HP 1652B/HP 1653B Logic Analyzers

Getting Started Guide

HEWLETT PACKARD

Getting Started Guide

HP 1652B/HP 1653B Logic Analyzers

HEWLETT PACKARD

Manual Set Part Number 5952-0176

Printed in the U.S.A. January 1990

Introduction

About this book...

Welcome to Hewlett-Packard logic analyzers. The HP 1652B and HP 1653B Logic Analyzers are more than just another logic analyzer. They are an analyzer and oscilloscope in one instrument. With this combination, you have expanded measurement capabilities.

The user interface of the HP 1652B/HP 1653B was designed to be as easy to operate as possible. The use of "pop up" windows help lead you through setups and measurements without having to memorize a lot of steps. As you read this guide and the other references about the logic analyzer, you will see just how easy to use the HP 1652B/HP 1653B really is.

We will not cover every feature and function of the HP 1652B/HP 1653B Logic Analyzer in this guide. The purpose of this *Getting Started Guide*, is to get you going quickly, by giving you the basic user interface information along with a couple measurement examples. The detailed information will be found in the *HP 1652B/HP 1653B Front-Panel Operation Reference.*

If you are new to logic analyzers and digitizing oscilloscopes, or just need a refresher, we think you will find *Feeling Comfortable With Logic Analyzers* and *Feeling Comfortable With Digitizing Oscilloscopes* valuable reading. These books will help eliminate any misconceptions or confusion you may have about the analyzer's applications, and will show you how to get the most out of it's measurement functions.

How to use this guide

This guide is organized into two types of information. The first three chapters cover basic user interface information. If you are an experienced HP logic analyzer user, but new to this family of logic analyzers, you may want to read just the first three chapters and then go directly to the *HP 1652B/HP 1653B Front-Panel Operation Reference.*

The next three chapters simulate basic measurements in a task-oriented format. These example measurements, illustrate menu setup, in a step by step order commonly followed when solving digital system problems. If you are new to HP logic analyzers and logic analysis, you should include the measurement examples as part of your reading.

You will see illustrations of menu setups interspersed with text. If you are going through a sequence for the first time, you should refer to the illustrations while reading the text, to aid your understanding. If you are a fast learner and remember the first three chapters on the user interface, you can follow through the examples by just referring to the illustrations.

Contents

Chapter 1: Introducing the HP1652B/HP 1653B

Chapter 2: Getting to Know the Front Panel

Chapter 3: Gettin to Know the Main Menus

Chapter 4: Using the State Analyzer

Chapter 5: Using the Timing Analyzer

Chapter 6: Using the Oscilloscope

Chapter 7: What's Next?

Appendix A: Logic Analyzer Turn-on Check List

Appendix B: Making Hardcopy Prints

1

Introducing the HP1652B/HP 1653B

What Are the HP 1652B and HP 1653B?

The HP 1652B and HP 1653B logic analyzers are general-purpose logic analyzers with built-in oscilloscopes. If you're familiar with the HP 1652B/HP 1653B or any HP digitizing oscilloscope, you'll recognize many features that you've grown to know and love. If you're new to our analyzers and oscilloscopes, the HP 1652B/HP 1653B is easy to get to know.

The only differences between the HP 1653B and the HP 1652B, are the number of data channels and the speed of the state analyzer. The HP 1652B has 80 data channels and 35 MHz state analysis while the HP 1653B has 32 data channels and 25 MHz state analysis. Both analyzers have 100 MHz timing analysis along with 100 MHz single-shot and repetitive (repetitive single-shot) oscilloscope measurement capabilities. Because the differences are small between both instruments, they use the same set of manuals.

Some of the key features shared by both the HP 1652B and HP 1653B are listed below:

- Transitional or glitch timing mode on all channels.
- Simultaneous state/state or state/timing modes.
- Glitch detection on all channels.
- Marker measurements.
- Timing and Scope Autoscale.
- Scope Automeasure.
- Pattern, edge, and glitch triggering.
- Overlapping of waveforms.
- Small lightweight probing.
- Time and number of states tagging.
- State Compare, Waveform, and Chart.

This *Getting Started Guide* covers only a few of the logic analyzer's features. You will find details of all the features of the HP 1652B and HP 1653B in the *HP 1652B/HP 1653B Front-Panel Operation Reference Manual.*

Getting Ready to Operate

If you have just unpacked your new HP 1652B/1653B logic analyzer, please take a few minutes to completely read this chapter. It tells you how to prepare your logic analyzer for applying power and turning it on. If you are learning how to use the logic analyzer and it is already turned on, start with chapter 2, "Getting to Know the Front Panel".

Initial Inspection

Inspect the shipping container for damage. If the shipping container or packaging materials are damaged, you should keep them until the contents of the shipment have been checked for completeness and the instrument has been checked mechanically and electrically.

If there is damage to the instrument, refer to the service manual for the proper procedure for contacting the nearest service center.

Accessories

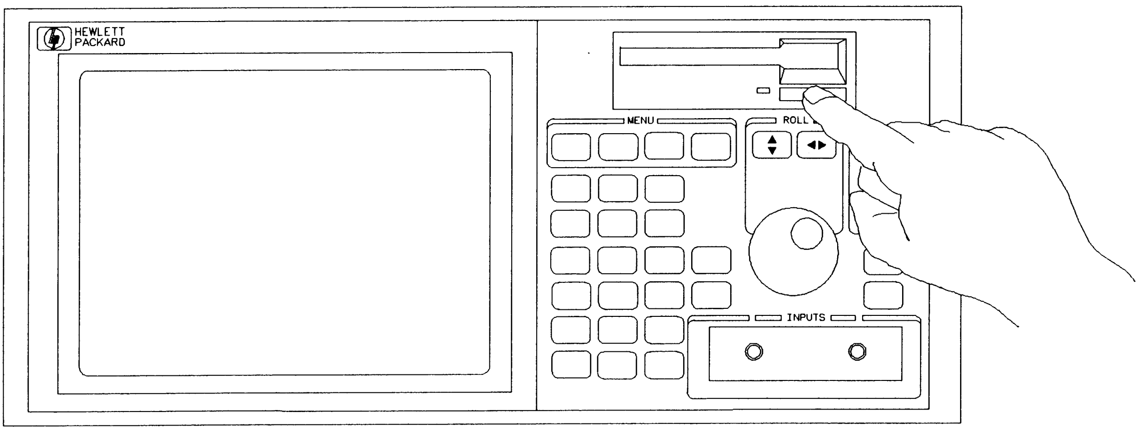
In addition to checking the instrument for damage, you should also check to see that the accessories supplied with it are complete.

The *HP 1652B/HP 1653B Front-Panel Operation Reference* Manual lists all the accessories shipped with the HP 1652B/HP1653B logic analyzers. If any of these items are missing contact your nearest Hewlett-Packard Sales Office.

Removing Yellow Shipping Disk

Your logic analyzer is shipped with a protective yellow shipping disk in the disk drive. Before you can insert the operating system disk you must remove the yellow shipping disk.

Slide the white locking tab to the right and press the disk ejection button as shown in the following figure. The yellow shipping disk will pop out part way so you can pull it out of the disk drive.



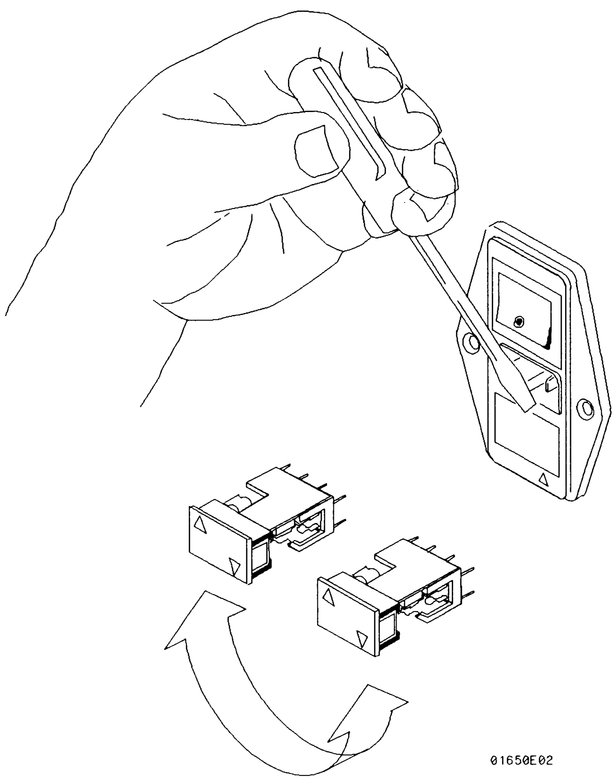
01652E06

Selecting the Line Voltage

The line voltage selector has been set at the factory to the line voltage used in your country. It is a good idea to check the setting of the line voltage selector so you can become familiar with what it looks like. If the setting needs to be changed, follow the procedure in the next paragraph.

Caution

You can damage the logic analyzer if the fuse module is not set to the correct position.

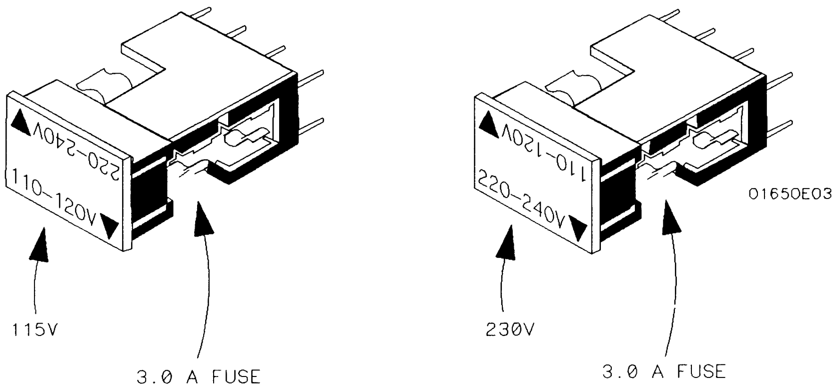


01650E02

You change the line voltage setting by pulling the fuse module out and re-inserting it with the proper arrows aligned. To remove the fuse module, carefully pry at the top center of the module, as shown in the figure above, until you can grasp it and pull it out by hand.

Checking for the Correct Fuse

If you need to check for the correct fuses, remove the fuse module and look at the amperage and voltage of each fuse. The following figure will help you locate the 115 V and 230 V fuses. To remove the fuse module, carefully pry at the top center of the module until you can grasp it and pull it out by hand. (Refer to, "Selecting the Line Voltage" on the previous page.)



Connecting the Power Cable

The HP 1652B/HP 1653B comes with a 3-wire power cable. When you connect the cable to an appropriate AC power receptacle, a ground is provided for the instrument cabinet. The type of power cable you receive with the instrument depends on your country. Refer to appendix D of the *HP 1652B/HP 1653B Front-Panel Operation Reference* manual for power cord types.

Warning

To avoid possible shock hazard, you must connect the instrument to a properly grounded 3-wire receptacle.

Operating Environment and Ventilation

You may operate your logic analyzer in a normal lab or office environment, but don't block its ventilation. You must provide an unrestricted airflow for the fan and ventilation openings in the rear of the logic analyzer. However, you may stack the logic analyzer under, over, or in-between other instruments as long as the surfaces of the other instruments aren't needed for their ventilation. If you intend to use it in another type of environment, refer to Appendix F in the *HP 1652B/HP 1653B Front-Panel Operation Reference Manual.*

Loading the Operating System

Before you can operate the logic analyzer, it must transfer its operating system from disk memory to its internal memory. This is called "loading the operating system" or "booting."

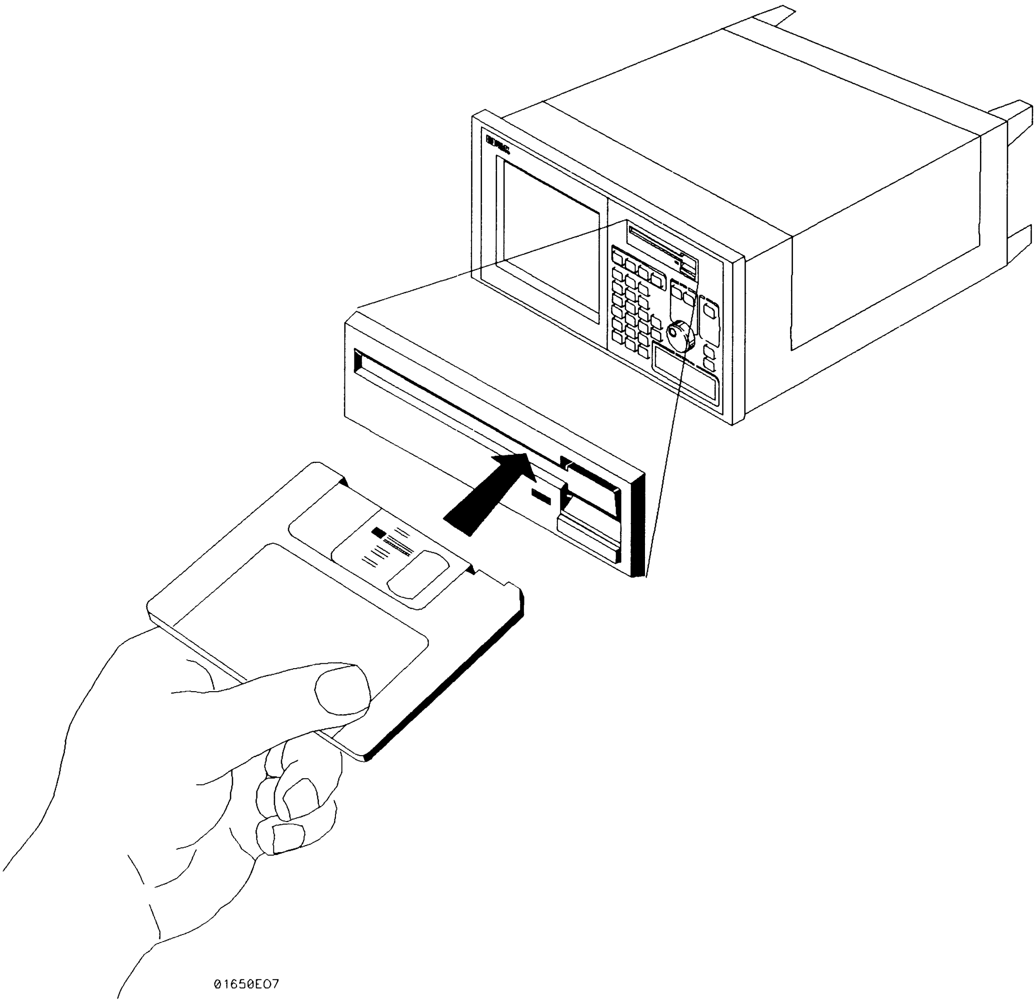
The logic analyzer operating system is a set of instructions that control the operation of the instrument. The operating system resides on a 3.5-inch flexible disk. You received two identical operating system disks. You should mark one of them **Master** and store it in a safe place. Mark the other one **Work** and use only the work copy. This will provide you with a back-up in case your work copy becomes corrupt.

Caution

To prevent damage to your operating system disk, DO NOT remove the disk from the disk drive while it is running. Remove it only after the indicator light has gone out.

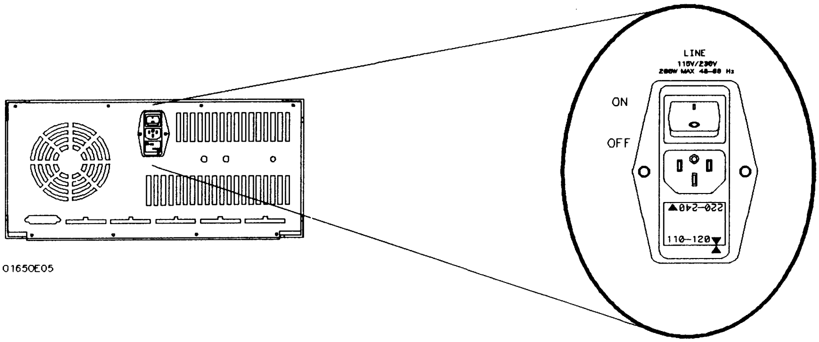
Installing the Operating System Disk

To load the logic analyzer's operating system, you must install the disk as shown below **before** you turn on the power. When the disk snaps into place, the disk eject button will pop out.



Line Switch

The line switch is on the rear panel. You turn on the logic analyzer by pressing the "1" on the rocker switch. Make sure the operating system disk is in the disk drive before you turn it on. If you forget the disk, don't worry, you won't harm anything. You will merely have to repeat the turn-on procedure with the disk in the drive.

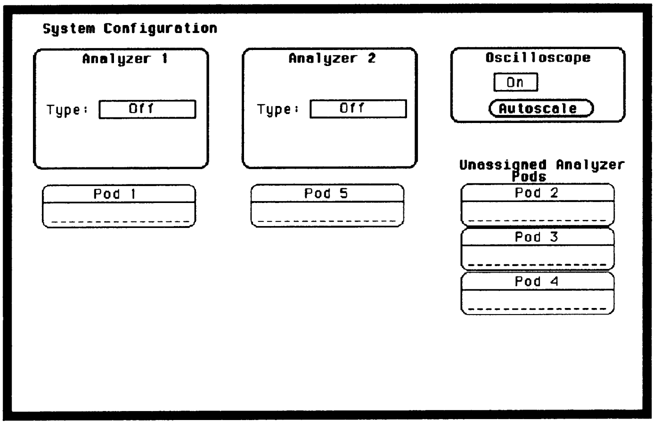


01650E05

Power-up Self-Test

When you turn on the logic analyzer, it performs a series of self-tests. When it has successfully completed these tests, it loads the operating system into memory from the disk.

When the logic analyzer has completely loaded the operating system it displays the System Configuration menu as shown below.

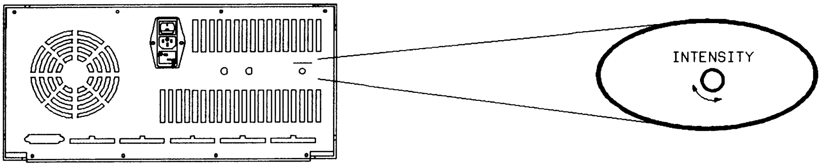


Note

This is the HP 1652B System Configuration menu. If you have an HP 1653B, the only difference is pod 1 will be assigned to analyzer 1 and pod 2 will be assigned to analyzer 2. There won't be any pods in the **Unassigned** area of the display.

Adjusting the Display Intensity

Once you have turned on the instrument, you may want to set the display intensity to a different level that's more comfortable for you. You do this by turning the INTENSITY control on the rear panel.



01650E06

Note

A high intensity level setting may shorten the life of the CRT in your instrument.

Summary

Now that you have unpacked, inspected, and begun operating the logic analyzer, the next step will depend on your needs. If you are a first-time logic analyzer user who wanted to get the instrument running before reading *Feeling Comfortable with Logic Analyzers* and *Feeling Comfortable with Digitizing Oscilloscopes*, you should read them now. If you are familiar with logic analysis, read either the rest of this *Getting Started Guide* or the *HP 1652B/HP 1653B Front-Panel Operation Reference* Manual.

2

Getting to Know the Front Panel

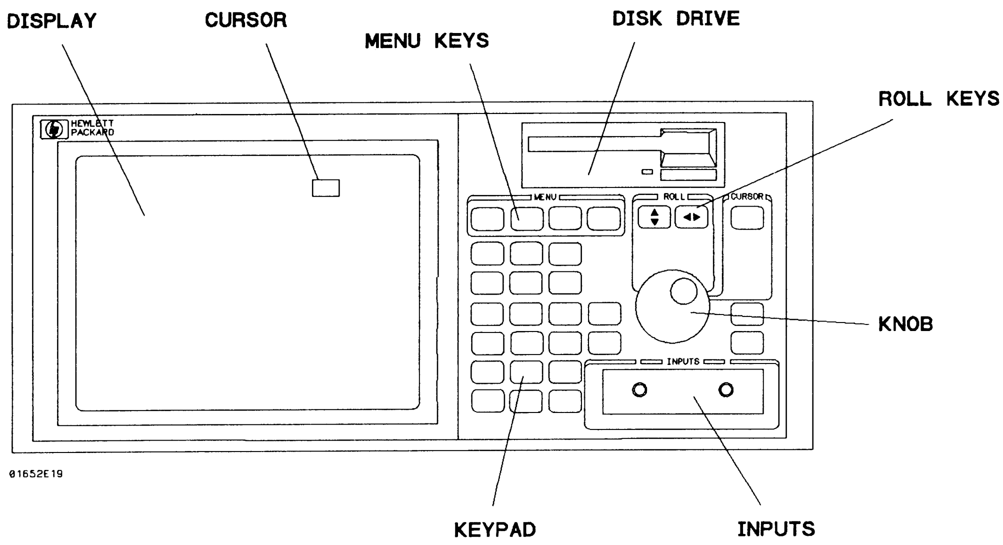
Introduction

The HP 1652B/HP 1653B logic analyzers have been designed to be very easy to use. The controls are located logically by function so you can learn how to use them quickly and easily.

This chapter divides the front panel into these functional areas and gives you an overview of each area.

Front Panel Organization

The functional areas of the front panel are the display, disk drive, menu keys, roll keys, cursor, keypad, knob and input.



Display

The display is where the various menu screens appear. Menu screens show analyzer and oscilloscope configurations as well as measurement results. Menus have many different functions you can execute. Most functions are activated by highlighting a field in the menu with the cursor, and then pressing the SELECT key.

Cursor

The cursor is a movable indicator on the display that allows you to access desired fields in each menu. It changes the field where it resides from the normal white background to the dark background (inverse video). The knob moves the cursor to the field (function) you wish to activate.

Menu Keys

The MENU keys allow you to quickly move between the major menus that are available. What menus are available depends on which part of the HP 1652B or HP 1653B you are currently using.

When the **FORMAT/CHAN** key is pressed, the following menus appear:

- State Format Specification (if you are in the state analyzer)
- Timing Format Specification (if you are in the timing analyzer)
- Channel (if you are in the oscilloscope)

When the **TRACE/TRIG** key is pressed, the following menus appear:

- State Trace Specification (if you are in the state analyzer)
- Timing Trace Specification (if you are in the timing analyzer)
- Trigger (if you are in the oscilloscope)

When the **DISPLAY** key is pressed, the following menus appear:

- State Listing (if you are in the state analyzer)
- Timing Waveforms (if you are in the timing analyzer)
- Waveforms (if you are in the oscilloscope)

When the **I/O** key is pressed, the same menu appears no matter which part of the HP 1652B or HP 1653B you are using. The I/O menu makes available all the Input /Output operations.

Disk Drive

The disk drive is used by the HP 1652B/HP 1653B to load the operating system every time the logic analyzer is turned on. You will use the disk drive to load configurations or store configurations, acquired data, and inverse assemblers for later use. The disk drive uses 3.5-inch flexible disks. More information on loading files is found in Appendix B of this guide. Complete details on the disk drive and its functions can be found in the *HP 1652B/HP 1653B Front-Panel Operation Reference* manual.

Roll Keys

When part of the listing or waveforms data is off screen, the ROLL keys define which way the knob will move the displayed data. You will use these keys and the knob to roll displayed data up/down or left/right to view data that is off-screen.

Knob

The Knob has four major functions depending on what menu or pop-up menu you are in. The KNOB allows you to do the following actions:

- Move the cursor from field to field within the System Configuration and main menus.
- Roll the display left or right and up or down.
- Position the cursor on options within pop-up menus.
- Increment/decrement numeric values in numeric entry pop-ups.

Inputs

The HP 1652B and HP 1653B has both front-panel inputs and rear panel inputs/outputs. The front panel INPUTS 1 and 2 are for signal input to the oscilloscope. BNC cables or oscilloscope probes can be used.

Logic analyzer probe inputs, labeled by POD and CLOCK, are located on the rear-panel. In addition, a Probe Compensation Signal, and EXTERNAL TRIGGER IN/OUT BNCs are on the rear-panel.

Keypad The keypad has keys for entering data as well as the **RUN** and **STOP** keys (for acquiring waveform data). When entering numeric data, use the **CHS** key to change the sign of the number. When entering patterns, the **DON'T CARE** key enters an X ("don't care") in place of a regular digit. Pressing **CLEAR ENTRY** replaces the entry with the default value.

Summary

Now that you are acquainted with the front panel organization, you will be able to decide where you want to go next. If you are just starting to learn logic analysis, you should read this entire guide. If you are experienced in logic analysis, you should continue to read chapter 3 to become more familiar with the operation of the front panel before you turn to the reference manual.

3

Getting to Know the Main Menus

Introduction

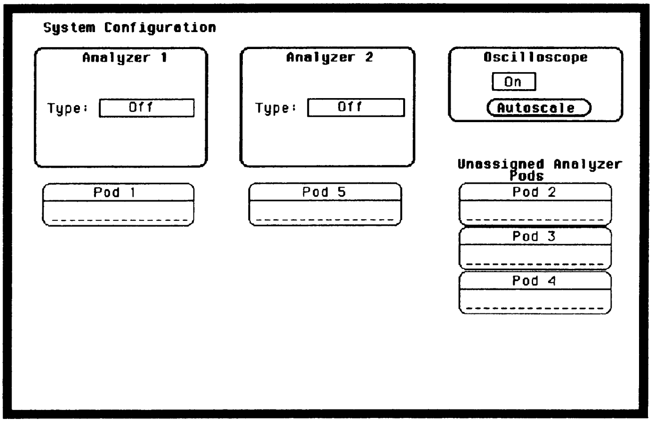
This chapter introduces you to the main menus of the HP 1652B and HP 1653B. The main menus are used to configure the logic analyzer and display the measurement results. In this chapter, you will learn how to move between the different machines (state analyzer, timing analyzer, and oscilloscope) and how to move between menus within each machine. You will also learn how to quickly get back to the System Configuration menu from any of the main menus.

In addition to learning the main menus, you will be introduced to some common pop-up menu types. There is more information on the use of all the pop-up types in the example exercises in chapters 4 through 6.

Exploring the System Configuration Menu

The first exercise in this chapter starts from the System Configuration menu. If you are not in the System Configuration menu as shown below, turn the power switch off, then back on.

An HP 1652B is used in this example. If you have an HP 1653B, there will only be two pods. The following procedure is the same for both logic analyzers.



Using the Cursor, Knob and SELECT Key

Most of the logic analyzer operation is initiated by placing the **cursor** on the field you want to interact with and pressing the **SELECT** key. To move the cursor from field to field, just turn the **knob.**

Depending on the field type (immediate execute or pop-up) pressing **SELECT** will either execute a function or open a pop-up menu.

Configuring the Analyzer

In the following exercise, you will use a selector type pop-up to assign Analyzer 1 as a state analyzer and Analyzer 2 as a timing analyzer. You will use the Alpha Entry pop-up to assign unique names to each analyzer.

1. Select the **Type:** field of Analyzer 1, then select **State** from the selector pop-up.

Notice, once you have assigned Analyzer 1 as State, a new field appears directly above the Type: field. This new **Name:** field, is where you will assign the analyzer's unique label.

2. Select the **Name:** field. An Alpha Entry pop-up will appear.
3. Use the knob to place the cursor over the letter you want, then press the **SELECT** key. As you repeat this step for all the letters in the label, the label will be spelled out, left to right, inside the bracket at the bottom of the pop-up.

To make changes or corrections in the Alpha Entry field, place the under-score marker under the character you want to change.

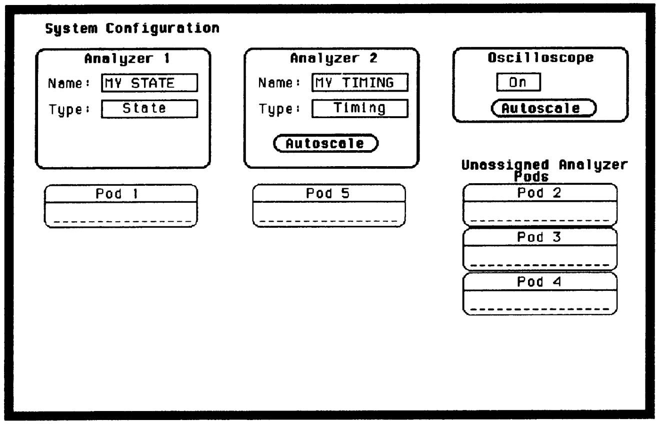
To move the under-score marker to the left, place the cursor over the left arrow and press **SELECT** once for each backspace.

To move the under-score marker to the right, you either place the cursor on a desired character and press **SELECT,** or place it on the right arrow and press **SELECT.**

You can also use the **ROLL** keys and the **knob** to move the underscore marker.

4. When you are finished spelling out the label, select **Done.**
5. Repeat steps 1 through 5, assigning Analyzer 2 as **Timing.**

The figure below is what your System Configuration menu should look like. For this example, we have labeled Analyzers 1 "MY STATE" and Analyzer 2 "MY TIMING."

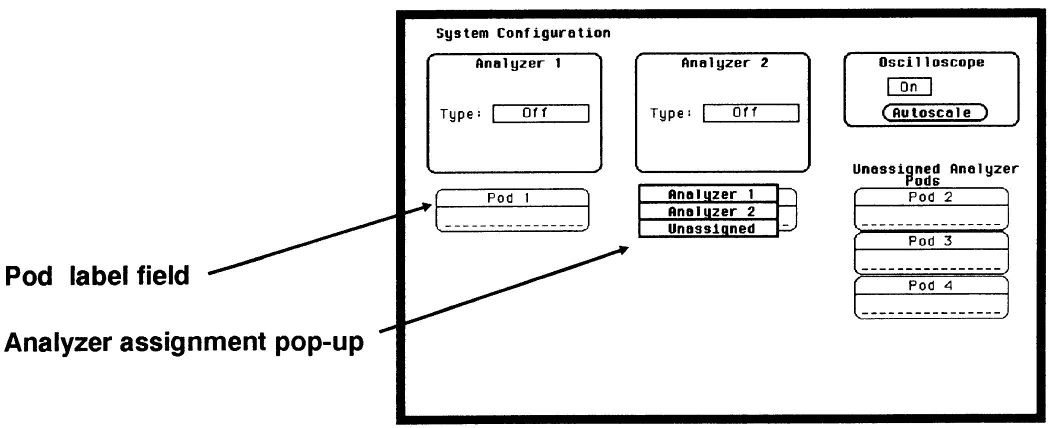


Pod Assignments

Pod labels in the System Configuration menu match the inputs on the rear panel. Pods are assigned to the analyzer machines as follows.

1. Position the cursor over the **Pod** you want to assign, as shown below, then press the **SELECT** key.

2. When the selection pop-up appears, select the **Analyzer** you want the pod assigned to. The pod will automatically move under the analyzer machine you select.



Autoscale

The System Configuration menu has two Autoscale fields. One is for the timing analyzer, and one is for the scope. If you select either **Autoscale** from the System Configuration menu,the logic analyzer will display a pop-up with the choices of **Cancel** and **Continue.** The **Cancel** allows you to change your mind before the autoscale is executed.

