KineticSystems Company, LLC

FATIGUE MASTER 7000

February 17, 2003

KINETICSYSTEMS CO. LLC

FATIGUE MASTER 7000

DIGITAL LOAD CONTROL SYSTEM

Operator's Manual

TABLE OF CONTENTS

Page No.

CHAPTER 6

CHAPTER 7

CHAPTER 8

CHAPTER 9

Page No.

CHAPTER 10

CHAPTER 11

CHAPTER 12

APPENDICES

APPENDIX A

CHAPTER 1

INTRODUCTION

Cyber Systems' Fatigue Master 7000 Digital Load Control System provides the latest in structural test equipment available today. State-of-the-art hardware and sophisticated software techniques are combined with three decades of Cyber's experience in the field of structural test systems. The result of this combination is a structural test system capable of meeting the unique requirements of the aerospace and aircraft testing industry.

The Fatigue Master 7000 is completely modular in design, allowing it to be easily configured to accommodate a wide range of requirements, including hundreds of control channels, independent and simultaneous control of multiple tests, and/or acquisition of up to 2000 or more auxiliary data transducers.

All setup, control, and test monitoring is performed from a centralized area using a combination of one or more video display terminals and/or X-Windows terminals. Contemporary menu-driven software together with the UNIX System V Operating System readily handle the multi-tasking, multi-user requirements of a structural test environment.

Overall monitoring and control of test parameters is enhanced by front panel controls & displays available with each servo controller. Graphic software and the high-performance 32-bit X-Windows terminal(s) provide the added capability to monitor on-line test data in several graphic formats, including strip charts, X-Y plots, and bar graphs.

The use of specialized operational modes within the application software allow both static and fatigue type testing applications to be performed with the same degree of ease.

Microprocessors, together with advanced software methods, allow the Fatigue Master 7000 to offer many features which were unavailable until now. Some of these features include automatic bridge balancing, automatic feedback scaling, automatic valve balancing, and automatic loop parameter tuning. These features eliminate the tedious task of manually adjusting control knobs, ultimately reducing the number of man hours and cost associated with setup.

This chapter describes each of the components of the Fatigue Master 7000 Digital Load Control System, and identifies the key features of the test definition software used to set up, monitor, and review tests.

As **Figure 1-1** illustrates, the basic architecture of Cyber's Fatigue Master 7000 includes six major building blocks:

- a Motorola UNIX-based host computer & VDT(s)
- a real-time controller referred to as the TMTM
- one or more Test Control Panels
- one or more digital servo controllers
- one or more "X-Windows" terminals (a.k.a. X-terminals)
- an optional Cyber Auxiliary Data Acquisition System (ADAS)

Though other components such as computer peripherals and discrete I/O hardware are also employed in the system, focusing on the major elements individually described below will simplify the understanding of the overall system operation.

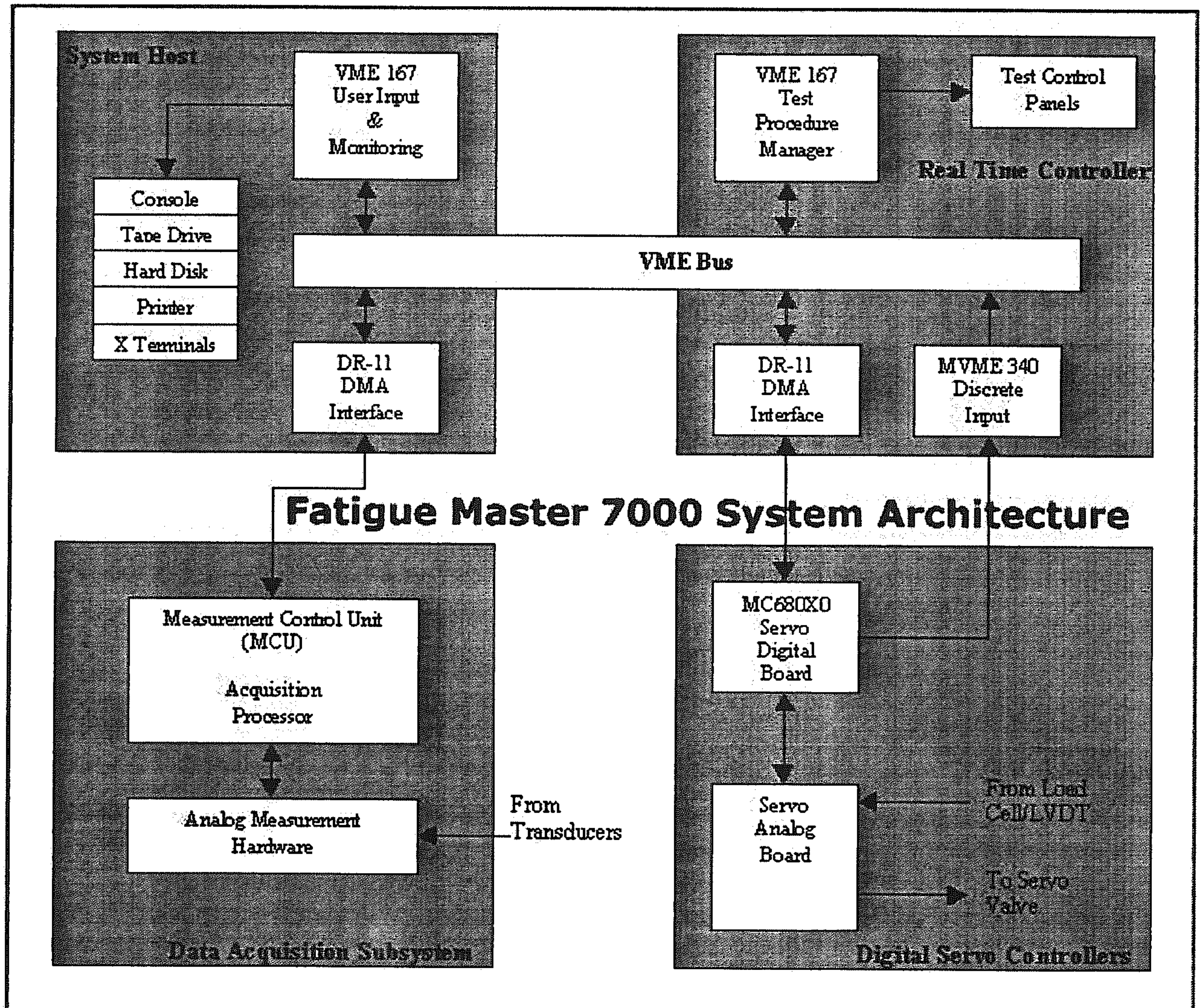


Figure 1-1 FM7000 Architecture

The Host Computer

The **host** is a Motorola Delta Series computer employing a 68040 or 68030 processor and the VME bus architecture. The **host**, in conjunction with one or more standard video display terminals (VDTs) or X-terminals, is primarily used for all non-time-critical functions, including test setup, tabular & graphic data display, data recording, hardcopy output, etc.. All system/test control is accomplished through simple "user-friendly" interface methods such as pull-down menus, spreadsheets, dialogue boxes, etc..

The Motorola **host** computer is configured as a rack-mountable chassis, equipped with the following items:

- 520 MB hard disk unit (upgradeable to larger sizes, optionally)
- 150 MB streaming cartridge tape drive unit
- Optional network communication provisions (NSE which includes Ethernet-TCP/IP, NFS, RFS, etc.) to X-Windows terminals and other compatibly equipped computers. This capability is normally provided only when Cyber's real-time graphics package is configured with the system or when an interface to external computers is required.

The **host** computer contains the application software used to define channel parameters and set up test procedures. The host application software, as well as all firmware used throughout the system, is written in the high-level "C" language.

The computer can store both operator-defined tests and large spectrum load sequences created on a different machine and downloaded via a data link (ethernet or RS-232) or magnetic tape. When you run a test, the host computer transfers this information to the **TMTM**, which actually executes each test procedure step. On-line data regarding the test may be continuously supplied to the host computer for monitoring purposes.

The Test Monitor & Transition Module (TMTM)

The Test Monitor & Transition Module (**TMTM**) is the heart of the FM7000. The TMTM is a real-time controller comprised of three VME processors (68020s) dedicated to performing the real-time execution and synchronization of the test profile, as well as preprogrammed conditional actions in response to "out-of-limit" servo conditions or digital inputs. It continually monitors the servo controllers and overall test operation, while simultaneously reporting system conditions, malfunctions, and faults as they occur.

The **TMTM** also is the communication channel between the servo controllers and the host computer. It uses a digital serial interface which allows remote placement of the servo controllers up to 300 feet from the TMTM and host computer. The communication link between the servo controllers and the TMTM is redundant, with two distinct and dedicated channels. Upon detection of a failure in one channel, the **TMTM** automatically switches to the alternate channel and issues a shutdown to the servo controllers.

The **TMTM** also monitors 16 discrete inputs and controls 16 discrete outputs per test. Though several are reserved for specific functions, most are available to be used with external control and monitoring devices in the test control environment. Typical uses include:

- Hydraulic interlocks which allow the Fatigue Master 7000 and the hydraulic subsystem to communicate. For example, in an emergency the Fatigue Master 7000 may send a dump command to the hydraulic system via a discrete output. Or it may receive a "hydraulics on" discrete input that inhibits load cell calibration.

- General purpose fault indicators that use discrete input signals from auxiliary monitoring devices, such as break wires or limit switches, to trigger immediate corrective action.

- Test article discrete controls that may activate or sense actuators or pumps integral to the test article. If the test article, for example, is required to undergo a pressurization cycle during the test procedure, a discrete output can be used to activate a compressor at the correct time. The load sequence could be scheduled to hold until a discrete input connected to a pressure sensing switch is set, and then resume automatically.

Test Control Panel

For purposes of test article protection, certain test control processes are left under manual control and executed only at the Test Control Panel. The complete test environment may contain multiple Test Control Panels, each controlling only the channels dedicated to a specific test. Each test can also employ redundant control through the use of two Test Control Panels. Typically, one is placed in a control room, while the other is out on the lab floor near the test article.

Once you have defined a test and calibrated the system, you may activate the hydraulic system from the Test Control Panel. The hydraulic system is interlocked; you must first turn the system on, then select low pressure, and then select high pressure. You must use the Test Control Panel when you want to start the test or ramp to pre-defined load conditions prior to starting. During test execution, you may use the Test Control Panel to temporarily hold the test procedure at a specific point or to issue an emergency stop command to the hydraulic actuators.

The Test Control Panel also allows you to scale the test parameters during test execution. You may execute the test one step at a time, simultaneously scale all channels between 1% and 100% of their respective programmed load, or execute the test at between 1% and 200% of the programmed rate. The panel also includes four user-definable endpoint counters that display the current test step. Figure 1-3 shows the front panel of the Test Control Panel.

The Model 7030 Digital Servo Controller

Each dual-channel servo controller in the test environment interprets load instructions from the TMTM and drives up to two load actuators. The servo controller contains built-in circuitry that enables it to execute complex load processes independently of the TMTM. Feedback information from the servo loop is modulated by both digital and analog processes to provide accurate, high-speed, and adaptive loop control.

The servo controller continually monitors loop error and feedback, and responds automatically according to programmed instructions when the feedback and/or error signals exceed specified levels.

The servo loop includes a software summing junction in addition to the hardware error junction, thus allowing the use of programmed algorithms to enhance performance. The Model 7030 Digital Servo Controller includes the following features, all of which are operator-controlled via the Fatigue Master 7000 software:

- Automatic bridge balancing
- Automatic feedback scaling
- Automatic valve balancing
- Oscillation suppression & auto-tuning
- Dynamic null pacing (inner-error pacing)
- Endpoint tolerance checking
- Built-in hydraulic simulation

X-Windows Terminals (Optional)

When the FM7000 is configured with Cyber's Real-Time Graphics Package, the system includes one or more high-performance Network Computing Devices (NCD) X-Terminals. These units operate in conjunction with the host computer via a thin-line or thick-line Ethernet link. They provide enhanced control & display through the use of X-Windows software. Display options include real-time graphical formats which consist of:

- scrolling strip chart display of servo loop parameters (real-time & replay)
- bar graph display of the load feedback signal in engineering units or %
- *X-Y plot of up to 4 ADAS channels vs. one load channel on the same plot

When any of the above graphics formats is requested from the setup software (which is running as an independent "window" on the display), a new, additional window appears which displays data in the requested format. Each graphic window may then be manipulated in standard ways as allowed by X-Windows & Motif (minimize, maximize, move, resize, etc.) and as desired by the operator.

*********************************** NOTE ***********************************

X-Y plots are available only when Cyber's Auxiliary Data Acquisition Subsystem is included in the overall system configuration. In addition, x-y plots are restricted to graphs of stress vs. strain (i.e. load channel vs. auxiliary data acquisition channel).

****

System Peripherals

As standard, the FM7000 is delivered with a dot matrix printer for hardcopy output of tabular reports. If the system is configured with Cyber's optional real-time graphics package, a high resolution laser printer is substituted in place of the dot matrix unit. The laser printer provides hardcopy output of tabular & graphic data.

Transferring/Monitoring Data via Communication Networks

The Motorola Delta Series host computer may be optionally configured to have complete networking capabilities through the use of Network Services Extension (NSE) software. This package provides the following, in part:

- Ethernet - TCP/IP interface capabilities
- Network File System (NFS) software for direct file access between X-Window terminals (clients) and the host (server)
- Remote File Sharing (RFS) software to allow compatible computers to use each others' file systems "transparently".

With this configuration, the Fatigue Master 7000 can be an integral part of a Local Area Network (LAN) for such purposes as file transfer and remote monitoring. If this is the case, a separate Ethernet controller module is normally provided with the host computer.
The system can also facilitate post-test file transfer via supplied RS-232 ports. Using standard file-transfer software (e.g. KERMET) resident in the host computer, data files may be transferred to and from external computers.

The Fatigue Master 7000 Application Software

Much of the operation of the test control system is controlled by instructions that you define and enter into the computer using the Fatigue Master 7000 software. The application software resides on the host computer hard disk as delivered.

This manual provides you with all you need to know to use the software to set up, calibrate, execute, and monitor a test. Some of the key operator features of the Fatigue Master 7000 software include:

- Channel definition
- Spectrum definition
- Programmed conditional actions
- Channel calibration
- Test execution control
- Test monitoring displays and hardcopy reports

As you go through this manual, each function is described in detail.

CYBER'S AUXILIARY DATA ACQUISITION SYSTEM (OPTIONAL)

The Fatigue Master 7000 may also be configured with an optional Cyber Auxiliary Data Acquisition Subsystem (ADAS), for applications requiring synchronized measurement and automatic correlation of external test transducers. Cyber's ADAS operates as an integral part of the Fatigue Master 7000, providing low-level multiplexing and signal conditioning of up to 2048 external transducers.

Chapter 8 provides a detailed description of the ADAS, including instructions on how to setup up and calibrate the ADAS, as well as how to monitor, print, and record ADAS data. Referring back to Figure 1-1 to see how the ADAS is integrated with the FM7000.

When the FM7000 is configured with the ADAS, the following items are included:

- A Measurement Control Unit (MCU)
- A DR11-W interface board within the host computer
- One or more multiplexed data acquisition channels with amplification.
- One or more excitation power supplies for bridge type input channels
- Analog-to-Digital conversion

Digital interface to the MCU

Measurement Control Unit (MCU)

The MCU is the heart of the ADAS system. In addition to providing the interface between the host computer and the measurement hardware, the MCU controls all aspects of data acquisition, including the channel scan rate, post-amp gain settings, data transfer, and channel setup. Presently, a bi-directional DMA interface is used for communication between the host computer and MCU. Setup and control commands flow from the host to MCU, while acquired data is transferred from the MCU to the host. With earlier systems, the host used a serial RS-232 link to send commands to the MCU. **Figure 1-2** shows the ADAS architecture.

The scanning rate of the ADAS is 20,000 samples-per-second (i.e. one channel every 50 æseconds). Thus, the time to scan all channels depends on the number of channels provided with the system. Acquired data is transferred to the host and recorded to disk according to preprogrammed entries in the test procedure. This high rate per channel provides good correlation of data within a complete scan. However, to utilize all the data taken at this rate, continuously, is unnecessary. Therefore, the MCU is presently designed to take a scan of all channels at a rate of approximately 20 times per second (every 50 milliseconds). The system uses every 10th scan (~ every 500 milliseconds) to continually update the test data displays.

Recording ADAS data is described in **Chapter 8.**

The DR11-W Interface

The DR11-W is a bi-directional interface which controls the transfer of data from the MCU to the host, and the transfer of control and scan list data from the host to the MCU. It provides the means for the MCU to operate in a bidirectional DMA mode with the host.
Multiplexed Preamplifier

The mux preamp provides low level amplification of the multiplexed input signal (i.e. low level transducer output). It has gain ranges of 1, 16, or 256.

Post Amp/ADC

The Post Amp/ADC card performs two major functions:

- provides powers of 2 amplifier gains (1, 2, 4, 8, 16, 32) which further amplify the preamp output signal
- Performs analog-to-digital conversion of the high level signal

Calibration Supply

supplies the precision voltage calibration supply used in calibrating the data channels.

Fiber Optic Data I/O

performs interface and data conversion between the MCU and the other master chassis cards

Fiber Optic Adapter

accepts input from the MCU in the form of dual fiber optic cables (receive / transmit) converts serial data to parallel and interfaces with Fiber Optic Data I/O card

Address Driver

performs address decoding for a particular chassis within an RMU

The Multiplexed Input Channels (MUX channels)

The MUX cards configured in the front end measurement hardware provide the signal conditioning and input multiplexing of low level transducer signals (i.e. strain gauges, thermocouples, voltage, etc.). Each MUX card receives and conditions eight (8) inputs and provides local multiplexing. Final levels of multiplexing are performed on the PGA card.

Each input channel accommodates differential signals in the range of ñ5mV to ñ10.24 Volts full scale. Twelve programmable gain ranges between X1 and X2048 are provided. Twelve (12) bit or 16-bit digitizing resolution is available.

Integral bridge completion and shunt calibration is incorporated with each input channel to minimize the number of connections in the signal path, increasing reliability. Terminal lugs are provided for installing a single-point shunt calibration resistor and bridge completion resistors for 1-arm, 2-arm or full bridge configurations. Appendix G provides information on connecting your transducer.

Excitation Power

The ADAS is often configured with one or more excitation power supplies. These units are used in conjunction with the MUX cards and provide common voltage excitation supply for each group of 32 channels. The voltage output of each supply is manually adjustable from 0 to 15VDC.

The ADAS Software

The operation of the Auxiliary Data Acquisition System is controlled by instructions and entries that you define using the Fatigue Master 7000 application software enabled with the ADAS software option. Included in this option is a separate main menu selection labeled ACQUIRE, as well as additional hardcopy reports and appendages to the LCS Test Data display/report.This chapter provides you with all the necessary information needed to set up, calibrate, monitor, and record data using the AuxDAS. Some of the main features of the AuxDAS software include:

- DAS channel definition
- DAS channel calibration
- DAS rosette setup (optional)
- Defining Computed Channels (optional)
- Test monitoring and hardcopy reports

CHAPTER 2

USING THE FATIGUE MASTER 7000 TO CREATE A TEST

Assuming you've turned on the system, as described in Appendix B & C, you should be ready to become familiar with the FM7000 application software and create a test. The first part of this chapter describes the various types of interfaces used in the application software, while the second part describes how to create a new test. Other tasks, including opening existing tests, saving tests, and closing tests, are also described.

The Fatigue Master 7000 software is menu-driven. This means you access each of the software functions by making successive selections from menus that list different options. When you select an option from a menu, the system responds in one of the following ways, depending on the particular option selected:

- The system displays a sub-menu from which you select another option.
- The system displays a work screen which you use to enter or view information regarding the test.
- The system executes the function identified by the option and returns to the same screen, or to a higher-level menu.

The remainder of this manual describes the functions of the Fatigue Master 7000, identifying the entries you make to achieve specific results. This chapter focuses on the general rules for interacting with the system, selecting functions, plus saving and retrieving test files.

Figure 2-1 shows the Fatigue Master 7000 Main menu. This menu is the starting point for all system operations. It is the first menu displayed when you access the system, and it gives you access to these functions:

- **File** lets you open and close tests, create new tests and save them, and exit from the Fatigue Master 7000. The **File** option is discussed in detail later in this chapter.
- **Display** is used to enter, view, or change test parameters. This option is described throughout this manual.
- **Static** lets you create test profiles and perform branching functions using the FM7000's static mode software. These options are described in detail in Chapter 4.
- **Control** lets you execute the test, with options to verify the test definition, start up and monitor the test, and control the individual test channels. These functions are discussed in various chapters, including 6, 7, 9, 10, and 11.
- **Reports** offers a selection of several reports which includes setup data, load conditions, test procedures, and test data. Reports is described in Chapter 13.
- **Acquire** is used to set up the Auxiliary Data Acquisition Subsystem (ADAS) channels, verify input parameters, and calibrate the ADAS. This menu selection is only functional for systems configured with Cyber's auxiliary data acquisition subsystem. Acquire is described in Chapter 8.
- **Cs** gives the software version number and release date.

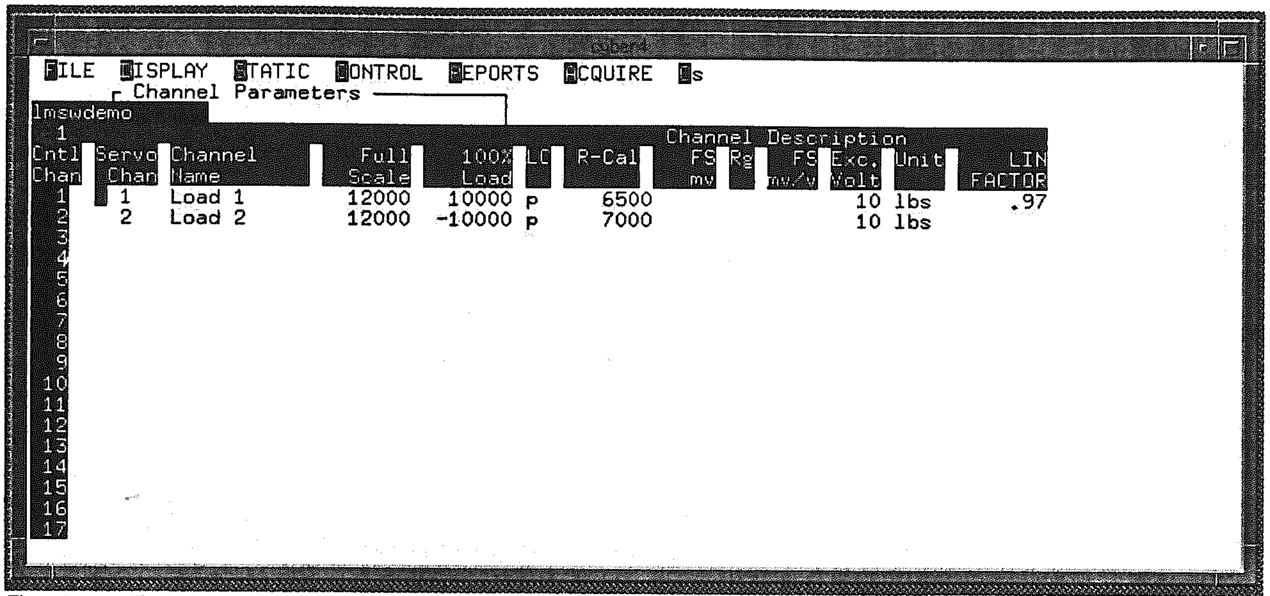


Figure 2-1 The Main Menu Bar

Selecting Menu Options

Whenever the Main menu is displayed, one of the options in it is shown in a highlighted block. This block is a pointer that is used to select different functions; you can move it from one item to the next by pressing the left arrow key or the right arrow key. To select an item from the menu, follow these steps:

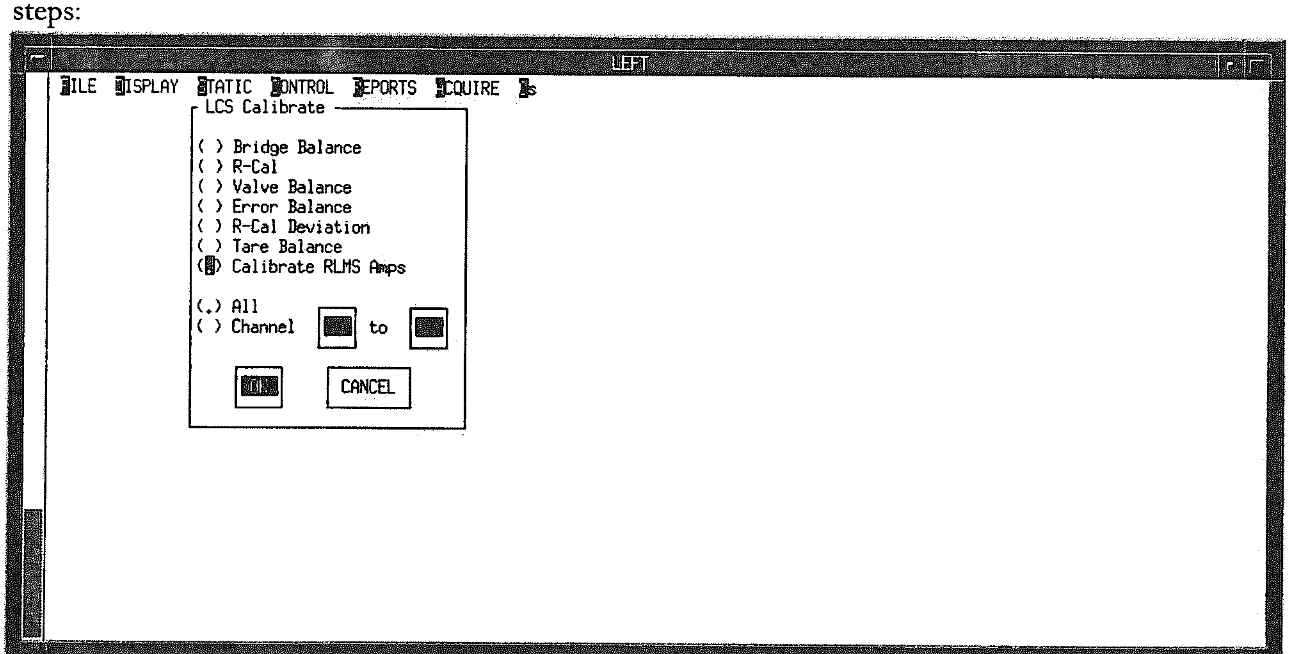


Figure 2-2 Example of a Pull Down Menu

Step 1: Move the highlighted block to the desired Main menu bar option.

Step 2: Press either **RETURN** or the down arrow key. The screen displays the list of functions that may be accessed through that menu choice, as shown in **Figure 2-2.** The highlighted block is located at the first selection in the function list.

Step 3: Move the highlighted block to the desired function by pressing the up arrow or the down arrow key.

Step 4: When the highlighted block is at the desired function, press RETURN to select it.

Alternatively, you can select an item in a menu list simply by typing the first letter of the function. In some lists, however, two or more functions begin with the same letter: in this situation, typing the letter selects the first of the functions beginning with that letter.

When a list of functions is displayed below a menu option, you can press the left arrow key or the right arrow key to display the list of functions available through the adjacent menu options. To return to the Main menu and erase all function lists, press the ESCAPE key (i.e. the F11 key).

USING THE FATIGUE MASTER 7000 SCREENS

After you select a function, the system typically displays a screen that you use to enter or view test information. There are two main types of screens: dialog boxes and spreadsheets. How you enter or edit information depends on the type of screen displayed.

Using Dialog Boxes

Dialog boxes are easy to recognize: a line surrounds the input area of the screen. The line is broken at the upper left where the screen name is inserted. **Figure 2-3** shows a typical dialog box.

With dialog boxes, you enter information on the screen and then store the information by pressing RETURN. Pressing ESCAPE does not store "just entered" information. The Master screen in **Figure 2-3** shows three types of input that are possible with a dialog box-type screen: entry field, button, and checklist.

- Field input areas are shown as highlighted blocks; you simply type information in these highlighted fields.
- Button inputs, sometimes called radio buttons, are pairs of parentheses ( ) corresponding to alternative choices. A dot is located in one of the pairs (□); you use the arrow keys to move the dot up or down, leaving it within the parentheses that correspond to your selection.
- Checklist input areas are pairs of square brackets [ ] that correspond to non-exclusive choices; you select listed items by pressing the space bar, which causes an "x" to appear within the brackets [x]. To de-select the item, press the space bar again: the "x" disappears. With checklist choices, you may select any number of the available options.

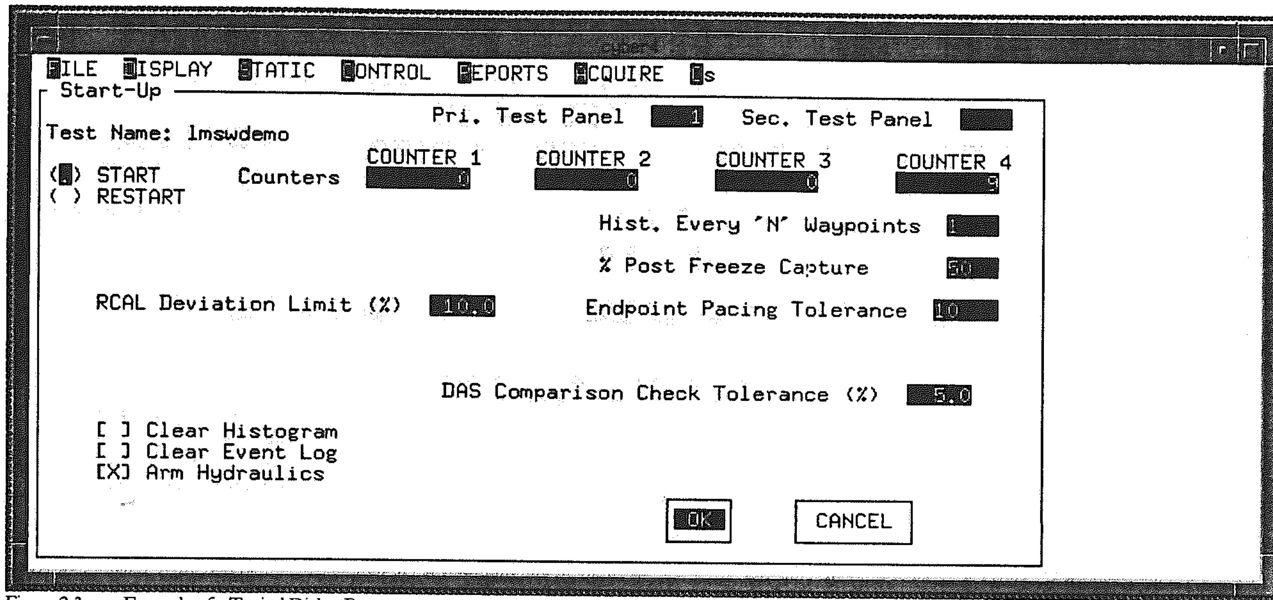


Figure 2-3 Example of a Typical Dialog Box

There is a fourth type of input used with dialog boxes, "list selection". With this type of input, you use the arrow keys to move to a highlighted block, selecting one option from a list of available options. List selection is often used when the selections are not always the same, as when you select a test name from a list of available names. It is similar to the process used to select functions from Main menu options.

When you access a dialog box, the cursor (a small flashing underscore symbol on a alphanumeric console or a colored box on an X-Terminal) is located in the first input area. The cursor shows the input area where your keyboard entries (typed characters, button selection, checklist entries, or list selection) will affect the screen. You access the input areas sequentially, entering appropriate information in each, and then moving on to the next. To move the cursor to the next input area, press **TAB**; to move it to the previous input area, press **SHIFT-TAB**. On X-Terminals, use **<CONTROL> B** to go backwards.

Dialog boxes also contain "action boxes". These are small boxes, with labels such as **OK**, **SELECT**, or **CANCEL**, that determine how the system responds when you finish with the screen. One of the action boxes is highlighted; this is the default action box. You can accept the default action or select another by moving the cursor to a different action box. Use of these boxes is optional. When you press **RETURN** the system executes the action of the dialog box regardless of where the cursor is. **OK** or **SELECT** accepts the information displayed in the input areas on the screen. **CANCEL** cancels the screen, discarding any changes you made and retaining the original information displayed on the screen.

To override the default action box, move the cursor to the alternate action box, using **TAB** or **SHIFT-TAB**, before pressing **RETURN**. Be careful using **RETURN** with a dialog box screen. If you accidently press this key, the current action box is executed, and may result in incomplete or incorrect information being stored for the test. To leave a dialog box without storing new entries, simply press **ESCAPE**.

Using Spreadsheet Screens

The second type of input screen is the spreadsheet-type screen, Figure 2-4, where your entries are alphabetic or numeric characters typed into spreadsheet cells.

With these screens, the up/down/left/right arrow keys are used to move the cursor to the desired cell, which is indicated by a highlighted block. After typing a cell entry, press **RETURN**, an up or down **arrow** key, **TAB**, or **SHIFT-TAB**, (<**CONTROL B**> on X-Terminals) to store that entry. When you finish with the screen, press ESCAPE to store all entries. Each subsequent press of the **RETURN** key will move the active cell in the direction of the last arrow key. Thus, if the last arrow key pressed was the down arrow, each subsequent depression of the **RETURN** key will move the cell vertically down.

*********************************** NOTE ***********************************

The side arrow keys will move you to the next cell provided you are not editing the present cell. In this case, the side arrow keys will move the cursor within the cell.

****

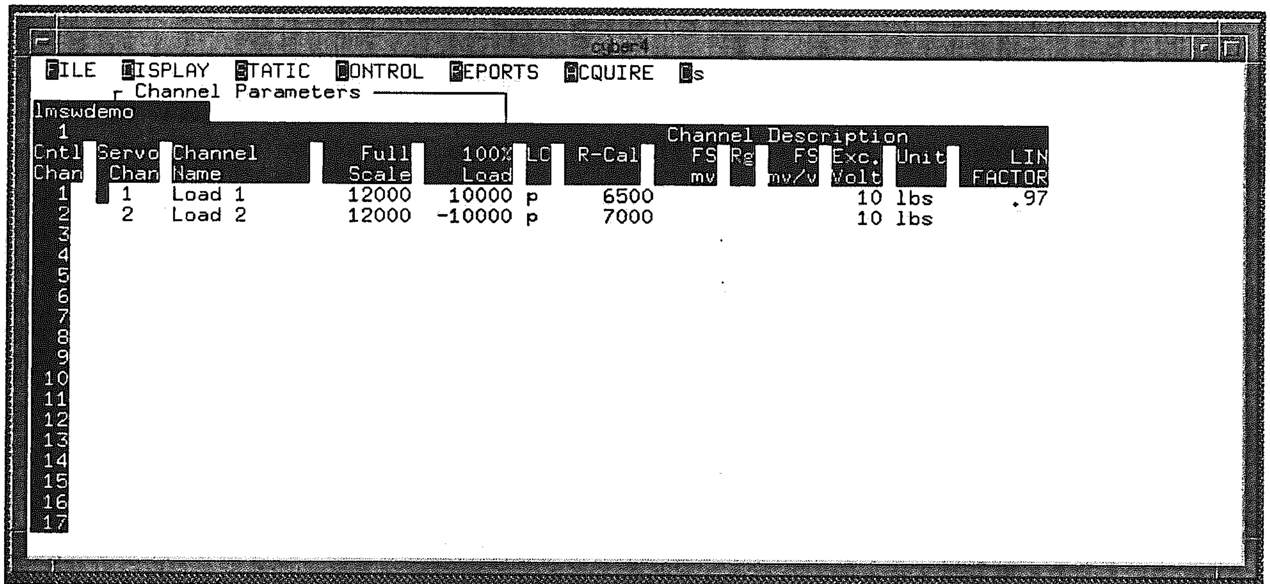


Figure 2-4 Typical Spreadsheet Screen

Using Test Files

Tests are created and stored under a single test name. To access the test definitions, or to execute a test, you must first open the test; to leave the test, you must close it. When you open a test, the system makes the test definition information available to you. You can view or modify the channel and test definitions, and download the test parameters to the TMTM and servo controllers. When you close the test, you remove the ability to view or modify the test information.

Closing a test has no effect on the test after it has been started. Actual test control is accomplished by the TMTM, servo controllers, and test control panel(s). The information in the host computer isn't necessary for operation after it has been downloaded.

This section describes the steps you follow to:

- Create a new test.
- Open an existing test.
- Save a test.
- Close a test.
- Print a test.

************************************* NOTE **********************************
Depending on whether or not passwords and access levels have been assigned (see Appendix H), your individual access level may not be able to access all of the screens. The ability to change test information at your site may be restricted.
****

Creating a New Test

When you create a new test, you simply follow these steps:

Step 1: Access the File menu, New option. The New Test screen is displayed, Figure 2-5.

Step 2: Enter the test name in the Test Name field.

********************************** NOTE ***************************
Spaces are invalid and must be accomplished using underscores.
****

Step 3: Select the Test Type using the arrow key. If you select Static, the additional screens needed to run a static test will be accessible. If you select "Fatigue", only the standard screens will be accessible. Differences between selecting Fatigue or Static are described in Chapter 4.

Step 4: Press **RETURN**. The system creates the new test, automatically opens it, and returns to the Main menu.

Even though the new test has been created, it has no test definition information associated with it. The File operation has only created the test directory under which individual files will be stored. You must still enter the test parameters, as described in **Chapter 3**, before executing the test.

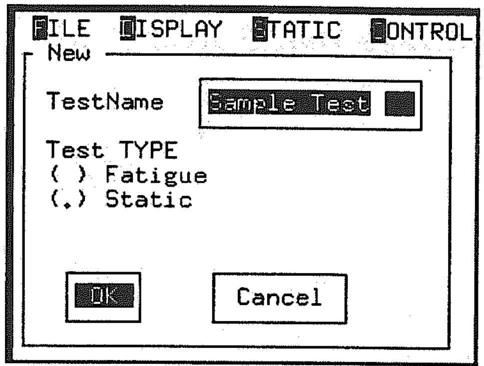


Figure 2-5 New Test Screen

Opening an Existing Test

When you open an existing test, you select the test from a list of available tests. Follow these steps:

Step 1: Access the File menu, Open option. The Open Test screen is displayed, Figure 2-6. The dialog box on this screen has one input field and two action boxes. The SELECT action box is the default.

Step 2: Use the up arrow key or the down arrow key to highlight the desired test. Also, if many tests are listed, use <NEXT SCREEN/PREV SCREEN> or <NEXT PAGE/PREV PAGE> to go through the list more quickly.

Step 3: Press **RETURN**. After a short pause, the system opens the test and returns to the Main menu. Each time a test is opened, a temporary directory is created, and all setup files are copied from the original testname directory to the newly created temporary directory.

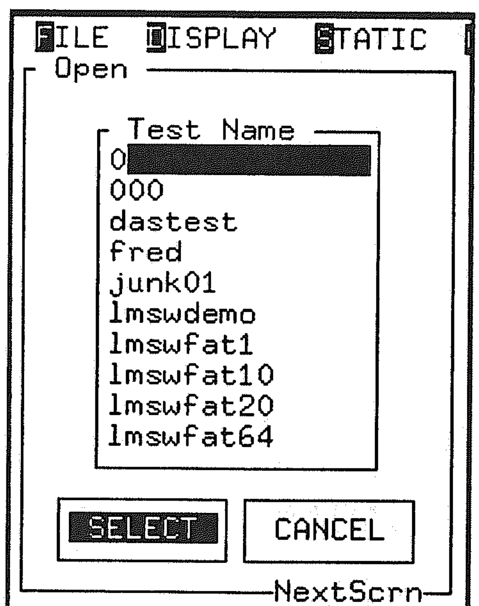


Figure 2-6 Open Screen

Saving a Test

Whenever you make changes to test definitions, you must save the changes before closing the test: if you don't, the changes are lost and are unavailable the next time you open the test. You can save a test in two ways:

- You can save the test using the original test name. This method updates the existing test definition with any changes you make prior to saving the test.

- You can save the test using a different test name. This method makes a copy of the test stored under the original name, and updates the copy with any changes you make prior to saving the test.

A test must be open before you can save it. It remains open, after you save it, until you close it. To save a test, follow these steps:

Step 1: Select the File menu option. The File menu is displayed. This menu differs from the File menu displayed when you are opening a test: since a test is already open. The New option and the Open option are omitted, and four options, Close, Save, Save As, and Quit are added.

Step 2: To save the test under the original test name, select the Save option and press RETURN. The system saves the test and returns to the Main menu. To save the test under a new name, select the Save As option. The system displays the Save As screen . Each time you save a test, the setup files are copied from the temporary directory to the testname directory.

Step 3:If you are saving the test under a new name, enter the new name in the TestName box, then press RETURN. The system saves the test under the new name and returns to the Main menu.

Closing a Test

When you finish working with a test, either defining the test parameters or running the test, you must eventually close the test. You may only close a test that is currently open.

************************************ NOTE ************************************

Closing a test is different in Password Protected Systems. Refer to Appendix H.

****

To close a test, simply select the File menu, Close option. If no changes were made to the setup since the test was last saved, the system closes the test and returns to the Main menu. Each time a test is closed, the temporary directory is deleted. If test items were altered, you will be asked if you wish to save changes as shown in Figure 2-9.

The system will also ask if you wish to save the Test Data file. If you wish to save the Test Data file, select YES and press return. Another dialog box, appears and prompts you to enter a filename under which this Test Data file will be recorded. All test data files for a given test are stored on the computer's hard disk under "/usr/dlcs/status/<testname>".

LEAVING THE FATIGUE MASTER 7000 APPLICATION SOFTWARE

Normally, the FM7000 runs continuously on the host computer; there is no need to exit from the software and return to the UNIX operating system.

************************************ CAUTION ******************************

Though the system can continue executing the current test once you've left the application software, exiting from the FM7000 may compromise the integrity of the test control environment, since visibility to test data is reduced.

****

Follow the steps below to exit the FM7000 application software:

Step 1: Select the File menu, Quit option.

Step 2: If the test was previously closed, the system leaves the application software and returns to the UNIX operating system shell.

If the test was not previously closed, a dialog appears and asks you if you wish to save changes. Take desired action by tabbing to **YES** or **NO** and pressing **RETURN**.

If you selected **YES**, the system will also ask you if you wish to save the Test Data file. If so, provide the requested information and press **RETURN**. The system leaves the application software and returns to the UNIX operating system shell.

CHAPTER 3

DEFINING THE CONTROL CHANNELS & TEST-

RELATED INFORMATION

The first task in running a test is to define the parameters that control it. This chapter takes you through that process, explaining the initial procedures required to set up a test. Many of the same steps described in this chapter may also be used to view or change the parameters of an existing test. If necessary, refer back to Chapter 2 for any questions you may have about interacting with the FM7000 screens.

The following steps are performed to set up a typical test:

1. Set up report headings
2. Assign channel numbers to the physical channels, and enter the full scale load, 100% load, feedback gain, and excitation voltage for each channel.
3. Set up loop parameters (proportional gain, integrator time constant, etc.)
4. Define the limits for feedback and loop-error for each channel.

*********************************** NOTE ************************************
Only item 2 above is required to run a test, since default values can be used for all other entries. It is also important to note that once you've entered the information described in this chapter and have downloaded the test (Chapter 6), many of the entries can not be changed without aborting the test and re-downloading. APPENDIX K provides a list of which parameters may be changed on-line.
****

??
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

□

DEFINING THE REPORT HEADINGS AND ENDPOINT COUNTERS

The FM7000 lets you create the headings that print at the top of hardcopy reports generated from the test. In addition, you may assign descriptive names to each of the endpoint counters. These names will also appear on the Master screen and Test Data display above the individual counter numbers.
There are a total of four counters. Counter 4 is a dedicated counter which is incremented at each step in the test procedure. Counters 1, 2, and 3 are user- definable and may be incremented by one or more steps in the test procedure. Each time counter 1, 2, or 3 is incremented, all higher-numbered counters are reset to zero.

Your test scenario should suggest appropriate names. For example, in an aircraft fatigue testing environment one might name the counters Lifetimes, Flights, Blocks, and Segments.

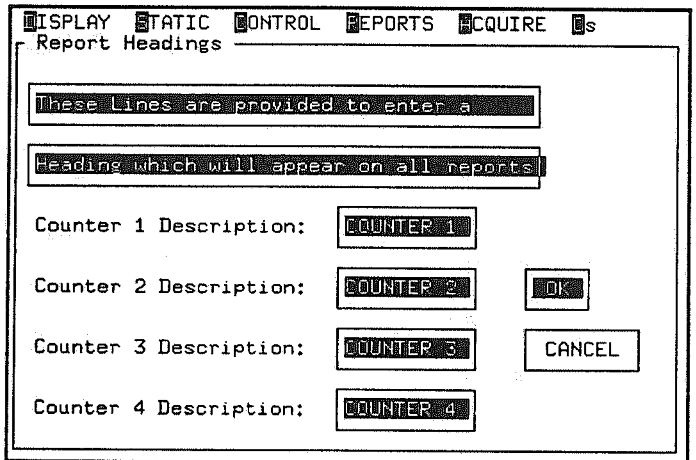


Figure 3-1 Defining Report Headings and Counter Labels

Follow these steps to create the report heading and name the endpoint counters:

Step 1: Select the Display menu, Report Headings option. The Report Headings screen, **Figure 3-1**, is displayed.

Step 2: Enter the report heading in the two boxes at the top of the screen. Use the upper box for the first line of the heading and the lower box for the second.

Step 3: Enter the counter names in the four counter description boxes. Remember that counter #4 is the lowest level counter, and is incremented the most frequently.

Step 4: Select OK or press **RETURN** when finished.

DEFINING TEST CONTROL CHANNELS

At this point you are ready to define the individual control channels to be used in the test. To accomplish this, you will perform the following tasks:

- Link a logical control channel number with a physical servo channel
- Define individual channel names for each channel in the test
- Define full scale values and 100% load values for each control channel
- Define how the feedback gain is to be established for each control channel (i.e.
- through an RCAL procedure or by setting the full scale feedback range directly).
- Define the excitation voltage and engineering units label
- Define the individual loop error and feedback limits for each control channel
- Define the individual control parameters for each control channel
- Define the individual tuning parameters for each control channel

Follow these steps to name each channel and enter its full scale load value:

Step 1: Select the Display menu, Channel Parameters option. The dialog box shown in **Figure 3-2** is displayed. Select Channel Description. The Channel Description screen, Figure 3-3, is displayed.

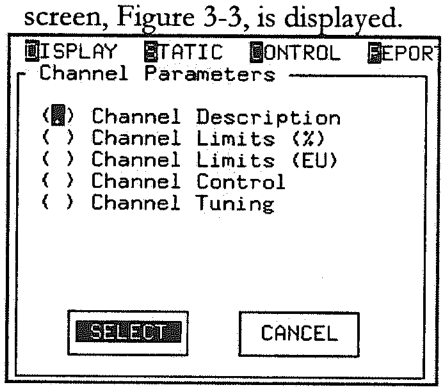


Figure 3-2 Channel Parameters Dialog Box

Step 3: Move the cursor to the line that corresponds to the control channel you want to enter. The control channel numbers are listed in the column at the left side of the screen: a total of 17 may be displayed per page. If the channel number you want to define is not shown on the screen, use the down arrow key or <NEXT PAGE> key to scroll to additional channels. Use the up arrow key or <PREV PAGE> key to return to the first group of channels.

Step 4: Enter the physical channel number in the column Servo Chan. This number corresponds to the physical location of the channel in the equipment cabinet.

Observe the following restrictions when entering Servo Chan:

- You must begin channel numbering with Cntl Chan #1.
- You must assign a logical channel (Cntl Chan) number to each servo channel used in the test.
- Servo Channels must be assigned to Control Channels (Cntl Chan) in a contiguous order; blank fields in the Servo Chan column are not allowed

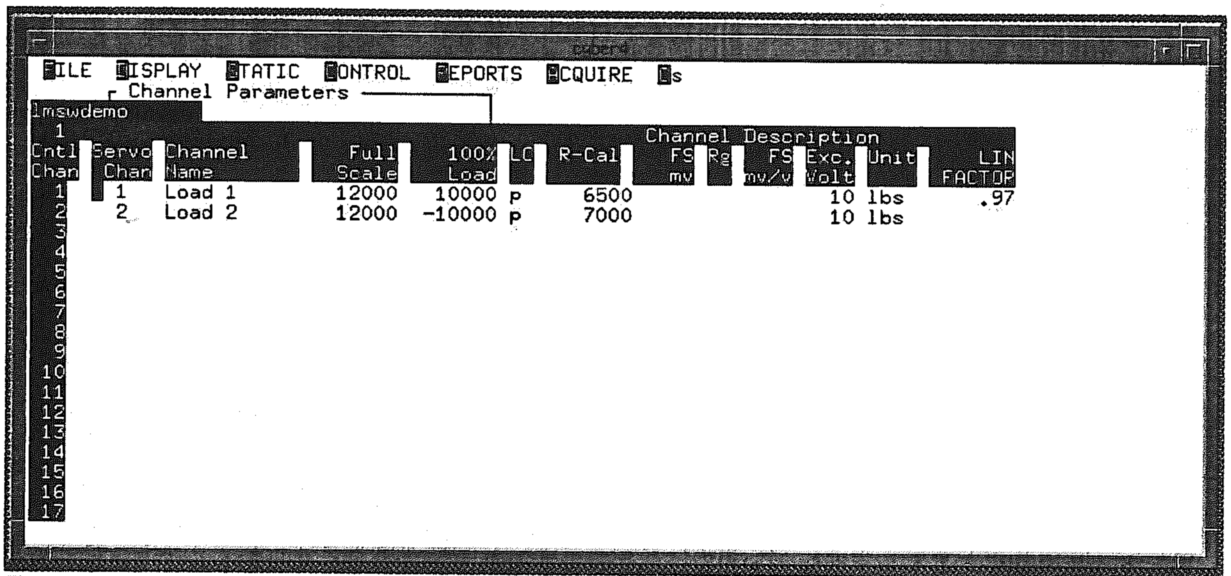


Figure 3-3 Channel Description Screen

Step 5: Enter a descriptive channel name which will be used throughout various data displays in the column labeled Channel Name.

Step 6: Enter the full scale for the channel in the column labeled Full Scale. This value may represent load or displacement. To the servo, the number entered for full scale is always equivalent to 10 Volts and 2000 counts.

Step 7: Enter the 100% Load value in the column labeled 100% Load. The 100% Load entry is made in engineering units according to structure design limit. This entry is used in conjunction with the Full Scale value. The 100% Load entry may range from 1 to 0.5 times the full scale entry, depending on the range of desired test load required in the test profile. For example, if 10,000 is entered as the 100% Load for a channel with 20,000 for full scale, this will allow for ñ200% test load (i.e. -20,000 to +20,000). However, if the full scale was 15,000 in the above example, only ñ150% test load would be permitted.

******************************** NOTE ********************************

If your test has been defined as static, you may simplify the setup procedure by using this column to specify which channels are to have positive load condition (command) values and which are to have negative load condition (command) values. Further information on how the sign (ñ) of this entry is used by the system is explained in Chapter 4, section Using the Static Mode Software to Create/Modify the Load Spectrum.

****

Step 8: Enter the method in which the load conditions are to be displayed; engineering units or as a percent of 100% Load. If engineering units are desired, type E in the E/P column. Type P if percent is desired. Though each channel can be individually defined with E or P, it is best to have all channels in the test use the same format (i.e. all P or all E).

Step 9: There are 3 methods available for establishing the feedback gain of each channel; through an automatic **RCAL** procedure for low-level bridge type transducers, by setting the full scale in millivolts, or by setting the full scale based on a "millivolt-per-volt of excitation" factor. Choose which method you wish to use and enter the appropriate information according to the following descriptions:

RCAL

□

If you are using a load cell or other low-level bridge-type transducer, you may use the RCAL method to calibrate the control channel. This procedure requires you to have installed the an RCAL shunt resistor on the rear of the servo chassis. If so, enter the corresponding RCAL value, in engineering units, in the column labeled RCAL. The RCAL value should equal the change in engineering units caused by the application of the RCAL shunt resistor across one arm of the load cell bridge.

