (using the Trend Monitoring application) the new limits are transferred to MPT and are displayed in the **Signals** tab in the MPT **Trend Parameters** window. Thus, if you store the procedure after running, these settings will be saved as well.

More About Smart Limits

- If you press **Reset** and start the procedure over, **Smart Limits** will *not* be calculated again.
- If you change the Specimen, **Smart Limits** will be calculated again (even if you load a previously run procedure).
- As mentioned above, if you change limits during the procedure, the new limits will be stored with the procedure if you decide to save the procedure after a run. However, if **Smart Limits** are enabled when you save, they will be calculated again at run time, thus effectively disabling the previously saved limits. (You may decide you want to disable **Smart Limits** before running the procedure.

Example

Suppose you want to run a sine wave for 20,000 cycles and you want trend to be calculated every 1,000 cycles.

To achieve this, use a Group process with a repeat count of 20. (If you want to repeat a Trend process, you must put it into a Group process paralleled by a command process, or use the Timed Mode.

Inside the Group process, you will have the Cyclic command process (in this case, a sine wave running for 1,000 cycles) and the Trend process running in parallel. The Start signal for both processes should be Group.Start. The Group is Done signal should be Trend.Done (assuming the name of your Trend process is "Trend"). The Interrupt signal for the Trend process should be Cyclic.Done.

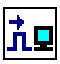
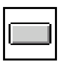
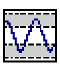
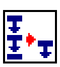
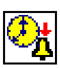
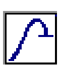
More About RMS, Standard Deviation, and Asymmetric Full Scales

RMS and Standard Deviation are always positive numbers (greater than or equal to zero). Given a range of –10 mm to –5 mm, RMS then would be in the range of 5 through 10, and Standard Deviation would be 0 through 10. This is important to remember when entering limit values, either as engineering units or as percentages. The **Smart Limits** controls in the **Trend** tab takes these physical relationships into account and checks user input accordingly.

Another idiosyncrasy occurs when working with temperature channels, and using Celsius within MPT and Fahrenheit in the output unit assignment set. In this case the lower limit for Standard Deviation defaults to 0° C; however, this equals 32° F. Because the Standard Deviation for slowly changing signals (such as temperature) is close to zero, in this scenario the lower limit for Standard Deviation (32° F) would very likely be tripped after the first pass. The best way to avoid this situation is to use the same unit assignment set within MPT and as output unit assignment set.

Event Processes

You use event processes to monitor test signals and issue event signals when conditions you specify are met.

Icon	Process Name	Description
	**Data Limit Detector**	Monitors one or more signals for sensor limits you specify. (See page 332.)
	**Digital Input Detector**	Detects digital transition states from external devices. (See page 340.)
	**Operator Event**	Enables operator interaction during the test. (For example, Operator events can display buttons the operator must click to continue the test). (See page 348.)
	**Peak/Valley Change Detector**	Detects changes in the peak and valley amplitudes of cyclic waveforms. (See page 354.)
	**Program Event**	Triggers other processes based on user-defined changes in the test state. (See page 364.)
	**Periodic Time Event**	Triggers other processes based on user-defined time events. (See page 367.)
	**Failure Detector**	Triggers other processes based on user-defined changes in the physical specimen's response. (See page 373.)

Data Limit Detector

The **Data Limit Detector** process monitors signals for the limits you specify. When any (or all) of the monitored signals exceed their limits, the process ends and generates a done signal. The done signal from this process is often used to trigger other processes.

You can configure the **Data Limit Detector** process to respond to a specific segment count, amount of time, or sensor signal value.

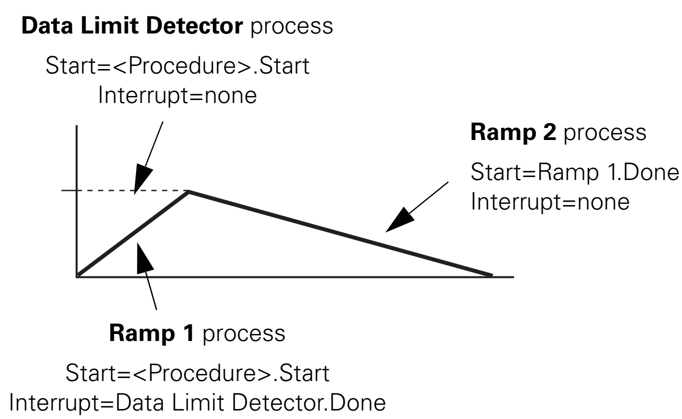
When you define a **Data Limit Detector** process, you select the signals you wish to monitor, set limits on those signals, choose how detector messages are logged, and choose what action, if any, you want to invoke when the defined limits are detected.

Example

Assume we want our program to:

- Ramp up to end level 1 at a specified rate (Ramp 1).
- Ramp down to end level 2 at a different rate once the output reaches end level 1 (Ramp 2).

We can use a **Data Limit Detector** process to trigger the ramp down process once our end level 1 is reached.

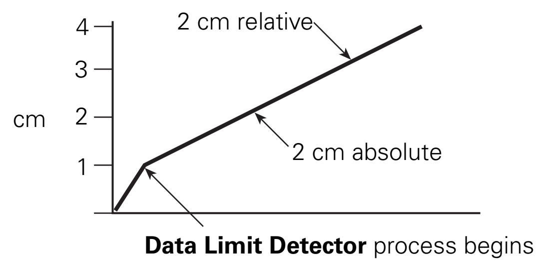


About Limit Modes

This process supports relative limits (starting when the process begins) and absolute limits (from a zero reference).

Example

Assume the test command ramps from 1 cm to 4 cm. An absolute limit value of 2 cm ends the process at 2 cm. A relative limit ends the process at 3 cm (2 cm relative to 1 cm).



Using actions with processes

The Station Manager includes an **Event-Action Editor** that lets you create user defined actions that you can link to test events.

The user defined actions that you create with the Station Manager, in addition to some built-in actions (Interlock, Station Power Off, Program Interlock, Program Stop, and Program Hold) are now available in the **Action** list boxes in the Data Limit Detector, Digital Input Detector, and Program Control processes.

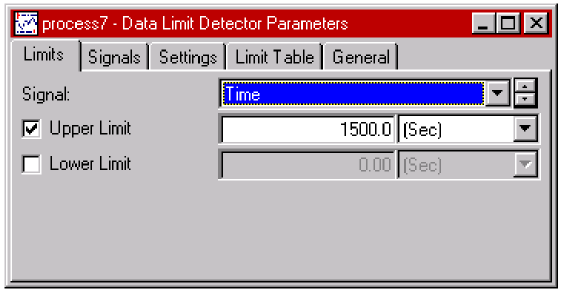
Creating actions when MPT is running

To define actions with the Station Manager **Event-Action Editor** (in the **Tools** menu), you must have the **Configuration** access level selected. To select the **Configuration** access level, you must exit MultiPurpose TestWare, and any other applications that may be running, such a Basic TestWare. Once you have exited the applications, you can select the **Configuration** access level and define actions as desired with the **Event-Action Editor**. Then, before you relaunch MultiPurpose TestWare, you must select any other access level other than **Configuration**.

The user defined actions you have defined will appear in the **Action** list boxes in the Data Limit Detector, Digital Input Detector, and Program Control processes.

Data Limit Detector Parameter Descriptions

Limits tab



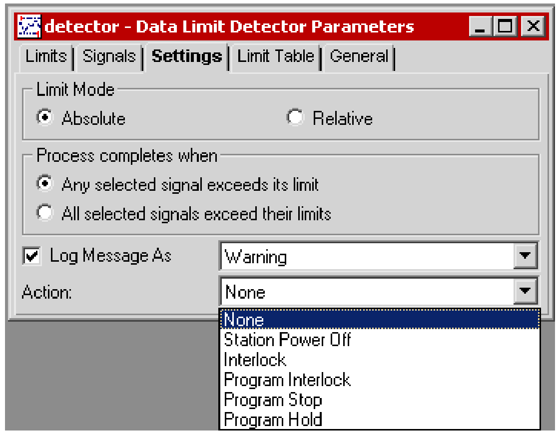
Limits Tab

ITEM	DESCRIPTION
Signal	Displays the signals that may be monitored for limit detection. Only signals moved to the **Included** column on the **Signals** tab appear in this list.
Upper/Lower Limit	Enables the upper and lower limit controls, which allow you to specify associated limit detector values. ***Note*** *If you want to temporarily disable a defined limit (for example, for a test run), uncheck the associated checkbox.*

Signals tab

Selects the signals that this process monitors. Use the left and right arrows to move desired signals to the **Included** list.

Settings tab



Settings Tab (part 1 of 3)

ITEM	DESCRIPTION
Limit Mode	Specifies whether the process limit values are absolute or relative to the signal value at the start of the process. See “About Limit Modes” on page 333.
Process completes when	Select **Any selected signal exceeds its limit** to end the process when any signal exceeds its limit. Select **All selected signals exceed their limits** to end the process once all signals have exceeded their limits at some time while during the process.

Settings Tab (part 2 of 3)

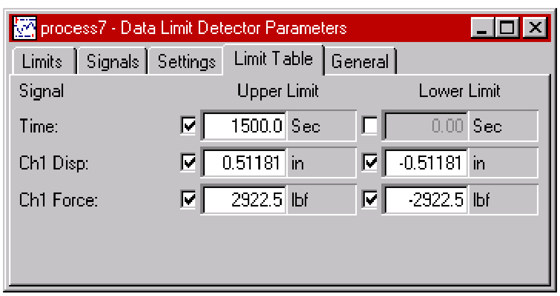
ITEM	DESCRIPTION
Log Message As	Assigns a severity level to the message (in ascending levels of severity: **Diagnostic**, **Information**, **Warning**, or **Error**) and makes the message available to the **MPT Specimen Log**. If the Station Manager **Station Log** and the Basic TestWare **Test Log** are configured to accept messages from all sources, then MPT makes the message available to them also. ***Note*** *Unlike Station Manager and Basic TestWare, MPT does not generate messages with a severity level of **Fatal Error**.* Each type of log will accept the message only if it has been configured to store messages with the assigned severity level. For instance, if the **MPT Specimen Log** is configured to accept messages with a minimum severity level of **Warning**, and you assign **Information** as the severity level for this message, then this message will not be recorded in the **MPT Specimen Log**. In that instance the **MPT Specimen Log** would accept messages having only **Warning** and **Error** severity levels. You can specify the minimum severity level that each type of log will accept. To do this: • For MultiPurpose TestWare, select the **MPT Options Editor**, then select the **Specimen Log** tab. • For Station Manager, select **Tools**, then select **Station Options**, then select the **Station Log** tab. • For Basic TestWare, select **Test Setup**, then select the **Test Log** tab.

Process Descriptions

Settings Tab (part 3 of 3)

ITEM	DESCRIPTION
Action	Specifies the action that occurs when the process completes.
None	The process ends without generating a message or performing an action.
Station Power Off	The process removes hydraulic power from the station. The test can be resumed after you reapply hydraulic power to the station.
Interlock	The process triggers a hydraulic interlock and shuts down the hydraulic power supply. The test can be resumed after you reset the interlock, reactivate hydraulic power, and apply hydraulic power to the station.
Program Interlock	The process triggers a program interlock. The test can be resumed after you reset the interlock.
Program Stop	The process stops the test. This action functions the same as clicking the **Stop** button on the **Station Controls** panel.
Program Hold	The process suspends the test until you click the **Run** button on the MPT control panel. This action functions the same as clicking the **Hold** button on the **Station Controls** panel.

Limit Table tab

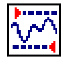


The **Limit Table** tab displays all the limits defined in this process. You can edit any data limit value in this window. To enable a limit, select the check box next to the signal name. To disable a limit, clear the check box next to the signal name.

General tab

See “General tab” on page 202.

How to Define a Data Limit Detector Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Signals** tab.
4. Move all the signals you want to monitor to the **Included** list. Move all the others to the **Available** list.
5. Click the **Limits** tab.
6. For each signal you want to monitor:
    A. In the **Signal** list, click the desired signal.
    B. Set the units of measurement and enter a value for the **Upper Limit** and/or a **Lower Limit**.

    Note *If you want to disable an upper or lower limit, clear the **Upper Limit/Lower Limit** check box.*
7. Click the **Settings** tab.
8. Under **Limit Mode**, set the limit mode.
9. Under **Process completes when**, set when the process ends.
10. Optional—Enable (check) the **Log Message As** control and select the type of message you wish to generate.
11. Optional—Select an action.
12. Click the **Limit Table** tab.
13. Verify and, if needed, edit limit values on this tab.

14. Click the **General** tab.
15. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
16. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
17. Optional—In the **Counter Type** list, click a counter type.
18. Optional—If you enabled a counter, type a **Counter Label**.
19. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Digital Input Detector

The **Digital Input Detector** process monitors digital input signals. When any (or all) of the monitored signals are received, the process ends and generates a done signal. The done signal from this process is used to trigger other processes.

When you define a **Digital Input Detector** process, you define the digital inputs you want to monitor, the required signal state, the trigger mode, how messages are logged, and what action, if any, you want the process to perform when it receives a monitored signal.

More About the Digital Input Detector Process

You must know what is connected to each digital input and what type of input is expected. See the Cabling chapter in the *Controller Installation and Calibration* manual for information about digital input connections.

Setting signal options

With the **Options** list (on the **Input Table** tab), you can specify what input signal state is required to trigger the detector process.

The following states are supported:

ITEM	DESCRIPTION
None	Disables the input.
High/Low	Detects transitions from a logic high state to a logic low state.
Low/High	Detects transitions from a logic low state to a logic high state.
Either	Detects any transition from high-to-low or low-to-high.
Channel Low	Detects the logic low state.
Channel High	Detects the logic high state.

Design considerations

If the signal is high when the process begins:

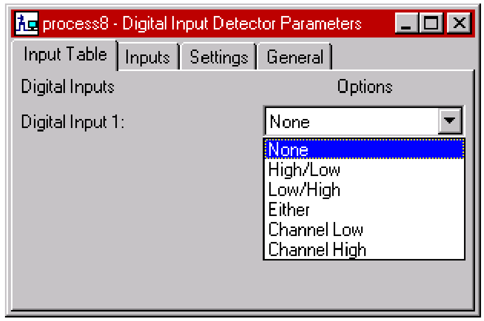
- The **High/Low** selection will cause the process to trigger when the signal becomes low.
- The **Low/High** selection will cause the process to trigger when the signal becomes low, then transitions to high.
- The **Either** selection will cause the process to trigger when the signal becomes low.
- The **Channel Low** selection will cause the process to trigger when the signal becomes low.
- The **Channel High** selection will cause the process to trigger immediately.

If the signal is low when the process begins:

- The **High/Low** selection will cause the process to trigger when the signal becomes high, then transitions to low.
- The **Low/High** selection will cause the process to trigger when the signal becomes high.
- The **Either** selection will cause the process to trigger when the signal becomes high.
- The **Channel Low** selection will cause the process to trigger immediately.
- The **Channel High** selection will cause the process to trigger when the signal becomes high.

Digital Input Detector Parameter Descriptions

Input Table tab



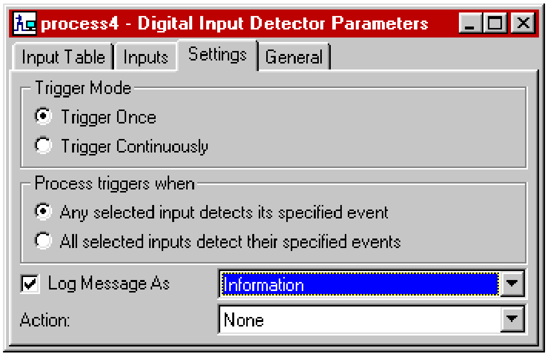
Input Table Tab

Item	Description
Digital Inputs	Displays the input signals being monitored by this process.
Options	Defines the type of signal event the process is looking for. See "Setting signal options" on page 340.

Inputs tab

Selects the signals that will be monitored by the Digital Input Detector process. Move signals to the **Included** list if you want to monitor them with this process.

Settings tab



Settings Tab (part 1 of 3)

ITEM	DESCRIPTION
Trigger Mode	**Trigger Once** causes the process to generate a trigger and a done signal when the specified digital event occurs. **Trigger Continuously** causes the process to generate a trigger signal when the specified digital event occurs. When you make this selection, the process will only terminate upon an interrupt or when its parent group or procedure ends. See "More About Trigger Signals" on page 107.
Process triggers when	Select **Any selected input detects its specified event** to end or trigger the process when any one of the inputs is detected. Select **All selected inputs detect their specified events** to end or trigger the process when all selected inputs are detected.

Settings Tab (part 2 of 3)

ITEM	DESCRIPTION
Log Message As	Assigns a severity level to the message (in ascending levels of severity: **Diagnostic**, **Information**, **Warning**, or **Error**) and makes the message available to the **MPT Specimen Log**. If the Station Manager **Station Log** and the Basic TestWare **Test Log** are configured to accept messages from all sources, then MPT makes the message available to them also. ***Note*** *Unlike Station Manager and Basic TestWare, MPT does not generate messages with a severity level of **Fatal Error**.* Each type of log will accept the message only if it has been configured to store messages with the assigned severity level. For instance, if the **MPT Specimen Log** is configured to accept messages with a minimum severity level of **Warning**, and you assign **Information** as the severity level for this message, then this message will not be recorded in the **MPT Specimen Log**. In that instance the **MPT Specimen Log** would accept messages having only **Warning** and **Error** severity levels. You can specify the minimum severity level that each type of log will accept. To do this: • For MultiPurpose TestWare, select the **MPT Options Editor**, then select the **Specimen Log** tab. • For Station Manager, select **Tools**, then select **Station Options**, then select the **Station Log** tab. • For Basic TestWare, select **Test Setup**, then select the **Test Log** tab.

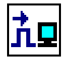
Settings Tab (part 3 of 3)

ITEM		DESCRIPTION
Action		***Note*** *For more information on Actions, see "Using actions with processes" on page 333*
		Specifies one of the following detector actions:
	None	The process ends without generating a message or performing an action.
	Station Power Off	The process removes hydraulic power from the station. The test can be resumed after you reapply hydraulic power to the station.
	Interlock	The process triggers a hydraulic interlock and shuts down the hydraulic power supply. The test can be resumed after you reset the interlock, reactivate hydraulic power, and apply hydraulic power to the station.
	Program Interlock	The process triggers a program interlock. The test can be resumed after you reset the interlock.
	Program Stop	The process stops the test. This action functions the same as clicking the **Stop** button on the **Station Controls** panel.
	Program Hold	The process suspends the test until you click the **Run** button on the **Station Controls** panel. This action functions the same as clicking the **Hold** button on the **Station Controls** panel.

General tab

See "General tab" on page 202.

How to Define a Digital Input Detector Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Inputs** tab.
4. Move all the signals you want to monitor to the **Included** list. Move all the others to the **Available** list.
5. Click the **Input Table** tab.
6. For each digital input signal listed, click the signal option in the **Options** list.
7. Click the **Settings** tab.
8. Under **Trigger Mode**, set the trigger mode.
9. Under **Process triggers when**, set when the process triggers.
10. Optional—Enable (check) the **Log Message As** control and select the type of message you wish to generate.
11. Optional—Select an action.
12. Click the **General** tab.
13. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)

14. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)

15. In the **Execute Process** box, type number of times you want the process to execute.

    Note *Repeating the process will have no effect if you selected **Trigger Continuously** in step 8.*

16. Optional—In the **Counter Type** list, click a counter type.

17. Optional—If you enabled a counter, type a **Counter Label**.

18. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Operator Event

The Operator Event process allows the operator to affect test progress through the use of custom buttons on a special control panel. It is useful for reminding the operator of particular test events or hazards.

Sample uses include:

- Prompting the operator before the test begins.
- Prompting the operator to perform an action before continuing the test.
- Triggering the beginning of a data acquisition process.
- Holding a test indefinitely.

When you define an **Operator Event** process, you must define the button ID, the button label, a message to the operator, the trigger mode, and how the event messages are logged.

More About the Operator Event Process

The MPT control panel can display up to four operator event buttons and messages simultaneously. This means only four **Operator Event** processes can be active at any one time.

If your operator events will not run simultaneously, you can use the same button for multiple **Operator Event** processes. The button label and message will change to coincide with the next active **Operator Event** process.

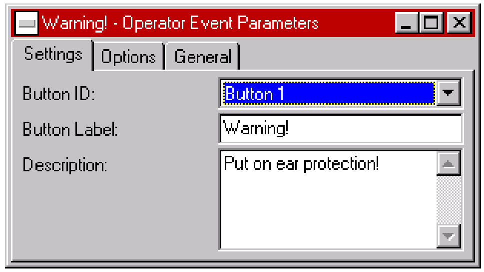
Operator Event example

Suppose a portion of the test you are designing results in a noise hazard for the operator.

You could configure an **Operator Event** process (labeled **Warning!**) to hold the test just before the hazardous portion begins. This would give the operator time to acknowledge the warning and put on ear protection. Once the operator is ready, he or she could click the **Warning!** button to resume the test.

Operator Event Parameter Descriptions

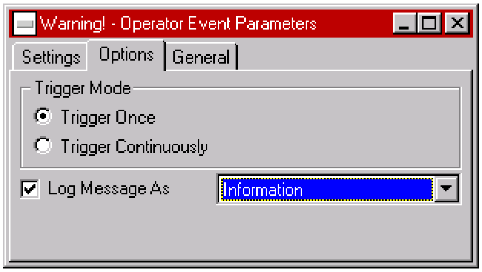
Settings tab



Settings Tab

ITEM	DESCRIPTION
Button ID	Selects one of four operator event buttons.
Button Label	Assigns a label to the button that appears on the MPT control panel (10-character maximum).
Description	Assigns the operator event message that is displayed next to the operator event button on the MPT control panel. ***Note*** *You may enter as much text as you like in the* ***Description*** *box. However, the MPT control panel will only display one line (about 50 characters), so enter the information you want the operator to see first. When you enable the* ***Log Message As*** *control (on the* ***Options*** *tab), the entire text in the* ***Description*** *box is entered in the* ***MPT Specimen Log*** *when the process completes or triggers.*

Options tab



Options Tab (part 1 of 2)

ITEM	DESCRIPTION
Trigger Mode	**Trigger Once** causes the process to generate a trigger and a done signal when the specified operator event occurs. **Trigger Continuously** causes the process to generate a trigger signal when the specified operator event occurs. When you make this selection, the process will only terminate upon an interrupt or when its parent group or procedure ends. See “More About Trigger Signals” on page 107.

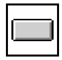
Options Tab (part 2 of 2)

ITEM	DESCRIPTION
Log Message As	Assigns a severity level to the message (in ascending levels of severity: **Diagnostic**, **Information**, **Warning**, or **Error**) and makes the message available to the **MPT Specimen Log**. If the Station Manager **Station Log** and the Basic TestWare **Test Log** are configured to accept messages from all sources, then MPT makes the message available to them also. ***Note*** *Unlike Station Manager and Basic TestWare, MPT does not generate messages with a severity level of **Fatal Error**.* Each type of log will accept the message only if it has been configured to store messages with the assigned severity level. For instance, if the **MPT Specimen Log** is configured to accept messages with a minimum severity level of **Warning**, and you assign **Information** as the severity level for this message, then this message will not be recorded in the **MPT Specimen Log**. In that instance the **MPT Specimen Log** would accept messages having only **Warning** and **Error** severity levels. You can specify the minimum severity level that each type of log will accept. To do this: • For MultiPurpose TestWare, select the **MPT Options Editor**, then select the **Specimen Log** tab. • For Station Manager, select **Tools**, then select **Station Options**, then select the **Station Log** tab. • For Basic TestWare, select **Test Setup**, then select the **Test Log** tab.