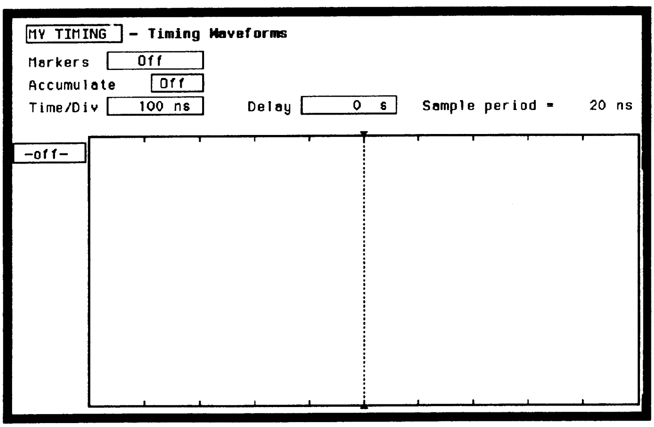
Note

When Autoscale is executed, instrument settings for any previous measurement will change. The new measurement will automatically be scaled for the best display of the waveform.

If you select **Continue,** the logic analyzer will display the Waveform menu of either the timing analyzer or the scope. The Waveforms display that is returned, depends on which Autoscale field you selected.

If there is no signal activity at the probe inputs, the Waveforms menu will not display data and the label to the left of the waveform area will be **-off-**. The figure below shows the waveforms menu for the timing analyzer after Autoscale was selected.

Name field



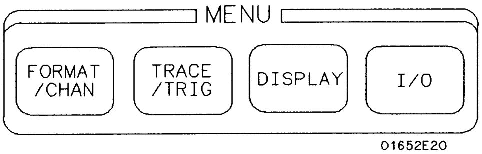
After an Autoscale, you can get back to the System Configuration menu by following the steps below.

1. Place the cursor on the **Name:** field in the upper left corner and press **SELECT.**

2. Place the cursor on the **System** field in the pop-up and press **SELECT**. You will now be back in the System Configuration menu.

Moving Around the Main Menus

The two logic analyzer types and the oscilloscope each have their own set of menus. You can enter these menus and move between them by pressing the **FORMAT/CHAN**, **TRACE/TRIG** and **DISPLAY** menu keys. These keys, as shown below, are in the area labeled MENU.



A fourth menu key labeled **I/O**, will bring up a pop-up menu that is independent of the analyzer and oscilloscope settings. The I/O functions available in this pop-up will be explained in detail in the *HP 1652B/HP 1653B Front-Panel Operation Reference* manual.

Accessing the Main Menus

When you turn Analyzer 1, Analyzer 2, or Oscilloscope on from the System Configuration menu, you also assign their set of main menus to the MENU keys.

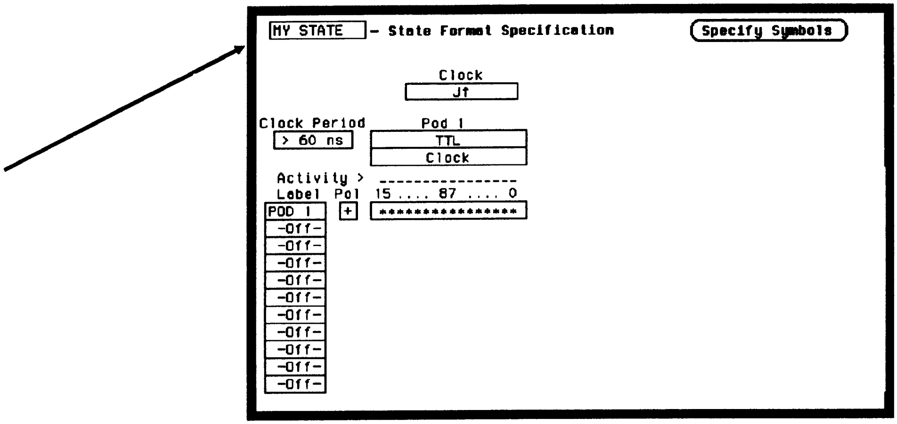
In the following exercise, you will access the Format Specification menus of both analyzer machines and the Channel menu of the oscilloscope.

1. Press the **FORMAT/CHAN** key. One of the following main menus will appear:

   - Scope **Channel** menu (if the scope is on).
   - **State Format Specification** menu (if the state analyzer is on).
   - **Timing Format Specification** menu (if the timing analyzer is on).

If, for example, the State Format Specification menu appears, but you wanted a FORMAT/CHAN menu of a different machine, continue with step 2.

2. Using the **Knob**, move the cursor to the analyzer **Name**: field (upper left corner) and press the **SELECT** key. See the figure on the following page.



3. From the pop-up that appears, select the name of the other analyzer or scope. The Format/Channel menu now appears for the new analyzer or scope.

If you want to move to the other menu types within the same machine, just press one of the other MENU keys defined below.

TRACE/TRIG

When you press the **TRACE/TRIG** key, you will have the following main menus available:

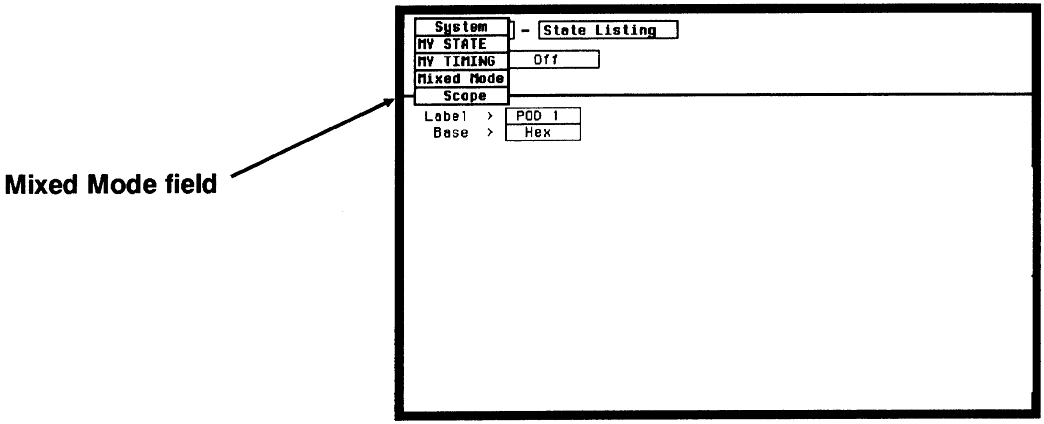
- Scope **Trigger** menu, if the scope is on.
- **State Trace Specification** menu, if the state analyzer is on.
- **Timing race Specification** menu, if the timing analyzer is on.

DISPLAY

When you press the **DISPLAY** key, you will have the following main menus available:

- Scope **Waveforms** menu, if the scope is on.
- **State Listing** menu, if the state analyzer is on.
- **Timing Waveforms** menu, if the timing analyzer is on.

When you are in any of the Waveforms or Listing menus, and you select the name field, you will notice an additional field called the **Mixed Mode** field.



The **Mixed Mode** field as shown above, is only available from the Waveforms or Listing menus. That is, because it is like an optional display menu. When the **Mixed Mode** field is selected, a combined display is returned. The use of the Mixed Mode display is shown in detail in Chapters 21, 26 and 27 of the *HP 1652B/HP 1653B Front-Panel Operation Reference* manual.

Returning to the System Configuration Menu

If at any time you think you are lost, or just want to get back to the System Configuration menu, follow the procedure below.

1. Press either the **FORMAT/CHAN, TRACE/TRIG,** or **DISPLAY** MENU key. The menu that appears, will have a field in the upper left corner.

2. Select the field in the **upper left corner**. A pop-up appears that lists all machines which are turned on. The machine names are either the labels which you have assigned, or the default names of **MACHINE 1, MACHINE 2, SCOPE** and **Mixed Mode.**

3. Place the cursor over the **System** field and press the SELECT key. You will be returned to the System Configuration menu.

Summary

In this chapter you learned how to get from the System Configuration menu into the main menus of the separate analyzer machines. From within the main menus, you where shown how to move between menus, then quickly get back out to the System Configuration menu.

As you configured the System Configuration menu, you where introduced to some common pop-up menus. As mentioned earlier, more information on pop-up menus will appear in the following chapters.

Chapters 4 through 6 are simulated measurement examples. These chapters will illustrate how the logic analyzer is configured and used to make typical measurements. If you are new to logic analysis, and need a little more practice with your new logic analyzer, try following along with these simulated measurement examples. If you are an experienced logic analyzer user, you can go to the *HP 1652B/HP 1653B Front-Panel Operation Reference* manual at this point.

4

Using the State Analyzer

Introduction

In this chapter you learn how to use the state analyzer by setting up the logic analyzer to simulate a simple state measurement. We give you the measurement results as actually measured by the logic analyzer, since you may not have the same circuit available.

The exercise in this chapter is organized in a task format. The tasks are in the same order you will most likely use them once you become experienced. The steps in this format are both numbered and lettered. The numbered steps state the step objective. The lettered steps explain how to accomplish each step objective. There is also an example of each menu after it has been properly set up.

How you use the steps depends on how much you remember from chapters 1 through 3. If you can set up each menu by just looking at the menu picture, go ahead and do so. If you need a reminder of what steps to perform, follow the numbered steps. If you still need more information about "how," use the lettered steps.

Problem Solving with the State Analyzer

In this example assume you have designed a microprocessor controlled circuit. You have completed the hardware, and the software designer has completed the software and programmed the ROM (read-only memory). When you turn your circuit on for the first time, your circuit doesn't work properly. You have checked the power supply voltages and the system clock and they are working properly.

Since the circuit has never worked before, you and the software engineer aren't sure if it is a hardware or software problem. You need to do some testing to find a solution.

What Am I Going to Measure?

You decide to start where the microprocessor starts when power is applied. We will describe a 68000 microprocessor; however, every processor has similar start-up routines.

When you power up a 68000 microprocessor, it is held in reset for a specific length of time before it starts doing anything to stabilize the power supplies. The time the microprocessor is held in reset ensures stable levels (states) on all the devices and buses in your circuit. When this reset period has ended, the 68000 performs a specific routine called "fetching the reset vector."

The first thing you check is the time the microprocessor is held in reset. You find the time is correct. The next thing to check is whether the microprocessor fetches the reset vector properly.

The steps of the 68000 reset vector fetch are:

1. Set the stack pointer to a location you specify, which is in ROM at address locations 0 and 2.

2. Find the first address location in memory where the microprocessor fetches its first instruction. This is also specified by you and stored in ROM at address locations 4 and 6.

What you decide to find out is:

1. What ROM address does the microprocessor look at for the location of the stack pointer, and what is the stack pointer location stored in ROM?

2. What ROM address does the microprocessor look at for the address where its first instruction is stored in ROM, and is the instruction correct?

3. Does the microprocessor then go to the address where its first instruction is stored?

4. Is the executable instruction stored in the first instruction location correct?

Your measurement, then, requires verification of the sequential addresses the microprocessor looks at, and of the data in ROM at these addresses. If the reset vector fetch is correct (in this example) you will see the following list of numbers in HEX (default base) when your measurement results are displayed.

+ 0000 000000 0000
+ 0001 000002 04FC
+ 0002 000004 0000
+ 0003 000006 8048
+ 0004 008048 3E7C

This list of numbers will be explained in detail later in this chapter in "The State Listing."

How Do I Configure the Logic Analyzer?

In order to make this state measurement, you must configure the logic analyzer as a state analyzer. By following these steps you will configure Analyzer 1 as the state analyzer.

If you are in the System Configuration menu you are in the right place to get started and you can start with step *2*; otherwise, start with step *1*.

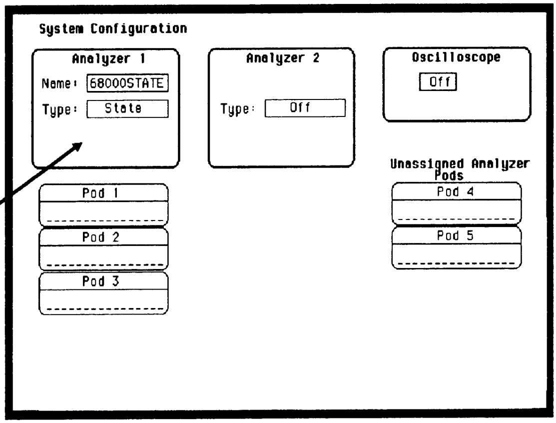
1. Using the field in the upper left corner of the display, get the System Configuration menu on screen.

   a. Place the cursor on the field in the upper left corner of the display and press SELECT.

   b. Place the cursor on **System** and press SELECT.

2. In the System Configuration menu, change the Analyzer 1 type to **State**. If Analyzer 1 is already a state analyzer, go on to step *3*.

Change to a State Analyzer



   a. Place the cursor on the **Type:** field and press **SELECT.**

   b. Place the cursor on **State** and press SELECT.

3. Name Analyzer 1 68000STATE (optional).

   a. Place the cursor on the **Name:** field of Analyzer 1 and press **SELECT.**

   b. With the Alpha Entry pop-up, change the name to 68000STATE.

4. Assign pods 1, 2, and 3 to the state analyzer.

   a. Place the cursor on the **Pod 1** field and press SELECT.

   b. In the Pod 1 pop-up, place the cursor on **Analyzer 1** and press SELECT.

   c. Repeat steps *a* and *b* for pods 2 and 3.

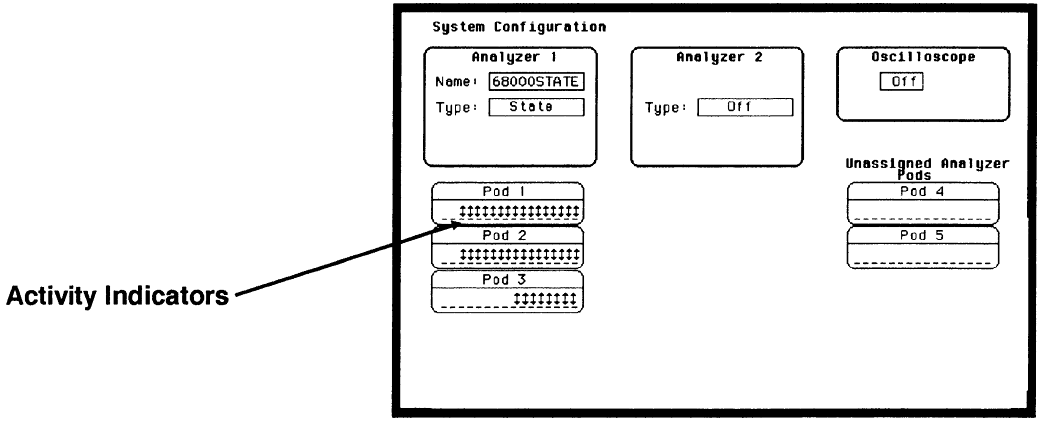
Connecting the Probes

At this point, if you had a target system with a 68000 microprocessor, you would connect the logic analyzer to your system. Since you will be assigning labels ADDR and DATA, you hook the probes to your system accordingly.

- Pod 1 probes 0 through 15 to the data bus lines D0 through D15.
- Pod 2 probes 0 through 15 to the address bus lines A0 through A15.
- Pod 3 probes 0 through 7 to the address bus lines A16 through A23.
- Pod 1, CLK (J clock) to the address strobe (LAS).

Activity Indicators

When the logic analyzer is connected and your target system is running, you will see **Activity Indicators** in the Pod 1, 2, and 3 fields of the System Configuration menu. This indicates which signal lines are transitioning.



Configuring the State Analyzer

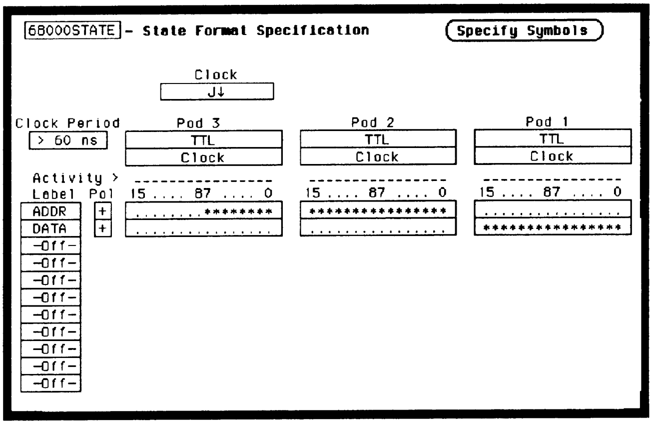
Now that you have configured the system, you are ready to configure the state analyzer. You will be:

- Creating two names (labels) for the input signals
- Assigning the channels connected to the input signals
- Specifying the State (J) clock
- Specifying a trigger condition

1. Display the State Format Specification menu.

   a. Press the **FORMAT** key on the front panel.

2. Name two labels, one ADDR and one DATA.



   a. Place the cursor on the top field in the label column and press SELECT.

   b. Place the cursor on **Modify label** and press SELECT.

c. With the Alpha Entry pop-up, change the name of the label to ADDR.

d. Name the second label DATA by repeating steps *a* through *c*.

3. Assign Pod 1 bits 0 through 15 to the label DATA.

a. Place the cursor on the bit assignment field below Pod 1 and to the right of DATA and press SELECT.

b. Any combination of bits may already be assigned to this pod; however, you will want all 16 bits assigned to the DATA label. The easiest way to assign is to press the CLEAR ENTRY key to un-assign any assigned bits before you start.

c. Place the cursor on the period under the 15 in the bit assignment pop-up and press SELECT. This will place an asterisk in the pop-up for bit 15, indicating Pod 1 bit 15 is now assigned to the DATA label. Repeat this procedure until all 16 bits have an asterisk under each bit number. Place the cursor on **Done** and press SELECT to close the pop-up.

d. Repeat step *c* for Pod 2 and the ADDR label to assign all 16 bits.

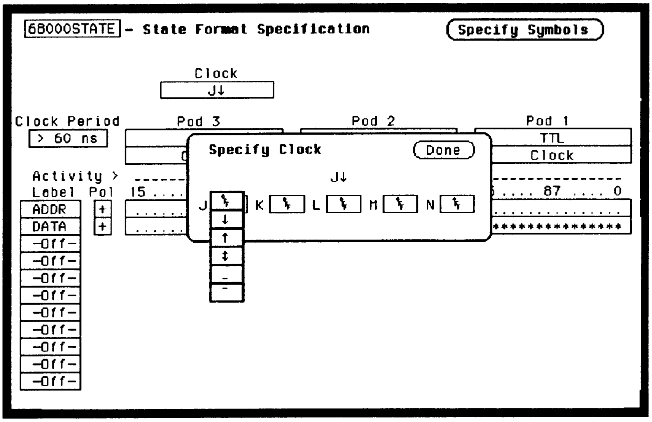
e. Repeat step *c* except you will assign the lower eight bits (0 - 7) of Pod 3 to the ADDR label.

Specifying the J Clock

If you remember from "What's a State Analyzer" in *Feeling Comfortable With Logic Analyzers*, the state analyzer samples the data under the control of an external clock, which is "synchronous" with your circuit under test. Therefore, you must specify which clock probe you will use for your measurement. In this exercise, you will use the J clock, which is accessible through pod 1.

1. Select the State Format Specification menu by pressing the **FORMAT** key.

2. Set the J Clock to sample on a negative-going edge.



a. Place the cursor on the **CLOCK** field and press SELECT.

b. Place the cursor on the box just to the right of J in the pop-up (labeled OFF) and press SELECT.

c. Place the cursor on ↓ and press SELECT.

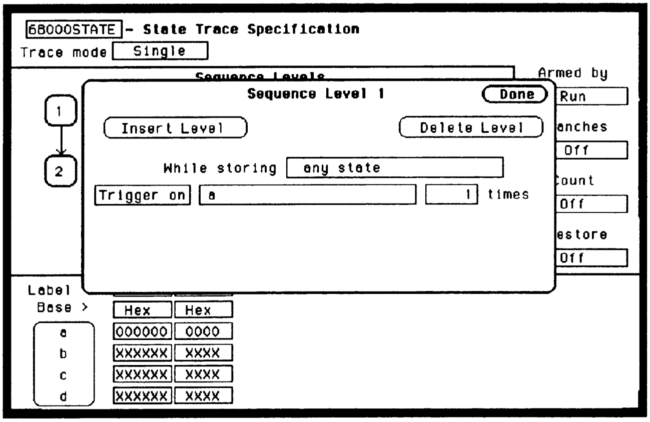
d. Place the cursor on **Done** and press SELECT.

Specifying a Trigger Condition

To capture the data and place the data of interest in the center of the display of the State Listing menu, you need to tell the state analyzer when to trigger. Since the first event of interest is address 0000, you need to tell the state analyzer to trigger when it detects address 0000 on the address bus.

1. Select the State Trace Specification menu by pressing the TRACE key.

2. Set the trigger so that the state analyzer triggers on address 0000. If the **Trigger on** option is not already "**a**" perform steps *a* through *d*. If the option is "**a**" skip to step *e*.



a. Place the cursor on the **1** in the Sequence Levels field of the menu and press SELECT.

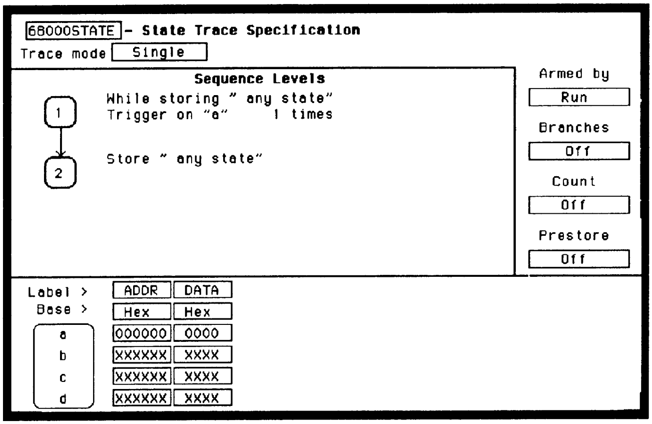
b. Place the cursor on the field to the right of the **Trigger on** field and press SELECT. Another pop-up appears showing you a list of "trigger on" options. Options *a* through *h* are qualifiers. You can assign them a pattern for the trigger specification.

c. Place the cursor on the "**a**" option and press SELECT.

d. Place the cursor on **Done** in the Sequence Levels pop-up and press SELECT.

e. Place the cursor on the field to the right of the "**a**" under the label ADDR and press SELECT.

f. With the keypad, press 0 (zero) until there are all zeros in the Specify Pattern: pop-up and then press SELECT.



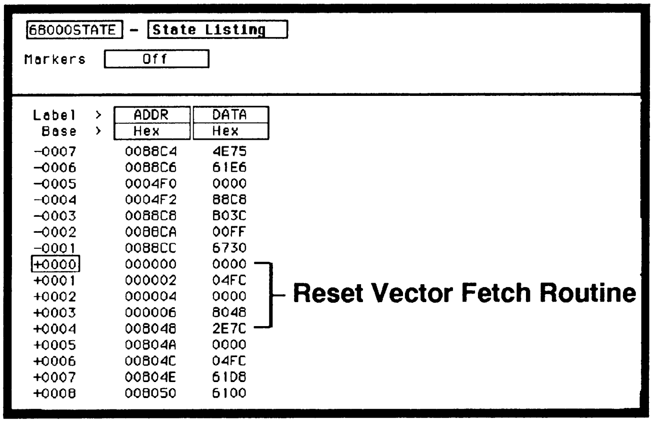
Your trigger specification now states: "While storing anystate trigger on "**a**" once and then store any state."

When the state analyzer is connected to your circuit and is acquiring data, it continuously stores until it sees 0000 on the address bus, then it will store anystate until the analyzer memory is filled.

Acquiring the Data

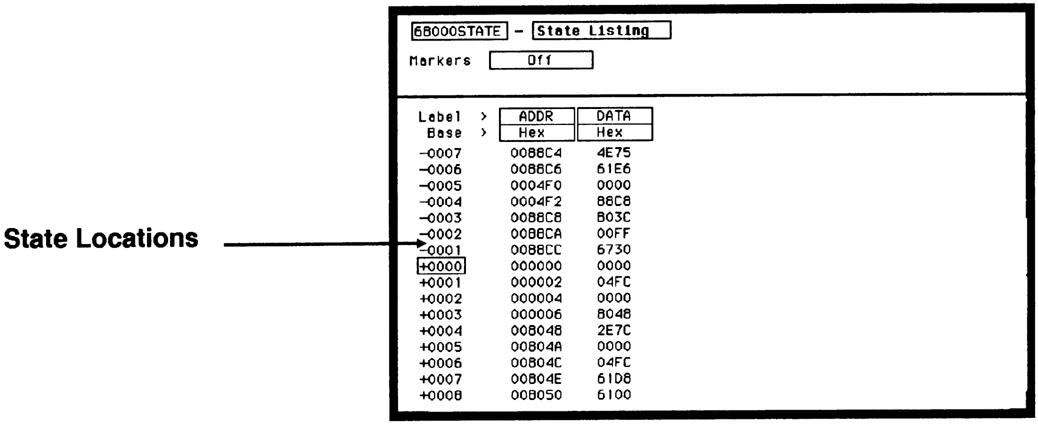
Since you want to capture the data when the microprocessor sends address 0000 on the bus after power-up, you press the RUN key to arm the state analyzer and then force a reset of your circuit. When the reset cycle ends, the microprocessor should send address 0000 trigger the state analyzer and switch the display to the State Listing menu.

We'll assume this is what happens in this example, since the odds that the microprocessor won't send address 0000 are very low.



The State Listing

The state listing displays three columns of numbers as shown:



The first column of numbers are the state line number locations as they relate to the trigger point. The trigger state is always on line + 0000 in the vertical center of the list area. The negative numbers indicate states occurring before the trigger and the positive numbers indicate states occurring after the trigger.

The second column of numbers are the states (listed in HEX) the state analyzer sees on the address bus. This column is labeled ADDR.

The third column of numbers are the states (listed in HEX) the state analyzer sees on the data bus. This column is labeled DATA.

Finding the Answer

Your answer is now found in the listing of states + 0000 through + 0004.

The 68000 always reads address locations 0, 2, 4, and 6 to find the stack pointer location and memory location for the instruction it fetches after power-up. The 68000 uses two words for each of the locations that it is looking for, a high word and a low word. When the software designer programs the ROM, he must put the stack pointer location at address locations 0 and 2. 0 is the high word location and 2 is the low word location. Similarly, the high word of the instruction fetch location must be in address location 4 and the low word in location 6.

Since the software design calls for the reset vector to set the stack pointer to 04FC and read memory address location 8048 for its first instruction fetch, you are interested in what is on both the address bus and the data bus in states 0 through 3.

The state listing below lists the codes reset vector search, in states 0 through 3 and the correct first microprocessor instruction in state 4.

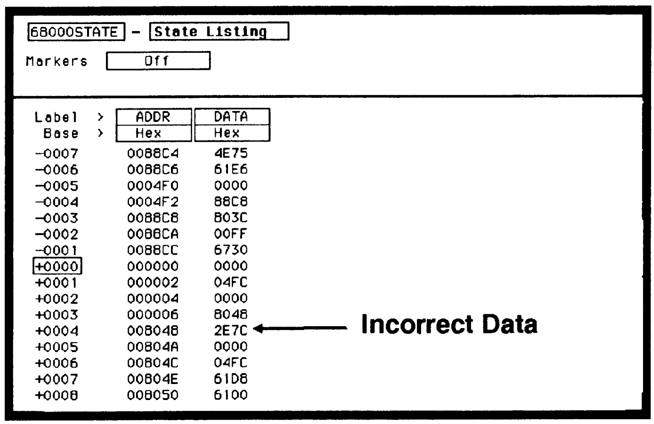
+ 0000 000000 0000
+ 0001 000002 04FC
+ 0002 000004 0000
+ 0003 000006 8048
+ 0004 008048 3E7C

You see that states 0 and 1 do contain address locations 0 and 2 under the ADDR label, indicating the microprocessor did look at the correct locations for the stack pointer data. You also see that the data contained in these ROM locations are 0000 and 04FC, which are correct.

You then look at states 2 and 3. You see that the next two address locations are 4 and 6, which is correct, and the data found at these locations is 0000 and 8048, which is also correct.

So far you have verified that the microprocessor has correctly performed the reset vector search. The next thing you must verify is whether the microprocessor addresses the correct location in ROM that it was instructed to address in state 4 and whether the data is correct in this ROM location. From the listing on your machine, you see that the address in state 4 is 008048, which is correct, but the instruction found in this location is 2E7C, which is not correct. You have found your problem: incorrect data stored in ROM for the microprocessor's first instruction.

+ 0000 000000 0000 (high word of stack pointer location)
+ 0001 000002 04FC (low word of stack pointer location)
+ 0002 000004 0000 (high word of instruction fetch location)
+ 0003 000006 8048 (low word of instruction fetch location)
+ 0004 008048 2E7C (first microprocessor instruction)



Summary

You have just learned how to make a simple state measurement with the HP 1652B Logic Analyzer. You have:

- specified a state analyzer
- learned which probes to connect
- assigned pods 1, 2, and 3
- assigned labels
- assigned bits
- specified the J clock
- specified a trigger condition
- acquired the data
- interpreted the state listing

You have seen how easy it is to use the state analyzer to capture the data on the address and data buses. You can use this same technique to capture and display related data on the microprocessor status control, and various strobe lines. You are not limited to using this technique on microprocessors. You can use this technique any time you need to capture data on multiple lines and need to sample the data relative to a system clock.

Chapter 21 of the *HP 1652B/HP 1653B Front-Panel Operation Reference* manual shows you how to use the logic analyzer as an interactive timing and state analyzer. You will see a simple measurement that shows you both timing waveforms and state listings and how they are correlated.

If you have an HP 1653B, you do not have enough channels to simultaneously capture all the data for a 68000. But, since you probably aren't working with 16-bit microprocessors, this example is still valuable because it shows you how to make the same kind of measurement on an eight-bit microprocessor.

5

Using the Timing Analyzer

Introduction

In this chapter you will learn how to use the timing analyzer by setting up the logic analyzer to simulate a simple measurement. We give you the measurement results as actually measured by the logic analyzer, since you may not have the same circuit available.

The exercise in this chapter is organized in a task format. The tasks are ordered in the same way you will most likely use them once you become an experienced user. The steps in this format are both numbered and lettered. The numbered steps state the step objective. The lettered steps explain how to accomplish each step objective. There is also an example of each menu after it has been properly set up.

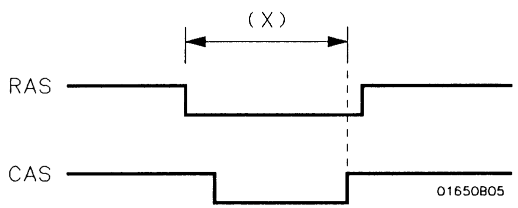
How you use the steps depends on how much you remember from chapters 1 through 3 . If you can set up each menu by just looking at the menu picture, go ahead and do so. If you need a reminder of what steps you need to perform, follow the numbered steps. If you still need more information about "how," use the lettered steps.