FULL SCALE, MILLIVOLTS (FS, MV)

If you know the full scale output of your transducer and wish to set the full scale feedback input range directly, enter the desired full scale value, in millivolts, in the column labeled FS, mv. If you enter the feedback full scale in this manner, you must also define the feedback range in the column labeled RGE. Enter H to select high level feedback (e.g. LVDTs) or L to select low level feedback (e.g. bridge type load cells).

NOTE: An entry made in FS, mv or FS, mv/v automatically reloads the feed back gain DAC in the servo upon download, overwriting any saved feedback gain setting in that channel.

FULL SCALE, MILLIVOLT-PER-VOLT (FS, MV/V)

If you wish to set the full scale based on the ratio of the change in load cell output to the change in excitation voltage input, enter the desired value in the column labeled FS mv/v. The result of multiplying this entry times the excitation voltage entry is what the feedback full scale millivolts will be set to.

****************************** [redacted] ****************************

The servo uses one of two available feedback paths; low level or high level. Selection of which path to use is made by a physical slide switch or jumper on the side of each servo module. Normally, the servo controllers are delivered with this switch in the low level position. However, if you choose to use the "FS, mv" with "H" selected, you must make sure that the High Level feedback path is selected.

Rules:If the full scale output of the transducer is between ñ40 mV, set the feedback switch the low level position and enter L in the RGE column.

If the full scale output of the transducer is between 0.1-10 Volts, set the feedback switch to the high level position and enter H in the RGE column.

****

Step 10: Enter the excitation voltage applied to the transducer in the column labeled Excit. Volt.

Step 11: Enter the engineering units, for example, "lbs" or "KIPs," in the column labeled Units.

Step 12: When you finish defining all channels, return to the sub-menu by pressing **ESCAPE** (i.e. F11).

SETTING CONTROL CHANNEL LIMITS

You can assign two types of limits to each control channel. These limits ensure that the performance of the servo loop remains within the tolerances that you define. In this procedure, you set:

- Loop error limits that restrict how much the loop feedback signal may deviate from the command signal.
- Maximum and minimum values of the feedback signal.

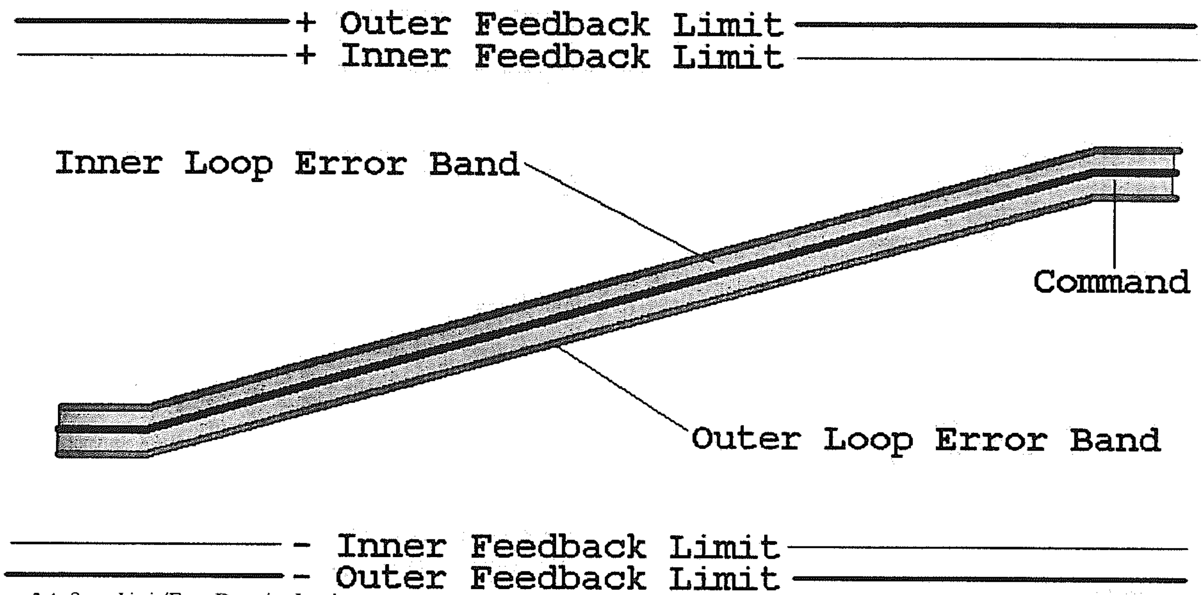


Figure 3-4 Servo Limit/Error Detection Levels

You set an inner and outer detection level, for both loop error and feedback limits, as shown in Figure 3-4. The inner value is typically used as a warning indication while the outer value is normally used for hydraulic shutdown. Outer error and outer limit detectors are latching conditions, while inner error and inner limit are non-latching conditions. You reset latched conditions by using the Reset Error Conditions selection button on the Master screen, discussed in Chapter 9 and Appendix I.

Exceeding any of the above limits causes the servo controller to issue a fault condition (Fault 1 or Fault 2) to the TMTM. Depending on how conditional actions are defined, the fault condition can trigger one or more of the following actions:

Freeze History Data
Hold
Print
Ramp to Condition 1
Ramp to Condition 2
Lock
Dump

Defining system responses to out-of-limit conditions is described in Defining Conditional Actions. in Chapter 5.

Follow these steps to set individual limits for each control channel:

Step 1: Select the Display menu, Channel Parameters option. When the dialog box is displayed, select Channel Limits (%). The Channel Limits (%) screen, **Figure 3-5**, is displayed.

******************************* NOTE ******************************

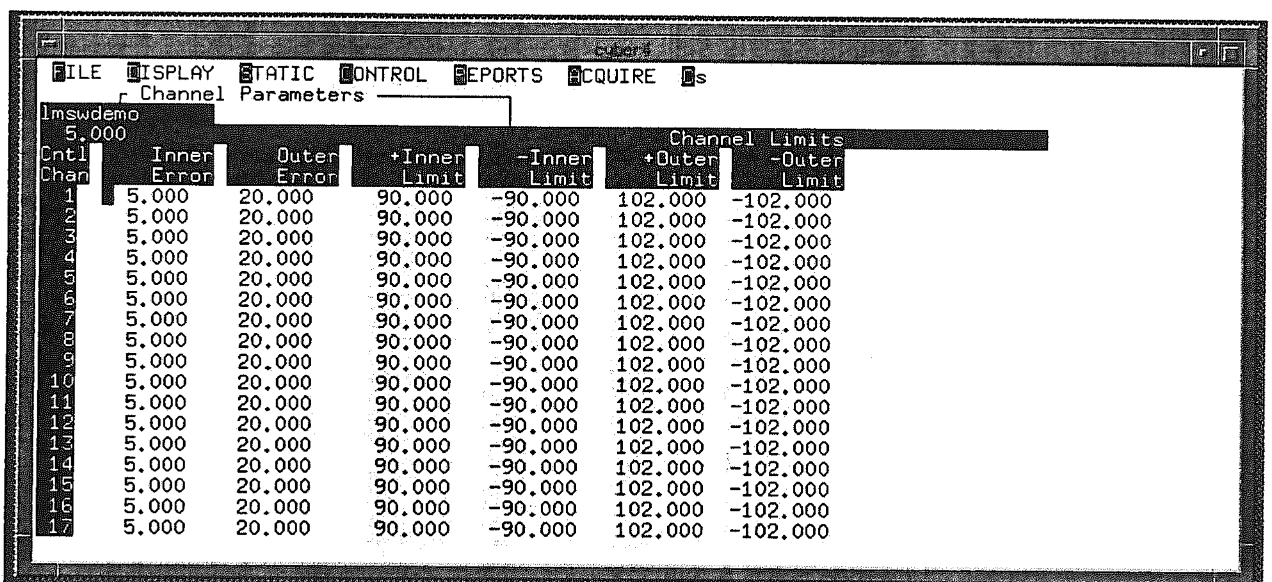
In order to correctly define or interpret the limits for each channel, you must be aware that each channel has a different min/max limit depending on the ratio (R) derived from [Full Scale ö 100% Load]. In all cases, the min/max limit is equal to ñ (R x 102.3%). For example, if full scale equals 10,000 lbs and 100% load equals 5,000 lbs, then R equals 2. Using the formula above, the min/max limit entry is equal to ñ(2 x 102.3%) = ñ204.6% of the 100% load entry. The allowable range of R is 1 ó Ró 2. Default values are automatically used for limit/error entries unless otherwise changed.

****

Step 2: Move the cursor to the line that corresponds to the controller channel you want to enter.

Step 3: Enter, as a percentage of the 100% load entry, the inner loop error in the Inner Error column, and the outer loop error in the Outer Error column.

Step 4: Enter, as a percentage of the 100% load entry, the positive inner feedback limit in the + Inner Limit column and the negative inner feedback limit in the - Inner Limit column. See note below.



Cntl Chan	Inner Error	Outer Error	+Inner Limit	-Inner Limit	+Outer Limit	-Outer Limit
1	5.000	20.000	90.000	-90.000	102.000	-102.000
2	5.000	20.000	90.000	-90.000	102.000	-102.000
3	5.000	20.000	90.000	-90.000	102.000	-102.000
4	5.000	20.000	90.000	-90.000	102.000	-102.000
5	5.000	20.000	90.000	-90.000	102.000	-102.000
6	5.000	20.000	90.000	-90.000	102.000	-102.000
7	5.000	20.000	90.000	-90.000	102.000	-102.000
8	5.000	20.000	90.000	-90.000	102.000	-102.000
9	5.000	20.000	90.000	-90.000	102.000	-102.000
10	5.000	20.000	90.000	-90.000	102.000	-102.000
11	5.000	20.000	90.000	-90.000	102.000	-102.000
12	5.000	20.000	90.000	-90.000	102.000	-102.000
13	5.000	20.000	90.000	-90.000	102.000	-102.000
14	5.000	20.000	90.000	-90.000	102.000	-102.000
15	5.000	20.000	90.000	-90.000	102.000	-102.000
16	5.000	20.000	90.000	-90.000	102.000	-102.000
17	5.000	20.000	90.000	-90.000	102.000	-102.000

Figure 3-5 Channel Limits (%) Screen

Step 5: Enter, as a percentage of the 100% load entry, the positive outer feedback limit in the **+ Outer Limit** column and the negative outer feedback limit in the **- Outer Limit** column.

Step 6: When you finish defining all channels, return to the Display menu by pressing the **ESCAPE** key (**F11**).

Displaying the Channel Limits (EU) Screen

Though the individual limits for each control channel are defined as a percent of each channels 100% Load entry, you may display the limits in equivalent engineering units.

************************************ NOTE ************************************
This screen will display zeros until you have downloaded the setup information and performed the necessary calibration procedures.
****

Select the Display menu, Channel Parameters option. When the dialog box is displayed, select Channel Limits (EU). The Channel Limits (EU) screen, is displayed.

If your test includes many channels, you may view any channel simply by using the PREV/NEXT selection.

If you wish to obtain a hardcopy of the display, select [PRINT] and press RETURN.

Setting Control Channel Parameters

The Fatigue Master 7000 includes a variety of ways to condition the performance of the channel. In this section, you set

- **Proportional gain** of the control loop amplifier.
- **Integrator time constant**. The integrator provides additional gain as the servo loop reaches the endpoint condition and the loop error approaches zero. You set the time for the integrator function to become active.
- **Rate compensation**. This function provides additional gain as a function of the rate of change of the command signal.
- **Piston area ratio**. This is the proportional difference between the piston area on both sides of the actuator (tension/compression). The system uses this ratio to compensate for slightly different performance characteristics that depend on which direction the piston is moving.
- **Input scaling**. This entry allows you to individually scale the command of any channel between 0 and 100% of its full scale value.
- **Input offset value**. This entry allows for applying a load offset to the command of each channel in engineering units.
- **Tare offset**. This entry allows you to establish a "test article zero" as part of the servo controller calibration by offsetting the weight associated with the actuator, fixturing, chains, etc..

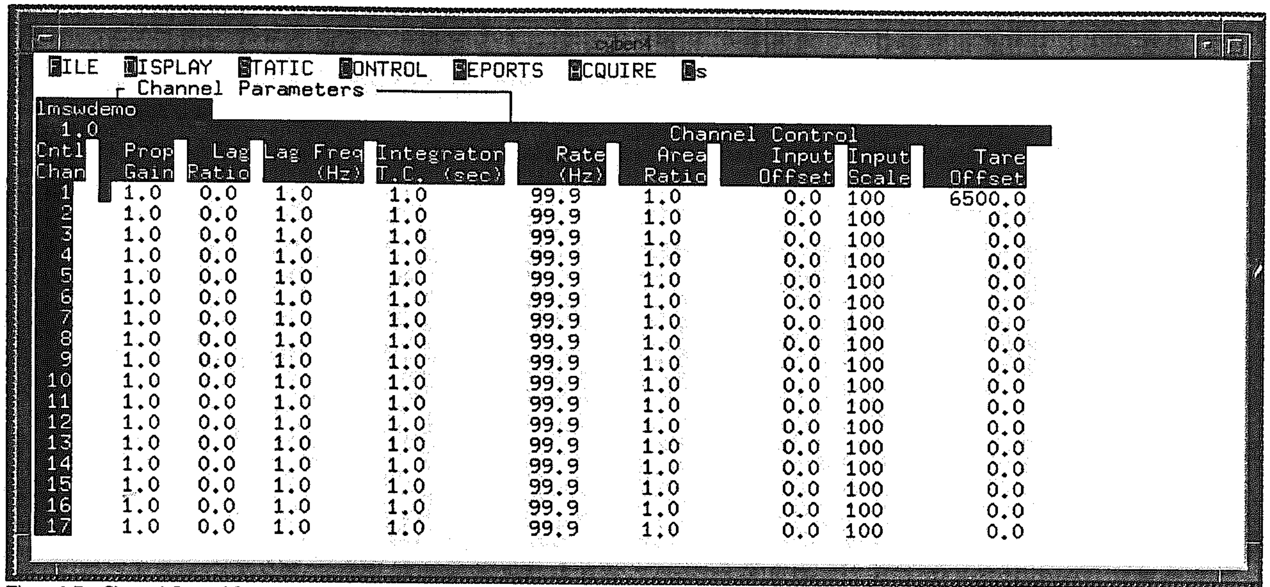
Follow these steps to set the channel gain and input parameters.

Step 1: Select the Display menu, Channel Control option. When the dialog box is displayed, select Channel Control. The Channel Control screen, **Figure 3-7**, is displayed.

Step 2: Move the cursor to the line that corresponds to the controller channel you want to enter.

Step 3: Enter the proportional gain of the control loop amplifier in the Prop Gain column. The allowable range of proportional gain is 0.25 to 127.9.

Step 4: Enter the integrator time constant in the Integrator T.C. (sec) column. This is the time required for the integrator to become active in removing loop error. The allowable range of the integrator time constant is 0.1 to 10 seconds. Please note that the integrators are enabled on a per channel basis, as well as on a global basis. Enabling/disabling the integrators is described in subsequent sections.



Cntl Chan	Prop Gain	Lag Ratio	Lag Freq (Hz)	Integrator T.C. (sec)	Rate (Hz)	Area Ratio	Input Offset	Input Scale	Tare Offset
1	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	6500.0
2	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
3	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
4	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
5	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
6	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
7	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
8	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
9	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
10	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
11	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
12	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
13	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
14	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
15	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
16	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0
17	1.0	0.0	1.0	1.0	99.9	1.0	0.0	100	0.0

Figure 3-7 Channel Control Screen

Step 5: Enter the rate compensation frequency in the Rate (Hz) column. This provides additional gain in the command signal when the command is changing rapidly, thus increasing dynamic response. This type of compensation is typically used in fatigue or cyclic testing applications. The allowable range of rate compensation is 0.1 to 99.9 Hz. For static tests, inhibit this feature by using the default value of 99.9 Hz.

Step 6: Enter the piston area ratio in the Area Ratio column. This is the ratio between the piston areas on the tension and compression sides of the hydraulic cylinder, respectively (i.e. A= Tension/compression). The allowable range of area ratio is 0.5 to 1. For example, if equal-area actuators are used, enter a 1 in this column. If the tension side of the actuator is half the area of the compression side, enter 0.5 in this column.

Step 7: If you want to scale the command signal between 0 and 100% of its full scale value, use the Input Scale column to enter a number between 0 and 100. This entry can come in very handy in the middle of the test, for example, if you want to exclude a channel(s) from further load application. This is easily accomplished by setting the input scale to 0%, thus causing all subsequent command signals for that channel(s) to be 0.

Step 8: Enter the input offset value in the Input Offset column. This is a static offset which gets summed with the command signal. The allowable range of input offset is - full scale to + full scale.

Step 9: If you wish to establish a "test article zero load condition" by offsetting the weight associated with the actuator, fixturing, chains, etc., enter this weight value in the Tare Offset column. This entry works in conjunction with the Tare Balance calibration routine described in Chapter 5.

Step 10: When you finish defining control parameters for all channels, press **ESCAPE (F11)** to return to the sub-menu.

Defining the Channel Tuning Parameters

The last step in defining the test channels is to set the tuning parameters. This section describes how you set:

- **Bode phase margin**. This is the difference between the phase-angle of the loop at unity gain and the phase-angle at 180 degrees. The Bode phase margin is used in the Bode estimation calculation of the pre-tune calibration function. The Bode estimator adjusts the proportional gain such that the servo loop maintains the specified phase margin. See note on page 3-14.
- **Enable/disable auto-tuning on a per channel basis**. When enabled, the system automatically adjusts the proportional gain of the loop based on the criteria selected.
- **Define the oscillation criteria**. This criteria includes the number and size of error signal peaks 6(oscillations) that may be encountered in a test step. If this criteria is met, the auto-tuning function automatically lowers the gain of the proportional amplifier until oscillations cease or until a minimum gain cap is met.
- **Enable/disable the integrator on a per-channel basis**. This provides additional gain for the valve driver as the loop reaches the endpoint condition.
- **Set the minimum and maximum gain caps for auto-tuning**. If auto-tuning is active, these entries limit the minimum and maximum gain range that the control channel can achieve during the test.

Follow these steps to set the channel tuning parameters:

Step 1: Select the Display menu, Parameters option. When the dialog box is displayed, select Channel Tuning. The Channel Tuning screen, **Figure 3-8**, is displayed.

Step 2: Move the cursor to the line that corresponds to the controller channel you want to enter.

Step 3: Enter the Bode phase-shift in the Bode Phase column. Once again, this is the difference between the phase-shift of the loop at unity gain and the phase-shift at 180 degrees. By setting the Bode Phase appropriately, you allow the Bode estimator to optimize the loop performance.

******************************** NOTE ********************************

Please use the default Bode Phase entry with this software update.

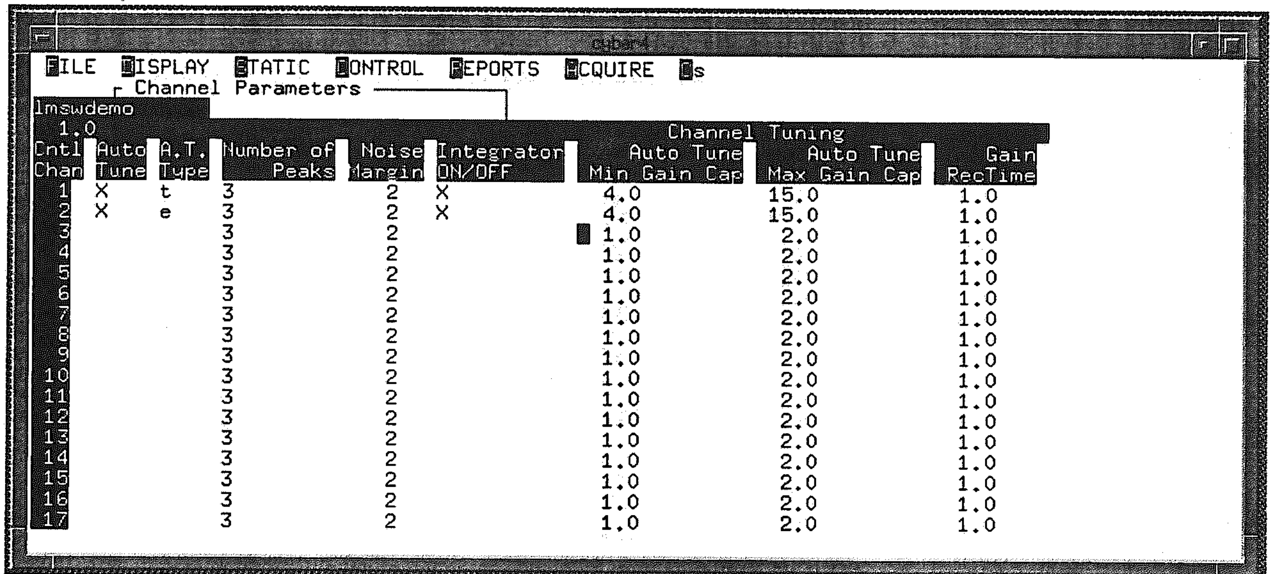
****

Step 4: Enable auto-tuning, if desired, by pressing the space bar while the cursor is in the Auto Tune column. Leave the column blank to disable auto-tuning for that channel. For additional information on how auto-tuning works, refer to **Appendix J**.

Step 5: Enter the maximum number of error signal peaks (or sign reversals) per cycle in the Number of Peaks column. If the system exceeds this number of peaks (i.e. oscillations), auto-tuning (if enabled) automatically reduces the gain of the proportional amplifier. The range of allowable entries is 1 - 10.

Step 6: Enter the cutoff level for defining an error signal peak in the Noise Margin column. The system ignores peaks that occur within this limit, counting only those peaks that exceed it. The range of allowable entries is 0 to full scale.

Step 7: Enable the loop integrator function for that channel, if desired, by pressing the space bar while the cursor is in the Integrator ON/OFF column. Leavethe column blank to disable the integrator for that channel.



Cntl Chan	Auto Tune	A.T. Type	Number of Peaks	Noise Margin	Integrator ON/OFF	Auto Tune Min Gain Cap	Auto Tune Max Gain Cap	Gain RecTime
1	X	t	3	2	X	4.0	15.0	1.0
2	X	e	3	2	X	4.0	15.0	1.0
3			3	2		1.0	2.0	1.0
4			3	2		1.0	2.0	1.0
5			3	2		1.0	2.0	1.0
6			3	2		1.0	2.0	1.0
7			3	2		1.0	2.0	1.0
8			3	2		1.0	2.0	1.0
9			3	2		1.0	2.0	1.0
10			3	2		1.0	2.0	1.0
11			3	2		1.0	2.0	1.0
12			3	2		1.0	2.0	1.0
13			3	2		1.0	2.0	1.0
14			3	2		1.0	2.0	1.0
15			3	2		1.0	2.0	1.0
16			3	2		1.0	2.0	1.0
17			3	2		1.0	2.0	1.0

Figure 3-8. Channel Tuning Screen

Step 8: Using the columns labeled Min Gain Cap and Max Gain Cap, enter minimum and maximum gain caps for that control channel, if desired. These caps are used in conjunction with the auto-tuning function and allow you to restrict the range of gain which the servo can establish for that channel. Oscillations will still cause the gain to be reduced, but the gain will go no lower than the minimum cap, even if oscillations persist. If gain is reduced due to an occasional oscillation, auto-tuning will "fight back" to bring the gain up to the maximum cap. If no entries are made, auto-tuning will adjust gain based on oscillation criteria only. The range of allowable entries is 0.25 - 127.9.

Step 9: When you finish defining tuning parameters for all channels, return to the sub-menu.

When you conclude this step, you have completed the definition of the test channels. You're ready to go on to defining the load spectrum.

???
REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
???

CHAPTER 4

DEFINING THE LOAD SPECTRUM

This chapter provides a step-by-step description on how to:

- create the load spectrum by manually defining the individual load conditions and test procedure
- use the FM7000 Static software to automatically create the load conditions and test procedure for static tests
- use the FM7000 Static software to modify the spectrum, on-line
- use the FM7000 Branching function to move around within a predefined test procedure, on-line

??
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

Understanding the FM7000 Spectrum

In order to aid your understanding, the following paragraphs provide a description of what information is included in the spectrum, and how the system uses this information.

The actual waveform generation by each servo controller involves several processes throughout each system element. The first step is accomplished through the operator's input of the setup data. This includes defining each control channel, as well as all of the information associated with the loading spectrum. Spectrum information consists of a Load Conditions file and a Test Procedure file. A load condition is a predefined set of command values for each control channel in the test. Below is an example of how load conditions are defined:

	Ch#1	Ch#2	Ch#3	'	Ch#n
load condition #1	1000	2000	-1500	'	1450
load condition #2	3000	-180	500	'	-600
'	'	'	'	'	'
'	'	'	'	'	'
load condition #n	-100	4000	3000	'	-900

When running static tests, the system allows for defining up to 8000 unique load conditions per test. For fatigue tests, virtually an unlimited number of load conditions may be used. Figure 4-2 provides an example of the screen used to manually define each load condition. The load conditions are eventually downloaded to the servo controllers, where they reside in memory.

The Test Procedure file is used to define the sequence in which the load conditions are to be applied. The Test Procedure screen, shown in Figure 4-3, allows for manually defining the individual test steps, each of which includes as a minimum:

- a block# and step #
- a load condition number
- a load transition time
- a command waveform type (i.e. ramp, haversine, or modified ramp).

For static tests, the test procedure file can include up to 2000 steps. The test procedure information is eventually downloaded to the LCS computer located in the TMTM chassis, where it resides in memory.

For long-term fatigue testing applications requiring more than 2000 test steps, the FM7000 can also operate in what is referred to as a "large spectrum" mode. This mode allows the test procedure and load conditions to be downloaded in "chunks", with a "chunk" being equivalent to 64 steps of the test procedure. When operating this way, the FM7000 can accommodate spectrum files with virtually an infinite number of load conditions and/or test procedure steps. For more information on the Large Spectrum mode, see Appendix G.

Initiating the test from the test control panel causes the TMTM to execute the test procedure one step at a time. With each step, a load condition number is broadcasted to all of the servo controllers, along with the preprogrammed transition time and waveshape. Each servo controller uses this information to generate its individual command waveform.

Up to this point we've only discussed the individual screens used to define the load spectrum, manually. However, the FM7000 includes software specifically tailored for setting up and running static tests. Through several simple operator entries, including the target test load, the loading rate, and a loading increment, the system automatically generates the desired loading profile, specifically the individual load conditions and test procedure steps. This eliminates the need to enter this information manually, thus expediting the overall setup procedure.

********************************** NOTE ***********************************

To simplify the manual, this section describes how to manually define the load spectrum using the Load Conditions and Test Procedure screens. Once you've become familiar with this approach, later sections describe how to use the Static Mode to automatically create the load conditions and test procedure.

****

Manually Defining the Load Conditions

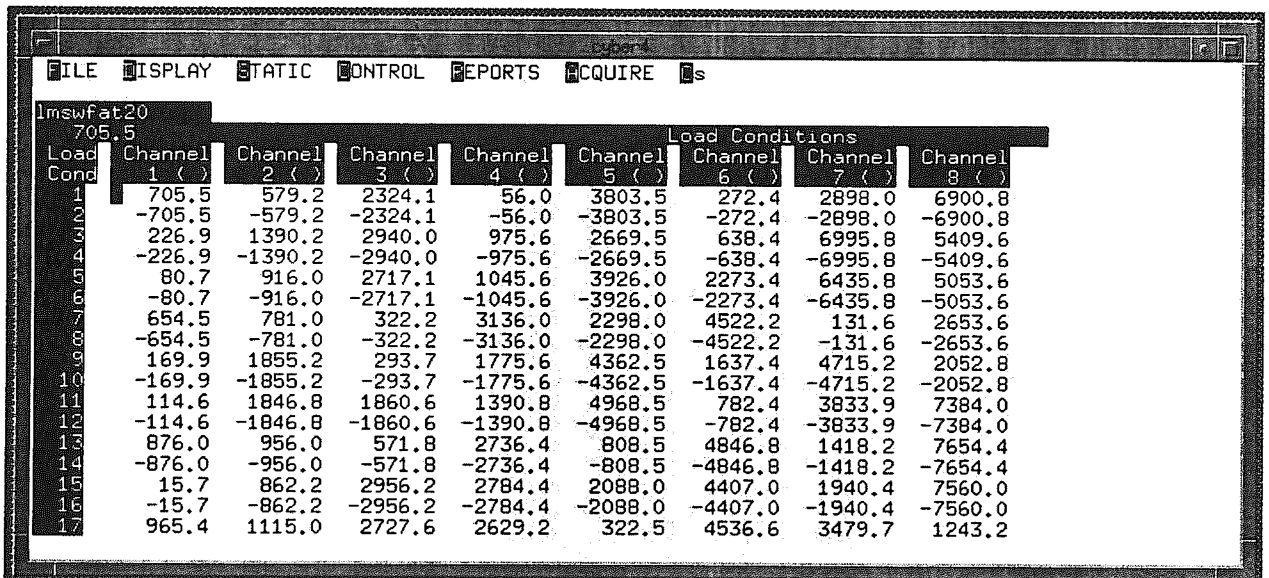
Load Conditions may be defined manually by following these steps:

Step 1: Select the Display menu, Load Conditions option. The Load Condition Channel Selection screen, is displayed. Allow sufficient time for the screen to appear as space is being allocated for 8000 load conditions.

Step 2: Enter the number of the range of channels for which you want to enter load conditions and select OK. For example, if you want to enter or modify the load condition for channel 20, enter 20. The Load Conditions screen, **Figure 4-1**, is displayed. This screen displays a block of up to 8 channels, including the channel you entered. So in our example above, channels 17 through 24 will be displayed.

Step 3: Move the cursor to the line corresponding to the condition number you want to define. Condition numbers are listed in the Load Cond column at the left side of the screen.

If the condition number you want isn't listed on the screen, press the down arrow key to view additional conditions. Using this screen, you can define a maximum of 8,000 load conditions for each channel



Load Cond	Channel 1 ( )	Channel 2 ( )	Channel 3 ( )	Channel 4 ( )	Channel 5 ( )	Channel 6 ( )	Channel 7 ( )	Channel 8 ( )
1	705.5	579.2	2324.1	56.0	3803.5	272.4	2898.0	6900.8
2	-705.5	-579.2	-2324.1	-56.0	-3803.5	-272.4	-2898.0	-6900.8
3	226.9	1390.2	2940.0	975.6	2669.5	638.4	6995.8	5409.6
4	-226.9	-1390.2	-2940.0	-975.6	-2669.5	-638.4	-6995.8	-5409.6
5	80.7	916.0	2717.1	1045.6	3926.0	2273.4	6435.8	5053.6
6	-80.7	-916.0	-2717.1	-1045.6	-3926.0	-2273.4	-6435.8	-5053.6
7	654.5	781.0	322.2	3136.0	2298.0	4522.2	131.6	2653.6
8	-654.5	-781.0	-322.2	-3136.0	-2298.0	-4522.2	-131.6	-2653.6
9	169.9	1855.2	293.7	1775.6	4362.5	1637.4	4715.2	2052.8
10	-169.9	-1855.2	-293.7	-1775.6	-4362.5	-1637.4	-4715.2	-2052.8
11	114.6	1846.8	1860.6	1390.8	4968.5	782.4	3833.9	7384.0
12	-114.6	-1846.8	-1860.6	-1390.8	-4968.5	-782.4	-3833.9	-7384.0
13	876.0	956.0	571.8	2736.4	808.5	4846.8	1418.2	7654.4
14	-876.0	-956.0	-571.8	-2736.4	-808.5	-4846.8	-1418.2	-7654.4
15	15.7	862.2	2956.2	2784.4	2088.0	4407.0	1940.4	7560.0
16	-15.7	-862.2	-2956.2	-2784.4	-2088.0	-4407.0	-1940.4	-7560.0
17	965.4	1115.0	2727.6	2629.2	322.5	4536.6	3479.7	1243.2

Figure 4-2 Load Conditions Screen

Step 4: Enter the load value for the condition in the column that corresponds to the selected channel. Depending on whether you specified E or P in the Channel Description screen, the load value may be entered in engineering units or as a percentage relative to the 100% Load specified in the Channel Description screen. The range of this entry is dependent on the ratio, R

R = [Full Scale/100% Load]

Specifically, this entry may be [-100% *R] to [+100% *R].

Step 5: Specify load conditions for other channels as desired. Each channel in a test must have the same number of load conditions. For example, if channel 11 has 14 load conditions, then all channels in the test must have 14 load conditions.

Step 6: When you finish, press the **ESCAPE** key (**F11**) to save your entries and return to the Display menu.

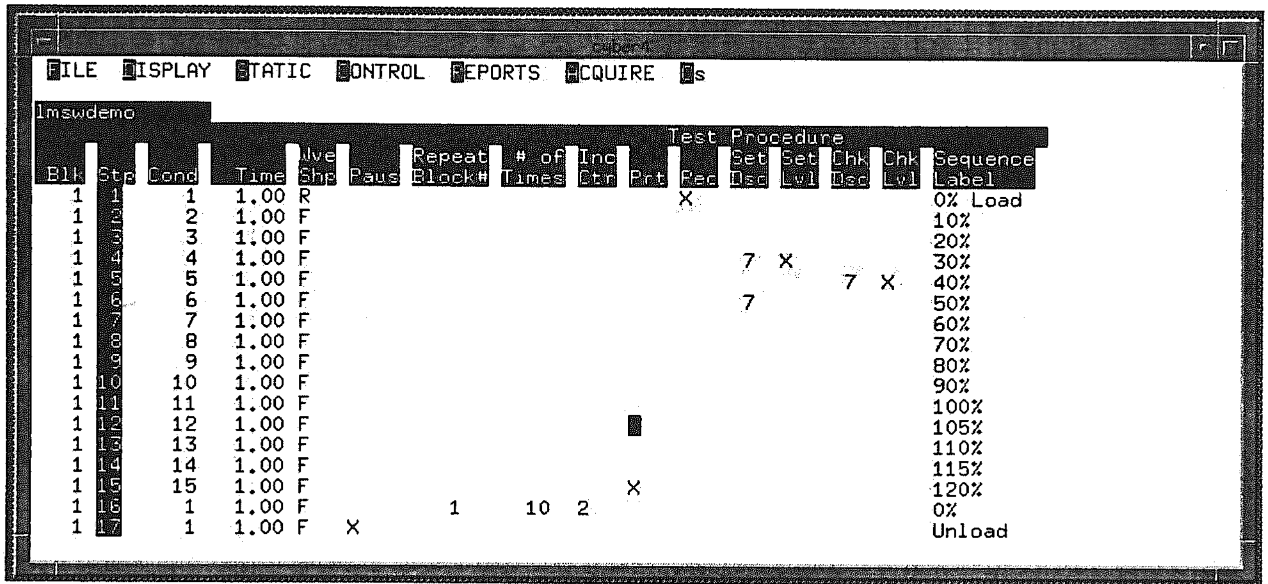
Defining the Test Procedure Steps

After defining the test channels and the load conditions, you are ready to define the test procedure. In this section, you:

- Define a load condition for each step in the test, and group the steps into multiple test blocks, if desired.
- Define the transition time for each step. All channels will undergo the transition from one load condition to the next in the same amount of time.
- Select the command waveform for each step in the test. The type of waveform determines how the command signal varies during the transition from one load condition to the next.
- Select specific actions for the system to take when it reaches each endpoint. You may make the system pause, repeat a block of test steps, increment an endpoint counter, print a test data report, record test data to disk, and/or set/reset discrete inputs & outputs.

Follow these steps to define each step of the test procedure:

Step 1: Select the Display menu, Test Procedure option. The system will display the Test Procedure screen shown in **Figure 4-2**.



Blk	Stp	Cond	Time	Wve Shp	Paus	Repeat Block#	# of Times	Inc Ctr	Prt	Rec	Set Dsc	Set Lvl	Chk Dsc	Chk Lvl	Sequence Label
1	1	1	1.00	R						X					0% Load
1	2	2	1.00	F											10%
1	3	3	1.00	F											20%
1	4	4	1.00	F							7	X			30%
1	5	5	1.00	F									7	X	40%
1	6	6	1.00	F							7				50%
1	7	7	1.00	F											60%
1	8	8	1.00	F											70%
1	9	9	1.00	F											80%
1	10	10	1.00	F											90%
1	11	11	1.00	F											100%
1	12	12	1.00	F											105%
1	13	13	1.00	F											110%
1	14	14	1.00	F											115%
1	15	15	1.00	F					X						120%
1	16	1	1.00	F		1	10	2							0%
1	17	1	1.00	F	X										Unload

Figure 4-2 Test Procedure Screen

**************************** NOTE ********************************

If you are planning on running a large spectrum fatigue test (i.e. more than 2000 test procedure steps), it is not recommended to look at and/ or change the test procedure file in the cyber program. Be aware that the test procedure file will be truncated to 2000 lines if you examine the Test Procedure in the cyber program and use the SAVE or SAVE AS options when exiting the test. To avoid the possibility of truncating the test procedure file, use the CLOSE option when exiting the test.

To make changes to large spectrum files without truncating them, requires that the changes be to the files at the UNIX operating system level. This requires a understanding of UNIX. If you do not possess this understanding, please call Cyber Systems for further instruction.

For static tests, you can display and make changes to the test procedure, and save them without truncating the test procedure.

****

Step 2: Enter the block number in the Block column. The test step number is automatically incremented within each block as you make entries. Test steps are individual transitions from one endpoint to the next. Sequences of test steps may be grouped into blocks. Up to 99 steps are permitted in each block.

Step 3: Enter the number of the load condition to be applied in this step in the Cond column. You must make sure that you only specify load condition numbers which have been previously defined in the Load Conditions screen.

Step 4: Enter the transition time, in seconds, in the Time column. This is the time the system will take to execute the transition from the previous load condition to the load condition specified in this step. Allowable Time entries are from 0.25 seconds to 999 seconds.

When you enter the time, the system automatically inserts a line for a new step, writes the new step number in the Step column, and moves the cursor to the Cond column for the new step. You are ready to enter the next step's load condition# and time. Continue to define the test steps in this way, entering the load condition and time only, until you want to define a new block or you want to override the default values for a step.

- To define a new block, simply move the cursor back to the Block column, using the arrow keys, and return to step 2.
- To override the default values for a particular test step, use the arrow keys to move the cursor to the desired column and type a new value. Steps 5 through 13 describe the remaining columns on the screen.

****************************** NOTE ****************************

If modifying an existing block number, you must "TAB through" the step column for each step within the newly defined block.

****

Step 5: Enter the code for the command waveform in the Shape column. You may enter "H" for haversine, "R" for ramp, or "F" for modified ramp. You may use upper-case or lower-case letters. **Figure 4-3** shows available waveforms.

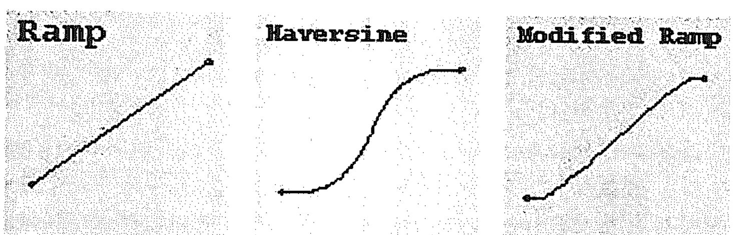


Figure 4-3. Command Waveforms

The following entries are optional and depend on individual spectrum requirements:

Step 6: If you want the test to pause at the completion of this test step, press the spacebar to enter 'X' in the Paus column. The test will stop when it completes this step, and must be restarted by pressing the RUN button on the test control panel. The "Prog Pause" LED will be illuminated on the test control panel.

Step 7: If you choose, you can repeat a specific block or group of blocks a number of times before continuing on to the next test step. This is accomplished by using the Rpt to Blk # column. This column works exactly like a "GOTO" statement in a basic program. Thus you can jump back to the first step of any previously defined block. If you wish use this function, enter the block number you wish to " go to".
The Rpt to Blk # column works in conjunction with the adjacent column labeled # of Times. This column is used to specify the number of times the system is to "go to" the first step of the specified block. When the specified number of repetitions have been completed, the test proceeds to execute the test steps which follow the step which included the repeat statement.

Step 8: If you want the test step to increment an endpoint counter, enter '1', '2', or '3' in the Incr Cntr column. Remember that every test step increments counter 4, and that incrementing any counter automatically resets higher-numbered counters to zero. Step 9:If you want a tabular printout of test data for each channel at the conclusion of this test step, press the space bar to enter 'X' in the Prt column. This entry works in conjunction with the Print Enable checkbox located on the Master screen. When the endpoint is reached, the system automatically prints a Test Data report.

Step 10: If you have defined a fatigue test and wish to record the endpoint data for the control channels (and DAS data, if configured) to the host computer's hard disk, enter an 'X' in the Rec column. This is a toggle-type function: the first step with an 'X' enables the record function; the next test step which has an 'X' disables the record function. For example, if steps #5 and #10 of a thirty step test procedure both had an "X" in the Rec column, the system will record endpoint data for steps 5, 6, 7, 8, and 9. The endpoint associated with Step 10 is not recorded. By using multiple "X"s, you can selectively record groups of steps throughout the entire test procedure. Using our example above, "X"s could also be entered on steps 20 and 25, or any other steps. In all cases, when the record function is enabled, test data for all control channels (and all DAS channels, if configured) is automatically recorded.

***************************** NOTE *****************************

This operation described above only applies to fatigue tests. For static tests, endpoint data is always recorded regardless of entries in the Rec column.

****

Step 11: To change the state of an output discrete at this test step, enter the discrete number in the Set Dsc column, and then set the desired state of the discrete in the Set Lvl column. Each output discrete is a Form-C relay. To set the discrete to the normally open state, press the space bar to enter 'X' in the Set Lvl column. To set the discrete to the normally closed state, leave the Set Lvl column blank.

Output discrete assignments are listed in Appendix F.

Step 12: To check the state of an input discrete at this step, enter the discrete number in the Chk Dsc column. To use the results of this check as the basis of the Discrete Synch conditional action, enter the desired state in the Chk Lvl column. The test will continue/hold based on the result of the check of this input discrete. If a true state is required for the test to continue (i.e not hold), press the space bar to enter 'X' in the Chk Lvl column. If a false state is required, leave the Chk Lvl column blank.

Input discrete assignments are listed in Appendix F.

See Chapter 5, Defining Safety & Operational Functions for further description on Discrete Synch.

Step 14: If you wish to have a label for the test step, enter a label in the Sequence Label column.

Step 15: As you define new test steps, the screen scrolls down to provide space for them. When you finish defining all of the test steps, press the **ESCAPE key (F11)** to return to the Display menu.

MODIFYING THE TEST PROCEDURE FILE

Most items in the test procedure are modified by simply tabbing to the field and re-entering the data (with the exception of block, as described earlier).

In the event that new steps need to be added or existing steps need to be deleted, refer to the following procedures:

Deleting a Line in the Test Procedure

Step 1: Using the up/down arrow keys, move the cell cursor to the line you wish to delete. The column in which the cell cursor is active has no effect on the delete function.

Step 2: Press "F6" function key to delete this line. The step numbering will be updated automatically.

Step 3: Exit Test Procedure screen and save the test.

Inserting a Line in the Test Procedure

Step 1: Using the up/down arrow keys, move the cell cursor to the existing line which you want to insert a new line after. The column in which the cell cursor is active has no effect on the insert function.

Step 2: Press "F7" function key to insert a new line after the line where the cursor is. The step numbers of all following steps will be automatically incremented (within their respective blocks) and an empty line will be provided for manual entry of the desired new step.

Step 3: Enter the new line by first entering the block number in the leftmost column. "Tab" to the right and observe that the step # is automatically set to the correct step number (i.e. one greater than the previous step number and one less than the following step #). Enter the desired load condition #, time, shape, etc..

Step 4: Exit Test Procedure screen and save the test.

*********************************** NOTE ************************************
The F6 and F7 keys only work for changes to the Test Procedure screen.
****

USING THE STATIC MODE SOFTWARE TO CREATE/MODIFY THE LOAD SPECTRUM

The FM7000 includes software specifically tailored for setting up and running static tests. Through several simple user entries, including the target test load, the loading rate, and a loading increment, the system automatically generates the desired load spectrum, specifically the load conditions file and test procedure file.

*********************************** NOTE ************************************
The static mode is simply an alternate method to defining the load profile, the test procedure and load conditions may be defined and/or modified manually, if preferred.
****

Since the Static Mode is only an extension of the standard FM7000 application software, the same setup and start up operations normally used with the system must be performed prior to using the Static Mode. These mandatory operations include:

- defining channel parameters (full scale, 100% load, limits, etc.)
- defining conditional action tables, if applicable
- performing calibration routines, if applicable
- verifying setup by selecting "Verify"
- performing the normal start up procedure using the "Start Up" screen

BUILDING A NEW TEST (Load Conditions & Test Procedure)

To build a new test using the Static Mode feature, follow these steps:

Step 1: Make a new test (under File) and select "Static" as the test type. You will now have access to the applicable screens for static setup.

Step 2: Select the Static menu, Static Mode option. The Static Mode screen, **Figure 4-4**, is displayed. Building a new test may be done either before or after the test is brought on-line (i.e. downloaded). However, a test profile must exist in order to pass Verify and initiate a download. Typically, the [Build New] option is used to automatically create the initial profile before downloading.

Step 3: Enter the desired load in the column labeled Target Test Load %. This entry defines the target load relative to the 100% Load entry made on the Channel Description screen. The range of this entry is dependent on the ratio, R, which equals "Full Scale/100% Load"

Specifically, this entry may be [-100% *R] to [+100% *R].

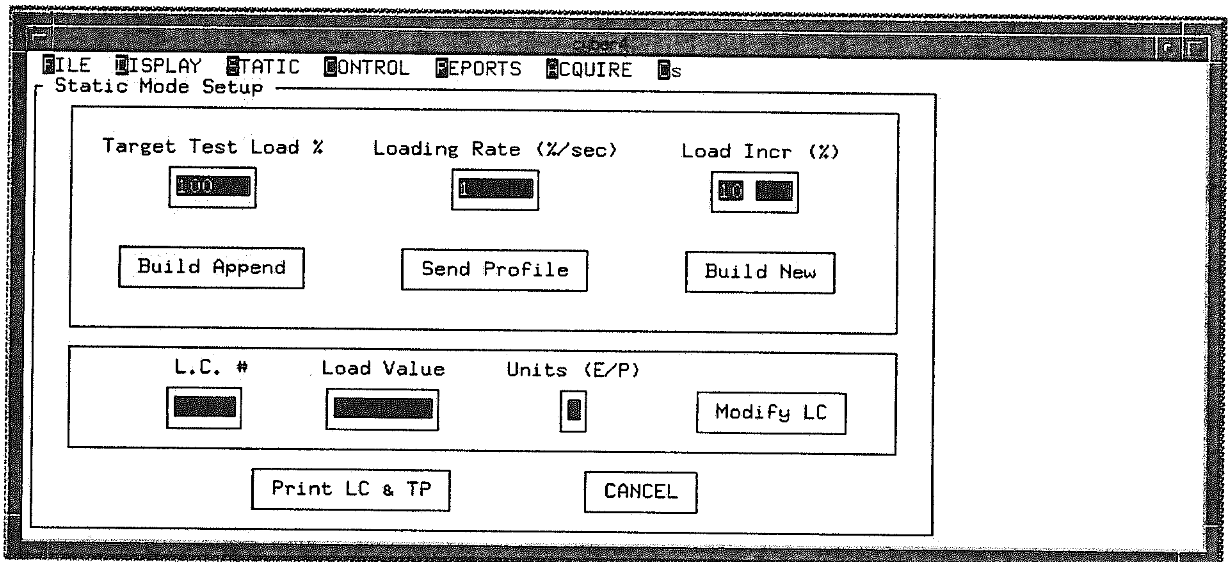


Figure 4-4 Static Mode screen

Step 4: Enter the desired loading rate in the column labeled Loading Rate (%/sec). This entry defines the rate at which the system will transition from the present load condition to the Target Test Load. The range is 0.5%/second to 200%/second.

Step 5: Enter the desired load increment in the column labeled Load Incr (%). This entry defines the size of the incremental loads used in the generation of the load profile. These incremental loads, which are actually individual load conditions, are primarily used for data recording and increased resolution for graphic displays (i.e. more data points). This entry can be between 1% and 200%, but must be smaller than the "Target Test Load %" entry.

********************************** NOTE *************************************

The actual transition time is determined from the quotient of Load Incr/Loading Rate. If this number is less than 0.25 seconds, the system will explicitly set the time transition value to 0.25 seconds.

****

Step 6: Select the Build New option and press **RETURN**. The message screen appears, indicating that the LC and TP files will be overwritten. Answer **YES** if you wish to continue.

The system clears the present temporary load conditions and test procedure files and creates new files according to the new target entries on the Static Mode Set UP screen. The load conditions file which is always created with L.C.#1 equal to "zero load", followed by the incremental load conditions per the target entries. The test procedure always uses L.C.#1 (zero load) in the first step, followed by the subsequent load conditions. A Programmed Pause is automatically inserted in the last step of the "just created" test procedure.

The new test profile (LC and TP) resides in the temp. directory for this test; but is not yet active. You must perform a save function, if you wish to make these temporary files be permanent files.

To make the newly created profile active, you must do one of two things:

1)download the test, if not already downloaded

or

2)proceed to step 7.

Step 7: Select the Send Profile option and press RETURN.

*************************** [redacted] ***********************************

The system must be in a Ramped, Paused (button or programmed), or Hold condition in order to initiate a [Send Profile]. If the system is Holding between two endpoints, performing a Build New will cause the system to not complete its current endpoint, but rather proceed to the first endpoint of new load spectrum from its present load.

****

This option causes the load conditions and test procedure to be verified and downloaded to the servos and TMTM, respectively. Messages indicating "Verifying Test" and "Transferring to TMTM" are displayed during this operation. Once the downloading of the LC and TP is completed, the test control panel's master displays will be blinking, indicating that you can now RUN, RAMP or PAUSE the test as desired. The test control panel displays blink and counters reset only for the on-line [Build New], [Send Profile] sequence. All other uses of [Send Profile], described later, do not cause the displays to blink and keep the counter values/test in its present location.

The following example illustrates how simple it is to automatically create a new test using **the [Build New]** function and show explicitly the load condition values and test procedure steps which get created.

Two control channels are defined as follows:

	Full Scale	100% Load
Ch 1	20,000	10,000
Ch 2	10,000	5,000

By using the following target entries:

Target Load %	Loading Rate (%/sec)	Loading Inc (%)
40	10	10

and selecting **[Build New]**, the system automatically creates the load conditions and test procedure steps shown below:

Test Procedure

Blk	Step	LC#	Time	Shape	Paus
1	1	1	1.00	F	
1	2	2	1.00	F	
1	3	3	1.00	F	
1	4	4	1.00	F	
1	5	5	1.00	F	X

Load Conditions

LC#	Ch 1(E)	Ch 2(E)
1	0	0
2	1000	500
3	2000	1000
4	3000	1500
5	4000	2000

The system created a "0" load condition plus 4 incremental load conditions between 10% and 40%. Each of these load conditions is automatically used in the test procedure along with the time and default waveshape (F). By having a "0" load condition, which is also always automatically used as the first step in the test procedure, when you press **RUN**, the first step will be "0%". Notice the program pause in the last step of the test procedure. You may go into either file (screen) directly and change load values, change time values, change waveshapes, add/delete load conditions and/or test procedure steps, insert actions in each test procedure step (record, print, increment counters, etc.), remove the pause statement from the last line, and make any other desired modifications. Simply performing another **[Send Profile]** will invoke these modifications.

USING THE STATIC MODE SCREEN TO APPEND THE LOAD SPECTRUM

Once a load spectrum is created, it can be modified either by manually editing the Test Procedure and/or Load Conditions screens, or by using the Static Mode screen to automatically append steps to the existing spectrum. The following section describes how to use the Build Append function.