General tab

See "General tab" on page 202.

How to Define an Operator Event Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Settings** tab.
4. In the **Button ID** list, click the button you want to use.
5. In the **Button Label** box, type in the button label that appears on the MPT control panel.
6. In the **Description** box, enter the message you want to display on the MPT control panel when this process completes or triggers.
7. Click the **Options** tab.
8. Under **Trigger Mode**, set the trigger mode.
9. Optional—Enable (check) the **Log Message As** control and select the type of message you wish to generate.
10. Click the **General** tab.
11. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
12. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
13. In the **Execute Process** box, type number of times you want the process to execute.

    Note *Repeating the process will have no effect if you selected **Trigger Continuously** in step 8.*

14. Optional—In the **Counter Type** list, click a counter type.
15. Optional—If you enabled a counter, type a **Counter Label**.
16. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Peak/Valley Change Detector

The **Peak/Valley Change Detector** process monitors signals for peak and valley changes.

In its default state, this process begins by detecting the amplitudes of the first peak and valley, which it uses as reference levels to establish the tolerance band.

- If the process is set to trigger once, when the process detects a peak or valley outside of the tolerance band, the process generates a trigger signal (its “done” signal) and ends.
- If the process is set to trigger continuously, when the process detects a peak or valley outside of the tolerance band, the process generates a trigger signal, and then automatically establishes new reference levels for the tolerance band. The process will keep triggering and establishing new reference levels until the process is interrupted.

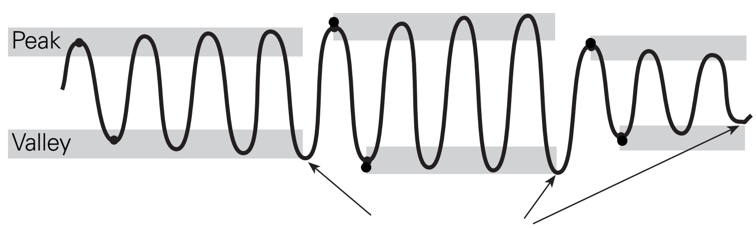
The trigger and done signals from the **Peak/Valley Change Detector** process are used to trigger other processes.

If desired, you can override automatic detection of reference levels and use the reference levels you specify. If you specify reference levels, the process will use them for the duration of the test.

This process automatically establishes peak and valley reference levels (indicated by the dots). The reference levels become the basis for the tolerance range (shown in grey).

If a peak or valley occurs outside of the tolerance band, the process generates a trigger signal and ends or establishes new reference levels.

If desired, you can enter your own reference values to be used throughout the test.



The detector triggers when a peak or valley occurs outside tolerance.

More About the Peak/Valley Change Detector Process

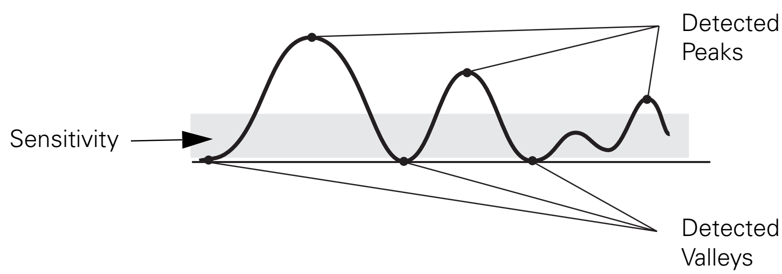
Sensitivity

A sensitivity value specifies how much the signal must change to detect a peak or valley. It can be used to keep the process from seeing signal noise as a new peak or valley.

- Setting the sensitivity too low may cause signal noise to be recognized as peaks and valleys.
- Setting the sensitivity too high may cause low amplitude signals to be missed.

Determines when a peak or valley is detected.

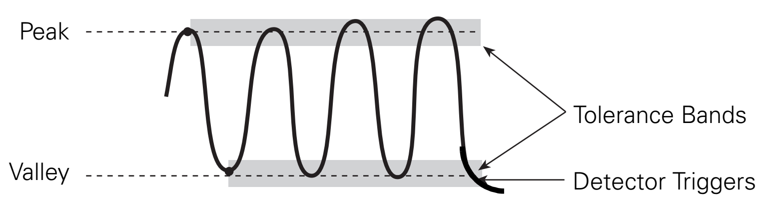
The sensitivity setting is the amount the signal must change before it is considered a new peak or valley.



Tolerance

The tolerance band specifies how much a peak or valley must deviate from the associated reference levels before the process generates a trigger signal.

For example, if the established peak value is +5 cm, and the tolerance is set to ±0.25 cm, the process will not generate a trigger signal unless the peak value registers greater than +5.25 cm or less than +4.75 cm.



Enable after

On the **Options** tab, you can use the **Enable After** command to specify how many feedback cycles will occur before the process begins to monitor the selected signal. This allows the signal to "settle-in" or stabilize before it is monitored.

If you select a channel for the **Channel Reference** on the **Command** tab, the process will begin counting cycles after the tapering is complete.

For example, if you specify 6 cycles, and the waveform begins with a taper that includes 2 cycles, the process will begin monitoring peaks and valleys on cycle 9.

If you do not use **Channel Reference**, the process will begin counting cycles immediately after the number of cycles you specify. For instance, if you specify 6 cycles, and the waveform begins with a taper that includes 2 cycles, the process will begin monitoring for peaks and valleys on cycle 7.

In either case, if you press stop and then run on the **Station Controls** panel, the process will begin the count over.

Reference values

On the **Options** tab, if you select **Default** under **Reference Values**, the first peak and valley detected by the process become the reference levels for the tolerance band.

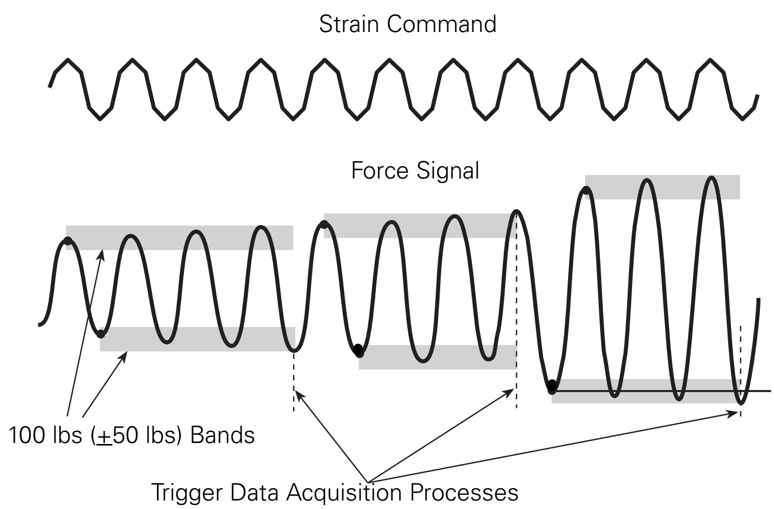
Note *The process will not generate a trigger signal based on the first peak or valley.*

On the **Options** tab, if you select **User-specified** under **Reference Values**, you enter the upper and lower level values for the tolerance band. This is a useful feature if it is possible that the first peak and valley used to establish the reference values could be unusually large or otherwise irregular. With **User-specified** reference values, the controller uses your preset reference values and compares them with the very first peak or valley, (at the start of the process or after the process resets itself after issuing a trigger). Because of this, in the **Trigger Continuously** mode, the process may generate more triggers if you use user-specified reference values than if you use default reference values.

Peak/Valley Change Detector Process Example

Suppose you want to acquire a cycle of data each time the peaks or valleys of an input signal change by a specified amount. Further, suppose you define a **Peak/Valley Change Detector** process with the following parameters:

- Trigger continuously
- Tolerance: ±50 lbs
- Sensitivity: 5 lbs



The test command cycles between ±0.0001 strain and you are monitoring a force signal.

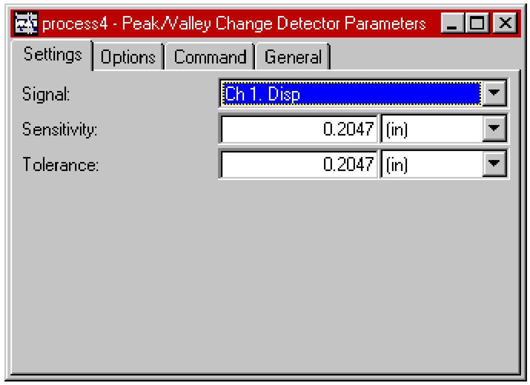
Each time the peaks or valleys on the force signal change more than ±50 lbs, the process generates a trigger signal that starts a data acquisition process.

The data acquisition process would use the **Peak/Valley Change Detector** process as its start trigger. A single data acquisition process could be run multiple times with this change detector.

Note *It is good practice to specify a number of cycles to execute before you begin monitoring to allow a "settling in" period.*

Peak/Valley Change Detector Parameter Descriptions

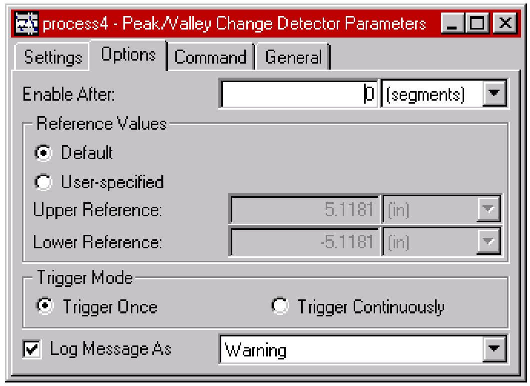
Settings tab



Settings Tab

ITEM	DESCRIPTION
Signal	Selects the signal you will monitor with the **Peak/Valley Change Detector** process.
Sensitivity	Defines how much the signal must change before it is considered a peak or valley. See “Sensitivity” on page 355.
Tolerance	Defines a tolerance band for the peaks and valleys. If a peak or valley occurs outside the tolerance band, the process triggers. See “Tolerance” on page 355.

Options tab



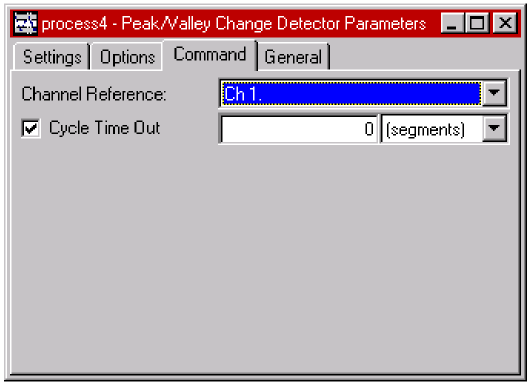
Options Tab (part 1 of 2)

ITEM	DESCRIPTION
Enable After	Specifies how many segments or cycles will occur before the process begins to monitor the selected signal. See "Enable after" on page 356.
Reference Values	Specifies how the upper and lower reference values are established. See "Reference values" on page 356.
Trigger Mode	**Trigger Once** causes the process to generate a trigger and a done signal when a peak or valley occurs outside the tolerance band. **Trigger Continuously** causes the process to generate a trigger signal when a peak or valley occurs outside the tolerance band. The process will keep triggering until the process is interrupted. See "More About Trigger Signals" on page 107.

Options Tab (part 2 of 2)

ITEM	DESCRIPTION
Log Message As	Assigns a severity level to the message (in ascending levels of severity: **Diagnostic**, **Information**, **Warning**, or **Error**) and makes the message available to the **MPT Specimen Log**. If the Station Manager **Station Log** and the Basic TestWare **Test Log** are configured to accept messages from all sources, then MPT makes the message available to them also. ***Note*** *Unlike Station Manager and Basic TestWare, MPT does not generate messages with a severity level of **Fatal Error**.* Each type of log will accept the message only if it has been configured to store messages with the assigned severity level. For instance, if the **MPT Specimen Log** is configured to accept messages with a minimum severity level of **Warning**, and you assign **Information** as the severity level for this message, then this message will not be recorded in the **MPT Specimen Log**. In that instance the **MPT Specimen Log** would accept messages having only **Warning** and **Error** severity levels. You can specify the minimum severity level that each type of log will accept. To do this: • For MultiPurpose TestWare, select the **MPT Options Editor**, then select the **Specimen Log** tab. • For Station Manager, select **Tools**, then select **Station Options**, then select the **Station Log** tab. • For Basic TestWare, select **Test Setup**, then select the **Test Log** tab.

Command tab



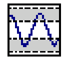
Command Tab

ITEM	DESCRIPTION
Channel Reference	Selects the command channel associated with the feedback signal that you want to monitor. If you select **None**, which is the default, the **Cycle Time Out** control is disabled. Use this feature if your procedure includes tapered waveforms. With the command channel, MPT can identify and avoid monitoring the transitions that occur during tapered cycles.
Cycle Time Out	***Note*** *This feature is enabled only if you select a channel in the **Channel Reference** box.* Counts cycles from the beginning of the process, including cycles in tapers. If the process does not detect a peak or valley during the specified number of cycles (or segments), it issues a trigger. If you enter a zero, the process will issue a trigger immediately. If you press **Stop** and then **Run** on the **Station Controls** panel, the process will start counting over again. ***Note*** *This feature can be used to detect a malfunction; for example, if a test is run when the **Master Span** (on the **Station Controls** panel) is set very low.*

General tab

See "General tab" on page 202.

How to Define a Peak/Valley Change Detector Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Settings** tab.
4. In the **Signal** list, click the signal you want to monitor.
5. In the **Sensitivity** box, set the units, and then enter a value.
6. In the **Tolerance** box, set the units, and then enter a value.
7. Click the **Options** tab.
8. Optional—If you want to delay the Peak/Valley monitoring process for a number of cycles to allow for a "settling in" period, enter a value in the **Enable After** box.
9. Under **Reference Values**, select **Default** or **User-specified**.

   If you select **User-specified Reference Values**, enter the desired **Upper** and **Lower Reference Level** values.
10. Under **Trigger Mode**, set the trigger mode.
11. Optional—Enable (check) the **Log Message As** control and select the type of message you wish to generate.
12. Optional—Click the **Command** tab, click a **Channel Reference** signal and enter a **Cycle Time Out** value.

13. Click the **General** tab.

14. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)

15. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)

16. In the **Execute Process** box, type number of times you want the process to execute.

    Note *Repeating the process will have no effect if you selected **Trigger Continuously** in step 10.*

17. Optional—In the **Counter Type** list, click a counter type.

18. Optional—If you enabled a counter, type a **Counter Label**.

19. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Program Event

The **Program Event** process is typically used to trigger other processes in response to a change in the test state.

There are seven test states that can generate a trigger and/or done signal in the **Program Event** process:

- **Running**—the state in which the procedure is controlling the machine and playing out a waveform.
- **Entering Hold**—the transition state between Running and Holding.
- **Holding**—the state in which the test is suspended (the actuator is not moving), but the test can be continued by clicking the **Run** button.

Note *This process sees a "paused" state in the same way that it sees a Holding state. That is, as the run indicator on the* ***Station Controls*** *panel blinks while "paused," this process will see the following sequence of transitions: Running>Entering Hold>Holding>Exiting Hold>Running.*

- **Exiting Hold**—the transition state between Holding and Running. It will typically complement what is done in the Entering Hold state.
- **Entering Stopped**—the transition state between Running/ Holding and Stopped.
- **Stopped**—the Stopped state. Once in this state, the actuators are fully stopped. If appropriate, the test can be continued by clicking the **Run** button, but it will not start up by itself.

  Note *The only MPT command process that will execute during a Stopped state is the Digital Output process.*

- **Exiting Stopped**—the transition state between Stopped and Running. It typically complements what is done in the Entering Stopped state.

More About the Program Event Process

Behavior with Stopped and Holding states

Most processes stop or suspend themselves when MPT enters a Stopped or Holding state. When a Program Event process detects one of these states, the process that is triggered will typically see that MPT is stopped or holding and refuse to do anything. Because it is common to use a Program Event process to trigger a Digital Output process that turns off an external device when the test is stopped, the Digital Output process is designed to function in a Stopped or Holding state.

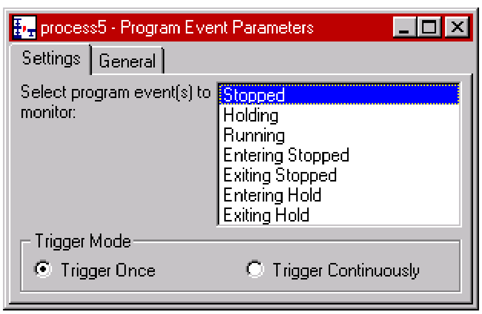
Detecting transitions after a Done Interrupt

When a procedure or group process is done, MPT sends out an Interrupt signal to all of the active child processes. Program Event processes cannot detect Entering Stopped and Stopped transitions that occur after a Done Interrupt.

Note *For more information about transitions, see* *"About Transition States" on page 173.*

Program Event Parameter Descriptions

Settings tab



Settings Tab (part 1 of 2)

ITEM	DESCRIPTION
Select program event(s) to monitor	Selects the program event(s) you want to monitor. You can choose as many program events as you want. To select more than one event, hold the **Ctrl** key down while clicking the desired events. To select a range of events, click the first event in the range, then hold the **Shift** key down and click the last event in the range.

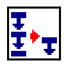
Settings Tab (part 2 of 2)

ITEM	DESCRIPTION
Trigger Mode	**Trigger Once** causes the process to generate a trigger and a done signal when a program event occurs. **Trigger Continuously** causes the process to generate a trigger signal when a program event occurs. The process will keep triggering on program events until the process is interrupted. See “More About Trigger Signals” on page 107.

General tab

See “General tab” on page 202.

How to Define a Program Event Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Settings** tab.
4. Hold the **Ctrl** key and click the program events you want to monitor.
5. Under **Trigger Mode**, set the trigger mode.
6. Click the **General** tab.
7. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
8. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
9. In the **Execute Process** box, type number of times you want the process to execute.

   Note *Repeating the process will have no effect if you selected Trigger Continuously in step 5.*

10. Optional—In the **Counter Type** list, click a counter type.
11. Optional—If you enabled a counter, type a **Counter Label**.
12. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Periodic Time Event

With the Periodic Time Event process, you can schedule arbitrary and repetitive times at which the process will produce a trigger, or a series of triggers. You can use these time based triggers to invoke an event, such as triggering another process, actuating an external device, etc.

When you define a Periodic Time Event process, you can specify periodic trigger times, the trigger mode, and whether or not the process writes a message to the message log when it invokes a trigger event.

Note *To produce a trigger the Periodic Time Event process must be active at the time you wish the trigger event to occur, so it is important to keep that in mind when designing your test.*

Periodic Time Event example

Suppose you need to start warming up a temperature chamber before a technician arrives at your facility, so that it has achieved and stabilized at the desired temperature by the time the technician begins testing. Further, suppose the technician is scheduled to perform tests at 8:00AM on Mondays, Wednesdays, and Thursdays beginning in March and continuing through May.

In this case you could use a Periodic Time Event process to trigger a Temperature Control process hours before the technician arrives on the days and in the months testing is scheduled.

Periodic Time Event Parameter Descriptions

Settings tab

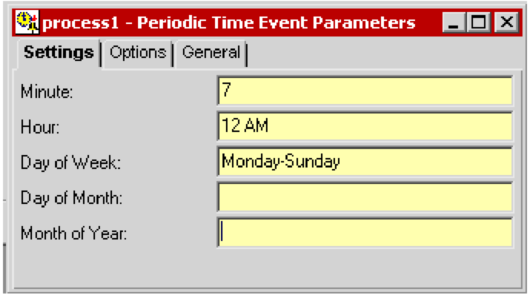
Double-click the process icon to open the process parameters window. Click each tab to access a different set of process parameters.

You may type in entries in any of the entry boxes or choose from the selection list. To display the selection list of an entry box, double-click on the box, or position the cursor in the edit field of the desired box and press the up or down arrow keys.

Each box may be empty (meaning any value), may contain a single value, or may contain a list of values separated by commas. Each value may be a single value or a range of values (two separate values joined by a dash). For example: 1-3, 5, 7, 8-13, 19 means 1 through 3, or 5, or 7, or 8 through 13, or 19.

While running or holding, the process checks the time (clock and calendar time) at which it begins and then periodically at one-minute intervals.

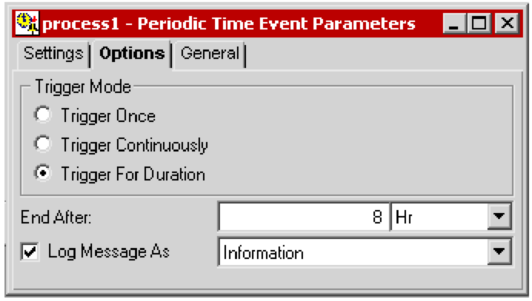
The process triggers when the current time matches the schedule criteria. For example, in the window shown below, the process will trigger Monday through Sunday (everyday) of every month at 12:07 AM.



Settings Tab

ITEM	DESCRIPTION
Minute	Accepts values from 0 to 59.
Hour	Contains selections displayed in 12 or 24 hour format based on the regional settings of the operating system. Accepts value entries in either the 12 or 24 hour format.
Day of Week	Contains selections displayed according to the regional settings of the operating system. Accepts entries in English or local language.
Day of Month	Accepts values from 1 to 31.
Month of Year	Contains selections displayed according to the regional settings of the operating system. Accepts entries in English or local language.

Options tab



Options Tab (part 1 of 2)

ITEM	DESCRIPTION
Trigger Mode	**Trigger Once** causes the process to generate a trigger and a done signal when the specified event occurs. **Trigger Continuously** causes the process to generate a trigger signal when the specified event occurs. When you make this selection, the process will only terminate upon an interrupt or when its parent group or procedure ends. See "More About Trigger Signals" on page 107. **Trigger For Duration** is the same as the **Trigger Continuously** control except you can specify the duration of time the process remains active. When selected, the **End After** control is enabled, which allows you to specify the time that the process will end in relation to when it invoked its first trigger.

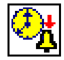
Options Tab (part 2 of 2)

ITEM	DESCRIPTION
Log Message As	Assigns a severity level to the message (in ascending levels of severity: **Diagnostic**, **Information**, **Warning**, or **Error**) and makes the message available to the **MPT Specimen Log**. If the Station Manager **Station Log** and the Basic TestWare **Test Log** are configured to accept messages from all sources, then MPT makes the message available to them also. ***Note*** *Unlike Station Manager and Basic TestWare, MPT does not generate messages with a severity level of **Fatal Error**.* Each type of log will accept the message only if it has been configured to store messages with the assigned severity level. For instance, if the **MPT Specimen Log** is configured to accept messages with a minimum severity level of **Warning**, and you assign **Information** as the severity level for this message, then this message will not be recorded in the **MPT Specimen Log**. In that instance the **MPT Specimen Log** would accept messages having only **Warning** and **Error** severity levels. You can specify the minimum severity level that each type of log will accept. To do this: • For MultiPurpose TestWare, select the **MPT Options Editor**, then select the **Specimen Log** tab. • For Station Manager, select **Tools**, then select **Station Options**, then select the **Station Log** tab. • For Basic TestWare, select **Test Setup**, then select the **Test Log** tab.

General tab

See "General tab" on page 202.

How to Define a Periodic Time Event Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Settings** tab.

Note *Double-clicking an entry box displays its selection list.*

4. Enter or select desired values in the **Minute**, **Hour**, **Day of Week**, **Day of Month**, and **Month of Year** entry boxes.
5. Click the **Options** tab.
6. Under **Trigger Mode**, set the trigger mode.
7. Optional—Enable (check) the **Log Message As** control and select the type of message you wish to generate.
8. Click the **General** tab.
9. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
10. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
11. In the **Execute Process** box, type number of times you want the process to execute.