Problem Solving with the Timing Analyzer

In this exercise, assume you are designing a dynamic RAM memory (DRAM) controller and you must verify the timing of the row address strobe (RAS) and the column address strobe (CAS). You are using a 4116 dynamic RAM and the data book specifies that the minimum time from when LRAS is asserted (goes low) to when LCAS is no longer asserted (goes high) is 250 ns. You could use an oscilloscope but since the timing analyzer will do just fine when you don't need voltage parametrics you decide to go ahead and use the logic analyzer.

What Am I Going to Measure?

After configuring the logic analyzer and hooking it up to your circuit under test, you will be measuring the time (x) from when the RAS goes low to when the CAS goes high, as shown below.



How Do I Configure the Logic Analyzer?

In order to make this timing measurement, you must configure the logic analyzer as a timing analyzer. By following these steps you will configure Analyzer 1 as the timing analyzer.

If you are in the System Configuration menu you are in the right place to get started and you can start with step *2*; otherwise, start with step *1*.

1. Using the field in the upper left corner of the display, get the System Configuration menu on screen.

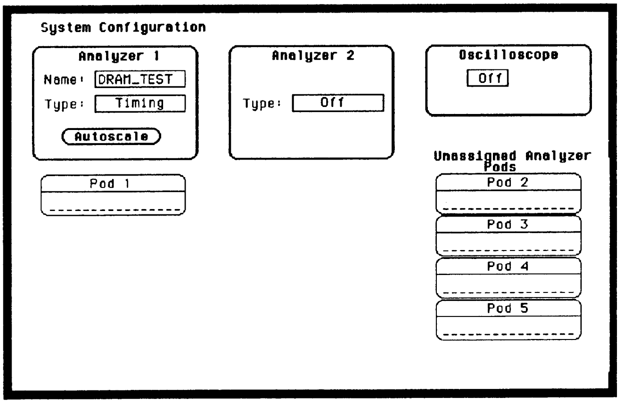
   a. Place the cursor on the field in the upper left corner of the display and press SELECT.

   b. Place the cursor on **System** and press SELECT.

2. In the System Configuration menu, change Analyzer 1 type to **Timing**. If analyzer 1 is already a timing analyzer, go on to step *3*.

   a. Place the cursor on the **Type:** field and press SELECT.

   b. Place the cursor on **Timing** and press SELECT.



3. Name Analyzer 1 "DRAM TEST" (optional)

   a. Place the cursor on the **Name:** field of Analyzer 1 and press SELECT.

   b. With the Alpha Entry pop-up, change the name to "DRAM TEST"

4. Assign pod 1 to the timing analyzer.

   a. Place the cursor on the **Pod 1** field and press SELECT.

   b. In the Pod 1 pop-up, place the cursor on **Analyzer 1** and press SELECT.

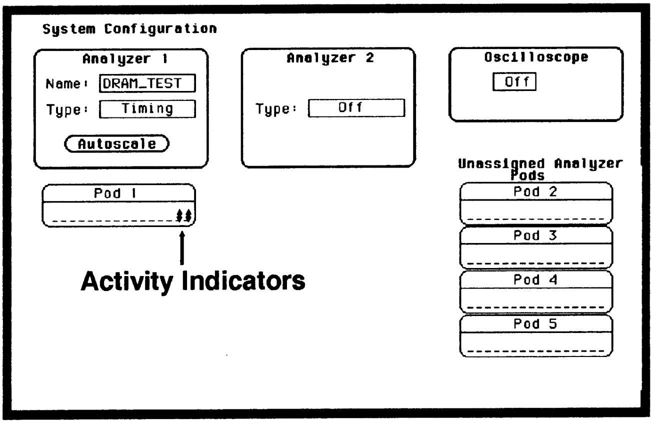
Connecting the Probes

At this point, if you had a target system with a 4116 DRAM memory IC, you would connect the logic analyzer to your system.

Since you will be assigning Pod 1 bit 0 to the RAS label, you hook Pod 1 bit 0 to the memory IC pin connected to the RAS signal. You hook Pod 1 bit 1 to the IC pin connected to the CAS signal.

Activity Indicators

When the logic analyzer is connected and your target system is running, you will see activity indicators, as shown below, at the right-most end (least significant bits) of the Pod 1 field in the System Configuration menu. This indicates the RAS and CAS signals are transitioning.



Configuring the Timing Analyzer

Now that you have configured the system, you are ready to configure the timing analyzer. You will be:

- Creating two names (labels) for the input signals
- Assigning the channels connected to the input signals
- Specifying a trigger condition

1. Display the Timing Format Specification menu.

   a. Press the **FORMAT** key on the front panel.

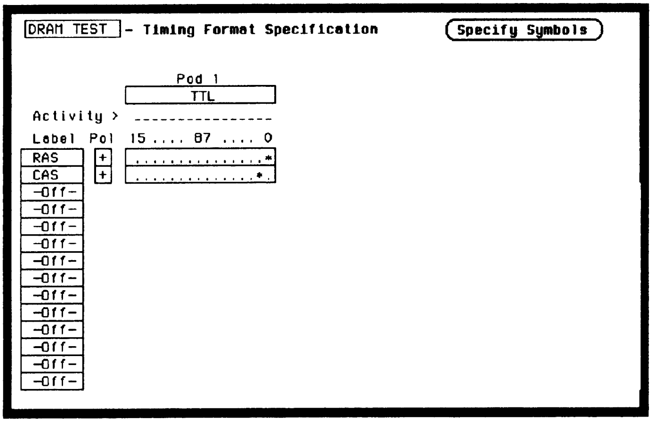
2. Name two labels, one RAS and one CAS.

   a. Place the cursor on the top field in the label column and press SELECT.

   b. Place the cursor on **Modify** label and press SELECT .

   c. With the Alpha Entry pop-up, change the name of the label to RAS.

   d. Name the second label CAS by repeating steps *a* through *c*.



3. Assign the channels connected to the input signals (Pod 1 bits 0 and 1) to the labels RAS and CAS respectively.

   a. Place the cursor on the bit assignment field below Pod 1 and to the right of RAS and press SELECT.

   b. Any combination of bits may be assigned to this pod; however, you will want only bit 0 assigned to the RAS label. The easiest way to assign bits is to press the CLEAR ENTRY key to un-assign any assigned bits before you start.

   c. Place the cursor on the period under the 0 in the bit assignment pop-up and press SELECT . This will place an asterisk in the pop-up for bit 0 indicating Pod 1 bit 0 is now assigned to the RAS label. Place cursor on **Done** and press SELECT to close the pop-up.

   d. Assign Pod 1 bit 1 to the CAS label by moving the cursor to bit 1 and pressing SELECT.

Specifying a Trigger Condition

To capture the data and then place the data of interest in the center of the display of the Timing Waveforms menu, you need to tell the logic analyzer when to trigger. Since the first event of interest is when the LRAS is asserted (negative-going edge of RAS), you need to tell the logic analyzer to trigger on a negative-going edge of the RAS signal.

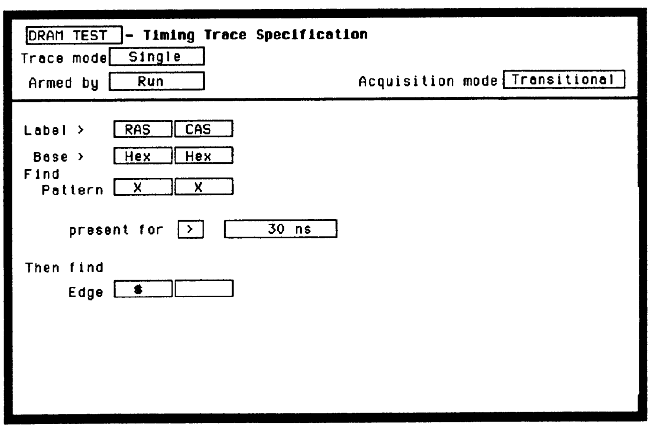
1. Select the Timing Trace menu by pressing the **TRACE** key.

2. Set the trigger so that the logic analyzer triggers on the negative-going edge of the RAS.

   a. Place the cursor on the **Then find Edge** field under the label RAS, then press SELECT.

   b. Place the cursor on the "." (period) in the pop-up and press SELECT once. Pressing SELECT once in this pop-up changes a period to $\downarrow$which indicates a negative-going edge.

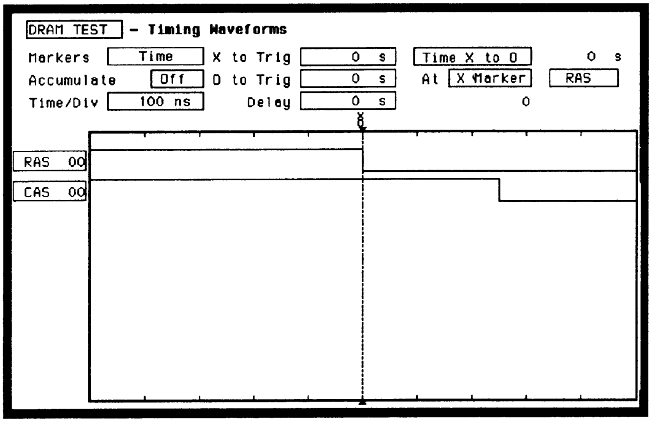
   c. Place the cursor on **Done** and press SELECT. The pop-up closes and a $ will be located in this field. The $ indicates an edge has been specified even though it can't be shown in the HEX base.



Acquiring the Data

Now that you have configured and connected the logic analyzer, you acquire the data for your measurement by pressing the RUN key. The logic analyzer will look for a negative edge on the RAS signal and trigger if it sees one. When it triggers, the display switches to the Timing Waveforms menu.

The RAS label shows you the RAS signal and the CAS label shows you the CAS signal. Notice the RAS signal goes low at or near the center of the waveform display area (horizontal center).



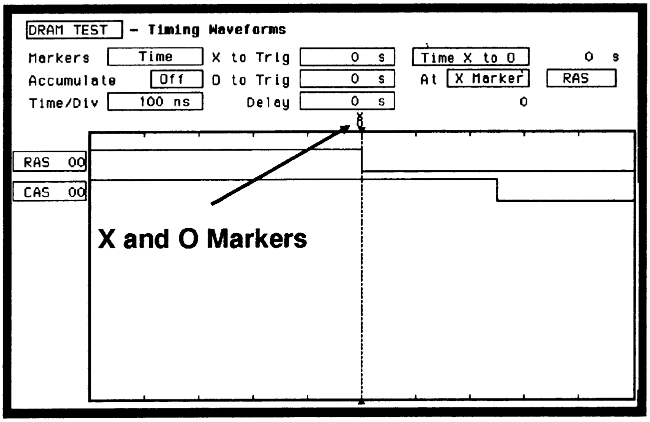
The Timing Waveforms Menu

The Timing Waveforms menu differs from the other menus you have used so far in this exercise. Besides displaying the acquired data, it has menu fields that you use to change the way the acquired data is displayed and fields that give you timing answers. Before you can use this menu to find answers, you need to know some of the special symbols and their functions. The symbols are:

- The X and O
- The ▼
- The vertical dotted line

The X and O

The X and O are markers you use to find your answer. You place them on the points of interest on your waveforms, and the logic analyzer displays the time between the markers. The X and O markers will be in the center of the display when X to trig (ger) and O to trig (ger) are both 0.000 s (see example below).

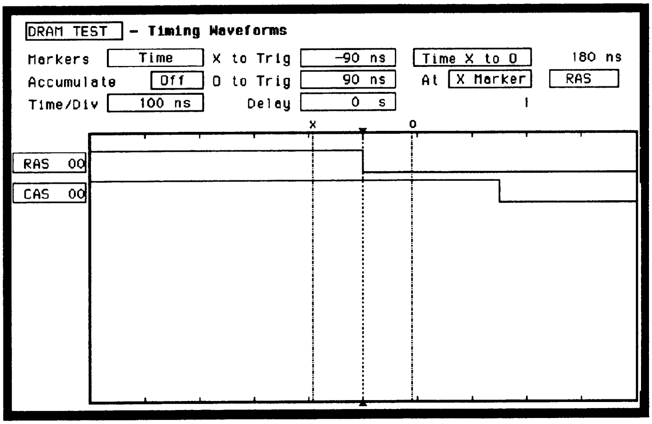


The ▼

The ▼ (inverted triangle) indicates the trace point. Remember, trace point = trigger + delay. Since delay in this example is 0.000 s, you will see the negative-going edge of the RAS signal at center screen under the ▼.

The Vertical Dotted Line

The vertical dotted line indicates the trigger point you specified in the Timing Trace Specification menu. The vertical dotted line is at center screen under the inverted triangle and is superimposed on the negative-going edge of the RAS signal.

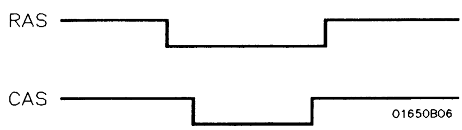


Configuring the Display

Now that you have acquired the RAS and CAS waveforms, you need to configure the Timing Waveforms menu for best resolution and to obtain your answer.

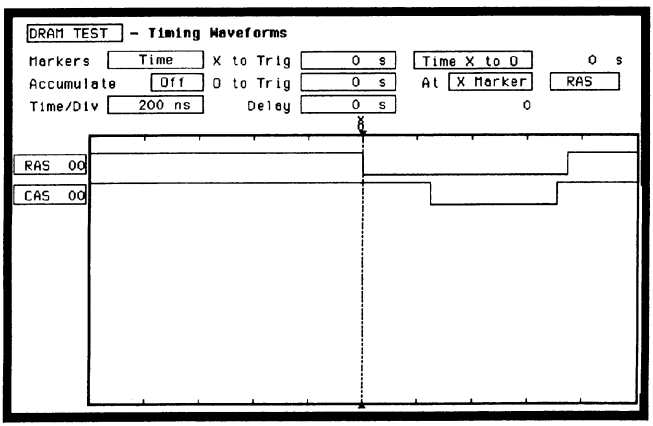
Display Resolution

You get the best resolution by changing the Time/Div to a value that displays one negative-going edge of both the RAS and CAS waveforms. Set the Time/Div by following these steps.



1. Place the cursor on **Time/Div** and press SELECT . The Time/Div pop-up appears, showing you the current setting.

2. While the pop-up is present, rotate the KNOB until your waveform shows you only one negative-going edge of the RAS waveform and one positive-going edge of the CAS waveform (see above). In this example 200 ns is best.



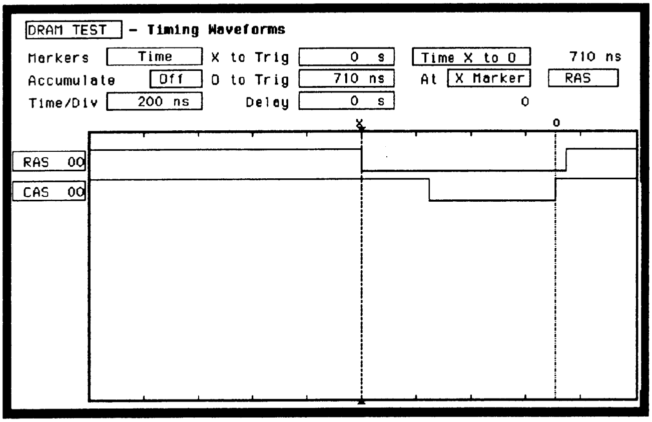
Making the Measurement

What you want to know is how much time elapses between the time RAS goes low and the time CAS goes high again. You will use the X and O markers to quickly find the answer. Remember, you specified the negative-going edge of the RAS to be your trigger point; therefore, the X marker should be on this edge if X to Trig = 0. If not, follow steps *1* and *2*.

1. Place the cursor on the **X to Trig** field and press SELECT . A pop-up will appear showing you the current time from the X marker to the trigger; however, you don't need to worry about this number now.

2. Rotate the KNOB to place the X marker on the negative-going edge of the RAS waveform and press SELECT . The pop-up closes and displays X to Trig = 0.000 s.

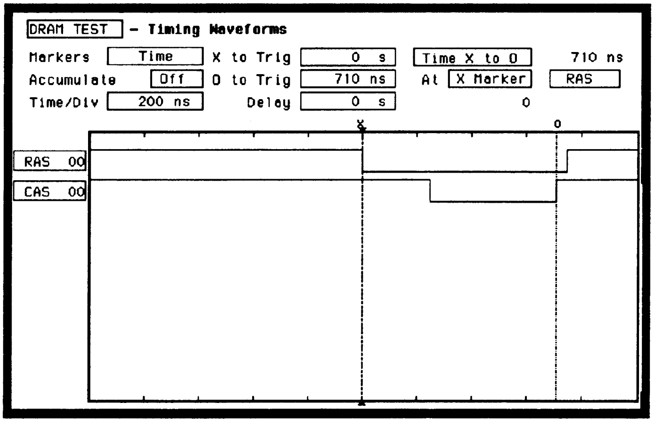
3. Place the cursor on **O to Trig** and press SELECT . Repeat step *2* except place the O marker on the positive-going edge of the CAS waveform and press SELECT. The pop-up closes and displays O to Trig = 710 ns.



Finding the Answer

Your answer could be calculated by adding the X to Trig and O to Trig times, but you don't need to bother. The logic analyzer has already calculated this answer and displays it in the **Time X to O** field.

This example indicates the time is 710 ns. Since the data book specifies a minimum of 250 ns, it appears your DRAM controller circuit is designed properly.



Summary

You have just learned how to make a simple timing measurement with the HP 1652B/1653B logic analyzer. You have learned to do the following:

- Specified a timing analyzer.
- Assigned pod 1.
- Assigned bits.
- Assigned labels.
- Specified a trigger condition.
- Learned which probes to connect.
- Acquired the data.
- Configured the display.
- Set the Time/Div for best resolution.
- Positioned the markers for the measurement answer.

You have seen how easy it is to use the timing analyzer to make timing measurements that you could have made with a scope. You can use the timing analyzer for any timing measurement that doesn't require voltage parametrics or doesn't go beyond the accuracy of the timing analyzer.

Using the Oscilloscope

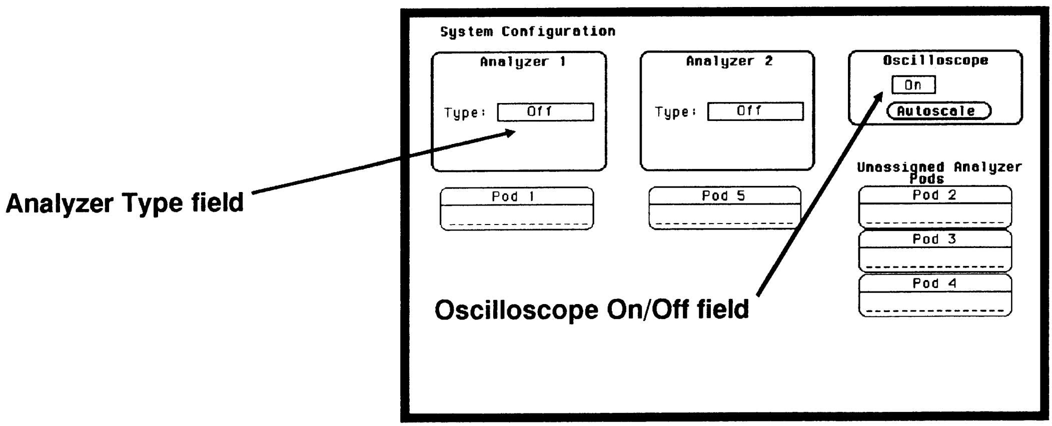
Introduction

This chapter uses a simple example to get you familiar with using the oscilloscope. We will be starting from the beginning with this exercise, so it's not necessary to have completed chapters 4 and 5.

As you follow through the menus setups in this exercise, you will use the Probe Compensation signal from the rear panel as the signal source for measurement. If you think you can complete this exercise by just following the illustrated menus, do so. If you need additional help, follow the numbered steps.

Getting to the Scope Menus

From the default System Configuration menu shown below, the scope should already be turned on. If the scope is not turned on, turn on the scope and turn off the analyzers. Now, get to the scope Channel menu.

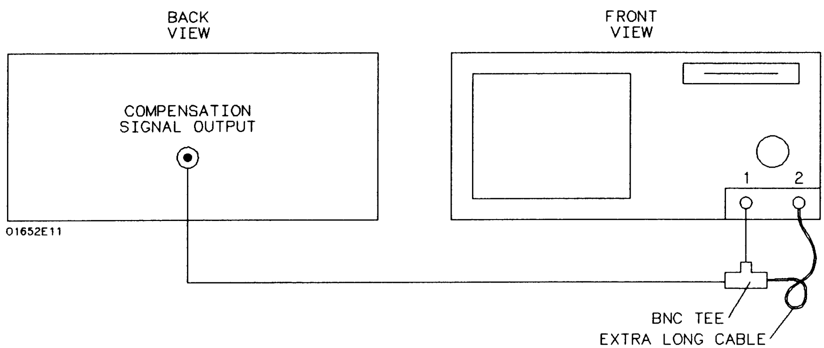


1. Toggle the Oscilloscope **On/Off** field to **On**.
2. Select the analyzer **Type**: field, then turn all analyzers **Off**.
3. Press the **FORMAT/CHAN** menu key.

Setting Up for the Measurement

Setting up for the measurement consists of two parts. The first part, is the actual hook-up to the "system under test." This first part is where you connect the scope probes and set up any external test equipment. The second part of the setup is the input attenuation and impedance setting of your oscilloscope. For the example in this chapter, we are using the Probe Compensation Signal from the rear panel.

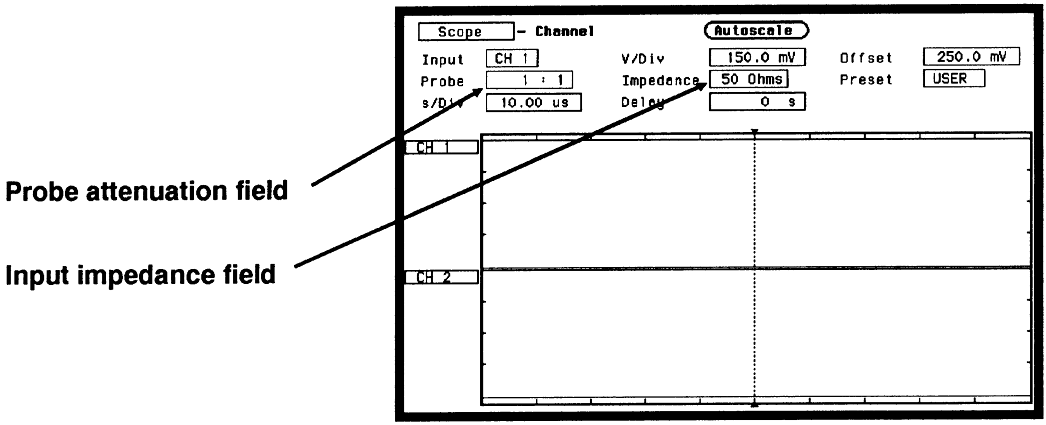
Connect two BNC cables and a BNC tee as shown below. An optional extra long BNC cable can be used to produce a delay on one of the channels.



Note

BNC cable length is not important, but cable impedance must be 50 Ω. impedance.

Set the **probe** attenuation field and the input **Impedance** field as shown below in the scope Channel menu.

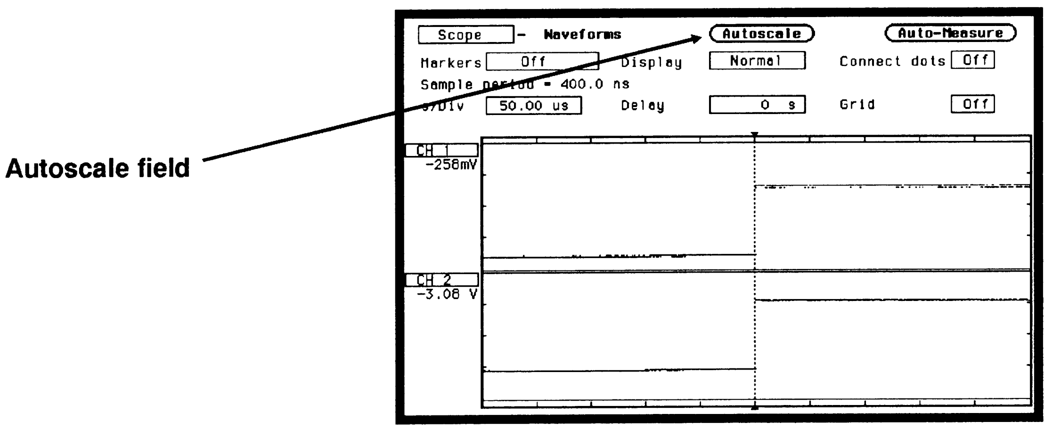


1. Select the **Probe** field. Using the knob or keypad, set the attenuation to **1:1**.

2. Select the **Impedance** field. Toggle to **50 Ohms**.

Making the Measurement

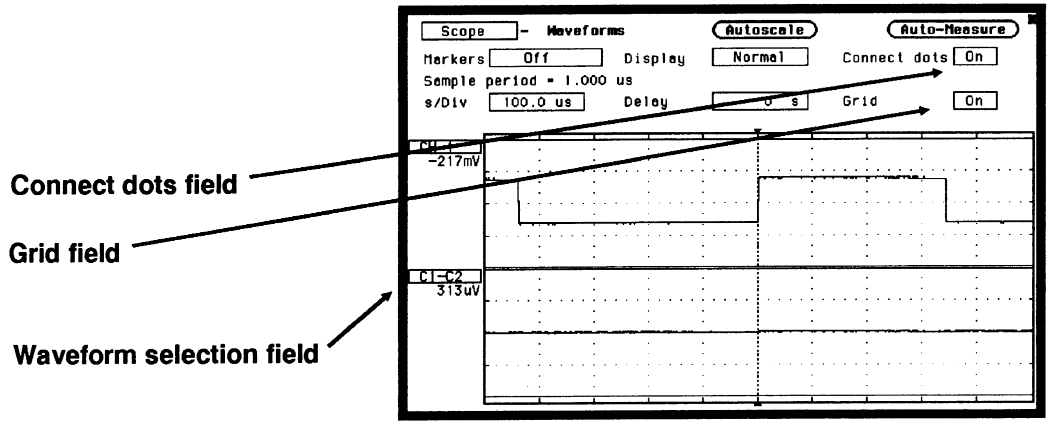
By selecting the **Autoscale** field, the measurement will automatically be scaled, positioned, and is displayed on the scope's Waveforms menu as shown below.



1. Select the **Autoscale** field, then select **Continue** from the pop-up.

Displaying the Results

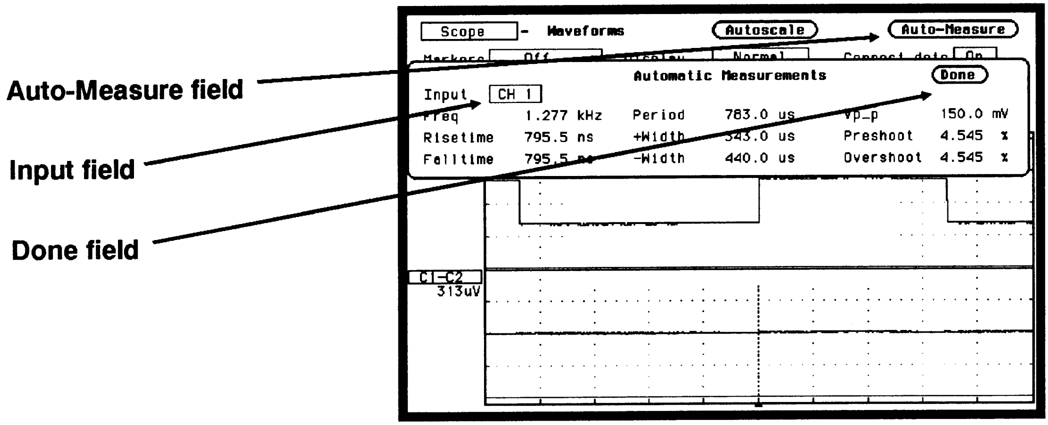
Now that you have the measurement results on screen, you can modify the Waveforms display to view the signals any way you want. The figure below shows just some of the many effects that occur when you change the Waveforms menu configuration.



1. Select the **Connect dots** field. Toggle to **On**.
2. Select the **Grid** field. Toggle to **On**.
3. Select the **s/Div** field. Using the knob or keypad, set the seconds per division to **100** μs.
4. Select the **CH 2** waveform selection field, then select the **Modify waveform** field from the pop-up. Select the **C1-C2** field from the pop-up.
5. Press the **RUN** key (not necessary when Run mode set to Repetitive).

Automatic Measurement Readouts

When the **Auto-Measure** field is selected, you get a parametric readout of nine parameters as shown in the figure below.



1. From the scope Waveforms menu, select the **Auto-Measure** field.
2. To get a readout for the other channel, move the cursor to the **Input** field, then press the **SELECT** key. The input will then automatically toggle to the other channel.
3. To exit the Auto-Measure pop-up, select **Done**.

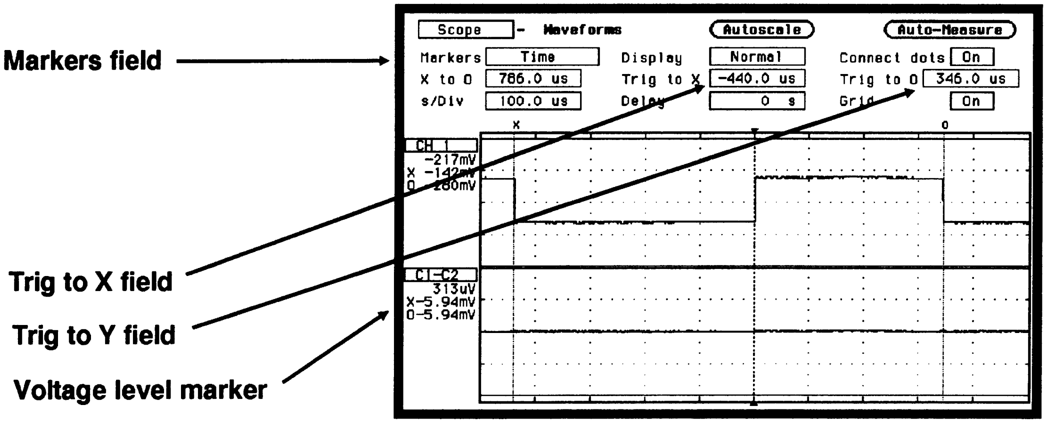
Using the Markers

Your oscilloscope has the ability to set markers (reference points) either manually or automatically. This may be important if your measurement requires that you know the exact time between different points on the waveform or from the point your scope triggered on the waveform.

In this exercise we set only the Time markers. For a complete description of the markers function, refer to the *HP 1652B\HP 1653B Front-Panel Operation Reference* manual.

From the scope Waveforms menu, select the **Markers** field. A pop-up appears that lists the different kinds of marker measurements available. As mention before, you will make just a Time marker measurement for now.

Set the Time markers menu to match the figure below.