When the [**Build Append**] option is selected from the Static Mode screen, the load conditions and test procedure steps created from new target entries are appended to their respective existing temporary files. The "Verify" program is automatically initiated to validate the spectrum information and a Programmed Pause is automatically inserted in the last step of the appended test procedure. Unlike Build New which resets the counters, [**Build Append**] does not reset counters.

Once a spectrum has been modified or appended, you simply perform another [**Send Profile**] and the modified load profile is verified and downloaded to the **TMTM** & servo controllers where it is ready for execution. Each time [Send Profile] is performed, it is logged in the Events Log.

The following examples illustrate the various uses of [**Build Append**] and show explicitly the Load Condition values and Test Procedure steps which get created.

EXAMPLE

Using the channel descriptions and existing profile created in Example#1 (page 16), if we now wish to go to 50% in 1% increments at a loading rate of 1% per second, do the following steps:

Step 1: Enter the following target entries:

Target Load %	Loading Rate (%/sec)	Loading Inc (%)
50	1	1

Step 2: Select **[Build Append]** and hit **RETURN**. The following Test Procedure and Load Conditions files are created:

Test Procedure							Load Conditions		
Blk	Step	LC#	Time	Shape	Pause		LC#	Ch 1(E)	Ch 2(E)
1	1	1	1.00	F			1	0	0
1	2	2	1.00	F			2	1000	500
1	3	3	1.00	F			3	2000	1000
1	4	4	1.00	F			4	3000	1500
1	5	5	1.00	F	X		5	4000	2000
1	6	6	1.00	F			6	4100	2050
1	7	7	1.00	F			7	4200	2100
1	8	8	1.00	F			8	4300	2150
1	9	9	1.00	F			9	4400	2200
1	10	10	1.00	F			10	4500	2250
1	11	11	1.00	F			11	4600	2300
1	12	12	1.00	F			12	4700	2350
1	13	113	1.00	F			13	4800	2400
1	14	14	1.00	F			14	4900	2450
1	15	15	1.00	F	X		15	5000	2500

Notice that **[Build Append]** used **LC#5** as its starting value to determine **LC#6** to **LC#15**. In general, it takes the target load (e.g. 5000 for Ch #1), subtracts the value of the last load condition in the file (for each channel, e.g. 4,000 for Ch #1), and inserts the necessary number of incremental values to give the desired load increments. The loading rate is turned into incremental step times in the test procedure. The time for each step (within a given build) is determined by:

Load Increment (%)/Loading Rate (%/sec)

Thus, producing 1.00 second step times in this example.

Provided the system is in a ramp, hold or pause state, a **[Send Profile]** will be allowed. **[Send Profile]** is the "on-line" way of making this profile executable and must be performed to do so. If the system was paused at the end of step #5, and the above actions were performed (i.e. Build Append with Static screen entries of 50, 1, 1, followed by a Send Profile), upon pressing the **RUN** button, the system would proceed from step #5 to step #6 (i.e. from 4000 to 4100 for Ch 1, and 2000 to 2050 for Ch 2). If no manual pauses are activated, the system will generate appropriate loads up to the load values of step 15, where it will pause. The operator also has the option to single step up to step 15 in 1% increments by running the system in the Pause mode, if desired. At the completion of step 15, the operator has the following options:

- Branch to an existing step in the test procedure
- Ramp to step by pressing the appropriate ramp button
- Append more steps to the test procedure
- Send Profile
- run the spectrum in reverse
- build an entirely new profile and start all over again
- or abort the test

Example [illegible]

Manually modifying an existing or newly created spectrum

Let's use example #1 and start with the case where the system runs to step #5 and pauses. We decide to go to 50% in 1% increments and do [Build Append] as in Example #2. We now have the same test procedure and load conditions files as in Example #2. Suppose we then decide the following:

- Ch 2 should not go above 45% (2250 lbs) while Ch 1 is allowed to go up to 49%
- The loading rate on both channels between 40% and 45% should be 10 times slower
- When Ch 1 reaches 4900 lbs both channels should ramp to zero load in 60 seconds

We would then perform the following manual modifications to the load profile:

Test Procedure							Load Conditions		
Blk	**Step**	**LC#**	**Time**	**Shape**	**Pause**		**LC#**	**Ch 1(E)**	**Ch 2(E)**
1	1	1	1.00	F			1	0	0
1	2	2	1.00	F			2	1000	500
1	3	3	1.00	F			3	2000	1000
1	4	4	1.00	F			4	3000	1500
1	5	5	1.00	F	X		5	4000	2000
1	6	6	**10.00**	F			6	4100	2050
1	7	7	**10.00**	F			7	4200	2100
1	8	8	**10.00**	F			8	4300	2150
1	9	9	**10.00**	F			9	4400	2200
1	10	10	1.00	F			10	4500	2250
1	11	11	1.00	F			11	4600	2300
1	12	12	1.00	F			12	4700	2350
1	13	13	1.00	F			13	4800	2400
1	14	14	1.00	F			14	4900	2450
1	15	**1**	**60.00**	**R**	X		15	5000	2500

Observations:

a) By going into the Load Conditions screen and making LC #11 through LC #14 for Ch 2 be 2250 lbs, Ch 2 will not go above 45% while Ch 1 will be allowed up to 49%.

b) By going into the Test Procedure screen and making the time for steps #6 through #10 (40% to 45%) be 10.00 seconds, the rate was reduced by a factor of 10.

c) By going into the Test Procedure screen and changing the LC # for step#15 from "15" to "1", the test will transition from LC #14 (49%, 45%) to LC#1 (0,0) in 60 seconds. Also, the waveshape will be a ramp.

Again, this modified spectrum will become executable only when **[Send Profile]** is performed.

Manual modification of the test procedure also applies for all columns to the right of the programmed "Paus" column. These include:

repeat to blk#
# of times
counter incrementing
programmed print
set discrete
check discrete

Additional steps may be manually added (i.e. typed in) to the test procedure and additional load conditions may be added to the Load Conditions screen, if desired.

USING THE STATIC MODE SCREEN TO MODIFY/CREATE LOAD CONDITIONS GLOBALLY

As mentioned earlier, the system allows the user to modify individual load conditions for all channels with a single entry. This feature is most useful for static tests with many channels, where the Load Condition must set the same value for each channel in the test. This is accomplished with the lower entries on the Static screen (**Figure 4-6**), each of which is described below:

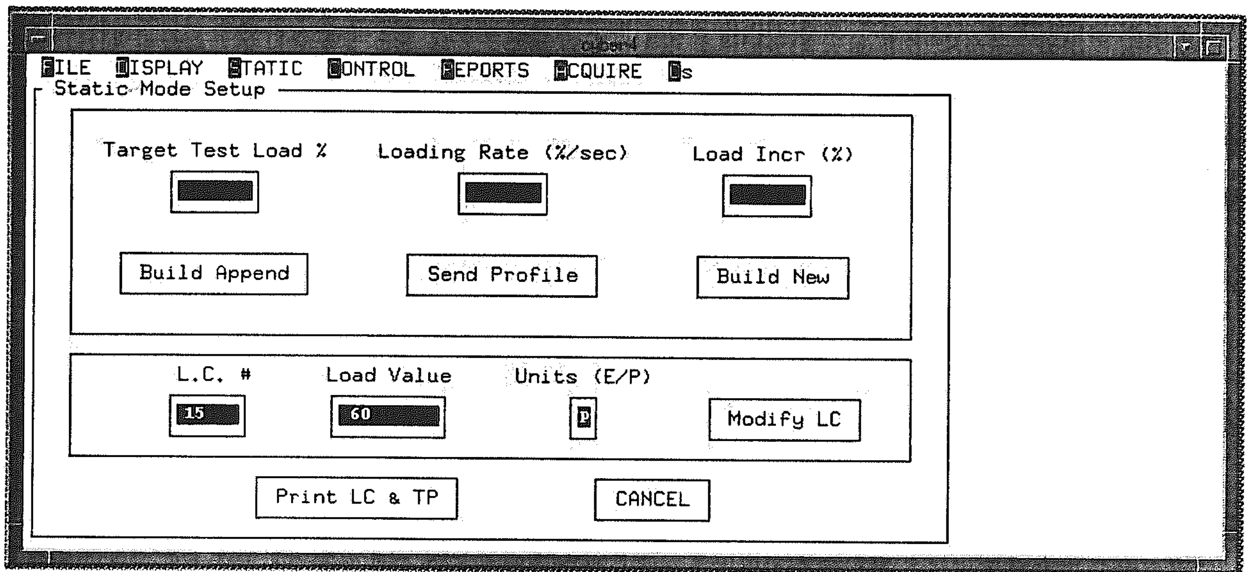


Figure 4-6 Using the Static Mode screen to Create/Modify a Load Condition

L.C. # This entry identifies the Load Condition which is to be modified or created.

Load Value This entry defines the new load value for all channels in the selected Load Condition.

Units (E/P) This entry allows the user to select whether the "Load Value" entry described above represents Engineering Units or a Percent value relative to the 100% Load entry.

Once the above entries have been made, selecting [**Modify LC**] initiates the modification and verification of the selected Load Condition.

In the course of verification, the system checks for overscale loads and non-existent LC#s. If the specified Load Value results in overscale values for one or more channels defined in the test, the system automatically sets such channels to their individual full scale value.

If the selected LC# is non-existent, the system will automatically create a new load condition with the specified load value and append it to the end of the existing load condition file. The next contiguous LC# is used for this new load condition, e.g. if a total of 12 load conditions exist in the load condition file and you enter 27 in the LC# box, the system will create a new load condition labeled 13, which is appended to the load conditions file. You may use this modified or newly created load condition within the test procedure, if desired. Once any modifications are made, you simply perform another [Send Profile] and the modified load profile is verified and transferred to the TMTM and servos, where it is ready for execution.

You may obtain a hardcopy of the test procedure and load conditions using the [Print TP & LC] software selection available on the Static Mode screen.

Using the Load Condition file in example #2, suppose we wished to make the final load (LC #15) be 60% instead of 50%. We would make the following global entries in the Static screen:

LC #	**Load Value**	**Units (E/P)**
15	60%	P

We would then move the cursor to **[Modify LC]** and press **RETURN**. The following changes to the Load Conditions file would be made:

Load Conditions		
LC#	**Ch 1(E)**	**Ch 2(E)**
1	0	0
2	1000	500
3	2000	1000
4	3000	1500
5	4000	2000
6	4100	2050
7	4200	2100
8	4300	2150
9	4400	2200
10	4500	2250
11	4600	2300
12	4700	2350
13	4800	2400
14	4900	2450
15	6000	3000

Also, if we wished to make LC #15 be 5,000 lbs for both channels, we would make entries of . . .

LC #	Load Value	Units (E/P)
15	5000	E

position the cursor on [Modify LC] and press **RETURN** to produce the following:

Load Conditions		
LC#	**Ch 1(E)**	**Ch 2(E)**
1	0	0
2	1000	500
3	2000	1000
4	3000	1500
5	4000	2000
6	4100	2050
7	4200	2100
8	4300	2150
9	4400	2200
10	4500	2250
11	4600	2300
12	4700	2350
13	4800	2400
14	4900	2450
15	**5000**	**5000**

USING THE BRANCHING FUNCTION WHILE RUNNING A STATIC TEST

When running a static test, the FM7000 allows you to branch (i.e. "GOTO") to a particular step within the present profile held by the TMTM. The steps given below, describe how to use this feature.

Step 1: Select the STATIC menu, BRANCHING option. The BRANCHING screen shown in **Figure 4-7**, is displayed.

Step 2: Enter the Block # and Step # you wish to have the system branch to (go to).

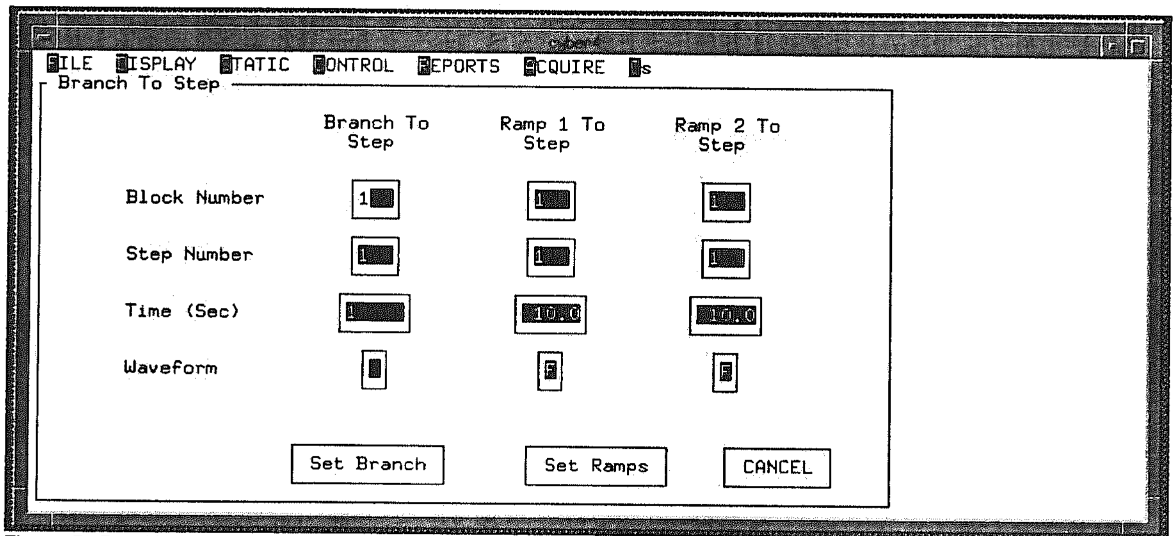


Figure 4-7 Branching Screen

Step 3: Enter the time and waveshape of the transition from where you are now to the specified Block/Step. The default waveshape is a Ferentz wave. However, you may specify a ramp (R) or Haversine (H).

****************************** NOTE ****************************

Branching may only be done when the system is in a Paused, Hold, or Ramped condition.

****

Step 4: Select "**Set Branch**" from the screen. The system automatically changes the pointer location to the specified block/step.

Step 5: Press the **RUN** button on the test control panel. The system will transition from its present location in the profile, to the location (i.e. block/step) specified in the branching screen and resume running according to the profile from there.

Using the Branching function to modify the progress of a predefined test spectrum

In order to understand how the branching function works, we've created a simple hypothetical test situation which will illustrate how useful this feature can be. Let's say we have a one channel test which we want to test to 100% in 10% increments. However, we're concerned about the test article once the load reaches 50%. There is a chance that our stress analysts may want us to continue from 50% to 100% in 5% increments, instead of 10%. In fact, they might even want us to unload the test article in 5% decrements, based on the stress data. **Figure 4-9** illustrates the three possible paths or "branches" we may have to execute. Most importantly, however, is that we must be able to change the load increment and direction "in mid-stream", without having to define new steps or load conditions. This is very simple to do with the Branching function.

Branch A
Once at 50% load, the operator can continue to 100% load with 10% increments, simply by executing steps 7 through 11

Branch B
Once at 50% load, the operator can continue to 100% load with 5% increments, simply by executing steps 12 through 21

Branch C
Once at 50% load, the operator can unload the test article in 5% decrements, simply by executing steps 22 through 31

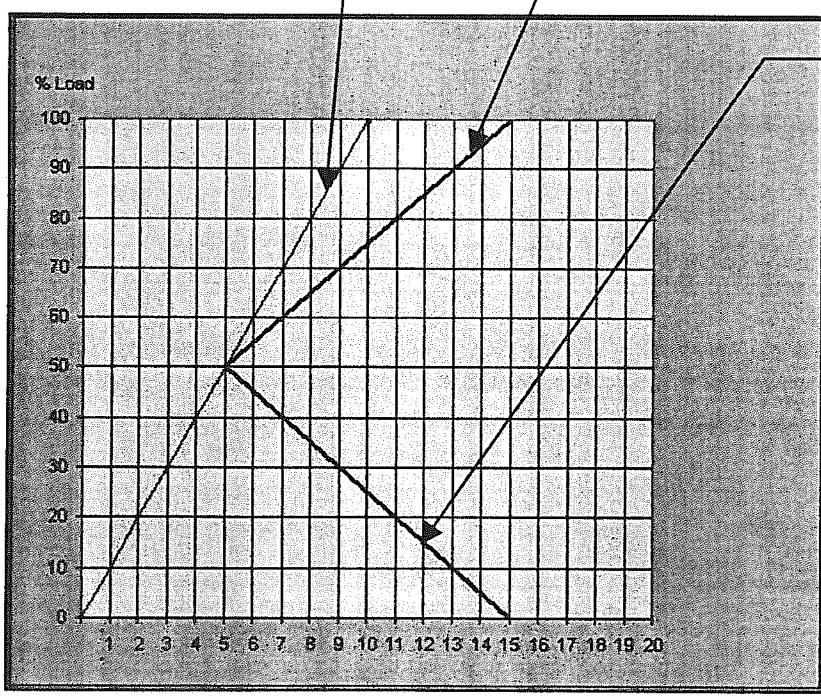


Figure 4-9 Branching Example

This time we'll use the test procedure screen and load conditions screen to manually define the spectrum information. However, we could have also used the static mode screen to create the same profile.

As you can see, we've defined several load conditions with the same value (e.g. load conditions #11 and #21 both equal 100). In a real test, this is not necessary. However, in order to make the example easy to understand, we've used an equal number of load conditions and test procedure steps.

Test Procedure							Load Conditions	
Blk	Step	LC#	Time	Shape	Pause		LC#	Ch 1(P)
1	1	1	1.00	F			1	0
1	2	2	1.00	F			2	10
1	3	3	1.00	F			3	20
1	4	4	1.00	F			4	30
1	5	5	1.00	F			5	40
1	6	6	1.00	F	X		6	50
1	7	7	1.00	F			7	60
1	8	8	1.00	F			8	70
1	9	9	1.00	F			9	80
1	10	10	1.00	F			10	90
1	11	11	1.00	F	X		11	100
1	12	12	1.00	F			12	55
1	13	13	1.00	F			13	60
1	14	14	1.00	F			14	65
1	15	15	1.00	F			15	70
1	16	16	1.00	F			16	85
1	17	17	1.00	F			17	80
1	18	18	1.00	F			18	85
1	19	19	1.00	F			19	90
1	20	20	1.00	F			20	95
1	21	21	1.00	F	X		21	100
1	22	22	1.00	F			22	45
1	23	23	1.00	F			23	40
1	24	24	1.00	F			24	35
1	25	25	1.00	F			25	30
1	26	26	1.00	F			26	25
1	27	27	1.00	F			27	20
1	28	28	1.00	F			28	15
1	29	29	1.00	F			29	10
1	30	30	1.00	F			30	5
1	31	31	1.00	F	X		31	0

As you can see, steps #1 through #6 will load the test article to 50% and pause. At that point, and depending on want our stress analysts say, we can let the system continue executing steps # 7 through #11. If necessary, we can also branch to step #12 or #21, depending on what the stress analysts decide.

The instructions below explain how you would branch from step#6 (or any other step) in our example test to any other step in the test procedure.

************************************** NOTE **********************************
In order to Branch, you must first make sure the system is in a Paused, Hold, or Ramped condition. In our example, at step # 6 the system was paused automatically by the programmed pause in the test procedure file.
****

Step 1: Select the STATIC menu, BRANCHING option.

Step 2: Enter the Block # and Step # you wish to have the system branch to (go to). For this example, we'll branch to step #12.

Step 3: Enter the time and waveshape of the transition from where you are now to the specified Block/Step. The default waveshape is a Ferentz wave. However, you may specify a ramp (R) or Haversine (H). For this example, let's maintain the 1.00 second transition time and Ramp waveshape.

Step 4: Select [Set Branch] from the Branching screen and press RETURN. The system automatically changes the pointer location from Block#1/Step#7 to Block#1/Step #12. Since we completed step#6 and were at 50%, the next step would have been #7.

Step 5: Press the RUN button on the test control panel. The system will transition from 50% to 55% in 1 second and resume running according to the profile from there, pausing at step#21 (100%).

Step 6: At this point, our stress analysts tell us to go back to 0% and start over. This is very easy with the branching function. Simply specify Block#1/Step#1 and an appropriate transition time/shape, then select "Set Branch". When you press the RUN button, the system will go to step#1 in the procedure (0%) and resume running from there.

DEFINING FATIGUE TESTS USING CYBER'S BLOCK LOADING METHOD OF SPECTRUM GENERATION

The system also allows for predefining a multi-tier spectrum (sometimes called Block Loading), consisting of load conditions, flights, and blocks. In order for you to use this feature, the test must be defined as a "fatigue" test (refer to Chapter 2). Below is a step by step description for defining a multi-tier spectrum.

Step 1: Define the load conditions. This is done using the Load Conditions screen.

Step 2: Group one or more load conditions to create individual flights. This is done by selecting "Flight Definition" from the DISPLAY pull down menu.

The Flight Definition screen shown in **Figure 4-11** appears. As you will notice, this screen looks very similar to the test procedure screen. The main difference is that Flt # has been substituted in place of the Block # column.

As the example screen shows, you can define multiple Flights, each of which can consist of one or more steps/load conditions. Each step can include any of the selective features such as incrementing counters, printing, setting/reading discretes, etc..

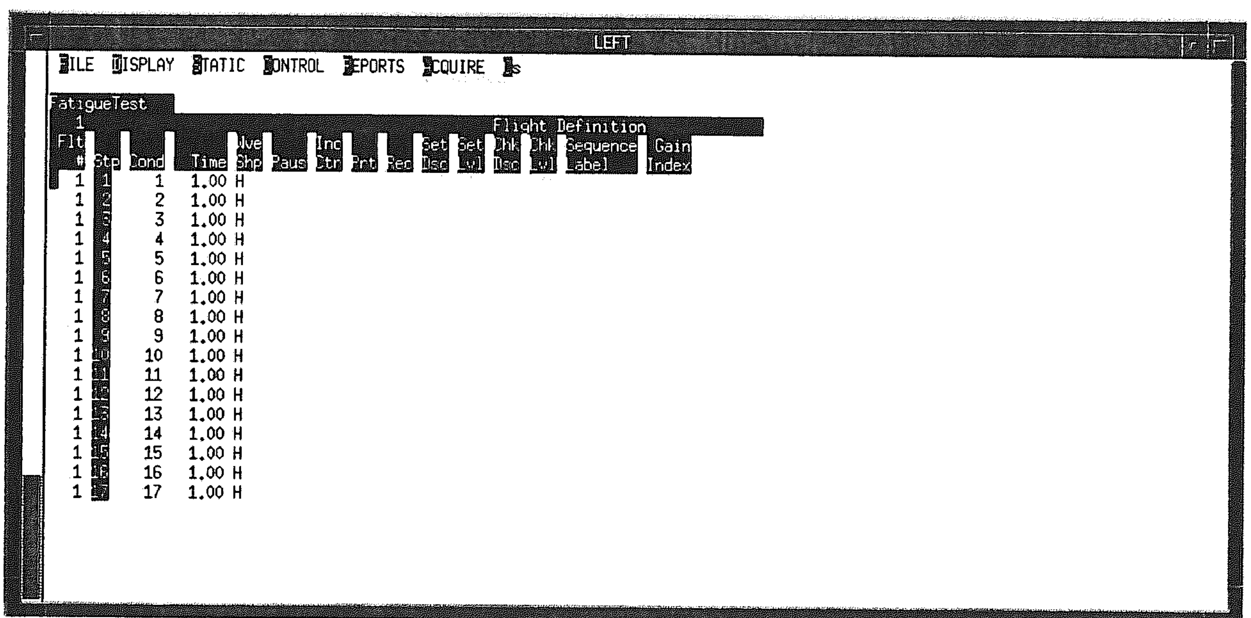


Figure 4-11 - Flight Definition Screen

Step 3: Select Block Setup from the DISPLAY pull down menu. A dialog box shown in Figure 4-12 appears. This dialog box gives you two options. Select "Block Definition" and press **RETURN**.

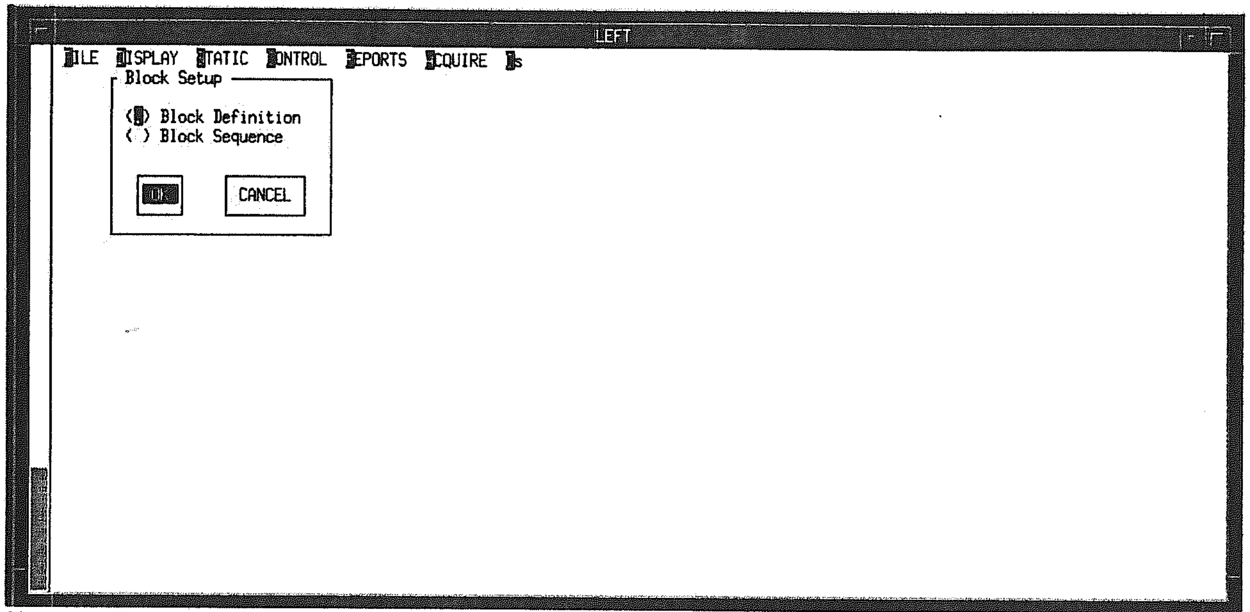


Figure 4-12 - Block Definition/Selection Dialog Box

The screen in **Figure 4-13** appears. This screen allows you to use the previously defined Flight Cycles to create "Blocks". As the example screen shows, each block can be defined with one or more Flight Cycles. Additionally, each Flight Cycle entry can be repeated a user-specified number of times.

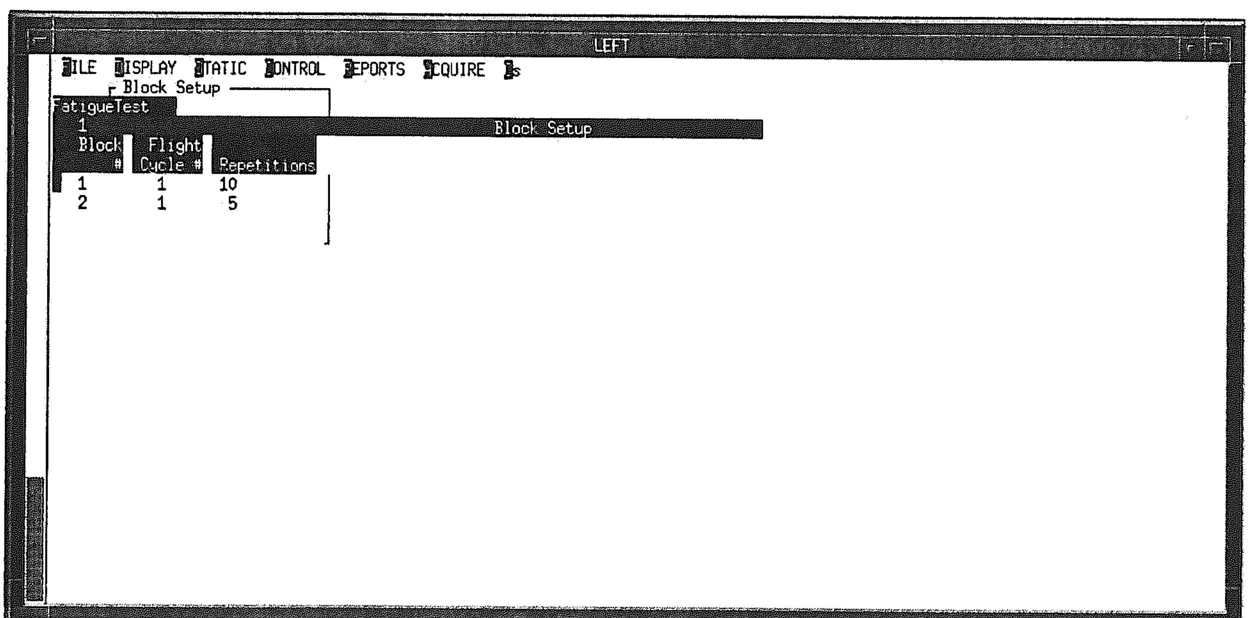


Figure 4-13 - Block Setup Screen

Step 4: When you have finished defining all of the individual Blocks, select Block Setup from the DISPLAY pull down menu. The dialog box shown earlier in Figure 4-12 appears again. This time select "Block Sequence" and press **RETURN**.

The screen in **Figure 4-14** appears. This screen allows you to specify the sequence in which the previously defined Blocks are to be executed. As the example screen shows, with each line on this screen you enter the desired Block # along with the number of times it is to be sequentially repeated.

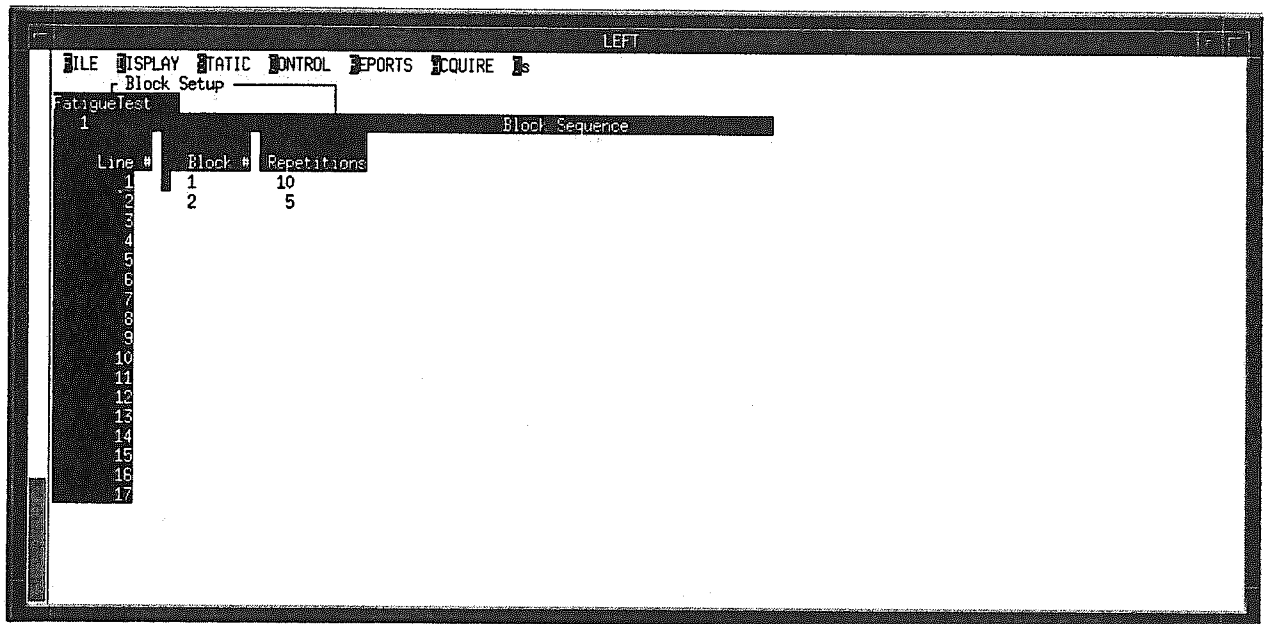


Figure 4-14 - Block Sequence Screen

Step 5: When you are finished defining the Block Sequence and attempt to leave this screen by pressing **F11**, a dialog box will appear and ask whether or not you want to re-generate the spectrum.

If you answer **YES** the system will re-generate the spectrum according to your entries. This re-generation will ultimately produce a test procedure file in the format you're familiar with.

You may look at the newly created test procedure file by selecting **"Test Procedure"** from the **DISPLAY** menu, or obtain hardcopies of the test procedure (all or sections) under the **REPORTS** menu. An example of the reports follows at the end of this chapter.

If you answer **NO**, any previously defined test procedure will remain unchanged.

??
REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

CHAPTER 5

DEFINING SAFETY & OPERATIONAL FUNCTIONS

This chapter describes how to make use of the individual safety and operational features offered with the Fatigue Master 7000, including:

how to define the actions which are to be taken when specific conditions arise in the system, such as out-of-limit servo conditions, discrete input states, pacing function timeouts, etc.

- how to define time-sensitive parameters, such as the dump time, ramp time, pacing times, etc.
- for fatigue tests, how to define the load condition the system ramps to when either of the Ramp buttons on the test control panel are pressed, or when the Dump or Ramps in the System Conditional Action Table are invoked.
- for static tests, how to define the step the system ramps to when either of the Ramp buttons on the test control panel are pressed, or when either of the Ramps in the System Conditional Action Table are invoked.

??
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

DEFINING CONDITIONAL ACTIONS

There are three types of conditional actions; channel actions, system actions, and discrete actions. They are similar in concept, but the three types differ in what conditions are tested and what range of reactions are possible. When faults or out-of-limit conditions occur, the system looks to the conditional action tables and executes the action that has been linked to the particular event or out-of-limit condition. This section explains how you define each type of conditional action.

Defining Channel Conditional Actions

Channel conditional actions define responses to faults detected by the servo controller on individual control channels. Fault responses include setting Fault 1 or Fault 2 conditions, which can trigger system conditional actions.

All channel errors are subject to a debounce time (also referred to as a persistence time) before the servo controller notifies the system of any fault conditions. Conditional actions may be defined for any of the following channel conditions:

- Inner error faults; when the error signal on a channel exceeds the inner error level defined for that channel.
- Outer error faults; when the error signal on a channel exceeds the outer error level defined for that channel. This is a latching condition.
- Inner limit faults; when the feedback signal on any channel exceeds the inner limit level defined for that channel.
- Outer limit faults; when the feedback signal on any channel exceeds the outer limit level defined for that channel. This is a latching condition.
- Discrete input faults; when the servo controller tests the state of an input discrete and finds that the discrete input state is different from the expected state. There are two fault conditions corresponding to discrete inputs 1 and 2 on the servo controller back panel.
- Excitation Change faults; when the servo controller detects a change in the excitation voltage of 100mV or greater. If the change is less than 100mV, this condition is not activated. The change is detected via the bridge input connector on the servo controller back panel.

Follow these steps to define channel conditional actions:

Step 1: Select the Display menu, Channel Conditional Actions option. The Channel Conditional Action Table screen, **Figure 5-1**, is displayed.

Step 2: Use the TAB key and space bar to link each channel condition to a Fault 1 or a Fault 2.

********************************** NOTE ******************************

1) Fault 1 is a latching fault condition and should be linked to latching channel conditions such as outer error and outer feedback limits.

2) Fault 2 is a non-latching fault and should be linked to non-latching channel conditions such as inner error & inner feedback limits.

3) You are not required to link every channel condition to a fault. For example, if you are not concerned with performing an action based on an excitation change, no "X" is required for that condition.

****

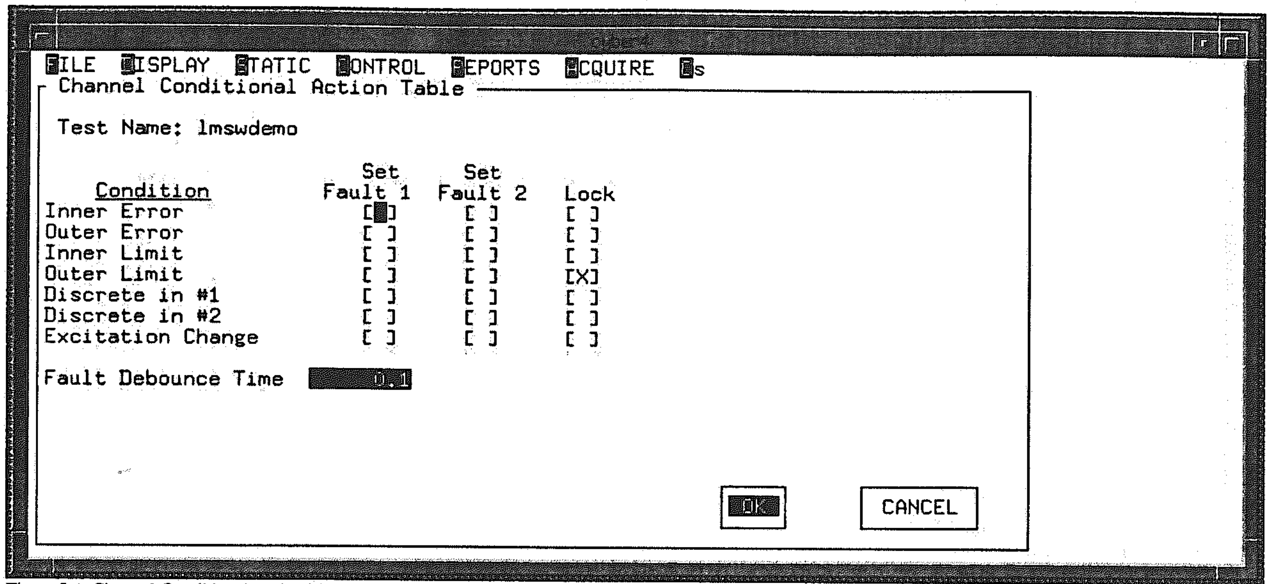


Figure 5-1. Channel Conditional Action Table Screen

Step 3: Enter the desired time delay in the Fault Debounce Time field at the bottom of the screen. The system uses 0.1 seconds as the default time. This entry corresponds to the minimum time the error condition must exist before the servo controller informs the system of the fault condition.

******************************** NOTE ******************************

Fault#1 and #2 (not the specific out-of-limit conditions) are used in the First Fault Ch# detection scheme (see Identifying the First Fault Channel in Chapter 14). The only time an out-of-condition will be active in flagging a particular channel as the 1st fault is when the error condition is linked to Set Fault 1 or Set Fault 2 as described above.

****

Step 4: When all channel conditional actions are defined, select OK to return to the Display menu.

Defining System Conditional Actions

Once the Channel Conditional Action Table is defined, you may go on to link the fault 1 and fault 2 conditions, as well as other system conditions, to various actions. The System Conditional Action Table defines system-wide responses to faults which may occur on individual channels or in ancillary systems such as the hydraulic system. Conditional actions may be defined for any of the following fault conditions:

- Fault 1 and Fault 2 are two predefined servo faults that are set in the Channel Conditional Action Table, discussed in the previous section. When these fault conditions occur, they may serve as the basis for a system-wide response.
- Discrete synch faults occur when the system tests the state of an input discrete and finds that the discrete state is different from the expected state, for greater than a user-defined duration of time.
- Hydraulic pressure faults occur when the hydraulic pressure is excessively low (i.e. from a "low-pressure" switch), as indicated by a "high" input at Discrete Input # 3.
- Hydraulic temperature faults occur when the hydraulic temperature is excessively high (i.e. from a "high-temperature" switch), as indicated by a low input at the Hydraulic Temp High discrete input.
- Inner pacing timeout faults occur when the load control system pauses more than a user-specified amount of time during a test step while waiting for slower channels to catch up. Inner pacing ensures that all channels operate synchronously, not only at endpoints but also in the transitions between endpoints.
- Endpoint tolerance timeout faults occur when a channel fails to reach the desired load condition value within a user-specified period of time.

************************************ [redacted] ******************************

Communication faults between the TMTM and the servo controllers are unconditional faults. Refer to Appendix I for an explanation of system response to communication faults.

****

Follow these steps to define System Conditional Action Table:

Step 1: Select the Display menu, System Conditional Actions option. The System Conditional Action Table screen, **Figure 5-2**, is displayed. The screen lists the possible system conditions at the left; the columns to the right define the available actions.

Step 2: Use the tab key to move the cell cursor to, and around, the desired condition line. The steps below explain how to select the actions.

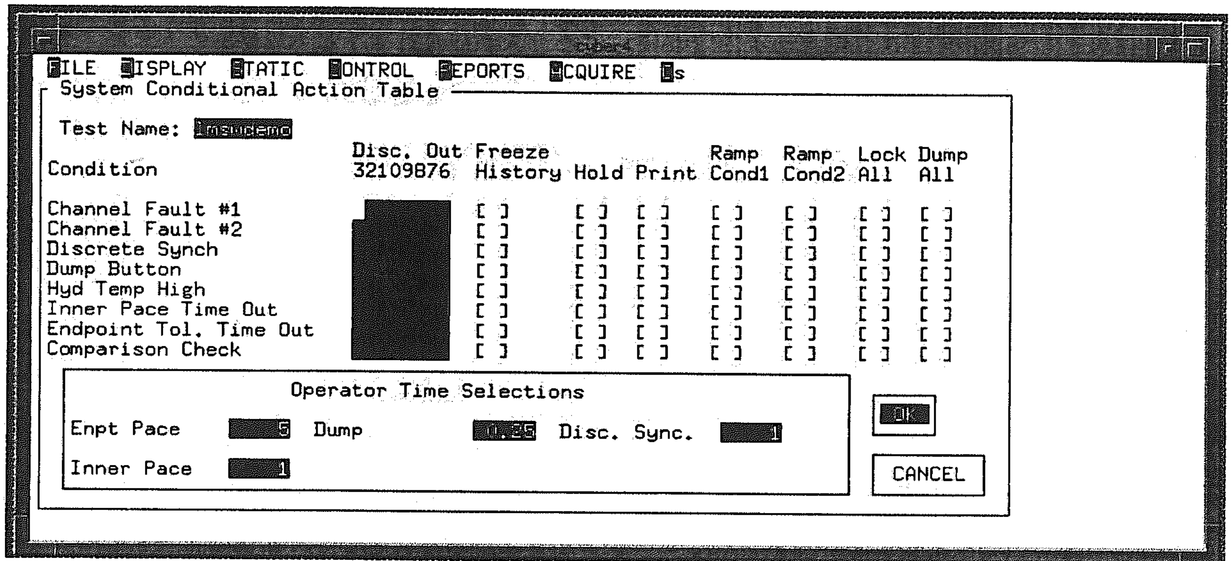


Figure 5-2. System Conditional Action Table Screen

Step 3: To set the state(s) of up to eight discrete outputs in the event of a specific system fault or condition, make the appropriate entries in the "Disc. Out, 32109876" column. The discrete outputs available are numbers 13 through 6. In the column description, the number "3" corresponds to discrete output #13, "2" to discrete output # 12, etc.. To set a discrete output to the normally open state, enter "H" in the corresponding column; to set a discrete to the normally closed state, enter "L". To leave a discrete in its existing state, enter "X" or leave the column blank. See Appendix B for further information on the discrete outputs.

Step 4: To "freeze" the history memory (i.e. Rotating Data Buffer (RDB)) in the servo controller at the time of a specific condition, press the space bar to enter 'X' in the Freeze History column in the row corresponding to the condition. When a Freeze History is activated, the system prevents further updates to the RDB (pending post-freeze % setting). The servo controller resumes updating the RDB when it receives a "Reset Error Conditions" command from the Master screen. After the data has been transferred to the host computer's hard disk, the RDB data is available for examination in tabular and/or graphic format.

********************************* [redacted] *****************************

Freeze History Data is intended to be used in conjunction with one of the conditional actions described in steps 5 through 9.

****

Step 5: To instantaneously hold all channels at their present load at the time of a specific condition, press the space bar to enter 'X' in the Hold column in the row corresponding to the condition.

Step 6: To get a printout of the load channel data at the time of the fault condition, enter X' in the Print column in the row corresponding to the condition.

Step 7: To ramp all channels to special load condition #1 whenever a specific condition occurs, press the space bar to enter 'X' in the Ramp Cond1 column in the row corresponding to the condition. The actual load condition that the servos ramp to depends on how the test has been defined (i.e. static or fatigue).

For fatigue tests, to specify the Ramp#1 load condition refer to the section labeled Defining Special Load Conditions for Fatigue Tests on page 11. Defining the time for this ramp is described in the next section labeled Defining Time-Sensitive Parameters.

For static tests, the Ramp#1 is defined as a specific test procedure step, along with a transition time. See *Defining Ramps for Static Tests* on page 13.

Step 8: To ramp all channels to special load condition #2 whenever a specific condition occurs, press the space bar to enter 'X' in the Ramp Cond2 column in the row corresponding to the condition. The actual load condition that the system ramps to when this action is invoked is defined
For fatigue tests, to specify the Ramp#2 load condition refer to the section labeled Defining Special Load Conditions for Fatigue Tests on page 11. Defining the time for this ramp is described in the next section labeled Defining Time-Sensitive Parameters.

For static tests, the Ramp#2 is defined as a specific test procedure step, along with a transition time. See *Defining Ramps for Static Tests* on page 13.

Step 9: To lock all channels at the current state at the time of a specific condition, press the space bar to enter 'X' in the Lock All column in the row corresponding to the condition. Activation of this function causes a servo "hold" and a corresponding "Lock All" relay on the discrete I/O panel to activate. Typically this function is used when the hydraulics actuators are designed to be physically locked in placed. Once a Lock All function has occurred, the function must be manually reset, as described in Resetting System Faults in Appendix D.

Step 10: To dump hydraulic pressure on all channels at the time of a specific condition, press the space bar to enter 'X' in the Dump All column in the row corresponding to the condition. When this function is activated, the servo controllers ramp to a predefined condition and a corresponding "Dump All" relay on the discrete I/O panel is activated. The actual load condition that the servos ramp to when this action is invoked is defined by the operator, as described in the section labeled Defining Special Load Conditions for Fatigue Tests on page 11. Defining the time for this ramp is described in the next section labeled Defining Time-Sensitive Parameters.

Once a dump has occurred, the function must be manually reset, as described in Appendix I.

DEFINING TIME-SENSITIVE PARAMETERS

The steps below describe each of the time-sensitive entries located at the bottom of the System Conditional Action Screen. All entries are in units of seconds.

Step 1: Enter the maximum time the system is to hold for endpoint errors to come into tolerance in the Enpt Pace field. If endpoint errors do not come into tolerance in this time, the system performs the specified action(s).

Step 2: In the Inner Pace field, enter the maximum time the system is to hold for an inner error. If the inner error persists beyond this time, the system performs the specified action(s).

Step 3: In the Dump field, enter the time for the servos to ramp to the dump condition (typically 0) when the Dump All command is issued. The time entered must not be shorter than 0.01 seconds

Step 4: In the Dscrt Synch field, enter the maximum time the system is to hold at endpoint for a discrete input to switch to an expected state. If the selected discrete input does not achieve the expected state within the time specified, a dscrt sync timeout occurs and the system performs the preprogrammed action(s) or holds where it is, if no action is programmed.

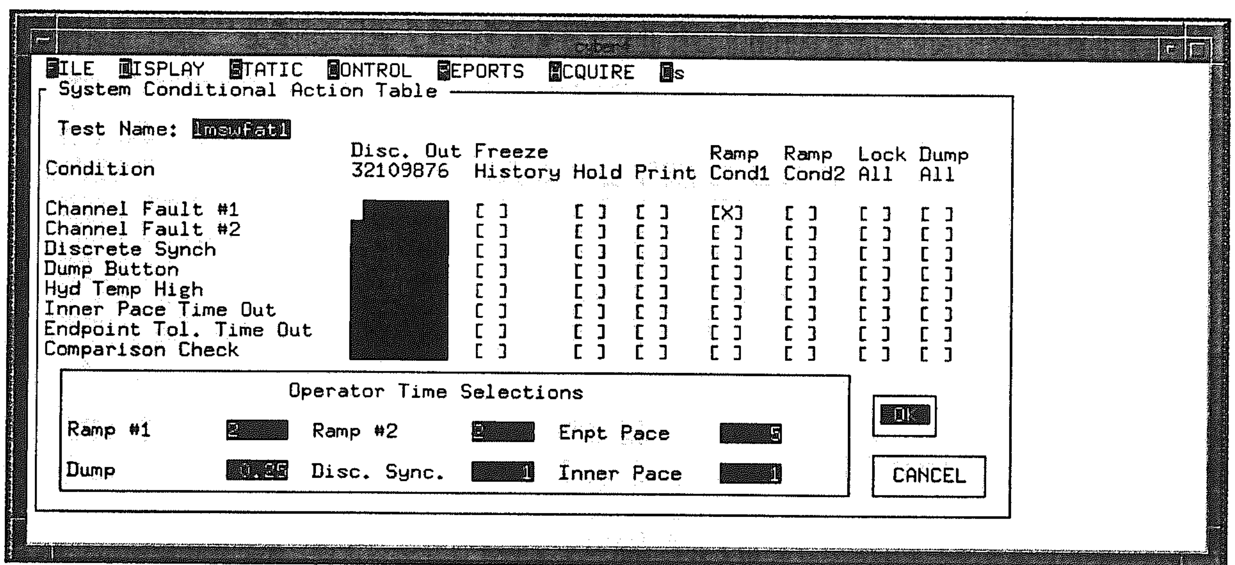


Figure 5-3 System Conditional Action Screen used with Fatigue Tests

All of the parameters described above pertain to both static and fatigue type tests. However, when running a test which has been created as a fatigue test, this System Conditional Action screen will also include two additional time entries; Ramp #1 and Ramp #2. See **Figure 5-3**. These entries work in conjunction with the RAMP buttons on the test control panel, as well as the "Ramp Cond1" and "Ramp Cond2" in the System Conditional Action Table.

When running a fatigue test, the ramp buttons can be used to ramp the system to a predefined load condition (also known as a special load condition). Defining special load conditions is described on page 11.

Step 5: Enter the ramp time for special load condition #1 in the Ramp #1 field. The time entered must not be shorter than 0.25 seconds.

Step 6: Enter the ramp time for special load condition #2 in the Ramp #2 field. The time entered must be no shorter than 0.25 seconds.

Step 7: When all system conditional actions are defined, select OK to return to the Display menu.

DEFINING DISCRETE CONDITIONAL ACTIONS

Discrete Conditional Actions allow the user to specify system responses to discrete input states. The same system-wide responses as in the System Conditional Action Table are found in the Discrete Conditional Action Table; the only difference is that the "conditions" are discrete inputs. As a minimum, discrete conditional actions may be defined for discrete inputs 6 through 13. Depending on your particular system configuration, you may have additional discrete inputs.

Follow these steps to define discrete conditional actions:

Step 1: Select the Display menu, Discrete conditional Actions option. The Discrete Conditional Action I screen, Figure 5-4, is displayed. The screen lists the discrete inputs (6-13) at the left; the screen's columns define the same conditional actions as the system conditional action screen.

Step 2: Refer to Defining System Conditional Actions, Steps 2 through 9 for a description of the various entries. Entries are made in an identical fashion, with the only difference being that an action is taken based on a discrete input state rather than a system condition. When an entry is made in any column for any discrete input, that input becomes active. If a "high" or open state is on that input, the conditional action is taken.

************************************ NOTE ************************************

Input discretes are "debounced". This means that the discrete action will be taken once for a transition to the active state of the input discrete. A discrete input must transition back to the non-active state before the system will "consider" it again to trip a discrete conditional action. Some systems have "short input" active low discretes. Verify that your particular configuration is as desired.