    Note *Repeating the process will have no effect if you selected **Trigger Continuously** in step 6.*

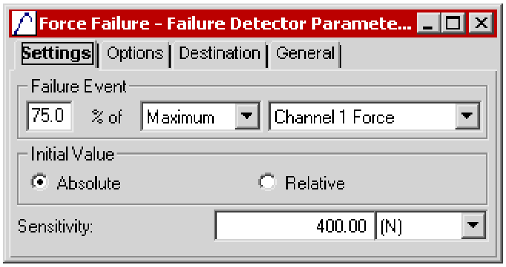
12. Optional—In the **Counter Type** list, click a counter type.
13. Optional—If you enabled a counter, type a **Counter Label**.
14. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Failure Detector

The **Failure Detector** process is detector process which monitors signals for occurrence failure events (peak or valley values), determines whether the failure events meet the criteria for specimen failure, and if so, completes the process.

Failure Detector Parameter Descriptions

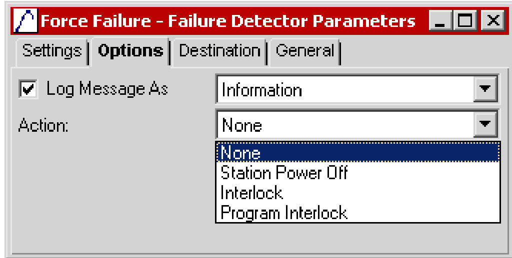
Settings tab



Settings Tab

ITEM	DESCRIPTION
Failure Event	
% of	Specifies the percentage of the monitored signal's maximum or minimum value that represents specimen failure. When the monitored signal reaches this level the process ends.
event type (**Maximum** or **Minimum**)	Selects the type of event the process looks for in the monitored signal. For example, if you select maximum, a maximum or peak level will trigger the process.
monitored signal	Selects the input signal for which the process monitors your failure criteria.
Initial Value	Specifies whether the process monitors the value of the selected signal relative to zero (**Absolute**), or relative to the value of the signal when the Failure Detector process started (**Relative**).
Sensitivity	Specifies how much the monitored signal must change before a level is considered a maximum event (peak) or a minimum event (valley).

Options tab



Options Tab (part 1 of 2)

Item	Description
Log Message As	Assigns a severity level to the message (in ascending levels of severity: **Diagnostic**, **Information**, **Warning**, or **Error**) and makes the message available to the **MPT Specimen Log**. If the Station Manager **Station Log** and the Basic TestWare **Test Log** are configured to accept messages from all sources, then MPT makes the message available to them also. ***Note*** *Unlike Station Manager and Basic TestWare, MPT does not generate messages with a severity level of **Fatal Error**.* Each type of log will accept the message only if it has been configured to store messages with the assigned severity level. For instance, if the **MPT Specimen Log** is configured to accept messages with a minimum severity level of **Warning**, and you assign **Information** as the severity level for this message, then this message will not be recorded in the **MPT Specimen Log**. In that instance the **MPT Specimen Log** would accept messages having only **Warning** and **Error** severity levels. You can specify the minimum severity level that each type of log will accept. To do this: • For MultiPurpose TestWare, select the **MPT Options Editor**, then select the **Specimen Log** tab. • For Station Manager, select **Tools**, then select **Station Options**, then select the **Station Log** tab. • For Basic TestWare, select **Test Setup**, then select the **Test Log** tab.

Options Tab (part 2 of 2)

ITEM		DESCRIPTION
Action		Specifies the action that occurs when the process completes.
	None	The process ends without generating a message or performing an action.
	Station Power Off	The process removes hydraulic power from the station. The test can be resumed after you reapply hydraulic power to the station.
	Interlock	The process triggers a hydraulic interlock and shuts down the hydraulic power supply. The test can be resumed after you reset the interlock, reactivate hydraulic power, and apply hydraulic power to the station.
	Program Interlock	The process triggers a program interlock. The test can be resumed after you reset the interlock.
	Program Stop	The process stops the test. This action functions the same as clicking the **Stop** button on the **Station Controls** panel.
	Program Hold	The process suspends the test until you click the **Run** button on the MPT control panel. This action functions the same as clicking the **Hold** button on the **Station Controls** panel.

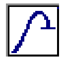
Destination tab

See “Data destination” on page 268.

General tab

See “General tab” on page 202.

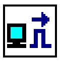
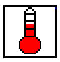
How to Define a Failure Detector Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Settings** tab.
4. Define your desired specimen failure criteria with the **Failure Event**, **Initial Settings**, and **Sensitivity** controls.
5. Click the **Options** tab.
6. Optional—Enable (check) the **Log Message As** control and select the type of message you wish to generate.
7. Optional—Select the desired action you wish to occur if the specimen condition matches your failure criteria and the process completes.
8. Click the **Destination** tab.
9. In the **Destination** list, click the destination for the acquired data. (If you click **User-specified data file**, enter a file name in the **User Data File** box.) See "Data destination" on page 268.
10. Type a description of the data file in the **Data Header** box.
11. Click the **General** tab.
12. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)

13. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
14. In the **Execute Process** box, type number of times you want the process to execute.
15. Optional—In the **Counter Type** list, click a counter type.
16. Optional—If you enabled a counter, type a **Counter Label**.
17. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

External Control Processes

You use external control processes to send signals to external controllers or logic devices.

ICON	PROCESS NAME	DESCRIPTION
	**Digital Output**	Controls digital outputs (see page 379).
	**Temperature Control**	Programs and/or controls an external temperature controller (see page 384).

Digital Output

The **Digital Output** process invokes the controller to issue digital signals that can be sent to external switches and logic devices.

Note *Before you configure a Digital Output process to signal an external device, ensure the signal values listed below are compatible with the device.*

The following signal options are available with the **Digital Output** process:

- **None**–The process does not send a digital signal, but the done signal can still be used to start and interrupt other processes.
- **Set**–The process generates a logic high signal (+12 V DC).
- **Clear**–The process generates a logic low signal (0 V DC).
- **Toggle**–The process inverts the current state (from low-to-high or high-to-low).
- **Pulse**–The process inverts the current state with a pulse signal, holds the inverted state for the time specified in the **Pulse Width** control, then returns to the original state.

  Note *Once the controller issues the pulse, the pulse will not be affected by the stop or hold controls on the **Station Controls** panel.*

More About the Digital Output Process

Most of the time, processes stop or suspend themselves when MPT enters a stopped or holding state. This is not true of the **Digital Output** process.

A **Digital Output** process will still function when MPT enters a Stopped or Holding state. It is common to use a **Program Event** process to trigger a **Digital Output** process (to turn off an external device) when the test is stopped or held.

Keep in mind

When you are configuring a **Digital Output** process for use in your procedure, keep in mind:

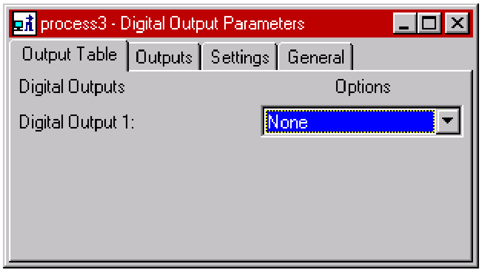
- You must allocate a digital output resource in the station configuration file in order to use the **Digital Output** process.
- You should choose an output signal option that is compatible with the external device connected.
- Outputs assigned for a **Digital Output** process cannot be manually triggered from Station Manager while MPT is in the execute mode.
- When the **Digital Output** process ends, the state of the external device will remain in the current state until a new station is opened or until another **Digital Output** process changes that state.

  Note *Use the **Pulse** option if you want to return the output to its initial state when the process is done.*

Process Descriptions

Digital Output Parameter Descriptions

Output Table tab



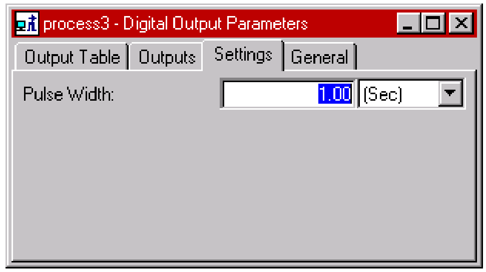
Output Table Tab

ITEM	DESCRIPTION
Digital Outputs	Lists the digital outputs included on the **Outputs** tab.
Options	Defines the type of digital signal that will be sent to the external device: • **None**–The process does not send a digital signal, but the done signal can still be used to start and interrupt other processes. • **Set**–The process generates a logic high signal (+12 V DC). • **Clear**–The process generates a logic low signal (0 V DC). • **Toggle**–The process inverts the current state (from low-to-high or high-to-low). • **Pulse**–The process inverts the current state with a pulse signal, holds the inverted state for the time specified in the **Pulse Width** control, and then returns to the original state. If you choose the **Set**, **Clear**, or **Toggle** selections, the process generates a done trigger as soon as it sends the selected digital signals to the external device. If you select the **Pulse** selection, the process does not generate the done signal until the pulse width time has expired and those outputs return to their original state.

Outputs tab

Selects the digital output channels that this process will control. Use the arrow keys to move the desired channels from the **Available** list to the **Included** list.

Settings tab



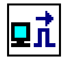
Settings Tab

ITEM	DESCRIPTION
Pulse Width	Specifies the duration of the pulse the controller sends to the external device. You may select a time in the range of 100 milliseconds to 2 seconds with an increment of 10 milliseconds. ***Note*** *This control is enabled only if you have a digital output channel using the* ***Pulse*** *option.*

General tab

See "General tab" on page 202.

How to Define a Digital Output Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Outputs** tab.
4. Move all the channels you want to send digital signals to the **Included** list. Move all the others to the **Available** list.
5. Click the **Output Table** tab.
6. Select an option for each digital output channel listed.

Note *If you select* ***Pulse*** *for any channel, you must specify a* ***Pulse Width*** *on the* ***Settings*** *tab.*

7. Click the **General** tab.
8. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
9. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
10. In the **Execute Process** box, type number of times you want the process to execute.
11. Optional—In the **Counter Type** list, click a counter type.
12. Optional—If you enabled a counter, type a **Counter Label**.
13. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Temperature Control

The **Temperature Control** process provides setpoint programming to external temperature controllers cabled to your test system.

Depending on how your hardware is configured, you may be able to:

- Use temperature feedback from a single thermocouple to maintain a specified temperature tolerance for a specified amount of time.
- Use temperature feedback from multiple thermocouples to ensure the desired temperature “soaks into” the specimen.

  Note *Typically, one thermocouple is placed in the chamber, and a second thermocouple is attached to the physical specimen.*

When you define a **Temperature Control** process, you must specify the desired end level (temperature), a transition time to the specified temperature, and the amount of time you want to maintain that temperature.

Note *The external temperature controller must be properly cabled to your controller in order for this process to work correctly. For cabling information, see the Controller Setup or Programmer Setup manual that accompanied your test system.*

More About the Temperature Control Process

High and low rate channels

The Model 793.00 System Software supports normal and “low-rate” programming channels. Since temperature controllers do not require frequent setpoint updates, they are often configured as low-rate channels.

By default, low-rate channels operate at a command rate of 25.6 Hz (1 Hz for Eurotherm temperature controllers connected to the TestStar IIs **J51** serial-port interface). For more information on low-rate channels, see the *Model 793.00 System Software* manual that accompanied your test system.

Note *If a process includes both normal (high-rate) and low rate channels, the channels will start and stop together. However, they may not stop at the same level. They have be off in phase as much as one low-rate tick.*

Programming multiple channels with your controller

Since low-rate channels do not require as much processing power, you can program a temperature controller on a low-rate channel and a servovalve on a normal-rate channel at the same time.

Controller options with your controller

Depending on the type of external controller you are programming, the **Temperature Control** process may be configured differently. Refer to the following sections for special considerations.

Eurotherm controllers

Some controllers support a special cabling configuration for Eurotherm temperature controllers. When a Eurotherm temperature controller is connected to your system, special **Temp Output 1** and **Temp Input 1** resources must be added to your .hwi file.

When configured this way, the **Temperature Control** process sends temperature commands to the Eurotherm controller through the J51 serial connector at 1 Hz, and receives temperature feedback (for data acquisition) through the same connector at 0.5 Hz (EI-BISYNCH protocol). In the temperature control process parameters window, you can set a **Control Tolerance** and a **Dwell Period**. If your configuration includes an additional analog input resource for an external temperature conditioner, you can also set the **Monitor Tolerance**.

Other controllers with Program w/Feedback

If the programming channel is configured as **Program with Feedback**, the process sends the MPT end level command to the temperature controller, and can monitor temperature feedback to maintain a **Control Tolerance** and a **Dwell Period**.

If your configuration includes an additional analog input resource for an external temperature conditioner, you can also set the **Monitor Tolerance**.

Note *The **Control Tolerance** and **Monitor Tolerance** features require independent feedback channels.*

Note *In most cases, **Program with Feedback** channels are configured as low-rate channels.*

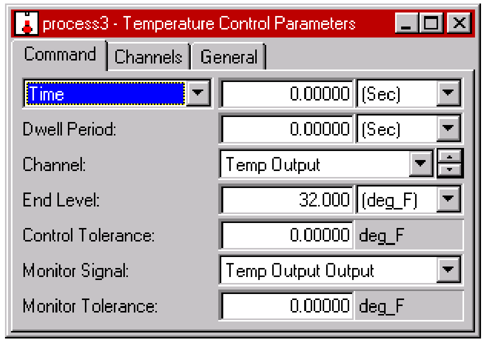
Other controllers with Program Only

If the programming channel is configured as **Program Only**, the process sends the MPT end level command to the temperature controller, which maintains the control loop. In this scenario, you cannot set a **Control Tolerance**, **Monitor Tolerance**, or limits on temperature feedback.

Note *In most cases, **Program Only** channels are configured as low-rate channels.*

Temperature Control Parameter Descriptions

Command tab



Command Tab (part 1 of 2)

ITEM	DESCRIPTION
rate type	Specifies the method with which the controller ramps the temperature to the specified end level. The following rate types are supported: • **Time** specifies the time to execute one segment. • **Frequency** specifies the time to execute a two-segment cycle (even though a single segment executes). • **Rate** specifies a constant rate between the starting level and the end level. ***Note*** *You cannot use **Rate** for a process applied to multiple channels. If you do, an error message will appear when the procedure is locked.*
Dwell Period	Specifies the amount of time the temperature feedback must stay within tolerance before the process terminates. The dwell period begins as soon as the temperature enters the specified control tolerance range. ***Note*** *If the temperature strays outside of the tolerance range during the dwell period, the dwell period resets. The dwell period starts over when the temperature reenters the tolerance range.* If you choose a dwell period of zero, the process will terminate as soon as the temperature feedback becomes within tolerance. If your station configuration supports the monitor feature, then the dwell period applies to both the control tolerance and monitor tolerance ranges.

Command Tab (part 2 of 2)

ITEM	DESCRIPTION
Channel	Specifies the channel to which the window applies.
End Level	Specifies the desired temperature—that is, the temperature you want the controller to achieve (and maintain if a dwell period is specified).
Control Tolerance	Specifies the amount the control temperature feedback may vary from the specified **End Level** during the **Dwell Period**. This feature requires feedback from the temperature sensor associated with the current channel—that is, the feedback used by the external temperature controller to achieve closed-loop control.
Monitor Signal	Specifies the temperature feedback signal associated with the monitor feature. This feedback comes from a second temperature sensor in the enclosure, which is conditioned by its own conditioner. This feedback is independent of the feedback used by the external temperature controller to achieve closed-loop control.
Monitor Tolerance	Specifies the amount the monitor temperature feedback may vary from the specified **End Level** during the **Dwell Period**. If you use the monitor feature, the **Temperature Control** process monitors the feedback associated with the **Control Tolerance** and the feedback associated with the **Monitor Tolerance** concurrently. If either feedback strays outside of its tolerance range the dwell period starts over.

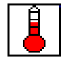
Channels tab

See “Channels tab” on page 202.

General tab

See “General tab” on page 202.

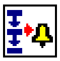
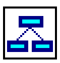
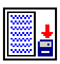
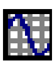
How to Define a Temperature Control Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Channels** tab.
4. Move all the channels you want to program with this process to the **Included** list. Move all the others to the **Available** list.
5. Click the **Command** tab.
6. Set the rate type (**Time**, **Rate**, or **Frequency**), set the units of measurement, and then type a value.
7. In the **Dwell Period** box, set the units of measurement for the dwell period, and then type a value.
8. In the **Channels** list, click each temperature control channel in succession.
9. In the **End Level** box, set the units of measurement for the end level, and then type a value.
10. In the **Control Tolerance** box, type a value.
11. Optional—In the **Monitor Signal** list, click the channel you want to monitor temperature feedback on (this signal should be from an independent temperature sensor).
12. Optional—In the **Monitor Tolerance** box, type a value.
13. Repeat steps 8 through 12 for each temperature control channel.

14. Click the **General** tab.
15. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
16. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
17. In the **Execute Process** box, type number of times you want the process to execute.
18. Optional—In the **Counter Type** list, click a counter type.
19. Optional—If you enabled a counter, type a **Counter Label**.
20. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Other Processes

You use other processes to perform a variety of miscellaneous functions.

Icon	Process Name	Description
	**Program Control**	Performs user-defined program actions such as Program Hold, Program Resume, Interlock, and user defined actions. (See page 391.)
	**Group**	Defines a process sub-group that may contain one or more other processes. (See page 396.)
	**Operator Information**	Enables operator data entry (to be written to a data file) during the test. (See page 398.)
	**Start Application**	Triggers another application to start during the test. (See page 406.)
	**Data Display**	Plots data signals on-screen during the test. (See page 411.)

Program Control

The **Program Control** process can be used to generate a message and/or perform a specified action. Once the process performs the specified action, the process ends.

Note *Generally, you use a detector process to start a Program Control process.*

Logging Program Control messages

If desired, you can save generated messages to the message logs.

Note *When logging messages via the Program Control process, messages from the process may appear in the log before the "Running" message that pertains to the procedure.*

More About the Program Control Process

Program Control process example

Suppose you wish to add a new capability to your existing procedure in which the message "'Failure Threshold Attained" is issued to the message log if displacement feedback reaches 2 cm.

To do this, you would add a Data Limit Detector process and a Program Control process to your procedure.

You would configure the Data Limit Detector process to monitor displacement feedback and attain its done state if displacement feedback equals 2 cm.

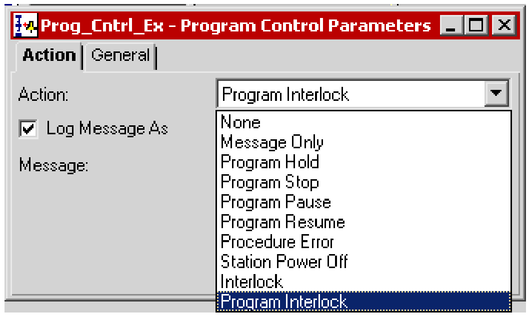
You would sequence the Program Control process to start when the Data Limit Detector process is done.

Then you would configure the Program Control process by selecting the **Message Only** action, and entering "Failure Threshold Attained" in the message entry box.

When you run a test with the modified procedure, the message "Failure Threshold Attained" will log if displacement feedback reaches 2 cm.

Program Control Parameter Descriptions

Action tab



Action Tab (part 1 of 3)

ITEM	DESCRIPTION
Action	***Note*** *For more information on Actions, see "Using actions with processes" on page 333* Specifies the action that occurs when the process completes.
None	The process ends without generating a message or performing an action.
Message Only	The process writes a message to the **MPT Specimen Log**. (You define the content of the message in the **Message** box.)
Program Hold	The process suspends the test until you click the **Run** button on the **Station Controls** panel. This action functions the same as clicking the **Hold** button on the **Station Controls** panel.
Program Stop	The process stops the test. This action functions the same as clicking the **Stop** button on the **Station Controls** panel.
Program Pause	Similar to the **Program Hold** action, but the test can be resumed by a **Program Resume** action from another Program Control process.
Program Resume	Resumes a test paused by a **Program Pause** action.
Procedure Error	The process stops the test and generates an error. The procedure goes to the error state and cannot be resumed. To run the procedure again, click the **Reset** button, then click the **Run** button on the **Station Controls** panel.
Station Power Off	The process removes hydraulic power from the station. The test can be resumed after you reapply hydraulic power to the station.

Action Tab (part 2 of 3)

ITEM	DESCRIPTION
Interlock	The process triggers a hydraulic interlock and shuts down the hydraulic power supply. The test can be resumed after you reset the interlock, reactivate hydraulic power, and apply hydraulic power to the station.
Program Interlock	The process triggers a program interlock. The test can be resumed after you reset the interlock.
Log Message As	Assigns a severity level to the message (in ascending levels of severity: **Diagnostic**, **Information**, **Warning**, or **Error**) and makes the message available to the **MPT Specimen Log**. If the Station Manager **Station Log** and the Basic TestWare **Test Log** are configured to accept messages from all sources, then MPT makes the message available to them also. ***Note*** *Unlike Station Manager and Basic TestWare, MPT does not generate messages with a severity level of **Fatal Error**.* Each type of log will accept the message only if it has been configured to store messages with the assigned severity level. For instance, if the **MPT Specimen Log** is configured to accept messages with a minimum severity level of **Warning**, and you assign **Information** as the severity level for this message, then this message will not be recorded in the **MPT Specimen Log**. In that instance the **MPT Specimen Log** would accept messages having only **Warning** and **Error** severity levels. You can specify the minimum severity level that each type of log will accept. To do this: • For MultiPurpose TestWare, select the **MPT Options Editor**, then select the **Specimen Log** tab. • For Station Manager, select **Tools**, then select **Station Options**, then select the **Station Log** tab. • For Basic TestWare, select **Test Setup**, then select the **Test Log** tab.

Action Tab (part 3 of 3)

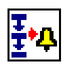
ITEM	DESCRIPTION
Message	Defines the content of the message written to the log.
Include Counters[1]	Includes counter information with the message written to the message log. ***Note*** *This option can be used to periodically log counter information.*

1. If the Program Control process is configured to include counters and log on a transition state change, your log will contain two similar messages each time the state changes. This occurs because all state transitions are automatically logged with counter information. Keep in mind that the automatic logs are displayed as Information, while the Program Control logs are displayed in the severity level you specify (Information, Warning, Error, or Diagnostic). This allows you discern between these log entries if desired.

General tab

See "General tab" on page 202.

How to Define a Program Control Process

1. Drag the  icon from the **Process Palette** to your Procedure table.
2. Double-click the process icon in your Procedure table.
3. Click the **Action** tab.
4. In the **Action** list, click the action for this process.
5. Optional—If you want to write a message to the log when this process triggers, select the **Log Message As** check box and click **Diagnostic**, **Information**, **Warning,** or **Error** in the list.
6. Type the message that you want to send to the various message logs when the process triggers.
7. Click the **General** tab.

8. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
9. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
10. In the **Execute Process** box, type number of times you want the process to execute.
11. Optional—In the **Counter Type** list, click a counter type.
12. Optional—If you enabled a counter, type a **Counter Label**.
13. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Procedure table.

Group

The Group process is basically a sub-procedure that runs within your main procedure (or within another Group process). On the Procedure table, you can double-click the Group process icon to display the Group Parameters window.

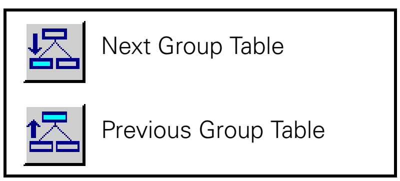
The Group table looks similar to the Procedure table. You add processes to it in the same manner.

For information on sequencing Group processes, see "Sequencing Group Processes" on page 116.

How to Move In and Out of a Group Process Window

When you click a Group process icon on a table and click the Next Group Table button on the MPT toolbar, a new Group table appears. You can also right-click on a Group process icon on a table and select Open Table to make a new table appear.