1. From the scope Waveforms menu select the **Markers** field. Select the **Time** field from the pop-up.
2. To set the X-marker, select the **Trig to X** field. Using the knob, move the marker to the left (negative time from trigger point) until the dashed line is aligned with the falling edge of the pulse.
3. To set the Y-marker, select the **Trig to Y** field. Using the knob, move the marker to the right (positive time from trigger point) until the dashed line is aligned with the falling edge of the next pulse.

Now that you have set the X and O markers on these edges of the waveform, the following time relevant information is available:

- Time between markers is displayed in the **X to O** field.
- Time between markers and trigger point is displayed in the **Trig to X** field and the **Trig to O** field.
- The voltage level of the waveform at the point the markers were placed is displayed under the channel label.

Summary

After finishing the exercises in chapters 4 through 6, you should now be familiar with your new logic analyzer. If you want more detailed information on how your logic analyzer operates, refer to the *HP 1652B/HP 1653B Front-Panel Operation Reference* Manual.

If you have a printer and would like to make hardcopy prints of configurations in any of the previous exercises, refer to Appendix B "Making Hardcopy Prints."

7

What's Next?

Now that you are familiar with the logic analyzer, you may want to try some of the basic measurements discussed in this book on your target system. Refer to the documentation for your microprocessor.

If you are comfortable with the basic measurements that you can perform with the HP 1652B and HP 1653B Logic Analyzers, you are ready for the *HP 1652B/HP 1653B Front-Panel Operation Reference.* This reference manual explains all the capabilities of both logic analyzers and their operation from the front panel. The *HP 1652B/HP 1653B Programming Reference* manual tells you how to operate both logic analyzers from a controller via the RS-232C or HP-IB interface.

A

Logic Analyzer Turn-on Check List

This appendix summarizes the steps you take to turn on the HP 1652B and HP 1653B logic analyzers. The details of the turn-on procedures are in Chapter 1 of this Guide.

1. Check the rear-panel line voltage indicator for the proper setting. Change the setting if necessary.
2. Make sure you have the proper 3-wire grounded AC power cable.
3. Make sure the rear-panel line switch if **Off**.
4. Connect the power cable to the rear-panel line connector and a properly grounded power receptacle.
5. Make sure the yellow shipping disk is removed from the disk drive.
6. Insert the operating system disk in the disk drive.
7. Turn the logic analyzer on with the rear-panel line switch.

When the logic analyzer completes its self-tests, it then loads the operating system from the disk. When the operating system has been completely loaded, the **System Configuration** menu will be displayed.

B

Making Hardcopy Prints

Introduction

The HP 1652B and HP 1653B Logic Analyzers allow you to print the configurations, waveforms, and listings. Whenever your printer is connected to your logic analyzer and you instruct it to do so, it will print what is currently displayed on screen.

This chapter shows you how to set up the logic analyzer's HP-IB and RS-232C interfaces and how to instruct the logic analyzer to make a print. If you have a Hewlett-Packard ThinkJet, QuietJet, or LaserJet series printer with the RS-232C interface, the RS-232C interface is already set up for you.

If you have another kind of printer, refer to your printer manual for its interface requirements and change your logic analyzer's interface configuration as instructed.

Hooking Up Your Printer

If your printer is already connected to the logic analyzer, skip to "Setting RS-232C for HP Printers" or "Setting HP-IB for HP Printers." If not, hooking up your printer is just a matter of having the correct HP-IB or RS-232C interface cable. Refer to the *Front Panel Reference Manual* you received with your logic analyzer.

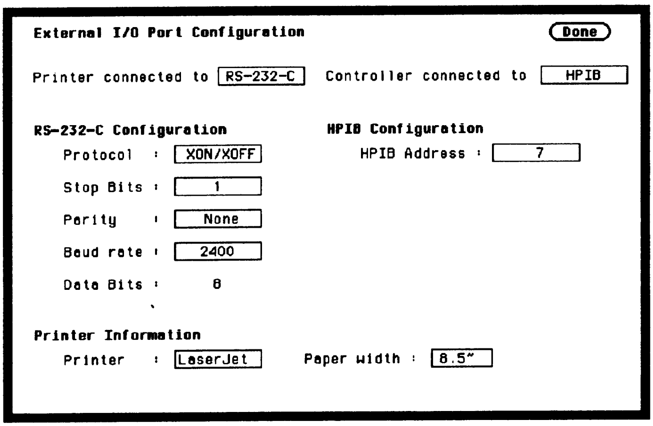
Setting RS-232C for HP Printers

All you have to do to set the interface for any of the previously listed Hewlett-Packard series printers with the RS-232C interface is to set the printer type in the **External I/O Port Configuration** submenu.

To set the printer type, follow these steps:

1. Display the I/O menu by pressing the **I/O** key.
2. Place the cursor on **I/O Port Configuration** and press **SELECT.**

You will see the following submenu:



3. If the **Printer connected to** field displays **RS-232C** skip to step 4. Otherwise, place the cursor in the **Printer connected to HP-IB** field and press **SELECT.** The **Printer connected to** switches from **HP-IB** to **RS-232C.**
4. Place the cursor on the printer series type and press **SELECT.**
5. Place the cursor on **Done** and press **SELECT.** The logic analyzer will display the menu that was displayed when you selected the I/O menu.

Setting RS-232C for Your Non-HP Printer

The following attributes of the RS-232C interface must be set to the correct configuration for your printer:

- Protocol
- number of stop bits
- parity type
- Baud rate
- paper width

You can set all of these attributes for your printer by following this procedure:

1. Press the **I/O** key to display the I/O menu.
2. Place the cursor on **I/O Port Configuration** and press **SELECT.**
3. Place the cursor on the attribute and press **SELECT.**
4. When the pop-up is open, place the cursor on the option your printer requires and press **SELECT.** The pop-up closes, placing your selection in the box. Repeat this step for all attributes that you need to change.
5. Place the cursor on **Done** and press **SELECT**. The logic analyzer will display the menu that was displayed when you selected the I/O menu.

Setting Hp_IB for HP Printers

The HP 1652B/HP 1653B interfaces directly with HP PCL printers supporting the printer command language. These printers must also support HP-IB and "Listen Always." Printers currently available from Hewlett-Packard with these features include:

- HP 2225A ThinkJet
- HP 2227B QuietJet
- HP 3630A option 002 PaintJet

Note

The printer must be in "Listen Always" mode when HP-IB is the printer interface. The HP 1652B/HP 1653B HP-IB port does not respond to service requests (SRQ) when controlling a printer. The SRQ enable setting for the HP-IB printer has no effect on the HP 1652B/HP 1653B operation.

For HP-IB printers, the **Printer connected to** field must be set to **HP-IB** in the I/O Port Configuration menu. You access this menu by first pressing the **I/O** key, then moving the cursor to the **I/O Port Configuration** field and pressing **SELECT**.

Starting the Printout

When you are ready to print, you will need to know whether there is more data than is displayed on screen. In cases where data is off screen (i.e., format specifications with all pods assigned to a single analyzer), you need to decide whether you want all the data or just the data is on screen.

If you want just what is on screen, start the printout with the **Print Screen** option. If you want all the data, use the **Print All** option. Both options are in the I/O menu.

Once you decide which option to use, start the printout by placing the cursor on the print option (screen or all) and pressing **SELECT.**

I/O MENU

- Done
- Print Screen
- Print All
- Disk Operations
- I/O Port Configuration
- External BNC Configuration
- Self Tests

Print Screen

The **Print Screen** option prints only what is displayed on screen at the time you initiate the printout. In the Print Screen mode, the printer uses its graphics capabilities so the printout will look just like the logic analyzer screen with only one exception: the cursor will not print.

Print All

The **Print all** option prints not only what is displayed on screen, but also what is off screen at the time you initiate the printout. In the Print All mode, the printout will be made in the text mode with only one exception: a timing waveform display will be printed in the graphics mode because it has no off-screen data.

Use this option when you want to print all the data in menus like:

- Timing and State Format Specifications
- State Trace Specifications
- State Listing

What Happens during a Printout?

When you press select to start the printout, the I/O menu pop-up disappears and an advisory **PRINT in progress** appears in the top center of the display. While the data is transferred to the printer, the logic analyzer's keyboard deactivates. When the logic analyzer has completed the data transfer to the printer, the advisory disappears and the keyboard reactivates.

Don't worry! The **Print in progress** advisory won't appear in your printout.

Summary

Now that you have configured the RS-232C or HP-IB interface for your printer, you can make hardcopy printouts of anything that the logic analyzer displays. This is a valuable feature when you need to keep records of configurations and measurements.

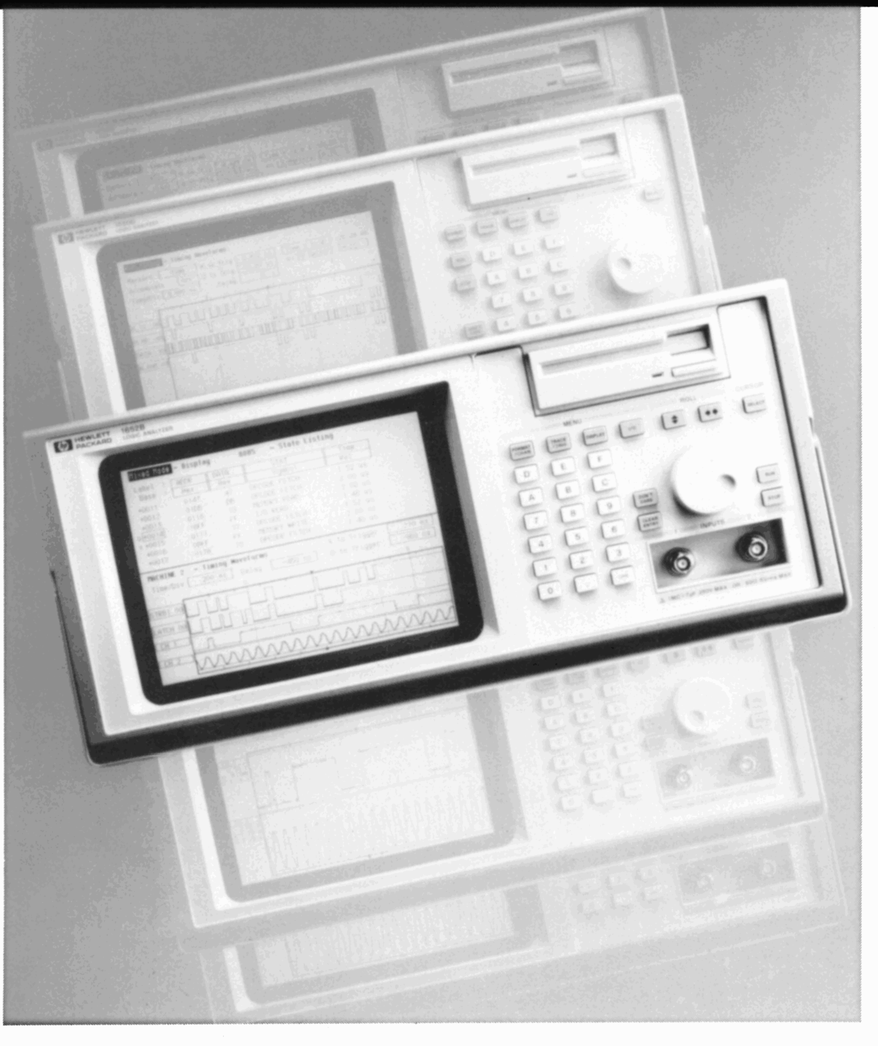
HEWLETT
PACKARD

HP PART NUMBER: 5952-0176

Printed in U.S.A.

HP 1652B/HP 1653B Logic Analyzers

Front-Panel Operation Reference
Volume 1 of 2



HEWLETT PACKARD

Front-Panel Operation Reference

Volume 1 of 2

HP 1652B/HP 1653B
Logic Analyzers

HEWLETT PACKARD

Manual Set Part Number 01652-90902

Printed in the U.S.A. November 1989

Printing History

New editions are complete revisions of the manual. Update packages, which are issued between editions, contain additional and replacement pages to be merged into the manual by the customer. The dates on the title page change only when a new edition or a new update is published.

A software code may be printed before the date; this indicates the version level of the software product at the time of the manual or update was issued. Many product updates and fixes do not require manual changes and, conversely, manual corrections may be done without accompanying product changes. Therefore, do not expect a one to one correspondence between product updates and manual updates.

Edition 1	November 1989	01652-90902

List of Effective Pages

The List of Effective Pages gives the date of the current edition and of any pages changed in updates to that edition. Within the manual, any page changed since the last edition is indicated by printing the date the changes were made on the bottom of the page. If an update is incorporated when a new edition of the manual is printed, the change dates are removed from the bottom of the pages and the new edition date is listed in Printing History and on the title page.

Pages	Effective Date
All	November 1989

Product Warranty

This Hewlett-Packard product has a warranty against defects in material and workmanship for a period of one year from date of shipment. During warranty period, Hewlett-Packard Company will, at its option, either repair or replace products that prove to be defective.

For warranty service or repair, this product must be returned to a service facility designated by Hewlett-Packard. However, warranty service for products installed by Hewlett-Packard and certain other products designated by Hewlett-Packard will be performed at the Buyer's facility at no charge within the Hewlett-Packard service travel area. Outside Hewlett-Packard service travel areas, warranty service will be performed at the Buyer's facility only upon Hewlett-Packard's prior agreement and the Buyer shall pay Hewlett-Packard's round trip travel expenses.

For products returned to Hewlett-Packard for warranty service, the Buyer shall prepay shipping charges to Hewlett-Packard and Hewlett-Packard shall pay shipping charges to return the product to the Buyer. However, the Buyer shall pay all shipping charges, duties, and taxes for products returned to Hewlett-Packard from another country.

Hewlett-Packard warrants that its software and firmware designated by Hewlett-Packard for use with an instrument will execute its programming instructions when properly installed on that instrument. Hewlett-Packard does not warrant that the operation of the instrument, software, or firmware will be uninterrupted or error free.

Limitation of Warranty

The foregoing warranty shall not apply to defects resulting from improper or inadequate maintenance by the Buyer, Buyer-supplied software, or interfacing, unauthorized modification or misuse, operation outside of the environmental specifications for the product, or improper site preparation or maintenance.

NO OTHER WARRANTY IS EXPRESSED OR IMPLIED. HEWLETT-PACKARD SPECIFICALLY DISCLAIMS THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

Exclusive Remedies THE REMEDIES PROVIDED HEREIN ARE THE BUYER'S SOLE AND EXCLUSIVE REMEDIES. HEWLETT-PACKARD SHALL NOT BE LIABLE FOR ANY DIRECT, INDIRECT, SPECIAL INCIDENTAL, OR CONSEQUENTIAL DAMAGES, WHETHER BASED ON CONTRACT, TORT, OR ANY OTHER LEGAL THEORY.

Assistance Product maintenance agreements and other customer assistance agreements are available for Hewlett-Packard products.

For any assistance, contact your nearest Hewlett-Packard Sales and Service Office. Addresses are provided at the back of this manual.

Certification Hewlett-Packard Company certifies that this product met its published specifications at the time of shipment from the factory. Hewlett-Packard further certifies that its calibration measurements are traceable to the United States National Bureau of Standards, to the extent allowed by the Bureau's calibration facility, and to the calibration facilities of other International Standards Organization members.

Safety This product has been designed and tested according to International Safety Requirements. To ensure safe operation and to keep the product safe, the information, cautions, and warnings in this manual must be heeded.

Introduction

About this manual...

Welcome to Hewlett-Packard logic analyzers. The HP 1652B/HP 1653B Logic Analyzer is more than just a logic analyzer. It is an analyzer and oscilloscope in one instrument. With this combination, you have expanded measurement capabilities.

This manual has been split into two volumes for better accessibility. Volume one contains general instrument information and operating reference information for the state analyzer. Also included is a state analyzer measurement example.

Volume two contains operating reference information and measurement examples for the timing analyzer and oscilloscope. To help put the total functionality of the instrument together, a measurement example for mixed mode operation (timing/state/scope) is included. Located in the back of volume two is the appendices which contain the seldom used information.

Information in both volumes is accessed easily by major tabs. All menu and field definitions are arranged by major function within each measurement type. In addition, both volumes have a master index.

The user interface of the HP 1652B/1653B was designed for the most intuitive operation as possible. Pop-up windows help lead you through setups and measurements so you won't have to memorize a lot of steps. As you read this manual and the other manuals about this logic analyzer, you will see just how easy the HP 1652B/1653B is to use.

If you aren't familiar with the HP 1652B/1653B Logic Analyzers, we suggest you read the HP 1652B/1653B Getting Started Guide. This guide contains tutorial examples on the basic functions of the logic analyzer and digitizing oscilloscope.

If you are new to logic analyzers and digitizing oscilloscopes, or just need a refresher, we think you'll find *Feeling Comfortable with Logic Analyzers* and *Feeling Comfortable with Digitizing Oscilloscopes* valuable reading. It will eliminate any misconceptions or confusion you may have about their application, and will show you how to get the most out of the measurement funtions.

Please take time to fill out the "Your Comments Please" questionaire. If it has already been used and you have any comments, address them to:

Hewlett-Packard
Atten: Publications Dept.
P.O. Box 2197
Colorado Springs, CO 80901-2197

Contents Volume 1

Chapter 1: General Information

Chapter 2: Probing

Chapter 3: Using the Front-Panel User Interface

Chapter 4: System Configuration Menu

Chapter 5: I/O Menu

Chapter 6: Disk Drive Operations

Chapter 7: Making Hardcopy Prints

Chapter 8: The State Analyzer

Chapter 9: State Format Specification Menu

Chapter 10: State Trace Menu

Chapter 11: State Listing Menu

Chapter 12: State Compare Menu

Chapter 13: State Chart Menu

Chapter 14: State Waveforms Menu

Chapter 15: State Analyzer Measurement Example

Index

1

General Information

Logic Analyzer Description

The HP 1652B/1653B logic analyzers are general purpose logic analyzers with oscilloscope measurement capabilities. These analyzers are designed as stand alone instruments for use by digital and microprocessor designers. Both the HP 1652B and HP 1653B have HP-IB and RS-232C interfaces for hardcopy printouts and control by a host computer. With faster state analysis, oscilloscope measurement capabilities and the improved features, the HP 1652B\53B analyzers will accommodate next generation design tasks.

The HP 1652B, is capable of 100 MHz timing and 35 MHz state analysis on 80 channels. The HP 1653B, is capable of 100 MHz timing and 25 MHz state analysis on 32 channels . You will use the same manual set regardless of whether you have an HP 1652B or an HP 1653B.

Both analyzers have the same 2-channel, 400-megasample/second, 100 MHz single-shot and repetitive single-shot digitizing oscilloscope measurement capabilities.

User Interface

First-time and casual users as well as experienced logic analyzer users will find the user interface easier to use than in previous generations.

The front panel is controlled by a front-panel keyboard, and the addition of a "KNOB" allows you to move the cursor or change settings more quickly than before. The timing analyzer (a close cousin of the oscilloscope) now has oscilloscope-type controls which more closely match the type of measurements you make with the timing analyzer. Information is displayed on a nine-inch white phosphor CRT.

Configuration Capabilities

The HP 1652B/1653B can be configured either as two independent machines (analyzers) or as two interactive machines. No matter how the analyzers are configured, up to two channels of oscilloscope measurement can be added. The configurations for each analyzer includes the following:

HP 1652B:

- Up to 80 channels state and up to two channels oscilloscope.
- Up to 80 channels timing and up to two channels oscilloscope.
- Two state machines with multiples of 16 channels per machine, with a combined maximum of 80 channels and up to two channels oscilloscope.
- One state and one timing machine with multiples of 16 channels per machine, with a combined maximum of 80 channels and up to two channels oscilloscope.
- Up to two channels of oscilloscope.

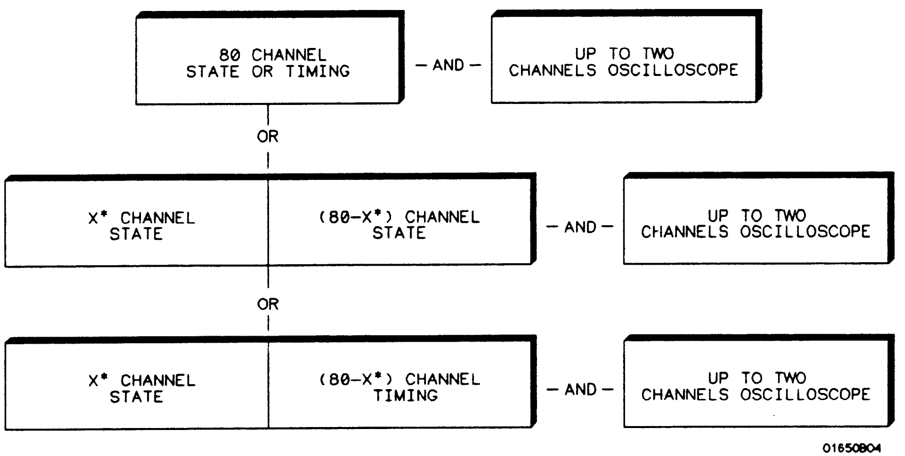


Figure 1-1. HP 1652B Configuration Capabilities

*multiples of 16 channels

HP 1653B:

- Up to 32 channels state and up to two channels oscilloscope.
- Up to 32 channels timing and up to two channels oscilloscope.
- Two state machines with multiples of 16 channels per machine, with a combined maximum of 32 channels and up to two channels oscilloscope.
- One state and one timing machine with multiples of 16 channels per machine, with a combined maximum of 32 channels and up to two channels oscilloscope.
- Up to two channels of oscilloscope.

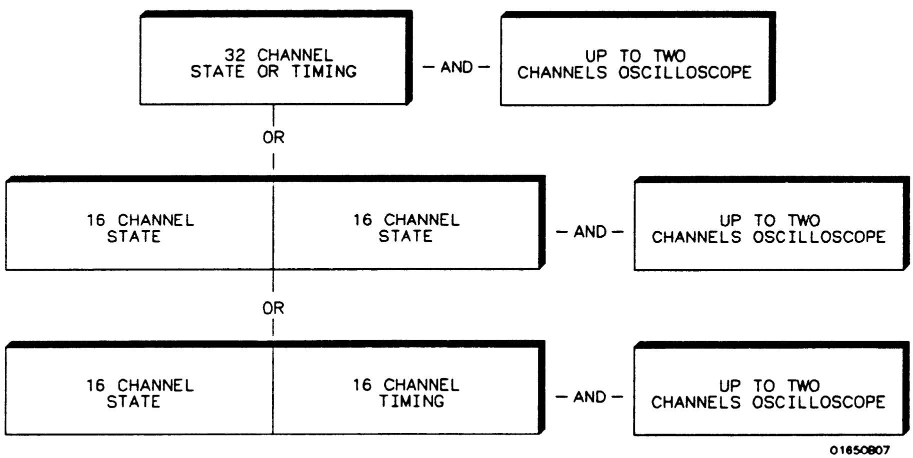


Figure 1-2. HP 1653B Configuration Capabilities

Key Features

A 3.5-inch disk drive is built into the instrument for storing logic analyzer and oscilloscope configurations and acquired data. The disk drive also provides a way of loading inverse assembly configuration files into the logic analyzer for easy configuring. Some common features of the logic analyzer and oscilloscope include lightweight passive probes for easy hook-up, mixed-mode display, HP-IB and RS-232C interfaces for programming and printer output.

Logic analyzer key features include:

- Transitional timing for extended timing analyzer memory.
- All channels can be used for state or timing.
- An external trigger BNC connector.
- Transitional or glitch timing modes.
- 1 k deep memory on all channels.
- Glitch detection.
- Marker measurements.
- Triggering and pattern qualification.
- Overlapping of timing waveforms.
- Eight sequence levels.
- Eight pattern recognizers.
- One range recognizer.
- Time and number-of-states tagging.
- Pre-store.
- Autoscale.
- Programmability.
- Cross-domain triggering.
- Interactive measurements.
- Oscilloscope-type controls in the timing analyzer.
- State Compare, Chart, and Waveform modes.

Oscilloscope key features include:

- 400 Megasample/second digitizing rate.
- 100 MHz single-shot (real-time) bandwidth.
- 4 ksamples per measurement per channel.
- Automatic waveform scaling.

- ECL and TTL presets.
- Automatic pulse parameter measurements.
- Channel-to-channel time interval measurements.
- Markers for time and voltage readouts.
- 6-bit resolution.
- Probe attenuation from 1:1 to 1000:1.
- 50Ω dc or 1 MΩ dc input coupling.
- Edge or immediate triggering.
- Delayed trigger by events and/or time.
- Trigger point marker displayed.
- Normal, average, or cumulative acquisitions.
- Connect-the-dots.
- Chan + Chan, Chan-Chan, and waveform overlay.

Accessories Supplied

Table 1-1 lists the accessories supplied with your HP 1652B/53B. If any of these accessories were missing when you received the logic analyzer from the factory, contact your nearest Hewlett-Packard office.

Accessory	HP Part No.	Quantity 1652B	1653B
Probe assemblies	01650-61608	5	2
Probe cables	01650-61607	5	2
BNC Adapter 90°	1250-0076	2	2
BNC-to-mini probe adapter	1250-1454	1	1
Grabbers (Note 1)	5959-0288	100	40
Probe Leads (Note 2)	5959-9333	85	34
Ground leads (long) (Note 2)	5959-9335	5	2
Ground leads (short) (Note 2)	5959-9334	10	4
RS-232C loop back adapter	01650-63202	1	1
Probe and probe cable numbering label card	01650-94303	1	1
Mini-probes 10:1, 1 MΩ, 6.5 pF, 1 m	HP 10430A	2	2
AC power cable	See Note 3	1	1
Operating system disk	Call	2	2
Operating and Programming manual set	01652-90902	1	1
Service Manual	01652-90905	1	1

Table 1-1. Accessories Supplied

Notes:

1. Package of 20 per part number. The quantity in the table only indicates what is shipped with the instrument.

2. Package of 5 per part number. These items are shipped assembled as a 01650-61608. The part numbers are provided for replacement orders. The quantity in the above table only indicates what is sent with the instrument.

3. The type of power cord you receive with your logic analyzer depends on your country. Complete information about power cord options is in Appendix D of this manual.

Available Accessories

In addition to the accessories supplied, there are a number of accessories available that will make your measurement tasks easier and more accurate. You will find these listed in *Accessories for HP Logic Analyzers.*

Manuals Supplied

The manuals supplied with your logic analyzer are as follows:

- *Feeling Comfortable with Logic Analyzers* - A primer on logic analyzers.
- *Feeling Comfortable with Digitizing Oscilloscopes* - A primer on digitizing oscilloscopes.
- *Getting Started with the HP 1652B/1653B Logic Analyzer* - A tutorial for new and casual users.
- *HP 1652B/1653B Front Panel Operation Reference Manual* - A complete operating manual.
- *HP 1652B/1653B Programming Reference* - A complete reference to programming commands.
- Service Manual - A guide to troubleshooting and module-level repair.

Turning On the Logic Analyzer

Before you turn your logic analyzer on, refer to Appendix D for information covering installation and set up of your logic analyzer.

Note

Do not turn on the logic analyzer before you remove the yellow shipping disk from the disk drive.

If you are unfamiliar with how to use the HP 1652B/1653B logic analyzers, refer to chapter 1 of the *Getting Started with the HP 1652B/1653B Logic Analyzer.*

2

Probing

Introduction

This chapter contains a description of the probing system of the HP 1652B/1653B logic analyzers. It also contains the information you need for connecting the probe system components to each other, to the logic analyzer and oscilloscope, and to the system under test.

Probing Options

You can connect the HP 1652B/1653B logic analyzers to your system under test in one of the following ways:

- HP 10320C User-Definable Interface (optional).
- HP 10269C with microprocessor specific modules (optional).
- The standard HP 1652B/53B probes (general purpose probing.)
- Direct connection to a 20-pin 3M® Series type header connector using the optional termination adapter (HP part number 01650-63201).

The HP 10320C User-Definable Interface

The optional HP 10320C User-Definable Interface module combined with the optional HP 10269C General Purpose Probe Interface allows you to connect the HP 1652B/1653B logic analyzers to the microprocessor in your target system. The HP 10320C includes a breadboard (HP 64651B) which you custom wire for your system.

Another option for use with the HP 10320C is the HP 10321A Microprocessor Interface Kit. This kit includes sockets, bypass capacitors and a fuse for power distribution. Also included are wire-wrap headers to simplify wiring of your interface when you need active devices to support the connection requirements of your system.

You will find additional information about the HP 10320C and HP 10321A in the *Accessories for HP Logic Analyzers* data sheet.

The HP 10269C General Purpose Probe Interface

Instead of connecting the analyzer probe tips directly to the signal lines, you may use the optional HP 10269C General Purpose Probe Interface. The HP 10269C allows you to connect the probe cables, without the probes, to connectors on the interface. When the appropriate preprocessor is installed in the interface, you will have a direct connection between the logic analyzer and the microprocessor under test. See figure 2-1 for a basic block diagram.

There are a number of microprocessor specific preprocessors available as optional accessories which are listed in the *Accessories for HP Logic Analyzers* data sheet. Appendix A of this manual also introduces you to preprocessors and inverse assemblers.

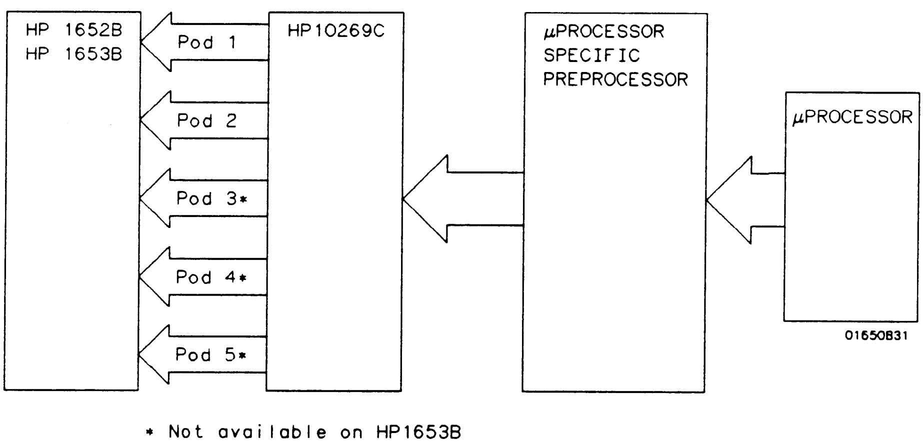


Figure 2-1. HP 10269C with Preprocessor

General Purpose Probing

General purpose probing involves connecting the logic analyzer and oscilloscope probes directly to your target system without using any interface. General purpose probing does not limit you to specific hook up schemes, for an example, as the probe interface does.

The Termination Adapter

The optional termination adapter (HP part number 01650-63201) allows you to connect the logic analyzer probe cables directly to test ports on your target system without the probes. However, since the probes contain the proper termination for the logic analyzer inputs, a termination must be provided.

The termination adapter shown below, is designed to connect to a 20 (2x10) position, 4-wall, low profile header connector, 3M® Series 3592 or equivalent.