****

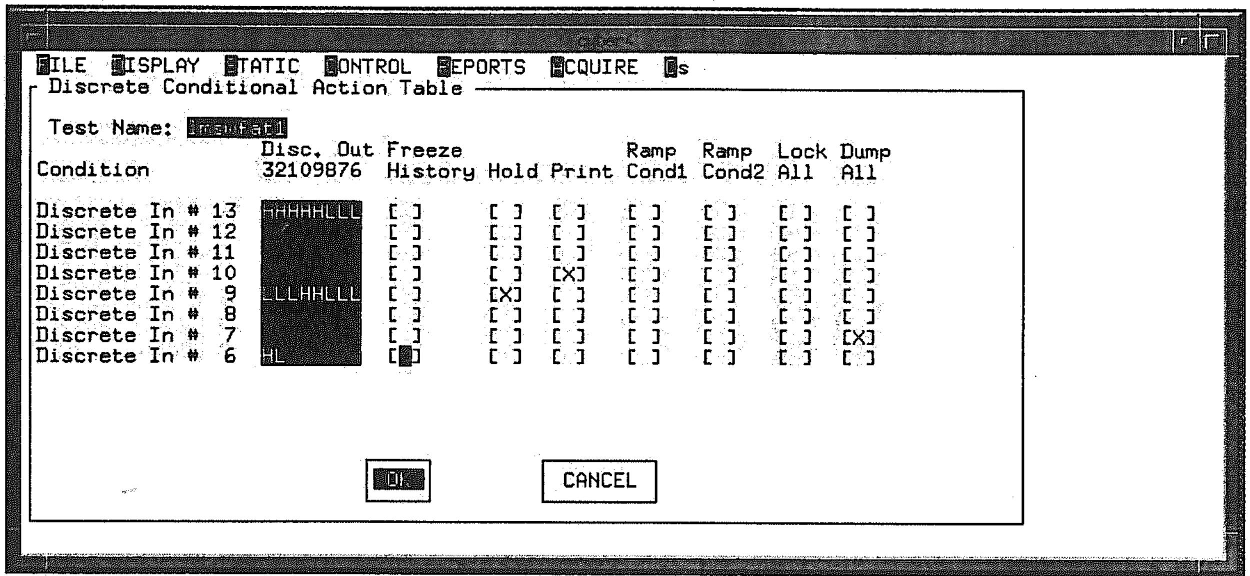


Figure 5-4. Discrete Conditional Action I Screen

Step 3: Some system configurations may have additional screens label Discrete Conditional Actions II and III. These screens provide additional discrete inputs which are set up the same way as those described above.

Step 4: When all discrete conditional actions have been defined, select OK.

DEFINING SPECIAL LOAD CONDITIONS FOR FATIGUE TESTS

For tests which have been defined as "fatigue", this section describes how to define:

- the load condition the servos ramp to when either of the Ramp buttons on the test control panel are pressed.
- the load condition the servos ramp to when either of the Ramps in the System Conditional Action Table are invoked.
- the load condition the servos ramp to when a Dump is activated.

All of the above items are defined by using the Special Load Conditions screen.

Follow these steps to define the special load conditions:

Step 1: Select the Display menu, Special Load Conditions option. The system prompts you to enter the desired channel number.

******************************** NOTE ******************************

Entry into this screen is only allowed in "fatigue" tests, as these values will be tied to the ramp buttons for fatigue tests.

****

Step 2: Enter the number of the channel for which you want to enter a special load condition and select OK. The Special Load Conditions screen, Figure 5-5, is displayed.

Step 3: Move the cursor to the line corresponding to the condition you want to define. Enter the load value that the system is to ramp to when the "**Ramp to 0**," "**Ramp to 1**," or "**Ramp to 2**" command is issued. If you don't enter a value, the system defaults to zero load.

******************************** NOTE ******************************

The "Ramp to 0" condition is the load condition that the system ramps to when a dump is initiated.

****

Step 4: Repeat steps 2 and 3 for each channel.

Step 5: The times associated with these ramp and dump conditions are setup on the System Conditional Action Screen, as described in labeled Defining Time Sensitive Parameters on page 11.

Step 6: When you finish, press **ESCAPE** to return to the Display menu.

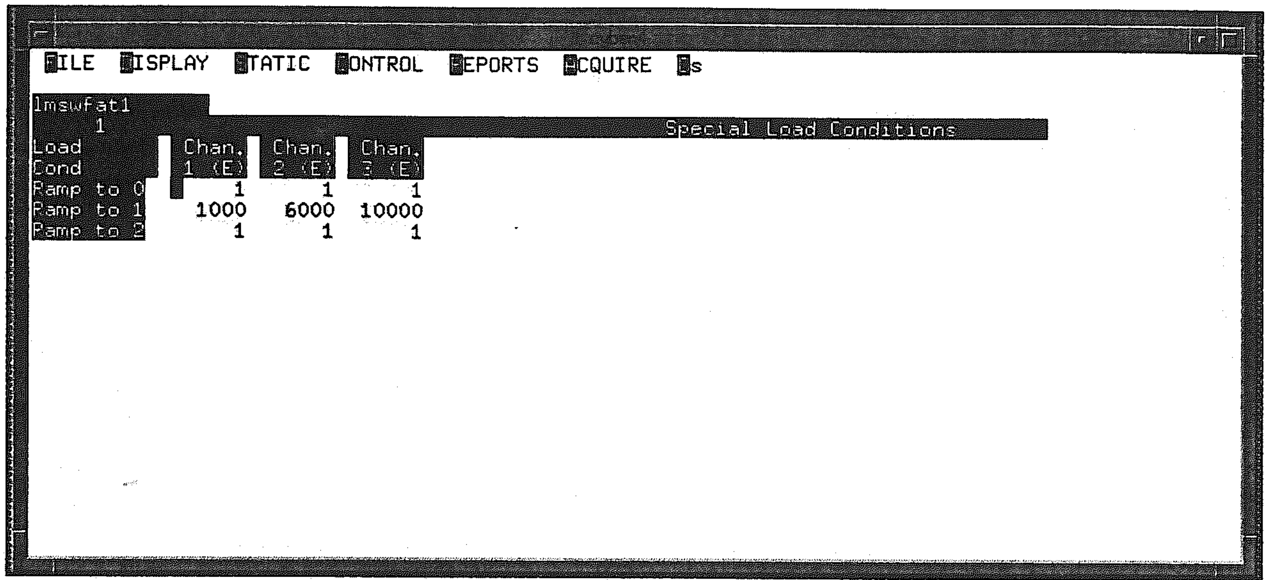


Figure 5-5. Special Load Conditions Screen

DEFINING RAMPS FOR STATIC TESTS

For tests which have been defined as "Static", this section describes how to define:

- the step the servos ramp to when either of the Ramp buttons on the test control panel are pressed.
- the step the servos ramp to when either of the Ramp actions in the System Conditional Action Table are invoked.

Both of the above items are defined by using the **[Set Ramps]** function.

************************************ NOTE ************************************

The **[Set Ramps]** function may be performed before or after a test has been downloaded.

****

Follow the steps given below to perform the Set Ramps function:

Step 1: Select the **STATIC** menu, **Branch To Step** option. The Branching screen shown in **Figure 5-6** is displayed.

Step 2: Under the column labeled Ramp 1 To Step, enter the block and step number you want the system to ramp to when a conditional action RAMP#1 action is invoked. The system uses Block 1 and Step 1 as the default location which is zero load for a load profile generated with the static software.

Step 3: Enter the time, in seconds, for this transition in the box labeled Time (Sec). The system uses 10 seconds as the default time.

Step 4: Enter the type of waveform to be used for this transition in the box labeled Waveform. The system uses a modified waveform (F) as the default. However, Ramp (R) or Haversine (H) may also be used.

Step 5: To define the Ramp 2 location, repeat steps 2 through 4 for the entries associated with the Ramp 2 To Step column.

Step 6: Select **[Set Ramps]** and press **RETURN**. At this point, pressing the either of the **RAMP** buttons on the test control panel will command the system to ramp to the preprogrammed block/step in the test procedure. The test will resume from the specified Block#/Step# when the **RUN** button is subsequently pressed.

Step 7: When you finish, press **ESCAPE** to return to the Display menu.

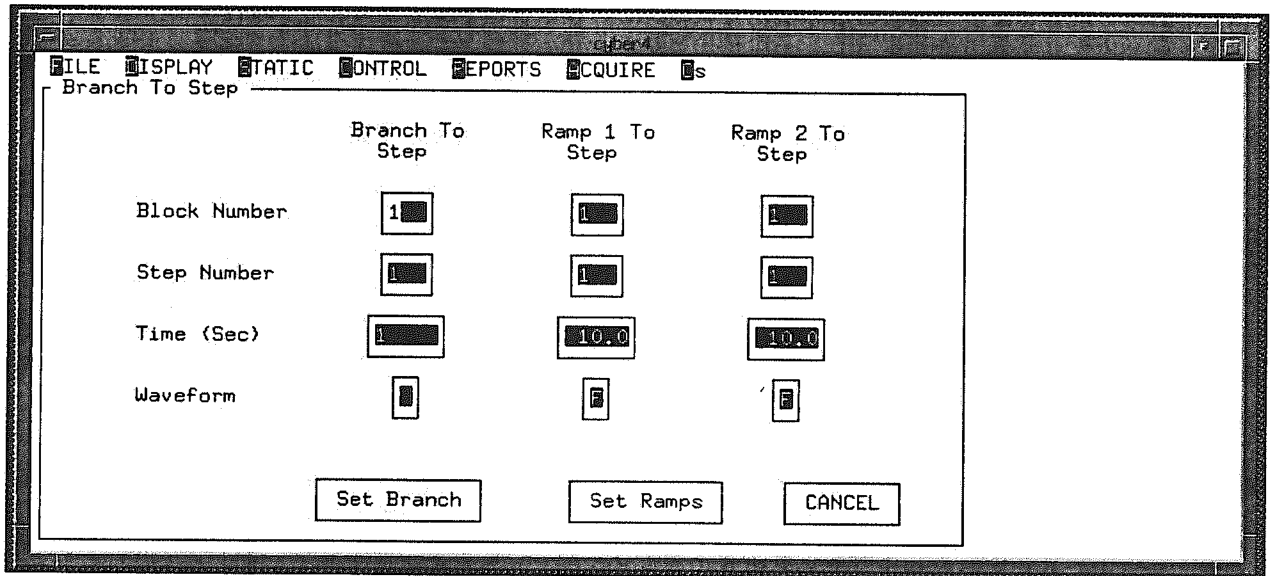


Figure 5-6. Using the Branching Screen to Define Ramps

??
BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

CHAPTER 6

PREPARING TO DOWNLOAD THE SETUP INFORMATION

When the test environment is ready and the load spectrum has been defined, you're ready to download the test definition files and run the test. This chapter explains the steps you follow to verify that the setup information is consistent and define the start up parameters.

You should be familiar with the information described in Chapter's 2 through 5 before beginning this chapter. If you have any questions about how to interface the servo valve and feedback transducer to the servo controller, refer to Appendix D.

???
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
???

Verifying the Setup Information

Defining test parameters can be a complex process with a high potential for error. Because of this, the Fatigue Master 7000 includes a verify program that performs an extensive crosscheck on each of the test parameters, ensuring that all required parameters are defined. Before executing a test, you must run the verify program; if the system locates any inconsistencies, specific on-screen messages identify the problem. The Fatigue Master 7000 will not allow you to start the test unless a successful verification is performed.

Verify the test parameters by selecting the Control menu, Verify option. This starts the verification process. As verification proceeds, the screen may display specific messages identifying inconsistencies in the test definitions. If inconsistencies exist, you must edit the test parameters to resolve each problem, and then re-run the verification function.

If the verification function completes its internal check without identifying any further inconsistencies, the program returns to the Control menu. When this occurs, verification is complete.

Defining Start Up Parameters

Before performing calibration functions and beginning a test, you must define certain parameters that relate to the test, including:

- which Test Control Panels are to be used with the test.
- choosing between starting a test from the beginning or restarting the test from a specific location in the test spectrum.
- how often data is to be recorded in each servo's rotating history data buffer.
- how much data before and after a "freeze event" should be recorded in the servo's rotating history data buffer and the DAS rotating history data buffer
- the tolerance for endpoint pacing; the process that ensures all channels reach the desired endpoint for each step before proceeding to the next step.
- the Valve Bias Limit which is used in conjunction with the automatic valve balance routine.
- the RCAL Deviation Limit which is used in conjunction with the RCAL Deviation routine.
- the Inner & Outer Redundant Check Limits for systems configured with this capability.
- whether to clear the tabular histogram data from an earlier execution of the test.
- whether to clear the Events Log from an earlier execution of the test.
- enabling hydraulic buttons on the Test Control Panel to activate hydraulic functions.
- enabling the "Engineering Units per Second" method of running a fatigue test.

Follow these steps to define the start up parameters:

Step 1: Select the Control menu, option Start Up. The Start Up screen, Figure 6-1, is displayed.

Step 2: Enter the number(s) of the Test Control Panel(s) used for the test in the fields labeled Pri. Test Panel and Sec. Test Panel. The number is determined by the physical jumper selection on the front of the Test Control Panel. If you use only one Test Control Panel, enter its number in the Pri. Test Panel field. If your system is only configured with one test, always enter 1.

For applications where redundant test control and monitoring is necessary, a secondary Test Control Panel can be used in conjunction with the primary panel to allow identical control and monitoring at a remote location. In such redundant configurations, each Test Control Panel is a mirror image of the other. For instance, if the RUN button is pressed at the primary panel, the secondary panel will indicate the same status by illuminating its RUN light.

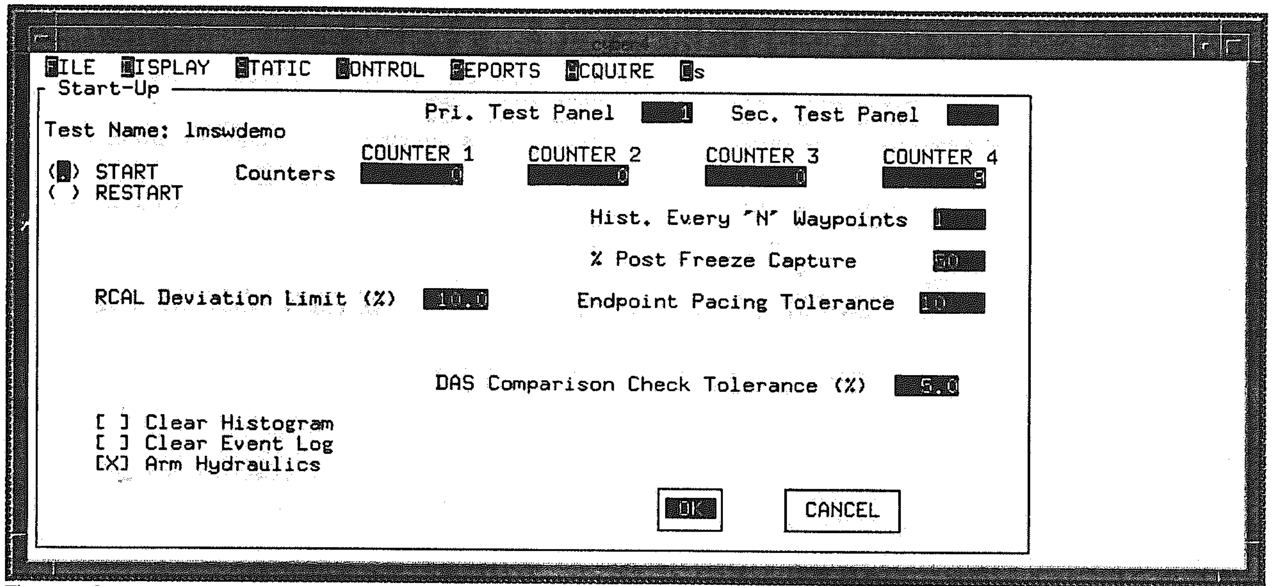


Figure 6-1. Start Up Screen

Step 3: Choose either **START** or **RESTART**, depending on whether you are starting the test at the beginning of the test procedure or are restarting a test which has been interrupted. If you are starting a test from the beginning go to step 5. If you are restarting a test at a point within the test procedure, follow step 4.

Step 4: If you are restarting the test, you must enter the counter values that correspond to point in the test from which you wish to resume. If appropriate, enter the numbers displayed on the Test Control Panel.

Enter the restart counter values in the fields labeled COUNTER 4 (SEGMENTS), COUNTER 3, COUNTER 2, and COUNTER 1. When a download is initiated, the system "fast-forwards" through the test to the point specified by the counter values. When the RUN button is pressed, all servo controllers used in the test will ramp to the load condition specified in the corresponding step of the test procedure and testing resumes.

***************************** NOTE ************************************

Your system may display different labels for the counters, depending on how you named them.

****

Step 5: If you plan to use the automatic Valve Balance feature during calibration, and wish to set a limit on how much valve bias can be applied in this routine, enter the desired limit in the Valve Bias Limit (%) field. The range of this entry is from 0 to 100% of full scale valve current. The default value for this entry is 20.0%.

Step 6: You may specify a limit in the RCAL Deviation Limit (%) field which is be used in conjunction with the RCAL Deviation check described in Chapter 7. The default value for this entry is 10.0%.

Step 7: Each servo controller channel includes a built-in Rotating Data Buffer (RDB) which is used to store critical loop parameters. The circulating buffer stores the following data at each sample point:

- **Command:** the spectrum command plus the analog command input (in engineering units). The analog command is typically zero.
- **Feedback:** the feedback signal (in engineering units)
- **Loop Error:** the loop error signal (in percent %)
- **Valve Current:** the actuator valve current in percent (%) of full scale valve current

The amount of history memory available with each servo channel is 16,384 samples (i.e. 4,096 samples for each of the 4 parameters). The Rotating Data Buffer is a circulating memory, where the memory locations are continually updated with the most recent data. The rate at which the Buffer is updated is determined by your entry in field labeled Hist. Every 'N' Waypoints.

Each servo controller has an internal update rate of 1,200 samples-per-second. Translating this to time (1/1,200) results in an 833 microsecond data sample period . . . one waypoint. Using the default value of 1 in the Hist. Every 'N' Waypoints field, causes the servo to store data at every waypoint (sample). If 25 is entered, the system will store every twenty-fifth data sample (or 25 x 833æs = every 21 milliseconds).

The number entered must be a positive integer. The larger the number, the less frequently data is stored and hence, the longer the time will be before the buffer begins to write over existing data. The list below gives some useful examples of how the sample rate can be defined:

Hist. Every 'N' Waypoints entry	Sample Interval	Total Buffer Time Available
1	833 us	3.4 Secs
25	21 ms	86 Secs
60	50 ms	3.4 Mins
200	167 ms	11.4 Mins

Step 8: When a system conditional action initiates a Freeze History, each servo controller assigned to the test is commanded to discontinue updating its respective history memory. You can control when the system stops adding data by entering a number from 0 to 100 in the field labeled % Post Freeze Capture. The number you enter represents the percentage of circulating memory (i.e. 4,096 data samples for each parameter) that is filled after the freeze occurs. If you enter "0", the History memory does not receive any new data after the freeze occurs; if you enter "100", the entire History memory receives new data after the occurrence of the freeze. Using the default value of 50, the system will maintain 50% of the existing data up until the freeze occurred, while continuing to fill the remaining 50% of the memory with post-freeze data.

Using the % Post Freeze Capture entry in conjunction with the Hist. Every 'N' Waypoints entry (update index), you can set the amount of time to acquire new data after the freeze occurs. For instance, using the formula and the example below, you can see how to determine this time:

[Update Index x waypoint time] x [% Post Freeze x Memory Size] = Recording Time
after freeze event

Example

Hist. Every 'N' Waypoints entry = 25
% Post Freeze Capture entry = 10% or 0.10
Waypoint time = 833 æseconds
Memory Size per Parameter = 4,096

[25 x 833æs] x [0.10 x 4,096] =8.54 seconds of data recorded after the freeze event occurs

If your system is configured with Cyber Auxiliary Data Acquisition System, the % Post Freeze Capture entry also works in conjunction with a 3 minute DAS Rotating Data Buffer (i.e. disk file) stored on the host computer's hard disk. This buffer is continuously updated with the 900 most recent scans of DAS data taken at 200 millisecond intervals. This updating continues until the buffer receives a "Freeze History" command. This is the same "Freeze History" which gets sent to the servo controllers based on entries in the System or Discrete Conditional Action Tables.

When this happens, the buffer continues to update with new scans such that the "% Post Freeze Capture" entry on the Start Up screen is satisfied. Thus, if the "% Post Freeze Capture" entry is set to 50%, the DAS rotating buffer will take 450 more scans (i.e 50% of the 900 total scans, approximately 1.5 minutes of data) before it stops updating. When the DAS rotating buffer is finished updating, you may request a hardcopy report for any DAS channel. The sample at which the freeze occurred is marked by an asterisk. This report is described in more detail in Chpaters 13 & 14.

Step 9: Enter the allowable deviation, as a percent of full scale, from the expected load at endpoint in the field labeled Endpoint Pacing Tolerance. When the Endpoint Pacing function is enabled from the Master screen, the servo controller compares the command value with the feedback data at each endpoint. The range for this entry is from 0 to 100% of full scale. The default value for this entry is 10. If the difference between the two values exceeds the amount specified in this field, test execution pauses. While waiting, the system continually tests the difference between the load cell feedback signal and the commanded load. If these subsequent comparisons fail for the length of time defined in the Enpt Pace field on the System Conditional Action Table screen, the conditional action defined in that table is executed.

Step 10: If you are running a fatigue test and wish to have the system determine the time for each step in the test procedure based on an "engineering units- per-second" factor, enter the desired loading rate in the field labeled Engineering Units/Second. The system takes this entry and divides it into the maximum delta load for each load transition to determine the transition time. Calculation of the maximum delta load for each load transition is performed during download of the test. As the test progresses, the maximum delta load for each step is read, divided by the Engineering Units/Second, and the transition time is determined. This entry works in conjunction with the EU per Sec box described in step 16 below.

Step 11: If your system is configured with Cyber's Auxiliary Data Acquisition and has been specifically set up to perform a redundant comparison between various DAS channels, enter the allowable tolerance for this check in the field labeled DAS Inner Redundant Limit (%). The range of this entry is from 0 to 100% of the DAS channel full scale. The default value for this entry is 5.0%. In the event that any comparison of DAS channels exceeds this tolerance, the MCU automatically generates a DAS Inner Redundant Limit fault condition and sets a dedicated discrete output on the MCU. This discrete output is typically tied to a discrete input on the FM7000 Discrete I/O panel. The Discrete Conditional Action Table may be used to linked to one or more actions to a DAS Inner Redundant Limit condition.

Typically this fault condition is linked to a cautionary action such as Hold. The second level of checking described in the next step is typically used for invoking more critical actions, such as Dump.

Step 12: Enter the allowable tolerance for a second comparison between various DAS channels in the field labeled DAS Outer Redundant Limit (%). The range of this entry is from 0 to 100% of the DAS channel full scale. The default value for this entry is 10.0%. In the event that any comparison of DAS channels exceeds this tolerance, the system automatically generates a DAS Outer Redundant Limit fault condition (second discrete output on the MCU) which may be linked to one or more actions in the Discrete Conditional Action Table.

Step 13: To clear archived tabular histogram data for the new execution of the test, use the space bar to enter an 'X' in the box labeled Clear Histogram. The default state (i.e. empty box) causes new data to be appended to the archived tabular histogram data.

Step 14: To clear the Events Log, use the space bar to enter an 'X' in the box labeled Clear Events Log. The default state (i.e. empty box) causes new information to be appended to the archived events log data.

Step 15: If the hydraulic buttons (On/Off, Low, High) on the Test Control Panel have been wired to your hydraulic subsystem, in order to enable them you must enter an 'X' in the box labeled Arm Hydraulics.

Step 16: If you plan to use the Engineering Units per Second feature described in step 10, use the space bar to enter an 'X' in the box labeled EU per Sec.

Step 17: When the start up parameters are defined and you are ready to download data, select OK or press RETURN. A message will appear on the screen stating "Transferring data to TMTM", indicating that the previously defined test data is now being downloaded to the TMTM and servo controllers. When this data transfer is completed, the message will disappear and the master AMPlitude & FREQuency displays on the selected Test Control Panel will blink on and off. If EU per Sec is selected, an additional message, "Generating Delta Loads for EU per Sec" will be displayed first.

You're now ready to perform the calibration of the control channels and/or start the test.

************************************ NOTES ************************************

If you have previously calibrated the control channels and are starting/restarting a test, and do not wish to re-calibrate the control channels, skip Chapter 7 and go to Chapter 9.

If you are preparing to run a test using the hydraulic simulators (i.e. hydraulics off), you do not have to, nor does the system allow you to, calibrate the control channels. Therefore, skip Chapter 7 and proceed to Chapter 9. That chapter also describes how to enable the hydraulic simulators.

****

??
BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

CHAPTER 7

CALIBRATING THE CONTROL CHANNELS

Calibration is the process that adjusts the components of the servo loop to achieve the following conditions in the control channel:

- The voltage across the unshunted load cell bridge is adjusted to zero volts (i.e. the bridge is balanced).
- The feedback amplifier gain is adjusted when the R-CAL shunt method of calibration is chosen during setup of the Channel Description screen.
- The system determines the valve current required to provide zero error (i.e., balance the actuator valve) with zero "0" command.
- The feedback signal caused by initial tare loads is offset in order to provide a test article zero condition.
- The command signal is adjusted until the loop error equals "0", thus allowing the hydraulic subsystem to be activated without causing abrupt changes to the actuator position. This is often referred to as a soft start.
- Optionally, you can have the system execute a built-in "pre-tune" spectrum and evaluate the performance of the control channel while hydraulics are active and the loop is closed. Using this information, the system determines the Bode gain and phase curves and sets the proportional amplifier and rate compensation gains appropriately. Please note, however, it is not necessary to perform the pre-tune calibration routine.

Appendix D offer additional information on the available calibration methods, as well as how to wire the servo controller to your valve and feedback transducer.

??
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

PERFORMING BRIDGE BALANCE CALIBRATION

If one or more of the control channels you have defined in your test is connected to a load cell bridge, follow these steps to perform the Bridge Balance procedure:

Step 1: Select the Control menu, Calibration option. The Calibration dialog box shown in **Figure 7-1** is displayed.

Step 2: Using the radio buttons, select the Bridge Balance function.

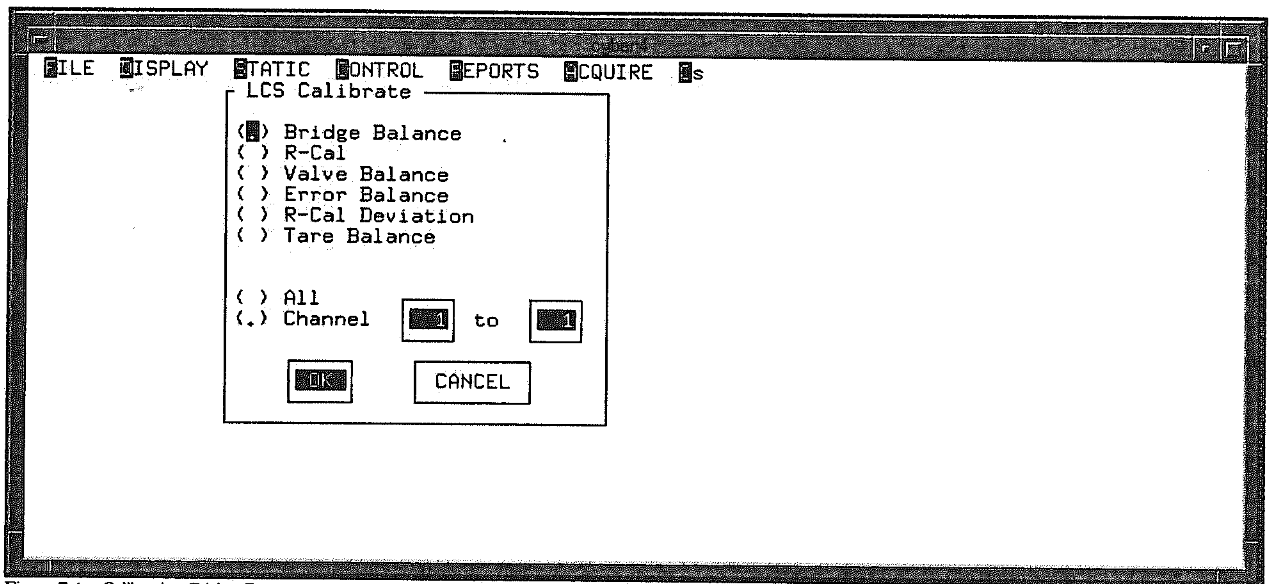


Figure 7-1 Calibration Dialog Box

Step 3: Select the channels to be calibrated:

- If all are to be done, enable the button in the All field.
- If a range of channels is to be done, enable the button in the Channel field. Then enter the first and last control channel numbers in the range in the two boxes to the right of Channel.

Step 4: Select OK or press RETURN to start the calibration process. While calibrating, the system displays a calibration message and the time remaining for completion of the calibration procedure.

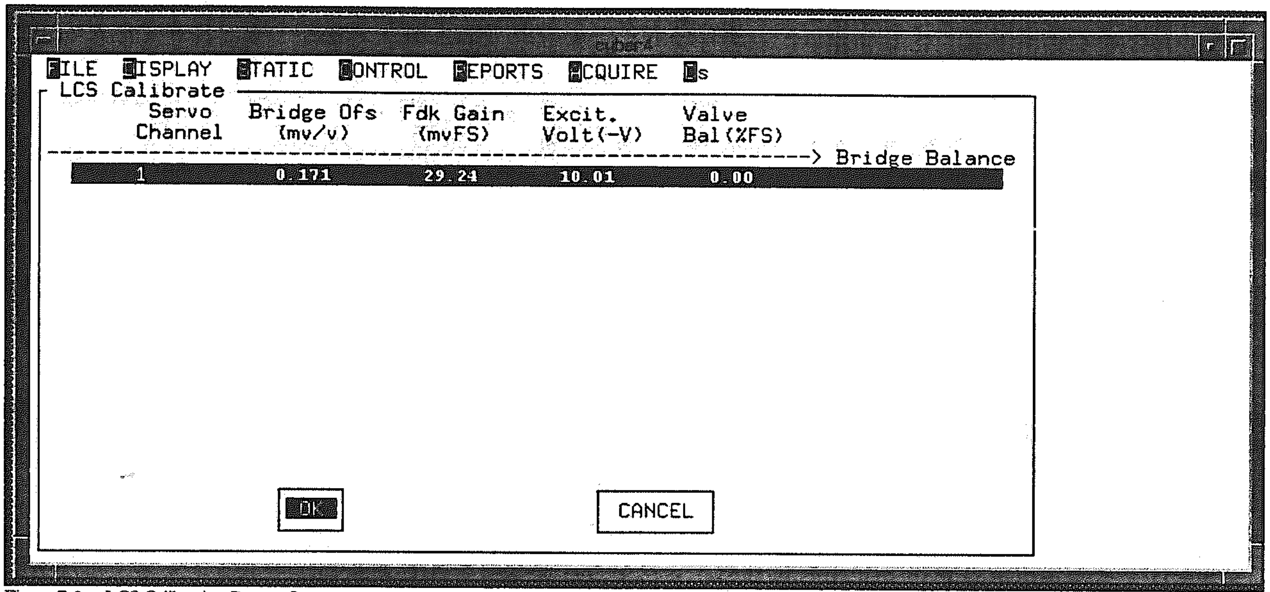


Figure 7-2 LCS Calibration Report Screen

Step 6: After the system has completed this function, the LCS Calibration report shown in Figure 7-2 is displayed. The column labeled Brdge Ofs (mv/v) indicates how much bridge offset was necessary to balance each of the control channels which were calibrated.

Step 7: Select **CONTINUE** to view additional channels not shown on the first page of the report.

Step 8: Press the **ESCAPE** key (**F11**) to exit the LSC Calibration screen.

PERFORMING THE R-CAL PROCEDURE

The **R-CAL** procedure invokes an algorithm within the servo controller which automatically adjusts the feedback amplifier to the proper gain setting.

If one or more of the control channels you have defined in your test is connected to a load cell bridge (or other bridge type transducer) and you wish to use the **R-CAL** procedure, follow these steps:

********************************** NOTE ************************************

In order to perform the **R-CAL** calibration procedure, you must have previously entered an R-CAL factor on the Channel Description screen (Chapter 3) prior to activating the R-CAL procedure. You must also install a corresponding **R-CAL** shunt resistor on the rear of the servo controller.

****

Follow these steps to perform the R-CAL procedure:

Step 1: Select the Control menu, Calibration option. The Calibration dialog box is displayed.

Step 2: Using the radio buttons, select the **R-CAL** function.

Step 3: Select the channels to be calibrated:

- If all are to be done, enable the button in the All field.
- If a range of channels is to be done, enable the button in the Channel field. Then enter the first and last control channel numbers in the range in the two boxes to the right of Channel.

Step 4: Select **OK** or press **RETURN** to start the calibration process. While calibrating, the system displays a calibration message and the time remaining for completion of the calibration procedure.

Step 5: When the **R-CAL** procedure has been completed, the LCS Calibration report is displayed on the screen. The column labeled Fdk Gain (**mvFS**) indicates what full scale feedback range was set for each channel calibrated.

Step 6: Select **CONTINUE** to view additional channels not shown on the page.

Step 7: Press the **ESCAPE** key (**F11**) to exit the LSC Calibration screen.

PERFORMING AN R-CAL DEVIATION CHECK

The **R-CAL** Deviation procedure performs the same function as the **R-CAL** procedure, without actually setting the feedback gain to the "just determined" value. Instead, the "just determined" value is compared to the original **R-CAL** results, and the difference between the two values is reported to the host. This information is displayed to the operator and allows him to verify the integrity of the load cell and feedback amplifier at various times throughout the duration of the test.

********************************** NOTE ***********************************

In order to perform the **R-CAL** Deviation procedure, you must have previously performed the **R-CAL** procedure.

****

Follow these steps to perform the R-CAL Deviation procedure:

Step 1: Select the Control menu, Calibration option.

Step 2: Using the radio buttons, select the R-CAL Deviation function.

Step 3: Select the channels to be calibrated:

- If all are to be done, enable the button in the All field.
- If a range of channels is to be done, enable the button in the Channel field. Then enter the first and last control channel numbers in the range in the two boxes to the right of Channel.

Step 4: Select OK or press RETURN to start the calibration process. While calibrating, the system displays a calibration message and the time remaining for completion of the calibration procedure.

Step 5: Once the check is completed, the R-CAL Deviation Report screen shown in Figure 7-3 is displayed. The column labeled Feedback Gain (mvFS) indicates what the new full scale feedback range would be if you were to perform the R-CAL procedure again.

The column labeled Deviation Gain (mvFS) indicates the amount of change in the feedback gain, in millivolts. The deviation in percent is given in the column label Deviation. If the amount of deviation for any channel exceeds the RCAL Deviation limit specified on the Start Up screen, a failed message is displayed under the Status column.

****************************** NOTE ******************************

The system does not prohibit you from running with channels which exceed this limit.

****

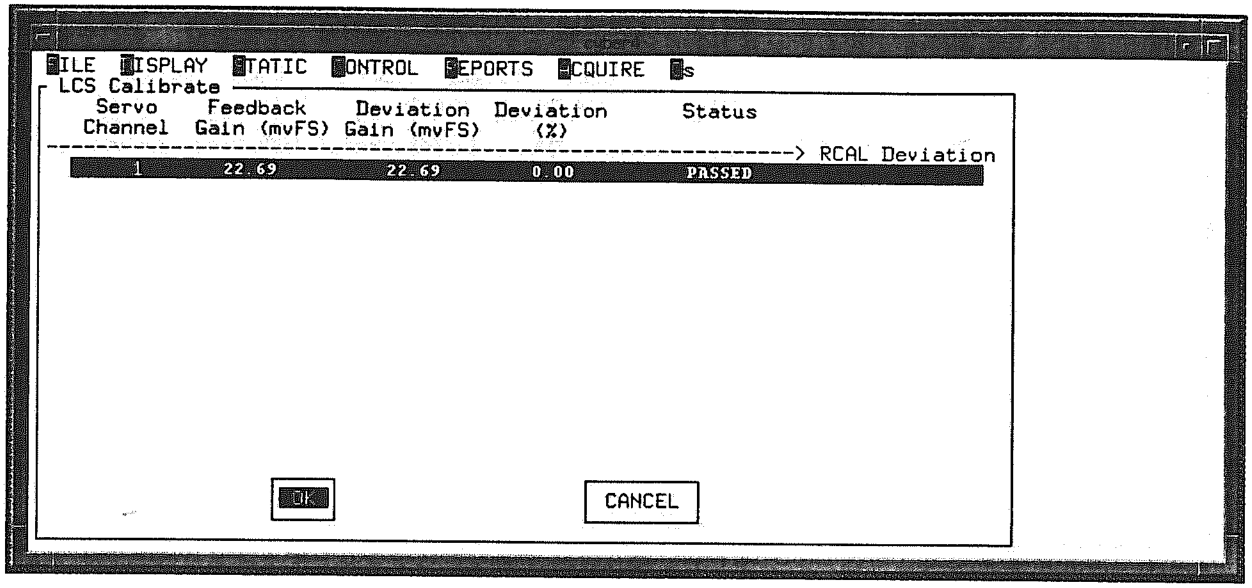


Figure 7-3 R-CAL Deviation Report Screen

Step 6 Select **CONTINUE** to view additional channels not shown on the page.

Step 7: Press the **ESCAPE** key to exit the LCS Calibration screen.

PERFORMING AUTOMATIC VALVE BALANCE

If you wish to have the system perform an automatic Valve Balance procedure, follow the steps outlined in this section. If you wish to adjust the valve bias manually, skip this section and refer to the next section labeled Adjusting the Valve Bias Manually.

Step 1: Select the Control menu, Calibration option. The Calibration dialog box is displayed.

Step 2: Using the radio buttons, select the Valve Balance function.

*********************************** NOTE *************************************

If you select this function, it must be performed while the hydraulics are active and the loop is closed, as described in Chapter 10, section labeled Activating Hydraulics.

****

Step 3: Select the channels to be calibrated:

- If all are to be done, enable the button in the All field.
- If a range of channels is to be done, enable the button in the Channel field. Then enter the first and last control channel numbers in the range in the two boxes to the right of Channel.

Step 4: Select **OK** or press **RETURN** to start the Valve Balance routine. While calibrating, the system displays a calibration message and the time remaining for completion of the calibration procedure.

Step 5: Upon completion of this routine, the LCS Calibration report will display the results under the columns labeled Valve Bal (%FS) and Valve Bias.

The Valve Bal (%FS) column lists the amount of valve current (relative to full scale valve current) required to balance the valve. The Valve Bias column is used to flag any channel which required more valve bias than specified on the Start Up screen. The default limit is 20.0% of full scale valve current.

***************************** NOTE **********************************

The system prohibits you from running the test with excessive valve bias.

****

Step 6: Select CONTINUE to view additional channels not shown on the first page of the report.

Step 7: Press **ESCAPE** to exit the LSC Calibration screen.

ADJUSTING THE VALVE BIAS MANUALLY

To adjust the valve bias manually from the servo front panel, follow these steps:

Step 1: Use the rocker switches on the selected servo front panel to select **VLVO** from the **SETUPCAL** category. **VLVO** stands for valve offset.

Step 2: While observing the control channel's feedback value, use the lower rocker switch to adjust the amount of valve offset injected into the valve driver circuit. You will know when the feedback reads approximately zero for this channel.

Step 3: Return the servo front panel display to the desired parameter.

Step 4: Repeat steps 1 through 3 for each channel you wish to adjust.

PERFORMING A TARE BALANCE

The Tare Balance procedure works in conjunction with the engineering unit value entered in the Tare Offset column of the Channel Control screen, and is intended to be used to offset any initial loads caused by actuators, fixturing, etc.. It adjusts the channel's bridge balance DAC until it reads the user-entered Tare Offset value on the feedback. The Tare Balance procedure also includes a second procedure, known as Error Balance, which automatically adjusts the command signal until "0" loop error is achieved. Performing Tare Balance allows the test to be initiated without any abrupt changes out at the actuators; effectively providing a soft start.

********************************** NOTE *************************************

In order to perform the Tare Balance calibration procedure, you must ensure that hydraulics are NOT active, that you have entered a Tare Offset factor, in engineering units, for each channel defined on the Channel Control screen (Chapter 3), and that the bridge has been zeroed and the feedback gain has been set to the desired value.

****

Follow these steps to perform the Tare Balance procedure:

Step 1: Select the Control menu, Calibration option.

Step 2: Using the radio buttons, select the Tare Balance function.

Step 3: Select the channels to be calibrated:

- If all are to be done, enable the button in the All field.
- If a range of channels is to be done, enable the button in the Channel field. Then enter the first and last control channel numbers in the range in the two boxes to the right of Channel.

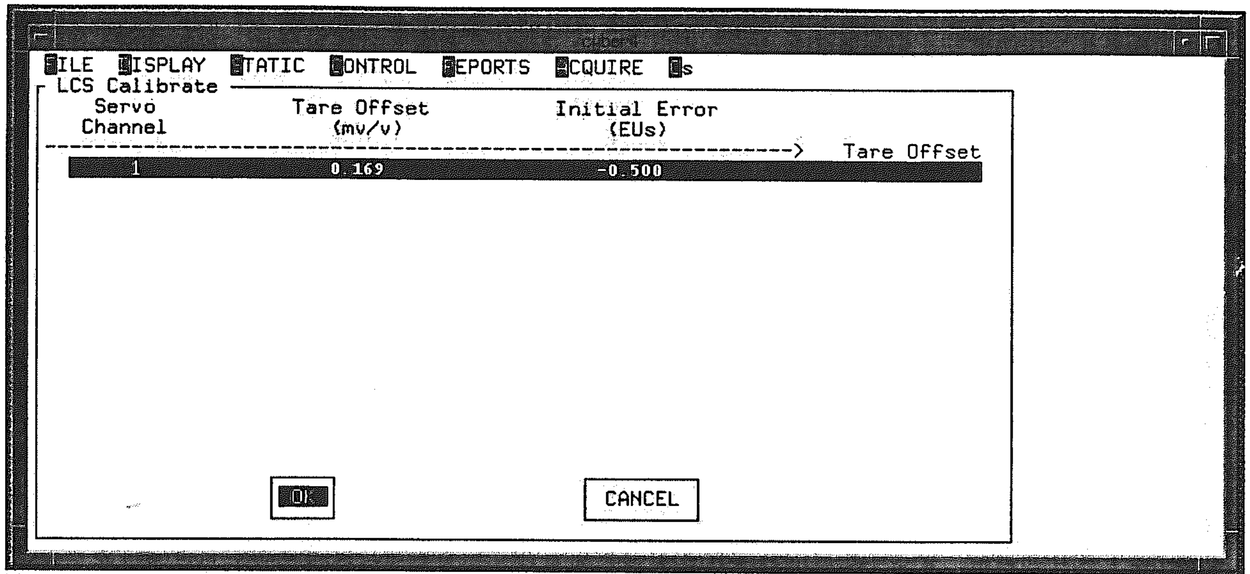


Figure 7-4 Tare Balance Report Screen

Step 4: Select **OK** or press **RETURN** to start the calibration process. While calibrating, the system displays a calibration message and the time remaining for completion of the calibration procedure.

Step 5: Once Tare Balance is completed, the Tare Balance report shown in Figure 7-4 is displayed. The column labeled Tare Offset (mv/v) indicates how much tare offset was used for each control channel. The column labeled Initial Error (EUs) indicates what the initial loop error was for each control channel prior to when the system performed the Error Balance portion of the calibration procedure.

Step 6: Select **CONTINUE** to view additional channels not shown on the first page of the report.

Step 7: Press the **ESCAPE** key to exit the LSC Calibration screen.

Step 8: Since the Error Balance portion of the routine adjusted the command in order to achieve "0" loop error, you may initiate the test according to the steps given in Chapter 10.

PERFORMING AN ERROR BALANCE

Although the Error Balance routine is an integral part of the Tare Balance procedure, there are times when you may want to perform the Error Balance procedure separately, such as after a hydraulic dump has occurred. In such instances, one or more channels may experience a non-zero loop error condition which, if not corrected, could cause abrupt loading upon re-activation of the hydraulic system. For this purpose, enabling the Error Balance routine automatically adjusts the command signal for each control channel until "0" loop error is achieved.

Follow these steps to perform the Error Balance procedure:

Step 1: Select the Control menu, Calibration option.

Step 2: Using the radio buttons, select the Error Balance function.

Step 3: Select the channels to be calibrated:

- If all are to be done, enable the button in the All field.
- If a range of channels is to be done, enable the button in the Channel field. Then enter the first and last control channel numbers in the range in the two boxes to the right of Channel.

Step 4: Select OK or press RETURN to start the calibration process. While calibrating, the system displays a calibration message and the time remaining for completion of the calibration procedure.

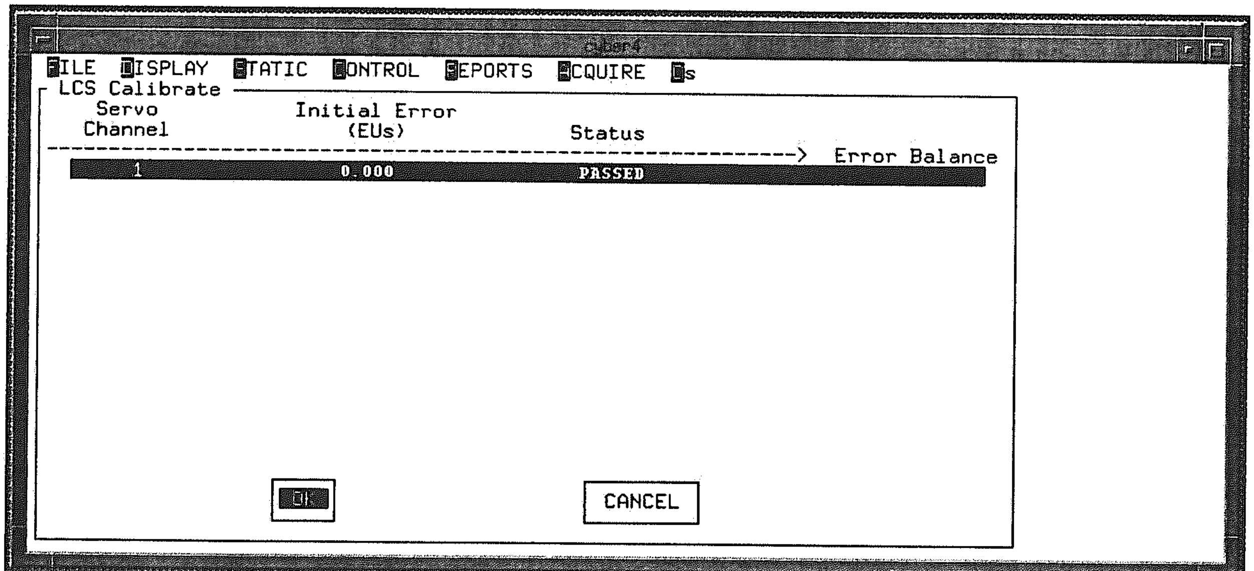


Figure 7-5 Error Balance Report Screen

Step 5: Once Error Balance is completed, the Error Balance report shown in Figure 7-5 is displayed. The column labeled Initial Error (EUs) indicates what the initial loop error was for each control channel, in engineering units, prior to calibration.

Step 6: Select **CONTINUE** to view channels not shown on the first page.

Step 7: Press the **ESCAPE** key (**F11**) to exit the LSC Calibration screen.

PERFORMING THE PRETUNE PROCEDURE

The system includes a Pretune function which applies a built-in spectrum to the control loop, determines its Bode gain and phase, and automatically sets the proportional gain and rate compensation of the channel.

If this calibration function is performed, a proportional gain cap for each selected channel is determined using Bode estimation. When auto-tuning is enabled, channels which have been pretuned will operate with a maximum proportional gain equal to the Bode gain cap. Channels which have been pretuned will retain their respective gain cap until a SETD (set defaults) is performed from the servo controller front panel.

**********************************NOTE*************************************

If auto-tuning is enabled without a pretune having been performed, the proportional gain which is used is based on oscillation criteria and min/max gain caps, not Bode estimation. This function must be performed with the hydraulic system on.

****

Follow these steps to perform the PRETUNE procedure:

Step 1: Select the Control menu, Calibration option.

Step 2: Using the radio buttons, select the **PRETUNE** function.

Step 3: Select the channels to be calibrated:

- If all are to be done, enable the button in the All field.
- If a range of channels is to be done, enable the button in the Channel field. Then enter the first and last control channel numbers in the range in the two boxes to the right of Channel.

Step 4: Select OK or press RETURN to start the calibration process. While calibrating, the system displays a calibration message and the time remaining for completion of the calibration procedure.

Step 5: Once the selected channels are calibrated, the system displays the Calibration Report screen, Figure 7-2, which shows the results of the most recent execution of the calibration function.

Step 6: Select **CONTINUE** to view channels not shown on the first page.

Step 7: Press the **ESCAPE** key (**F11**) to exit the LSC Calibration screen.

SUMMARY OF THE LCS CALIBRATION REPORT SCREEN

The following calibration information, which resides in non-volatile memory associated with each servo channel, is provided on LCS Calibration Report screen:

Servo Channel The physical channel number.

Bridge Ofs (mv/v) The equivalent mv/v value of the load cell bridge offset.

Max Prop Gain The maximum gain cap determined by the Bode estimator used in the pretune calibration procedure.

Fdk Gain (mvFS) The load cell full scale in millivolts as determined by the R-CAL procedure

Excit Volt (-V) The excitation voltage in volts.

Valve Bal (FS) The amount of valve current required to balance the valve, in percent(%) of full scale valve current.

********************************** NOTE **************************************

If a particular calibration function is not performed, the appropriate default values will be displayed.

****

??

BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.

??

CHAPTER 8

SETTING UP & CALIBRATING THE AUXILIARY DATA ACQUISITION SYSTEM

If your system is not configured with Cyber's Auxiliary Data Acquisition System, continue reading from Chapter 9. If your system is configured with Cyber's AuxDAS and you wish to use it, make sure you've read Chapter 1 of this manual to obtain a basic understanding of the individual elements associated with AuxDAS. Refer to page 14 of Appendix D for instructions on how to interface your transducers to the DAS hardware, as well as how to install the RCAL & bridge completion resistors for bridge type inputs. You should not continue with Chapter 8 until you have completed the interface as described in Appendix D.

Throughout this manual Cyber uses the abbreviations AuxDAS, ADAS, and DAS interchangeably to refer to the Auxiliary Data Acquisition System.

??
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

SETTING UP THE AUXILIARY DATA ACQUISITION SYSTEM

Cyber's Auxiliary Data Acquisition System operates in the same way as the rest of the Fatigue Master 7000 software. It's menu driven and uses a combination of spreadsheet screens and dialogue boxes to make entries and/or initiate specific functions. **Figure 8-1** shows the Fatigue Master 7000 ACQUIRE menu. This menu is the starting point for all DAS operations.

The first task in using the AuxDAS is to set up the data acquisition channels to be used in the test. When all desired values are entered, you must perform a Verify of the AuxDAS setup. This works in a similar fashion to the LCS Verify, providing a cross-check of the DAS setup data.

When AuxDAS Verify passes, you must then perform calibration of the data acquisition channels. Additionally, if the Rosettes, Computed Channels, and/or other available options are present and the use of one or more is desired, you must enter the necessary set up data.

This chapter provides a detailed step-by-step procedure for performing each of the above tasks.