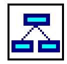
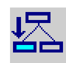
To move back to a Group process' parent table, click the Previous Group Table button on the toolbar:



Group Parameter Descriptions

General tab See "General tab" on page 202.

How to Define a Group Process

1. Drag the  icon from the **Process Palette** to your **Group** or **Procedure** table.
2. Click the **Group** process icon in your **Group** or **Procedure** table, and then click  on the **MPT Procedure Editor**.
3. Add processes to your Group table. Make sure to set the parameters for each process included in your group.
4. Set **Start** and **Interrupt** signals for each process in your group. Also, set the **Group is done** when signal.
5. Click  on the **MPT Procedure Editor** to return to the parent table.
6. Double-click the **Group** process icon in your Group or Procedure table.
7. Click the **General** tab.
8. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
9. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
10. In the **Execute Process** box, type number of times you want the process to execute.
11. Optional—In the **Counter Type** list, click a counter type.
12. Optional—If you enabled a counter, type a **Counter Label**.
13. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

Operator Information

The **Operator Information** process requires the operator to enter information at a specific time during the test. The information entered can be archived along with other data during the test.

Some examples of the types of things a test designer can do with the **Operator Information** process include:

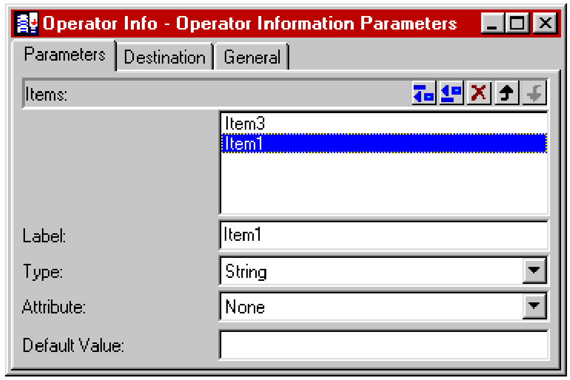
- Controlling the test flow by prompting the operator for information at any time during the test.
- Providing information about the physical specimen with the data file.
- Prompting the operator for specific information, such as:
  - Operator ID
  - Batch number
  - Workstation number
  - Part number
  - Operator comments

When an **Operator Information** process is triggered, it displays a window that requires the operator to enter the type of information specified by the test designer. As the test designer, you can choose whether the information is non-editable, or whether specific information, comments, or values must be entered when the window appears.

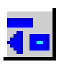
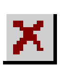
When the operator enters the required information in the window and clicks **OK**, the information is output to a test data file.

Operator Information Parameter Descriptions

Parameters tab



Parameters Tab (part 1 of 2)

ITEM	DESCRIPTION
Items	Displays the **Operator Information** process toolbar and your current items.
 **Add/Add after selection**	Creates a new item. When the **Items** list is empty, this button is labeled “Add”. Once an item has been added, it changes to “Add after selection.” If you select an item and then click this button, a new item will be added after the selected item.
 **Add before selection**	Adds a new item before the selected item.
 **Delete selection**	Deletes the selected item.
 **Move selection up**	Moves the selected item up one position.
 **Move selection down**	Moves the selected item down one position.
Label	Specifies the item label at run time. When you select an item, its label appears in the **Label** box. To rename an item, type the new name and then press **Enter**.

Parameters Tab (part 2 of 2)

ITEM	DESCRIPTION
Type	Specifies the type of information that will be displayed in the **Operator Information** window: **String**—Can be any arbitrary text string. **Real**—Can be any real number (includes decimal expressions, such as 1.2) within the defined range. If you select **Real**, the dimension boxes are enabled (see below). **Integer**—Can be any number (zero, 1, 2, and so on) within the defined range. **List**—A list of items from which the operator can select. This selection allow the designer to create a custom list for the operator.
Attribute	**None**—Operator can edit or delete the default entry. **Non-Editable**—Operator cannot edit text. **Non-Blank**—Operator must enter information. The entry cannot be blank.
Default Value	Specifies a default value displayed in the **Operator Information** data entry box. ***Note*** *The operator can change units when entering data, but MPT writes the data to the data file in the units selected here.*
Dimension *(Real Type only)*	Specifies the dimension for the **Default Value**.
Minimum/Maximum *(Real and Integer Types only)*	Enables the minimum/maximum limit to be enforced at runtime. ***Note*** *If the operator tries to enter a value outside the limits, the displayed value will revert to last valid limit value.*
Inclusive *(Real and Integer Types only)*	When selected, includes the specified minimum/maximum limit value. If this checkbox is cleared, the value you enter can be close to but cannot equal the limit value.
Selections *(List Type only)*	This is a toolbar identical to the Items toolbar. It allows you to create, delete, and choose selections available for the custom list.
Description *(List Type only)*	Allows you to edit the default list names generated by the Selections toolbar.

Destination tab

See "Destination tab" on page 270.

General tab

See "General tab" on page 202.

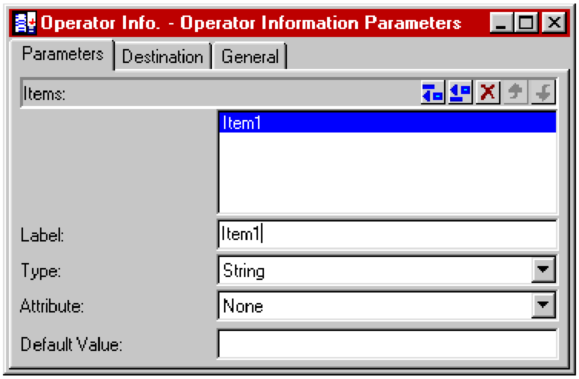
More About the Operator Information Process

Operator Information process example

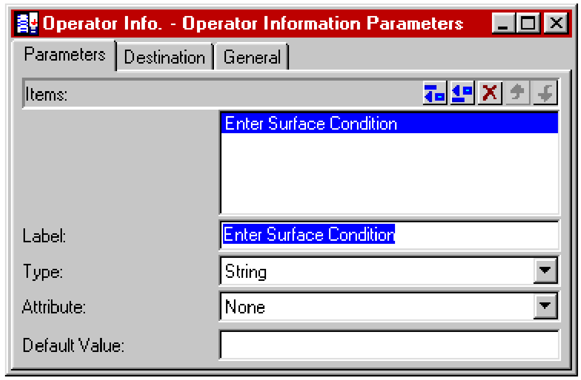
Assume you want to define an **Operator Information** process that requires the operator to enter information about the physical specimen condition before the data acquisition phase of the test starts. Also assume you want to save this information to a data file labeled "Surface Condition."

Perform the following steps to define the **Operator Information** process:

1. On the **Parameters** tab, click the **Add** button to create **Item1** in the **Items** list. In the **Type** list, click **String**.



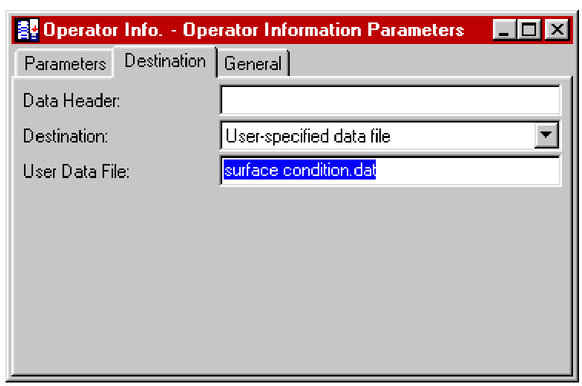
2. In the **Label** box, type **Enter Surface Condition**, and then press **Enter**.



3. Click the **Destination** tab.

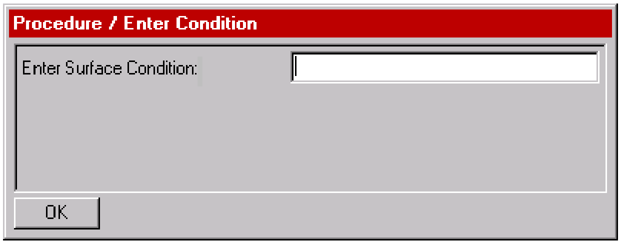
   A. In the **Destination** list, click **User-specified data file**.

   B. In the **User Data File** box, type **Surface Condition**, and then press **Enter**.

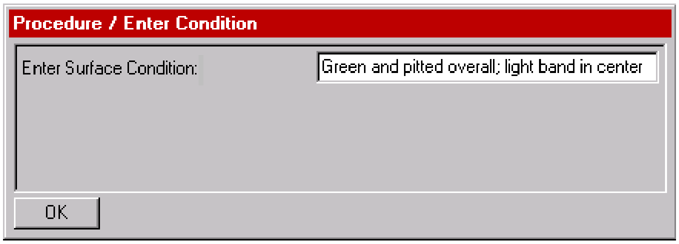


Process output

Now, the test will pause when the **Operator Information** process starts, and the Operator Information runtime window will appear. The data entry box will be labeled **Enter Surface Condition**.

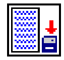


When the test is run, the operator will enter the condition of the physical specimen:



When the operator finishes entering the information and clicks **OK**, the information is written to a separate file (named **Surface Condition**) that is stored in the MPT specimen, and the procedure resumes with the next process.

How to Define an Operator Information Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Parameters** tab, and then click  to create an item.
4. In the **Label** box, type the text you want to appear when the runtime window is displayed.
5. In the **Type** list, click the item type, and then click the **Attribute**.
6. In the **Dimension** list, click the dimension (real types only).
7. In the **Selections** panel, add, arrange, and label custom list items (list types only).
8. In the **Default Value** box, type a default value.
9. Enter **Minimum** and **Maximum** values (real and integer types only).
10. Repeat steps 3 - 9 for each item you want to define.
11. Click the **Destination** tab.
12. Type a description of the data file in the **Data Header** box.
13. In the **Destination** list, click the destination for the acquired data. (If you click **User-specified data file**, enter a file name in the **User Data File** box.) See "Data destination" on page 268.
14. Click the **General** tab.

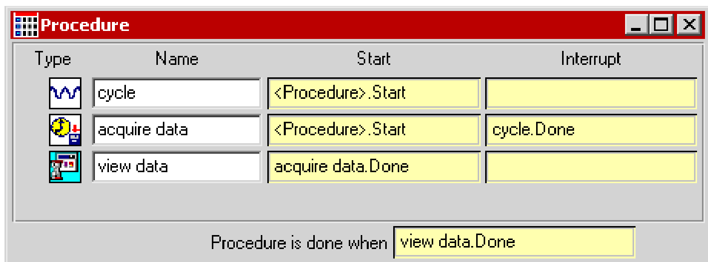
15. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
16. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
17. In the **Execute Process** box, type number of times you want the process to execute.
18. Optional—In the **Counter Type** list, click a counter type.
19. Optional—If you enabled a counter, type a **Counter Label**.
20. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

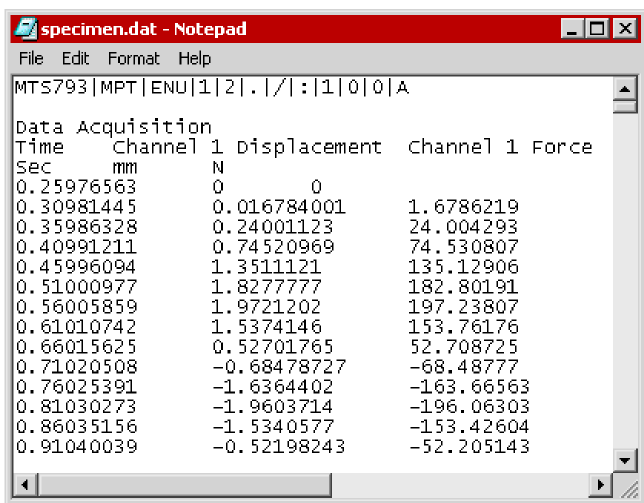
Start Application

With the **Start Application** process, you can launch another application when the **Start Application** is triggered, and specify arguments suitable for that application.

Example

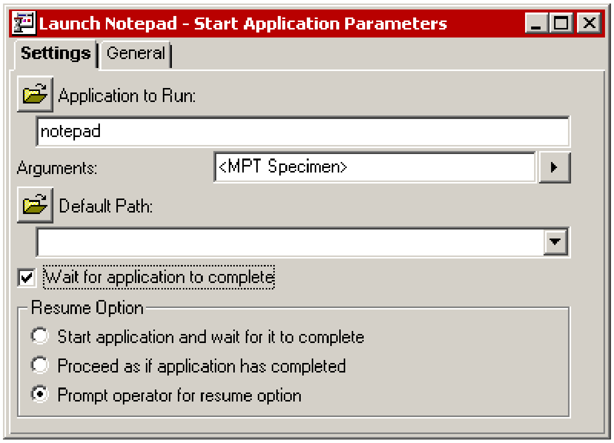
Suppose you want to create a test in which the procedure automatically displays data when the test is complete. With the **Start Application** process, you can invoke the Notepad application to display the specimen data file that contains the data acquired during the test, as shown below.





Start Application Parameter Descriptions

Settings tab



Settings Tab (part 1 of 3)

ITEM	DESCRIPTION
Application to Run	Specifies the application that will be launched. If you wish to run an application registered by Microsoft or available in the system path, you do not have to specify a full path because the process will find it automatically. For instance, if you want to select Microsoft Excel, you can simply enter “excel”. To launch Notepad, enter “notepad”; to launch Microsoft Word, enter “winword”. To run other applications, enter the full path to the application in this box, or browse for the application to find its path.

Settings Tab (part 2 of 3)

ITEM	DESCRIPTION
Arguments	Specifies an input or parameter to the selected application. The argument may be anything appropriate for the application, like a file name, command, etc. The arguments can include predefined placeholders that will be substituted with actual information when the process executes. The arrow button displays a menu of the available placeholders, as shown:  Selecting a placeholder will insert it into the arguments at the current position. For instance, if you want to display the contents of the specimen.dat file in the current MPT specimen, you would enter “notepad” as the Application to Run, and select “MPT Specimen” in the arguments list (MPT Specimen will appear between angle brackets in the entry box), and add a back slash character followed by “specimen.dat”. You could also simply enter “<MPT Specimen>\specimen.dat”. When you enter arguments directly, you must observe standard command line syntax.
Default	Specifies the default path to the selected argument. You can enter the default path by typing the desired path in the entry box, by browsing to the desired location, or by selecting one of the predefined directory locations in the Default Path list, as shown below: 
Wait for application to complete	When enabled, the process will wait until the application has completed before completing itself (becomes it sends out its “done” trigger). When disabled, the process will complete immediately after starting the specified application.

Settings Tab (part 3 of 3)

Item	Description
Resume Options	Defines how the process will resume when the procedure has been stopped and unlocked while waiting for the selected application to complete. **Start application and wait for it to complete.** This option causes the process to restart the application and complete when the application has completed. **Proceed as if application has completed.** This option causes the process to complete immediately. **Prompt operator for resume option.** This option displays a message which causes the operator to either: • Restart the application or • Proceed as if the application has completed.

General tab

See “General tab” on page 202.

How to Define the Start Application Process

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon in your Group or Procedure table.
3. Click the **Settings** tab.
4. Select the **Application to Run**, **Argument**, and **Default Path** as desired.
5. Optional—wait for the application to complete.
6. Click the **General** tab.
7. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
8. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
9. In the **Execute Process** box, type number of times you want the process to execute.
10. Optional—In the **Counter Type** list, click a counter type.
11. Optional—If you enabled a counter, type a **Counter Label**.
12. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

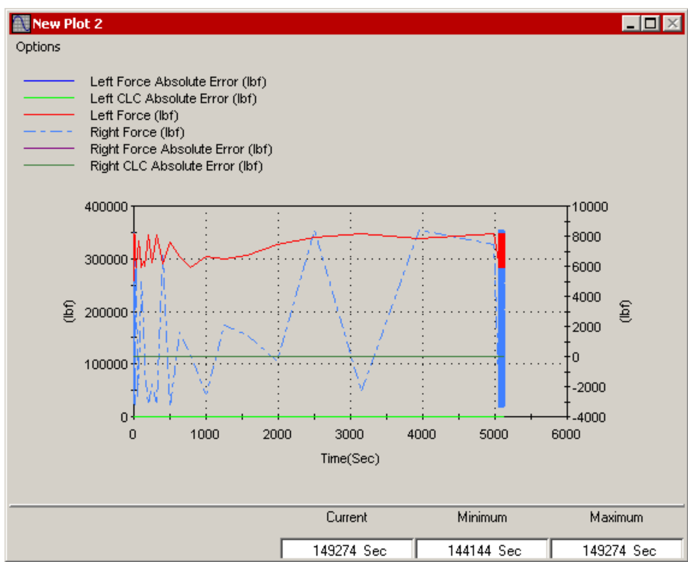
Data Display

The **Data Display** process acquires data on the signals you specify, and then plots the data on-screen during the test.

With this process, you can view events as they occur on multiple data channels. This process acquires data each time a selected signal changes a specified amount. This process must run in parallel with a command process; it cannot acquire data unless the command process is performing an action that it can measure.

Note *The **Data Display** process does not store data to a disk. The data that is acquired is used for display only.*

Within a single **Data Display** process, you can define any number of plot configurations, each of which can be enabled or disabled for the test. When the test starts, a plot window (similar to the Station Manger Scope window) appears on-screen for each enabled plot configuration.



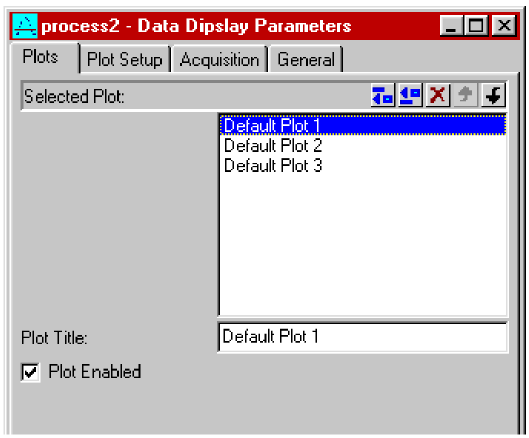
Plot Window Example

Data Display Parameter Descriptions

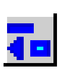
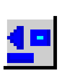
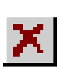
Plots tab

This window lets you create, delete, and rearrange plots using its toolbar icons.

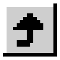
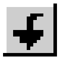
Note *To save the position of the plot window(s), you must save the procedure ('position' refers to the location of the plot on your PC monitor). If you move the plot window from its default position and exit the procedure without saving, the plot window will appear in its default (or last saved) position.*



Plots Tab (part 1 of 2)

ITEM	DESCRIPTION
Plots:	Displays the **Data Display** process toolbar and your current plots.
 **Add/Add after selection**	Creates a new plot. When the **Plots** list is empty, this button is labeled "Add". Once an item has been added, it changes to "Add after selection." If you select a plot and then click this button, a new plot will be added after the selected item.
 **Add before selection**	Adds a new plot before the selected item.
 **Delete selection**	Deletes the selected plot.

Plots Tab (part 2 of 2)

ITEM	DESCRIPTION
 **Move selection up**	Moves the selected plot up one position.
 **Move selection down**	Moves the selected plot down one position.
Plot Title	Specifies the plot window title.
Plot Enabled	Displays the plot when the test is run. ***Note*** *Only plots that are enabled are displayed on-screen during the test.*

Plot Setup tab

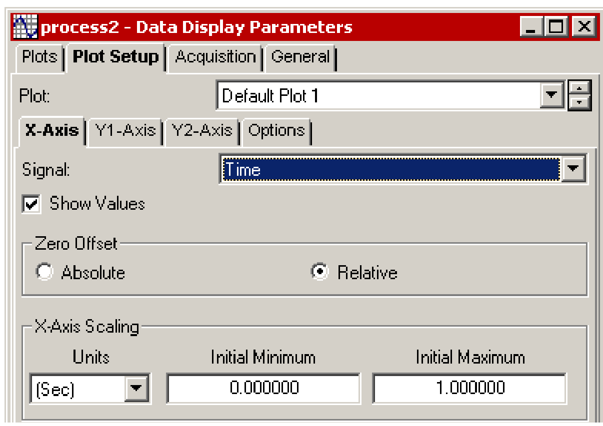
Double-click the process icon to open the process parameters window.

This window lets you customize the axes for each plot. Characteristics for each plot are defined separately. Plot characteristics are specified on four tabs: **X-Axis**, **Y1-Axis**, **Y2-Axis**, and **Options.**

Click each tab to access a different set of plot parameters.

Plot Setup Tab (part 1 of 9)

ITEM	DESCRIPTION
Plot	Specifies the plot being configured.
X-Axis Subtab	Determines settings for the horizontal or x-axis.



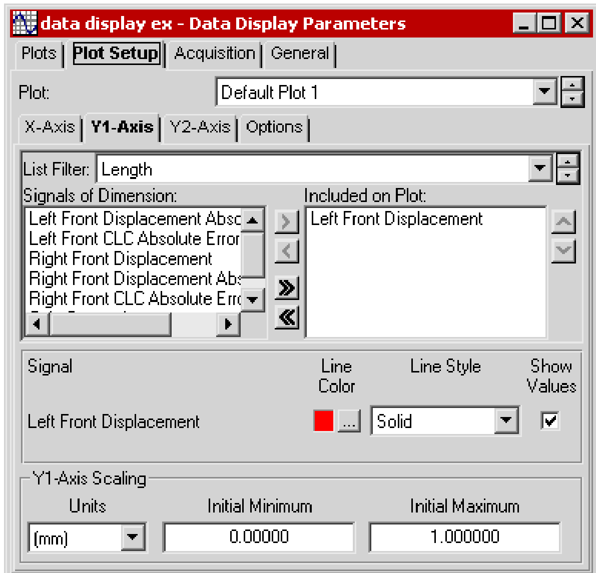
ITEM	DESCRIPTION
Signal	Specifies the signal for the x-axis of the plot.
Show Values	Displays the selected signal values in a table in the **Plots** window.

Plot Setup Tab (part 2 of 9)

ITEM	DESCRIPTION
Zero Offset	***Note*** *This control is most useful for continuously increasing signals, such as time, segment count, etc.* Determines the starting point: • Absolute: Plotted data begins where the output begins • Relative: Plotted data begins at zero Example: Suppose the Test Procedure includes a Cyclic Command process that precycles the specimen for 30 seconds prior to starting another Cyclic Command process that applies test cycles to the specimen, and a parallel Data Display process that displays a plot of the test cycles. In this scenario, if **Zero Offset** is set to **Absolute** and the x-axis signal is time, the plot will begin at 30 seconds, leaving a gap from 0 to 30 seconds on the plot. If **Zero Offset** is set to **Relative**, the plot will begin at 0 seconds.
Units	Specifies the units for the x-axis.
Initial Minimum/ Maximum	Sets the initial x-axis maximum and minimum values. The plots will automatically rescale if necessary.

Plot Setup Tab (part 3 of 9)

ITEM	DESCRIPTION
Y1/Y2 Axis Subtabs	Determines the settings for the left and right vertical axes. Selections for the **Y2-Axis** are the same as the **Y1-Axis**.

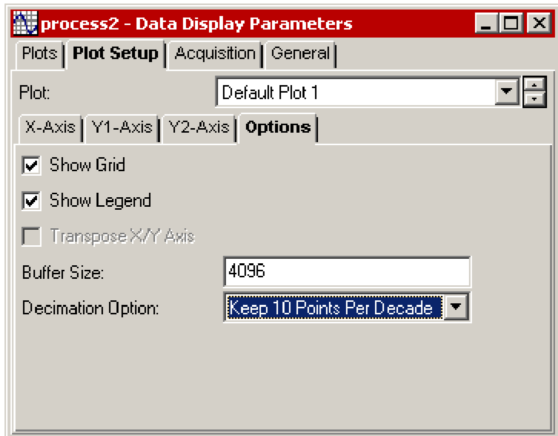


List Filter	Lists categories of signals you can plot, based on dimension. For example, if you select **Length**, the **Signals of Dimension** list will contain only signals with a length dimension. The **List Filter** ensures the signals you select to plot on a given axis share the same dimension.