To hook up the adapter, connect the termination adapter to the analyzer probe cable. Connect the other end of the adapter directly to your test port.

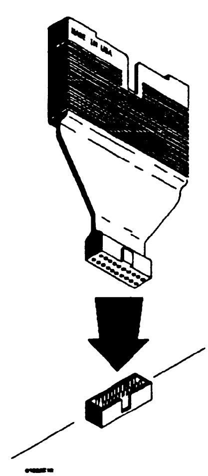


Figure 2-2. Termination Adapter

The HP1652B/1653B Probing System

The standard HP 1652B/53B probing system consists of logic analyzer probes and oscilloscope probes. Both have a passive design which means there are no active circuits at the outer end of the cable. The passive design also enables the pods and probes to be smaller and lighter, there by making them easier to use.

The logic analyzer probing system consists of flat ribbon probe cables, a probe housing, probe leads, ground leads and grabbers. This passive probing system is similar to the probing system used with high frequency oscilloscopes. It consists of a series R-C network (100 kΩ in parallel with 8 pF) at the probe tip, and a shielded resistive transmission line. The advantages of this system include the following:

- 2 ns risetime with ±5% perturbations
- 8 pF input capacitance at the probe tip
- Signal ground at the probe tip for higher speed timing signals
- Inexpensive removable probe tip assemblies

Probe Pod Assemblies

Probes and probe pod assemblies allow you to connect the logic analyzer to your system under test without the HP 10269C Probe Interface. This general purpose probing is useful for discrete digital circuits. Each pod, as they will be referred to for consistency, contains, 16 probes (data channels), one clock channel, and a pod ground. See the figure below.

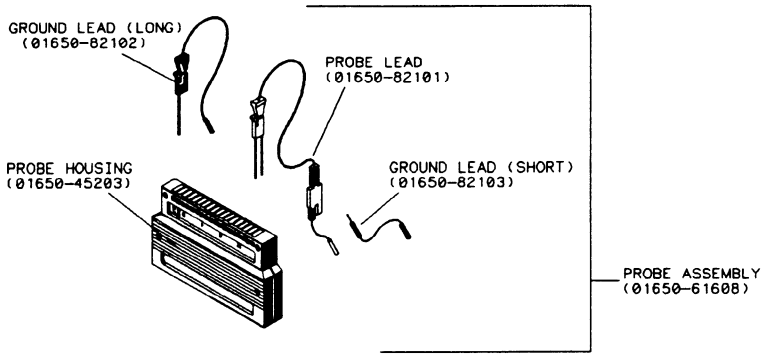


Figure 2-3. Probe Pod Assembly

Pod Grounding

Each pod is grounded by a pod ground lead that should always be used. You can connect the ground lead directly to a ground pin on your target system or use a grabber. The grabber connects to the ground lead the same way it connects to the probe lead. To connect the ground lead to grounded pins on your target system, you must use 0.63 mm (0.025 in.) square pins or round pins with a diameter of 0.66 mm (0.026 in) to 0.84 mm (0.033 in).

Probes

The probe consists of a 12-inch twisted pair cable and one grabber. The probe tip, which connects to the target system, has an integrated R-C network with an input impedance of 100 kΩ in parallel with approximately 8 pF. See figure 2-4 below.

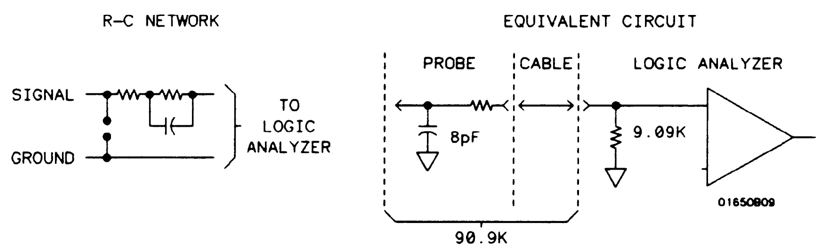
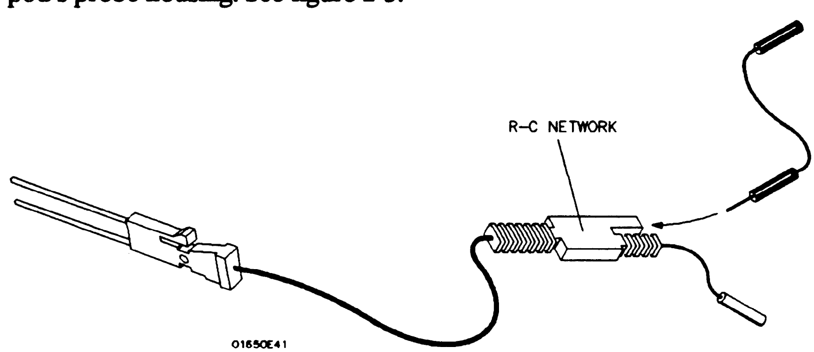


Figure 2-4. Probe Input Circuit

The other end of the probe has a two-pin connector that snaps into the pod's probe housing. See figure 2-5.



2-5. Probe

Probe Grounding

You can ground the probes in one of two ways. You can ground the probes with the pod ground only; however, the ground path won't be the same length as the signal path through the probe. If your probe ground path must be the same as your signal path, use the short ground lead (probe ground). The probe ground lead connects to the molded probe body via a pin and socket. You can then use a grabber or grounded pins on your target system the same way as the pod ground.

Note

For improved signal fidelity, use a probe ground for every four probes in addition to the pod ground.

If you need additional probe ground leads, order HP part number 5959-9334 from your nearest Hewlett-Packard sales office.

Grabbers

The grabbers have a hook that fits around IC pins and component leads and connects to the probes and the ground leads. The grabbers have been designed to fit on adjacent IC pins.

Probe Cable

The probe cable contains 17 signal lines, 17 chassis ground lines and two power lines (for preprocessor use) that are woven together into a flat ribbon that is 4.5 feet long. The probe cable connects the logic analyzer to the pods, termination adapter, or the HP 10269C General Purpose Probe Interface.

Both ends of the cable are alike, so you can connect either end to the pods or logic analyzer. Each cable is capable of carrying 0.60 amps for preprocessor power.

Caution

DO NOT exceed this 0.60 amps per cable or the cable will be damaged. Also, the maximum power available from the logic analyzer (all cables) is 2 amps at 5 volts.

Note

Preprocessor power is protected by a current limiting circuit. If current exceeds 2.3 amps, the circuit will open. The current limiting circuit will try to reset itself every 20 ms until the shorted condition is fixed.

Oscilloscope Probes

The two oscilloscope probes supplied with the HP 1652B/1653B Logic Analyzer are the HP 10433A Miniature Passive Probes. These small, lightweight probes allow measurements that were previously very difficult in densely populated circuits.

For complete information on the operation, maintenance, and adjustments of the miniature passive probes, be sure to read the operating note that is packaged with the probes.

Probe Inputs

Probe inputs are located on the front panel below the Knob. Input 1 (CH 1) is on the left. The probes may be connected directly to the BNC input connectors. The signal is dc coupled to the oscilloscope.

BNC cables can be connected directly to the BNC connectors. The HP 10503A 1.2 meter BNC-to-BNC cable is not provided with the instrument, but, you can order it, separately.

External Trigger BNCs

Inputs. The External Trigger Input allows the analyzer/scope trigger to be armed from an external TTL compatible source. Arming occurs when the normally active high status of the BNC is pulled low.

Outputs. The External Trigger Output provides the user access to the analyzer/scope trigger output pulse. The output pulse is a TTL compatible positive going pulse, that remains high from the time of trigger until the acquisition cycle is complete.

BNC cables can be connected directly to the BNC connectors. The HP 10503A 1.2 meter BNC-to-BNC cable is not provided with the instrument, but, you can order it, separately.

Compensation Signal Output

The Compensation Signal Output BNC is located on the rear panel. The Compensation Signal 50Ω output is ~1.2 kHz square wave with high amplitude near -200 mV and low amplitude near -400 mV when connected to a 50Ω load. This square wave is used for probe compensation adjustment (see your operating note for more information about probing) and is used in examples throughout this manual.

Signal Line Loading

Any signal line you intend to probe with the logic analyzer probes, must supply a minimum of 600 mV to the probe tip. The probes have an input impedance of 100 kΩ shunted by 8 pF. If the signal line is incapable of this minimum voltage, you will not only have an incorrect measurement, but the system under test may also malfunction.

Maximum Probe Input Voltage

The maximum input voltage of each logic analyzer probe is ±40 volts peak.

The maximum input voltage of the oscilloscope probes is ±250 volts dc at 1 MΩ setting and 5 volts rms at 50 Ω setting.

Pod Thresholds

Logic analyzer pods have two preset thresholds and a user-definable pod threshold. The two preset thresholds are ECL (- 1.3 V) and TTL (+ 1.6 V). The user-definable threshold can be set anywhere between - 9.9 volts and + 9.9 volts in 0.1 volt increments.

The pod thresholds of pods 1 and 2 in the HP 1653B and of pods 1, 2, and 3 in the HP 1652B can be set independently. The pod thresholds of pods 4 and 5 in the HP 1652B are slaved together. Therefore, when you set the threshold on either pod 4 or 5, both thresholds will be the same.

Connecting the Logic Analyzer to the Target System

There are four ways you can connect the logic analyzer to your target system: the probes (general purpose probing); the HP 10320C User-definable Interface; the HP 10269C with microprocessor specific preprocessor modules; and direct connection to a 20 pin 3M® Series type header connector using the optional termination adapter (HP part number 01650-63201).

Since the probe interface hookups are microprocessor specific, they will be explained in their respective microprocessor operating notes. The rest of this chapter is dedicated to general purpose probing with the logic analyzer probes.

Connecting the Probe Cables to the Logic Analyzer

You connect the probe cables to the probe cable connectors located on the rear panel of the logic analyzer. The probe cable connectors are keyed for proper orientation. You can connect either end of the cable to the rear panel since both ends of the cables are alike.

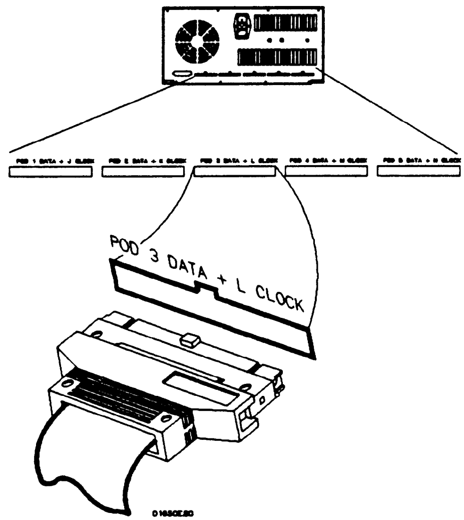


Figure 2-6. Probe Cable to Analyzer Connection

Connecting the Pods to the Probe Cable

The analyzer pods of the HP 1652B/53B differ from other logic analyzers in that they are passive (have no active circuits at the outer end of the cable). The pods, are the connector bodies (as shown below) that the probes are installed in when you receive your logic analyzer.

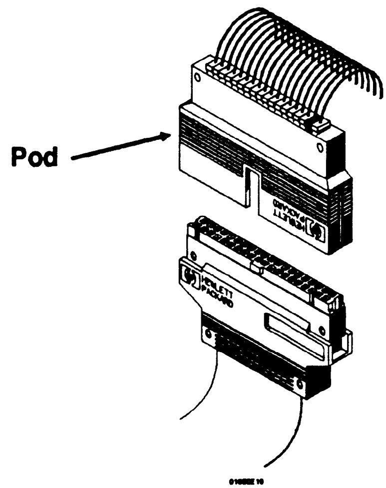


Figure 2-7. Connecting Pods to Probe Cables

To connect a pod to a cable, align the key on the cable connector with the slot on the pod connector and press together.

Disconnecting the Probes from the Pods

When you receive the logic analyzer, the probes are already installed in the pods. To keep them out of your way, disconnect them from the pod.

To disconnect a probe, insert the tip of a ball-point pen into the latch opening. Push on the latch while gently pulling the probe out of the pod connector as shown below.

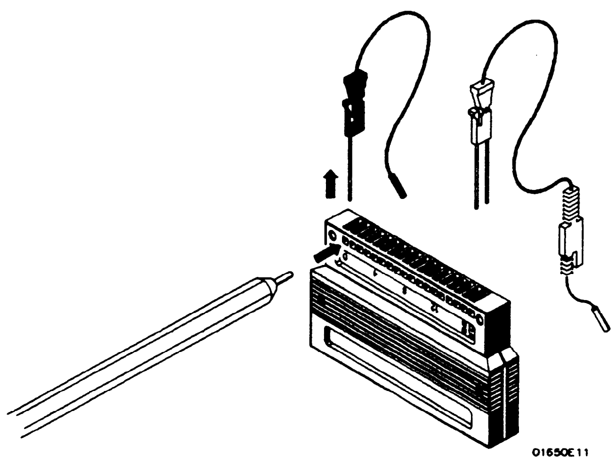


Figure 2-8. Disconnecting Probes From Pods

You connect the probes to the pods by inserting the double pin end of the probe into the pod. The probes and pod connector body are both keyed (beveled) so that they will fit together only one way.

Connecting the Grabbers to the Probes

Connect the grabbers to the probes by slipping the connector at the end of the probe onto the recessed pin located in the side of the grabber. If you need to use grabbers for either the pod or the probe grounds, connect the grabbers to the ground leads in the same manner.

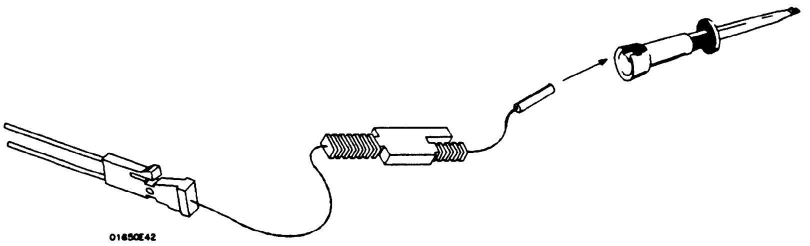


Figure 2-9. Connecting Grabbers to Probes

Connecting the Grabbers to the Test Points

The grabbers have a hook that fits around the IC pins and component leads. Connect the grabber to the test point by pushing the rear of the grabber to expose the hook. Hook the lead and release your thumb as shown below.

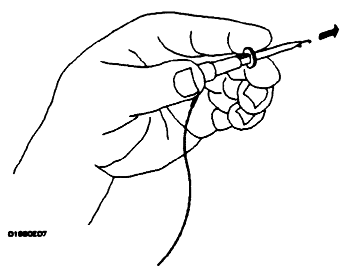


Figure 2-10. Connecting Grabbers to Test Points

Labeling Pods, Probes, and Cables

Included with your logic analyzer are self-adhesive labels for each pod, cable and probe. Use these sets of labels for identification.

Each set has labels for each end of the cable, a label for the probe housing, a label for the clock probe and 15 labels for each of the channels. The figure below, shows the correct placement of the labels.

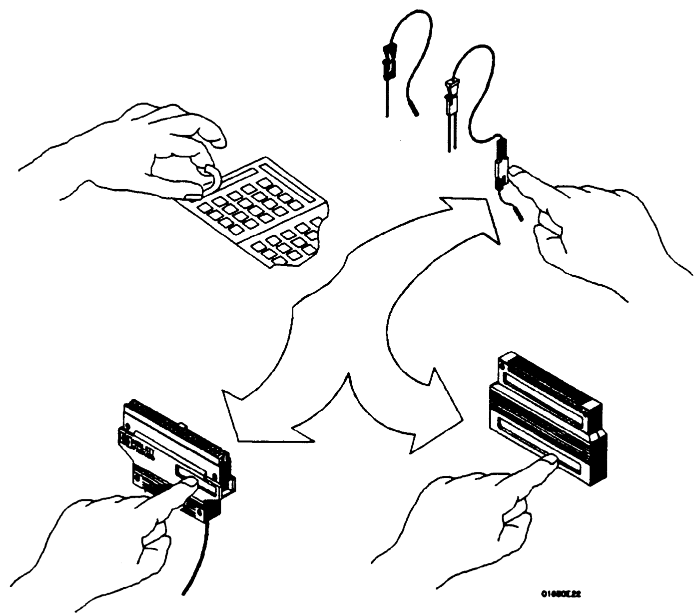


Figure 2-11. Labeling Pods, Probes and Cables

3

Using the Front-Panel User Interface

Introduction

This chapter explains how to use the front-panel user interface. The front and rear-panel controls and connectors are explained in the first part of this chapter followed by "How to use..." explanations of the front-panel user interface.

The front-panel user interface consists of front-panel keys, the KNOB, and display. The interface allows you to configure the logic analyzer, oscilloscope and each analyzer (machine) within the logic analyzer. It also displays acquired data and measurement results.

Using the front-panel user interface involves the following processes:

- Selecting the desired menu with the menu keys.
- Placing the cursor on the desired field within the menu by rotating the KNOB.
- Displaying the field options or current data by pressing the SELECT key.
- Selecting the desired option by rotating the KNOB or entering new data by using the KNOB or the keypad.
- Starting and stopping data acquisition by using the RUN and STOP keys.

Front-Panel Controls

In order to apply the user interface quickly, you should know what the front-panel controls do.

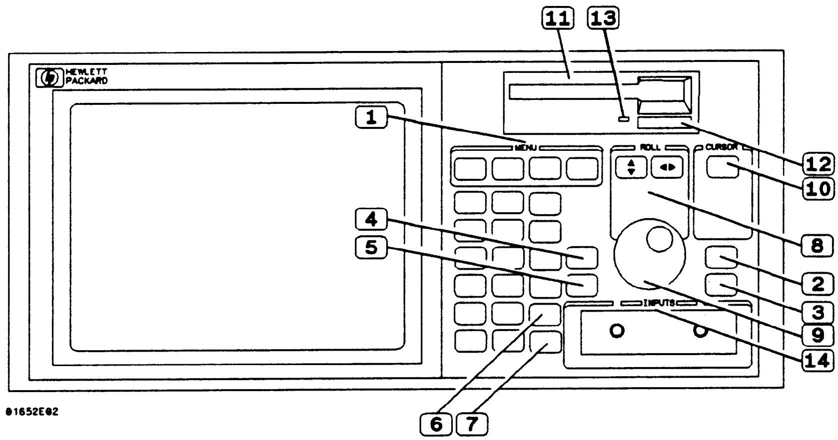


Figure 3-1. HP 1652B/53B Front Panel

(1) **Menu Keys.** The menu keys allow you to select the main menus in the logic analyzer. These keys are FORMAT/CHAN, TRACE/TRIG, DISPLAY, and I/O. The Format/Channel, Trace/Trigger, and Display keys will display the menus of either analyzer (machine) 1 or 2 respectively or the oscilloscope depending on what menu was last displayed or what you did in the System Configuration menu.

Format/Channel Menu Key. The FORMAT/CHAN menu key allows you to access either the Timing Format Specification, State Format Specification, or Oscilloscope Channel menus. You exit the Format/Channel menu by pressing another menu key or by returning to the System Configuration menu from this menu.

Trace/Trigger Menu Key. The TRACE/TRIG menu key allows you to access either the Timing Trace, State Trace, or Oscilloscope Trigger menus. You exit the Trace/Trigger menu by pressing another menu key or by returning to the System Configuration menu from this menu.

Display Menu Key. The DISPLAY menu key allows you to access either the Timing Waveforms display, State Listing display, or the Oscilloscope Waveforms display. You exit the Timing Waveforms, State Listing, and Oscilloscope Waveforms menus by pressing another menu key or by returning to the System Configuration menu.

I/O Menu Key. The I/O menu key allows you to access the I/O menu. You can access the I/O menu from any menu in either analyzer (timing or state) or oscilloscope, and at any time. Pressing the I/O menu key causes the I/O menu to pop up over any current menu on the display.

(2) **Run Key**. The RUN key allows you to initiate a data acquisition and display cycle. The analyzer (state or timing) is automatically forced into its display menu when a run is initiated. The oscilloscope will stay in its current menu when a run is initiated. The trace mode or run mode you select (in the Trace/Trigger menu) determines whether a single or multiple (repetitive) run occurs.

(3) **Stop key**. The STOP key allows you to stop data acquisition or printing. A single press always stops the data acquisition. The data displayed on screen depends on which acquisition mode (single or repetitive) was used to acquire the data. In the repetitive mode, STOP causes the old display to remain unchanged as long as the old data is not corrupt. In single mode, STOP causes any new data to be displayed. If printing a hardcopy, the STOP key stops the print.

(4) **Don't Care Key**. The DON'T CARE key allows you to enter don't cares in binary octal, and hexadecimal pattern specification fields. In Alpha Entry fields, this key enters a space and moves the underscore marker to the next space.

(5) **Clear Entry Key**. The CLEAR ENTRY key allows you to perform the following tasks:

- Return decimal values to the previous value in the decimal menu fields.
- Return values to don't cares in menu fields with number bases other than decimal.
- Clear Alpha Entry menus.
- Move the underscore marker or cursor to its original position in the menu fields.

(6) **Hex(adecimal) Keypad.** The HEX keypad allows you to enter numeric values in numeric entry fields. You enter values in the four number bases below:

- Binary
- Octal
- Decimal
- Hexadecimal

The A through F keys are used for both hexadecimal and alpha character entries.

(7) **CHS Key.** The CHS (change sign) key allows you to change the sign (±) of numeric variables.

(8) **Roll Keys.** When part of the data display is off screen, the ROLL keys define which way the KNOB will move the displayed data. These keys and the KNOB roll displayed data up/down or left/right so you can view off-screen data.

(9) **Knob.** The KNOB has four major functions depending on what menu or pop-up menu you are in. The KNOB allows you to do the following:

- Move the cursor from field to field within the System Configuration and main menus.
- Roll the display left or right and up or down.
- Position the cursor on options within pop-up menus.
- Increment/decrement numeric values in numeric pop-up menus.

(10) **Select Key.** The SELECT key allows you to open pop-up menus, choose options in them, cancel selections, and close pop-up menus. When the cursor is in a main menu (i.e. Format Specification) pressing the SELECT key either opens a pop-up, or toggles options (when there are only two options possible) in that field.

When a pop-up menu appears, the cursor will be on the current option. You use the KNOB to move the cursor to your desired option. Pressing the SELECT key tells the logic analyzer this is the option you want. This either automatically selects the option and closes the pop-up, opens another pop-up, or changes options. If the pop-up doesn't automatically close, it will contain the Done field. In this case you close the pop-up by placing the cursor on Done and pressing SELECT.

(11) **Disk Drive.** A 3.5 inch, double-sided, double density drive. Besides loading the operating system, it allows you to store and load logic analyzer configurations and inverse assembler files.

(12) **Disk Eject Button.** Press this button to eject a flexible disk from the disk drive.

(13) **Indicator Light.** This light is illuminated when the disk drive is operating. Wait until this light is out before removing or inserting disks.

(14) **Inputs 1 and 2.** Two BNC connectors allow the connection of oscilloscope probes and BNC cables for signal input to the oscilloscope.

Rear-Panel Controls and Connectors

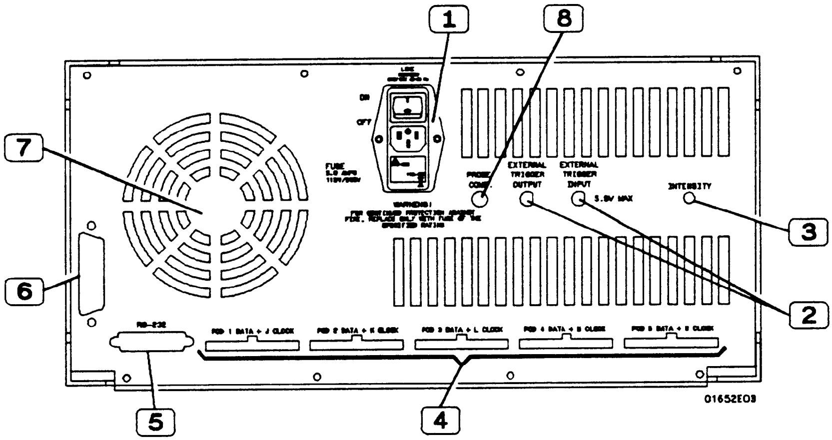


Figure 3-2. HP 1652B/53B Rear Panel

(1) **Line Power Module.** Permits selection of 110-120 or 220-240 Vac and contains the fuses for each of these voltage ranges. The On/Off switch is also part of the module.

(2) **External Trigger BNCs.** Provide arm out and arm in connections.

(3) **Intensity Control.** Allows you to set the display intensity to a comfortable level.

(4) **Pod Cable Connectors.** Keyed connectors for connecting the pod cables.

Note The HP 1653B rear panel has connectors for pods 1 and 2 only.

---

(5) **RS-232C Interface Connector.** Standard DB-25 type connector for connecting an RS-232C printer or controller.

(6) **HP-IB Interface Connector.** Standard HP-IB connector for connecting an HP-IB printer or controller.

(7) **Fan.** Provides cooling for the logic analyzer. Make sure air is not restricted from the fan and rear-panel openings.

(8) **Probe Compensation Signal Output.** Provides a signal for probe compensation adjustment.

The Cursor

The cursor (inverse video) highlights interactive fields within the menus that you want to use. Interactive fields are enclosed in boxes in each menu. When you rotate the KNOB, the cursor moves from one field to another.

How to Select Menus

You select the main menus by pressing the appropriate menu key. The main menu keys are:

- FORMAT/CHAN
- TRACE/TRIG
- DISPLAY
- I/O

When the menu is displayed, you can access fields within the menus.

The FORMAT/CHAN, TRACE/TRIG, and DISPLAY menu keys provide access to their respective menus. If more than one analyzer (machine) is on, or the oscilloscope is on, you see the selected menu of either analyzer 1, analyzer 2 or the oscilloscope depending on what type menu was last displayed (analyzer or scope), or what you did in the System Configuration menu. To switch from the machine 1 menu set to machine 2 (same analyzer) menu set or the oscilloscope menu set, select the desired analyzer or scope from the pop-up that appears when the field in the upper left corner of the main menu is selected. This pop-up is available in all main menus except the I/O menu.

The I/O menu differs from the other three main menus in that it is a pop-up menu that appears on top of the currently displayed menu when you press the I/O key.

How to Switch between the Analyzers and Oscilloscope

You can switch between the analyzers and oscilloscope in any main menu except the I/O menu. To switch between analyzers and scope, place the cursor on the field in the upper left corner of the FORMAT/CHAN, TRACE/TRIG, or DISPLAY (timing, state or scope) menu and press SELECT. A pop-up menu appears with the following options:

- System
- MACHINE 1 (or your analyzer name)
- MACHINE 2 (or your analyzer name)
- Mixed Mode (if two or more are on)
- Scope

Place the cursor on the opposite analyzer (machine), or scope and press SELECT. The logic analyzer will display the same menu type (i.e. format, trace, etc.) in the other analyzer (machine) or the scope menu. For example, if you were in the TRACE menu of machine 1, you will now see the TRIGGER menu of the scope or the TRACE menu of machine 2.

Returning to the System Configuration Menu

You can return to the System Configuration menu directly from the FORMAT, TRACE, or DISPLAY menus. To return to the System Configuration menu, place the cursor on the field in the upper left corner of any of these menus and press SELECT. The same pop-up menu appears with the following options:

- System
- MACHINE 1 (or your analyzer name)
- MACHINE 2 (or your analyzer name)
- Mixed Mode (if two or more are on)
- Scope

Place the cursor on System and press SELECT. The System Configuration menu is displayed.

How to Select Fields

You select fields within the main menus by placing the cursor on the desired field and pressing SELECT. Depending on what type of field you select, you will either see a pop-up menu or a new option in fields that toggle.

Pop-up Menus

The pop-up menu is the most common type of menu you see when you select a field. When a pop-up appears, you will see a list of two or more options. Two pop-up menu types are described in "How to Select Options" in this chapter.

How to Close Pop-up Menus

Pop-up menus without the Done option automatically close when you place the cursor on an option and press SELECT. After closing, the logic analyzer places your choice in the main menu field from which you opened the pop-up.

Pop-up menus that contain the Done option do not automatically close when you make your selection. To close the pop-up, you place the cursor on the Done option and press SELECT.

These two pop-up menu types are described in "How to Select Options" in this chapter.

How to Select Options

How to select options depends on what type of pop-up menu appears when you press select. When the pop-up appears, you will see a list of options. You select the option you want by placing the cursor on it and pressing SELECT. In most cases the pop-up menu closes and your desired option is now displayed in the field in the main menu.

There are also pop-up menus where each option within the pop-up menu has more than one option available. In these cases, when you place the cursor on one of the options and press SELECT, another pop- up will appear.

An example of one of these is the clock field in the State Format Specification menu. When you select the **clock field** in this menu it will pop-up and show you all five clocks (J, K, L, M, and N) for an HP 1652B or both clocks (J and K) for an HP 1653B.

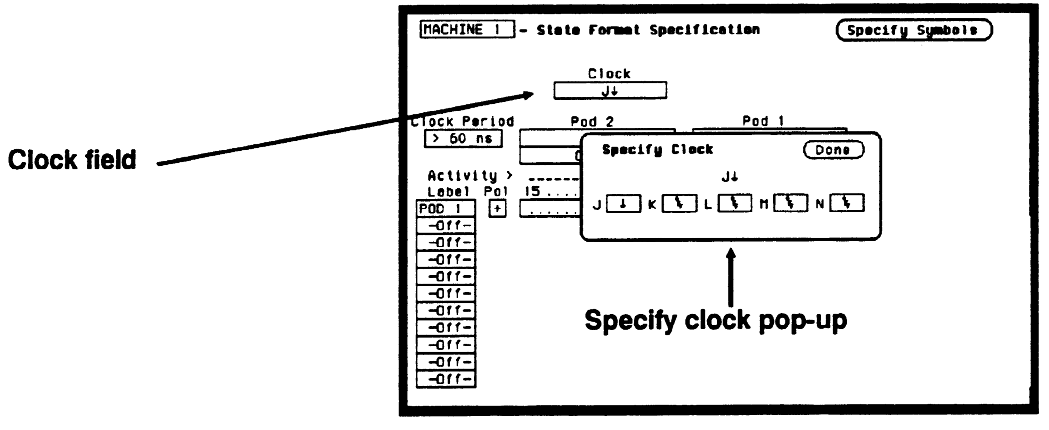


Figure 3-3. State Clock Pop-up Menu

When you place the cursor on one of the clocks and press SELECT, another pop-up appears, showing you the choices of clock specifications available.