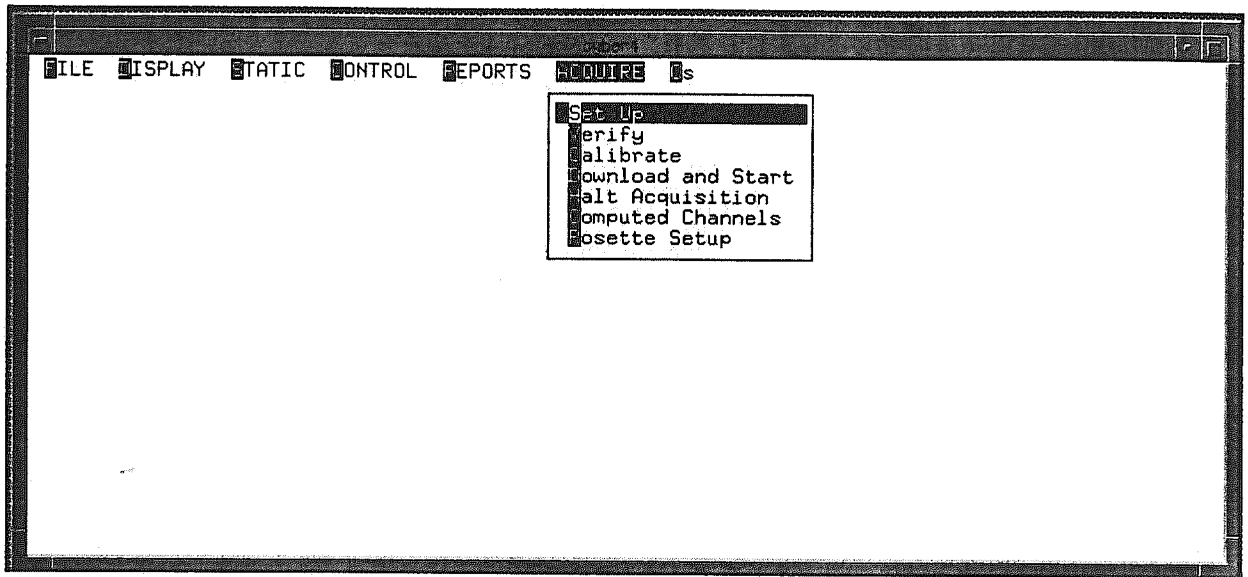


Figure 8-1 Acquire Menu

DEFINING THE DAS CHANNELS

The first task in using the DAS in conjunction with the LCS is to define the data acquisition channels to be used in the test. Channels may be defined in any order up to the maximum number in the system. In this process you:

- Enter the DAS channel number corresponding to its physical location in the chassis
- Assign a name to each input channel
- Specify the input type for each channel, such that the proper type of calibration is performed later on in the calibration procedure
- Enter the engineering units you wish to use for the data
- Enter the full scale and calibration factor in engineering units
- Define minimum and maximum limits on the data for each channel
- Specify the sequence of the DAS channels on the DAS Subset Test Data display/report
- Enable/disable redundant limit checking in the MCU on a per channel basis (if configured)

Follow these steps to set up the DAS channels:

Step 1: Select the **ACQUIRE** menu, **Setup** option. The AuxDAS Setup screen shown in **Figure 8-2** is displayed.

Step 2: Enter the channel # in the column labeled Chan No. This number can range from 8 to the maximum number of DAS channels configured with the system

☐

☐

Step 3: Enter the name of the channel (if desired) in the column labeled Chan Name.

Step 4: In the column labeled Chan Type, enter the channel type according to the following table:

Chan Type	Description	Calibration Functions Performed
0	Voltage Channel	The feedback gain is set according to the calibration factor
6	Voltage Channel with voltage suppression	Initial Offset is removed, then the feedback gain is set according to the calibration factor
7	Bridge Channel	Initial Offset is removed, then the RCAL shunt is invoked in order to establish the proper feedback gain for that channel

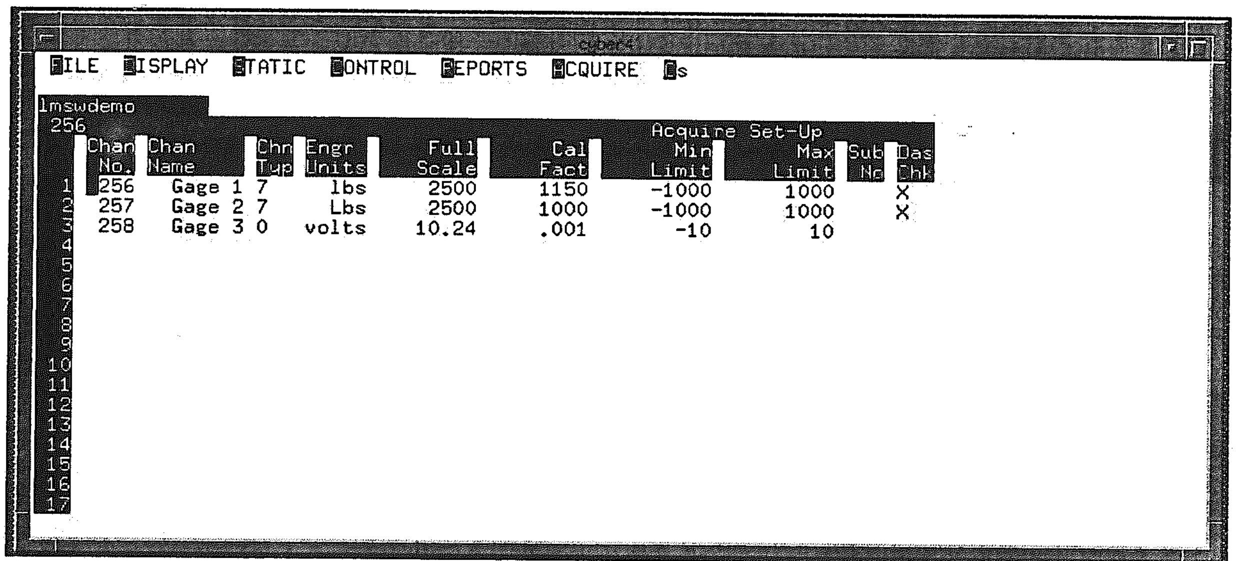


Figure 8-2 AuxDAS Setup Screen

Step 5: Enter the engineering units of each channel in the column labeled Engr Units.

Step 6: Enter the full scale for each AuxDAS channel in engineering units in the column labeled Full Scale.

Step 7: Enter the calibration equivalent value in engineering units in the column labeled Cal Fact. For bridge type channels (type 7), which are to be calibrated by using an RCAL shunt applied across one arm of the bridge, enter the engineering unit equivalent of the RCAL shunt resistor. For voltage channels (types 0 and 6), enter the engineering unit-per-millivolt relationship (e.g. 10 lbs/mV) for the transducer.

☐

Step 8: If you wish to flag a given channel for being out of range when it is read at endpoint, enter the appropriate numbers in the columns labeled Min Limit and Max Limit. If the channel's value is greater than the Max Limit, an "H" will be placed in the Limit column of the Test Status Report. If the value is less than Min Limit, an "L" appears in the Limit column of the Test Status Report. If the value is between Min Limit and Max Limit, the column is left blank.

Step 9: If you wish to reorganize the sequence of DAS channels, or select a subset of DAS channels to be used with the DAS Test Data display/report, enter the desired numerical position of the channel in the column labeled Subset No. For example, if your test has 20 DAS channels (8 through 27), but you're primarily concerned with channels 10 and 16, entering a "1" in the Subset No. column for channel 10, and a "2" in the Subset No. column for channel 16 will allow you to obtain an special version of the DAS Test Data display/report which only includes these two channels. The sequence of the channels will be in numerical order of the subset numbers. In our example above, channel 10 would be the first channel on the subset display/report, followed by channel 16. More information on the Subset DAS Test Data display/report is given in Chapter 11.

Step 10: Some system configurations include an additional field labeled **Check Active** or **Das Chk**. This field is used to enable/disable, on a per channel basis, redundant limit checking which is performed by the MCU. If your system includes this field and you wish to enable redundant limit checking in the MCU, use the space bar to enter an "X" in this field for the selected DAS channel(s). Leave the field blank in order to disable the redundant limit check. You may change which channels are in the limit check, but you must recalibrate the DAS in order for the change to take effect.

Step 11: When all channels have been defined, select [ESCAPE] to return to the ACQUIRE menu.

☐

☐

☐

☐

☐

DEFINING ROSETTES

□

□

A rosette is a particular physical orientation of strain gauges which allows the calculation of a series of stress, strain, and associated directional data for loads applied to a test article. The FM7000 ADAS accommodates the use of two widely used rosettes: rectangular and bi-directional (i.e. two gauge). **Figure 8-3** illustrates each configuration.

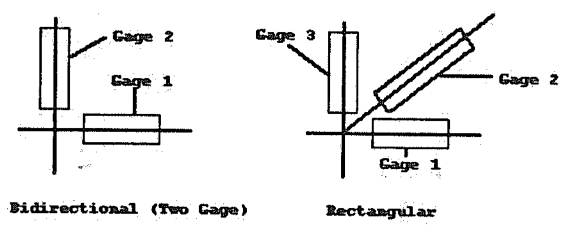


Figure 8-3 Rosette pictures

For each rosette, the FM7000 software computes a series of parameters based on reading each strain gauge, and directs the results to the Test Data display/report. The parameters computed are:

- Maximum and minimum strain
- The angle from gauge 1 axis to the maximum normal stress axis
- Maximum and minimum stress
- Shear stress

Follow these steps to set up rosettes:

Step 1: Select the **ACQUIRE** menu, **Rosette** option. The Rosette set up screen, **Figure 8-4** is displayed. You should notice the Chan No column which automatically numbers each rosette starting with 1000.

Step 2: Enter the name (if desired) of each rosette in the column labeled Chan Name. Step 3:Enter the type of rosette in the column Ros Type. For a rectangular rosette, enter "r", and for a bi-directional rosette, enter "b". Capital letters are also acceptable.

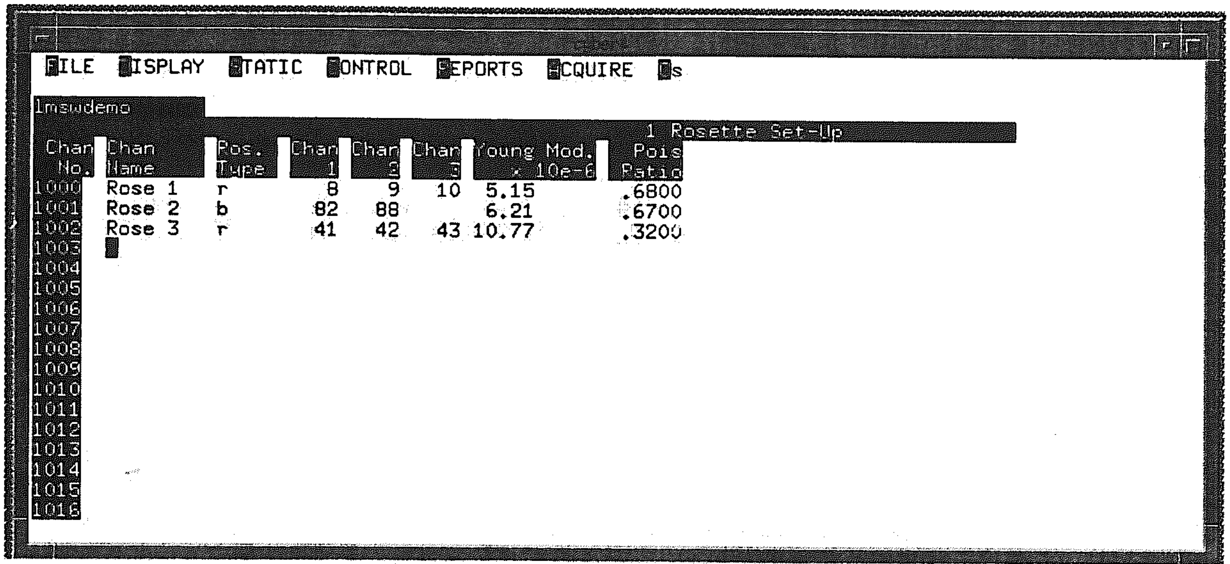


Figure 8-4 Rosette Setup

Step 4: In the following three columns: **Chan1**, **Chan2** and **Chan3**, enter the **DAS** channels assigned to that rosette. If the rosette is bi-directional, you must use only the **Chan1** and **Chan2** columns. The strain values read from these channels are used in the rosette computation.

******************************** NOTE ******************************

All channels entered in the Rosette setup must be independent (i.e. a channel may only be assigned to one rosette).

****

Step 5: Enter the elasticity constant for each rosette in the column labeled Young Mod x 10e6. The number entered is scaled by 106 before being used in computations.

Step 6: Enter the Poisson ratio of the test article material in the column labeled Pois Ratio. This factor accounts for strains produced normal to the direction of primary strain (i.e. secondary strains) in the rosette calculation.

Step 7: When all entries have been properly made, select [ESCAPE] to return to the Acquire menu.

DEFINING COMPUTED CHANNELS

□

□

When the FM7000 is configured with Cyber's DAS, the software allows mathematical manipulation of DAS channel data according to "computed channel" equations which you define. Up to 256 computed channels may be defined per test. These channels may be viewed under Test Data Display or replayed from disk.

To define one or more computed channels, follow the steps given below:

Step 1: Select the **ACQUIRE** menu, **COMPUTED CHANNELS** option. The Computed Channel setup screen shown in **Figure 8-5** is displayed. You should notice the Chan No column which automatically numbers each computed channel starting with the number #2000.

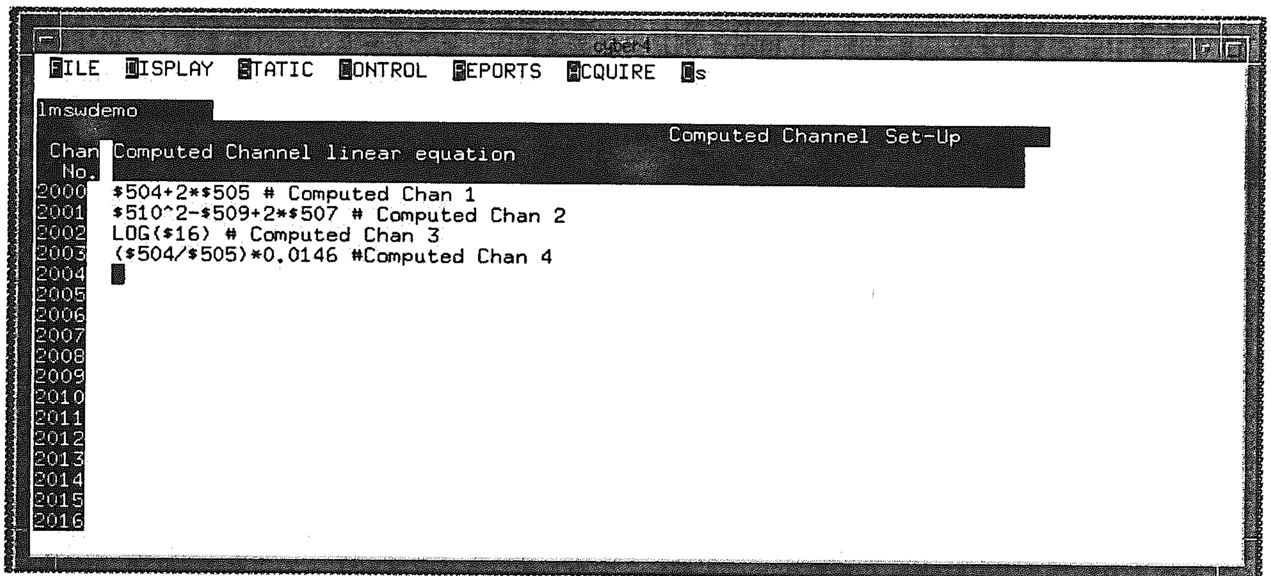


Figure 8-5 Defining Computed Channels

Step 2: Define the computed channel equation in accordance with the information described below and in **Step 3**. Each equation can include the following arithmetic operators:

Unary operators:

- \+ Positive sign
- \- Negative sign

Binary operators:

- \+ Addition
- \- Subtraction
- * Multiplication
- / Division
- ^ Raise to the power
- ( ) Parenthesis

You may also use transcendental functions within each equation. A list of the available functions and their syntax is shown below. The parentheses must always be used.

ABS( )	Takes the absolute value of the function.
ACOS( )	Finds the angle in radians whose cosine is the function.
ASIN( )	Finds the angle in radians whose sine is the function.
ATAN( )	Finds the angle in radians whose tangent is the function.
COSH( )	Finds the hyperbolic cosine of the function in radians.
COS( )	Finds the cosine of the function in radians.
EXP( )	Finds the value of e raised to the power of the function.
LOG10( )	Finds the log base 10 of the function.
LOG( )	Finds the natural log of the function.
POW10( )	Finds the value of ten raised to the power of the function.
ROUND( )	Rounds off the function to the nearest integer value.
SINH( )	Finds the hyperbolic sine of the function in radians.
SIN( )	Finds the sine of the function in radians.
SQRT( )	Finds the square root of the function.
SQR( )	Finds the square of the function.
TANH( )	Finds the hyperbolic tangent of the function in radians.
TAN( )	Finds the tangent of the function in radians.
TRUNC( )	Removes the fractional part of the function.

Step 3: When defining computed channels, the following rules must be adhered to:

Rule #1 A "$" is used to indicate a DAS channel number. For example, $8 indicates DAS channel # 8.

Rule #2 DAS channels used within computed channel equations must first be defined in the ACQUIRE Set Up screen.

Rule #3 The equation must be continuous; no spaces between factors and operators. Below are examples of acceptable and unacceptable equations:

$8+500/$9*1000	acceptable
$8 + 500 / $9 * 1000	unacceptable

Rule #4 Operations within the equation are not prioritized; the equation is strictly computed from left to right. For example, in the equation below, the system will add 500 to the data from Channel #8, divide the sum by Channel #9, and multiply the quotient by 1000.

$8+500/$9*1000

Rule #5 Use parenthesis to prioritize operations within the equation.

Rule #6 Multiplication is not implied by parenthesis. Therefore, a multiplication sign "*" must be used between factors and parenthesis.

Rule #7 Computed channels must be defined in a contiguous manner. For example, 2000, 2001, 2002, etc..

Rule #8 A previously defined computed channel may be used in a subsequent one. For example, computed channel #2004 may include computed channel #2001 in its equation. However, computed channel #2004 can not be used within other computed channels. This restriction is often referred to as single level nesting.

Rule #9

Computed channels used within an equation are identified by a preceding "#". For example:

Comp. Chan	Equation
2001	2000*$4
.	. .
.	. .
2005	$7+15/8+#2001

Rule #10 No recursive nesting is allowed. For example:

□

□

Comp. Chan	Equation
2001	$8+#2001

□

□

□

Step 4: Enter a name for each computed channel in the column labeled Chan Name. Capital letters or lower case letters are acceptable.

Step 5: If you wish to edit a computed channel equation, you must move the cell cursor to the desired line and re-type the entire equation again.

Step 6: When all entries have been properly made, select [ESCAPE] to return to the **ACQUIRE** menu. Remember, up to now, all of the computed channel information you entered is being held in temporary files, use the **SAVE** function to permanently save this information to disk.

Defining Displacement Transducer Inputs

□

□

Your system may include this displacement transducer option which is designed to acquire BCD data from up to 64 displacement transducers, in synchronization with the FM7000 load profile. This is accomplished through the use of an MVME340 discrete I/O board in the Motorola host computer and external multiplexing hardware provided by the customer. Six discrete outputs are used to address the appropriate displacement transducer channel, while 25 discrete inputs read the BCD data (24 + sign). These discretes are accessed at the back of the host through a dedicated connector panel (see Appendix A for the pin assignments).

The operator assigns the desired displacement transducer channels to a particular test through the use of a spreadsheet entry screen. At each load profile endpoint, all 64 channels are "scanned"; however, only the data from the displacement channels defined in the test setup is merged with the standard system endpoint data of that test (i.e. LCS & ADAS). This data is available for display, hardcopy, and/or recording. If you are running multiple independent tests, each of which has one or more displacement transducers defined in its setup, the system will automatically merge the data from the displacement transducer channels defined in each test with the corresponding TS file.

To set up displacement channels, perform the following steps:

Step 1: Select the **ACQUIRE** menu, **Disp. Setup** option. The Displacement Transducer Spreadsheet screen, **Figure 8-6** is displayed.

Step 2: Enter the desired displacement transducer channel number in the column labeled **Chan. No.**. Enter the channel's name (if desired) in the column labeled **Chan Name**. Up to 64 channels may be entered for a given test with channel numbers ranging from 1 to 64.

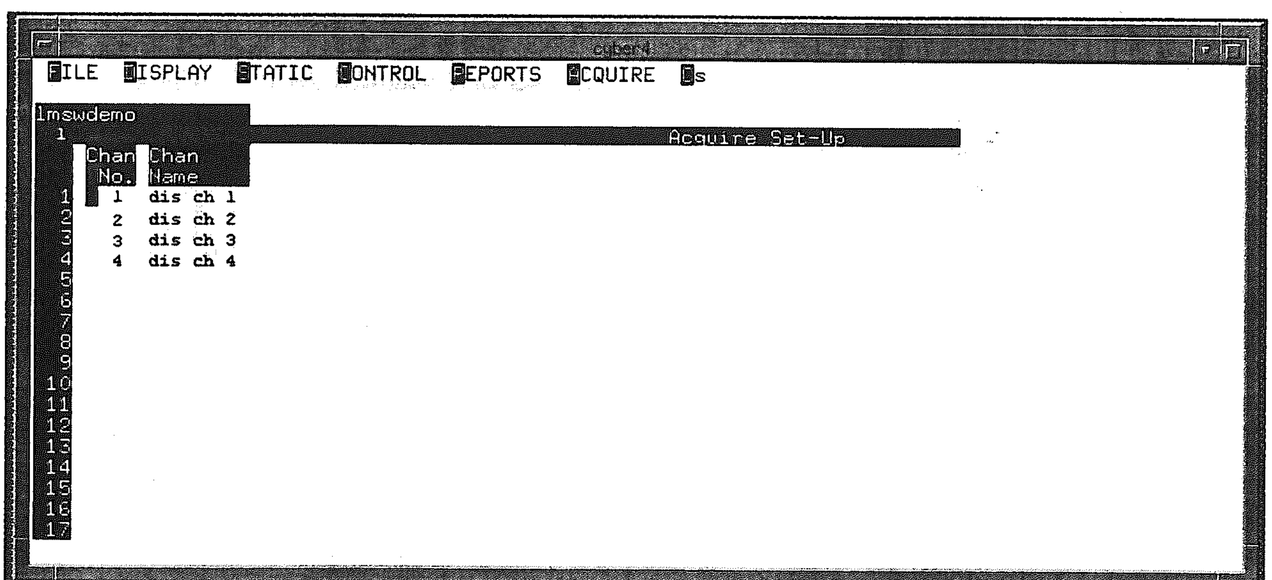


Figure 8-6 Displacement Transducer Setup Screen

Step 3: When all channels have been entered, press the **ESCAPE** key (**F11**) to return to the **ACQUIRE** menu.

Recording and monitoring of the displacement transducer data is covered in subsequent chapters.

VERIFYING THE ADAS SETUP

When all ADAS setup has been completed, you are ready to initiate a cross-check of the entries made using the ADAS Verify program. Any setup errors encountered (e.g. missing entries, duplicate entries, out of range entries, etc.) are flagged and displayed. You must correct any errors and re-initiate ADAS Verify before proceeding. When no more setup errors are present, the display returns to the ACQUIRE menu.

To activate the ADAS Verify cross-check procedure, simply select the **ACQUIRE** menu, **VERIFY** option. The program runs as described above.

CALIBRATING THE DAS FOR THE FIRST TIME

When a successful Verify has been performed, you are ready to calibrate the ADAS. The primary functions of the calibration procedure are:

1. Determine the optimum gain for each channel to allow for maximum resolution.

2. Determine correction and conversion coefficients to provide maximum accuracy.

Two modes of calibration are available: balance and drift. The more common of the two, balance calibration, is used for selecting optimum gains and correcting for any inherent offset. The drift calibration procedure, on the other hand, is normally performed after the original balance calibration (e.g. typically months) and determines how much each channel has drifted from its original (balance) calibration readings, providing an additional correction factor. With the precision, low-drift analog hardware of the ADAS, drift calibration is seldom necessary.

Follow these steps to calibrate the ADAS of the FM7000:

Step 1: Select the **ACQUIRE** menu, **Calibrate** option. The calibration dialog box, **Figure 8-7** is displayed.

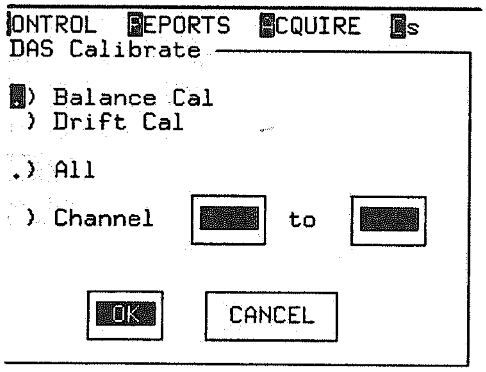
☐

Step 2: Select the mode of calibration to be performed, **Balance Cal** or **Drift Cal**. Do not select **Drift Cal** unless a balance calibration has previously been performed.

Step 3: Select the channels to be calibrated.

- If all are to be done, keep the button in the All field.
- If a range of channels is to be done, place the button on the Channel field and enter the first and last ADAS channel numbers in the range in the two boxes to the right of Channel.

Figure 8-7 DAS Calibration Dialog Box



Step 4: Select **OK** or press **[RETURN]** to start the calibration process. While calibrating, the system displays which calibration sub-task is being performed.

Step 5: Once the selected DAS channels are calibrated, the system displays the ADAS Calibration Report screen, **Figure 8-8**, which shows the results of the most recent execution of the ADAS Calibration function.

Test Description:

Test Name: LMSW Demo
DAS Balance Display

Last Calibration Date/Time: Sept 30 1998 17:44:51

Chan No.	FullScale (mv)	Gain	Offset (Counts)	Res (Counts)	R-CAL (EU)	Scale Fac A(EU/mv)	Balance Offset(mV)	Zero Read Z(EU)
504	20	512	-12	6	938.7	108.125	-0.015	1.782
505	20	512	-15	-5	1249.8	144.012	0.012	-2.989
506	20	512	15	-2	1561.5	179.865	0.004	-2.415

Figure 8-8 DAS Calibration Report Screen

The following information is provided on the ADAS calibration Report Screen:

Channel No	The ADAS channel number
Full Scale (mV)	The Full Scale in millivolts as chosen by the R-CAL procedure
Gain	The gain setting for that channel as determined by the R-CAL procedure; inversely proportional to FS (mV)
Offset (counts)	The value, in counts, of the DAC setting for this channel.
Res (counts)	The value in counts of any offset remaining after the offset DAC is set.
R-CAL (EU)	The calibration factor as entered on the Setup screen.
Scale Fac A (EU/mV)	The scale factor used in converting a voltage reading into engineering units for each channel
Balance Offset (mV)	A corrective offset value added to the readings on that channel
Zero Read Z (EU)	a corrective zeroing factor applied to the engineering unit values on each channel

Step 6: Select **[CONTINUE]** on the calibration results display by pressing **[RETURN]**. This allows you to view all of the DAS channels which have been calibrated. If you do not wish to see the results, select **[CANCEL]**.

When the last page of the report has been displayed and you have pressed **[RETURN]** or by selecting CANCEL, the system exits the calibration display to return to the **ACQUIRE** menu. What happens next depends on the type of MCU configured with your system, Z-80 based or 68000-based. Both configurations are described below:

Z-80 Based MCU (prior to 1992)

If your system is configured with Cyber's Z-80 based MCU, the host computer transfers the scan list to the MCU via the serial RS-232 link. This can be observed by watching the "Host Transmitting" LED on the front of the MCU, which is brightly illuminated during this phase.

Once the scan list transfer is completed, the MCU immediately begins scanning the DAS channels. This is indicated by a series of flickering LEDs on the MCU. If you have previously downloaded the

LCS test setup information from the Start Up screen (Chapter 6), the DAS Test Data display should update approximately every half second.

The actual scanning occurs at a UNIX-dependant interval (about 50 ms), i.e. the host repeated issues scan/halt commands to the MCU. This tells the MCU to take a scan all the DAS channels in the system, then wait for the next scan command. The rate at which the channels are scanned is 20,000 channels-per- second. When DAS recording is enabled (in the test procedure and Master screen), the host records the endpoint data of the scan closest to the endpoint.
68000-Based MCU

If your system is a later configuration of the FM7000 which uses the 68000-based MCU, the host computer transfers the scan list to the MCU via DMA. Depending on the length of your scanlist (i.e. number of DAS channels in the system), this will take anywhere from 3 to 5 seconds.

Once the scan list transfer is completed, the MCU immediately begins scanning the DAS channels. This is indicated by a series of rapidly flickering LEDs on the MCU. If you havepreviously downloaded the LCS test setup information from the Start Up screen (Chapter 6), the DAS Test Data display should update approximately every half second for small and medium systems and every two seconds for large systems.

The actual scanning is independent of the host/UNIX. Instead, the MCU operates in an intervalometer mode, taking a scan of all channels every 50-200 milliseconds depending on the number of DAS channels in the system. The rate at which the channels are scanned is always 20,000 channels-per second. The DAS data which gets recorded is the scan which resides in shared memory at the time the load control endpoint occurs (i.e. within one interval time (50-200 ms) of endpoint occurrence).

Step 7: If further calibration is necessary, select the appropriate function.
□

□

*********************************** NOTE **************************************
Verification and calibration of the DAS may take place before or after LCS setup data has been downloaded to the TMTM. Furthermore, the DAS may be re-calibrated while the test is active, provided that the system is in either a paused, ramped, or hold condition.
****

□

RE-CALIBRATING ONE OR MORE DAS CHANNELS

□

□

To re-calibrate one or more DAS channels, follow the steps given below:

□

□

Step 1: Select the **ACQUIRE** menu, **Calibrate** option. The calibration dialogue is displayed.

□

Step 2: Select the mode of calibration to be performed, **Balance Cal** or **Drift Cal**. Do not select **Drift Cal** unless a balance calibration has previously been performed.

□

□

Step 3: Select the channels to be calibrated.

- If all are to be done, keep the button in the **All** field.
- If a range of channels is to be done, place the button on the Channel field and enter the first and last ADAS channel numbers in the range in the two boxes to the right of Channel.

Step 4: Select **OK** or press **Return** to start the calibration process. While calibrating, the system displays which calibration sub-task is being performed.

□

□

Step 5: Once the selected channels are calibrated, the system displays the DAS Calibration Report screen with the results of the most recent DAS calibration. The report includes all of the DAS channels in the test. Channels which were not re-calibrated maintain their original results, while re-calibrated channels display new results.

HALTING THE DAS SCANNING/ACQUISITION

If you wish to stop the DAS scanning, select the **ACQUIRE** menu, **HALT ACQUISITION** option. A dialog box is displayed which states "Acquisition Halted". Press **[RETURN]** to halt the acquisition process or press the **ESCAPE** key (**F11**) if you choose not to. If you pressed **[RETURN]**, you will notice that the LEDs on the MCU have stopped flickering. If you wish to resume scanning, without re-calibrating the DAS, follow the procedure given in the proceeding section. If you wish to re-calibrate one or more of the DAS channels, follow the instructions given earlier in this chapter.

DOWNLOADING THE SCAN LIST MANUALLY

□

□

There may be times when you wish to download the scan list without necessarily re-calibrating the DAS. Some examples of when this might be required, include:

- after the DAS scanning/acquisition has been halted and you do not wish to re-calibrate the DAS
- when recovering from a powered-down state and you do not wish to re-calibrate the DAS
- if the "smm" process was killed, but you've restarted it and wish to resume scanning again, without re-calibrating the DAS.

In order to deal with these situations, the FM7000 includes a means of manually downloading the scan list, without re-calibrating the DAS. To do this, select the ACQUIRE menu, Download and Start option. The scan list is downloaded and the MCU begins scanning immediately.

*********************************** NOTE ************************************

The DAS must have been calibrated at some point earlier in the test, in order for the necessary calibration files to be on the host disk.

****

ACTIVATING THE AUXILIARY DATA ACQUISITION SYSTEM

To activate the DAS for data to be recorded along with LCS data, you need only to either calibrate the DAS or manually download the scan list prior to pressing the **RUN** button for that test. Calibration or manual downloading may be done before or after the test setup is downloaded to the TMTM. Once the RUN button is pressed, the test "locks in" pertinent data (i.e. the total number of DAS channels defined for this test) needed to write to the test data file on the hard disk.

If DAS is activated after the system has been set to **RUN**, the DAS data will be viewable but will not be recorded to the test data file. Only the LCS data will be recorded to the file in this case.

If the total number of DAS channels is increased in mid-test and the DAS is recalibrated, the additional channels will not be recorded. Only the total number of channels as "locked in" after the initial pressing of the RUN button will be recorded.

RETRIEVING RECORDED ADAS DATA

Data from the AuxDAS is attached to the LCS Test Status data (i.e., Test Data display/report). Therefore, it may be stored to disc, viewed, and printed in the same manner as the standard (LCS) Test Data display/report. Rosette data, Computed Channel data, Pressure data, and/or Displacement data are additional appendages to the Test Data display/report which your particular system may or may not include. Chapter 11 provides additional detail on retrieving AuxDAS data.

??
BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

CHAPTER 9

ESTABLISHING PRE-RUN & RUN-TIME CONDITIONS

There are several test conditions which you may want to establish prior to initiating the loading process. These include:

- resetting any error conditions which may have occurred prior to or after the test was downloaded
- enabling/disabling the Conditional Action Tables for systems configured with this feature
- simulating channel faults for the purpose of verifying programmed conditional actions
- enabling or disabling certain run-time functions, such as auto-tuning, endpoint pacing, inner error pacing, etc.
- resetting individual discrete outputs
- scaling the preprogrammed transition time (rate) and/or load for all control channels

This chapter describes each of these conditions separately.

??
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

RESETTING ERROR CONDITIONS

Before a test can be initiated or resumed, you must reset any error conditions which may have occurred. Error conditions can occur several ways, including:

- when the test setup information is downloaded to the TMTM/servos
- when the hydraulic simulators within the servo controllers are enabled/disabled
- when out-of-limit conditions are exceeded during normal operation of the system (i.e. outer error, ñouter limits)
- when simulated fault conditions are activated (see next section in this chapter)

Follow these steps to reset error conditions:

Step 1: Select **MASTER** from the **CONTROL** menu. The Master screen shown in **Figure 9-1** is displayed.

Step 2: Move the cursor to the Reset Error Conditions position and press **RETURN**. The TMTM sends a message to all servos in the test to reset the latched error status, which results in the following:

- The associated lights on the servo front panel(s) are extinguished.
- The error status in the TMTM is reset.
-The number(s) in the First Fault Ch# boxes on the Master screen are removed (i.e. if the error condition was linked to a Channel Conditional Action).

******************************** NOTE ******************************
The error condition must be removed before the error can be reset. For example, if an "outer limit" or "outer loop error" is still present at the time a Reset Error Conditions is initiated, the servo(s) will immediately detect another error condition (e.g. outer limit, outer error). The same applies to discrete input states which have been linked to preprogrammed actions.
****

In addition, Reset Error Conditions works in conjunction with the following:

- Freeze History Data. Selecting Reset Error Conditions unfreezes the history data buffer and allows new data to be stored.
- Simulated Faults. This option resets the simulated faults set by Set Fault #1 and Set Fault #2.
- Rearms the RUN button for test resumption after latching faults (e.g. Dump)

Step 3: Press the **[ESCAPE]** key **(F11)** to exit the Master screen.

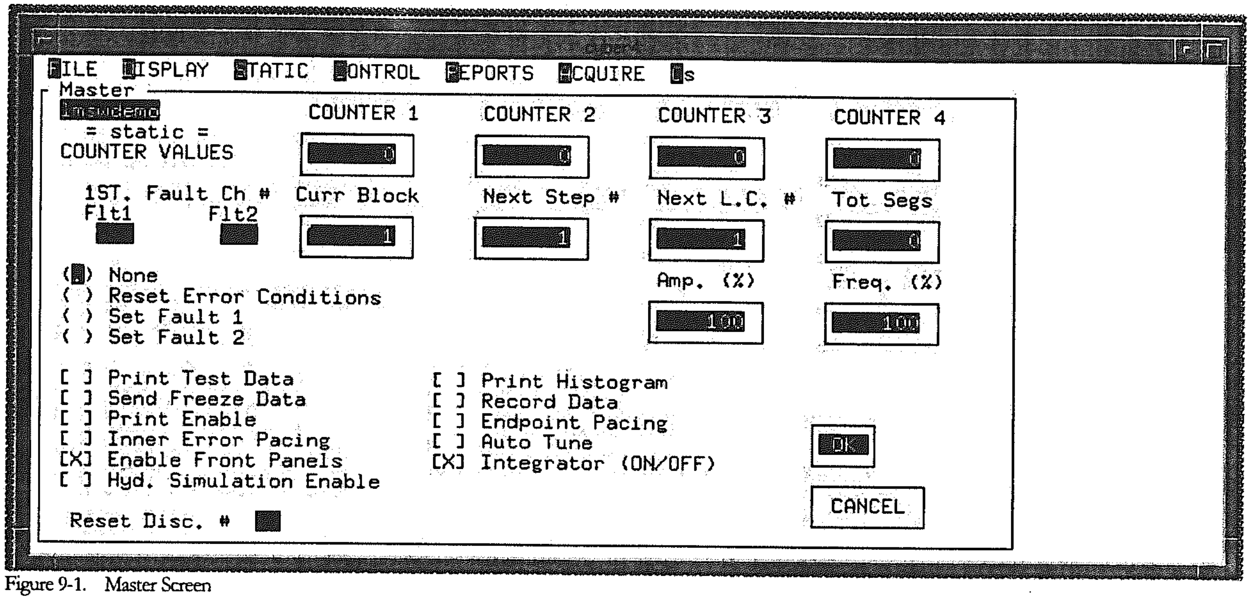


Figure 9-1. Master Screen

ENABLING/DISABLING THE CONDITIONAL ACTION TABLES

Some systems are equipped with a special feature which allows you to enable/disable the Conditional Action Tables prior to, or during, a test run. With such system configurations, the Conditional Action Tables are disabled until the test has been downloaded and the appropriate AUX button on the test control panel has been activated.

This feature avoids unnecessarily triggering programmed actions which may occur due to pre-existing servo error conditions which are present upon downloading the test information to the TMTM & servos. It also allows you to disable the Conditional Action Tables:

- when you wish to allow the system to exceed previously set limits without performing any programmed conditional actions
- when you run the test in simulation and don't want the system to perform conditional actions

Refer to Appendix A to determine if your system includes this feature. Appendix A will also indicate which AUX button (AUX#1 or AUX#2) on the test control panel is dedicated as the Limit Enable button.

************************************ NOTE ************************************

If you do not enable the Conditional Action Tables from the appropriate AUX button, no programmed actions will be taken for any conditions listed in the Conditional Action Tables. However, if a fault condition occurs and you manually pause or hold the test some time after, the next press of the RUN button is disarmed until you Reset Error Conditions.

****

SETTING FAULT CONDITIONS

In order to verify the system response to each of the servo's fault conditions (i.e. Fault #1 and Fault #2), the system includes a means of simulating these faults, even though no servo error conditions actually exist.

Follow the steps given below to set fault conditions:

Step 1: Select **MASTER** from the **CONTROL** menu. The Master screen shown in **Figure 9-1** is displayed.

Step 2: If your system configuration allows you to enable/disable the Conditional Action Table, as described in the previous section, press the **AUX#1** button to enable this function.

Step 3: If you wish to simulate a **Fault#1** condition, move the cursor to the **Set Fault 1** position and press [**RETURN**]. The system issues a command to all servos in the test which causes each servo to set Fault #1. This fault condition is identified by the TMTM, causing the system to perform any programmed actions that have been linked to a Fault#1 condition.

Step 4: If you wish to simulate a **Fault#2** condition, move the cursor to the **Set Fault 2** position and press [**RETURN**]. The system issues a command to all servos in the test which causes each servo to set Fault #2. This fault condition is identified by the TMTM, causing the system to perform any programmed actions that have been linked to a Fault#2 condition.

Step 5: Press [**ESCAPE**] to exit the master screen.

ENABLING/DISABLING RUN-TIME FUNCTIONS

Prior to initiating a test, or while a test is running, you may enable/disable various run-time functions which control how the test will be conducted.

Follow the steps given below to enable/disable run-time functions:

Step 1: Select **MASTER** from the **CONTROL** menu. The Master screen shown is displayed.

Step 2: Enabling/disabling of each run-time function requires you to select or de-select the appropriate check-box. A run-time function is enabled when an "X" appears in the square brackets next to the function. To disable a function, move the cursor to the corresponding brackets and press the space bar to erase the "X". To re-enable the function, press the space bar again: the "X" reappears.

******************************* [redacted] **********************************

Some of the available functions are intended to be used "as required" during the test and should not be enabled prior to pressing RUN. Some functions activate modes of operation or global features which may be enabled prior to or after pressing RUN. Lastly, some functions, namely one -Send Freeze Data, is intended to be enabled only after other events have occurred.

****

Step 3: Select one or more of the following functions, if desired:

- **Print Test Data** sends a Test Data Report of the next endpoint to the printer. This is a "one-shot" function; it is automatically disabled after the report is printed for the next endpoint, i.e. the "**X**" goes away when you press **[RETURN]**. You must re-enable the function whenever you want a Test Data Report for an upcoming endpoint. This function is typically enabled during the test, not prior to pressing RUN. You may also select this function while Puased at an endpoint to obtain that endpoint's data.
- **Print Histogram** sends tabular histogram data at the next endpoint to the printer. This is a "one-shot" function; it is automatically disabled after the report is printed for the next endpoint, i.e. the "**X**" goes away when you press **[RETURN]**. You must re-enable the function whenever you want a histogram report at the next endpoint. This function is typically enabled during the test, not prior to pressing **RUN**.
- **Send Freeze Data** prompts selected servo channels to transfer their circulating memory contents to the host's hard disk, one at a time. You can activate this function if the system is in the **RUN** mode. Refer to Chapter 14 for more information on retrieving/examining the history data.
- **Record Data** stores the endpoint Test Data on the system hard disk at the completion of the current endpoint. This is a "one-shot" function; it is automatically disabled after the endpoint data is record, i.e. the "**X**" goes away when you press **[RETURN]**. You must re-enable the function whenever you want to store the endpoint Test Data for upcoming endpoints. This function is typically used for endpoints which have not been set up to be recorded in a fatigue test's test procedure file. This function is typically enabled during the test, not prior to pressing **RUN**.
- **Print Enable** allows the automatic printing of the Test Data Report at steps within the test procedure which call for a printout. This is a latching function; you may enable or disable it only from the Master screen. This function can be enabled before or after pressing **RUN**.
- **Endpoint Pacing** forces the system to wait for all channels to reach the correct endpoint load (within the tolerance specified on the Start Up screen) before continuing to the next step. The length of time the system waits is specified in the System Conditional Actions screen, discussed in Chapter 5. Endpoint pacing is a latching function; you may enable or disable it only from the Master screen. This function can be enabled before or after pressing **RUN**.
- **Inner Error Pacing** prevents one or more control channels from lagging behind the others. The system accomplishes this by monitoring the inner error signal of each control channel in the test. When an inner error limit has been exceeded by one or more channels, the system halts the command function generation for all channels. When the channel(s) are within their allowable inner error limit, the system resumes command function generation. The length of time that the system allows the inner error limit to be exceeded is specified in the System Conditional Actions screen, discussed in Chapter 5. Inner error pacing is a latching function; you may enable or disable it only from the Master screen. This function can be enabled before or after pressing **RUN**.

Step 3 (Cont):

- **Auto Tune** enables the automatic tuning algorithm in the servo controllers. Enabling this function means that it is operational only on those channels for which the auto-tuning function is programmed on the Channel Tuning screen, as discussed in Chapter 3. Auto-tuning is a latching function; you may enable or disable it only from the Master screen. This function can be enabled before or after pressing **RUN**.

- **Hyd. Simulation Enable** turns on the hydraulic simulator in each servo controller. This function can be enabled only when the actual system hydraulics are off (i.e. the Ext Hyd Press LED on the Test Control Panel is NOT illuminated). It is a latching function, and may disabled from the Master screen.
- **Enable Front Panels** enables the controls on the servo front panels and the Master Rate/Freq controls on the test control panel. If disabled, the system allows the test operator to monitor servo parameters from the front panel of the servo controller(s), without the ability to change the values of the servo parameters from the front panel. The same applies to the Master Rate/Freq controls on the test control panel. This is a latching function; you may enable/disable it only from the Master screen. This function can be enabled before or after pressing **RUN**.
- **Integrator** (**ON/OFF**) controls the integrator within each servo controller. Enabling this function means that it is operational only on those channels for which the integrator was selected on the Channel Tuning screen, discussed in Chapter 3. This is a latching function; you may enable or disable it only from the Master screen. This function can be enabled before or after pressing **RUN**.

******************************** NOTE ********************************

Integrators will turn on only if hydraulics (i.e. the Ext Hyd Press LED is illuminated) are on. If hydraulics are turned off with integrators enabled, all integrators will be disabled, and must be re-enabled as described above. The X' in the checkbox indicates the last commanded state of the integrators. If integrators are shut off due to deactivation of hydraulics (this occurs within each servo channel), the state of this checkbox does not change. Actual integrator status is viewable on the Channel Status display which provides a real-time update of the settings and data for that channel. The operator must re-command integrators on (by pressing RETURN on the Master Screen) upon reactivation of hydraulics for the integrators to come back on. Once the Ext Hyd Press LED is illuminated, the integrators will take on the state of the integrator checkbox (subject to Channel Tuning entries) whenever RETURN is pressed on the Master Screen ([X] = ON, [ ] = OFF).

****

Step 4: When you finish setting run-time parameters, press [**RETURN**] to execute your enable/disable selections.

Step 5: Press [**ESCAPE**] to return to the **CONTROL** menu.

RESETTING INDIVIDUAL DISCRETE OUTPUTS

Individual discrete outputs on the rear of the discrete I/O panel may be reset from the Master screen. Using the input box label **RESET DISC. #** at the lower left of the Master screen, enter the number of the discrete output you wish to reset. When you press return, the system resets the Form-C relay associated with the selected discrete output.

ADJUSTING THE MASTER AMPLITUDE & RATE CONTROLS

It is sometimes helpful to have the master amplitude and/or frequency of the test be at lower levels prior to pressing RUN. This allows the hydraulics actuators to come up to load and speed gradually. In order to do this, the FM7000 includes Master controls on the test control panel which to allow you adjust the rate and amplitude of the test prior to pressing RUN and/or while the test is running.

Changing the Amplitude of the Test

Using the "**Master AMP**" control on the test control panel, you can simultaneously vary the command for all channels from 1% to 100% of their individual preprogrammed values. The Master screen display the selected setting in the block labeled **AMP(%).** See note below.

Changing the Frequency of the Test

Using the master "**FREQ**" control on the Test Control Panel, you can vary the speed of the test from 1% to 200% of the preprogrammed transition times. The Master screen displays the selected setting in the block labeled **RATE(%).** See note below.

********************************** NOTE ***************************************

If the master Amplitude or Frequency settings are changed while the test is transitioning from one load condition endpoint to the next, the new settings will not be applied until the present transition is completed. If "Enable Front Panels" on the Master Screen is not selected, the displays will show "**LOCKED**" when either rocker switch is pressed.

****

??

BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.

??

CHAPTER 10

INITIATING THE LOADING PROCESS

If you have performed the necessary steps given in the preceding chapters, you are ready to initiate the loading process. The following sections provide instruction on how to activate the hydraulic subsystem and control the principle test functions using the Test Control Panel.

If you have enabled the "**Hyd. Simulation Enable**" selection on the Master screen and wish to run your test under hydraulic simulation, skip the next two sections below on activating hydraulics and begin reading from the section labeled Running the Test.

??
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

ACTIVATING HYDRAULICS

The hydraulic subsystem may be activated in one of two ways. The first method involves the hydraulic system controls on the Test Control Panel. These controls are used to activate relays on the discrete I/O panel, which, if hydraulic control is wired through the Test Control Panel, will turn on the hydraulic subsystem. For these buttons to be active, the box labeled "**Arm Hydraulics**" on the Startup Screen must contain an X' prior to downloading the test.

The second method is used when hydraulic control is not wired to the Test Control Panel hydraulic buttons. In this case, you must turn on the hydraulics using the appropriate external procedure.

In either case, to inform the system that hydraulic pressure is on, a pressure switch, or equivalent, must be wired into discrete input #1. When a low (i.e. shorted) input is at discrete input #1 (some systems use opposite state), the system knows that hydraulics are on and indicates so by illuminating the "Ext Hyd Press" LED on the appropriate Test Control Panel.

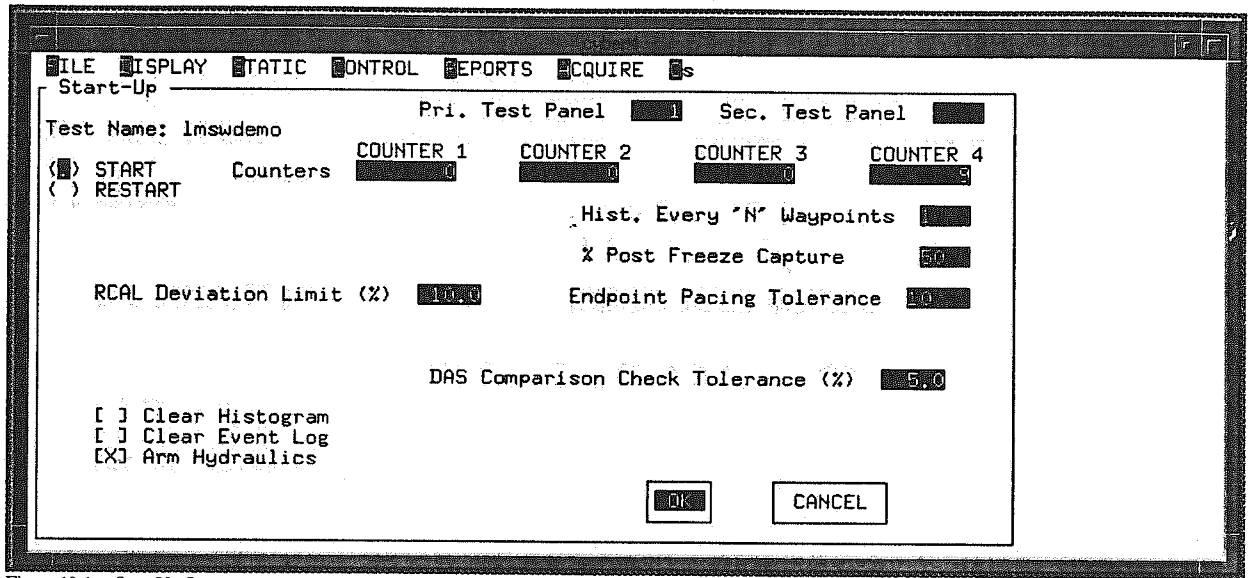


Figure 10-1 Start-Up Screen

In the "**Hydraulics On**" state, the following conditions and interlocks exist:

- The **'Ext Hyd Press'** LED on the Test Control Panel is illuminated.
- **Bridge Balance** and **R-CAL** procedures for the control channels can not be performed.
- Integrators are enabled to be turned on.
- **Valve Balance** and **Pre-Tune** procedures may be performed.
- Built-in hydraulic simulators are disabled; if they were on, they will be turned off when the system is told that hydraulics are on and will be disallowed from coming on as long as hydraulics are on.

USING THE TEST CONTROL PANEL BUTTONS

If hydraulic control is wired to the Test Control Panel, follow these steps to activate the hydraulic subsystem:

Step 1: Press the **ON** button on the Test Control Panel. This would typically be wired to hydraulic manifold and used to enable or apply pressure to that manifold.

Step 2: Select low hydraulic pressure (e.g. 600 psi) by pressing the **LOW** button. This assumes that your manifold has a "low pressure" input with appropriate pressure regulation to apply low pressure to the test fixture hydraulics.

Step 3: Select high hydraulic pressure (e.g. 3,000 psi) by pressing the **HIGH** button. This enables full hydraulic pressure to be applied to the test fixture.

Once hydraulics are on, you can proceed with the execution of your test. The following sections describe each of the test control buttons located on the Test Control Panel and how to use them to accommodate your specific test requirements.

RUNNING THE TEST (RUN BUTTON)

Depending on how you want to execute your test procedure, your next step may be to activate the **RUN** button. However, when the **RUN** button is pressed, the system will proceed to execute the test procedure, as previously defined. For instance, if your test procedure includes 11 steps, each specifying increasing increments of load (e.g. 0% 10%, 20%, 30% . . . 100%), pressing the **RUN** button will initiate a continuous loading procedure without any further intervention on your part. This is commonly referred to as normal operation or the "run" mode.