Plot Setup Tab (part 4 of 9)

ITEM	DESCRIPTION
Signals of Dimension/ Included on Plot	Displays signals on the plot with unit dimensions matching the current List Filter selection. Move signals to the **Included on Plot** list if you want to plot them on this axis. ***Note*** *Only signals with the same dimension can be plotted on the same axis.*
Signal	Displays the signals that were added to the **Included on Plot** list. ***Note*** *This control, and those listed below it, are not displayed until you move a signal to the **Included on Plot** list.*
Line Color	Specifies the line color.
Line Style	Specifies the line style.
Show Values	Shows or hides the selected signal values in a table below the plot (in the plot window).
Units	Specifies the units for the Y1 or Y2 axis.
Initial Minimum/ Maximum	Sets the Y1 or Y2 axis initial maximum and minimum values. The plot will autoscale if necessary.

Plot Setup Tab (part 5 of 9)

ITEM	DESCRIPTION
Options Subtab	Lets you define visual characteristics of the plot and buffer size, and choose which data is displayed. 
Show Grid	Hides or displays the grid in the plot window.
Show Legend	Hides or displays the legend in the plot window.
Transpose X/Y Axis	Transposes the X and Y1 axes. ***Note*** *This option is disabled if both the Y1 and Y2 axes are used.*
Buffer Size	Sets the size of the display buffer, which ranges between 4096 and 16,384. ***Note*** *When the buffer is filled, room for new data in the buffer is based on the Decimation Option (data is deleted according to the current Decimation Option).*

Plot Setup Tab (part 6 of 9)

ITEM	DESCRIPTION
Decimation Option	Decimation refers to discarding a portion of the data in the display buffer when it fills. Each option controls the flow of data through the display buffer in a unique way, and creates a corresponding type of display on the plot. Each option follows the same general process: • The buffer fills with data • The data is decimated (reduced) according to the option selected • The retained data is shifted to the front of the buffer to form a contiguous block • The process is repeated **Every Other Point**—This option causes the buffer to accept acquired data until it fills, perform a decimation of every other point, shift data, and repeat. This option decreases the resolution of the initial data as a function of the number of times the buffer fills. *First pass:* When the buffer fills for the first time, it accepts every point of acquired data until the buffer is full, then performs a decimation by discarding every other point in the buffer. The retained points are shifted to the front of the buffer to form a contiguous block, leaving the back half of the buffer empty and ready to receive new data. *Second pass:* The buffer fills the second time as before until full, then performs another decimation and data shift. *Subsequent passes:* The process is repeated in subsequent passes in the same way until stopped.

Plot Setup Tab (part 7 of 9)

ITEM	DESCRIPTION
	Reduced Rate—This option causes the buffer to accept acquired data until it fills, perform a decimation of every other point, shift data, accept new data at one-half the rate used on the previous pass (every point on the first pass, every second point on the second pass, every fourth point on the third pass, etc.), and repeat. This option results in data resolution that is constant regardless of the number of times the buffer fills. *First pass:* When the buffer fills the first time, it accepts every point of acquired data until the buffer is full, then performs a decimation to the entire buffer by removing every other point. The retained points are shifted to the front of the buffer to form a contiguous block, leaving the back half of the buffer empty and ready to receive new data. *Second pass:* When the buffer fills the second time, it accepts every other point of acquired data until the buffer is full, then performs a decimation to the entire buffer by removing every other point. The retained points are shifted to the front of the buffer as before. *Third pass:* When the buffer fills the third time, it accepts every forth point of acquired data until the buffer is full, then performs a decimation to the entire buffer by removing every other point. The retained points are shifted to the front of the buffer as before. *Subsequent passes:* The process is repeated until stopped, with the factor used by the buffer to accept new data doubling with each subsequent pass, and the decimation always removing every other point in the buffer.

Plot Setup Tab (part 8 of 9)

ITEM	DESCRIPTION
	Discard Oldest Points—This option causes the buffer to accept acquired data until it fills, perform a decimation that discards the oldest points in the buffer, shift data, and repeat. If the selected signal is something that is always increasing such as time, the result is a scrolling effect. This method of discarding points prevents the plot from redrawing on any given update cycle if the number of points acquired is relatively small. *First and subsequent passes:* On any given pass, the buffer accepts every point of acquired data until the buffer is full, then performs a decimation by discarding the oldest points in the buffer. The retained points are shifted to the front of the buffer to form a contiguous block, leaving the back half of the buffer empty and ready to receive new data. The number of points discarded is the greater of the number of points that have been acquired per update (internally determined time) or 10% of the buffer size. As the old points are discarded, new points are acquired. The number of points acquired is dependent on the acquisition rate (which varies depending on the acquisition signal and increment value).

Plot Setup Tab (part 9 of 9)

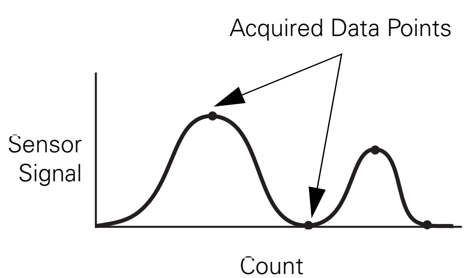
ITEM	DESCRIPTION
	Keep 10 Points Per Decade—This option causes the buffer to accept acquired data until it fills, perform a decimation that retains ten points per decade of data, shift the data, and repeat. Unlike the other decimation options, this method does not discard points retained from previous passes, and is especially suited to viewing trends in peak/valley data. *First pass:* Suppose the buffer initially fills with 4096 points of data. In this case: • 10 Points are retained in the first decade (1-10) • 10 Points are retained in the second decade (11-100) • 10 Points are retained in the third decade (101-1000) • 6 Points are retained in the forth decade (1001-10,000) In this example, the buffer retains 36 points on the first pass and discards the remainder. Only 6 points are retained from the forth decade because the buffer holds only 4096 points, and the next point scheduled for retention is outside of the points stored in the buffer. After the decimation, the retained points are shifted to the front of the buffer to form a contiguous block. *Second and subsequent passes:* In subsequent passes, the buffer iterates between filling and decimating points until the 4 remaining points in the fourth decade are retained. After that, the buffer begins acquiring 10 points in the fifth decade, etc. The passes continue on in the same manner until the process is stopped. Because the interval between each desired point is logarithmic, fewer points are retained with each successive pass.

Acquisition tab

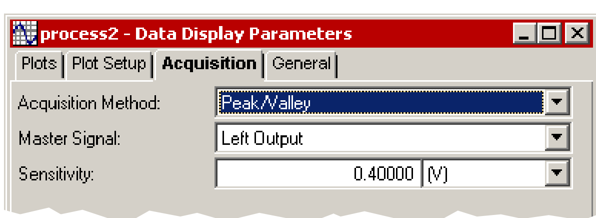
This window lets you select the method with which the process acquires data to plot.

Acquisition Tab (part 1 of 3)

ITEM	DESCRIPTION
Acquisition Method:	
Peak/Valley	The **Peak/Valley Acquisition Method** acquires data on slave signals when the software detects a peak or valley in the specified master signal. When selected, the application: • Permits only channel count signals to display on the x-axis (the signal list on the **X-Axis** subtab of the **Plot Setup** tab will display only channel count signals). • Displays two traces per selected y-axis signal (selected on the **Y1-Axis** and **Y2-Axis** subtabs of the **Plot Setup** tab); one displays peaks, the other displays valleys. • Selects logarithmic scaling on the x-axis by default. You can change the scaling in the **Properties** window (right-click on the plot to access the **Properties** window).



When you select the **Peak/Valley Acquisition Method**, you must define the master signal and sensitivity.



ITEM	DESCRIPTION
Master Signal	Selects a signal that determines when data is acquired.

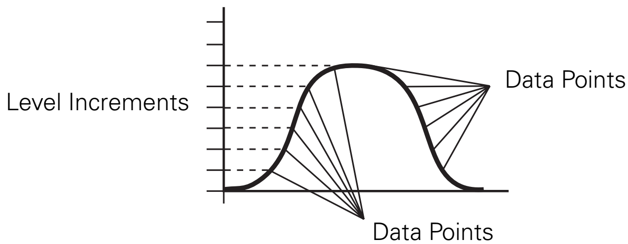
Acquisition Tab (part 2 of 3)

Item	Description
Sensitivity	Specifies how much the master signal must change to detect a peak or valley. Setting the sensitivity too low may cause signal noise to be recognized as peaks and valleys. Setting the sensitivity too high may cause low amplitude signals to be missed.

Acquisition Method:

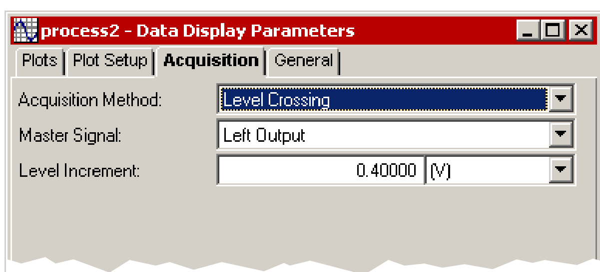
Level Crossing

The **Level Crossing Acquisition Method** acquires data each time the selected master signal changes by a specific amount.



Data is acquired each time the signal moves by the specified level increment

When you define a **Level Crossing Acquisition Method**, you must select the master signal and define the level increment at which data is acquired.



Master Signal

Selects a signal that determines when data is acquired.

Note *When you select **Time** you get timed data acquisition, when you select any of the feedback signals you get level-crossing data acquisition.*

Acquisition Tab (part 3 of 3)

ITEM	DESCRIPTION
Level Increment	Defines how much the signal must change before acquiring data. Initially a default level increment is displayed, which for **Time** is 50 mSec, for **Count** is 1 segment, and for feedback signals such as **Actuator Displacement** is a percentage of the signal's full scale value. Enter the desired level increment and select appropriate units. The level increment must be greater than zero.

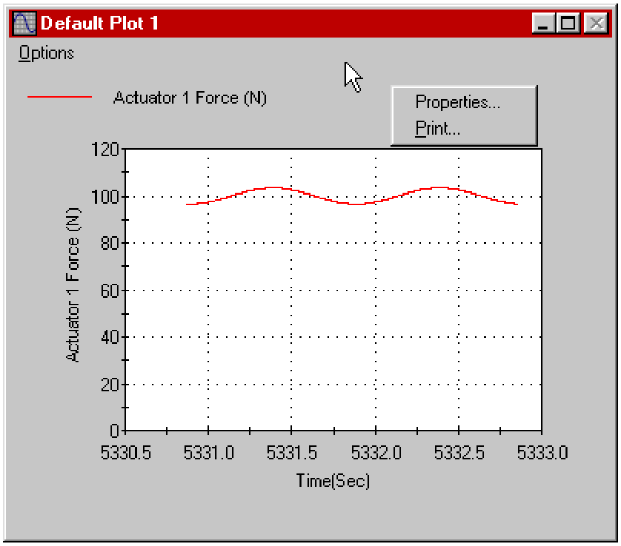
General tab

See "General tab" on page 202.

Additional Plot Properties

Accessing the Properties and Print menu

When you right-click on the plot a menu will appear with two selections, **Properties** and **Print**, as shown below.



If you select **Print**, the **Page Setup** window will appear. Complete the desired selections and click **OK**. The plot will be sent to the printer.

If you choose **Properties**, the **Chart Properties** window will appear.

Chart Properties Axes and Chart Properties General Tabs

These windows (not shown) allow you to define various properties of the x and y axes, as well as colors and fonts for each plot. The selections in these windows take effect on the plot when you press **OK**.

Zooming Options

You can zoom in on a plot's axis by pressing the **Shift** key while holding down the left mouse button, and then moving the mouse to the area of interest and releasing the button.

You can zoom in on a part of the trace by pressing the **Ctrl** key while holding down the left mouse button, and then moving the mouse to the area of interest and releasing the button.

Return to default

To return to the default plot display, press "**r**." This will remove all interactive scaling and zooming.

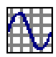
Working with the Data Display Process

Data Display Characteristics

This process is used to view data. The data will not be saved to disk.

- Plots can be enabled and/or disabled.
- Plot settings can be saved.
- The plot will appear when the procedure is locked. Likewise, it will disappear when the procedure is unlocked.

Note *To save the position of the plot window(s), you must save the procedure ('position' refers to the location of the plot on your PC monitor). If you move the plot window from its default position and exit the procedure without saving, the plot window will appear in its default (or last saved) position.*

1. Drag the  icon from the **Process Palette** to your Group or Procedure table.
2. Double-click the process icon on the table.
3. Click the **Plots** tab.
4. Select an existing plot or create a new plot.
   A. Select an existing plot—Highlight the desired plot.
   B. Create a new plot—Click one of the add plot buttons .
5. Enter a name for the plot in the **Plot Title** box.
6. Click the **Plot Enabled** check box to enable the plot. (If the box is not checked the plot will be disabled.)
7. Click the **Acquisition** tab.
8. Select an **Acquisition Method**.

9. If you select the **Peak/Valley Acquisition Method**:

   A. Select a **Master Signal** from the pull down menu.

   B. Select a **Units** type for the **Sensitivity**.

   C. Enter the desired value for the **Sensitivity**.

   If you select the **Level Crossing Acquisition Method**:

   A. Select a **Master Signal** from the pull down menu.

   B. Select a **Units** type for the **Level Increment**.

   C. Enter the desired value for the **Level Increment**.

10. Click the **Plot Setup** tab.

11. Click the **X-Axis** tab.

12. Select the desired **Signal**.

13. Check the **Show Values** check box if you want the values to be displayed on the plot.

14. Select the desired **Zero Offset** setting.

15. Select the desired **Units**.

16. Enter the desired **Initial Minimum** and **Initial Maximum** values.

17. Click the **Y1-Axis** tab.

18. In the **List Filter** box, select the desired dimension category.

19. Double-click the individual signals you want to appear on the **Y1-Axis**.

20. For each signal included on the plot, define the **Line Color** and **Line Style** controls as desired, and enable the **Show Values** control as desired.

21. Click the **Y2-Axis** tab.

22. In the **List Filter** box, select the desired dimension category.

23. Double-click the individual signals you want to appear on the **Y2-Axis**.

24. For each signal included on the plot, define the **Line Color** and **Line Style** controls as desired, and enable the **Show Values** control as desired.
25. Click the **Options** tab.
26. Check the **Show Grid** box if you want the grid to appear on the plot.
27. Check the **Show Legend** box if you want the legend to appear on the plot.
28. Check the **Transpose X/Y Axes** if you want to exchange the position of the axes.
29. Enter the desired buffer size.
30. Select the desired **Decimation Option**.
31. Click the **General** tab.
32. In the **Name** box, type a name for this process. (You can also name the process in the Procedure table.)
33. Select the **Process Enabled** check box. (If this box is cleared, the process will not execute.)
34. In the **Execute Process** box, type number of times you want the process to execute.
35. Optional—In the **Counter Type** list, click a counter type.
36. Optional—If you enabled a counter, type a **Counter Label**.
37. Close the parameters window, and set the desired **Start** and **Interrupt** signals for the process in the Group or Procedure table.

Defining Additional Chart Properties

1. Right-click on the plot. The **Chart Properties** window will appear.
2. Click the **Axis** tab.
3. Select the desired **Axis** you wish to format.
4. Check the desired **Auto** formatting boxes and enter corresponding values for each.
5. Click **Save Defaults** if you want to save these settings as default settings.
6. Click the desired colors and font selections on the **General** tab.

Important *The **Graph Area Color** and the **Background Color** must be different colors. Your text will not appear if they are the same color.*

Fatigue Monitoring Application

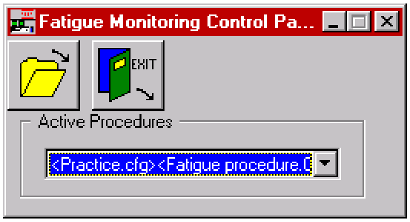
The Fatigue Monitoring application plots fatigue data (received from an MPT **Fatigue** process) in real-time. While the **Fatigue** process is running, you can use the Fatigue Monitoring application to adjust data limits in real-time.

When you run a procedure that contains a **Fatigue** process, the **Fatigue Monitoring Control Panel** appears minimized on the Windows taskbar. To display the **Fatigue Monitoring Control Panel** and the rest of the Fatigue Monitoring windows, click the icon on your Windows taskbar.

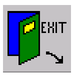
About the Fatigue Monitoring Control Panel

The **Fatigue Monitoring Control Panel** allows you to choose which procedure for which to display monitoring information. It also includes a toolbar that allows you to open fatigue data files and exit the Fatigue Monitoring application.

Note *When you run a test that includes a Fatigue process, the Fatigue Monitoring application runs in a minimized state.*



Fatigue Monitoring Control Panel

ITEM	DESCRIPTION
	Opens the **Open Fatigue Data File** window. You use this window to open an existing fatigue data file and display its contents. If a data file has been opened before, the display will be updated to show the new fatigue data.
	Quits the Fatigue Monitoring application. ***Note*** *If data has already been received, a message will prompt you to confirm the Exit command.*
Active Procedures	Specifies a procedure to display.

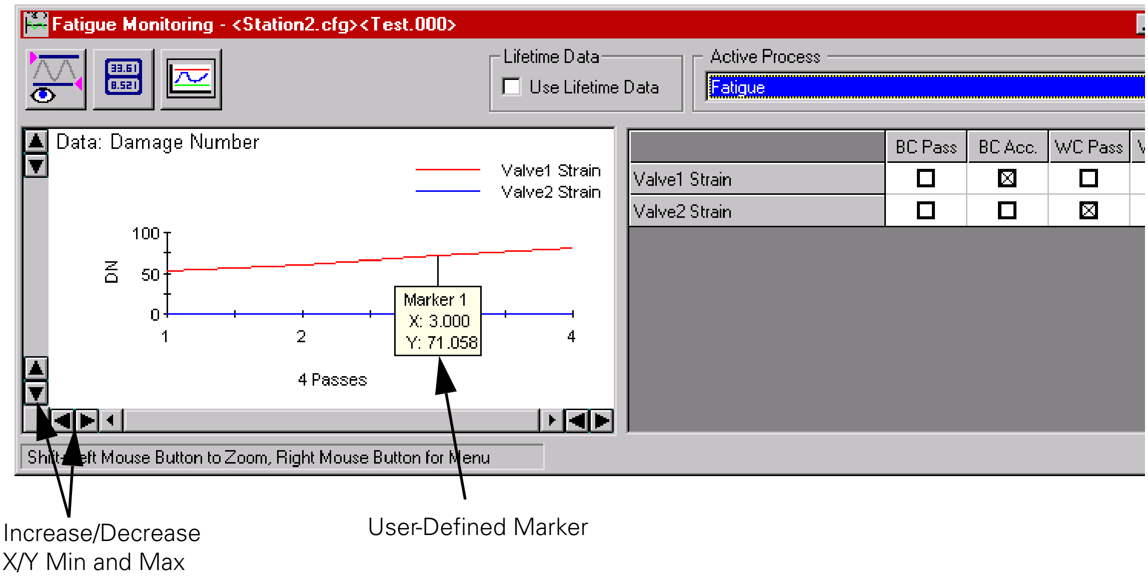
About the Fatigue Monitoring Window

The **Fatigue Monitoring** window consists of your fatigue plot, a channel selection box, and a toolbar. If you are running a new procedure, the **Procedure File View** displays. If you open an existing data file, the **Data File View** displays.

The current damage values and limit settings are displayed on the data plot. If you are creating a new process, the system will plot only the first channel in the list by default.

Note *If just one fatigue value has been plotted, and you have specified limits, the limits are also displayed on the plot. If multiple values are plotted, the limits are not displayed.*

Procedure File View

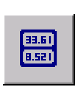
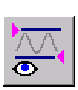
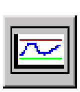


Note *A left mouse-click on the trace will display it coordinates.*

Controls

The **Fatigue Monitoring** main window controls are described in the following table.

Fatigue Monitoring Window (part 1 of 2)

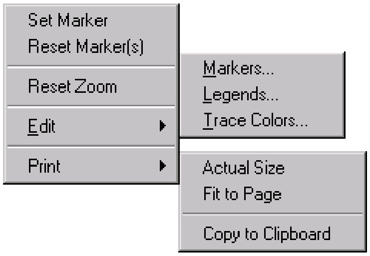
ITEM	DESCRIPTION
	Displays the **Numerical Display** window (see "About the Numerical Data Window" on page 439).
	Displays the **Limit Settings** window (see "About the Limit Settings Window" on page 436).
	Shows or hides the channel value information.

Fatigue Monitoring Window (part 2 of 2)

ITEM	DESCRIPTION
Active Process	Selects the process or file to view. You can have multiple processes or multiple files per window.
Lifetime Data	Displays "percentage of lifetime" instead of damage number.
Channel Selection	Specifies the damage value plotted for a particular signal. Up to six different values can be plotted at the same time. • Clicking the **Pass/Acc.** button at the head of a column selects the top six values in that column. • Clicking on the channel name selects all damage values for this channel. • To select values randomly (for example, Channel 1 BC Pass and Channel 2 WC Acc), press the **Ctrl** key while clicking on the check boxes.

Context menu

Other commands are available on the Fatigue Monitoring application context menu. To access the menu, right-click the fatigue plot.



Fatigue Monitoring Context Menu

Item	Description
Set Marker	Places a graphic marker on your plot window. After you enable the **Set Marker** control, click on the desired location of the desired trace to place the marker at that location.
Reset Zoom	Returns a zoomed plot to the default view.
Edit	**Markers**—Opens the **Marker Editor** window that can be used to specify marker attributes **Legends**—Opens the **Legend Editor** window that can be used to specify legend attributes. **Trace Colors**—Opens the **Trace Color Editor** window that can be used to specify the color for each signal trace.
Print	**Actual Size**—Prints the window as is. **Fit to Page**—Scales and prints the window on a single page. **Copy to Clipboard**—Copies the window to the clipboard.

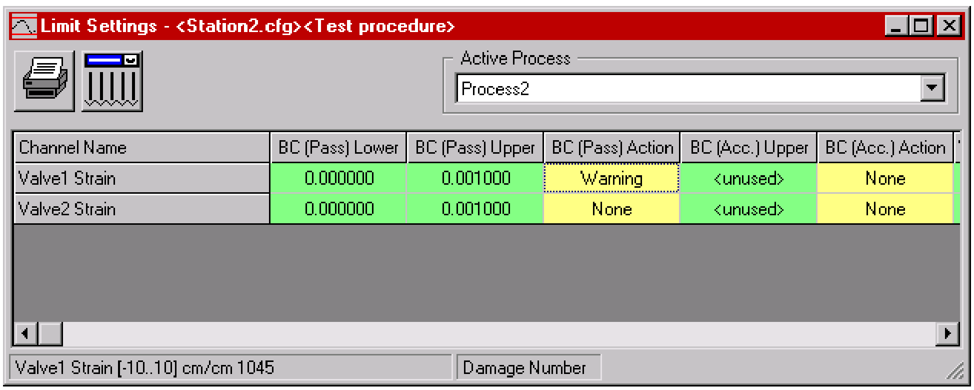
Zooming data

If you want to zoom in on plot data, hold the shift key, and drag a selection box around the area of interest. To reset the display, right-click the plot, and then click **Reset Zoom**.