Clock specification pop-up

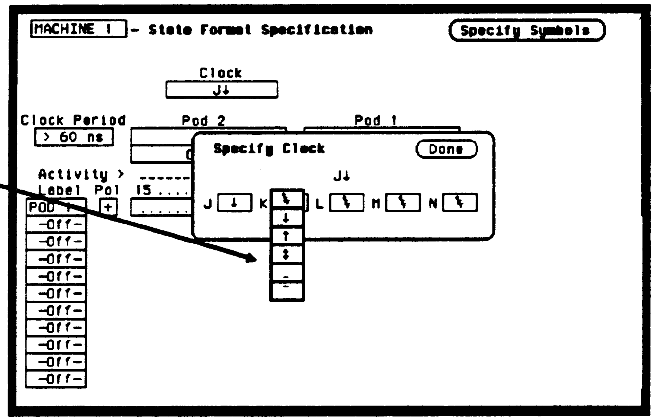


Figure 3-4. State Clock Pop-up with K Pop-up Open

When you choose one of these specifications and press SELECT, this pop-up will close, however, the original clock pop-up still remains open. When finished specifying the choices for the clocks, you close the original pop-up menu by selecting Done and pressing SELECT.

Toggle Fields

Some fields will toggle between two options "off" and "on". When you place the cursor on one of these fields and press SELECT, the displayed option toggles to the other choice and no additional pop-up appears.

How to Enter Numeric Data

There are a number of pop-up menus in which you enter numeric data. The two major types are as follows:

- Numeric entry with fixed units (i.e. volts).
- Numeric entry with variable units (i.e. ms, μs, etc.).

An example of a numeric entry menu in which you only enter the value with fixed units is the pod threshold pop-up menu.

You can set the pod thresholds to either of the preset thresholds (TTL or ECL) or to a specific voltage from - 9.9 V to + 9.9 V.

To set pod thresholds to a specific voltage, place the cursor in the threshold portion of the pod field (TTL, ECL, or User-defined) of any pod and press SELECT.

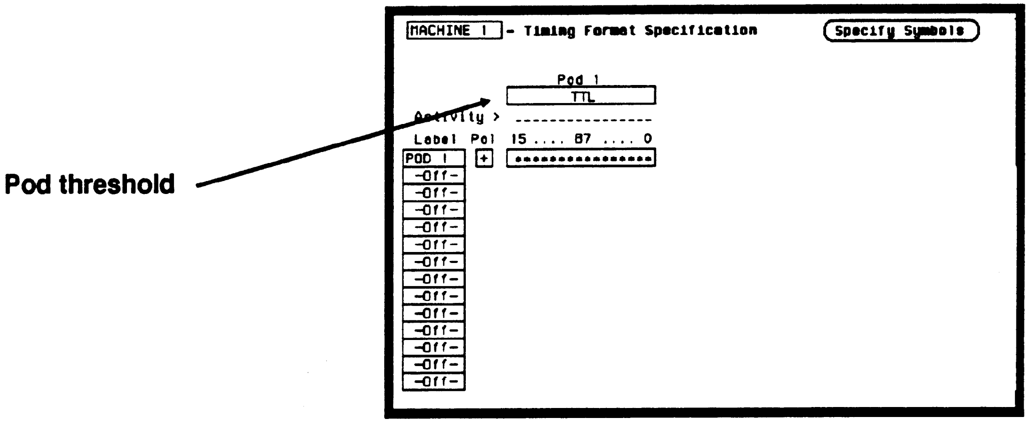


Figure 3-5. Pod Threshold

Select the **User-defined** option and another pop-up appears for you to specify the pod threshold voltage.

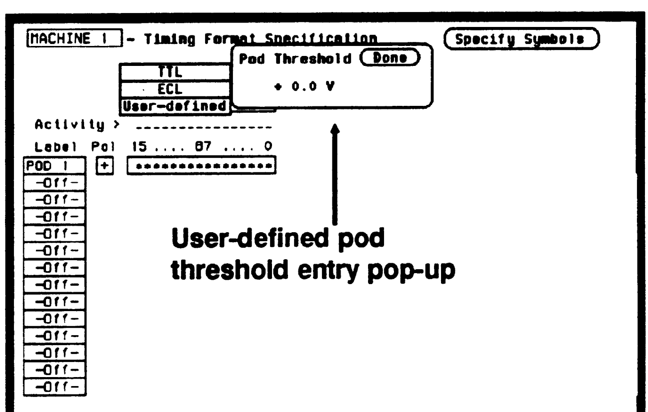


Figure 3-6. User-Defined Pop-up

You can select your desired threshold by rotating the KNOB until your desired threshold voltage is displayed. Rotating the KNOB increments or decrements the value in small steps. Or you can change the value with the keypad. It allows you to make large value changes quickly. Entering the new value from the keypad replaces the previous value.

If you want a negative voltage for the threshold, press the CHS (change sign) key on the front panel. The minus (−) sign will appear in the pop-up.

Notice, the cursor stays in the upper right corner of the pop-up over Done. When you press SELECT, the pop-up will close and your new threshold will be placed in the Pod field.

In another type of numeric entry pop-up menu you must specify the units as well as the numeric value. The pattern duration specification in the Timing Trace Specification menu is an example. When you place the cursor on the value in the **present for** _____ field and press SELECT, you will see the following pop-up:

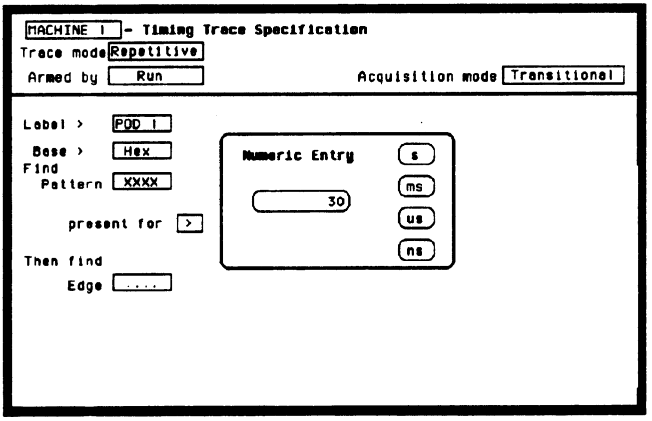


Figure 3-7. Numeric Entry Pop-up

You enter a new value from the keypad. When you have entered your desired value, you can change the units (i.e., ns, μs, ms, s) by rotating the KNOB.

Once you select the new value and the units, close the pop-up by pressing SELECT. The new value and the units will be displayed in the **present for** ______ field.

In all numeric entry fields except the pod threshold field, you can open the pop-up without pressing SELECT. To open the pop-up without pressing SELECT, place the cursor on the field and press any number that particular field accepts. The pop-up will appear with the new number in the pop-up.

Note Any time the cursor is on one of the numeric entry fields and you unintentionally press a key that the field accepts, the pop-up will appear and the number you pressed will replace your current value. To close the pop-up and return the original value, press the CLEAR ENTRY key.

How to Enter Alpha Data

You can customize your analyzer configuration by giving names to several items:

- The name of each analyzer.
- Labels.
- Symbols.
- Filenames.
- File descriptions.

For example, you can give each analyzer a name that is representative of your measurement. The default names for the analyzers within the logic analyzer are MACHINE 1 and MACHINE 2. To rename an analyzer, place the cursor on the name you wish to change in the System Configuration menu and press SELECT. You will see the Alpha Entry pop-up menu:

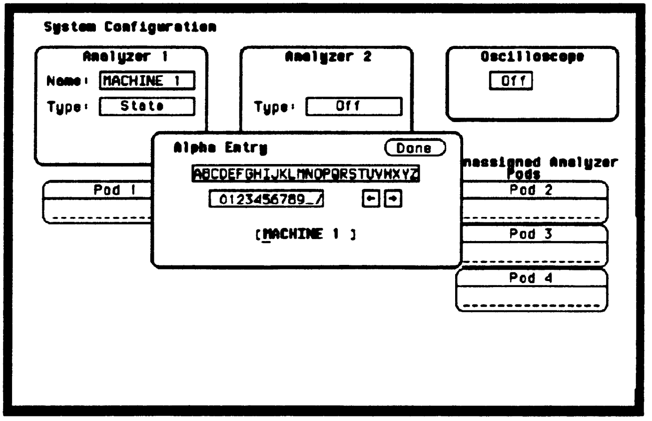


Figure 3-8. Alpha Entry Pop-up

The top two lines enclosed in boxes in the pop-up contain the complete alphanumeric set you use for names in these types of fields. The bottom line (enclosed in brackets) contains the name that existed when you opened the Alpha Entry pop-up. To enter alpha characters in the brackets (where the default or old name appears) position the cursor on the desired character and press SELECT. The new character will be placed in the brackets where the underscore marker is located. If you want to place a new character in the brackets at a location not marked by the underscore marker, move the underscore marker to where you want the new character to be placed. Moving the underscore marker is explained in "Changing Alpha Entries."

Note

You can also make direct keypad entries. Your selection will be placed where the underscore marker is in the box.

Changing Alpha Entries

To make changes or corrections in the Alpha Entry field, position the underscore marker under the character you want to change.

To move the underscore marker to the left, place the cursor over the left arrow and press SELECT once for each backspace.

To move the underscore marker to the right, you either place the cursor on a desired character and press SELECT, or place it on the right arrow and press SELECT.

You can also use the ROLL keys and the KNOB to move the underscore marker. To use this alternate method press the left/right ROLL key and rotate the KNOB until the underscore marker is under the desired character. To return the KNOB to controlling the cursor's movement, press the left/right ROLL key again or press SELECT.

If you want to erase the entire entry and place the underscore marker at the beginning of the name box, press the CLEAR ENTRY key on the front panel.

If you want to replace a character with a space, place the underscore marker under that character and press the DON'T CARE key on the front panel.

How to Roll Data

To roll data, you press either the left/right or up/down ROLL keys and rotate the KNOB. The roll function is only available when there is more data in the menu than can fit on screen. If there is off-screen data, pressing the ROLL keys causes an indicator to appear in the upper left corner of the display and activates the roll function of the KNOB. If there is no off-screen data, the indicator will not appear.

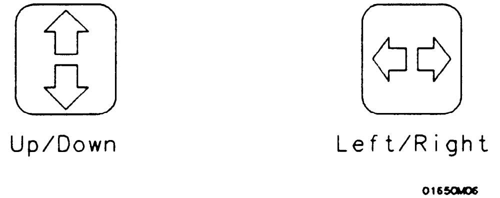


Figure 3-9. Roll Function Keys

One example of a menu with off-screen data is the STATE LISTING menu. The state listing can contain up to 1024 lines; however, the display is only capable of showing you 16 lines at a time. To roll the off-screen data, press the up/down ROLL key and then rotate the KNOB to view the off-screen data.

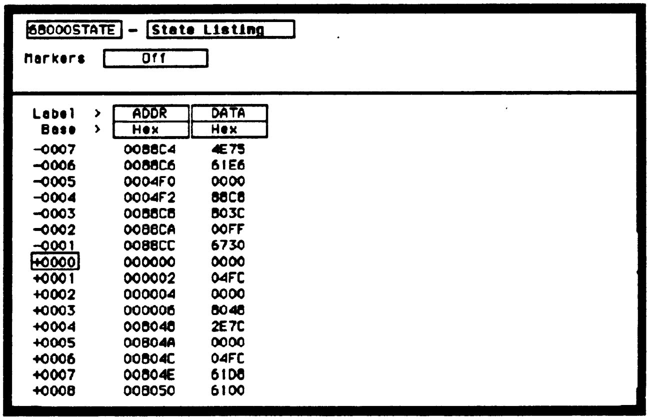


Figure 3-10. Typical State Listing Menu

Assignment/ Specification Menus

There are a number of pop-up menus in which you assign or specify what you want the logic analyzer to do. The basic menus of this type are as follows:

- Assigning pod bits to labels
- Specifying patterns
- Specifying edges

Assigning Pod Bits to Labels

The bit assignment fields in both state and timing analyzers work identically. The convention for bit assignment is as follows:

* (asterisk) indicates assigned bits
. (period) indicates un-assigned bits

Note

If you don't see any bit assignment fields, it merely means you do not have any pods assigned to this analyzer. Either switch analyzers or assign a pod to the analyzer you are working with.

To assign bits in these menus, place the cursor on one of the bit assignment fields and press SELECT. You will see the following pop-up menu:

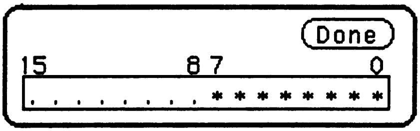


Figure 3-11. Bit Assignment Pop-up

Place the cursor on the left-most asterisk or period in the pop-up that you want to change and press SELECT. The bit assignment toggles to the opposite state of what it was when the pop-up first opened. Move the cursor one bit to the right. Holding the SELECT key, repeats the bit assignment. You close the pop-up by placing the cursor on Done and pressing SELECT.

Specifying Patterns

The Specify Patterns fields appear in several menus in both the timing and state analyzers. Patterns can be specified in one of the available number bases, except ASCII.

The convention for "don't cares" in these menus is an "X" except in the decimal base. If the base is set to decimal after a "don't care" is specified, a $ character is displayed.

An example of a Specify Patterns field is the **Find Pattern____** field in the Timing Trace Specification menu.

When you place the cursor on the **Find Pattern** ____ field and press SELECT, you will see the following pop-up menu appear.

Figure 3-12. Find Pattern ________ field Pop-up

When the pop-up is open, enter your desired pattern from the keypad (including don't cares). When you finish entering your pattern, close the pop-up by pressing SELECT.

Specifying Edges

You can select positive-going (↑), negative-going (↓), or either edge (↕) as part of your trigger specification. You specify edges in the Timing Trace Specification menu by placing the cursor on the **Then find Edge** field under the desired label and pressing SELECT. You will see the following menu.

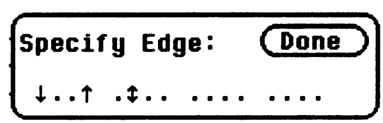


Figure 3-13. Edge Pop-up

You will notice a number of periods in the pop-up menu. Each period represents an unassigned bit for each bit assigned to the label. Don't be alarmed if you see a different number of unassigned bits, it merely means the number of bits in your label is different than the number in the label for this example.

To select a desired edge, place the cursor on your desired bit position in the pop-up and press SELECT until you see the desired edge, or unassign (.) the bit. Pressing SELECT changes the bit sequentially from (.) to ↓ to ↑ to ↕ and back to (.).

4

System Configuration Menu

Introduction

This chapter describes the System Configuration menu and pop-up menus within the System Configuration menu.

The purpose and functions of each field are explained in detail, and we have included illustrations and examples to make the explanations clearer.

System Configuration Menu

The System Configuration menu can be considered a system level menu in that it contains fields that you use to turn the scope on or off and start the configuration process for both analyzer 1 and analyzer 2. You use this menu to do the following:

- Turn analyzer machines and scope on or off.
- Specify analyzer type (timing and state).
- Assign pods to the individual machines within the logic analyzer.
- Initiate Autoscale in both the oscilloscope and timing analyzer.
- Name each analyzer.

In this menu, you configure your logic analyzer in one of nine ways:

- Timing analyzer only.
- State analyzer only.
- Up to two scope channels.
- Two state analyzers.
- One timing analyzer and one state analyzer.
- Timing analyzer with up to two scope channels.
- State analyzer with up to two scope channels.
- Two state analyzers with up to two scope channels.
- One timing analyzer, one state analyzer and up to two scope channels.

The System Configuration menu for the HP 1652B Logic Analyzer is shown below.

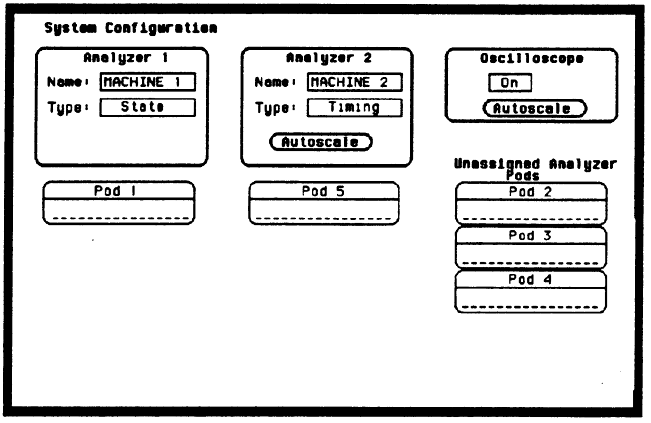


Figure 4-1. System Configuration Menu For HP 1652B

Accessing the System Configuration Menu

The System Configuration menu is the default display when the logic analyzer is turned on and the operating system has loaded. Once the logic analyzer or scope is on and you are in a menu other than the System Configuration menu, you access the System Configuration menu by placing the cursor in the system access field in the upper left corner and press SELECT. This field will be displaying either the scope, Machine 1, Machine 2, or a user-defined name for the current analyzer machine before you press SELECT.

You then place the cursor on System in the pop-up menu and press SELECT. When the pop-up closes the System Configuration menu will be displayed.

System Configuration Menu Fields

The System Configuration menu fields are described in the following paragraphs.

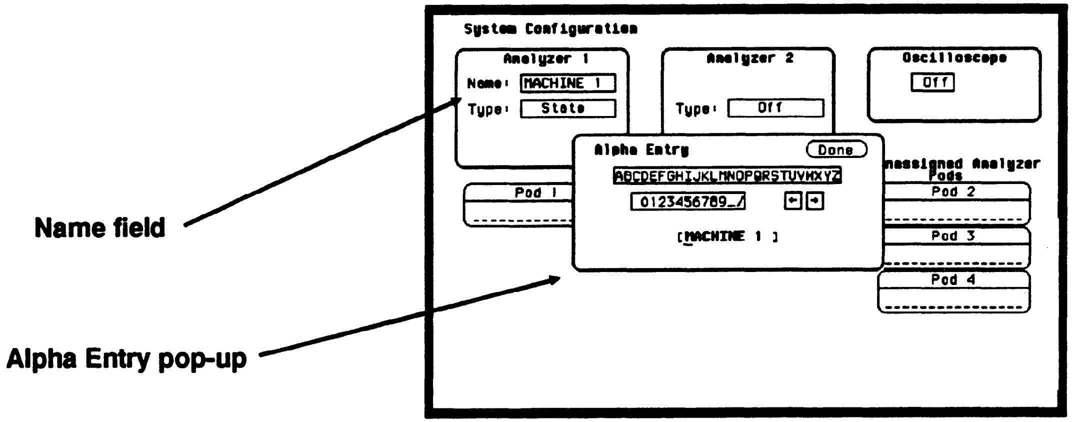


Figure 4-2. Alpha Entry Pop-up Menu

Name

You name an analyzer by selecting the **Name** field under it. An Alpha Entry pop-up menu will open as shown above. The pop-up contains a row of alpha characters, a row of numeric characters, two arrows, and a box at the bottom of the menu in which the name appears. In the name box is an underscore marker. This marker indicates in what space your next selection will be placed.

You can name the analyzer in one of two ways. The first way is to position the cursor over the desired character in the pop-up using the KNOB, then press SELECT. The character appears in the name box.

The second method is to use the keypad on the front panel. With this keypad you can enter the letters A through F and the numbers 0 through 9 instead of using the characters in the pop-up.

The arrows in the pop-up move the underscore marker forward or backward. To move the marker forward, position the cursor over the right-pointing arrow and press SELECT. To backspace the marker position the cursor over the left-pointing arrow and press SELECT.

You can also move the underscore marker with the ROLL keys and the KNOB. Pressing the left/right ROLL key activates the marker. Rotating the KNOB places the marker under the desired character.

You can replace a character with a space in one of two ways. Position the cursor over the space in the pop-up and press SELECT, or press the DON'T CARE key on the front panel.

If you want to erase the entire entry and place the underscore marker at the beginning of the name box, press the CLEAR ENTRY key on the front panel. When you have entered the correct name, position the cursor over Done and press SELECT.

Type

The **Type** field defines the machine as either a state analyzer or a timing analyzer. When this field is selected, a pop-up selector menu appears. You choose the machine type by using the KNOB to move the cursor within the menu to the desired selection and pressing SELECT.

Type Pop-up

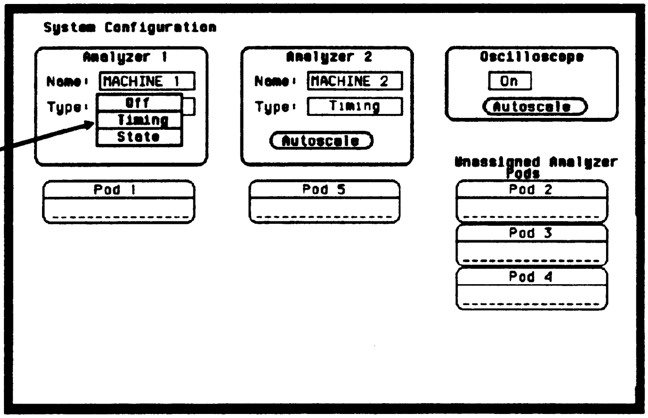


Figure 4-3. Type Pop-up Menu

Scope On/Off

The scope defaults to Off. To turn the scope on or off, simply move the cursor over the **On/Off** field and press select. Scope measurement may be added to any analyzer configuration.

Autoscale

Autoscale provides a starting point for setting up a measurement. The **Autoscale** field appears for the timing analyzer in the System Configuration menu only. When you select Autoscale, a pop-up appears with two options: Cancel and Continue. If you select **Cancel**, the autoscale is cancelled and control is returned to the System Configuration menu.

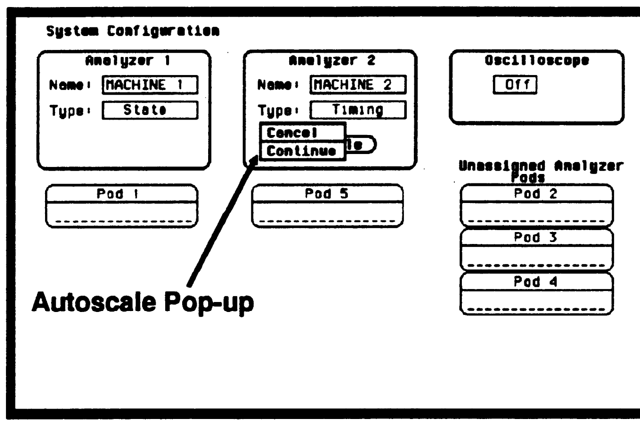


Figure 4-4. Autoscale Pop-up Menu

If you choose **Continue**, autoscale configures the Timing Format, Trace Specification, and the Timing Waveforms menus. Autoscale searches for channels with activity on the assigned pods and displays them in the Waveforms menu.

Autoscale for the scope is located in all main menus. When **Continue** is selected, the Channel, Trigger, and Waveforms displays are automatically configured. More information on scope autoscale is located in chapter 23, "Channel Menu."

Note

Choosing Autoscale erases all previous configurations in the timing analyzer and scope, and turns the other analyzer (state) off if it was on. If you don't want this to happen, select Cancel in the pop-up.

Pods

Each pod can be assigned to one of the analyzers. When the HP 1652B Logic Analyzer is powered up, Pod 1 is assigned to Analyzer 1 and Pod 5 is assigned to Analyzer 2. When the HP 1653B is powered up, Pod 1 is assigned to Analyzer 1 and Pod 2 is assigned to Analyzer 2.

To assign a pod, position the cursor on one of the pod fields and press SELECT. With the pop-up that appears, you can assign the pod to Analyzer 1 , Analyzer 2 , or Unassign it. Pressing the SELECT key closes the pop-up.

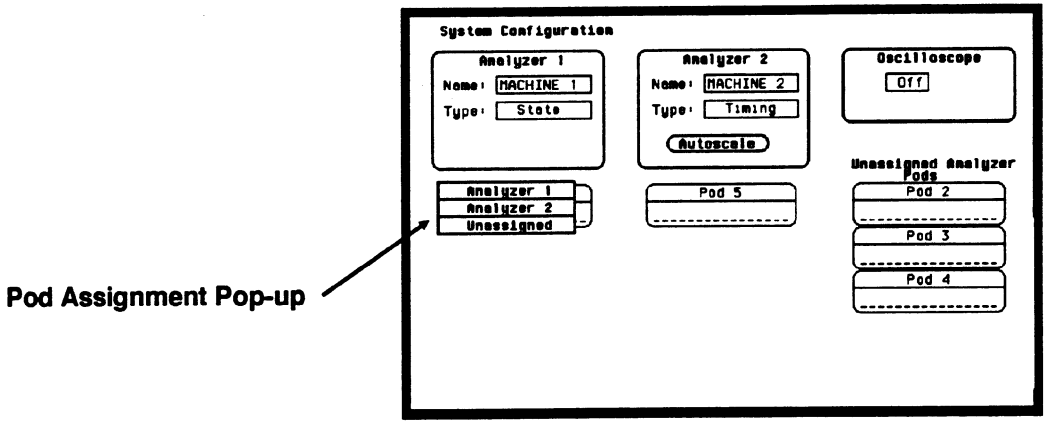


Figure 4-5. Pod Assignment Pop-up Menu

Where to Go Next

When you complete the system level configuration for the logic analyzer in this menu, you need to complete the individual analyzer configurations for analyzer 1, analyzer 2, or scope. To configure an individual analyzer you will normally configure the Format menu first and then the Trace menu. For the scope you configure the Channel menu first and then the Trigger menu.

Configuration menus for the timing analyzer start at chapter 16. For the state analyzer, menus start at chapter 8 and for the scope, start at chapter 22.

5

I/O Menu

Introduction

This chapter describes the I/O and pop-up menus that you will use on your logic analyzer. The purpose and functions of each menu are explained in detail, and we have included many illustrations and examples to make the explanations clearer.

The I/O menu allows you to perform I/O tasks with your logic analyzer. The tasks you can do with this menu areas follows:

- Print screens.
- Perform disk operations.
- Configure the HP-IB Interface.
- Configure the RS-232C Interface.
- Enable the analyzer to perform external triggering.
- Run self tests on the analyzer.

Accessing the I/O Menu

You can access the I/O menu from any other menu in the system by pressing the I/O key on the front panel. Use the KNOB to roll the cursor through the menu. When the cursor is positioned over the option you desire, press SELECT. It lists the following options:

- Done
- Print Screen
- Print All
- Disk Operations
- I/O Port Configuration
- External BNC Configuration
- Self Test

To exit the I/O menu, position the cursor over the **Done** option and press SELECT. This returns you to the menu you were in before you pressed the I/O key.

Print Screen

When you select the **Print Screen** option, the information on the screen is frozen and the message "PRINT in progress" appears at the top of the display. This message will not print. Only the STOP key is operational while data is being transferred to the printer. If you wish to stop a printout before it is completed, press the STOP key.

Print All

The **Print All** option prints not only what is displayed on screen but what is below, and, in the Format Specification, what is to the right of the screen at the time you initiate the printout.

Note

Make sure the first line you wish to print is on screen when you select Print All. Lines above screen will not print.

Use this option when you want to print all the data in menus like:

- Timing Format Specification
- State Format Specification
- State Trace Specification
- State Listing
- Disk Directory
- Symbols

If there is information below the screen, the information will be printed on multiple pages. In Timing and State Format Specifications, the print will be compressed when necessary to print data that is off-screen to the right.

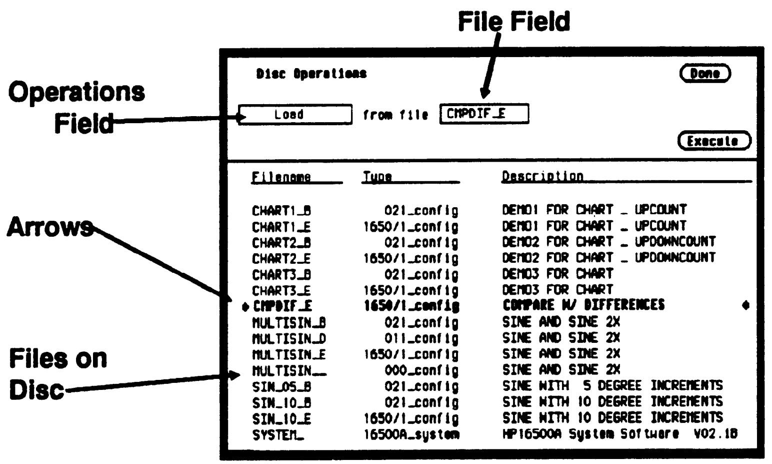
When you select the Print All option, the information on the screen is frozen, and the message "PRINT in progress" appears at the top of the display. This message will not print. If you wish to stop the printout before it is completed, press the STOP key on the front panel.

Disk Operations

The Disk Operations option allows you to perform operations on your disk and with the files on your disk. For example, you can load a file from your disk, store a file to your disk, or format a disk. The following pages describe the disk operations. For additional information on the disk operations, refer to Chapter 6, "Disk Drive Operations."

When you select Disk Operations , a new menu pops up. This menu is divided in two sections separated by a horizontal line. The top section displays the disk operation that is to be performed and the file or files that will be affected.

The bottom section displays the files on the disk in alphabetical order. It also states the type of the file and a description, if one was specified at storage. If no disk is in the disk drive or if the disk is not a supported format, the appropriate message will be displayed.



Halfway down the bottom display are arrows at each side of the screen. These arrows tell you which file is to be operated on. To roll through the list of files, press the up/down ROLL key and rotate the KNOB. The file that is between the arrows in boldface type also appears in the FILE field in the top section of the display.

The top section of the menu contains different types of fields. Pressing the Done field exits the Disk Operations menu and the I/O menu, returning you to the menu you were in before you pressed the I/O key. The field on the left-most side of the display is the operations field. It tells you which disk operation is to be performed. Next to that will usually be one or two file fields that tell you which file or files are to be acted upon. For several operations another field will appear in the top section.

The Execute field executes the disk operation appearing in the operations field. For non-destructive operations, when Execute is selected the operation is immediately performed. For destructive operations a pop-up appears with two options: Cancel and Continue. Cancel lets you change your mind before the action is taken preventing any data from being lost mistakenly. Continue executes the operation.

If you select the operations field, you will see a pop-up menu with nine options for disk operations, as shown. Each operation will now be discussed in detail.

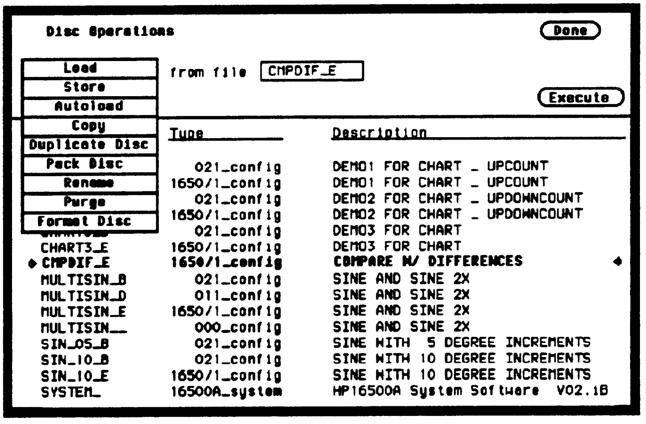


Figure 5-2. Disk Operations Pop-up Menu

Load

The Load operation allows you to load configuration files (including symbol tables), and inverse assemblers from a disk. Executing a Load operation loads the logic analyzer with the file whose name appears in the File field in the top section of the Disk Operations menu. Loading symbol or inverse assembler replaces those that are linked to the current configuration.

When a Load operation is executed, a message "Loading file from disk" appears at the top of the display. After the file has been loaded, this message is replaced by "Load operation complete."