Since the **RUN** button is an alternate-action switch, pressing the **RUN** button while a test is running will hold the test instantaneously, without completing the current endpoint transition. This is known as the "**HOLD**" mode. Pressing **RUN** again will cause the system to resume the test from the point at which it was held. The Hold function is described in subsequent sections.

PAUSING THE TEST (PAUSE BUTTON)

Once you have pressed the **RUN** button and the test is running, should you want to pause the test at completion of the current endpoint transition, press the **PAUSE** button. This function latches; the system completes the present load transition and holds at that endpoint. If you press the **RUN** button while the PAUSE function is still active, the test executes the next step and holds at the following endpoint. Repetitive depression of the **RUN** button allows you to incrementally step through the predefined test procedure. This procedure is commonly referred to as operating in "**PAUSE**", or "**Single Step**" mode and is typically used in static loading applications.

In addition to push-button activation of the pause mode, the system can be placed in the pause mode through a programmed pause command defined within the test procedure (refer to Chapter 4)

To resume normal test operation from a manually-activated pause mode, press **PAUSE** again (to unlatch the pause function), then press **RUN**. The test is placed in the run mode and resumes normal operation. To resume from a programmed pause mode, simply press **RUN**.

HALTING THE TEST (HOLD BUTTON)

To halt the execution of a test which is underway, press the **RUN** button on the Test Control Panel. The test execution instantaneously stops at the current waypoint within the transition, and the **HOLD** light is illuminated. The system will not complete the transition until it has been taken out of the "**HOLD**" mode. To resume testing, press the **RUN** button again.

The test may also halt as a result of a programmed conditional action **HOLD**. To restart the test after a programmed conditional action **HOLD**, press the **RUN** button.

RAMPING THE TEST **(RAMP-TO BUTTONS)**

Cyber's Test Control Panel provides two user-definable ramp-to buttons; **RAMP-TO COND 1** and **RAMP-TO-COND 2**. These buttons perform different functions depending on whether or not the test is defined as a static test or as a fatigue test.

For tests defined as static, the **RAMP** buttons are used to branch to (i.e. go to) predefined steps in your test procedure. The section labeled ***Defining Ramps for Static Tests*** in Chapter 5 describes how to program each **RAMP** button for use in static tests.

For tests defined as fatigue, each **RAMP** button operates in conjunction with a preprogrammed load condition. The section labeled ***Defining Special Load Conditions for Fatigue Tests*** in Chapter 5 describes how to program each **RAMP** button for use in fatigue tests.

In both cases, pressing either **RAMP** button will command all control channels in the test to their respective preprogrammed step/load condition.

ENABLING AUXILIARY FUNCTIONS (AUX 1 & AUX 2 BUTTONS)

The **AUX** buttons on the test control panel are reserved for special functions. Some of these functions include **Limit Enable** and **Reverse**. Each function is described separately below.

Limit Enable Button

If your system is configured with this feature, the system allows you to enable/disable the Conditional Action Tables prior to, or during, a test run. With such system configurations, the Conditional Action Tables are disabled until the test has been downloaded and the appropriate AUX button on the test control panel has been activated. This feature avoids unnecessarily triggering programmed actions which may occur due to error conditions created upon downloading the test information to the TMTM & servos..

If hydraulics are on (i.e. Ext Hyd Press LED illuminated) and Limit Enable is active, it may not be turned off. Only when hydraulics go off will you be allowed to turn off **Limit Enable**.

To determine if your system includes this feature and which **AUX** button to use, refer to Appendix A.

Reverse Button

If your system is configured the **Reverse** feature, the system allows you to reverse the direction that the system proceeds through the test profile. One of the **AUX** buttons on the test control panel is used for this purpose. Refer to Appendix A to see which one. This button is only functional with tests defined as **Static**.

Though this feature may be used as required, the main reason this capability was added to the system was to allow unload a static test article using the same increments which were used to load it.

If you are presently running a test and wish to reverse the loading direction, the system must first be paused at an endpoint or held between endpoints. Once held or paused, press the **REVERSE** button. Pressing **RUN** causes the system to execute the test procedure in reverse order, starting from the current step in the profile. If you wish to reverse through the profile, pausing at each endpoint, press the **PAUSE** button. If not (i.e. the system is in **RUN** mode), the **REVERSE** button press is ignored.

You may take the system out of reverse mode at any time, as long as the system is held or paused. Press the **PAUSE** button (or **RUN[HOLD]**), then press the **REVERSE** button. The system is now back in the normal mode (i.e. forward loading). Pressing the **RUN** button will cause the system to execute the test procedure from the current step in the profile.

[redacted]The **"Rpt to Blk #"** and **"Inc Counter"** columns of the Test Procedure are ignored when operating in reverse.

ACTIVATING A HYDRAULIC SHUTDOWN (EMERGENCY STOP BUTTON)

The Test Control Panel has an **EMERGENCY STOP** button which is typically wired to the lab's emergency shutdown circuitry. When this button is pressed, the FM7000 performs a Dump. When this occurs, all servo controllers ramp to zero, the dump relay triggers, and a dedicated hardwired contact (tied to the button) is activated. The **DUMP** contact is intended to be the emergency shutoff mechanism for the hydraulic system.

TEST CONTROL PANEL INDICATORS

The list below describes what each light on the test control panel indicates when illuminated:

HOLD	indicates system has been placed in **Hold** state due to pressing of the **RUN/HOLD** button or via a programmed action
ENDPT	indicates system is at an endpoint
ERROR PACING	indicates that the system is presently performing the inner error pacing and/or endpoint pacing functions
DSCRT SYNC	indicates that the system waiting for one or more discrete inputs to achieve their expected state within the discrete sync time specified on the System Conditional Action Table.
PROG PAUSE	indicates that the system has been placed in a paused state via a programmed pause in the test procedure
VALVE CABLE	indicates that the valve cable to one or more control channels in the test has been disconnected with hydraulics on. Checking the Events Log (Chapter 14) will reveal which channel(s) have valve cable problems.
COMM ERROR	indicates that the system has experienced one or more communication problems with one or more servo channels on the SDLC link
FAULT	indicates that the system has experienced a **Fault #1** or **Fault #2** condition. Checking the Events Log (Chapter 14) will reveal which fault occurred and any actions which may have been taken.
HYD TEMP	if wired appropriately, this light indicates that an external customer-supplied, temperature sensor has detected that the hydraulic oil temperature is high
EXT HYD PRESS	if wired appropriately, this light indicates that hydraulic pressure is active
DUMP	indicates that the system has performed a dump procedure, either due to a programmed conditional action, embedded conditional action such as a communication error, or manual activation via the Emergency Stop button.

??
BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

CHAPTER 11

ON-LINE MONITORING OF ALPHANUMERIC DATA

While the test is underway, the Fatigue Master 7000 allows you to monitor its progress in three different ways:

- You can follow the progress of the test by viewing the real-time display on the Master screen. In addition, you can use the Master screen to enable or disable certain run-time test functions "on the fly" while the test is underway.
- You can view a variety of tabular displays that show the present state of the control channels, DAS channels and other types of test data.
- If your system is configured with Cyber's Real-Time Graphics Subsystem, you also have the option to monitor the test using various graphic displays. Refer to Chapter 12 to learn how to setup and use the graphic displays with your test.

??
BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

VIEWING THE MASTER SCREEN

Access the Master screen by selecting the **Master** from the **CONTROL** menu.

The Master screen shown in **Figure 11-1** has three primary functions:

- You monitor this screen during test execution to follow the progress of the test and to retrieve data from the test. It's helpful to leave this screen displayed while the test is running as it gives the operator an overview of which part of the test procedure is being executed.
- You use this screen to view the state of, and/or enable/disable certain run-time functions.
- You use this screen to identify which control channel(s) were first to be polled with fault conditions.

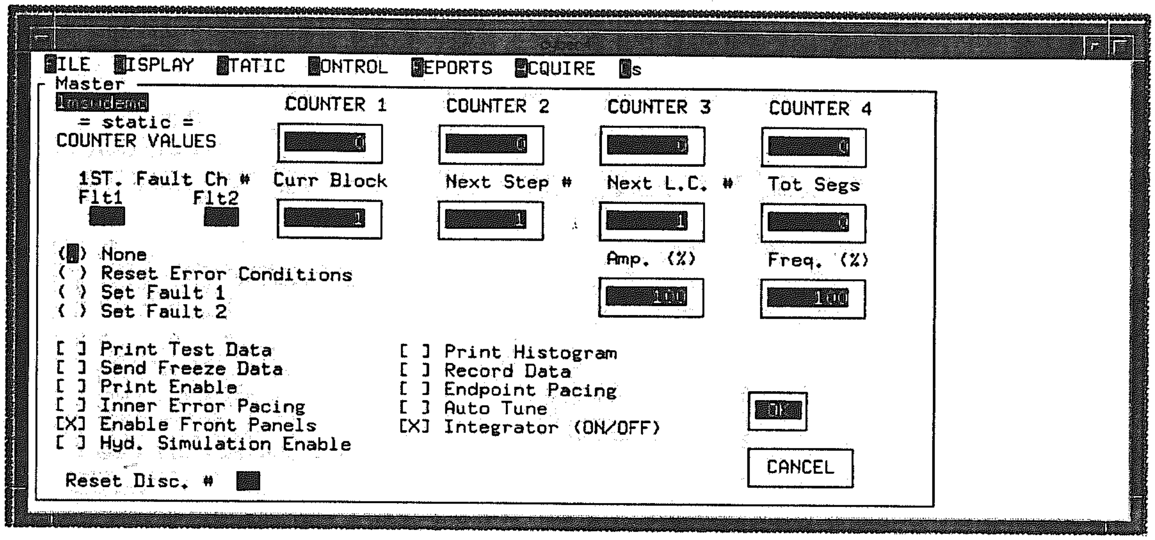


Figure 11-1. Master Screen

The Master screen allows you to view various test related data including:

- The current values of Counters #1 through #4
- The current block and step number
- The current load condition being executed
- The physical channel number of the first detected servo controller to initiate a fault#1 condition or fault#2 condition
- The total number of executed segments
- The master rate and amplitude scaling factors
- The present state of various run-time functions such as auto-tuning, endpoint pacing, etc.. You may also enable/disable the state of these run-time functions while the test is underway. See Chapter 9 for more details.

VIEWING TABULAR TEST DISPLAYS

The Fatigue Master 7000 provides extensive data monitoring through various on-line displays, each of which may be accessed from the **DISPLAY** menu. Each display is described separately below:

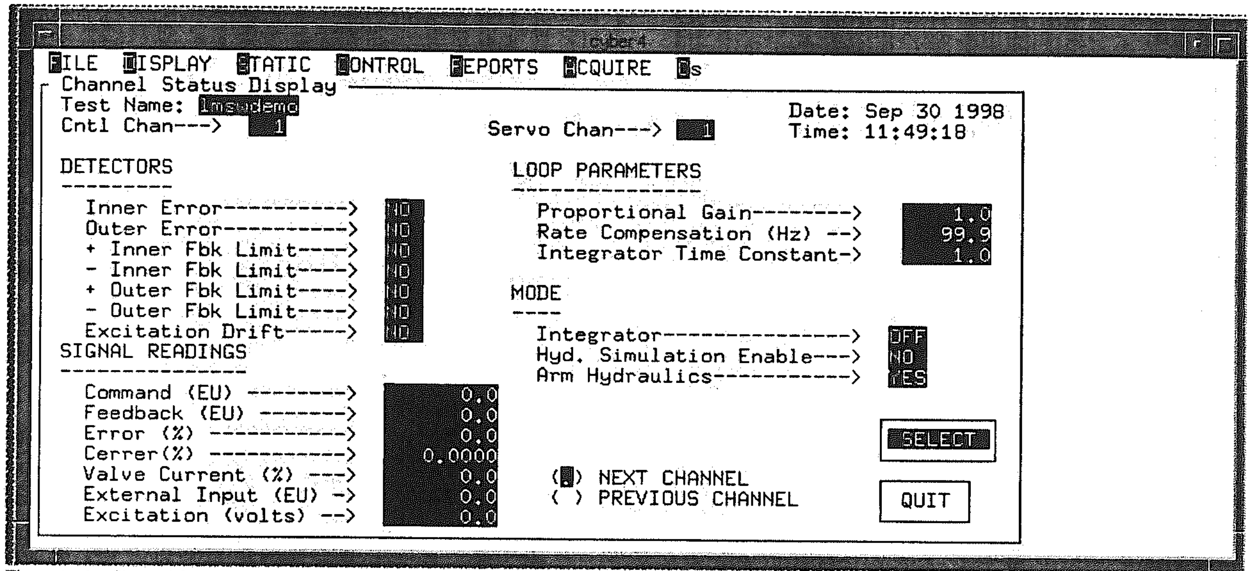


Figure 11-2. Channel Status Display

Channel Status Display

This screen provides real-time display (i.e. updated twice per second) of the loop parameters, tuning parameters, error/limit detector status, and operating modes (e.g. hydraulics, integrators, simulators, etc.) for a selected control channel.

Follow the steps given below to view the Channel Status display:

Step 1: Select **Channel Status** from the **DISPLAY** menu. **Figure 11-2** provides an example of a typical Channel Status display.

Step 2: You may view Channel Status information for any channel in the test by **selecting "Next Channel"** or **"Previous Channel"** and pressing [**RETURN**].

Step 3: Press the **ESCAPE** key (**F11**) or select **QUIT** to exit the Channel Status screen.

TEST DATA DISPLAYS

Depending on your particular system configuration, there are several Test Data displays which allows you to view various types of test data available with the FM7000, including LCS data, DAS data, Rosette data, Computed Channel data, Displacement Xducer data and Pressure data. Some of these displays may not be available with your particular system software.

Unlike the Channel Status display, which only provides information for one control channel at a time, Test Data displays provide test information for multiple channels, simultaneously. Each type of data is displayed on a separate Test Data display screen, all of which clearly display the test name, current date and time. The following sections describe each Test Data display separately.

LCS Test Data Display

The **LCS Test Data** Display provides a listing of tabular data for all control channels in the test. The rate at which this display is updated is dependent upon how the test was defined when it was created. If the test is defined as "**static**", the **LCS Test Data** display is updated on a continuous basis (2 times per second). If the test is defined as a "fatigue" test, this display is updated at each endpoint in the test procedure. With static tests or fatigue tests, the **LCS Test Data** display contains the following information:

- The current values of the Counter #1 through #4
- The total number of segments executed during the test
- The servo channel number and engineering units are also displayed for each control channel.
- Command, feedback, and % error data for up to 9 channels is displayed per page.
- Follow the steps given below to view the LCS Test Data Display:

Step 1: Select Test Data from the **DISPLAY** menu.

Step 2: Select the LCS radio button (which is the default selection), and press [**RETURN**]. The screen shown in Figure 11-3 is displayed:

Step 3: You may view information for any control channel in the test by selecting "Next Channel" or "Previous Channel" and pressing [**RETURN**].

Step 4: To obtain a hardcopy, select **PRINT** and press [**RETURN**].

Step 5: Press [**ESCAPE**] or select **QUIT** to exit the **LCS Test Data** screen.

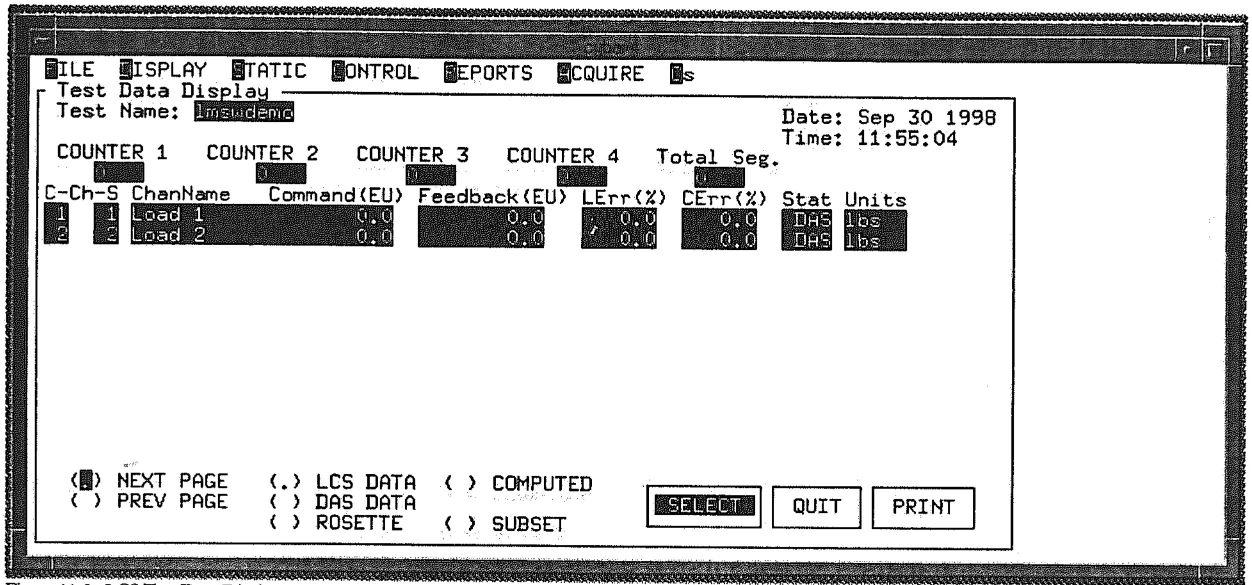


Figure 11-3. LCS Test Data Display

DAS Test Data Display

If your system is configured with DAS, you may also obtain a Test Data display for DAS data. Unlike the update rate of the LCS Test Data display, which is dependent on the test type (fatigue or static), the **DAS Test Data** display is always updated twice a second, regardless of the test type. However, when recording is enabled (via test procedure), recording of the DAS data is only done at endpoints.

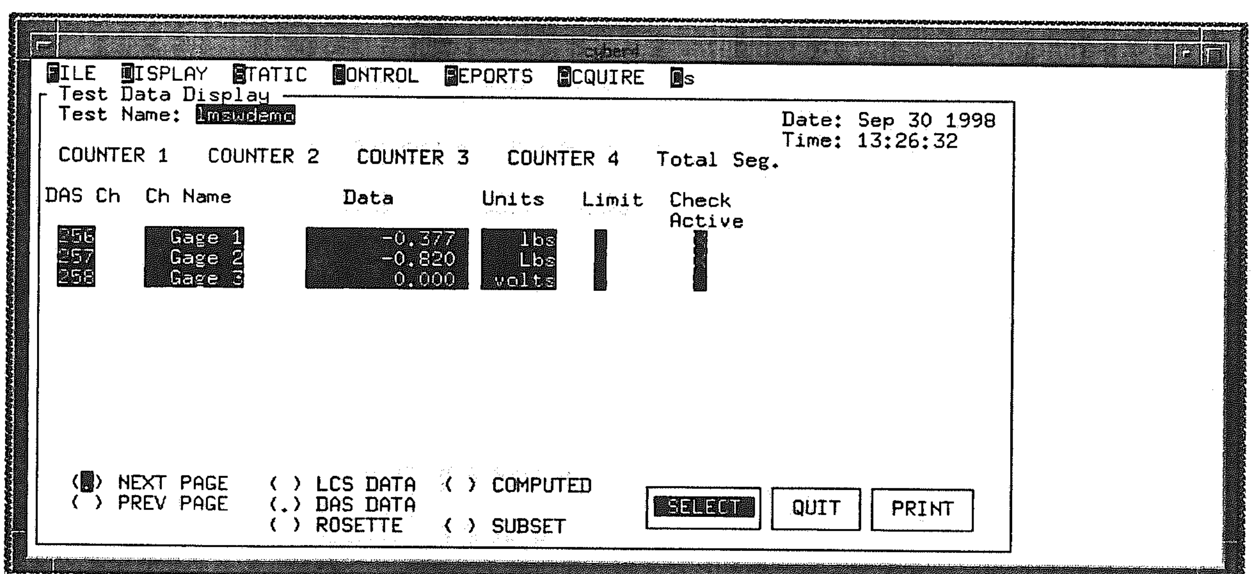


Figure 11-4. DAS Test Data Display

DAS Test Data display includes the following information for each DAS channel currently displayed:

- The channel name, as defined on the ACQUIRE Set Up screen
- The channel data and engineering units
- The limit status. An 'H' in this column indicates that data for that channel exceeds the upper limit previously defined in the ACQUIRE Set Up screen. Similarly, if an 'L' is displayed, data is less than the lower limit previously defined. If the data is within range, this column is blank.

Follow the steps given below to view the DAS Test Data display:

*********************************** NOTE ************************************

The following steps require you to have previously calibrated the DAS channels and downloaded the setup information from the Start Up screen.

****

Step 1: Select **Test Data** from the **DISPLAY** menu.

Step 2: Select the **DAS** radio button and press [**RETURN**]. The screen shown in **Figure 11-4** is displayed:

Step 3: You may view Test Status information for any DAS channel in the test by selecting "**Next Channel**" or "**Previous Channel**" and pressing [**RETURN**].

Step 4: If you wish to obtain a hardcopy of the data presently displayed, using the **TAB** key, select **PRINT** and press [**RETURN**].

Step 5: Press [**ESCAPE**] or select **QUIT** to exit the **DAS Test Data** screen.

Rosette Test Data Display

If your system is configured with Cyber's Auxiliary Data Acquisition Subsystem and you have defined one or more **ROSETTE CHANNELS** (refer to Chapter 8), you may view the results of these rosette channels in the form of a Rosette Test Data display.

For each rosette defined during setup, the FM7000 software computes a series of parameters based on reading each strain gauge at endpoint, and directs the results to the Test Data display/report.

The parameters computed are:

- Maximum and minimum strain
- The angle from gauge 1 axis to the maximum normal stress axis
- Maximum and minimum stress
- Shear stress

The data in the Rosette Test Data display is updated on a continuous basis, two times per second, regardless of the test type (static or fatigue).

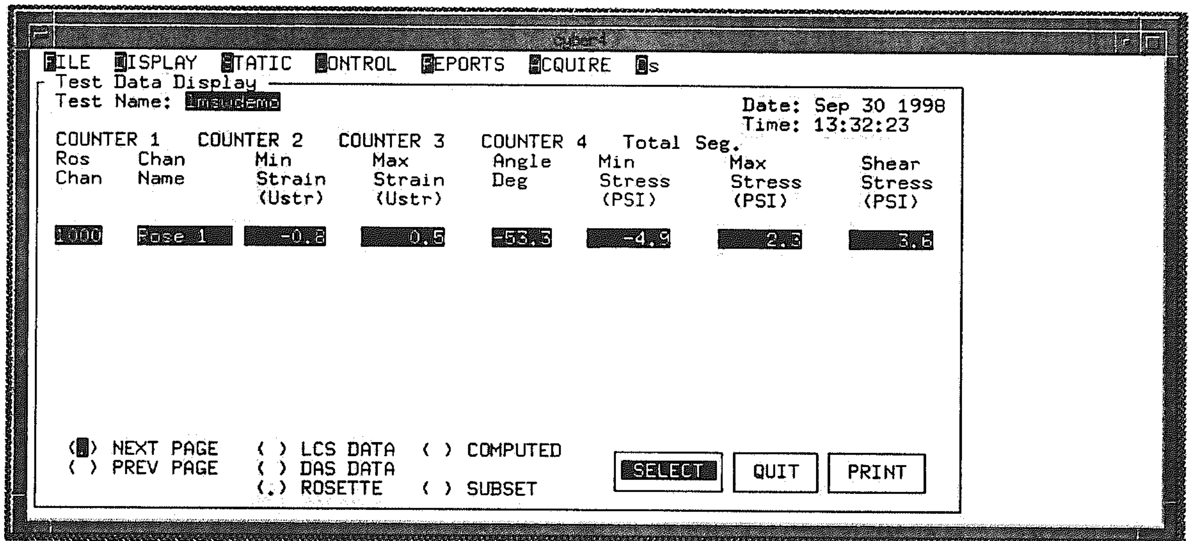


Figure 11-5. Rosette Test Data Display

Follow the steps given below to view the Rosette Test Data display:

*********************************** [redacted] *************************************

The following steps require you to have previously calibrated the DAS channels and downloaded the setup information from the Start Up screen.

****

Step 1: Select **Test Data** from the **DISPLAY** menu.

Step 2: Select the **ROSETTE** radio button and press [**RETURN**]. The screen shown in **Figure 11-5** is displayed.

Step 3: You may view Test Status information for any rosette channel in the test by selecting "**Next Channel**" or "**Previous Channel**" and pressing [**RETURN**].

Step 4: If you wish to obtain a hardcopy of the data presently displayed, using the **TAB** key, select **PRINT** and press [**RETURN**].

Step 5: Press [**ESCAPE**] or select **QUIT** to exit the **Rosette Test Data** screen.

Computed Channel Test Data Display

If your system is configured with Cyber's Auxiliary Data Acquisition Subsystem and you have defined one or more **COMPUTED CHANNELS** (refer to Chapter 8), you may view the results of these computed channels from the Computed Channels Test Data display. This display is updated on a regular basis, twice per second, for both static and fatigue type tests.

Follow the steps given below to view the Computed Channel display:

*********************************** NOTE ************************************

The following steps require you to have previously calibrated the DAS channels and downloaded the setup information from the Start Up screen.

****

Step 1: Select Test Data from the **DISPLAY** menu.

Step 2: Select the **COMPUTED** radio button and press **[RETURN]**. The screen shown in Figure 11-6 is displayed.

Step 3: You may view Test Status information for any computed channel in the test by selecting "Next Channel" or "Previous Channel" and pressing **[RETURN]**.

Step 4: If you wish to obtain a hardcopy of the data presently displayed, select **PRINT** and press **[RETURN]**.

Step 5: Press **[ESCAPE]** or select **QUIT** to exit the COMPUTED Test Data screen.

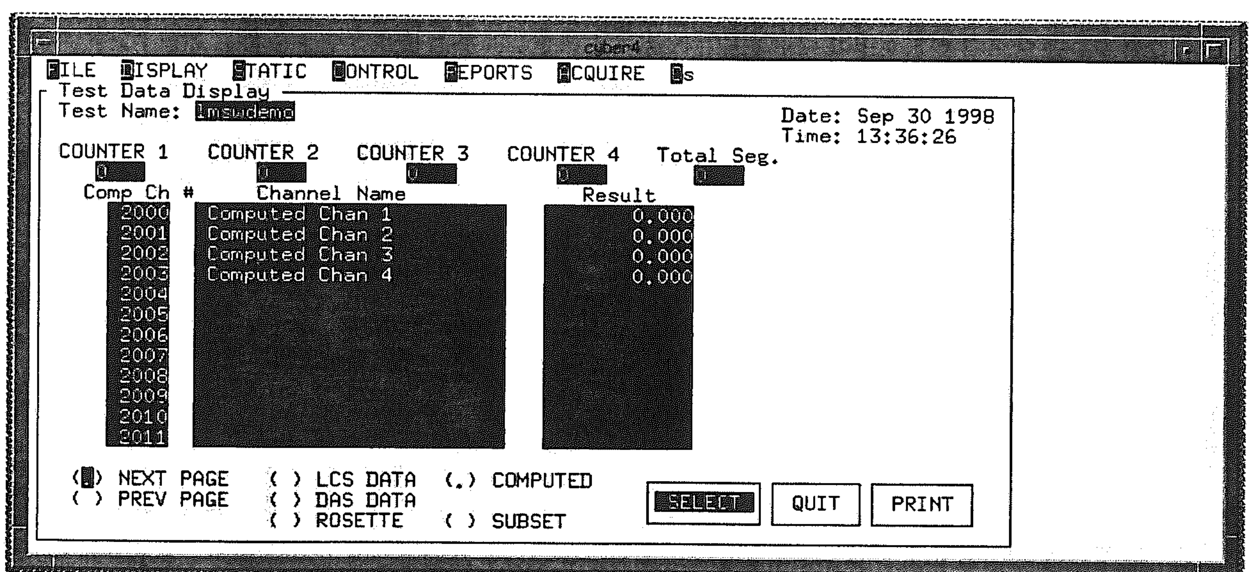


Figure 11-6. Computed Channel Test Data Display

SUBSET Test Data Display

If your system is configured with Cyber's Auxiliary Data Acquisition Subsystem and you have previously defined a subset group of DAS channels on the **ACQUIRE** Set Up screen (refer to Chapter 8), you may view a special version of Test Data display. This display is known as the **SUBSET** Test Data display. DAS channels in this display are in the order of subset numbering in the "Subset" column of DAS Setup. The data in this display is always updated on a regular basis, twice per second, for both static and fatigue type tests.

Follow the steps given below to view the Subset Test Data Display:

*********************************** NOTE ************************************

The following steps require you to have previously calibrated the DAS channels and downloaded the setup information from the Start Up screen.

****

Step 1: Select **Test Data** from the **DISPLAY** menu.

Step 2: Select the **SUBSET** radio button and press [**RETURN**].

Step 3: You may view data for any DAS channel in the subset group by selecting "**Next Channel**" or "**Previous Channel**" and pressing [**RETURN**].

Step 4: If you wish to obtain a hardcopy of the data presently displayed, using the **TAB** key, select **PRINT** and press [**RETURN**].

Step 5: Press [**ESCAPE**] or select **QUIT** to exit the **SUBSET** Test Data screen.

PRESSURE Test Data Display

If your system is configured with Cyber's Auxiliary Data Acquisition Subsystem and Cyber's Redundant Load/Pressure Check, you may view the results of this redundant check on the **PRESSURE** Test Data display.

In short, the redundant pressure/load check computes the actuator's load (i.e. force) from data acquired by pressure sensors on either side of the actuator. This pressure data is used in a PcAc -PtAt calculation, where Pc and Ac are the pressure and area of the compression side, respectively, and Pt and At are the pressure and area of the tension side, respectively. The result of this calculation is load (i.e. force).

The redundant pressure/load check compares this "computed load" against the measured load (via the servo's buffered output). If the difference between the computed load and measured load exceeds a user-specified tolerance, as a percent of 100% load, a "redundant check fail" condition is created which may be linked to one or more conditional actions.

Follow the steps given below to view the Pressure Test Data display:

************************************ NOTE ************************************

The following steps require you to have previously calibrated the DAS channels and downloaded the setup information from the Start Up screen. Since the measured load (via the servo's buffered output) is used in the redundant load calculation, the control channels in the test must also be calibrated in order for the results to be correct.

****

Step 1: Select **Test Data** from the **DISPLAY** menu.

Step 2: Select the **PRESSURE** radio button and press [**RETURN**]. The Pressure Test Data screen is displayed.

For each channel used in the redundant load check, the Pressures Test Data display indicates the following items:

- the servo channel number and control channel number
- the measured load, as measured by the ADAS via the servo controller's buffered feedback output.
- the computed load, as computed from the PcAc -PtAt relationship described above.
- the pressure on the tension side of the actuator, as measured by the ADAS.
- the pressure on the compression side of the actuator, as measured by the ADAS.
- the difference (% error) between the measured load and the computed load. Please note that this number is determined by the host computer and is for monitoring purposes only. The actual difference (% error) between the measured load and computed load, which gets used in the redundant check, is determined by the MCU. The MCU makes this comparison multiple times per second. In the event that the actual % error determined by the MCU exceeds the "Redundant Limits" specified on the Start Up screen, the MCU sets a discrete output. These MCU discrete outputs are tied directly to separate discrete inputs on the FM7000 discrete I/O panel. Activation of either of these discrete inputs is interpreted by the system as a "Inner or Outer Redundant Check Fail" condition. These conditions are included in the System Conditional Action table and may be used to trigger one or more preprogrammed actions.
- the status of the redundant check. Below is a list of the various status messages which may appear in this column, and their meaning:

    1. Failed The % error between the measured load and computed load exceeded the "Inner Redundant Limit" specified on the Start Up screen.

NA The channel is not presently (or Not Actively) being used in the redundant pressure/load check, as specified in the AREA screen under the ACQUIRE menu

BlankIf nothing is displayed, this means that the channel is included in the redundant pressure/load check, and has not yet exceeded the "**Inner Redundant Limit**" specified on the **Start Up** screen.

Step 3: You may view data for any channel used in the redundant load check by selecting "Next Channel" or "Previous Channel" and pressing [**RETURN**].

Step 4: If you wish to obtain a hardcopy of the data presently displayed, using the TAB key, select **PRINT** and press [**RETURN**].

Step 5: Press the **ESCAPE** key **(F11)** or select **QUIT** to exit the **PRESSURE** Test Data screen.

DISPLACEMENT XDUCER Test Data Display

Appendix A indicates whether or not your system includes this screen. If so, follow the steps given below to view the Displacement Xducer Test Data display:

*********************************** [redacted] ************************************

The following steps require you to have previously:

1)calibrated the DAS channels
2)enabled the Transducer Enable checkbox on the Master screen
3)downloaded the setup information from the Start Up screen.

****

Step 1: Select **Test Data** from the **DISPLAY** menu.

Step 2: Select the **Displacement Xducer** radio button and press **[RETURN]**. The Data screen is then displayed.

Step 3: You may view data for any channel in the displacement xducer group by **selecting "Next Channel"** or **"Previous Channel"** and pressing **[RETURN]**.

Step 4: If you wish to obtain a hardcopy of the data presently displayed, using the **TAB** key, select **PRINT** and press **[RETURN]**.

Step 5: Press the **ESCAPE** key **(F11)** or select **QUIT** to exit the **Displacement Transducer** Test Data screen.

HISTOGRAM DISPLAY

This tabular display shows the distribution of endpoint load values achieved from the beginning of the test, up to the current endpoint.

To view the tabular Histogram display, follow the steps given below:

Step 1: Select **Histogram** from the **Display** menu. Since only eight control channels can be displayed on the screen at one time, you must enter a control channel number within the group of channels you are interested in. For example; if you wish to view a tabular Histogram display for control channels 13 and 14 (which happen to be in the second group of channels from 9 to 16), you may enter any number between 9 and 16 to obtain the corresponding display.

Step 2: Once you have entered a control channel number, press **[RETURN]**. The screen displays a tabular Histogram such as that shown in **Figure 11-9**.

Step 3: You may view the entire range of load values, from -100% to +100%, simply by using the keyboard's up/down arrow keys or the next screen/previous screen keys.

Step 4: f you wish to view other channels not included on the initial display, press the **ESCAPE** key **(F11)** and select **Histogram** from the **Display** menu. At this point, enter the desired control channels.

Step 5: Interpreting the data included on the **Histogram** display is easy. The left-most column represents the 1% buckets of the histogram. There are a total of 202 buckets, 200 of which are used to represent the range from -100% to +100% of full scale, while the remaining two buckets are used for counting the number of times the loads were greater than full scale.

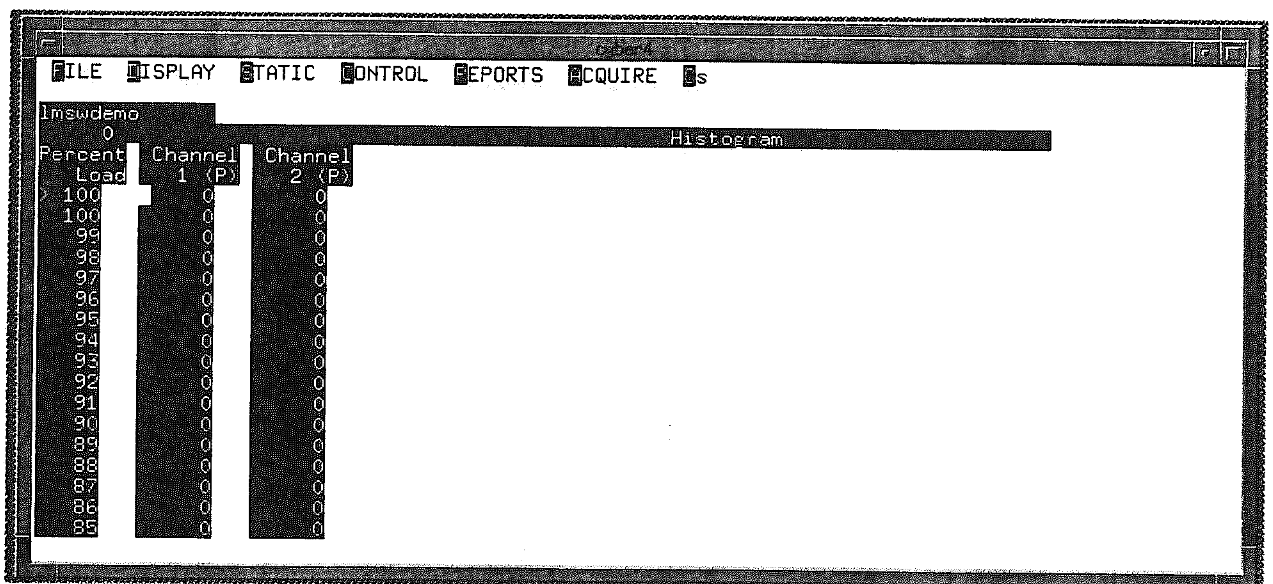


Figure 11-9. Tabular Histogram Display

??
BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

CHAPTER 12

USING REAL-TIME GRAPHIC DISPLAYS

Available Graphic Formats

If your system is configured with Cyber's Real-Time Graphics Subsystem, three different types of graphic displays are available:

Strip Charts. These plots display a continuous graph of various selectable parameters associated with a control channel, including the command, feedback, error, and valve current signals. You may also display the contents of the servo history data buffer in the form of a strip chart. This is described in detail in Chapter 14.

Bar Graphs. This displays shows the feedback, in engineering units or as a percentage of full scale, for up to 24 control channels, simultaneously.

X-Y Plots. If your system is configured with Cyber's Auxiliary Data Acquisition System, then you also have the ability to display DAS data and control data in the form of an X-Y plot.

Each type of graphic display is described separately in this chapter.

??

BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.

??

Since graphic screens can not be displayed on an alphanumeric terminal (e.g. WYSE-185), you must use an X-Terminal to perform the following operations.

To access the X-Window Graphs menu, select the **Display** menu, **Graphs** option. The X-Window Graphs menu, **Figure 12-1**, is displayed.

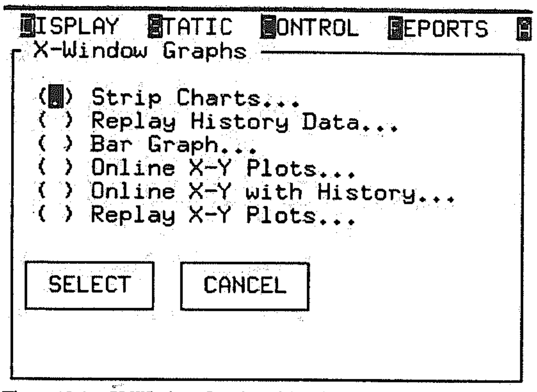


Figure 12-1. X-Window Graphs Dialog Box

DISPLAYING STRIP CHART

The FM7000 graphics software allows you to display an on-line strip chart of selected control channels. This feature may be used in place of, or in conjunction with, the analog strip chart recorders often used in test laboratories. For each chart, you may:

- Select the control channel to display.
- Select the signal to display; (feedback, error, command, or valve current).
- Select how the vertical scale of the plotted data is displayed.

Follow these steps to define the strip chart parameters:

Step 1) Select Graphs, Strip Charts option. The dialog box in **Figure 12-2** appears.

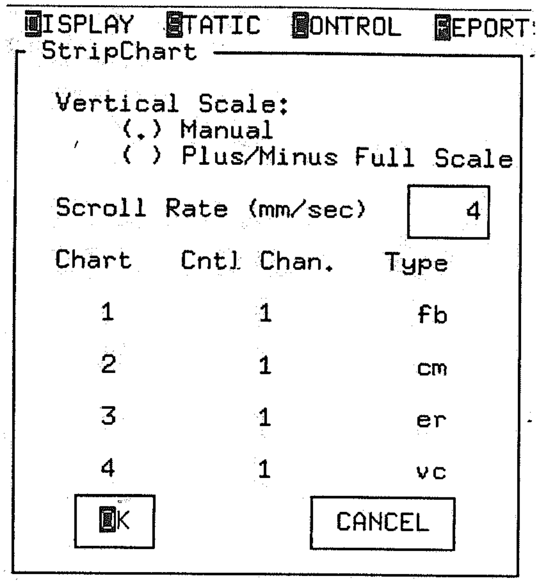


Figure 12-2 Strip Chart Dialog Box

Step 2)

Using the "**Vertical Scale**" radio button, select how the vertical scale of each strip chart is to be displayed. If "**manual**" is selected, another dialog box appears later on which will prompt you to enter the desired minimum and maximum vertical scales for each strip chart. Additional information on "**Manual**" scaling is provided in **step 5.**

If "**+- Full Scale**" is selected, the vertical scale for the selected channels will range from ñ full scale. Please note that full scale for each selected channel refers to the corresponding engineering unit full scale value entered in the Channel Description screen (see Chapter 3).

NOTE:The +/- full scales for Valve Current range from +102.3 to -102.4 % of full scale valve current.

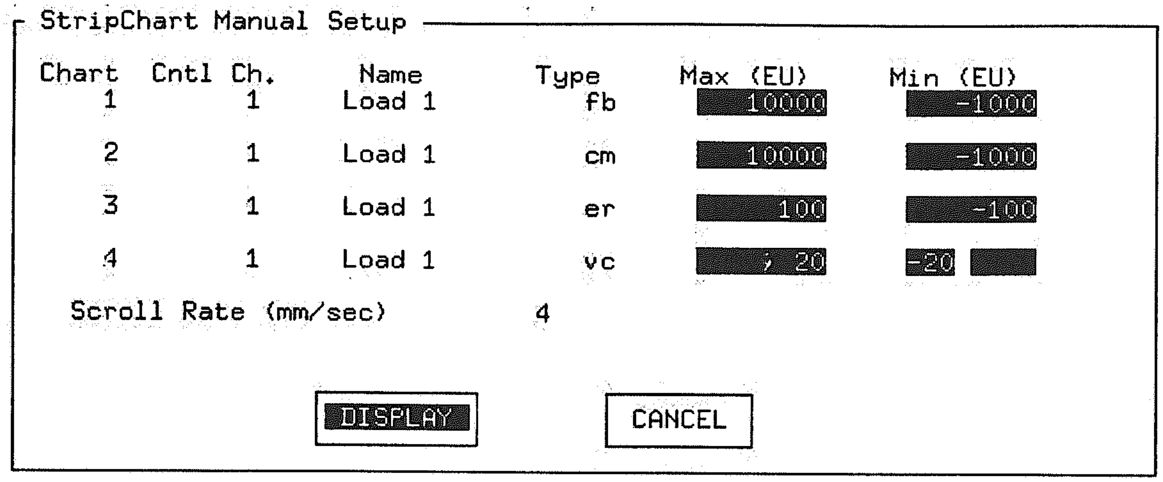


Figure 12-3 Manual Scaling Dialog Box

Step 3) Enter the strip chart scroll rate in millimeters-per-second in the box labeled "**Scroll Rate, min/sec**". This entry ranges from 1 mm/sec (which is slow) to 100 mm/sec (which is fast).

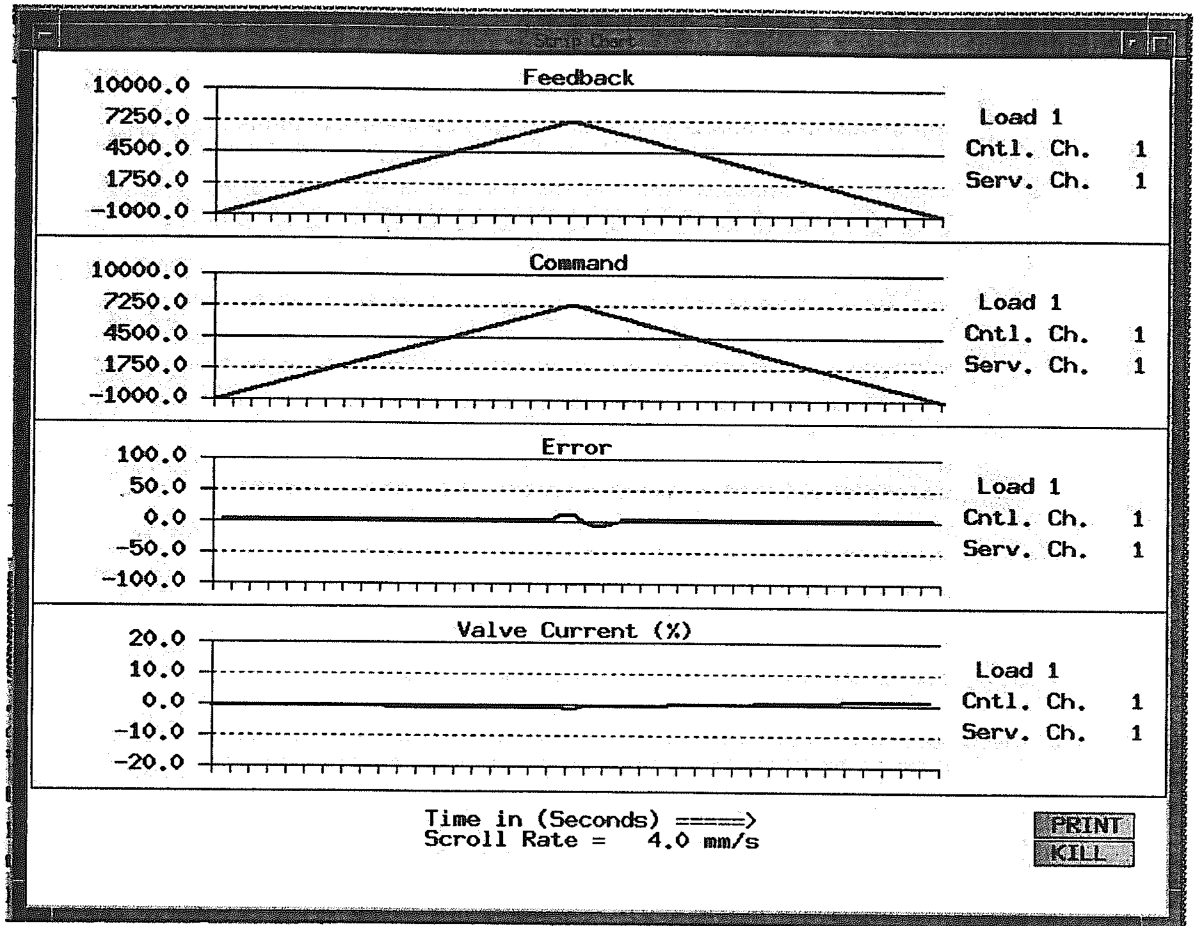


Figure 12-4 Example of a Typical Strip Chart Display

Step 4) Using the "**Log Chan**" column, enter the logical channel number for up to four charts. For each chart, specify the control parameter to be plotted by entering one of the following labels in the "**Parameter**" box:

fb (feedback)
- **cm** (command)
- **er** (loop error)
- **vc** (valve current)

If less than four channels are to be plotted, simply leave the remaining "Log Chan" entries empty.

Step 5) Once all entries have been made, select **OK** and press **RETURN**. If "**Manual**" was the selected scaling method in step 2, the dialog box in **Figure 12-3** will appear. Enter the individual "**Max**" and "**Min**" vertical scale values in engineering units for each desired chart. Once all manual scaling entries have been made, select **DISPLAY** and press **RETURN**.

The graph will appear as a message is sent to the selected servo channel(s) to start sending loop data at a rate of 10 times per second. As this data is received by the host, the strip chart begins to scroll from left to right.

If "**+- Full Scale**" was the selected scaling method, a strip chart window similar to the one shown in **Figure 12-4** will appear on the X-terminal. Using the mouse, you may move and/or resize the window, as desired.

******************************** NOTE ****************************

The number of strip chart setups displayed in this dialog box is equal to the number of "Log Chan" entries made in the prior dialog box.

****

Step 6) You may move and/or resize the window (using the mouse), as desired.

Printing Strip Charts

To obtain a hardcopy of the strip chart display, move the mouse pointer to the "**PRINT**" box in the lower right-hand corner of the window and press the left-most button on the mouse. You should hear a "beep" from the X-terminal. This indicates that the X-terminal is processing the window, converting it into the proper print format, and transferring the data to the host. Once in the host, the converted data is shipped to the postscript laser printer. The ready LED on the laser printer should begin to blink. Due to the large quantity of data contained in the window, it will take several minutes for the printer to generate the hardcopy.

Killing Strip Charts

To kill a particular strip chart window, move the mouse arrow to the "**Kill**" button in the lower left-hand corner of the window and press the right-most button on the mouse. That graphic window will disappear, but all other windows on the X-Terminal will remain. Also, killing the graph stops the flow of data from the servos.

DISPLAYING BAR GRAPHS

Cyber's Real-Time Graphics software allows you to display the servo controller's feedback signal as a bar graph display. Each control channel is represented by a bar. Up to 24 channels can be displayed simultaneously. The bar graph can be set up to display the feedback data in engineering units or as a percent of 100% load. A horizontal blue line above/below each bar indicates the target command for each endpoint, while horizontal red lines above/below each bar represent the ñ outer feedback limit settings.

Follow these steps to select the channels to include in the bar graph:

Step 1) Select **Graphs**, **Bar Graphs** option. The dialog box illustrated in **Figure 12-5** is displayed.

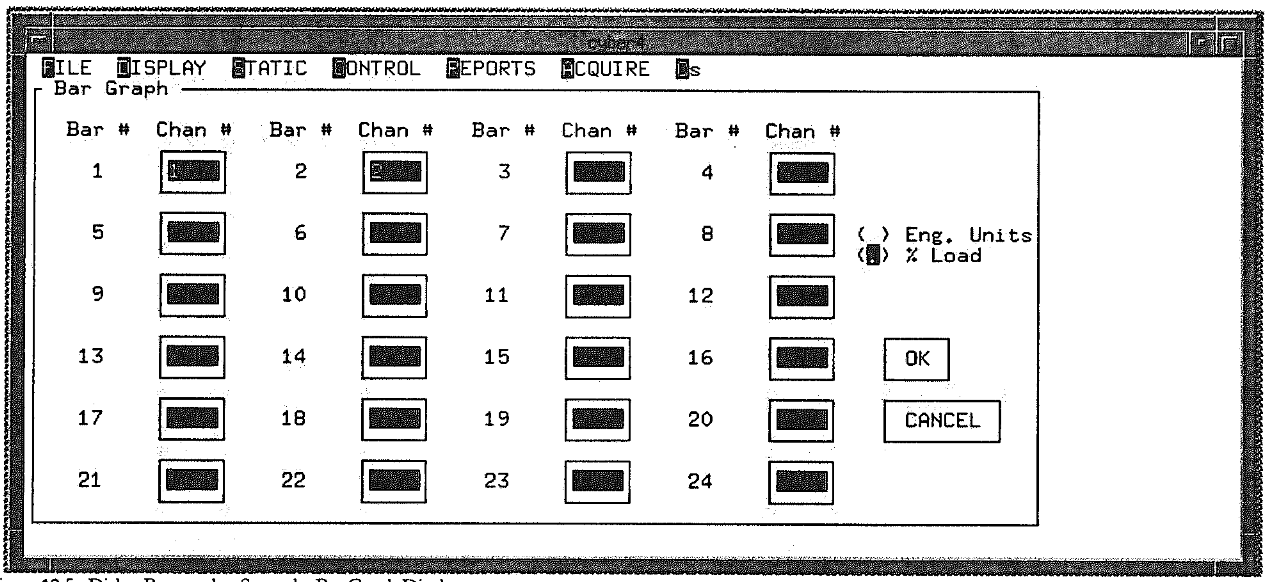


Figure 12-5 Dialog Box used to Setup the Bar Graph Display

Step 2) Enter the logical channel number for each channel in the test you wish to include in the bar graph display. Up to 24 control channels may be entered; if you wish to display fewer than 24 channels, leave the unused channel entries blank.

Step 3) Using the "**Eng. Units/% Load**" radio button, select how you want the bar graph data displayed.

Step 4) Select **OK** or press **RETURN** and a window will appear on the X-terminal with the selected channels displayed as a bar graph. This window will also display the name and channel number associated with each bar. The channel's name and number will also be displayed. The width of the individual bars varies depending on the number of channels entered. Each bar is updated with 4 loop data messages per second.