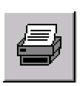
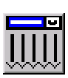
About the Limit Settings Window

All limit adjustments are made within this window. For each signal, you can select a best and worst case:

- Upper Limit (Pass)
- Lower Limit (Pass)
- Limit Action (Pass)
- Upper Limit (Accumulated)
- Limit Action (Accumulated)



Limit Settings Window (part 1 of 2)

ITEM	DESCRIPTION
	Prints the current limit settings in a tabular form.
	Applies the selected action to all channels, and redisplays the **Fatigue Monitoring** window.
Active Process	Specifies the current process or file.

Limit Settings Window (part 2 of 2)

ITEM	DESCRIPTION
Upper/Lower Limit values for channels	For a data file: Displays the limit values. For a procedure file: Displays editable limit values. To edit them, double-click a value, then type the new value. Press the **Enter** key to accept changes.
Limit Action	For a data file: Displays the action settings. For a procedure file: Displays editable action settings. To change a setting, double-click it. A list appears with the following options: **None**—Limit is disabled. **Warning**—A warning message is issued and a message is written to the **MPT Specimen Log**. **Stop**—The process stops and a message is written in the **MPT Specimen Log**. **Trigger**—The process triggers another process.
Status bar	Displays the name, full scale and dimensions of the selected channel.

About best case/ worst case

The terms “worst case” and “best case” refer to the methods MPT uses to analyze each histogram bin. Each histogram bin has an associated cycle count that increments whenever a data value occurs within the bin boundaries. Therefore, when analyzing the damage content of a histogram, the MPT analysis assumes a maximum or minimum bin value for each cycle that is counted. The bin size (resolution) affects the best case/worst case answers. The larger the bin size, the larger the potential for errors in the damage calculations. In general, assuming maximum values for a bin results in a maximum damage (worst case) scenario. Likewise, assuming minimum values for a bin results in a minimum damage (best case) scenario.

Worst case

“Worst case” assumes that the cycles in the bin have the largest range and tensile mean (for strain life and generic stress life damage models) and that the cycles “hang” from the bounding hysteresis loop in a way that yields the maximum (tensile) mean stress (for strain life damage models only).

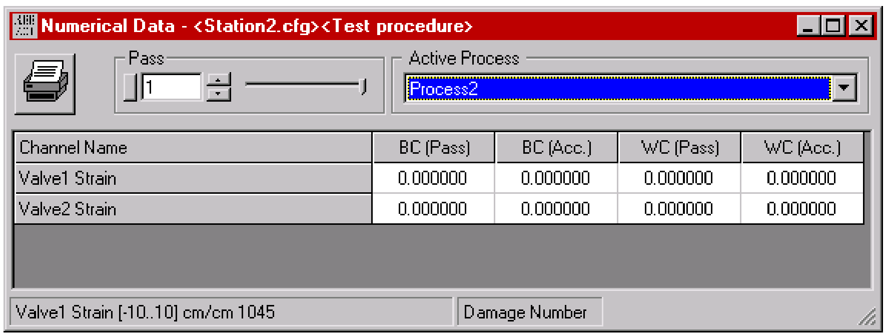
Best case

“Best case” assumes that the cycles in the bin have the smallest range and tensile mean (for strain life and generic stress life damage models) and that the cycles “hang” from the bounding hysteresis loop in a way that yields the minimum (tensile) mean stress (for strain life damage models only).

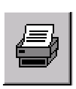
About the Numerical Data Window

The **Numerical Data** window displays the current or past values for all damage data for one pass. This view is updated with the data of a new pass only when the last pass is displayed. If you use the pass slider to look at a previous pass, this view will not change when new pass data is received. You can use the slider again or click on the pass number field to display the latest pass.

Red or purple numbers in the numerical data boxes indicate that these values are outside the defined limits. Red denotes “upper limit tripped” and purple denotes “lower limit tripped.”



Numerical Data Window

Item	Description
	Prints the current limit settings in a tabular form.
Pass	Selects the pass for which you want to display data. To show numerical data for a particular pass, enter the desired pass number, and then press return.
Active Process	Specifies the desired process or file.
Status bar	Displays the name, full scale and dimensions of the selected channel.

Trend Monitoring Application

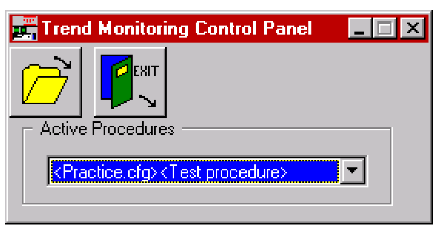
The Trend Monitoring application plots trend data (received from an MPT **Trend** process) in real-time. While the **Trend** process is running, you can use the Trend Monitoring application to adjust limits in real-time.

Note *Limit changes are transmitted back to the MPT application which does the actual limit-checking.*

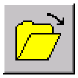
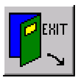
When you run a procedure that contains a **Trend** process, the **Trend Monitoring Control Panel** appears minimized on the Windows taskbar. To display the **Trend Monitoring Control Panel** and the rest of the Trend Monitoring windows, click the icon on your Windows taskbar.

About the Trend Monitoring Control Panel

The **Trend Monitoring Control Panel** allows you to choose which procedure for which to display trend information. It also includes a toolbar that allows you to open trend data files and exit the Trend Monitoring application.



Trend Monitoring Control Panel

ITEM	DESCRIPTION
	Displays the **Open Trend Data File** window: You use this window to open an existing trend data file and display its contents. If a data file has been opened before, the display will be updated to show the new trend data.
	Quits the Trend Monitoring application. ***Note*** *If data has already been received, a message will prompt you to confirm the Exit command.*
Active Procedures	Specifies a procedure to display.

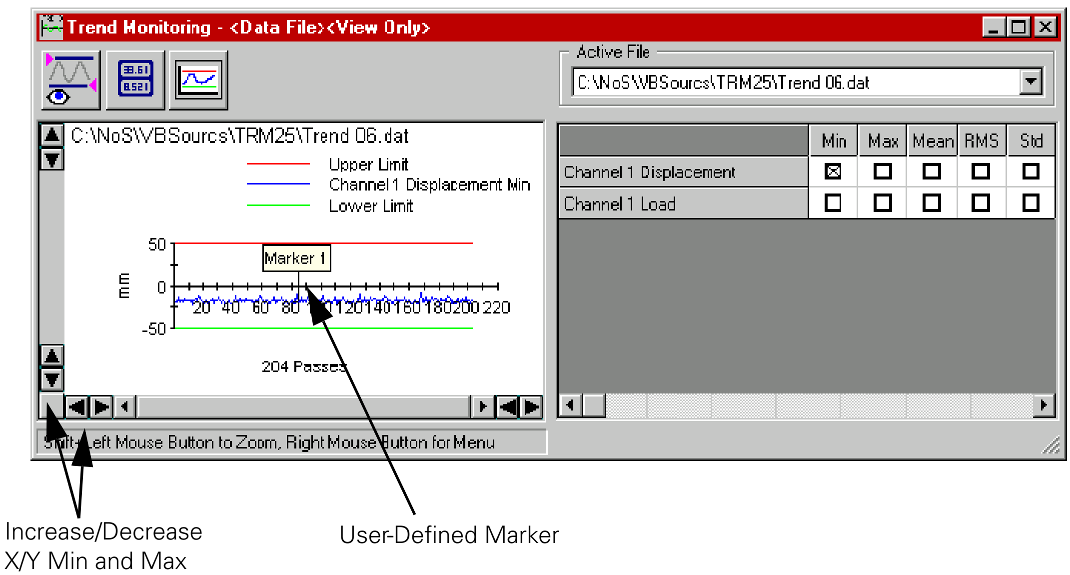
About the Trend Monitoring Window

The **Trend Monitoring** window consists of your trend plot, a channel selection box, and a toolbar. If you are running a new procedure, the **Procedure File View** displays. If you open an existing data file, the **Data File View** displays.

The current statistical values and the actual limit settings are displayed on the data plot. If you are creating a new process, the system will plot only the first channel in the list by default

Note *If just one statistical value has been plotted, and you have specified limits, the limits are also displayed on the plot. If multiple values are plotted, the limits are not displayed.*

Data File View

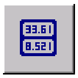
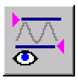
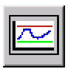


Note *A left mouse-click on the trace will display it coordinates.*

Controls

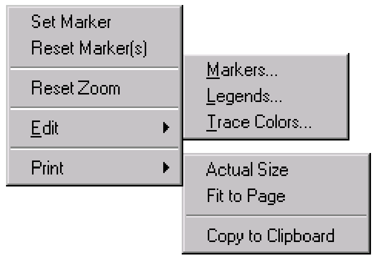
The **Trend Monitoring** main window controls are described in the following table.

Trend Monitoring Window

ITEM	DESCRIPTION
	Displays the **Numerical Display** window (see "About the Numerical Data Window" on page 447).
	Displays the **Limit Settings** window (see "About the Limit Settings Window" on page 445).
	Shows or hides the channel selection information.
Active Process	Selects the process or file to view. You can have multiple processes or multiple files per window.
Channel Selection	Specifies the statistical value plotted for that signal. Up to five different values (Min, Max, Mean, RMS, Std.) can be plotted at the same time. • Clicking the **Pass/Acc.** button at the head of a column selects the top six values in that column. • Clicking on the channel name selects all statistical values for this channel. (Note that the scaling of the Y-axis on the plot changes to percentage of full scale if more than one value is selected.) • If you want to select values randomly, press the **Ctrl** key while clicking on the check boxes.

Context menu

Other commands are available on the Trend Monitoring application context menu. To access the menu, right-click the trend plot.



Trend Monitoring Context Menu

Item	Description
Set Marker	Places a graphic marker on your plot window. After you enable the **Set Marker** control, click on the desired location of the desired trace to place the marker at that location.
Reset Zoom	Returns a zoomed plot to the default view.
Edit	**Markers**—Opens the **Marker Editor** window that can be used to specify marker attributes **Legends**—Opens the **Legend Editor** window that can be used to specify legend attributes. **Trace Colors**—Opens the **Trace Color Editor** window that can be used to specify the color for each signal trace.
Print	**Actual Size**—Prints the window as is. **Fit to Page**—Scales and prints the window on a single page. **Copy to Clipboard**—Copies the window to the clipboard.

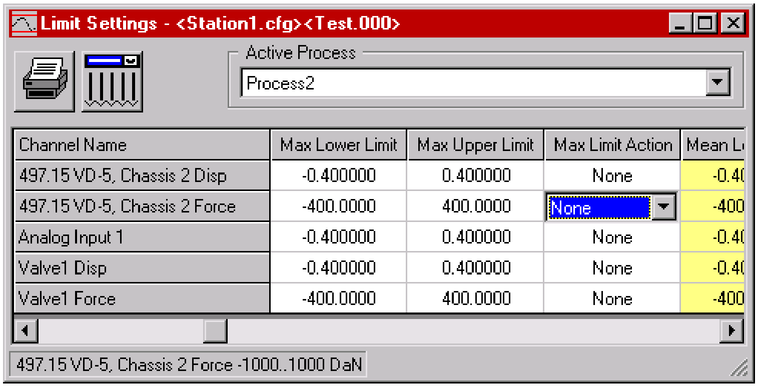
Zooming data

If you want to zoom in on plot data, hold the shift key, and drag a selection box around the area of interest. To reset the display, right-click the plot, and then click **Reset Zoom**.

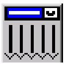
Process Descriptions

About the Limit Settings Window

All limit adjustments are made within this window. For each channel, you can select Lower Limit, Upper Limit, and Limit Action.



Limit Settings Window (part 1 of 2)

ITEM	DESCRIPTION
	Prints the current limit settings in a tabular form.
	Applies the selected action to all channels, and redisplays the **Trend Monitoring** window.
Active Process	Selects the desired process or file.

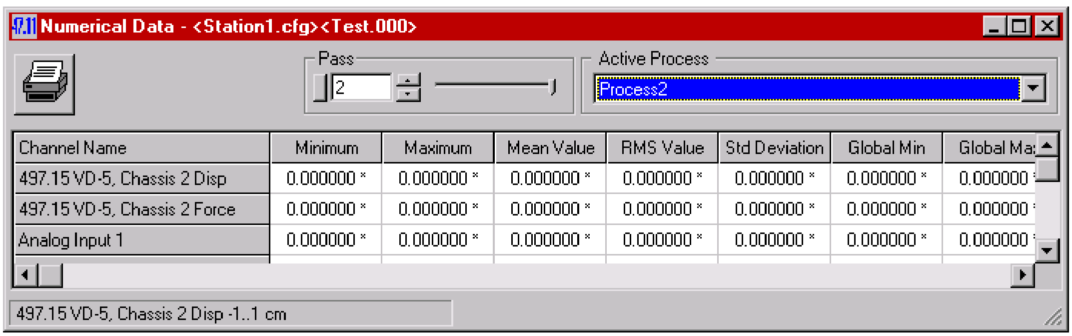
Limit Settings Window (part 2 of 2)

ITEM	DESCRIPTION
Upper/Lower Limit	For a data file: Displays the limit values. For a procedure file: Displays editable limit values. To edit them, double-click a value, then type the new value. Press the **Enter** key to accept changes.
Limit Action	For a data file: Displays the action settings. For a procedure: Displays editable action settings. To change a setting, double-click it. A list appears with the following options: **None**—Limit is disabled. **Warning**—A warning message is issued and a message is written to the **MPT Message Log**. **Stop**—The process stops and a message is written in the **MPT Message Log**. **Trigger**—The process triggers another process.
Status bar	Displays the name, dimensions, and full scale of the selected channel.

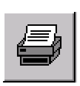
About the Numerical Data Window

This **Numerical Data** window shows the current or past values for all statistical data for one pass. This view is updated with the data of a new pass only when the last pass is displayed. If you use the pass slider to look at a previous pass, this view will not change when new pass data is received. You can use the slider again or click on the pass number field to display the latest pass.

Red or purple numbers in the numerical data boxes indicate that these values are outside the defined limits. Red denotes “upper limit tripped” and purple denotes “lower limit tripped.



Numerical Data Window

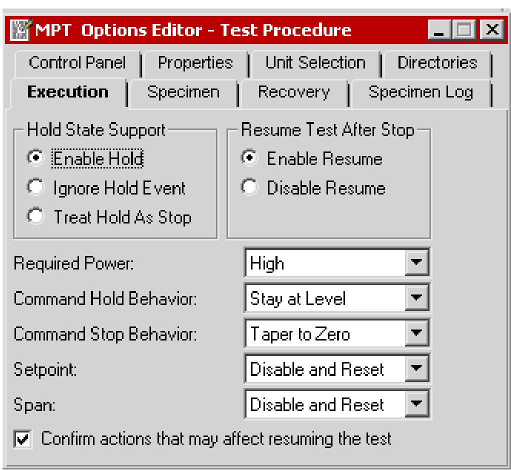
ITEM	DESCRIPTION
	Prints the current limit settings in a tabular form.
Pass	Selects the pass for which you want to display data. To show numerical data for a particular pass, enter the desired pass number, and then press return.
Active Process	Selects the desired process or file.
Statistical Values	Displays the name, full scale and dimensions of the selected channel.
Status bar	Prints the current limit settings in a tabular form.

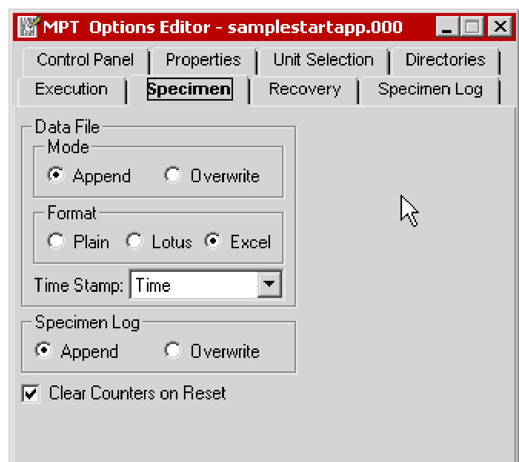
Appendix A
Default Templates

When you create a new test procedure with the MPT application, it inherits the process layout and **MPT Options Editor** settings defined by the default procedure template (default.000).

Note *As shipped from MTS, the default procedure template does not include process information, that is, the default procedure window is blank.*

In some instances, you may want to change the default template to suit your particular test environment.





All of the settings in the **MPT Options Editor** are saved in the default template (default.000). The settings in the **Execution** and **Specimen** tabs shown here are examples of preferences that MTS has set at the factory. You can customize the default template so new test procedures open with your own preferences.

If the default template is not found, the MPT application will use the built-in template to define new procedures. For more information, see "Template Error Messages" on page 452.

Customizing a Default Template

You can change the settings and processes defined by the default template to suit your particular test environment.

How to Customize the Default Template

1. In Station Manager, from the **Applications** menu, select MPT.
2. From the MPT file icon pull-down menu, click **Open Procedure**.
3. In the **Open Procedure** window, right-click the **default.000** file, and then click **Properties**.
4. In the **Properties** window, clear the **Read Only** check box, and then click **OK**.
5. In the **Open Procedure** window, double-click the **default.000** file to open it.
6. On the MPT control panel toolbar (or the **MPT Procedure Editor**), click the **MPT Options Editor** icon. Customize the settings on each tab as desired. (These settings will be used by new tests.)

   Note *When shipped, the default.000 file does not contain process information—that is, the procedure table is blank. If you add processes to the default.000 file, new test procedures will open with those processes preloaded in the procedure window.*

7. From the MPT file icon pull-down menu, click **Save Procedure** to save the customized default template. This new customized template will be used when creating any new procedures.
8. Close the file.
9. From the MPT file icon pull-down menu, click **Open Procedure**.
10. In the **Open Procedure** window, right-click the **default.000** file, and then click **Properties**.
11. In the **Properties** window, select the **Read Only** check box, and then click **OK**. This protects the new custom template from being inadvertently changed.

Using Multiple Default Templates

You may create multiple unique default templates for your testing needs. However, each template must be named default.000, and must be stored in a separate directory—you cannot save more than one default.000 file in a given directory.

How to Create and Use Multiple Default Templates

1. Open your current default template (default.000).
2. Modify the default template as desired.
3. From the MPT file icon pull-down menu, click **Save Procedure As**.
4. Use the **Save Procedure As** window to save the file to a different directory, with the label default.000. (All default templates must be named default.000.)

If you want MPT to use the alternate default template:

1. On the MPT control panel toolbar, click the **MPT Options Editor** icon.
2. Click the **Directories** tab, and then click **Template Files**.
3. Use the directory controls to locate the folder where your new default template is saved, and then click **Set Default** to update the registry with the new default template location.

The alternate default template will be used for the next procedure you create. MPT will always use the default template that has been assigned in the **Directories** tab of the **MPT Options Editor**.

Note *It is good practice to designate your default templates as "read only" so they cannot be inadvertently changed.*

Template Error Messages

MPT displays various error messages if it cannot locate a default template file.

The built-in template

If MPT cannot locate the default template, it will display a default error message, and create a procedure based on its built-in template.

The built-in template uses factory-set preferences that cannot be changed.

If the default template has been misplaced in another directory, you can use the file browser in the **Directories** tab of the **MPT Options Editor** to find the file. Once you locate the default template you can relink it to the MPT application.

If the default template has been deleted, you can create a new procedure with the desired attributes and save it as "Default.000."

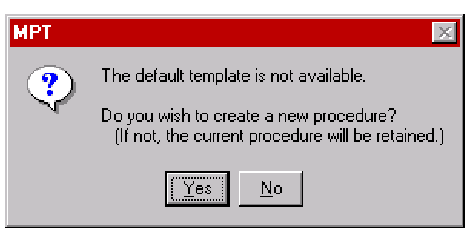
When you start MPT

The following message is displayed when you start the MPT application and MPT cannot find a default.000 file in the directory specified on the **Directories** tab:

When you click **OK**, MPT will use its own *built-in template* to open a test procedure.

When you select New after opening a procedure

The following message is displayed in the event you open a test procedure, select **New** from the **File** menu, and the MPT application cannot find a default.000 file in the path specified for template files:

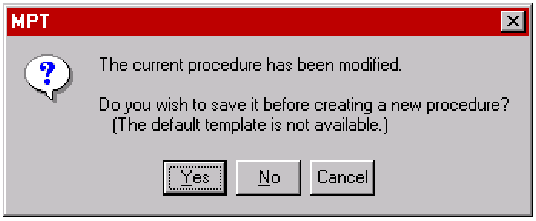


If you click **Yes**, MPT will open a new test procedure using the built-in template.

If you click **No**, MPT will remain in the current procedure.

When you select New while editing a procedure

The following message is displayed in the event that you edit a test procedure without saving, click **New** from the **File** menu, and the MPT application cannot find a default.000 file in the path specified for template files:



If you click **Yes**, MPT will save your changes to the current test procedure, and open a new test procedure using the built-in template.

If you click **No**, MPT will close the current test procedure without saving your changes, and open a new test procedure using the built-in template.

Appendix B
Excel File Converter

The MTS Excel File Converter is a utility that optimizes MPT data files for viewing with Microsoft® Excel.

Note *For more information on MPT data files, see "Specimen.dat" on page 74.*

When this utility is installed and enabled, Microsoft Excel can interpret MultiPurpose TestWare (and Basic TestWare) data files more intelligently than it normally interprets text files.

For example, the MTS Excel File Converter can be used to:

- Permit loading an unlimited number of rows of data—up to the memory capacity of your system. (Microsoft Excel has a limit of 65,535 rows of data on each sheet.) The converter splits the data into multiple sheets.
- Scan the data file for certain labels and use smart processing to determine when data should be written to a new worksheet. (This keeps your data organized more efficiently.)
- Enable **.dat**, **.t31**, and **.f31** files to be browsed and opened directly from the Microsoft Excel **Open** window.

Compatible data file formats

The Excel File Converter works best with tab-delimited data files (Excel format). If you want to use the Excel File Converter, you should make sure your data files are written in the Excel format. For more information on specifying your data file format, see "Formatting the data in the MPT specimen data file" on page 75.

Comma-delimited data files (Lotus format) can be processed by the converter, but the format of the resulting spreadsheet may not be desirable. Space-delimited data files (Plain format) will not be processed by the converter.

Note *Files in "Lotus" format will be read in, but the format of the resulting spreadsheet may not be desirable. If your Windows operating system's regional settings specify that decimal points should be commas (as is typical in parts of Europe) the "Lotus" format will not be readable. Use the "Excel" format whenever possible.*

Excel File Converter

How to Install the Excel File Converter

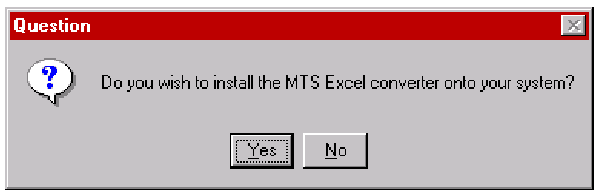
The Excel File Converter is included with the Model 793.00 System Software. During the system software installation, you will asked if you want to install the MTS Excel File Converter.

If you did not install it during the main software installation, insert the Model 793.00 System Software disc into your CD-ROM drive and follow these steps:

1. Follow this path using Windows NT Explorer:

   Your CD-ROM drive > Excel Converter > Setup.exe

   The following window appears:



2. Click **Yes**.

3. In the **Select Directory** window, specify the location for the MTS Excel File Converter utility, and then click **Next**.

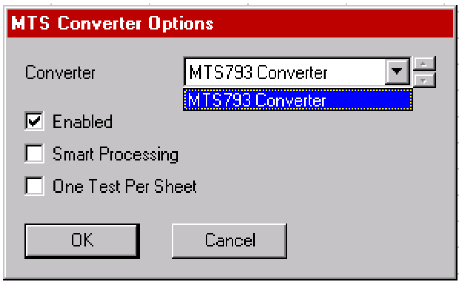
4. Click **OK** to acknowledge a successful installation.

How to Set Excel File Converter Options

Specify your Excel File Converter options to customize your data files in some useful ways.