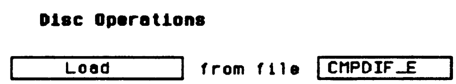


Figure 5-3. Load Operation

Store

The Store operation allows you to store all the setup information, data and inverse assembler links for the analyzer in a configuration file. You cannot store information for only one of the internal analyzers. The information and data present in the logic analyzer at the time the Store is initiated is stored on the disk.

When you select Store from the operations pop-up menu, the top section of the Disk Operations menu looks similar to that shown in figure 5-4. In addition to the operations and file fields, there is a File description field. You can write an optional description of the file you are storing in this field. A file description is not necessary but may help identify a file in the future.

When you name the file that you are storing, you must begin the file name with a letter. The name can contain up to ten characters. It can be any combination of letters and numbers, but it cannot contain any spaces.

Entering a file description is similar to naming a file with three exceptions: you can enter up to 32 characters, start the description with a number, and enter spaces.

When you Execute the Store operation, the message "Storing configuration to disk" appears at the top of the display. After the file has been stored, the message is replaced with "Store operation complete" and the file name appears in the bottom section of the Disk Operations menu with its file type and a description, if you gave it one.

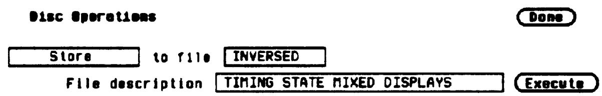


Figure 5-4. Store Operation

Autoload

The Autoload operation allows a specified configuration file to be loaded at power up. When you select Autoload, the top section of the Disk Operations menu looks similar to that shown below. A field appears next to the operation field. When you select this field, a pop-up menu appears with the choices Enable and Disable . Enable causes the specified file to be automatically loaded at power up. Disable prevents any file from being loaded at power up.

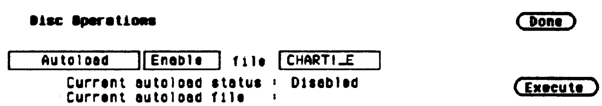


Figure 5-5. Autoload Operation

The file name in the file field can be changed with one of two methods. One method is to press the up/down ROLL key and rotate the KNOB to scroll through the list of files until the name of the desired file appears in the file field. The other method is to select the file field and use the Alpha Entry pop-up menu and the front-panel keypad to enter the name.

Below the operations and file fields are two information lines. The first line indicates the status of autoload (Enable or Disable), and the second line tells you which file, if any, is enabled for autoload. When you select either Enable or Disable the autoload status of a file will not change until you select Execute.

When you select Execute, after selecting Enable, the file whose name appears in the file field is selected for autoloading. The autoload status line will say Enable, and the autoload file line will state the name of the file.

Also, a file labeled AUTOLOAD is added to the bottom section of the display. This file is not a configuration file. It contains information the logic analyzer needs to load the chosen file at power up. If you disable autoloading, the file labeled AUTOLOAD does not disappear. You must Purge it to erase it from your disk. The Purge disk operation is covered later in this chapter. If Autoload is disabled, the logic analyzer will load the default configuration at power up.

Copy

The Copy operation allows you to copy a file to the same disk or to another disk. When you select Copy, the top section of the Disk Operations menu will look similar to that below.

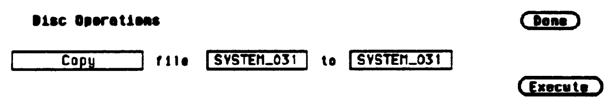


Figure 5-6. Copy Operation

Notice that there are two file fields. You can specify the file you are copying from and the file you are copying to. When you select either file field, you will get an Alpha Entry pop-up menu. You can use this menu and the keypad on the front panel to enter the name of the file. For the file that you are copying from, it is usually easier to use the up/down ROLL key and the KNOB to select one of the files on the disk rather than to use the Alpha Entry menu.

When you select Execute you will see a pop-up that tells you to insert the disk onto which you want to copy the file. There are also two fields in the pop-up. One is labeled **Continue**. You select Continue after you have inserted the disk and are ready to copy the file. The other field is labeled **Stop**. Selecting the Stop field halts the copy and returns you to the Disk Operations menu.

If you insert the destination disk and select **Continue**, the file will be copied. If the file is long, you might have to swap the source and destination disks again. The logic analyzer tells you if you need to reinsert the source disk to continue copying the file. You can also copy to the same disk, making the source and destination disk the same.

Duplicate Disk

The Duplicate Disk operation allows you to duplicate all the files on one disk to another. When you select this option, only the operations field appears in the top section of the Disk Operations menu. The disk is automatically formatted in this operation.

Figure 5-7. Duplicate Disk Operation

When you select Execute, you will see a pop-up with a message telling you what occurs when a disk is duplicated. The pop-up also contains two fields: Cancel and Continue. Cancel stops the duplicating process and returns you to the Disk Operations menu. Continue executes the operation. If you select **Continue**, the display goes blank except for the message "Insert source disk - hit select when ready." Insert the disk you want to duplicate and press SELECT. After the logic analyzer reads the disk, it displays the message "Insert destination disk - hit select when ready." Insert the disk to which you want to copy and press SELECT. The analyzer will tell you that it's writing to the disk.

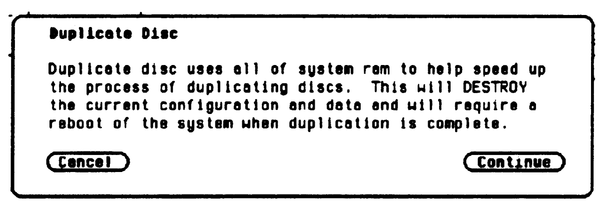


Figure 5-8. Duplicate Disk Pop-up Menu

The process of duplicating a disk is an iterative one; i.e., more than one swapping of disks may be necessary before all files are transferred. If this is the case the logic analyzer will repeat the message telling you to insert the source disk. Insert the source disk and press SELECT. The analyzer remembers where it stopped duplicating the first time and starts reading from that location. When the analyzer is ready, insert the destination disk and press SELECT. You will never have to swap disks more than three times.

After the duplication process is complete, the logic analyzer displays a message telling you what to do next. If you want to copy another disk, press the FORMAT key on the front panel. The analyzer will repeat its message to insert the source disk. If you do not want to copy any more disks, insert the system disk and press the SELECT key. This reboots the system.

Note

Duplicating a disk destroys any existing configurations and data on the destination disk. Make sure that the disk to which you are duplicating is the correct disk.

Pack Disk

The Pack Disk operation reorganizes the files on the disk, making room for more. When a file is purged, it is not removed from the disk even though it doesn't appear in the Disk Operations menu. Packing a disk moves files up, creating space at the bottom of the disk memory.

When you select **Pack Disk**, the top section of the Disk Operations menu looks similar to that shown below. Selecting Execute starts the process. After the packing is completed, the message "Disk packing complete" appears at the top of the screen.

Figure 5-9. Pack Disk Operation

Rename

The Rename operation lets you rename a file. When you select this option, the display will look similar to that shown in figure 5-10.

You will see a file field that tells you what the old name of the file is, and a file field that tells you what the new name will be. If you select either one of the file fields, an Alpha Entry pop-up menu appears. You can use this menu and the keypad on the front panel to enter the name of the file. For the field with the old file name, it is usually easier to use the up/down ROLL key and the KNOB to select the desired file rather than to use the Alpha Entry pop-up menu.

To start the rename operation, select Execute. The file will be renamed and relocated alphabetically in the file list in the bottom section of the Disk Operations menu.

If you try to rename a file with a name that already exists, a message will tell you that a file already exists with that name, and the file will not be renamed.

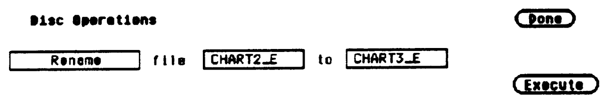


Figure 5-10. Rename Operation

Purge

The Purge operation allows you to delete a file from a disk. When you select this option, the display will look similar to that shown below.

The file field contains the name of the file to be purged. You can change the file in this field either by positioning the cursor on the field and selecting it to access an Alpha Entry pop-up menu, or by using the up/down ROLL key and the KNOB to move among the files.

When you select Execute you will see a pop-up with the choices **Cancel** and **Continue.** Cancel lets you stop the Purge operation and returns you to the Disk Operations menu. Continue purges the file whose name appears in the file field.

Note A purged file cannot be recovered. Make sure the file that is being purged is the correct one.

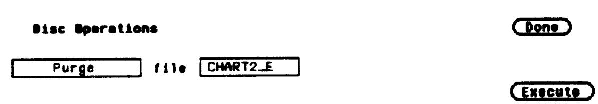


Figure 5-11. Purge Operation

Format Disk

The Format Disk operation formats a disk, purging all previous files on the disk. When you select this option, the display will look similar to that shown in figure 5-12.

Selecting **Execute** gives you a pop-up with the choices **Cancel** and **Continue**. **Cancel** stops the format operation and returns you to the Disk Operation menu. If you select **Continue**, the disk will be formatted. The message "Disk format in progress" will appear at the top of the screen. When the formatting is complete, all the files will be deleted.

Note

Formatting a disk purges all the files on the disk. Make sure the disk is the correct one to be formatted because purged files cannot be recovered.

Figure 5-12. Format Disk Operation

I/O Port Configuration

The I/O Port Configuration option in the I/O menu enables you to configure the logic analyzer for sending configuration, waveforms and listings to a printer or controller via HP-IB or RS-232C.

When you place the cursor on the External I/O Configuration option and press SELECT, you will see the menu shown in figure 5-13.

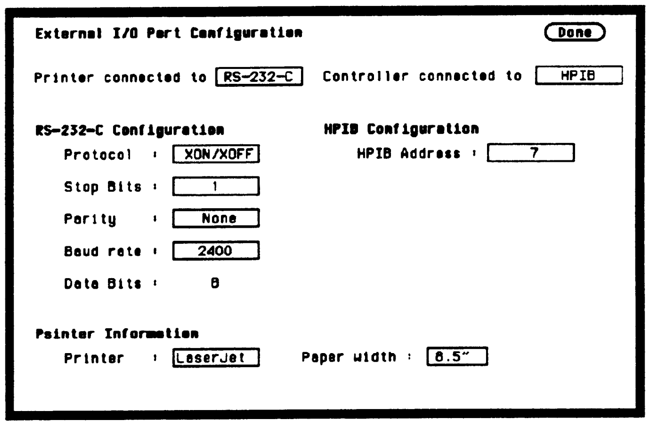


Figure 5-13. External I/O Port Configuration Menu

The HP 1652B/53B is equipped with a standard RS-232C interface and an HP-IB interface that allows you to connect to a printer or controller. Connecting a controller gives you remote access for running measurements, up-loading and down-loading configurations and data, and outputting to a printer. The controller interface is explained in more detail in the *HP 1652B/1653B Programming Reference Manual.*

Various HP-IB and RS-232C graphics printers can be connected to the logic analyzer. Configured menus as well as waveforms and other data can be printed for complete measurement documentation. The printer interface is explained in more detail in Chapter 7.

Configuring the Interfaces

You configure the HP-IB or RS-232C interfaces for a controller or a printer by first selecting the I/O menu. Then you select the **I/O Port Configuration** field to display the External I/O Port Configuration menu. When the menu appears, select either field at the top of the menu to switch the interfaces between a printer and a controller. Whenever you change the configuration for one interface, the other interface automatically changes to the opposite configuration.

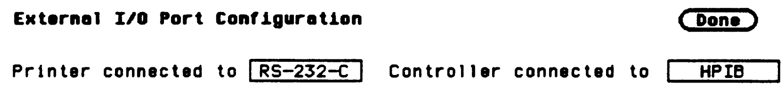


Figure 5-14. Interface Configurations

The HP-IB printer must be set to **Listen Always** for the HP-IB interface. In this mode, no HP-IB addressing is necessary. There are two fields at the bottom of the menu that allow you to select the printer type and paper width.

The HP-IB Interface

The Hewlett-Packard Interface Bus (HP-IB) is Hewlett-Packard's implementation of IEEE Standard 488-1978, "Standard Digital Interface for Programmable Instrumentation." The HP-IB is a carefully defined interface that simplifies the integration of various instruments and computers into systems. It uses an addressing technique to ensure that each device on the bus (interconnected by HP-IB cables) receives only the data intended for it. To accomplish this, each device is set to a different address and this address is used to communicate with other devices on the bus.

Selecting an Address. The HP-IB address can be set to 32 different HP-IB addresses, from 0 to 31. Simply choose an address that is compatible with your device and/or software. The default is 7.

To select an address:

1. Select the External I/O Port Configuration menu and place the cursor in the field directly to the right of **HP-IB Address:**. Press SELECT and an Integer Entry pop-up appears. See figure 5-15.

Integer Entry Pop-up

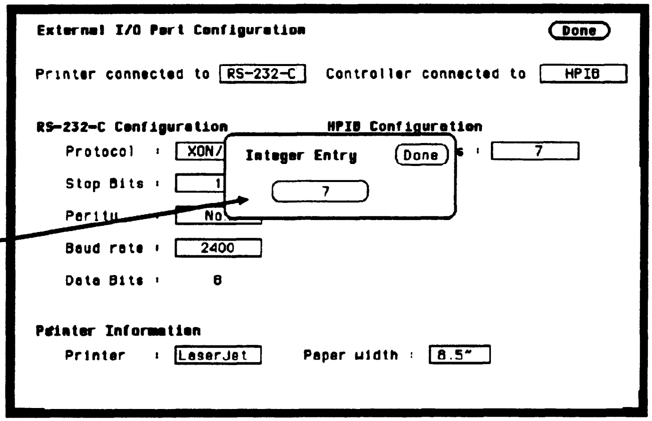


Figure 5-15. Integer Entry Pop-up

2. When the pop-up appears, either rotate the knob or use the keypad to enter the address. If you enter an address greater than 31, the address will default to 31 when you select **Done.**

3. When you are finished entering the HP-IB address, select **Done.** The pop-up closes, placing your selection in the appropriate field.

The RS-232C Interface

The RS-232C interface is Hewlett-Packard's implementation of EIA Recommended Standard RS-232C, "Interface Between Data Terminal Equipment and Data Communications Equipment Employing Serial Binary Data Interchange." With this interface, data is sent one bit at a time and characters are not synchronized with preceding or subsequent data characters. Each character is sent as a complete entity without relationship to other events.

Protocol. Protocol governs the flow of data between the instrument and the external device. The protocol options are **None** and **XON/XOFF.** The default setting is XON/XOFF.

None
XON/XOFF

Figure 5-16. Protocol Pop-up Menu

With less than a 5-wire interface, selecting **None** does not allow the sending or receiving device to control how fast the data is being sent. No control over the data flow increases the possibility of missing data or transferring incomplete data. With a full 5-wire interface, selecting **None** allows a hardware handshake to occur. With a hardware handshake, hardware signals control data flow. The HP 13242G cable allows the HP 1652B/1653BA to support hardware handshake.

With **XON/XOFF**, the receiver controls the data flow. By sending XOFF (ASCII decemal 19) over it's transmit data line, the reciever requests that the sender disables data transmission. A subsequent XON (ASCII decimal 17) allows the sending device to resume data transmission

Data Bits. Data bits are the number of bits sent and received per character that represent the binary code of that character. The HP 1652B/53B supports 8-bit only.

Stop Bits. Stop bits are used to identify the end of the character. The number of stop bits must be the same for both the controller and the logic analyzer. The options are **1**, **1.5**, or **2** stop bits per character. The default setting is 1.

1
1 1/2
2

Figure 5-17. Stop Bits Pop-up Menu

Parity. The parity bit detects errors as incoming characters are received. If the parity bit does not match the expected value, the character is assumed to be incorrectly received. The action taken when an error is detected depends on how the interface and the device program are configured.

Parity is determined by the requirements of the system. The parity bit may be included or omitted from each character by enabling or disabling the parity function. The options are **None, Odd,** or **Even.** The default setting is None.

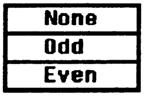


Figure 5-18. Parity Pop-up Menu

Baud Rate. The baud rate is the rate at which bits are transferred between the interface and the peripheral. The baud rate must be set to transmit and receive at the same rate as the peripheral, or data cannot be successfully transferred. The available baud rates are 110 to 19.2k. The default setting is 9600.

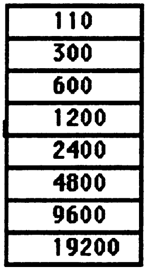


Figure 5-19. Baud Rate Pop-up Menu

Printer. You can specify which printer you are using by selecting the **Printer** attribute field and choosing one of the options in the pop-up. The options are **ThinkJet, QuietJet, LaserJet,** and **Alternate.** Alternate allows you to use an Epson® compatible printer. The default printer option is **ThinkJet.**

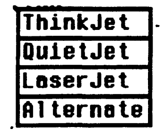


Figure 5-20. Printer Pop-up Menu

Paper Width. The logic analyzer offers two options for paper width: **8.5** and **13.5** inches. Selecting the **Paper Width** attribute field gives you a pop-up with which you can make your choice.

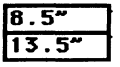


Figure 5-21. Paper Width Pop-up Menu

The HP ThinkJet and HP LaserJet series printers require a paper width of 8.5 inches and the HP QuietJet series printers require a paper width of 13.5 inches. If you have an HP ThinkJet or HP LaserJet printer but have set the paper width to 13.5 inches, the logic analyzer tells the printer to compress the print so it will fit on a page. The results may not be satisfactory. If you have an Epson® compatible printer, check your printer manual to see which size is required.

External BNC Configuration

On the rear panel of the logic analyzer are two BNC connectors. One BNC is an input for an external trigger source. The other is used to output a trigger source. The **External BNC Configuration** option in the I/O menu identifies one of the two internal machines or scope to be the trigger source for an external instrument.

When you select this option you will see a field next to the words "BNC output armed by." Selecting this field gives you a pop-up with either two or three options. One option is **Off**. This indicates that the logic analyzer will not trigger an external instrument. The other options are the internal analyzers, listed by name. You can select the analyzer for triggering your external instrument by using the KNOB to position the cursor on the appropriate name and pressing SELECT.

If for some reason both of the internal analyzers are off, selecting the **External BNC Configuration** option gives you the message "BNC output armed by : Off (note: both machines are off)."

Self Test

The **Self Test** option in the I/O menu allows you to run a self test on the logic analyzer. The self test is on the PV disk. Selecting this option gives you a pop-up telling you what effect the self test has on the analyzer. The pop-up also contains two fields: **Cancel** and **Start Self Test. Cancel** lets you change your mind about running the self test. Selecting this field returns you to the I/O menu. Selecting the **Start Self Test** field causes your logic analyzer to load the self test from the disk and run through it. Before selecting this field you must insert the master disk with the self test on it.

Note

Running the self test destroys all current configurations and data. Make sure that you save any important configurations on a disk before running any of the self tests.

For a description of the individual self tests, refer to appendix E, in volume 2 of this manual.

6

Disk Drive Operations

Introduction

This chapter describes the disk operations of the HP 1652B/53B in a task format. The disk operations are described in detail in chapter 5.

The Disk Operations Available

Nine disk operations are available:

- Load - Instrument configurations and data can be loaded from the disk. Inverse assemblers can be loaded.
- Store - Instrument configurations and data can be stored on disk. System files cannot be stored.
- Autoload - Designates a configuration file to be loaded automatically the next time the HP 1652B/53B is turned on.
- Copy - Any file on the disk can be copied from one disk to another or to the same disk.
- Duplicate Disk - All files from one disk are copied to another disk. The directory and all files on the destination disk will be destroyed with this operation. The copied files are packed on the new disk as they are copied.
- Pack Disk - This function packs files on a disk. Packing removes all empty or unused sectors between files on a disk so that more space is available for files at the end of the disk.
- Rename - Any filename on a disk can be changed to another name.
- Purge - Any file on a disk can be purged (deleted) from the disk.
- Format Disk - Any two-sided 3.5-inch floppy disk can be formatted or initialized. The directory and all files on the disk will be destroyed with this operation.

Although default values are provided for these disk operations, you may have to specify additional information. This information is entered by selecting the appropriate fields displayed for each disk operation. Disk operations are initiated by selecting the Execute field. If there is a problem or additional information is needed to execute an operation, an advisory appears near the top center of the screen displaying the status of the operation (an error message prompts to swap disks, etc.).

If executing a disk operation could destroy or damage a file, another pop-up appears with the options Cancel and Continue when you select Execute. If you don't want to complete the operation, select Cancel to cancel the operation. Otherwise, select Continue and the operation will be executed.

Accessing the Disk Menu

To display the Disk Operations menu, press the I/O menu key.

When the I/O pop-up menu appears, place the cursor on Disk Operations and press SELECT. You will see the Disk Operations menu.

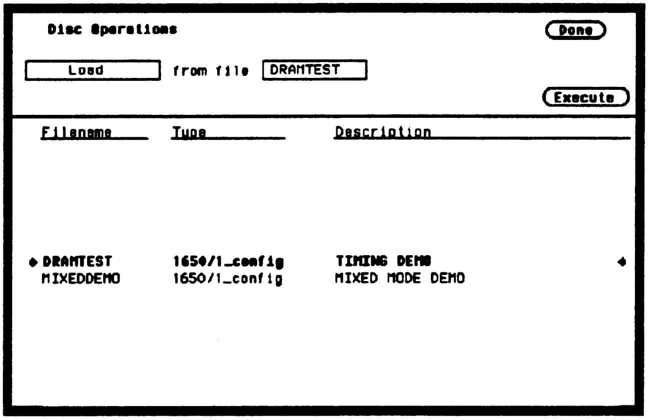


Figure 6-1. Disk Operation Menu

Selecting a Disk Operation

To select a disk operation, place the cursor on the field directly below Disk Operations and press SELECT. You will see the following pop-up:

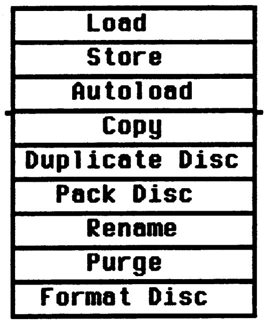


Figure 6 - 2. Disk Operations Pop-up Menu

When the pop-up appears, place the cursor on the operation you want and press SELECT. After you select an option, the pop-up closes and displays the fields required for your operation. For example, select Store. The Disk Operations menu now looks like this:

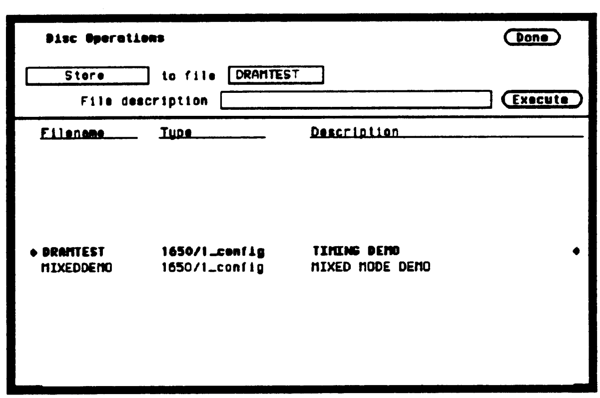


Figure 6-3. Store Operation

Disk Operation Parameters

The disk operation parameters consist of the information that the disk operation acts upon. They tell the logic analyzer the names, types, and descriptions of files. To change these parameters, select the appropriate field and the field will either toggle to the opposite function or a pop-up will appear. If a pop-up appears, select the appropriate option or enter data with the keypad.

To initiate the disk operation function you have selected, place the cursor on Execute. A pop-up appears with Continue and Cancel. To continue, place the cursor on Continue and press SELECT. To cancel place the cursor on Cancel and press SELECT. The Autoload, Pack Disk, and Rename functions immediately execute because they are not destructive to the files. These functions do not give you the Cancel and Continue options.

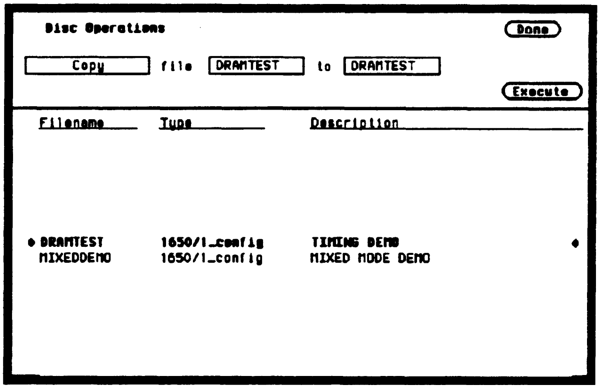


Figure 6-4. Disk Operation Parameters

Installing a Blank Disk

Included with the HP 1652B/53B is a blank 3.5-inch flexible disk for your own use. To install the blank disk, hold the disk so that the Hewlett-Packard label is on top and the metal auto-shutter is away from you. Push the disk gently, but firmly, into the front disk drive until it clicks into place.

Note

The HP 1652B/53B disk drives use the gray Hewlett-Packard double-sided disks, which can be ordered in a package of ten with the Hewlett-Packard part number 92192A. DO NOT use single-sided disks with the HP 1652B/53B.

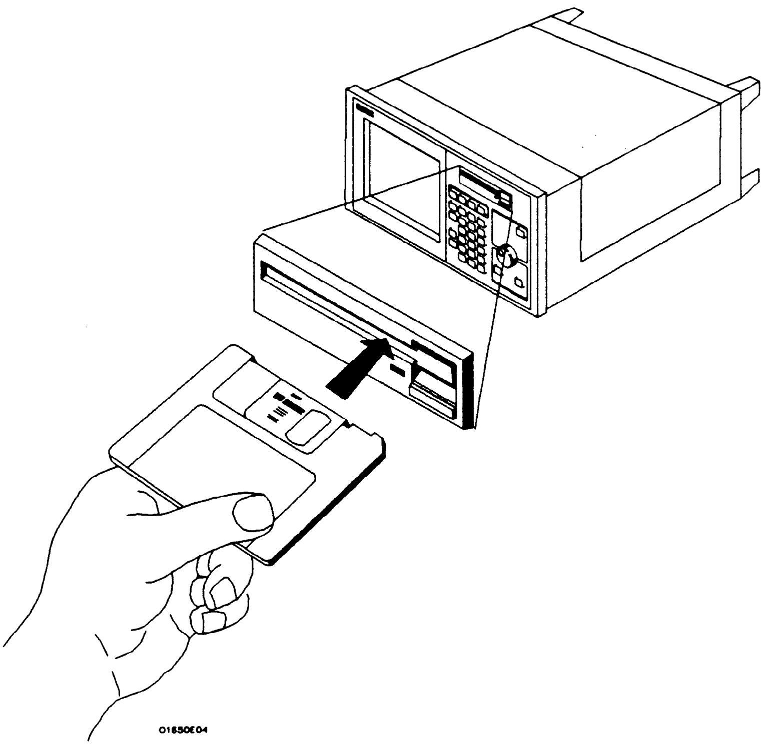


Figure 6-5. Installing a Disk

Formatting a Disk

Before any information can be stored on a new disk, you must first format it. Formatting marks off the sectors of the disk and creates the LIF (Logical Interchange Format) directory on the disk. If you initiate a Duplicate Disk operation, the logic analyzer will automatically format the destination disk.

Note

The HP 1652B/53B does not support track sparing. If a bad track is found, the disk is considered bad. If a disk has been formatted elsewhere with track sparing, the HP 1652B/53B will only read up to the first spared track.

Select the Format Disk operation.

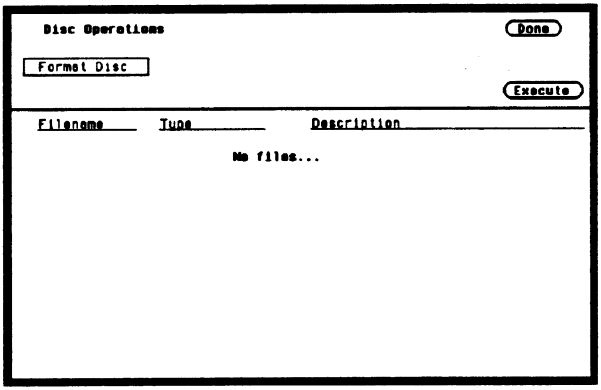


Figure 6-6. Format Disk Operation

After the Format Disk operation menu appears, the instrument reads the disk and shows its condition. One of three conditions can exist:

- If this is a new disk, or a disk formatted by a disk drive not using the LIF format, the menu will display UNSUPPORTED DISK FORMAT on the lower portion of the menu.
- If the disk is already formatted, but has no files, the menu will display No Files.

- If the disk already has files, a list of file names appear on the lower portion of the menu along with a file type and description.

If any of the listed files need to be saved, copy them to another disk before initiating the Format Disk function. To initiate the Format Disk function, select Execute. When the pop-up appears, select Continue and the instrument will format the disk. Otherwise, select Cancel to cancel the Format Disk operation.

Caution

Once you press Continue, the Format Disk operation starts and permanently erases all the existing information from the disk. After that, there is no way to retrieve the original information.

Storing to a Disk

The Store operation allows you to store your configurations and data to a file with a description of its contents. You must assign a file name for each file in which you wish to store data.

Select the Store operation.

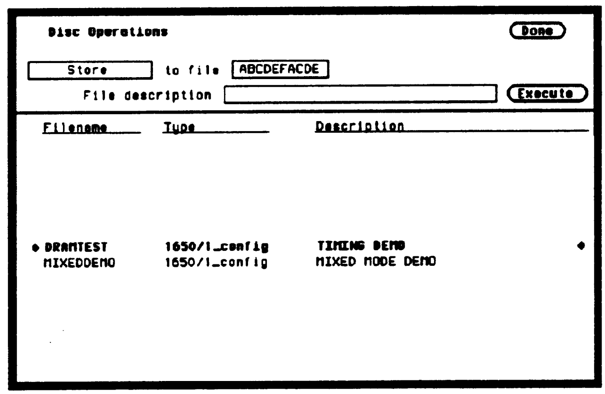


Figure 6-7. The Store Operation

To name your file, place the cursor on the field to the right of "to file" and press SELECT. The Alpha Entry pop-up appears.

Enter a filename that starts with a letter and contains up to ten characters. It can be any combination of letters and numbers, but there can be no blank spaces between any of the characters.

Entering a file description is the same process as naming a file except you can enter up to 32 characters, start the description with a number, and enter spaces between characters.

Note

The field for "file description" makes it easier to identify the type of data in each file. This is for your convenience but you can leave this field blank.

When you have completed entering the file name and file description, you initiate the store operation by placing the cursor on Execute and pressing SELECT. A pop-up appears with Continue and Cancel. To continue, place the cursor on Continue and press SELECT. To cancel, place the cursor on Cancel and press SELECT.

Caution

If you store a new configuration and data to an existing file, they are written over the original information "DESTROYING" the original information in that file.

The Load Operation

The Load operation allows you to load previously stored configuration and data from a file on the disk.

Select the Load operation.