Figure 12-6 and **12-7** are examples of the bar graph display. **Figure 12-6** has been set up to display the data in engineering units, while **Figure 12-7** displays the data as a percent of the 100% Load entry (also referred to as Test Load) on the Channel Parameters screen (Chapter 3).

With both types of bar graph displays, the corresponding target command value (endpoint value for the endpoint the channel is trying to reach) is represented as a blue line above each bar (or below, depending on the direction of the signal). Each bar also includes a set of red line which represents the ñouter feedback limits defined in the Channel Limits screen (Chapter 3).

Printing Bar Graphs

To print a Bar Graph, move the mouse arrow to the "**PRINT**" box in the lower right- hand corner of the window and press the left-most button on the mouse.

Killing Bar Graphs

To kill an Bar Graph, move the mouse arrow to the "**Kill**" box in the lower right-hand corner of the window and press the left-most button on the mouse. The window will disappear, also stopping the flow of loop data messages any other displayed windows will remain.

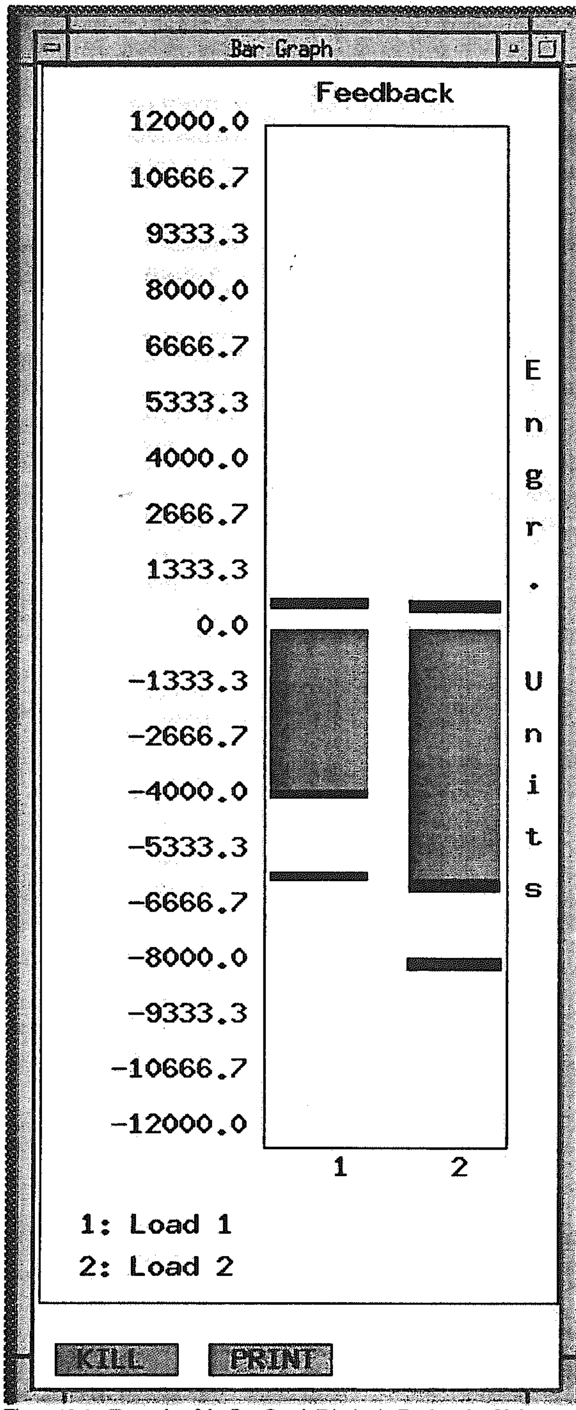


Figure 12-6 Example of the Bar Graph Display in Engineering Units

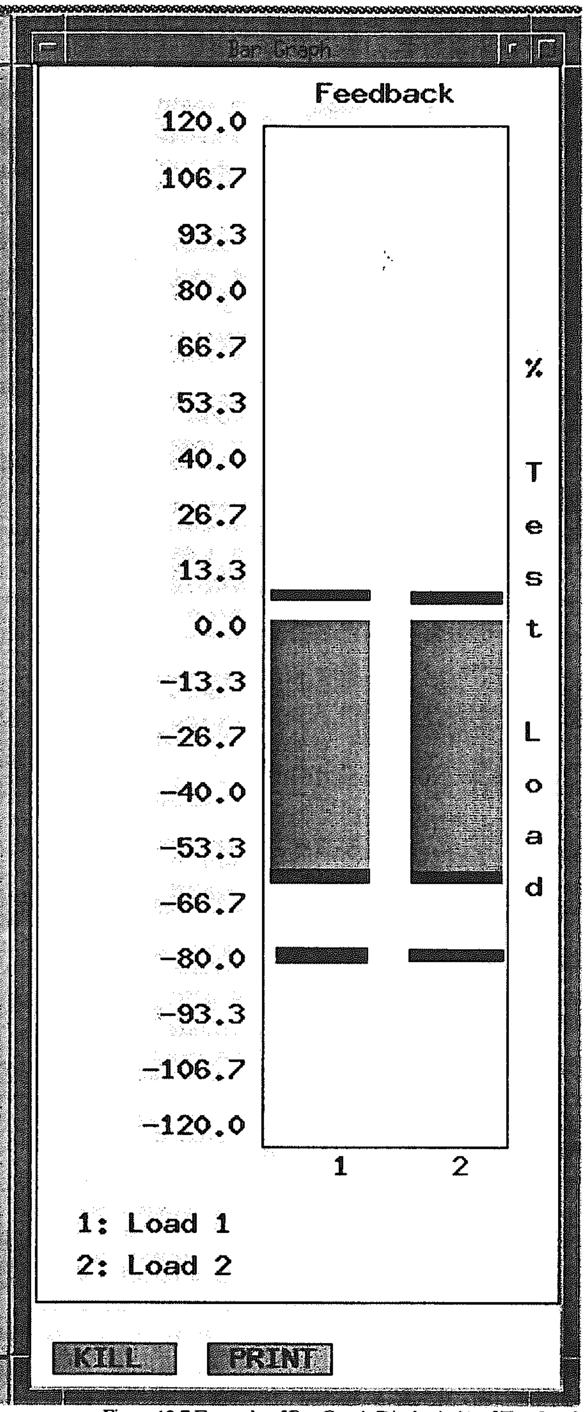


Figure 12-7 Example of Bar Graph Display in % of Test Load

DISPLAYING X-Y PLOTS

For systems configured with Cyber's ADAS, the graphics subsystem provides X-Y plots of up to four y-axis channels versus one versus x-axis channel. These plots are intended to be used primarily for static testing applications.

With all systems, the y-axis channels represent separate DAS channels. However, depending on the way the system software was configured at the factory, the x-axis channel may represent a control channel or a DAS channel. See Appendix A to find out if your system allows the x-axis channel to represent a control channel or DAS channel.

The FM7000 offers several different types of X-Y plots, including On-Line X-Y plots, On-Line X-Y with History, and Replay X-Y Plots. Below is a brief description of how each plot works:

On-Line X-Y plots:	this type of plot displays data which begins at the time the plot was requested and ends when the plot is killed
On-Line X-Y with History:	this type of plot appends on-line data with previously data recorded
Replay X-Y Plots:	this is a plot of previously recorded data

All three display types are plotted on an endpoint basis. Each type is described separately in the following sections.

On-Line X-Y Plots

On-Line X-Y plots are typically used when you want to display test data as it is being obtained. These displays plot data beginning at the time of the request, up until the time the display is killed. One example of when you would use this feature would be during a static test, as the system is approaching a critical load. Displaying the On-Line X-Y plot at this time would allow the test operators to focus on the present load transition.

Follow these steps to display an On-Line X-Y plot:

Step 1) Select **Graphs, On-Line X-Y Plots** option. The dialog box in **Figure 12-8** is displayed.

Step 2) To help distinguish each of the Y-axis channels in the plot, different symbol (e.g circles, squares, triangles, etc.) are used for each of the Y-axis channels. This helps to easily identify each Y-axis channel on a hardcopy, which is typically black & white, as well as on monochrome X-terminals. Since each graph is plotted on an endpoint basis, you may define how often a symbol is to be plotted (i.e. every 1 endpoint, every 2 endpoints, etc.).

Enter the symbol update rate in the entry labeled Y-axis symbol update rate: At every _ endpoint(s). This is a global parameter used on all on-line XY plots.

```
 ISPLAY  TATIC  ONTROL  EPORTS  CQUIRE  s
 X-Window Graphs
 Online X-Y Plots
   Y-axis symbol update rate: At every 1  endpoint(s)
  [ ] Straight Line prediction
           X1          Y1            X2           Y2

               OK                  CANCEL
```

Figure 12-8 Setting Up Global Parameters for On-Line X-Y Plots

Step 3) This dialog box also allows you to define a straight line prediction which will be plotted along with the test data. This straight line prediction is defined as a pair of X and Y coordinates. Once again, this is a global parameter used on all on-line XY plots.

If desired, enter X' in the checkbox and then enter the first XY coordinate of the straight line in the entry boxes labeled X1 and Y1. Enter the second XY coordinate of the straight line in the entry boxes labeled X2 and Y2.

Step 4) Press **RETURN**. The dialog box shown in **Figure 12-9** appears. This dialog box is used to specify which channels are to be plotted, how the vertical axis is to be scaled, and what labels are to be given to the x and y axes. Depending on your configuration, the labels may be automatically filled in as you tab the label fields. The labels used would be the X and first Y channel's units entry from the DAS Setup screen.

```
 ISPLAY  TATIC  ONTROL  EPORTS  CQUIRE  s
 Online X-Y Plots
  Vertical Scale:
      (.) Manual
      ( ) +/- Full Scale
 DAS Ch. (X)  DAS Ch. (Y1)  DAS Ch. (Y2)  DAS Ch. (Y3)  DAS Ch. (Y4)
      8            9             10            16            17
   X-axis label PSI        Y-axis label STRAIN

               OK                  CANCEL
```

Figure 12-9 Dialog Box used to Set Up an On-Line X-Y Plot

Step 5) Using the "**Vertical Scale**" radio button, select the desired method of scaling for the X-Y plots. The "**Manual**" selection is described further in step 10. If "**+- Full Scale**" is selected, the system uses the full scale value of the selected channel as the x-axis scale. As for the y-axis scale, the system uses the largest full scale of the DAS channels to be plotted.

Step 6) Enter the x-axis channel.

******************************** NOTE ********************************

If your system requires the x-axis to be a control channel, this dialog box will have an entry labeled LCS Ch. (X). Enter the desired control channel.

If your system uses the ADAS to measure the buffered feedback output at the rear of each servo controller, each ADAS data scan includes all of the test transducers, as well as the control channel feedback signals (i.e. the loads). This provides more flexibility and allows the x-axis channel to be any ADAS channel (i.e. one that represents a control channel OR one that represents a external test transducer such as a strain gage.) If your system has been configured in this way, the dialog box will have an entry labeled DAS Ch. (X). Enter the desired DAS channel for the x-axis.

****

Step 7) Enter up to four (4) y-axis channels to be plotted in the entries labeled:

DAS Ch. (Y1) **DAS Ch. (Y2)** **DAS Ch. (Y3)** **DAS Ch. (Y4)**

Step 8) Enter a label for the x-axis and y-axis, if desired. These labels are used on displays and hardcopies of the X-Y plot.

Step 9) If **+-Full Scale** was the selected scaling method back in step 5, continue on from step 11.

Step 10) Select **OK** or press **RETURN**. Since "**Manual**" was the selected scaling method, the dialog box illustrated in **Figure 12-10** is displayed.

The dialog box provides information on how the selected channels were defined during setup (i.e. their full scale value and names) and entries for defining the x- and y-axis. Using the displayed full scales as a reference, enter the desired Max and Min values for the x-axis in the column labeled DAS X.

Using the displayed full scales for each DAS channel as a reference, enter the desired Max and Min values for the common y-axis in the column labeled Common Y axis.

All entries are in the engineering units of the specified channels. If multiple DAS channels are entered, the engineering units of the Y-axis will correspond to the entries made in Common Y-Axis column.

******************************** NOTE ********************************

There is only one set of scaling entries (i.e. + extreme and - extreme) for plots containing multiple y-axis channels. The Full Scale value, Channel Number, and Channel Name are displayed on the plot for reference.

****

Step 11) Select **OK** or press **RETURN**. A window similar to the one shown in **Figure 12-11** will appear.

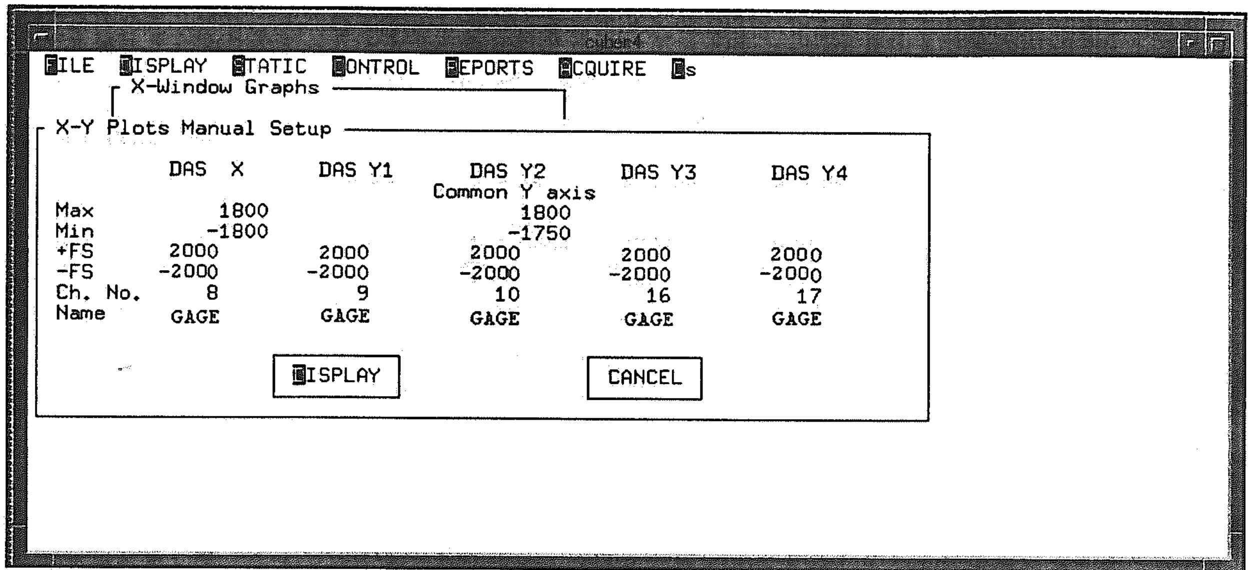


Figure 12-10 Dialog Box used to Manually Define the X-Y Plot Scales

************************************** NOTE **********************************

The X-Y PLot is updated under the following conditions:

1) only at endpoints
2)the ADAS must have been previously calibrated
3)the ADAS must be scanning

****

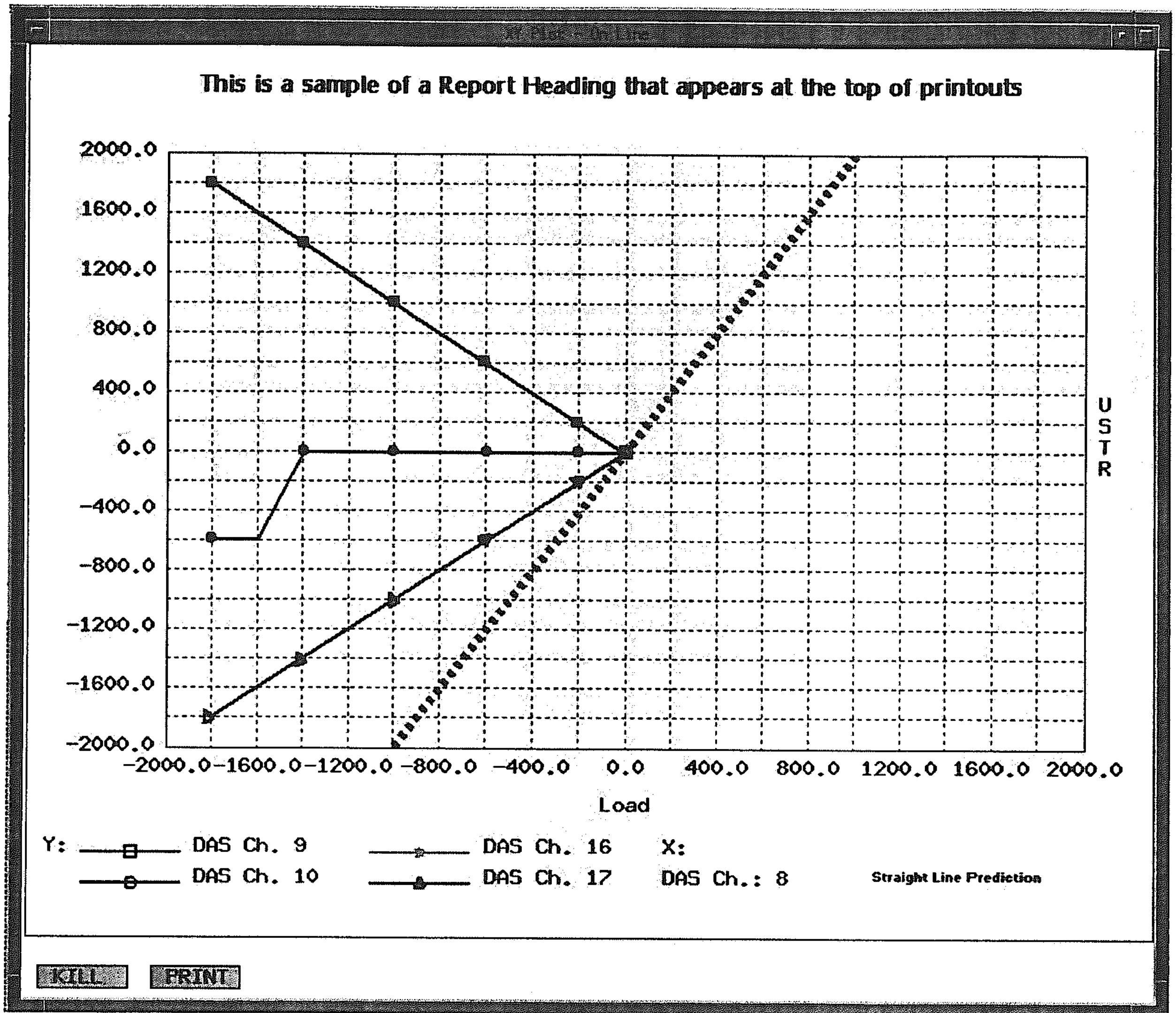


Figure 12-11 Example of an On-Line XY Plot

Killing X-Y Plots

To kill an X-Y Plot, move the mouse arrow to the "**Kill**" box in the lower right hand corner of the window and press the left-most button on the mouse. The window will disappear; any other displayed windows will remain.

Printing X-Y Plots

To print an X-Y Plot, move the mouse arrow to the "**PRINT**" box in the lower right- hand corner of the window and press the left-most button on the mouse.

ON-LINE X-Y PLOTS WITH HISTORY

This feature allows you to plot previously recorded test data combined with on-line test data. One example of when this feature would be useful is during a static test, if you wish to see a plot of the test data from the beginning of the test, up to the present load condition, and then continue to plot data for subsequent endpoints as the test proceeds. Follow these steps to obtain an On-Line X-Y Plot with History.

Follow these steps to use the Replay X-Y Plot feature:

Step 1) From the **DISPLAY** menu, select the **Graphs** option..

Step 2) Select **Graphs, On-Line with History** option. A dialog box is displayed.

Step 3) Enter the symbol update rate. The default rate is every 10 endpoints.

Step 4) If you wish to have a straight line prediction on the plot, enter an "X" in the check box labeled [ ] Straight Line prediction. Also enter the pair of XY coordinates for the prediction line.

Step 5) Select **OK** or press **RETURN**. The dialog box appears.

Step 6) Using the Vertical Scale radio button, select the desired method of scaling for the X-Y plots. The "Manual" selection is described further in step 12. If "ñ Full Scale" is selected, the system uses the full scale value of the selected channel as the x-axis scale. As for the y-axis scale, the system uses the largest full scale of the DAS channels to be plotted.

Step 7) Enter the x-axis channel.

******************************** NOTE ******************************

If your system requires the x-axis to be a control channel, this dialog box will have an entry labeled LCS Ch. (X). Enter the desired control channel.

If your system uses the ADAS to measure the buffered feedback output at the rear of each servo controller, each ADAS data scan includes all of the test transducers, as well as the control channel feedback signals (i.e. the loads). This provides more flexibility and allows the x-axis channel to be any ADAS channel (i.e. one that represents a control channel OR one that represents a external test transducer such as a strain gage.) If your system has been configured in this way, the dialog box will have an entry labeled DAS Ch. (X). Enter the desired DAS channel for the x-axis.

****

Step 8) In the column labeled **DAS Ch. (X),** enter the channel number to be used for the x-axis. This channel must be included in your test.

Step 9) Enter up to four (4) ADAS channel number(s) in the columns labeled:

DAS Ch. (Y1) **DAS Ch. (Y2)** **DAS Ch. (Y3)** **DAS Ch. (Y4)**

Step 10) Enter a label for the x-axis and y-axis, if desired. These labels are used on displays and hardcopies of the X-Y plot.

Step 11) If **"+-Full Scale"** was the selected scaling method back in **step 6**, continue on from **step 13**.

Step 12) Select **OK** or press **RETURN**. Since **"Manual"** was the selected scaling method, the dialog box illustrated in **Figure 12-15** is displayed.

The dialog box provides information on how the selected channels were defined during setup (i.e. their full scale value and names) and entries for defining the x- and y-axis. Using the displayed full scales as a reference, enter the desired Max and Min values for the x-axis in the column labeled DAS X.

Using the displayed full scales for each DAS channel as a reference, enter the desired Max and Min values for the common y-axis in the column labeled Common Y axis.

All entries are in the engineering units of the specified channels. If multiple DAS channels are entered, the engineering units of the Y-axis will correspond to the entries made in Common Y-Axis column.

********************************** NOTE ******************************

There is only one set of scaling entries (i.e. + extreme and - extreme) for plots containing multiple y-axis channels. The Full Scale value, Channel Number, and Channel Name are displayed on the plot for reference.

****

Step 13) Select **OK** or press **RETURN**. A window similar to the one shown in **Figure 12-16** will appear. This display includes all of the previously recorded test data for the channels defined in the setup screen.

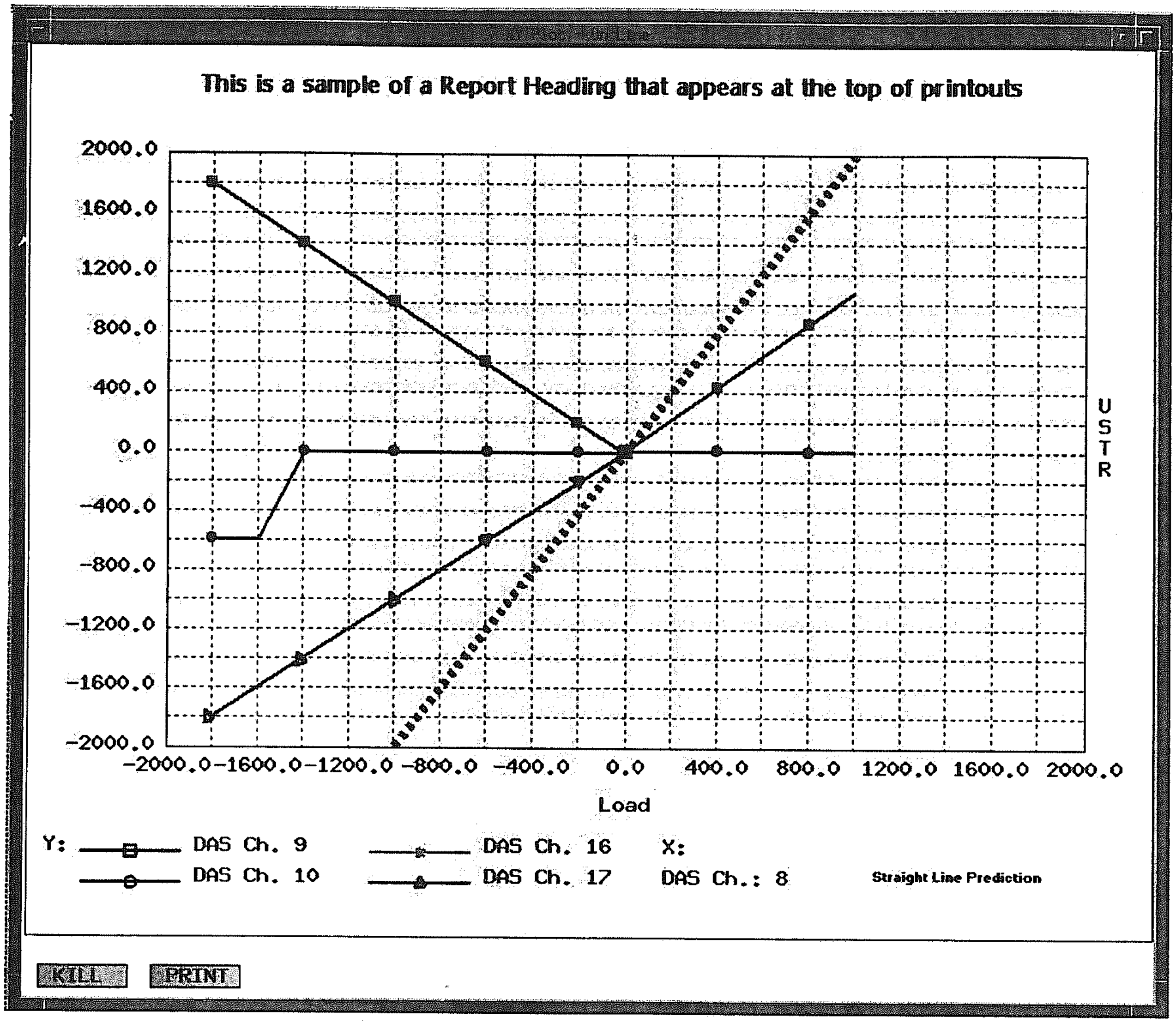


Figure 12-16 Example of an X-Y Plot with History

Step 14) As your test continues and the system reaches the next endpoint, new data is appended to the historical data (i.e. the DAS endpoint data which was previously recorded to the disk data file). An example of this is shown in Figure 12-17. The plot in this figure is the same as the one in Figure 12-16. However, if you look closely at the left side of the plot, you will notice that a new endpoint has been plotted from -1000.0 to +1000.0.

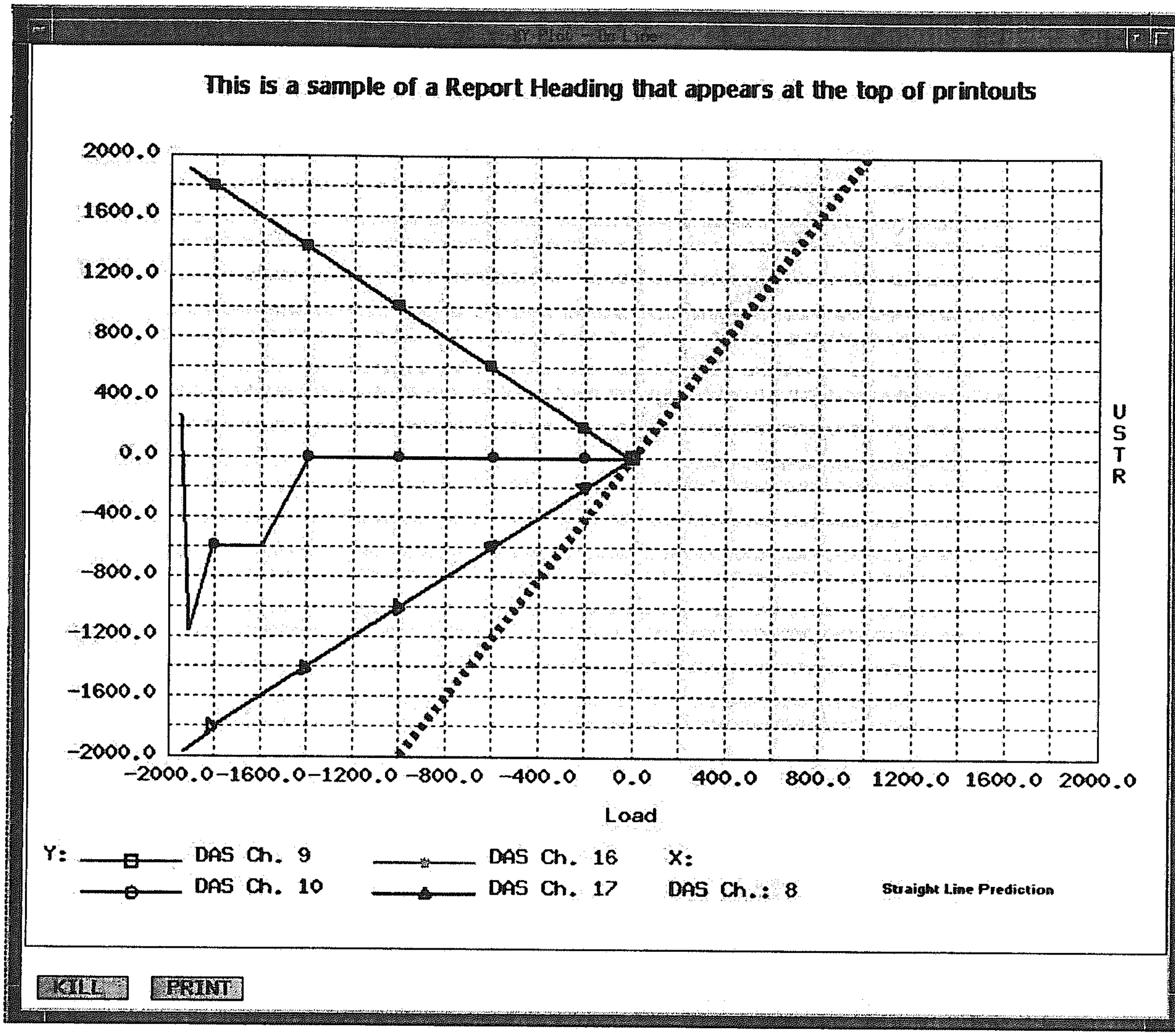


Figure 12-17 Same Plot as in Previous Figure, Except On-Line Data has been Added

************************************** NOTE **********************************

The X-Y Plot with History will continue to update under the following conditions:

1) only at subsequent endpoints
2)the ADAS must have been previously calibrated
3)the ADAS must be scanning

****

REPLAY X-Y PLOTS

Replay XY plots are of previously recorded test data only. They are most useful after a test has been completed and you wish to see a plot of the recorded test data for the entire test.

The first step in using the **Replay XY** plot feature is to set up which X channel and Y channel(s) are to be included in each plot. In fact, the system allows you to define up to 200 different plots. With each plot you select the scaling method for each axis. You may also define a straight line prediction for each plot, if desired. Once setup is completed, you have the option to submit the list of predefined plots as a batch job to the printer OR select a predefined plot for display.

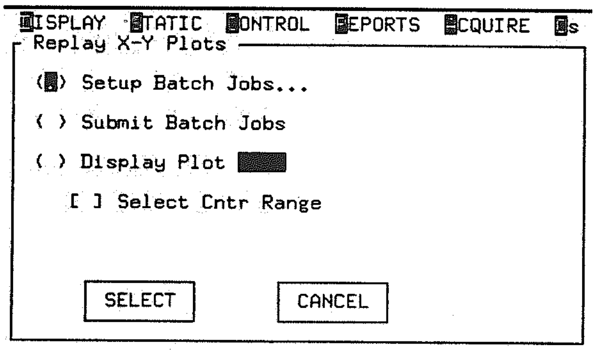


Figure 12-18 Dialog Box used to Select the Replay XY Plot Function of Interest

Follow these steps to use the Replay X-Y Plot feature:

Step 1) Select **Graphs, Replay X-Y Plots** option. The dialog box shown in **Figure 12-18** is displayed.

Step 2) Use the radio button to select "**Setup Batch Job**" and press **RETURN**.The spread sheet screen shown in **Figure 12-19** appears.

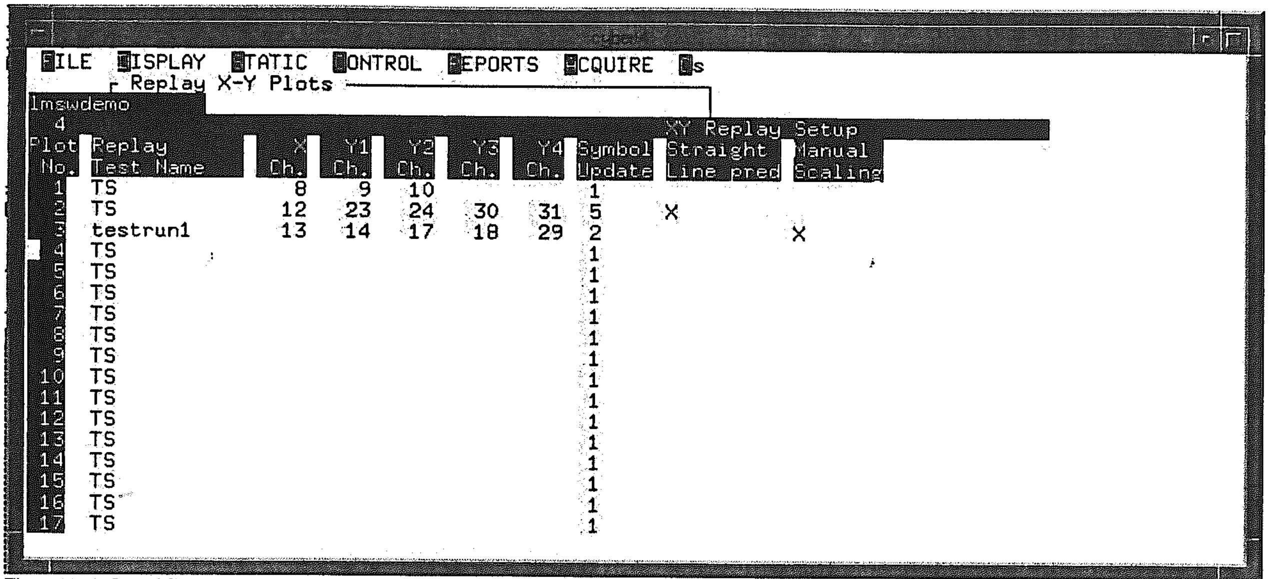


Figure 12-19 Spread Sheet used to Set Up one or more Replay XY Plots

Step 3) To set up a **replay X-Y** plots you must identify which Test Data file the data is to come from. The spread sheet is born with **TS** as the default test data file name. It is this file which contains the recorded data for the present test run. Each time the test is downloaded, this file is reinitiailzed.

If your wish to plot data from a previously recorded test data file and not the on-line file **TS**, enter the appropriate file name in the **Replay Test Name** column.

Step 4) In the column labeled **X Ch.**, enter the channel number to be used for the x-axis. This channel must have been used in the test for which the Test Data file in the **Replay Test Name** column was created.

Step 5) Enter up to four (4) ADAS channel number(s) in the columns labeled:

Y1 Ch. **Y2 Ch.** **Y3 Ch.** **Y4 Ch.**

Step 6) Enter the desired symbol update rate in the column labeled **Symbol Update**. The default rate is 1; a symbol is plotted at every endpoint.

Step 7) If you wish to define a straight line prediction, tab over to the column labeled **Straight Line pred** and enter an "X" by pressing the space bar. Press **RETURN**. The dialog box will appear, prompting you to enter the pair of XY coordinates for the prediction line. Enter the coordinates and press **RETURN**.

Step 8) If the desired scaling for the plot is to use "+- **Full Scale**" for both axes, leave the Manual Scaling column blank and go to step 10.

Step 9) If manual scaling is desired for the plot, tab over to the column labeled Manual Scaling and enter an "X" by pressing the space bar. Press **RETURN**. The dialog box will appear. Enter the min and max scale factors for each X and Y axes in the columns labeled **X Min**, **X Max**, **Y Min**, and **Y Max**. These entries must be in engineering units.

Step 10) Repeat steps 4 through 9 for each of replay plot you wish to set up.

Step 11) Press **ESCAPE** to exit the screen. This will temporarily save the data. Use the system's save functions to save the setup permanently.

Submitting Batch Jobs to the Laser Printer

Once the replay XY plots have been setup as described in the previous section, you may initiate the automatic X-Y Plot printing feature, otherwise referred to as Submit Batch Jobs. To do this, perform the following steps:

Step 1) Select **Graphs**, **Replay X-Y Plots** option. The dialog box shown in **Figure 12-22** is displayed.

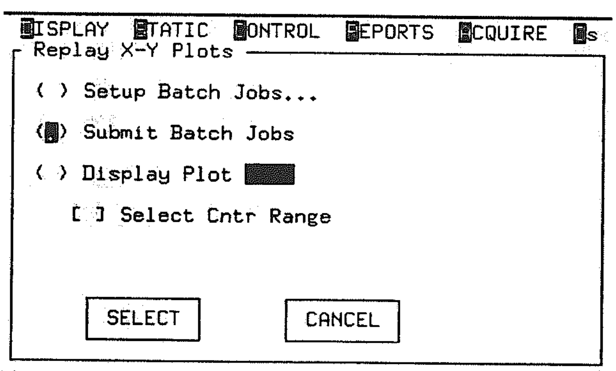


Figure 12-22 Dialog Box used to Select the Submit Batch Jobs Option

Step 2) Use the radio button to select "Submit Batch Job" and press **RETURN**. Another small dialog box will appear, prompting you to enter a range of plots to be submitted to the laser printer. Use the Plot# from the Setup Batch Jobs spread-sheet screen to specify the range. If only one plot is desired (e.g. plot 7), specify the range as [7] to [7].

Step 3) Press **RETURN** to start the process. Each plot submitted will be temporarily displayed on the X-terminal while printing is in progress. When the first plot is finished printing, the next replay-plot will be submitted. This process continues until all plots in the specified range are completed.

DISPLAYING REPLAY X-Y PLOTS

The system also allows you to display a previously defined Replay XY plot, without printing it. To do this, perform the following:

Step 1) Select **Graphs, Replay X-Y Plots** option. The dialog box shown in **Figure 12-24** is displayed.

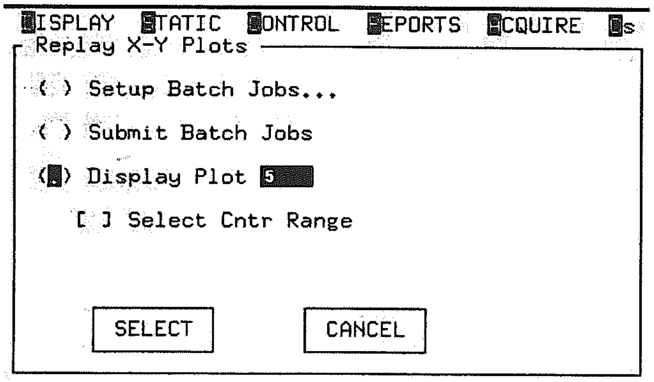


Figure 12-24 Dialog Box used to Select a Replay XY Plot for Display

Step 2) Move the radio button to Display Plot.

Step 3) Tab to the entry box and enter the plot number from the Setup Batch Job screen to be displayed. Please note that the plot must first be defined on the Setup Batch Job screen before it can be displayed.

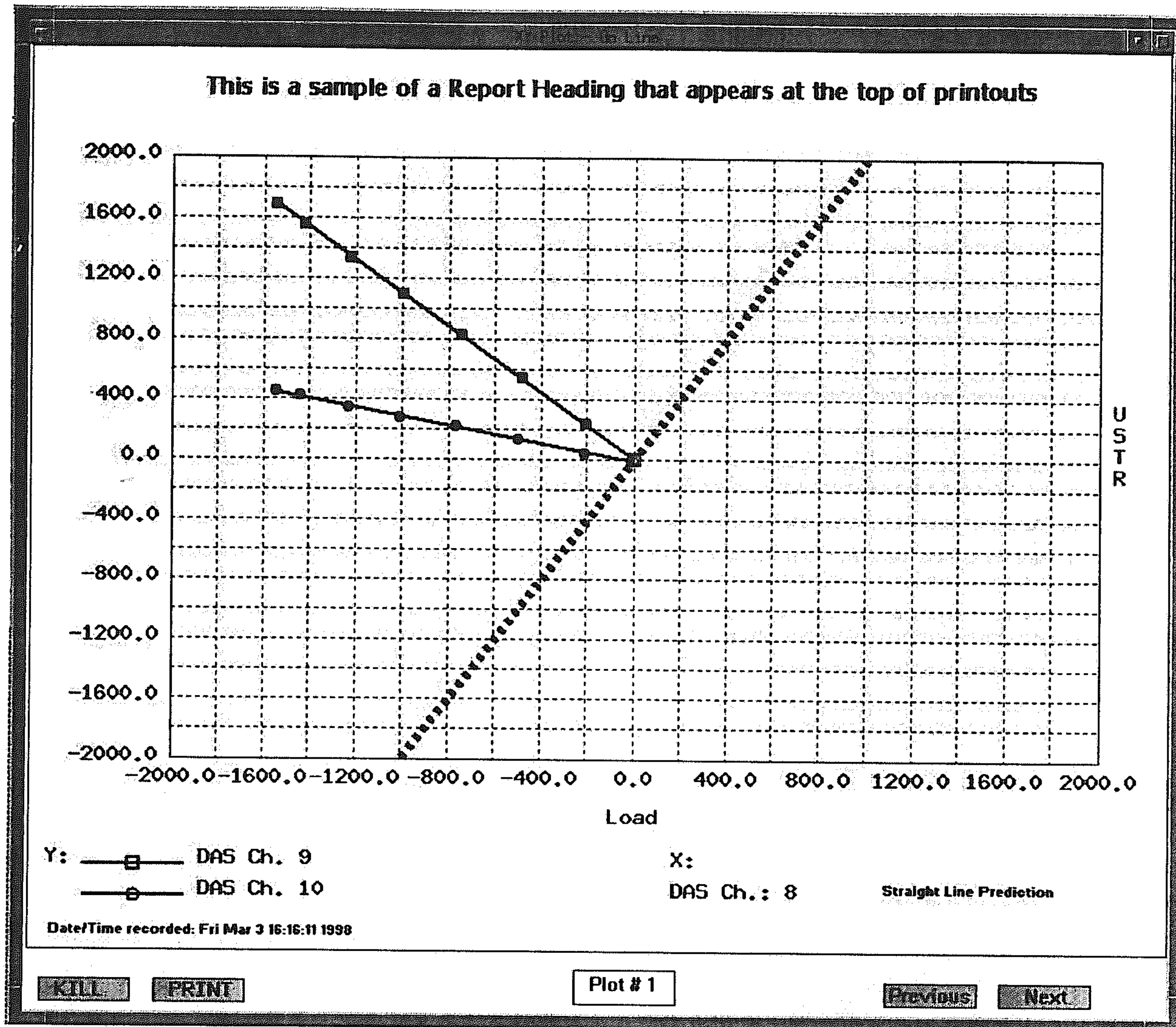


Figure 12-25 Example of a Replay XY Plot Display

Step 4) Select **OK** and press **RETURN**. The selected plot will be displayed on the X-terminal. **Figure 12-25** provides an example of a **Replay XY Plot** display.

Step 5) If you wish to display other previously defined Replay XY plots, use the [**NEXT**] and [**PREVIOUS**] buttons at the lower right-hand side of the window to page through the plots as defined in Setup Batch Job. Simply place the mouse pointer on either selection and click the left-most button on the mouse.

********************************** NOTE ******************************

This plot may be killed or printed in the same manner as on-line X-Y plots.

****

CHAPTER 13

PRINTING TEST REPORTS

You can print a variety of reports that describe setup data, load conditions, test procedures, calibration results and test data. If fact, some reports are hardcopy outputs of the test data displays described in the preceding sections. To view the available reports, select REPORTS from the main menu bar. The report options are displayed as shown in **Figure 13-1**.

```
REPORTS  ACQUIRE  s
 Load Conditions
 Special Load Conditions
 Replay Test Data
 Replay Histogram
 Replay History Data
 Replay DAS Data
 LCS Setup
 LCS Calibration
 RCAL Deviation
 Kill Report
 Test Procedure
 DAS Setup
 DAS Calibration
 COMP Channels
 DAS Rosette
```

Figure 13-1. Reports Menu

Below is a list of available test reports, simply select the desired report from the REPORTS menu and supply any requested information. If your printer is on-line, the report will print automatically.

- **Load Conditions** prompts you to enter the desired control channel, and then prints all of the load conditions values for eight contiguous channels including the selected channel.
- **Replay Test Data** prints previously recorded endpoint data (command, feedback, and % error) for each channel in the test from a variety of sources depending on how your system is configured. When this selection is made, the screen shown if Figure 13-2 will appear. Enter the appropriate Test Data File name and the range of counter values which correspond to the endpoints you wish to print. If you do not enter a different Test Data File name, the system uses the TS file as the default Test Data file.

If your system is not configured with DAS, selecting **OK** or hitting **RETURN** will cause an endpoint data report of LCS data (command, feedback, % error) to be printed for the selected counter range.

If your system does have DAS, hitting will produce a selection screen (**Figure 13-3**) on which you may select any or all of the reports listed.

NOTE: You will only get meaningful data from the reports for which an appropriate setup was performed.

- **Replay Histogram** prompts you to enter the desired control channel number, and then prints a report listing the number of times each possible endpoint value (as a percentage of full scale) has been achieved up to the current endpoint being executed. If the test is completed, the system will print the entire Replay Histogram Report for the selected control channel(s) (i.e. group of eight contiguous channels).
- **Replay History Data** prompts you to enter the physical channel, and the range of waypoints to display, and then prints the selected channel's history data. In order to obtain this report, the system requires you to have previously performed a Freeze History/Send Freeze Data procedure. Refer to Chapter 14 for more information regarding this report.

****************************** NOTE *********************************

Be sure to enter the physical channel number when requesting this report.

****

- **Replay DAS Data**This report provides a printout of a 3 minute DAS Rotating Data Buffer (i.e. disk file) stored on the host computer's hard disk. The file consists of the 900 most recent scans of DAS data taken at 200 millisecond intervals. A report is allowed only if the DAS rotating data buffer receives a "Freeze History" command. This is the same "Freeze History" which gets sent to the servo controllers based on entries in the System or Discrete Conditional Action Tables.

The buffer is updated with new scans according to the "% Post Freeze Capture" entry on the Start Up screen. When the DAS rotating buffer is finished updating and you request a report, the system prompts you to enter the Data Sample Range (from 1 to 900), as well as the DAS channel number. A tabular report will be printed of the requested DAS channel, plus the five subsequent DAS channels in the setup list. Also, the sample at which the freeze occurred will be marked by an asterisk. For more information on this buffer, refer to Chapter 14, Examining the DAS History Data.

- **LCS Setup** prints the same information entered in the Channel Parameter screens. When you select this option, a second screen is displayed that lets you select up to four different reports, including:
    - Channel Descriptions
    - Channel Limits
    - Channel Control
    - Channel Tuning

- **LCS Calibration Report** prints the Load Control System calibration values that are presently storedin each servo channel's non-volatile memory.
- **LCS Deviation Report** prints the results of the last RCAL Deviation check.
- **Kill Report** is used to cancel printing of the current report. This feature is useful with lengthy printer jobs.
- **Test Procedure** lists the sequence of test steps that make up the test.
- **DAS Setup** identifies the channels from which the data acquisition system will receive test data. This report is available only with systems configured with Cyber's Auxiliary Data Acquisition System. This report is only available on systems configured with Cyber's Auxiliary Data Acquisition System.

- **DAS Calibration Report** prints the Data Acquisition System calibration results. This report is available only with systems configured with Cyber's Auxiliary Data Acquisition System.
- **COMP Channels** prints out the information entered in the Computed Channels setup screen. This report is only available on systems configured with Cyber's Auxiliary Data Acquisition System.
- **DAS Batch Report** provides a hardcopy output and/or ASCII file output of the DAS data in a more tailored format. Please note that this report may not be included in your system configuration. To see if this feature is included with your system, see Appendix A.
- **DAS Rosettes** provides a report of various rosette-related data, including:

  - rosette number
  - rosette name
  - rosette type
  - calculated minimum and maximum strain
  - gage angle (in degrees)
  - minimum and maximum stress
  - shear stress

This report is only available on systems configured with Cyber's Auxiliary Data Acquisition System.

To print any of the above listed test reports, simply select the desired report from the REPORTS menu and supply any requested information. If your printer is on-line, the report will print automatically.

??
BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

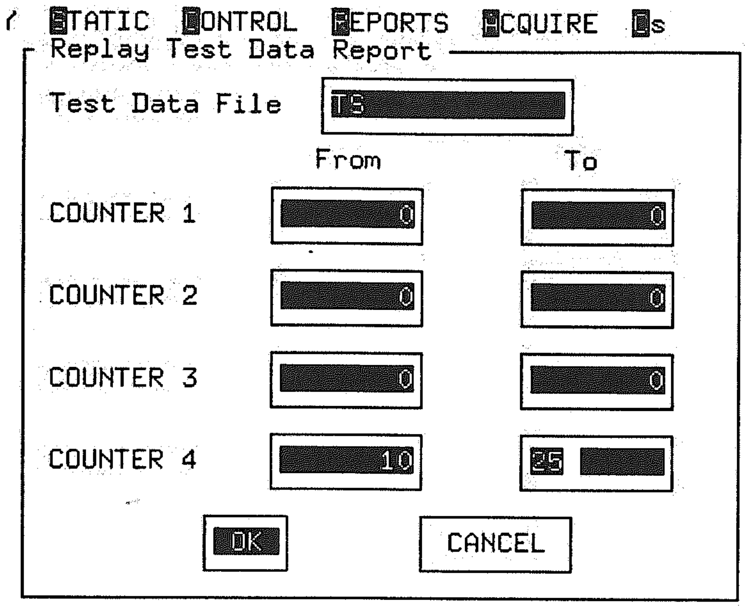


Figure 13-2 Specifying the Counter Range for Replay Reports

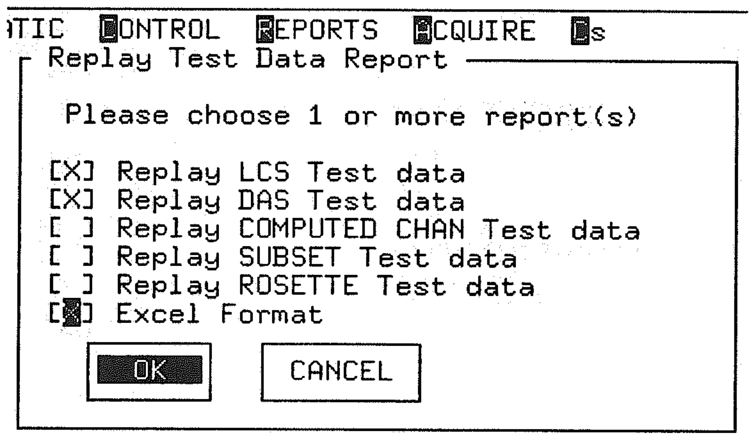


Figure 13-3 Replay Test Data Report Selection

CHAPTER 14

DETERMINING THE CAUSE OF SYSTEM REACTIONS AND EVALUATING TEST EVENTS

In the course of running your test, you will undoubtedly be faced with the task of determining and/or evaluating the cause of various system reactions or test events. Such tasks may include:

- determining which control channel was first to be polled with an Fault#1/Fault#2 condition
- determining the cause (e.g. out-of-limit condition, discrete input response, communication failure, etc.) of such test events as dumps, holds, ramps, etc..
- evaluating the sequence of operational steps taken by the operator prior to the system performing an action (e.g. dump, hold, etc.)
- examining the data captured (i.e. frozen) in the history rotating data buffer for one or more control channels.
- examining the DAS data captured in the systems rotating DAS buffer following a system "freeze" event

The steps necessary to perform these tasks are explained in this chapter.

??