1. Start the Microsoft Excel application. (A new spreadsheet will appear.)

2. On the Microsoft Excel **Tools** menu, click **MTS Converter Options**.



3. Select options as desired. (See Option descriptions for descriptions of the Excel File Converter options.)

Option descriptions

Enabled—This check box must be checked to enable the File Converter.

Smart Processing—If this box is enabled, the File Converter looks for certain labels. When it finds one of these labels, and there is not the required number of rows left in the current sheet, it will start filling a new worksheet. These are the labels it looks for, along with their respective numbers of rows:

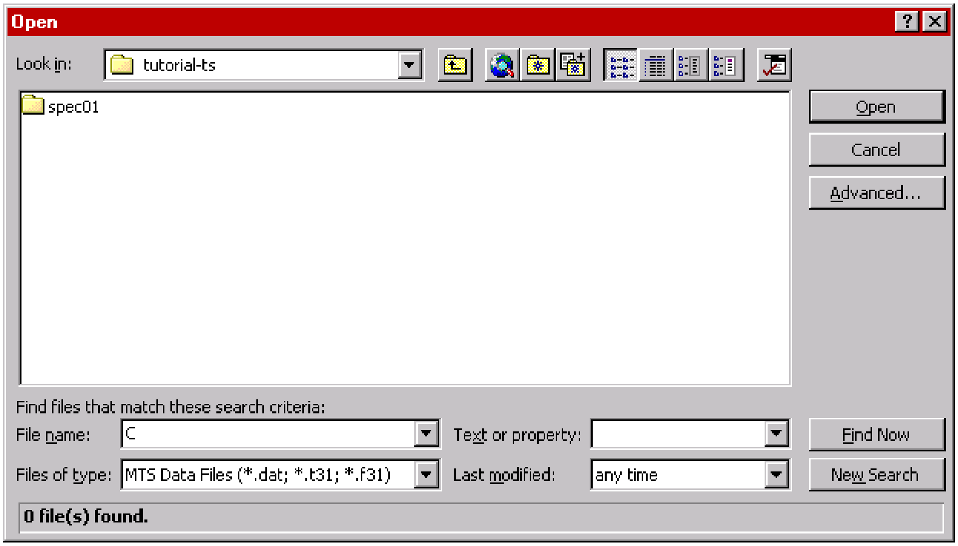
- Data Acquisition—4,000 rows
- Dynamic Characterization—2,000 rows
- Static Deflection—1,200 rows

Note *Smart processing will recognize German or French equivalents, based on the regional settings of your Windows operating system setting.*

One Test Per Sheet—If this box is enabled, each time the File Converter detects a new test within a data file, it will cause that test data to start filling a new Excel sheet.

How to Use the Excel File Converter

1. On the Microsoft Excel **File** menu, click **Open**.



2. In the **Files of type** list, click **MTS Data Files (*.dat; *.t31; *.f31)**.

3. Locate the desired data file, and click **Open**. The data file should appear in a new worksheet. If the Excel **Text Import Wizard** window appears, refer to the troubleshooting section that follows.

Troubleshooting

If the Excel **Text Import Wizard** window appears when you open a .dat file, ensure the following:

- The Excel File Converter is installed properly. If not, reinstall it. (See “How to Install the Excel File Converter” on page 456.)
- The Excel File Converter is enabled. If not, enable it. (See “How to Set Excel File Converter Options” on page 457.)
- Your data file format is tab- or comma-delimited. If not, the file is incompatible with the Excel File Converter. Make sure you generate all new data files in the Excel format. (See “Compatible data file formats” on page 455).

Appendix C
Profiles

Profiles

About Profiles

A profile is an ASCII text file that defines a series of command elements in a spreadsheet-style (or grid-style) format.

A profile grid contains a number of rows, with each row defining a single command element. Command elements may be a single segment or series of segments (cycles). Each segment or cycle is defined by its waveshape, end levels, rate type (time, frequency, or rate), and count.

A single profile can (and often does) contain the command content of an entire procedure.

Channel 1

	Time (Sec)	Count (segments)	Shape	Level1 (mm)	Level2 (mm)
0	3.0000	3.0000	Ramp	-10.0000	7.0000
1	4.0000	3.0000	Ramp	3.0000	6.0000
2	5.0000	3.0000	Ramp	4.0000	-8.0000
3					
4					
5					

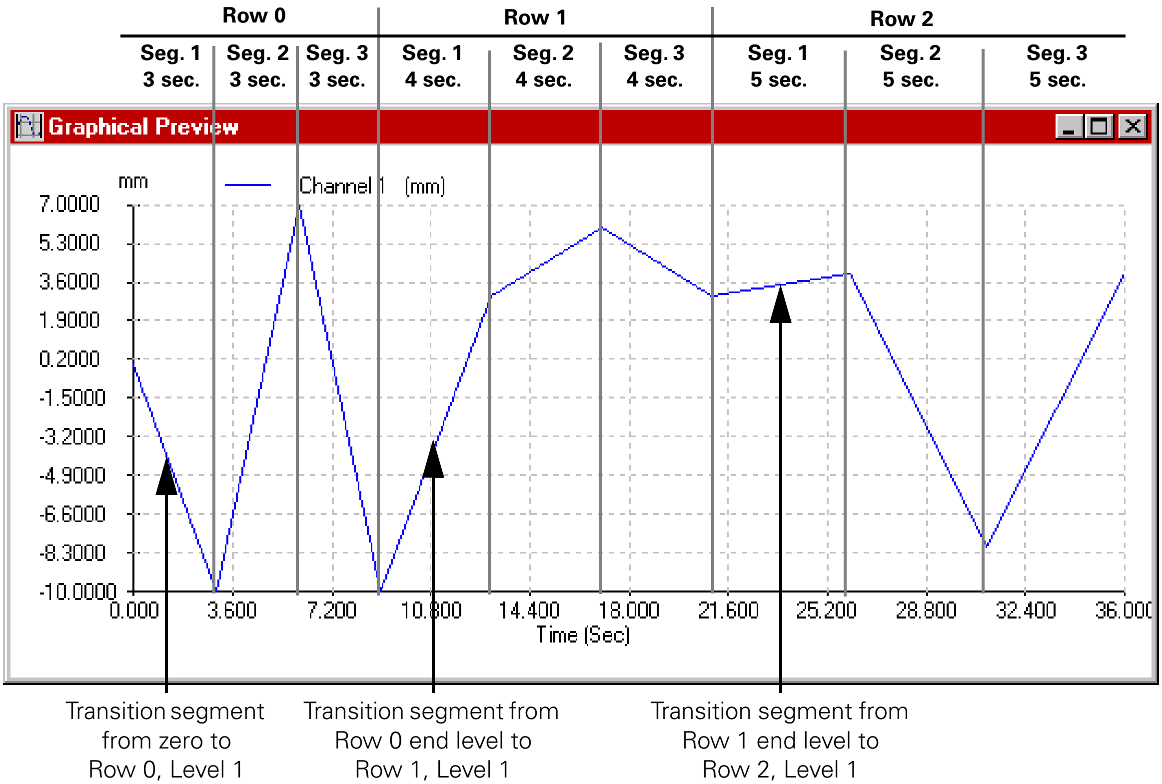
Sample Profile Grid

Profile output example

This example is based on the sample grid shown on the previous page. In this example:

- Row 0 specifies a ramp waveform shape, 3 segments, 3 seconds per segment, Level 1 of -10 mm, and Level 2 of 7 mm.
- Row 1 specifies a ramp waveform shape, 3 segments, 4 seconds per segment, Level 1 of 3 mm, and Level 2 of 6 mm.
- Row 2 specifies a ramp waveform shape, 3 segments, 5 seconds per segment, Level 1 of 4 mm, and Level 2 of -8 mm.

The file is read in sequence (row by row) to produce a waveform:



Note *The first segment of each row is always a "transition" segment from the end level in the previous row to Level 1 in the current row. At the beginning of the procedure, the first segment is a transition from the current level to Level 1 in Row 0. (See "Level data" on page 470.)*

Creating Profiles

The MPT Profile Command and Profile with ALC processes support block-arbitrary profiles and phase profiles.

You can create a profile using one of the following tools:

- A text editor
- A spreadsheet application
- The MTS Model 793.11 Profile Editor application

Block-arbitrary profiles

A block-arbitrary profile defines a particular command segment (or a series of segments) for each channel. Each segment can have a different shape, rate, repeat count, and amplitude.

A block-arbitrary profile can define several channels of data since there is no interaction between the data channels. For more information, see "Block-Arbitrary Profile Example" on page 471.

Phase profiles

A phase profile also defines a particular command segment (or a series of segments) for each channel. However, the segment shape, time base and count must be the same for each channel.

A phase profile can define several channels of data. Each row of segment data can have Phase Lag defined for it. The Phase Lag of each channel is referenced to the lowest Phase Lag defined in a channel. For more information, see "Phase Profile Example" on page 473.

General Profile Syntax Requirements

If you create a profile using the Profile Editor application, the syntax of the profile is managed automatically by the Profile Editor application.

If you are creating a profile with an external spreadsheet or text editor application, you should review the following profile syntax requirements before you start:

- Profiles should be saved with a .blk extension so the default file filter in MPT will recognize them.
- You must separate profile entries with a space or a tab (when using a text editor).
- Blank lines between rows may be used to improve readability.
- The profile must start with the header data definition, then with channel and command data definitions for each succeeding channel defined in the profile. (Header data, Channel 1 data, Channel 1 command data, Channel 2 data, Channel 2 command data, and so forth.)
- Keywords are not case-sensitive.
- Do not leave a space between the keyword and the equals sign (=). However, you can insert a space after the equals sign to improve readability.
- Use header keywords (FileType, Date, and so on) only once.
- Use channel keywords (level, frequency, and so on) only once for each channel definition.

Header Data Syntax

The following is an example of proper header data syntax:

FileType= Block-Arbitrary

Date= Fri Aug 12 07:55:44 2000

Description= this is a sample test profile

ActionList= <DO 1 on>, <DO 1 off>, counter1, “counter 2”

Channels= 1

KEYWORD	SUGGESTIONS	COMMENTS
FileType=	Block-Arbitrary or Phase	This required entry must be first.
Date=	Last modified date and time	This optional entry can be in any form. It can be omitted.
Description=	User-defined description of the file	This optional entry can be in any form (quotation marks are not needed) on one line. It can be omitted.
ActionList=	See “Action and Counter Syntax” on page 465.	This optional entry applies if you have actions or counters associated with your profile. It can be omitted.
Channels=	Number of channels in profile	This required entry must be followed by channel and channel data definitions for each channel in the profile.

Action and Counter Syntax

Observe the following syntax requirements when adding actions or counters to a profile file.

- Actions and counters are defined by the keyword Action. This attribute may be a constant value or a column of values.
- If the Action column is the last column in the file, the action value may be left blank. If the Action column is not the last column, the user must specify the empty string (“ ”) as a placeholder (any other string will be interpreted as a unique counter name).
- If the string contains embedded spaces, then counter names must be enclosed in quotes (“ ”) . If the name is one word, the quotes are not necessary.
- Action names must be delimited by a left angle bracket (<) and a right angle bracket (>). The Profile process will use these delimiters to distinguish between event actions and counters. If the delimiters are missing, these strings will be interpreted as new counters. If the delimiters are used for a counter, the process will interpret the name as an event action. Neither of these syntax errors will generate an error, but they may cause unexpected results.
- The keyword ActionList= must be in the file header if any actions are defined. It must include all event action names defined in the file; counter names are optional. The names must be separated by a comma. This list cannot exceed 256 characters without containing a space. MTS recommends separating items in the list with a comma (required) and a space.

Profiles

Example with counters and actions

FileType= Block-Arbitrary
Date= Wed_Jul 05 15:11:32
ActionList= <DO on>, <DO off>, “Counter 1”, Counter2
Channels= 1

Channel(1)= Channel 1
Frequency= 1 Hz
Count= 1 segments
Shape= Sine
Level2= 0.0 mm

Level1 mm	Action
5.0000	<DO Off>
-5.0000	“Counter 1”
8.0000	
-8.0000	
2.000	Counter2
-2.000	
5.0000	
-5.0000	
1.0000	
-1.0000	<DO On>

Channel Header Syntax

Following the header data definition, you must define the first control channel. After the channel definition, you must define the channel data for that channel (see “Channel Data Syntax” on page 467).

The following is an example of proper channel data syntax:

Channel(n)= channel name

Max= maximum value and units

Min= minimum value and units

KEYWORD	COMMENTS	SUGGESTIONS
Channel(n)=	Names the channel; n is the channel number	This is a required entry and must be first. The channel name is not case-sensitive.
Max=	Placed in the file by the Profile Editor and displayed on the **File Summary** tab; useful for determining an appropriate range with which to control the channel.	This is an optional entry that specifies the maximum output for the channel. It can be omitted.
Min=	See "**Max=**".	This is an optional entry that specifies the minimum output for the channel. It can be omitted.
Dimension=	Placed in the file by the Profile Editor to determine the dimension of a given unit.	This is optional but recommended entry. It can be omitted.

Profiles

Channel Data Syntax

Following the channel definition, you must define the command data for that channel.

Channel data attributes can be entered two ways—as constants or as individual values. Constants are declared before any individual parameters are listed, and they require the equal sign (=) to be appended to the end of its keyword. Any keyword can be defined as a constant. The following is one example of the command data syntax where one keyword is defined as a constant. Each row of data defines a command element.

The following is an example of proper command data syntax:

Shape= Sine

Frequency	**Count**	**Level1**	**Level2**
Hz	**Cycles**	**mm**	**mm**
10	**100**	**5**	**-5**
5	**150**	**10**	**0**
1	**0.5**	**0**	**0**

KEYWORD	RANGE	COMMENTS
Frequency, Time, or Rate[1]	Undetermined	This entry is required. Only one of these three can be specified. Rate must reflect the rate of change of the Level dimension. See "Rate type" on page 469.
Count	Greater than 0	This optional entry specifies the number of repeats. The units can be segments or cycles, and the value must be greater than 0. If this option is omitted, the command will execute once. See "Count" on page 469.
Shape	Ramp, Sine, Square, TrueSine, SquareTapered, RampTapered, SineTapered, or TrueSineTapered	This entry is optional. When not assigned as a constant, each segment can have a different shape. The default is Sine. See "Shape" on page 470.
Level1	Undetermined	This entry is required. See "Level data" on page 470.
Level2	Undetermined	This entry is required for more than one segment. See "Level data" on page 470.
Phase Lag[2]	0–360°	This optional entry specifies the phase lag for phase profiles. The default is 0. See "Phase lag" on page 470.

1. Rate is not supported by phase profiles.
2. Phase lag is not supported by block-arbitrary profiles.

Profiles

Rate type

A rate type determines the time base of the waveform. Three types of rate expressions (keywords) are supported.

- **FREQUENCY** (Hz, cps)
- **TIME** (msec, sec, minutes, and so on)
- **RATE** (units per time; for example, kips/sec)

A constant rate type of TIME= 2.5 sec assigns 2.5 seconds to each row of segment data. When a constant is defined, the related column (Frequency/Rate/Time) is not allowed. A variable rate type identifies a column of time values, where each row can have a different time base.

Count

The count lets you repeat a cyclic waveform a specific number of times. The **COUNT** keyword can repeat segments or cycles.

- 1 cycle = 2 segments
- A cycle count of 0.5 is the same as a single segment.
- Repeating a segment an even number of times ends at Level 2.
- Repeating a segment an odd number of times ends at Level 1.

The count can be set to a constant such as COUNT= 1 SEGMENTS. This executes each Level 1 parameter once. It is useful for profiles that issue a series of arbitrary end levels. In this case the values in Level 2 are not used.

Note *The True Sine segment shape always starts and stops at its mean level, and will cause a step if the previous row did not end at its mean level.*

Profiles

Shape

A waveshape defines how to go from the current end level to the next end level. The **SHAPE** keyword can define a waveshape as a constant at the beginning of the channel (SHAPE= RAMP), or it can define a column of data to assign a shape to each row.

Level data

The level data defines the end level of the rows. The file requires a Level 1 value for each row. If a Level 2 value is included, multiple segments are being specified (cycles). Cycles execute from the current level to Level 1, then to Level 2.

Note *The first segment of each row is always a "transition" segment from the end level in the previous row to the Level 1 value in the current row. At the beginning of the procedure, the first segment is a transition from the current level to Level 1 in Row 0.*

Phase lag

The Phase file type is the same as the Block-Arbitrary file type, except that phase relationships between channels can be specified. When Phase is used, the following requirements must be observed:

- The waveform rate, shape, and count must be the same for all phased channels. They must be defined with the first channel.
- The phase of one channel is compared to another; the higher value lags the lower value.
- Each row of data defines a waveform. The first segment of each of the waveforms is stretched to achieve the phase lag.

Block-Arbitrary Profile Example

Header data

Filetype= Block-Arbitrary

Date=10-Dec-00

Description= Two Channel Test

Channels= 2

Channel 1 header

Channel(1)= Right Front

Frequency= 1 Hz

Shape= TrueSine

Channel 1 data

Level1	Level2	Count
in	in	segments
5	-5	10
10	-10	20

Channel 2 header

Channel(2)= Left Front

Frequency= 2 Hz

Shape= Ramp

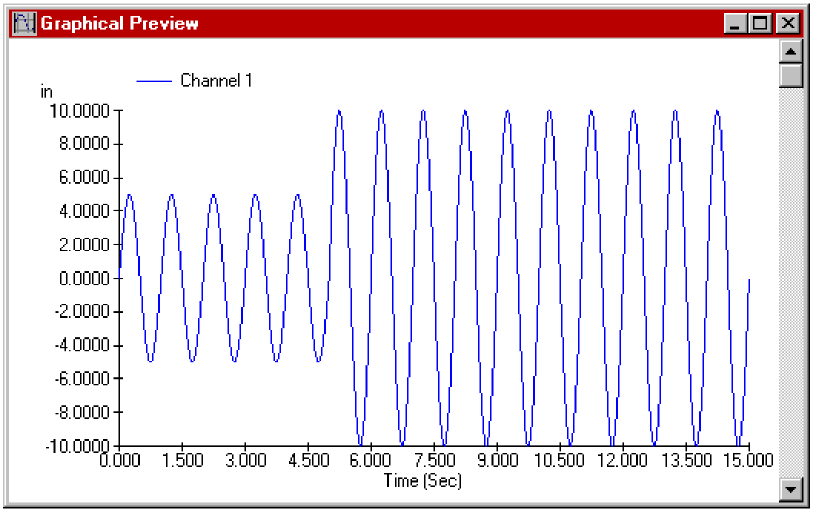
Channel 2 data

Level1	Level2	Count
lbf	lbf	segments
5000	-5000	15
10000	-10000	20
2000	-200	20

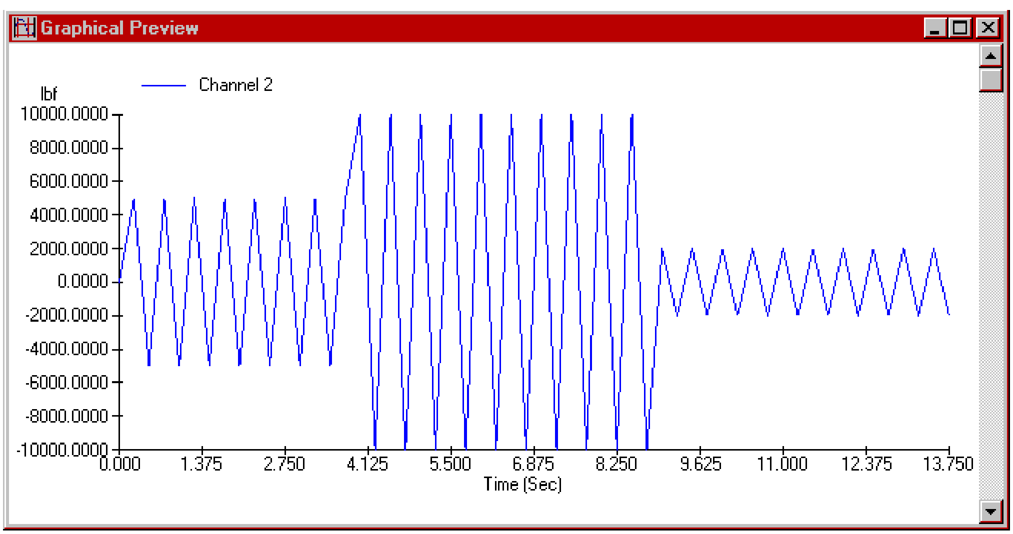
Profiles

Profiles

Graphical Preview Channel 1



Graphical Preview Channel 2



Phase Profile Example

Header data

FileType= Phase

Date= 10-Dec-00

Channels= 2

Channel 1 header

Channel(1)= Left Front

Frequency= 1 Hz

Shape= Sine

PhaseLag= 0 deg

Channel 1 data

Count	Level1	Level2
segments	mm	mm
4	5	-5
4	8	-8
4	5	-5
4	2	-2

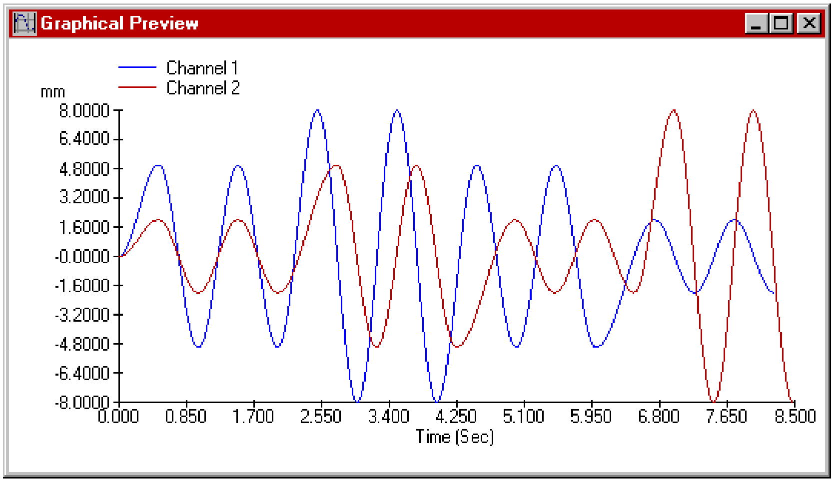
Channel 2 header

Channel(2)= Right Front

Channel 2 data

Level1	Level2	PhaseLag
mm	mm	deg
2	-2	0
5	-5	90
2	-2	180
8	-8	90

Graphical Preview Channels 1 and 2

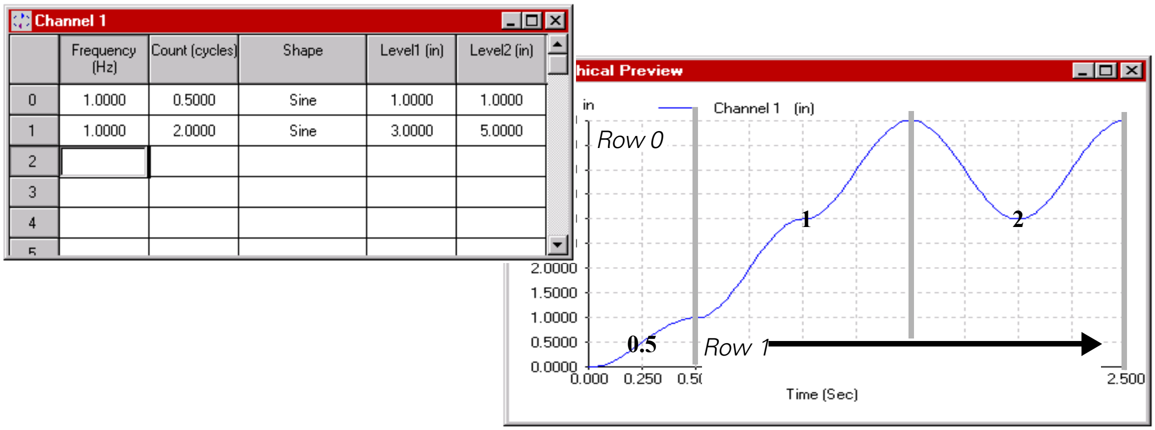


Profiles

About Cycle and Segment Behavior

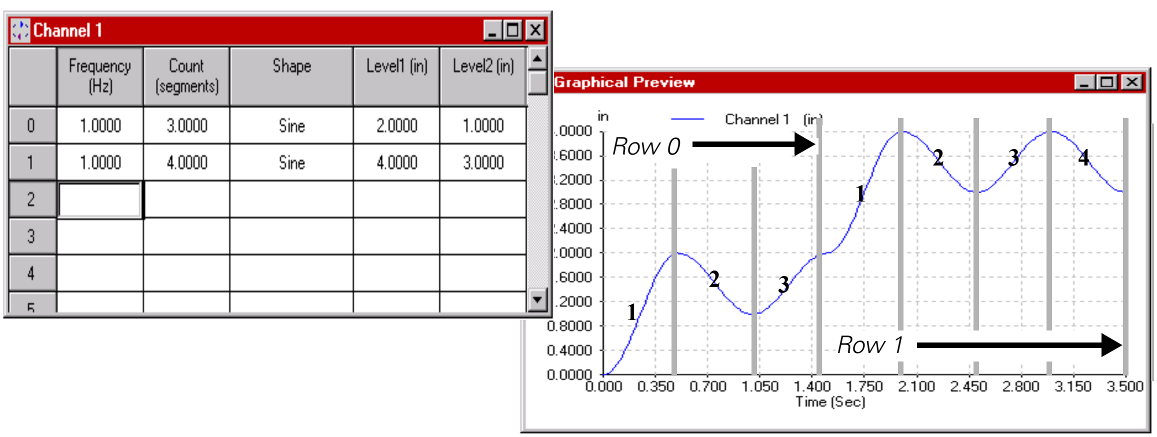
Cycle behavior

The first cycle in a row begins at the current level, travels to level 1, then travels to level 2. Subsequent cycles in the row travel from level 2 to level 1 and back to level 2. If a row has a cycle count of 0.5, the cycle begins at the current level and travels just to level 1.



Segment behavior

The first segment in a row begins at the current level and travels to Level 1. Subsequent segments in the row travel from one level to the next.

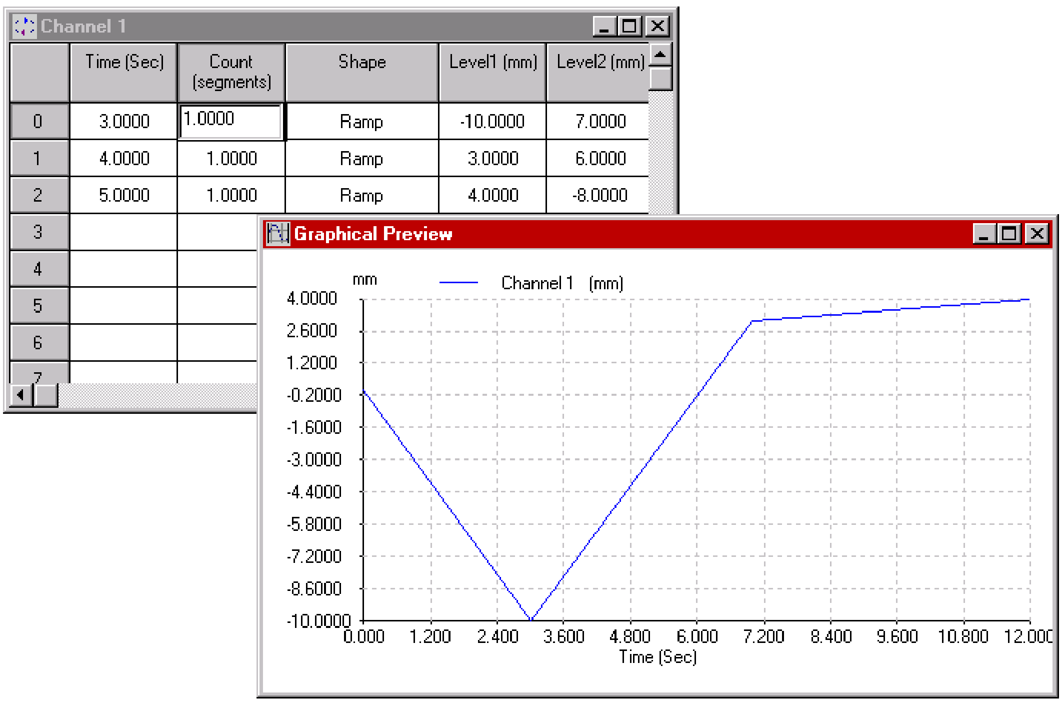


Segment behavior with Segment Count of 1

The first segment assigned to each row is always a “transition” segment from the end level in the previous row to the **Level 1** value in the current row. Thus, if you specify a **Count** of 1 segment, the **Level 2** values will be ignored. See “Level data” on page 470.

The illustration shows that since the first segment in each row is a transition from one row to the next, Level 2 values are ignored. Because only one segment has been assigned to each row, that segment is used to transition between rows.

For a more detailed discussion of segment behavior, see page “Profile output example” on page 461.



About Transitions

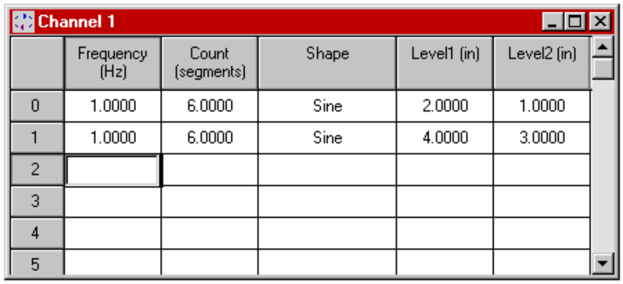
Note *The following information applies only to Sine, Ramp, and Square waveshapes. True Sine and tapered waveshapes always start and stop from their mean level.*

It is useful to understand how MPT generates transitions between rows, and how to add segments to modify the transitions.

The first segment (or the first half of the first cycle) in a row travels from the current level to the Level 1 value. It has special significance because it is the portion of the waveform that makes the transition between the rows of the profile.

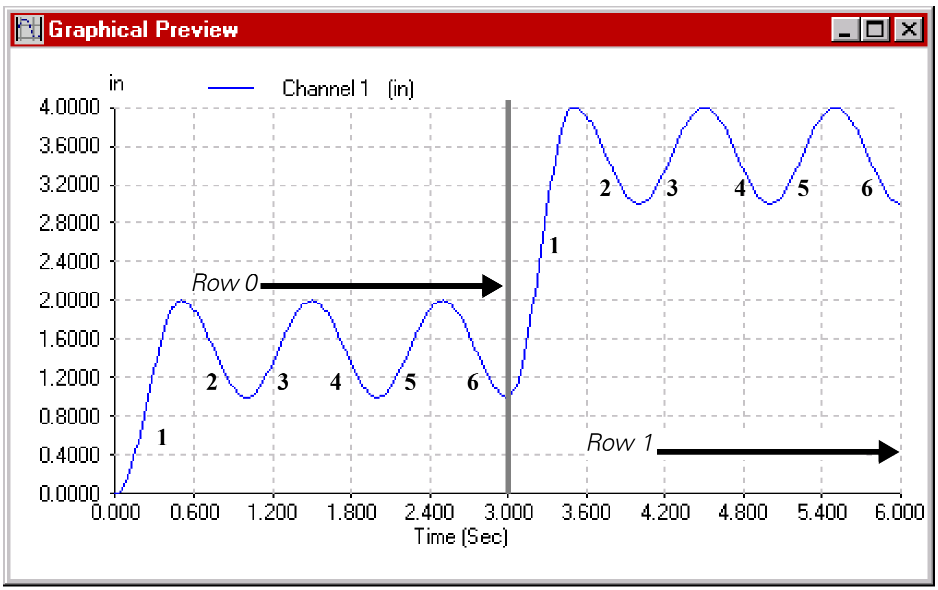
More About Automatic Transitions

This MTS Profile Editor example shows how MPT automatically generates the transition segments between the rows of the sample profile (Row 0, Segment 1 and Row 1, Segment 1).



Channel 1

	Frequency (Hz)	Count (segments)	Shape	Level1 (in)	Level2 (in)
0	1.0000	6.0000	Sine	2.0000	1.0000
1	1.0000	6.0000	Sine	4.0000	3.0000
2					
3					
4					
5					



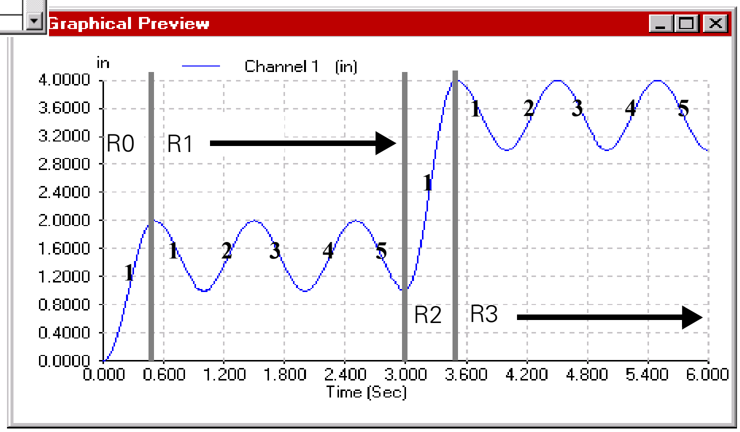
More About Manual Transitions

Sometimes it is desirable to change the characteristics of row transitions without affecting the remainder of the profile. To do this, you must manually add transition segments to replace automatically generated transition segments, as shown in the MTS Profile Editor examples below.

Channel 1

	Frequency (Hz)	Count (segments)	Shape	Level1 (in)	Level2 (in)
0	1.0000	1.0000	Sine	2.0000	
1	1.0000	5.0000	Sine	1.0000	2.0000
2	1.0000	1.0000	Sine	4.0000	
3	1.0000	5.0000	Sine	3.0000	4.0000
4					
5					

This profile is similar to the one on the previous page, but two rows have been added to create two "editable" transition segments. To keep the number of segments the same, the rows that previously specified six segments now specify five segments, and their level values are reversed.

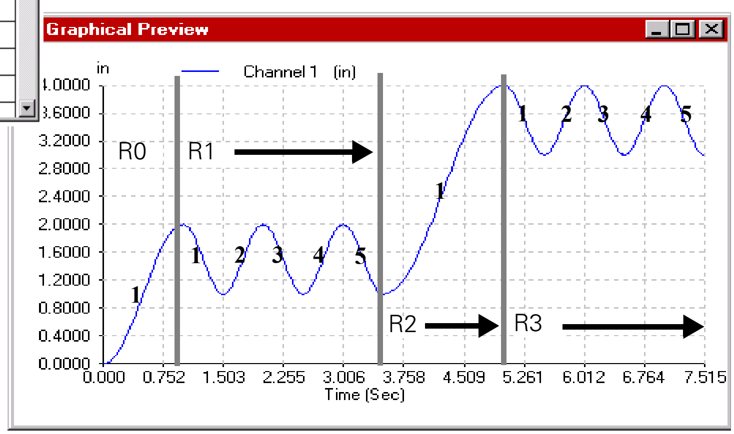


The plot appears identical to the one shown on the previous page, but now the transition segments can be edited.

Channel 1

	Frequency (Hz)	Count (segments)	Shape	Level1 (in)	Level2 (in)
0	0.5000	1.0000	Sine	2.0000	
1	1.0000	5.0000	Sine	1.0000	2.0000
2	0.3300	1.0000	Sine	4.0000	
3	1.0000	5.0000	Sine	3.0000	4.0000
4					
5					

This profile is similar to the one above, but the frequency values of the added transition segments have been changed.



The plot shows the new frequencies of the added transition segments.

Working with Dimensions

Using standard dimensions

You create a profile by entering the end level values you desire the actuator to achieve during the course of the profile. For end level values to be meaningful, they must be associated with a dimension and unit type. With the Profile Editor application, you specify the dimension and unit type separately (for instance, length and cm). When you create profiles with other applications (such as text editors), you select only the unit type, which has an inherent dimension (for instance, cm, which is associated with length).

Note *MTS recommends that you include the keyword "Dimension=" in your profiles to clarify unit type, because some unit types occur in more than one dimension.*

When you start MPT against a station configuration, MPT inherits resources—including the control mode dimensions—defined in the station configuration file.

When you select the control mode for a profile channel with the Profile process, the dimension of the profile end levels must match one of the control mode dimensions available to the current station configuration.

Example

Suppose you create a station configuration that uses the force dimension for a force control mode, and the length dimension for a displacement control mode. In addition, you create a profile in which you select a unit type associated with the length dimension to define end levels.

Next, you start MPT against the station configuration, and with the Profile process, on the **Command** tab, you select the profile you have created. On the **Mapping** tab, you select a **Profile Channel**. In this scenario, only displacement will be available in the **Control Mode** list.

Also, the **Level Multiplier** box will display 100% by default, which means MPT will play out the end level values in the profile without scaling them. In this case, if the first end level values in the profile are 2 cm and 9 cm, they will be played out as 2 cm and 9 cm. If you changed the **Level Multiplier** value to 200%, MPT would play out the these values as 4 cm and 18 cm.

Using normalized dimensions

If you create a profile using a *normalized* dimension, such as ratio, percentage, unitless, or volts, you can select any of the control modes available to the station configuration when selecting the control mode for the profile channel.

When you do this, you define the full-scale value of the normalized dimension by selecting one of the control modes and its associated engineering dimension available in the **Control Mode** list on the **Mapping** tab, along with a unit type and a **Level Multiplier** value. MPT converts all end level values in the profile to command levels accordingly.

Example

Suppose you create a station configuration that uses the force dimension for a force control mode, and the length dimension for a displacement control mode. In addition, suppose you create a profile in which you select the volts dimension to define end levels.

Next, you start MPT against the station configuration, and with the Profile process, on the **Command** tab, you select the profile you have created. On the **Mapping** tab, you select a **Profile Channel**. In this scenario, both displacement and force will be available in the **Control Mode** list.

Suppose you select displacement, and enter a value of 5 cm in the **Level Multiplier** box. This will cause all of the end level values in the profile to be converted to displacement values. For volts, the full-scale value equals 10 volts, which MPT will play out as 5 cm. So in this case, if the first end level values in the profile are 2 volts and 9 volts, they will be played out as 1 cm and 4.5 cm.

Using the same profile, suppose you change the **Control Mode** selection to force, and enter a **Level Multiplier** value of 100 N. This will cause all of the end levels in the profile to be converted to force values. In this case, MPT will play out 10 volts as 100 N. So if the first end level values in the profile are 2 volts and 9 volts, they will be played out as 20 N and 90 N.

Profiles

Full-scale values of normalized dimensions

Dimension	Full-Scale Value
Ratio	1:1
Percentage	100%
Unitless	1.0
Volts	10V

Using normalized dimensions to define control modes

You can use any of these normalized dimensions to define a control mode in a station configuration. If you do this, and then attempt to play out a profile that also uses a normalized dimension, MPT will treat it as a standard dimension when you define a Profile process.

Example

Suppose you create a station configuration that uses volts as a dimension for a volt control mode, which is associated with a conditioner that is cabled to a displacement sensor with a full-scale displacement of ±200 cm.

Next, you create a profile in which you select the volts dimension to define end levels.

Then you start MPT against the station configuration, and with the Profile process, on the **Command** tab, you select the profile you have created.

On the **Mapping** tab, you select a **Profile Channel**, and note that volts is available in the **Control Mode** list, and that the **Level Multiplier** box displays 100%.

In this scenario, if the first end level values in the profile are 1 V and 9 V, they will be played out as 20 cm and 180 cm. If you change the **Level Multiplier** value to 50%, MPT would play out these values as 10 cm and 90 cm.

Appendix D
MPT Shortcuts

You can use an MPT shortcut to perform a number of tasks automatically when you start the MPT application.

For instance, you can use MPT shortcuts to:

- Automatically load a specific procedure
- Automatically load a specific specimen
- Launch MPT as an editor (edit only mode)
- Automatically display the **MPT Procedure Editor** when MPT is launched
- Connect to a specific station
- Connect to a specific test system

To specify the MPT shortcut parameters, you must create an MPT shortcut, and then edit the *shortcut command line*.

About the Shortcut Command Line

Note *The syntax of the shortcut command lines are critical. If you do not use a supported shortcut command line, the shortcut will not work.*

Parameter and prefix descriptions are listed as follows:

PARAMETER	PREFIX	EXAMPLE
Load a Procedure file	/Procedure[1], or /P	/P default.000
Load a Specimen	/Specimen[1], or /Sp	/Sp spec03
Launch MPT as an Editor (no execution)	/EditOnly	
Launch the Procedure Editor with MPT	/Editor [x:y:w:h:style]	mpt /editor 400:400:300:300:StyleMinimized
Connect to a Station	/Station[1], or /St	/St station1.cfg
Connect to a System	/System[1], or /Sy	/Sy 148.150.203.191
Controller Name	/Controller name	/Controller 001
Display the MPT command-line options	/Help, or /H	

1. If the parameter name contains spaces (for example, *test 257.000*), the name must be enclosed in quotation marks ("test 257.000").

MPT Shortcuts

Command line practices

When you are editing the command line, keep in mind:

- The order of the parameters is not important.
- Prefixes can be spelled out or abbreviated in the command line.
- The prefix and the parameter must be separated by a space.
- If the parameter name contains a space (for example, **my first test.000**), it must be enclosed in quotation marks.
- The option names can be abbreviated to anything which is unque.

About the /EditOnly command

The /EditOnly option will launch MPT as an editor only. When you use this option, the MPT Specimen Log and Edit/Execute toolbar buttons will be disabled. MultiPurpose TestWare is not embedded in Station Manager in this mode. Since it is not embedded, it can be run on a different PC or as a second copy of MPT, etc.

About the /Editor [...] command

The /Editor [...] option will launch MPT in the embedded mode with the **Procedure Editor** already displayed. The syntax of the optional arguments (x:y:w:h:style) that specify the initial size and placement of the **Procedure Editor** window are as follows:

- "x" specifies the x-coordinate of the upper left-hand corner of the window
- "y" specifies the y-coordinate of the upper left-hand corner of the window
- "w" specifies the width of the window in pixels
- "h" specifies the height of the window in pixels
- "style" specifies whether the window is maximized or minimized

An example of the full specification is:

"mpt /editor 400:400:300:300:StyleMinimized"

Omitting arguments

Any of these arguments may be omitted, but the ":" must remain as a placeholder if an argument is omitted in the middle of the string. Arguments at the end can be omitted.

If an argument is omitted, the application will use a default value. For example, suppose you are not sure what size you want the **Procedure Editor** window to be, but you are sure you want it to appear in the upper left corner. In this case, you would enter:

"mpt /editor 10:10"

Likewise, suppose you want the **Procedure Editor** window to appear in the default location, but you want its size to be smaller than its default size. It this case, you would enter:

"mpt /editor ::200:200"

Entering invalid values for your screen

Also, if you enter x or y coordinate values that are not valid for your screen (values that would place part or all of the **Procedure Editor** window off the screen), MPT will override your values so that the window appears fully on your screen.

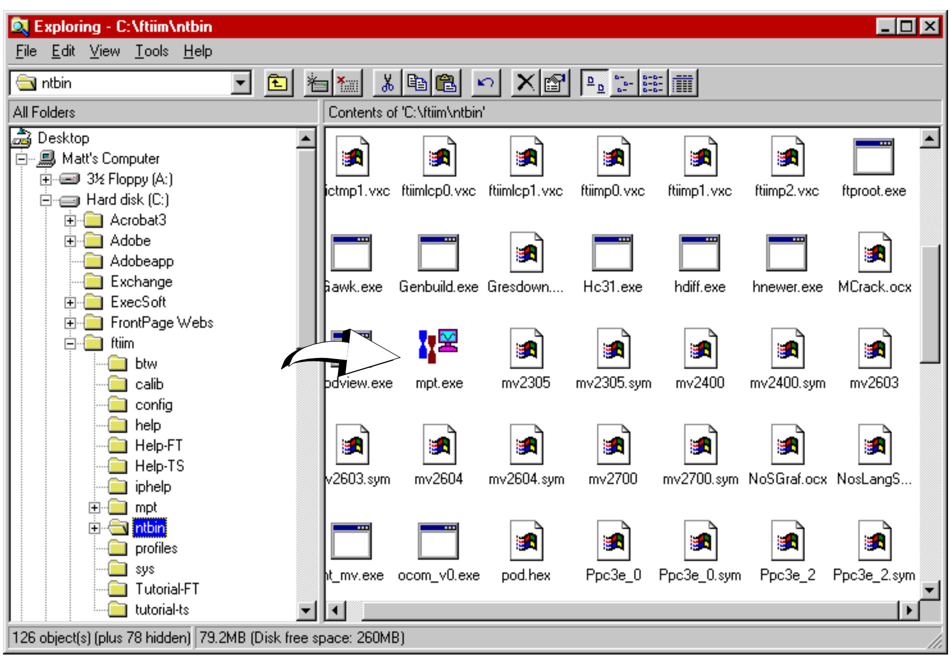
Likewise, if you inadvertently enter negative values for width or height, MPT will use the default (positive) values for width and height instead of the negative values.

Creating an MPT Shortcut

Before you can edit the shortcut command line, you need to create a MPT procedure shortcut.

How to Create an MPT Shortcut

1. Start **Windows® Explorer**.
2. Open the folder where your system software is installed.
3. Open the **ntbin** folder.
4. Right-click the **mpt.exe** icon, and then click **Create Shortcut**. This creates a shortcut file in the **ntbin** folder.
5. Drag the shortcut icon to your desktop.
6. Optional—Right-click the shortcut icon, and then click **Rename**. Rename the shortcut, and then press **Return**.



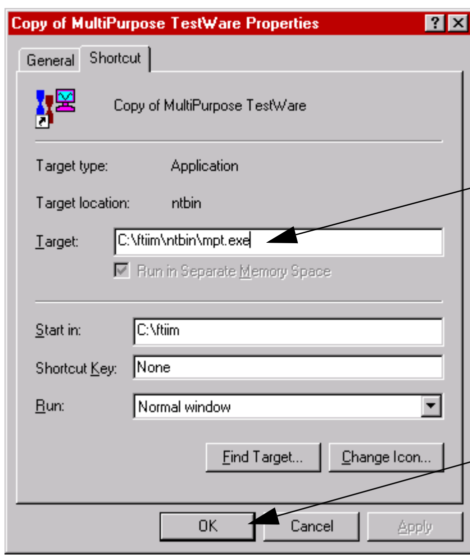
Editing the Shortcut Command Line

Edit the shortcut command line to define your desired procedure parameters.

How to Edit the Shortcut Command Line

1. Right-click the shortcut icon you created on your desktop, and then click **Properties**.
2. In the **Properties** window, click the **Shortcut** tab.
3. In the **Target** box, type your command line.

   Note *There must be a space between **mpt.exe** and your command line addition.*
4. Click **OK**.



Type your command line here. Be sure to put a space between **mpt.exe** and the start of your command line.

Click **OK.**

Test your shortcut

To test your shortcut command line, make sure the Station Manager application is started and the proper configuration file is open. Then double-click your shortcut icon.

Index

Symbols

A

B

C

D

E

F

G

H

I

L

M

N

O

P

R

S

T

U

V

W

Y