BEFORE PERFORMING ANY OF THE OPERATIONS DESCRIBED IN THIS CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.

??

IDENTIFYING THE FIRST FAULT CHANNEL

The FM7000 provides two first-fault indicators; **First Fault #1** and **First Fault 2**.. In the event of an out-of-limit servo condition, the **First Fault #1** indicator will identify the control channel which was first to be polled with a Fault#1 type error (e.g. outer error or +- outer limits). Similarly, the First Fault #2 indicator will identify the control channel which was first to be polled with a Fault #2 type error (e.g. inner error or ñ inner limits). Both indicators are displayed on the Master screen.

EXAMINING THE TEST EVENTS LOG

Since there are numerous events which can occur when using the system, the FM7000 includes an on-going log of various system conditions and test-related events, including:

- activation of any pushbutton on the test control panel
- all system conditional actions, giving the cause and the action taken
- all discrete conditional actions, giving the number of the discrete that tripped and the action taken
- miscellaneous system events such as valve cable failures and abort/start test information, sending profiles, setting branches/ramps

For each event recorded in the Events Log, the system automatically tags the event with the time of day and test counter values that it occurred. This allows the user to easily retrace the sequence of test conditions occurred, what actions the system took, and what actions, if any, the operator took.

Follow the steps below to examine the Events Log display:

Step 1) Select **Events Log** from the **DISPLAY** menu. A display similar to the example in **Figure 14-1** will appear on the screen.

Step 2) The top of the events log is displayed. You may use the **Prev/Next** selections to page to the end of the log (i.e. most recent events). Upon exit and re-entry of the events log, the display will be positioned where it was when you last exited.

******************************** [redacted] ********************************

If you did not select Clear Events Log on the Start Up screen prior to downloading the current test, be aware that the Events Log may contain events which occurred in previous executions of the same test file.

****

Step 3) If you wish to have a message logged in the Events Log, move the cursor the message field and type the desired message. Two lines are available for this purpose. Use Tab to move to the next line. Once you have typed in your message, move the cursor to the **ENTER** field and press **[RETURN]**. Your message is automatically log with the time & date, and present counter values.

If you wish to obtain a printout of the Events Log, move the cursor to the **PRINT** selection and press **[RETURN]**.

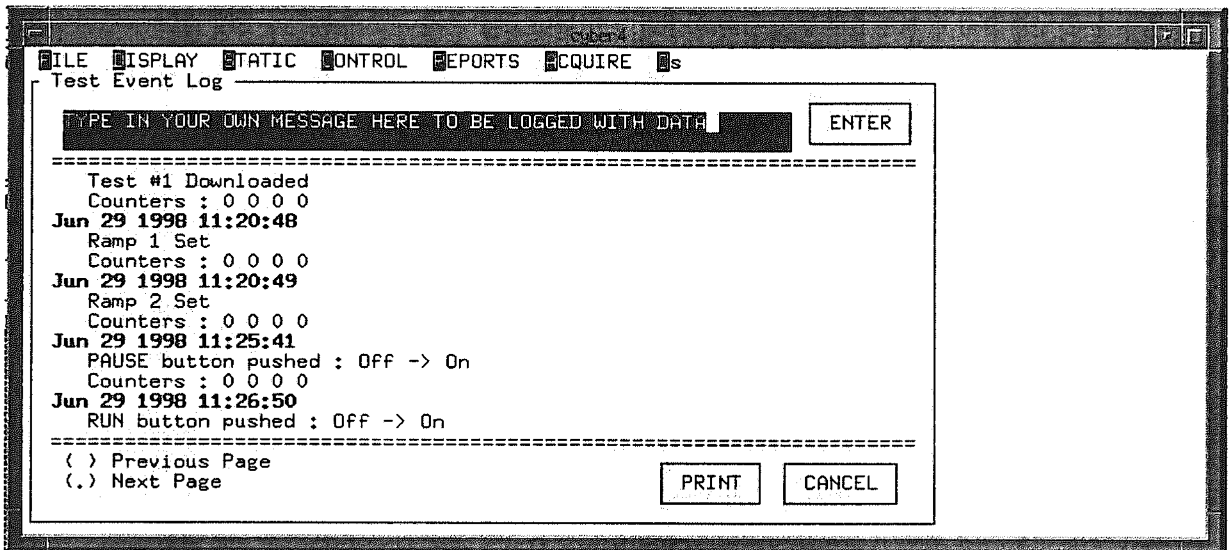


Figure 14-1 Events Log Display

EXAMINING THE SERVO HISTORY DATA

As you will recall from **step 7** in **Chapter 6** of this manual, each servo controller includes a built-in history rotating data buffer to maintain a record of the data leading up to a test article failure or other system/test event. This rotating data buffer stores 4,096 data points for each of 4 parameters per channel (16,384 total). The four parameters recorded in the history rotating data buffer include command, feedback, loop error, and valve current. Data in this buffer is continuously over written until "frozen" as part of a conditional action. Thus, at any time, this buffer always contains the most recent data.

The rate at which the data was recorded in the buffer, as well as the post-freeze percentage, depends on how you specified the history data recording parameters on the Start Up screen during set up. Refer to Chapter 6 for more information on how these parameter settings are used by the system.

If you have linked one or more system conditions to the"freeze" action given in the System Conditional Action Table, and a freeze history has been triggered, you may view the captured data by following the sequence of tasks given below:

TASK #1 TRANSFERRING THE SERVO HISTORY DATA TO THE HOST COMPUTER

Even though the system may have commanded the servo controllers to freeze their history rotating data buffers, you can not examine the history data until it is transferred to the host computer. To accomplish this data transfer, follow the steps given below:

Step 1) Select **MASTER** from the **CONTROL** menu.

Step 2) Using the **TAB** key, move the cursor to the **Send Freeze Data** checkbox and press the **space bar**.

Step 3) Press [**RETURN**]. If the system is in a non-running mode (e.g. Pause, Hold, etc.), it will display the dialog box shown in **Figure 14-2**, prompting you to enter the range of servo channels you wish to transfer data from. The system transfers the history data in each servo controller (one channel at a time) to the host for recording to the hard disk.

****************************** NOTE ******************************

Be sure to specify physical channel numbers. You must also make sure the channels you are requesting are included in the test.

****

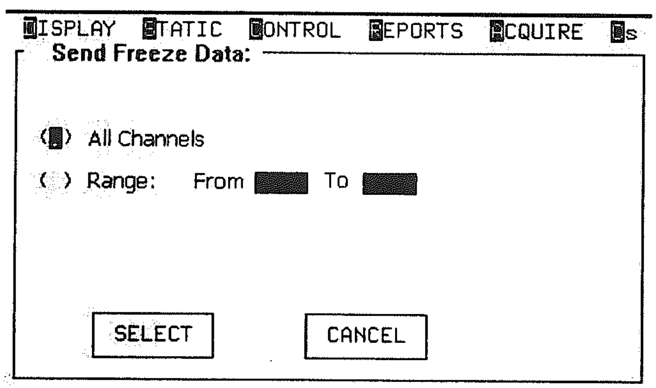


Figure 14-2 Dialog Box used to Initiate the Transfer of the Servo History Data

Step 4) The system will display a message which indicates that the transfer is taking place, along with the amount of time remaining before the history data for all of the selected channels has been transferred.

Step 5) Once the entire transfer is completed, you may examine the history data graphically in the form of a strip chart and/or as a tabular hardcopy report. Each format is described in the proceeding sections.

TASK #2 DISPLAYING THE SERVO HISTORY DATA GRAPHICALLY

If you have previously transferred the captured history data up to the host, you may now display this data in the form of a strip chart. The main steps you must complete are:

- define the vertical scale of the plotted data for each chart to be dispiayed.
- define the time range which represents the data of interest
- select up to four control channels to be displayed on the chart.
- select the signal for each chart; (feedback, error, command, or valve current).

Follow these steps to define the history strip chart parameters:

Step 1) Select **Graphs, Replay History Data**. The dialog box in **Figure 14-4** appears.

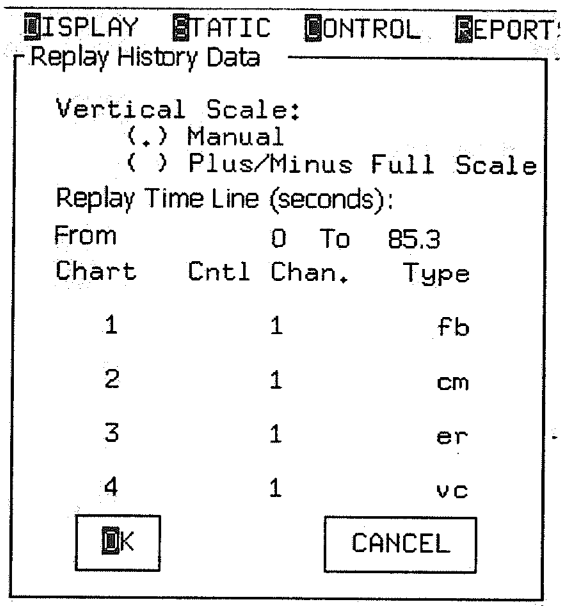


Figure 14-4 Dialog Box used to setup a Replay History Data Strip Chart

Step 2) Using the "**Vertical Scale**" radio button, select how the vertical scale of each strip chart is to be displayed.

If "**manual**" is selected, another dialog box will appear later on which will prompt you to enter the desired minimum and maximum vertical scales of each selected strip chart channel. If "+- Full Scale" is selected, the vertical scale for the selected channels will range from positive full scale to negative full scale. Please note that full scale for each selected channel refers to the corresponding engineering unit full scale value entered in the Channel Description screen.

Step 3) You must now indicate the range of data (in terms of time) that you are interested in displaying. This is accomplished by entering a beginning time and ending time in the entry labeled **Replay Time line (seconds):.**
The default range is from 0 to the total time to fill the rotating data buffer. The total time to fill the rotating data buffer will vary according to the **Record Every "N"** Waypoints entry on the Start Up screen prior to download. Using the default range will provide a strip chart display which represents the entire contents of the history data buffer.

Step 4) Using the "**Log Chan**" column, enter the logical channel number for up to four simultaneous scrolling strip charts in a single window. For each chart, specify the control parameter to be plotted by entering one of the following labels in the "**Parameter**" box:

- **fb** (feedback)
- **cm** (command)
- **er** (loop error)
- **vc** (valve current)

If less than four channels are to be plotted, simply leave the remaining "**Log Chan**" entries empty.

Step 5) Select **DISPLAY** and press **RETURN**. If "**Manual**" scaling was the selected scaling method, another dialog box will appear, prompting you to define the vertical axis for each chart. Fill in the required information and press **RETURN**. A strip chart similar to the one shown in **Figure 14-5** will appear on the X-terminal.

The x-axis shows the selected time range. Additionally, a vertical line indicates the point in time at which the fault occurred (i.e. Fault#1 or Fault#2). For the example display shown in **Figure 14-5**, the entire buffer (all 4,096 samples) is displayed.

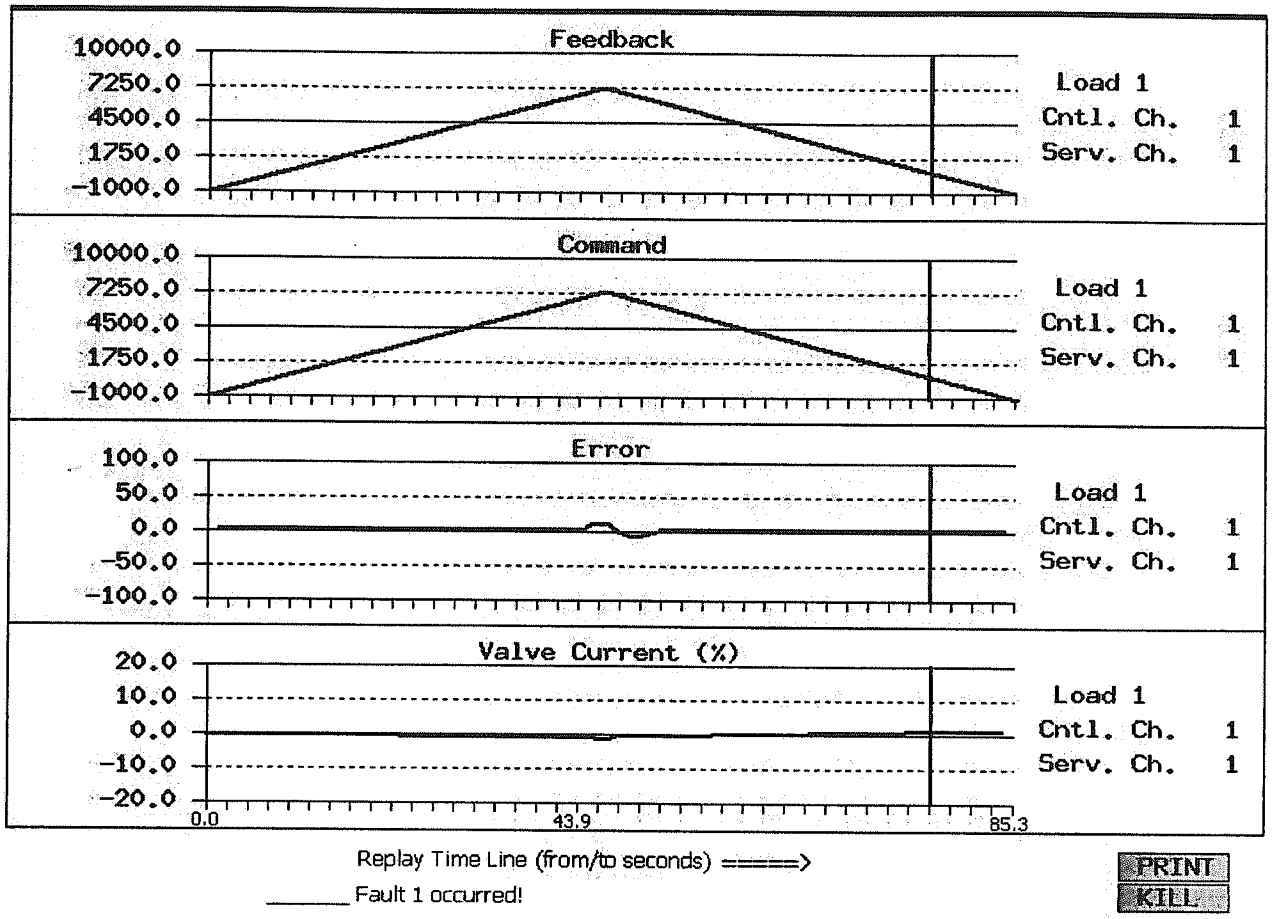


Figure 14-5 Strip Chart Display with Full Time Range

Step 6) If you wish to examine a moment in time more closely (e.g. around the fault), simply kill the graph and select **Graphs, Replay History Data** again. This time, enter a beginning and ending time which more closely suit the moment in time you are interested in.

Step 7) To view other channels, repeat **steps 1** through **6** using the desired servo channel numbers.

****************************** NOTE *********************************

If you repeat steps 1 through 6 for different servo channels, you must make sure that the history data for the newly selected channels was previously transferred to the host computer. If not, you must first transfer the history data for the selected channels as described in the previous section.

****

Obtaining a Hardcopy of the Servo History Data Strip Chart

In order to obtain a hardcopy of the history data strip chart, use the left-most mouse button and click on the [**PRINT**] selection at the lower right-hand side of the screen. For more information on how hardcopies of graphic screens refer to Chapter 12.

Killing the Strip Chart

To kill a particular strip chart window, move the mouse arrow to the "**Kill**" button in the lower left-hand corner of the window and press the right-most button on the mouse. That graphic window will disappear, but all other windows on the X-Terminal will remain.

TASK #3 OBTAINING A TABULAR REPORT OF THE SERVO HISTORY DATA

If the servo history data has been transferred to the host and you wish to get a tabular hardcopy report of this data, follow the steps described below:

Step 1) Select **Replay History Data** from the **REPORTS** menu. The dialog box shown in **Figure 14-7** appears.

Step 2) Enter the physical channel# for the selected channel.

****************************** NOTE **********************************

Be sure to enter the physical channel number when requesting this report.

****

Step 3) Enter the range of way-points to be included in the report. Remember, the servo controller's rotating history data buffer has a total of 4096 sample points, each of which includes the command, feedback, error, and valve current readings. If you want all the samples, enter a range from 1 to 4096.
If you have set up the post-freeze parameter on the Start Up screen, and you don't wish to see all the data, you may ask for a range of samples around the freeze point. For example, if the post-freeze parameter is defined as 50%, the first 2048 samples represent data prior to the freeze event, while samples 2049 to 4096 represent data following the freeze event. Therefore, if you were only interested in data closest to the freeze event, you might ask for a waypoint range from 2000 to 2100.

Step 4) Press **[RETURN]**. The report should begin printing shortly

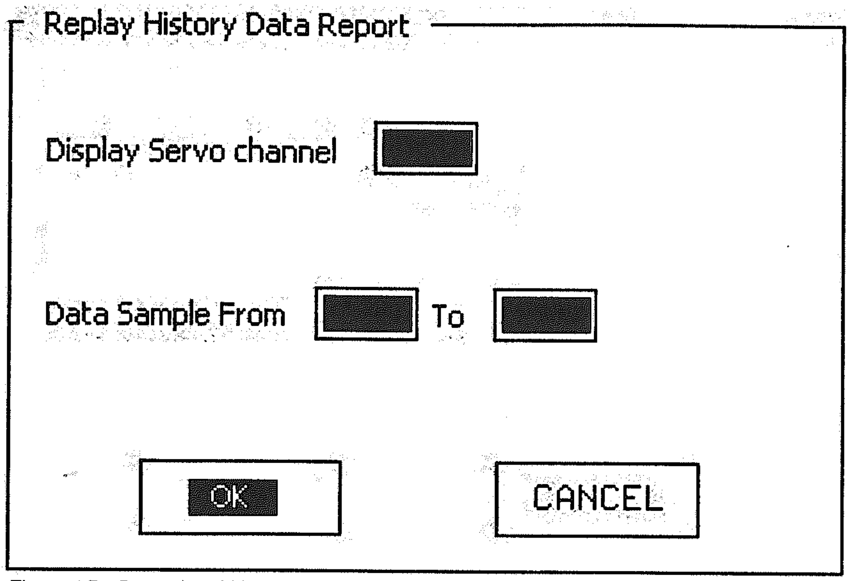


Figure 14-7 Requesting a Tabular Report of the Servo History data

EXAMINING THE DAS HISTORY DATA

If your test includes one or more ADAS channels and you have programmed the conditional action table(s) to **"freeze history"** based on one or more conditions, you may obtain a hardcopy report of the DAS data before and after the freeze event. This report provides a printout of a 3 minute DAS Rotating Data Buffer (i.e. disk file) stored on the host computer's hard disk under "/usr/dlcs/dasfrz/<testname>". The file consists of the 900 most recent scans of DAS data taken at 200 millisecond intervals. A report is allowed only if the DAS rotating data buffer receives **a "Freeze History"** command. This is the same **"Freeze History"** which gets sent to the servo controllers based on entries in the System or Discrete Conditional Action Tables.

Once the DAS rotating data buffer receives a freeze command, it continues to update with new scans such that the **"% Post Freeze Capture"** entry on the **Start Up** screen is satisfied. Thus, if the **"% Post Freeze Capture"** entry is set to 50%, the DAS rotating buffer will take 450 more scans (i.e 50% of the 900 total scans, approximately 1.5 minutes of data) before it stops updating.

When the DAS rotating buffer is finished updating you may request a report as described below:

Step 1) Select the **REPORT** menu, **Replay DAS Data** option. The dialog box shown in **Figure 14-9** appears.

Step 2) Enter the Data Sample Range (from 1 to 900).

Step 3) Enter the number for the desired DAS channel.

Step 4) Press **RETURN**. A tabular report will be printed of the requested DAS channel, plus the five subsequent DAS channels in the setup list. Also, the sample at which the freeze occurred will be marked by an asterisk.

********************************** NOTE ******************************

If you request a report before the DAS rotating buffer receives a freeze command, the message "DAS data has not been frozen" will be displayed. Also, if a report request is made while the DAS rotating buffer is updating with the **"% Post Freeze Capture"** samples, the message **"DAS data freeze currently in progress - try again in X seconds"** is displayed, where X is the amount of time remaining for the DAS rotating buffer to stop updating.

****

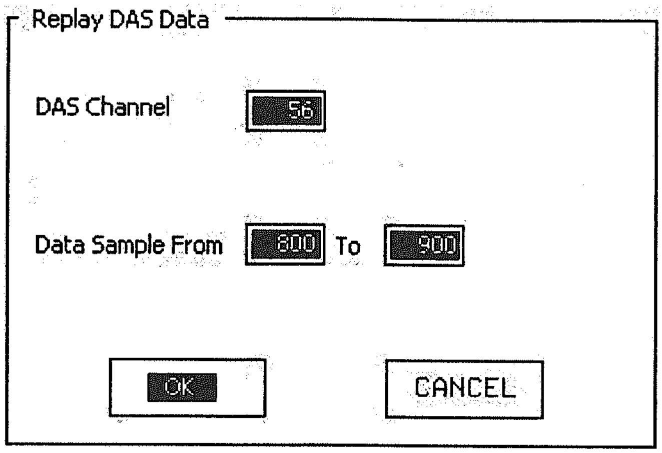


Figure 14-9 Requesting a DAS History Data Report

??
BEFORE PROCEEDING TO THE NEXT CHAPTER, REMEMBER TO CHECK APPENDIX A FOR SPECIAL FEATURES OR MODIFICATIONS WHICH MAY BE ASSOCIATED WITH YOUR PARTICULAR SYSTEM.
??

CHAPTER 15

ABORTING THE TEST

This chapter explains how to abort an active test.

************************************* [illegible] ************************************

If you have downloaded a test and no longer wish to run that particular test, or if a test runs to completion, you must perform the "Abort Test" procedure. Simply closing a test file does not cause the system to remove it from the TMTM's memory. In fact, such a test is still active.

****

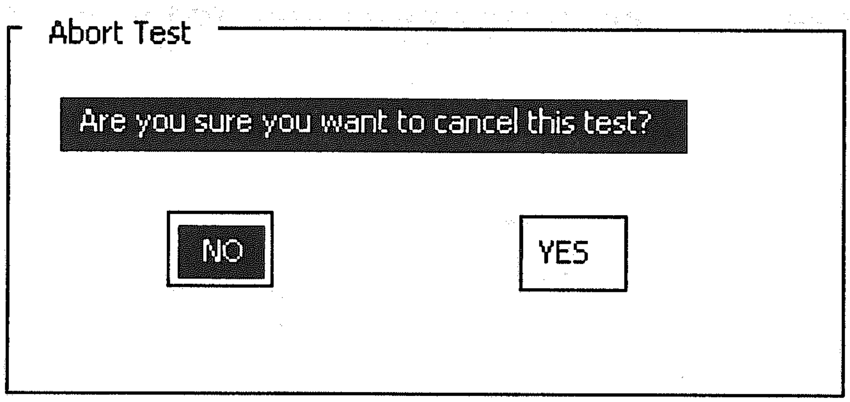


Figure 15-1. Abort Test Screen

To end the execution of the test and remove it from the TMTM memory, follow the steps given below:

Step 1: Make sure the test is not running (i.e. paused, held, or ramped) and is in a hydraulic safe condition, i.e. ramped to zero load.

Step 2: Select the **Abort Test** option in the **CONTROL** menu. The Abort Test screen, **Figure 15-1**, is displayed. A dialog box on the screen prompts "Are you sure you want to cancel this test?".

Step 3: If you wish to follow through with aborting the test, tab to [**YES**] and press [**RETURN**]. The system returns to the main menu bar.

Step 4: If you wish to make changes to the setup files or spectrum, you may do so. However, you must re-download the test again as described in Chpater 6.

Step 5: If you wish to close the test at this point, select **CLOSE** from the **FILE** menu. The test file is now closed.

Step 6: If you wish to leave the **CYBER** application software in order to enter the UNIX Operating System or to powerdown the host computer, select **QUIT** from the **FILE** menu. You are now in the UNIX Operating System. Refer to Appendix M if you wish to powerdown the host computer.

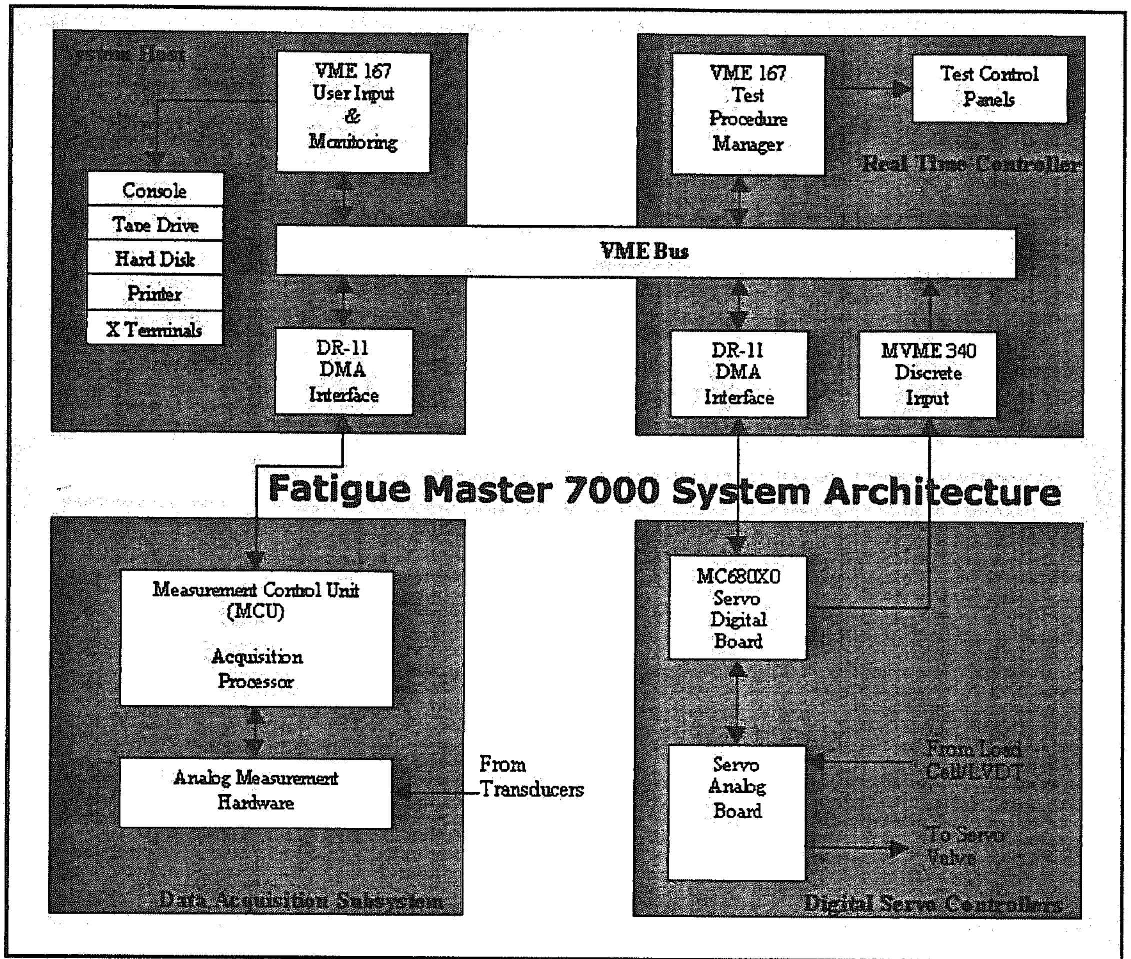
Cyber Systems, Inc.

Fatigue Master 7000

Digital Load Control System

OPERATOR'S MANUAL APPENDIX A

For Northrop Grumman



Fatigue Master 7000 Block Diagram

Purpose

The purpose of this addendum is to supplement the FM7000 Operator's Manual with features unique to the application used by Northrop Aircraft

It will cover the following aspects of system operation:

Section I	**Manual Goto Percent Load.**
Section II	**Excel Data Reports.**
Section III	**About WS_FTP Client**
Section IV	**Lag Lead Compensation**
Section V	**Tare Balance**

Northrop 40 Channel Digital Load Control System

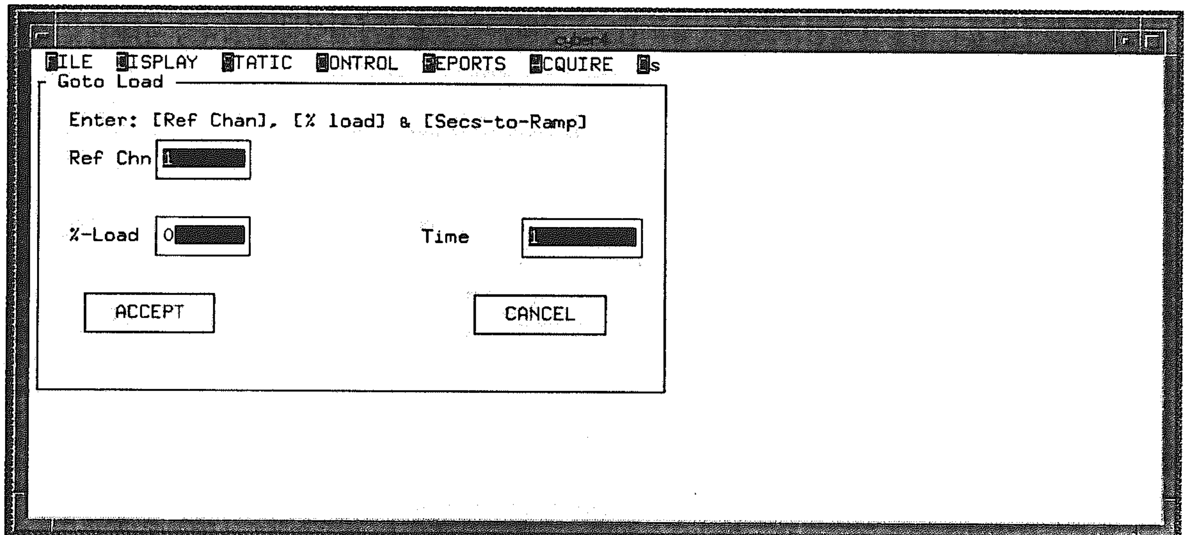
User Login:	fmlcs
Password:	cybsys1

MANUAL GOTO PERCENT LOAD

Similar to the ***Branch to Step*** feature, also located under the **Control** Menu, this selection allows the operator to specify a Percent Load value to which the Servo will ramp to after the **RUN** button is pressed.

Requirements:

- Must be a Static Test
- Load Conditions must be defined in Percent Load. (This mode is selected by going to ***Display=>Channel Parameters=>Channel Definitions***, and entering "***P***" under the ***LC*** column.)
- Percent Load value entered must be defined in the Load Condition Table.
- Step to Load Condition must be defined in Test Procedure



Goto Dialog Box

1. **Ref Chn:** Is the logical servo channel to be used for the Load Condition Table Lookup. In a multiple servo channel test this would be the servo channel that you wish to reference.
2. **%-Load:** Percent load value to look-up.
3. **Time:** Ramptime in seconds. Must be greater or equal to .25 seconds.
4. **Accept:** Execute ***GOTO*** function. You will be prompted with entered information and asked whether you wish to proceed. If the operator continues the lookup will take place and any errors conditions from the lookup will be displayed. If there are no errors the lookup is successful and the operator can then press **RUN** on the Test Control Panel which will cause the system to ramp to the specified Percent Load Condition.
5. **Cancel:** Cancel current request and close window

Operation:

Define Static Test:

1. Create a *New Test* and select *Static Test* from the New Test dialog.
2. Define Servo Channels and select Percent Load for the Load Condition Table.
3. Define the *Load Condition Table* and *Test Procedure Table* either manually or using the Static Mode Software to automatically create the test.

Tip:

Define Load Conditions and Test Procedure steps to cover all loads. I.E. -100 to 100 in 1 percent increments. This allows the operator to select any Percent Load value on the fly.

Run the Test

After the Setup is complete download the test to the Servo Controllers using the Start-up dialog. When the *Test Control Panel* appears, press the **PAUSE** button to place the system into a *Single Step* mode. Perform any Calibration and balancing that may be required and optionally ramp up to the 1st load condition.

Select ***Control=>Goto Load,*** to activate the ***Goto*** Dialog box.

Enter the desired information in the dialog box and select **ACCEPT.** Answer **Yes** to the User Prompt. The system will verify the user input. The goto will fail and the user notified if.

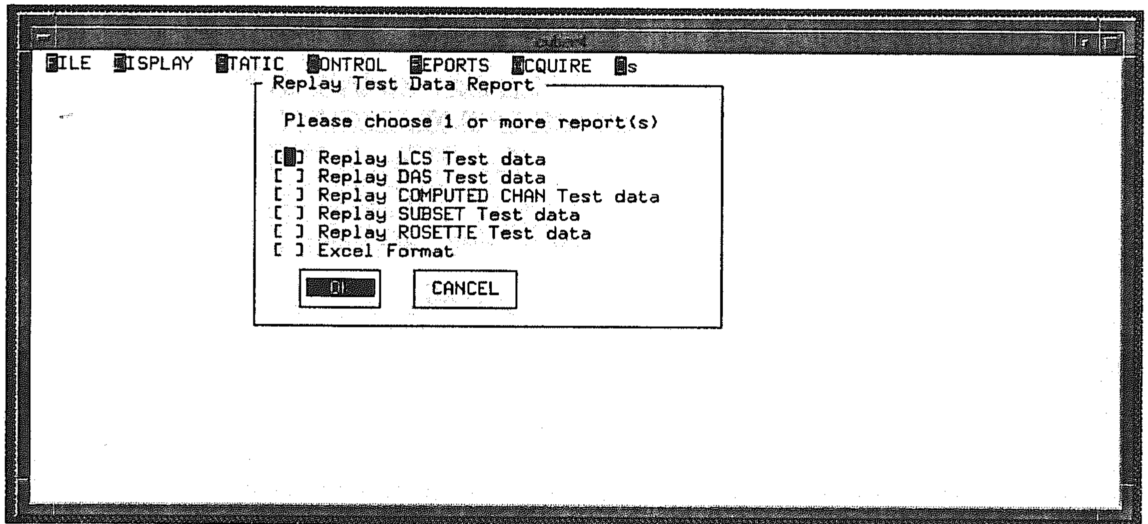
1. The system is not in a **Pause** or a **Hold** state
2. The ***Reference Channel*** was not found
3. The ***Percent Load*** value was not found in the *Load Condition Table.*
4. The ***Load Condition*** was not found in the *Test Procedure.*
5. The ***Time*** value entered was less than .25 seconds.

No response from the system will occur if the lookup was successful and the user is free to press the **RUN** button on the *Test Control Panel* to cause the system to ramp to the specified load. The ***Goto*** Dialog box will remain on the screen to allow the user to quickly enter the next load condition and will remember the last ramptime entered. The user can close this window by either selecting the **CANCEL** button or hitting the **ESC** key on the keyboard.

TEST DATA IN TABULAR FORMAT EXPORTABLE TO EXCEL

Test data generated by the FM7000 Load Control System at Endpoint may be formatted for easy importation to Microsoft's Excel Spreadsheet program.

Replay Test Data



Replay Test Data Report Screen

An Excel formatted file may be generated by selecting ***Replay Test Data*** from the ***Reports*** Menu. Select the data that you wish to include in the report and select ***Excel Format***. If ***Excel Format*** is not selected the report will output to the Printer.

After the operator selects **OK**, the system will ask for a range of counter values to include in the report.

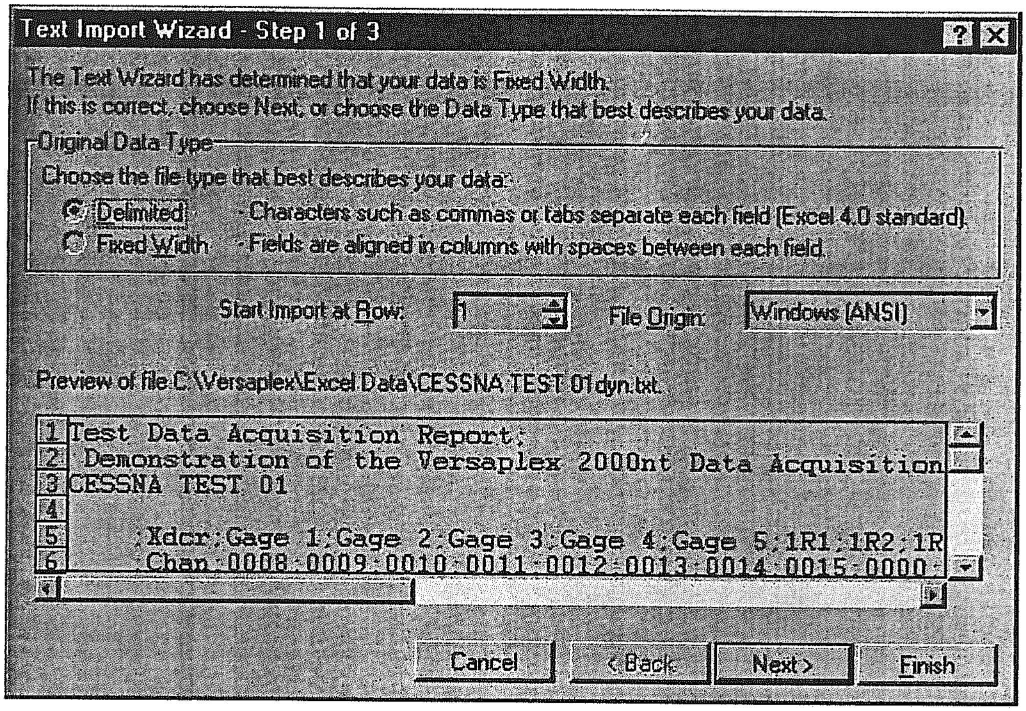
EXPORTING THE EXCEL FILE TO A PC

Transferring the excel formatted file to the PC is accomplished by the use of an FTP Client program. **WS_FTP** is such a client that is available on your Load Control PC system and the operator should familiarize himself with its operation.

Follow these steps using **WS_FTP** as a example.

1. Log on to LMSW using the FTP client
2. Navigate to **/tmp** directory on the Unix box
3. In this directory there may be a number of files. Files with the prefix of ***excel*** are the ones of interest. Look at the creation date of each excel prefixed file and determine which one has the data that you requested. Using WS_FTP you may rename this file either at this point or after you have transferred it to the PC. **IMPORTANT:** When you rename this file, give the file a **.txt** extension. I.E. JastRun1.txt. This is an extension that Excel knows how to handle.
4. Using **WS_FTP** transfer the file to the PC using the **ASCII** mode transfer, to a directory that you have prepared.

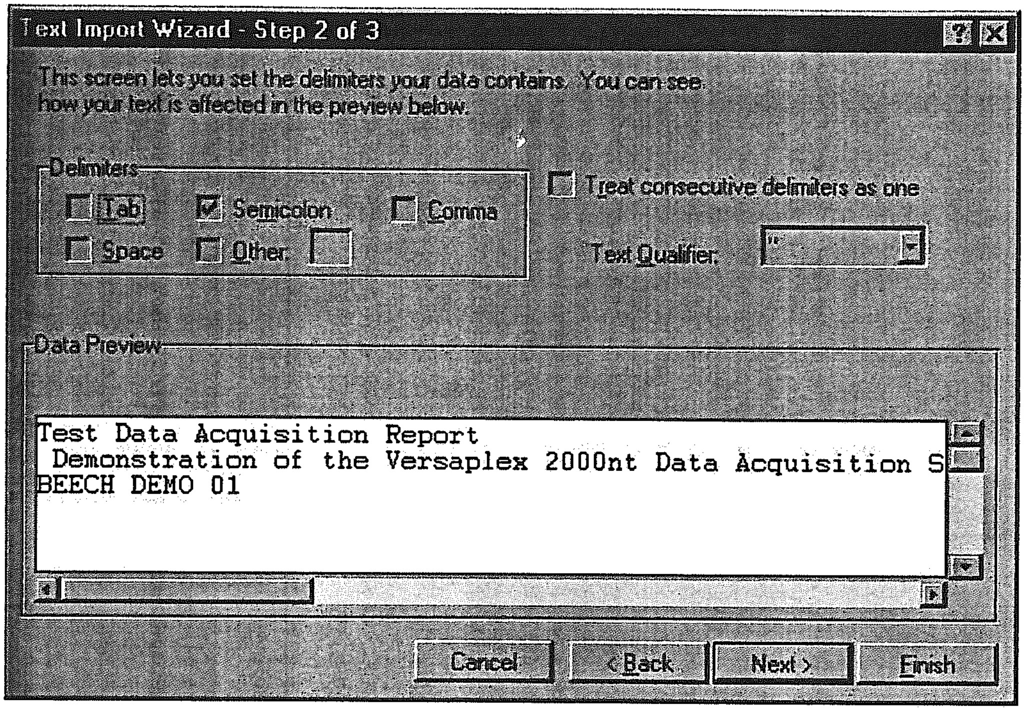
OPENING THE FILE IN EXCEL.



After using Excel to open the desired test;
Example:
C:\EXCEL DATA\JastRun1.TXT

The test operator will be presented with *Excel's Text Import Wizard.* Select **Delimited** as the *Original Data Type* and then press the **Next>** command button.

OPENING THE FILE IN EXCEL: (CONT.)



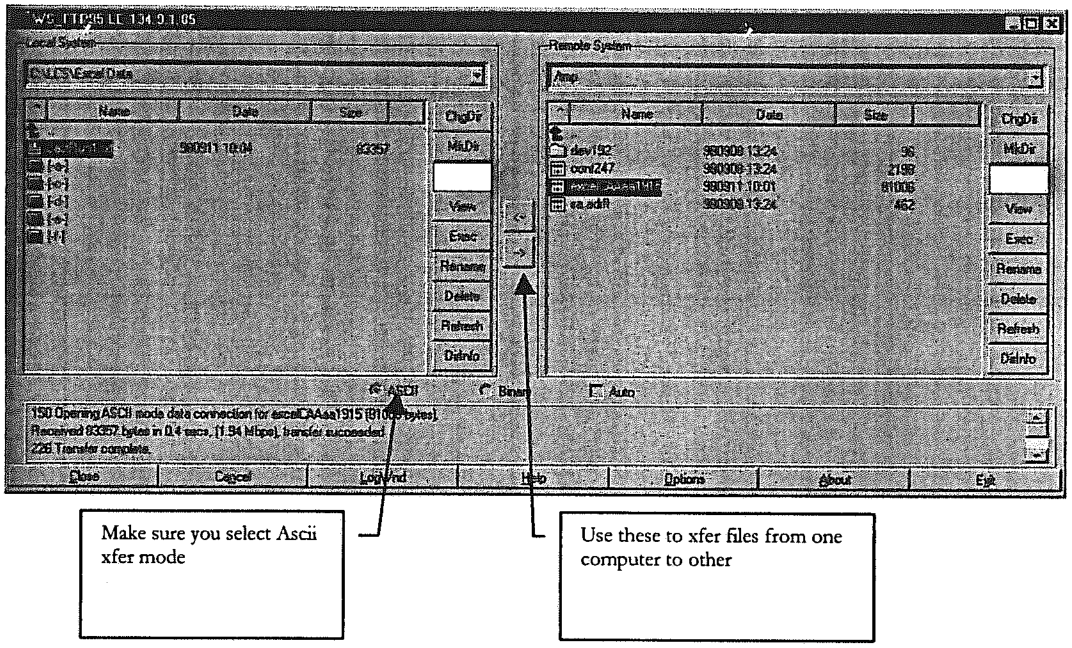
Step 2:

Select **<u>Semicolon</u>** from the list of **<u>Delimiters</u>** and press the **<u>Finish</u>** command button.

Excel will now open the file and the data will be formatted as *Row* and *Column* data.

WS_FTP - FTP CLIENT PROGRAM

WS_FTP is a FTP client program that is located on the PC used as the X-Windows terminal to your Unix box.



Description

This application is a File Transfer Protocol (FTP) client application for Windows Sockets and was designed to take full advantage of the point and click capabilities of the Windows environment

Lag – Lead Compensation

Each servo controller contains a feature called Lag Lead Compensation. This compensation occurs in the forward path of the control loop, which is between the loop error junction and the valve drive output. This path is usually just proportional gain, which is gain that the operator can set to control loop response and accuracy. This gain is normally set as high as possible to get best DC accuracy and quick response, but is limited because excessive gain causes loop oscillations.

The response of the Lag Lead Compensation is controlled by the Lag Frequency and the Lag Ratio which are entered on the Channel Control screen. A pole is at the Lag Frequency and creates a 6db per octave slope extending to the recovery (lead) frequency. Due to the 6db per octave slope, the ratio of the lag and recovery frequencies is the same as the ratio of the gain above the recovery frequency to the gain below the lag frequency. The Lag Ratio can be visualized either way.

The advantage of Lag Lead compensation is that the loop gain at DC and low frequencies can be made higher than the proportional gain can be set. Suppose there is an offset in the hydraulic servo valve of 10%. This means that a valve current of 10% of the rated current is required to set the valve to zero flow, which is what happens in a static condition. If, due to stability limitations, the proportional gain could not be set higher than five, the loop would have a static error of 2%. This 2% error times the loop gain of five creates the required 10% valve current.

If Lag Lead Compensation was used, a lower static error could be achieved. Suppose a Lag Frequency of 0.2HZ and a Lag Ratio of 4 was set. This means the DC gain will be four times higher than the proportional gain setting. In the example above, the DC forward gain will become 20 and the recovery frequency will be at 0.8HZ. This means the static loop error will go down to 0.5%.

Increasing the Lag Ratio alone has the effect of increasing the phase shift associated with the Lag Lead response. The further apart the lag and recovery frequencies are (higher lag ratio) the greater will be the phase shift associated with the lag lead response. Increasing the lag ratio also has the effect of moving the recovery frequency up closer to the loop bandwidth. The effect of this is to lower the loop phase margin, and therefore the stability of the control loop.

To maintain good loop stability, the lag frequency can be lowered when lag ratio is raised. In the above example where the lag frequency is 0.2Hz and the lag ratio is four, the recovery frequency is 0.8HZ. Lowering the lag frequency to 0.1HZ and raising the lag ratio to 8 keeps the recovery frequency at 0.8HZ.

The control accuracy during fatigue test cycling at or below the Lag Frequency can also be improved by the use of Lag Lead Compensation due to the increased loop gain at the cycling frequency.

The response of the Lag-Lead network in the open loop gain path of the servo controller is as shown in **Figure A.** The benefit of this is that the maximum loop gain that the loop can tolerate and still have good loop frequency stability does not have to be the maximum dc loop gain. Additional gain, and therefore static and dynamic accuracy, can be created by the Lag-Lead compensation without spoiling the loop frequency stability.

The transfer function of the Lag-Lead network is as follows.

$$\text{Open loop gain} = LG * \frac{Wz}{Wp} * \frac{(S + Wz)}{(S + Wp)}$$

Where

LG is the Loop Gain set by the operator.

Wp is the Lag frequency in radians per second

Wz is the Lead frequency in radians per second

S is the signal frequency in radians per second

$\frac{Wz}{Wp}$ is the Lag Ratio

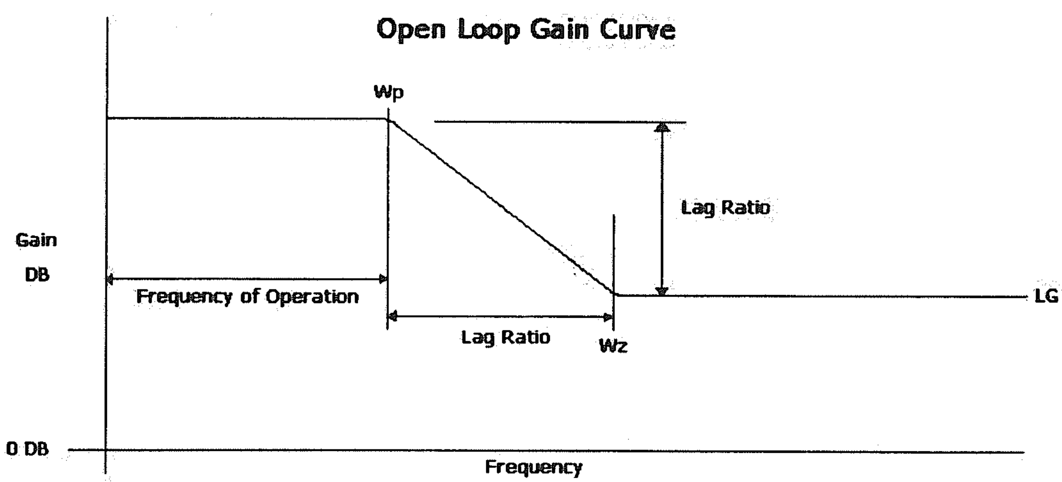


Figure A

Tare balance

The Tare Load is the load applied to the test article due to the weight of the fixture between the load cell and the test article.

The Tare Balance routine adjusts the offset of the load cell output so that it corresponds to the load applied to the test article instead of the load seen by the load cell.

POLARITY DEFINITION OF TARE

If a positive command to the servo control loop causes a force on the test article in the same direction as the tare weight, then the tare load is positive and is entered as a positive value.

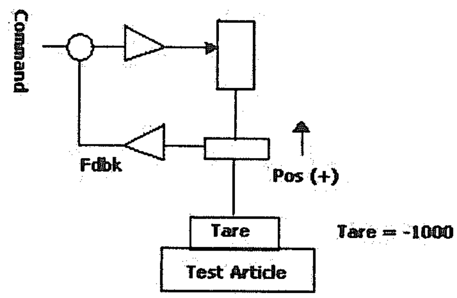
CALCULATION OF THE TARGET ADC VALUE

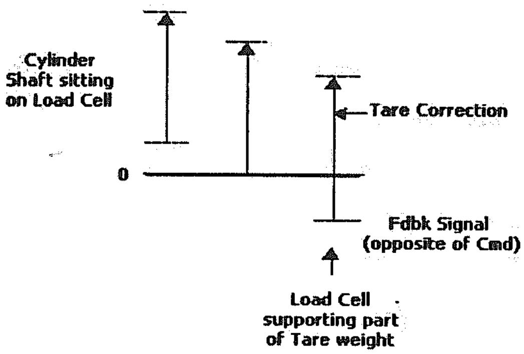
1. It is assumed that a **Zero Balance** and an **RCAL** have already been done on the load cell with it unpinned from the fixture. The value set into the balance **DAC** in the **Zero Balance** routine is to be saved forever until a new **Zero Balance** routine is run.

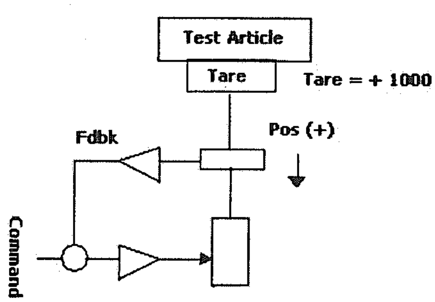
2. At the start of the Tare Balance routine, set the **Balance DAC** to the **Zero Balance** value. Read the **Feedback ADC** value. This is **Initial FBK**.

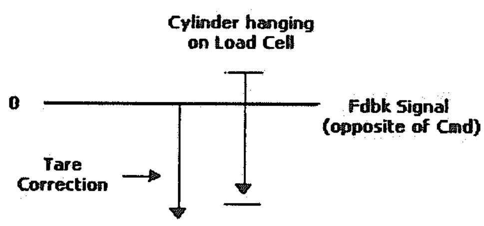
3. The equation for the **Target Tare ADC** value in the Tare Balance routine is as follows.

Target Tare ADC value = **Initial FBK** - (2000 * (tare load in EU)/(Servo Full Scale in EU))


Command
Fdbk
Pos (+)
Tare
Tare = -1000
Test Article


Cylinder Shaft sitting on Load Cell
Tare Correction
0
Fdbk Signal (opposite of Cmd)
Load Cell supporting part of Tare weight


Test Article
Tare
Tare = + 1000
Fdbk
Pos (+)
Command


Cylinder hanging on Load Cell
0
Fdbk Signal (opposite of Cmd)
Tare Correction