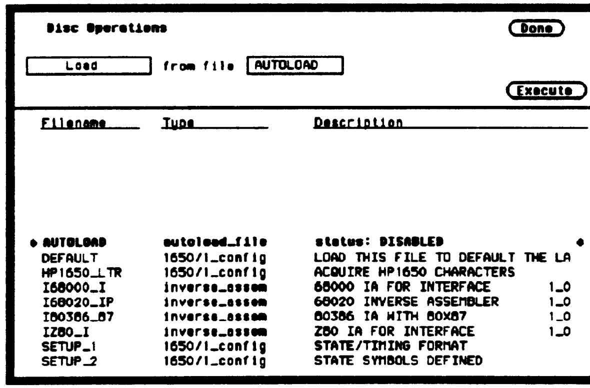


Figure 6-8. The Load Operation

Note

The Load operation is type dependent. This means that you cannot load a system file. For example, if you try to load the file "SYSTEM_," an advisory "Warning: Invalid file type" appears in the top center of the display.

To load the desired file, press the up/down ROLL key and rotate the KNOB until the desired file appears in the field to the right of "from file."

Another way to enter the name of the file in the field to the right of "from file" is to select this field. When the Alpha Entry pop-up appears, enter the correct filename.

Renaming a File

The Rename operation allows you to change the name of a file. The only restriction is that you cannot rename a file to an already existing filename.

Select the Rename operation. When you have completed entering a new file name and description, you initiate the Rename operation by placing the cursor on Execute and pressing SELECT.

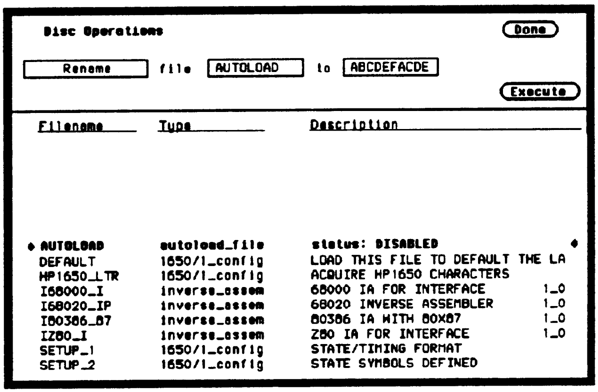


Figure 6-9. Renaming a File

Use either the KNOB or the Alpha Entry pop-up to enter the filename you wish to change in the field to the right of "file."

Move the cursor to the field to the right of "to" and press SELECT. When the Alpha Entry pop-up appears, enter the new file name. When you have completed entering the new file name, you initiate the rename operation by placing the cursor on Execute and pressing SELECT. The rename operation immediately executes and when it is completed, an advisory "Rename operation complete" is displayed.

The Autoload Operation

Autoload allows you to designate a configuration file to be loaded automatically the next time the HP 1652B/53B is turned on. When the Autoload operation is enabled, your designated configuration file is loaded instead of the default configuration file. This process allows you to change the default configuration of certain menus to a configuration that better fits your needs.

Select the Autoload operation. To enable Autoload, select the Disable field and when the pop-up appears, select Enable.

With the up/down ROLL key and KNOB or the Alpha Entry pop-up enter the name of the configuration file you wish to load in the field to the right of "File" and select Execute. The Autoload function is Enabled as shown after "Current Autoload status:" on the display.

Note

When power is applied to the logic analyzer, Autoload On or Off is determined by the presence of an enabled autoload file on the disk. If an enabled autoload file is present on the disk, the logic analyer will load this configuration file instead of the standard configuration file.

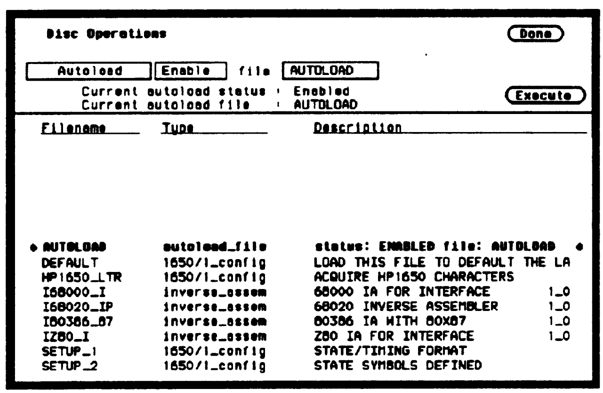


Figure 6-10. Autoload Operation Enabled

To disable the Autoload operation, select enable and when the pop-up appears, select disable. When the pop-up closes, select Execute and the Autoload function is disabled.

Purging a File

Select the Purge operation to Purge (delete) a file. With either the up/down ROLL key and KNOB or the Alpha Entry pop-up enter the file you wish to purge in the field to the right of "file." Select Execute and when the pop-up appears, select Continue and the file is purged from the disk.

Caution

Once EXECUTED, the Purge operation permanently erases the file. After that, there is no way to retrieve the orginal information.

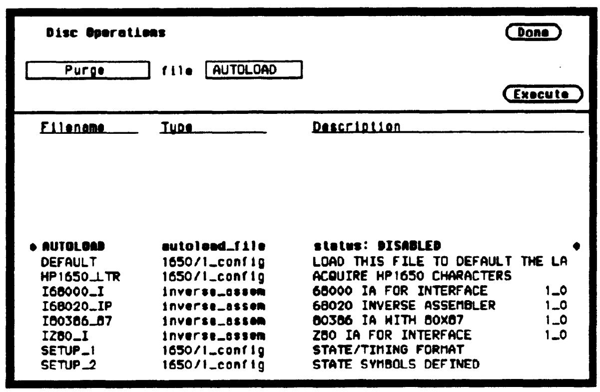


Figure 6-11. Purging a File

Copying a File

The Copy operation allows you to copy a file to the same disk or another disk. Select the Copy operation. With either the up/down ROLL key and the KNOB or the Alpha Entry pop-up, enter the filename you wish to copy in the field to the right of "file." Select the field to the right of "to" and when the Alpha Entry pop-up appears, enter the name of the file you want to "copy to."

You can also copy a file to the same filename on another disk. To do this, select the "To" filename field, press the CLEAR ENTRY key place the cursor on Done and press SELECT. This copies the original filename in the "To" filename field.

Select Execute to start the copy operation. A pop-up appears with instructions on what to do with the disks. Since you can copy a file to the same disk or another disk, simply follow the instructions as they apply to your situation and select Continue to continue.

- When "Insert the source disk" appears, remove the source disk and insert the destination disk into the disk drive if you are copying the file to another disk. The cursor is located on "Continue," so to continue, press SELECT; otherwise, place the cursor on "Stop" and press SELECT. If you are copying to the same disk, press "Continue" without moving the disk.

If the file cannot be copied in a single operation, the instruction "Insert the source disk" will appear in the pop-up. Remove the destination disk, re-insert the source disk and select Continue. The logic analyzer reads another segment of the source file. It will then tell you when to re-insert the destination disk and continue.

Note

If the source file is large (ie. System file) you should use the Duplicate Disk operation. Duplicating large files using the Copy operation requires changing disks many times. This invites the possibility of losing track of the disk changes, which will destroy part or all of the files on the source disk.

When the copy operation is complete, you will see the new file name in the directory. The new file name will be inserted in the directory in alphabetical order.

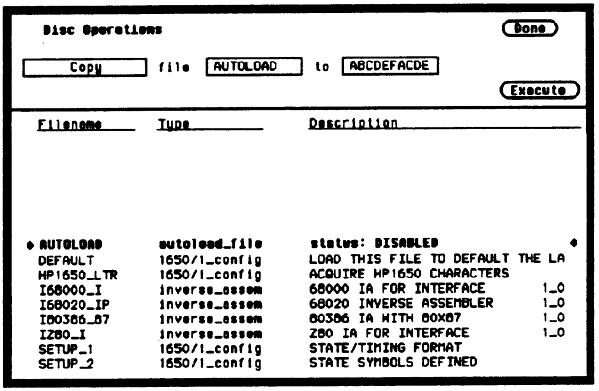


Figure 6-12. Copy File Operation

The Pack Disk Operation

By deleting files from the disk and adding other files, you end up with blank areas on the disk (between files) that are too small for the new files you are creating. The Pack Disk operation packs the current files together, removing unused areas from between the files so that more space is available for files at the end of the disk.

Select the Pack Disk operation. To pack the disk, select Execute.

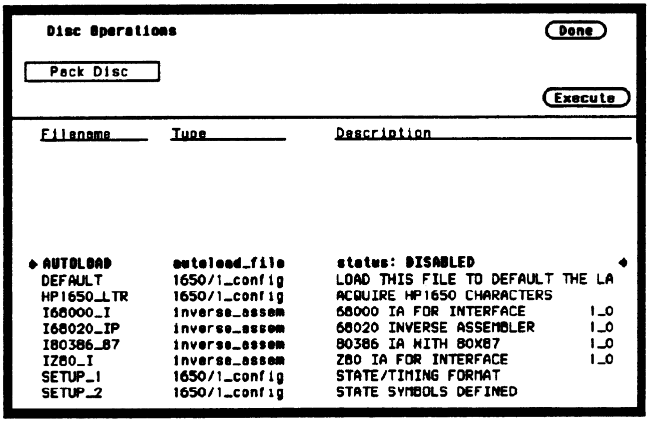


Figure 6-13. The Pack Disk Operation

Duplicating the Operating System Disk

The Duplicate Disk operation allows you to duplicate all the files on one disk to another disk. You use this operation to make a back-up copy of your important disks so you won't lose important data in the event the disk wears out, is damaged, or a file is accidently deleted.

Select the Duplicate Disk operation and press Execute. When the pop-up appears you will see the following advisory:

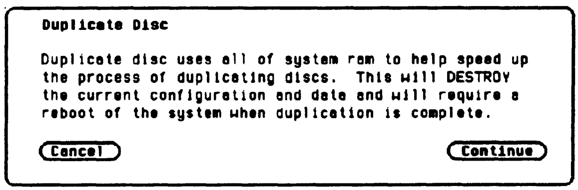


Figure 6-14. Duplicate Disk Pop-up

Caution

The original directory and files on the destination disk are destroyed by the DUPLICATE DISK operation.

To continue, select Continue. The instruction "Insert disk to be copied-hit select when ready" will be displayed. Insert the source disk and press SELECT. The logic analyzer reads the source disk and displays "Reading from source disk. Please wait..."

When the logic analyzer has filled memory or has read the entire source disk, it displays "Insert destination disk-hit select when ready." Remove the source disk, insert the destination disk and press SELECT. When the logic analyzer starts writing to the destination disk, you will see "Writing to destination disk. Please wait..."

Note

If the destination disk has not been formatted, the logic analyzer will automatically format the disk before it writes to it.

If the amount of data on the source disk exceeds the available memory in the logic analyzer, the logic analyzer will display "Insert the source disk-hit select when ready" again, and you will need to repeat the process of inserting the source disk, then the destination disk. Follow the directions on screen until the entire disk is duplicated.

When the entire disk is duplicated, you will see "Hit FORMAT key to copy another disk or insert system disk and hit SELECT to reboot." If you are finished duplicating disks, insert the system disk and press SELECT. The logic analyzer will load the system file and return you to the System Configuration menu.

7

Making Hardcopy Prints

Introduction

The HP 1652B/1653B Logic Analyzers allow you to print configurations, waveforms, and listings. Whenever your printer is connected to the logic analyzer and you instruct it to do so, it will print what is currently displayed on screen or all data in the menus having off-screen data.

This chapter shows you how to set up the logic analyzer's HP-IB and RS-232C interfaces for printers. If you have a Hewlett-Packard ThinkJet, QuietJet, or LaserJet series printer with the RS-232C interface, the RS-232C interface is already set up for you with the exception of the printer type and page width.

If you have another kind of printer, refer to your printer manual for its interface requirements and change the logic analyzer's interface configuration as instructed.

Supported Printers

The HP 1652B/1653B logic analyzers will support the following printers with HP-IB or RS-232C capabilities. For the following RS-232C printers, these configurations should be used:

- HP ThinkJet (RS-232C switches set for HP controllers)
- HP QuietJet (factory settings)
- HP LaserJet (factory settings)
- Alternate

Alternate Printers

In addition to HP printers, the logic analyzers support Epson® compatible RS-232C printers. These alternate printers must support graphics.

When the logic analyzer's RS-232C configuration is set for alternate printers, it transmits data to the printer in the Epson® format.

Printers incompatible with either HP or Epson data transfer formats will not work with the HP 1652B/1653B logic analyzers.

Hooking Up Your Printer

If your printer is already connected to the logic analyzer, skip to "Setting the RS-232C for HP Printers" or "Setting the HP-IB for HP Printers" in this chapter. Otherwise hooking up your HP printer is just a matter of having the correct HP-IB or RS-232C interface cable. Refer to the figure below.

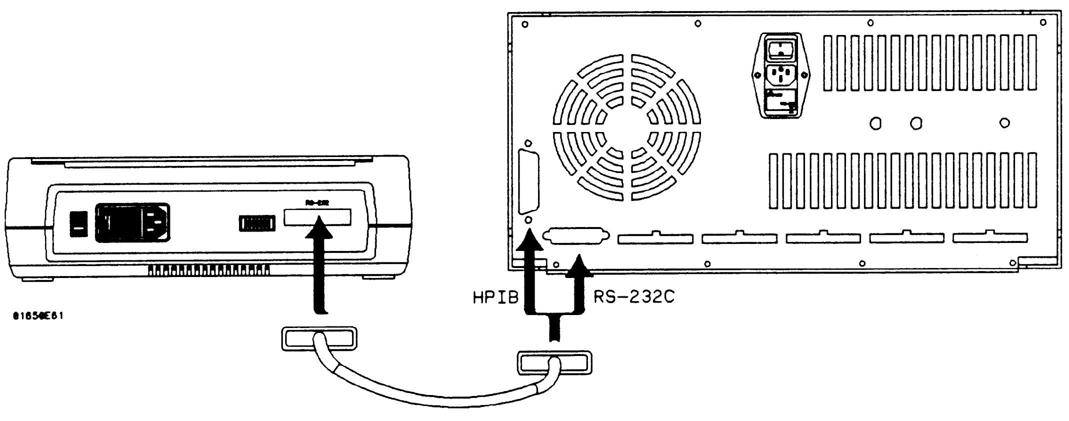


Figure 7-1. Logic Analyzer to Printer Hook-up

The type of connector on the printer end of the interface cable is determined by the kind of printer.

HP-IB Printer Cables

You can use any standard HP-IB cable to connect the logic analyzer to the printer. The specific HP-IB cable only depends on the length you need.

RS-232C Printer Cables

You can use either an HP 13242G or HP 92219H cable to connect the logic analyzer to the printer. However, the HP 13242G is the preferred cable since it can be used with either no protocol (hardware handshake) or XON/XOFF.

HP 13242G Cable

The HP 13242G cable has standard DB-25 connectors on each end and is wired for hardware handshake. The cable schematic is shown below.

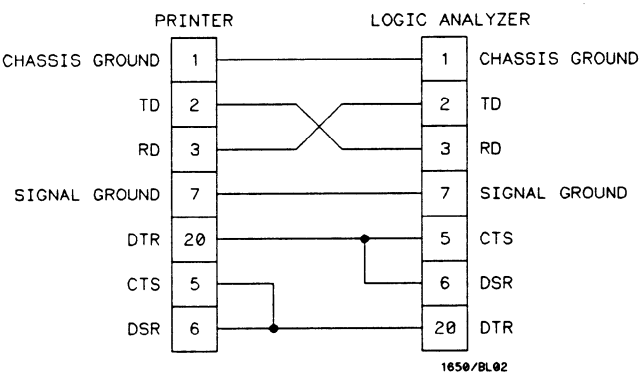


Figure 7-2. HP 1342G Cable Schematic

Note

HP 13242G cable ends are the same, therefore it doesn't matter which end of the cable is connected to which piece of equipment.

HP 92219H Cable

The HP 92219H cable has standard DB-25 connectors on each end and is wired for XON/XOFF handshake. The cable schematic is shown below.

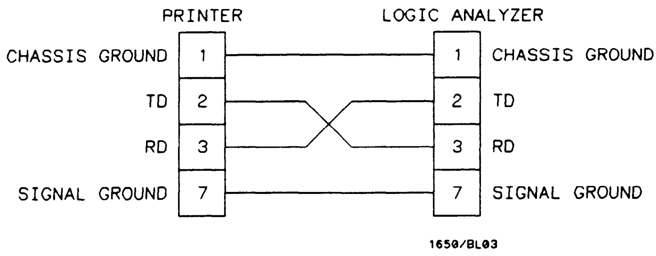


Figure 7-3. HP 92219H Cable Schematic

Setting HP-IB for HP Printers

The HP 1652B/53B interfaces directly with HP PCL printers supporting the printer command language. These printers must also support HP-IB and "Listen Always." Printers currently available from Hewlett-Packard with these features include:

- HP 2225A ThinkJet
- HP 2227B QuietJet
- HP 3630A option 002 PaintJet

Note

The printer must be in "Listen Always" when HP-IB is the printer interface. The HP 1652B/53B HP-IB port does not respond to service requests (SRQ) when controlling a printer. The SRQ enable setting for the HP-IB printer has no effect on the HP 1652B/53B operation.

For HP-IB printers, the Printer connected to field must be set to HP-IB in the I/O Port Configuration menu. You access the I/O Port Configuration menu by first accessing the I/O menu, then the I/O Port Configuration.

Setting RS-232C for HP Printers

All three series of HP printers (HP ThinkJet, HP LaserJet, and HP QuietJet) use the logic analyzer's RS-232C default configuration with only one or two changes depending on which printer you have.

Since the logic analyzer's default RS-232C configuration is set for the HP ThinkJet printer, no changes are needed for the HP ThinkJet.

For RS-232C printers, the Printer connected to field must be set to RS-232C in the I/O Port Configuration menu. You access the I/O Port Configuration menu by first accessing the I/O menu, then the I/O Port Configuration.

Listed below, are the changes you need to make for other HP printers:

- Printer type for the HP LaserJet and HP QuietJet.
- Paper width for the HP QuietJet.

You access the printer type and page width fields by first accessing the I/O menu, then the I/O Port Configuration menu.

Setting RS-232C for Your Non-HP Printer

The following attributes of the RS-232C interface must be set to the correct configuration for your printer:

- Protocol.
- Number of data bits.
- Number of stop bits.
- Parity type.
- Baud rate.
- Paper width.

You access these fields by first accessing the I/O menu then the I/O Port Configuration menu.

Setting Paper Width

Paper width is set by toggling the **Paper width : ____** field in the I/O Port Configuration menu. It tells the printer that you are sending up to 80 or 132 characters per line (only when you Print All) and is totally independent of the printer itself.

- If you select 132 characters per line (13.5 inches) when using other than an HP QuietJet selection, the listings are printed in a compressed mode. Compressed mode uses smaller characters to allow the printer to print more characters in a given width.
- If you select 132 characters per line (13.5 inches) on an HP QuietJet, it will print a full 132 characters per line.
- If you select 80 characters per line for any printer, a maximum of 80 characters are printed per line.

RS-232C Default Configuration

You can use the logic analyzer's default configuration (except for printer type and paper width) for all supported printers if you haven't changed the printer's RS-232C configuration.

The logic analyzer's default configuration is:

Protocol: XON/XOFF
Data Bits: 8
Stop Bits: 1
Parity: none
Baud rate: 9600
Printer: ThinkJet
Paper width: 8.5 inches

Recommended Protocol

The recommended protocol is XON/XOFF. This allows you to use the simpler three-wire hook-ups.

Starting the Printout

When you are ready to print, you need to know whether there is more data than is displayed on screen. In cases where data is off screen (i.e., format specifications with all pods assigned to a single analyzer), you need to decide whether you want just the data that is on screen or all the data.

If you want just what is on screen, start the printout with the Print Screen option. If you want all the data, use the Print All option. Both options are in the I/O menu. Once you decide which option to use, start the printout by placing the cursor on the print option (screen or all) and pressing SELECT.

I/O MENU

- Done
- Print Screen
- Print All
- Disk Operations
- I/O Port Configuration
- External BNC Configuration
- Self tests

Figure 7 - 4. I/O Menu

Print Screen

The Print Screen option prints only what is displayed on screen at the time you initiate the printout. In the Print Screen mode, the printer uses its graphics capabilities and the printout will look just like the logic analyzer screen with only one exception: the cursor will not print.

Print All

The Print All option prints not only what is displayed on screen, but also what is below, and, in the Format Specification, what is to the right of the screen at the time you initiate the printout.

Note

Make sure the first line you wish to print is at the top of the screen when you select Print All. Lines above the screen will not print.

Use this option when you want to print all the data in the following menus:

- Timing Format Specifications.
- State Format Specifications.
- State Trace Specifications.
- State Listing.
- Symbols.
- Disk Directory.

What Happens During a Printout?

When you press SELECT to start the printout, the I/O menu pop-up disappears and an advisory "PRINT in progress" appears in the top center of the display. While the data is transferred to the printer, the only useable key is the STOP key. When the logic analyzer has completed the data transfer to the printer, the advisory "PRINT complete" appears and the keyboard becomes useable again.

The PRINT in progress advisory won't appear in your printout. If you press STOP while the data is being transferred to the printer the transfer stops and the data already sent will print out. This causes an incomplete printout.

Connecting to Other HP Printers

The HP 1652B/53B can also be used with Hewlett-Packard printers that have RS-232C interface options. Simply connect the printer with the HP 13242G cable. Refer to table 7-1 for the appropriate selection for the RS-232C configuration of the HP 1652B/53B.

Table 7-1. HP Printer Selection

For this HP Printer	Select this Printer in I/O Port Configuration menu
HP 2631	QuietJet
HP 2671	ThinkJet
HP 2673	ThinkJet

The above printers should work with the HP 1652B/53B logic analyzers. However, no tests have been made to verify that they will work completely. Therefore, proper operation is neither promised nor supported by Hewlett-Packard.

8

The State Analyzer

Introduction

This chapter introduces the state analyzer and contains the state analyzer menu maps.

- Chapter 9 explains the State Format menu
- Chapter 10 explains the State Trace menu
- Chapter 11 explains the State Listing menu
- Chapter 12 explains the State Compare menu
- Chapter 13 explains the State Chart menu
- Chapter 14 explains the State Waveform menu
- Chapter 15 gives you a basic State Analyzer Measurement example

The State Analyzer (An Overview)

The state analyzer acquires data synchronously using the system-under-test to clock the acquired data. The acquired data is displayed in a list form in the State Listing menu and in waveform form in the State Waveform menu. The state analyzer differs from the timing analyzer in that the acquisition clock is provided by the system-under-test instead of the internal acquisition clock used by the timing analyzer. Therefore, the State Waveform menu displays the state waveforms referenced by states per division and not seconds per division as in the timing analyzer.

State Analyzer Menu Maps

The State Analyzer menu maps show you the fields and the available options of each field within the six menus. The menu maps will help you get an overview of each menu as well as provide you with a quick reference of what each menu contains.

State Format Menu Map

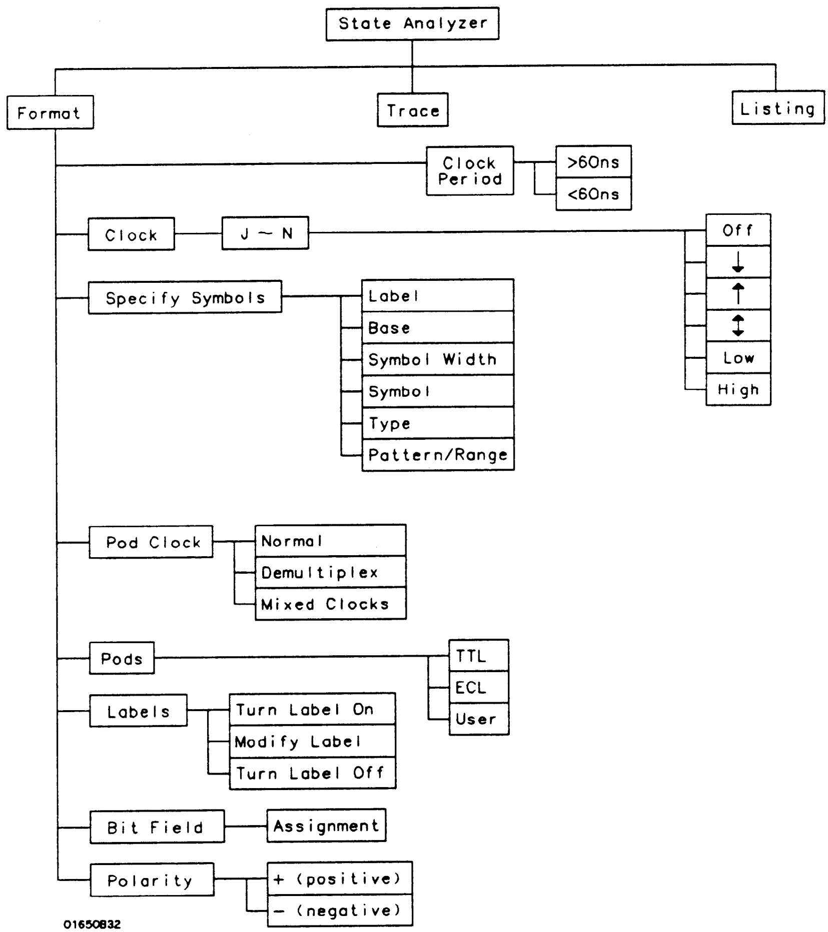


Figure 8-1. State Format Menu Map

State Trace Menu Map

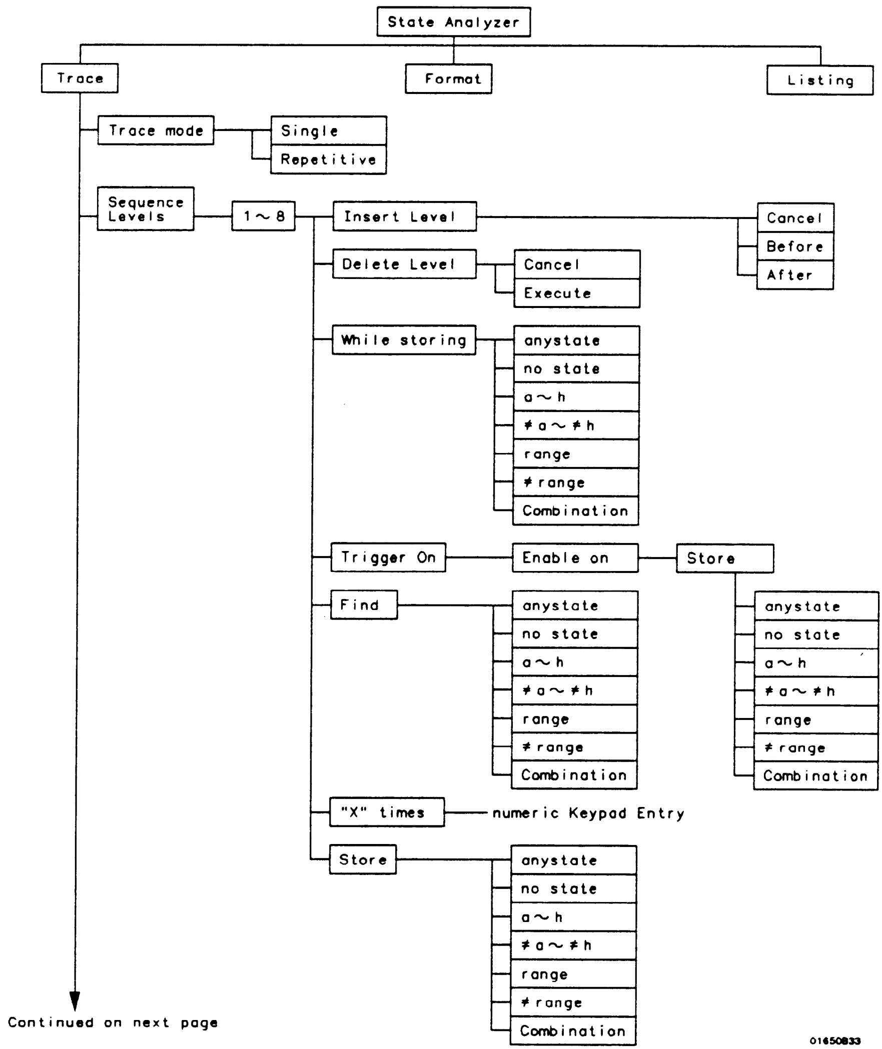


Figure 8-2. State Trace Menu Map

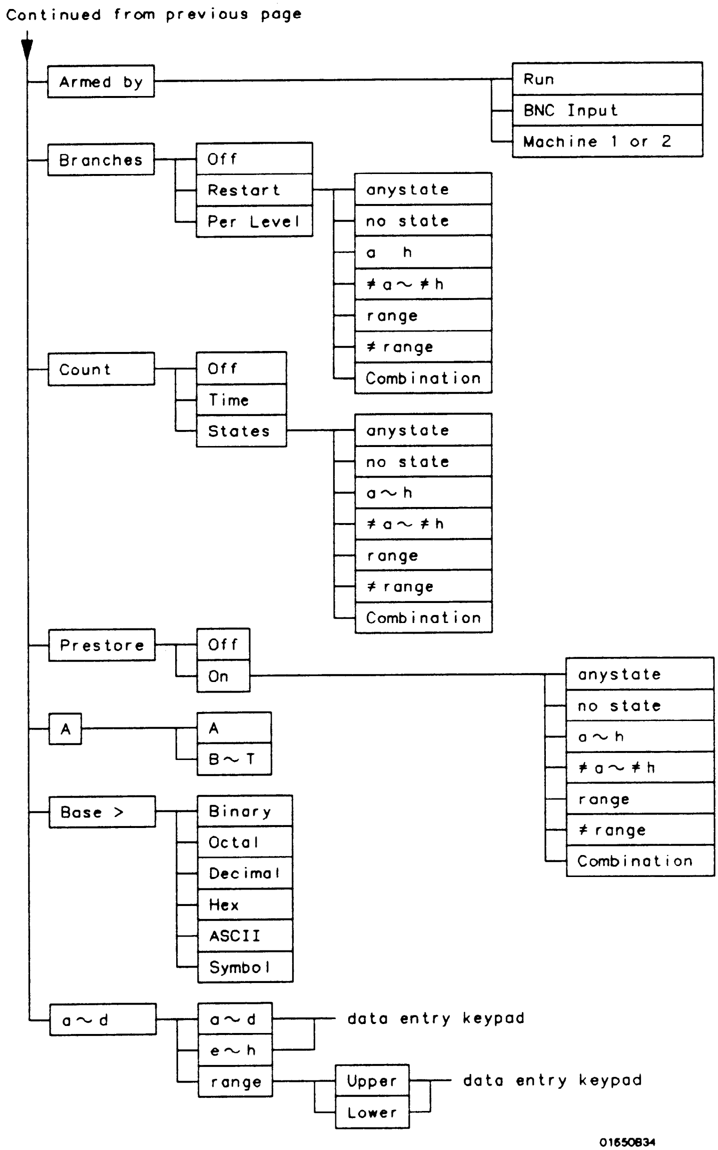
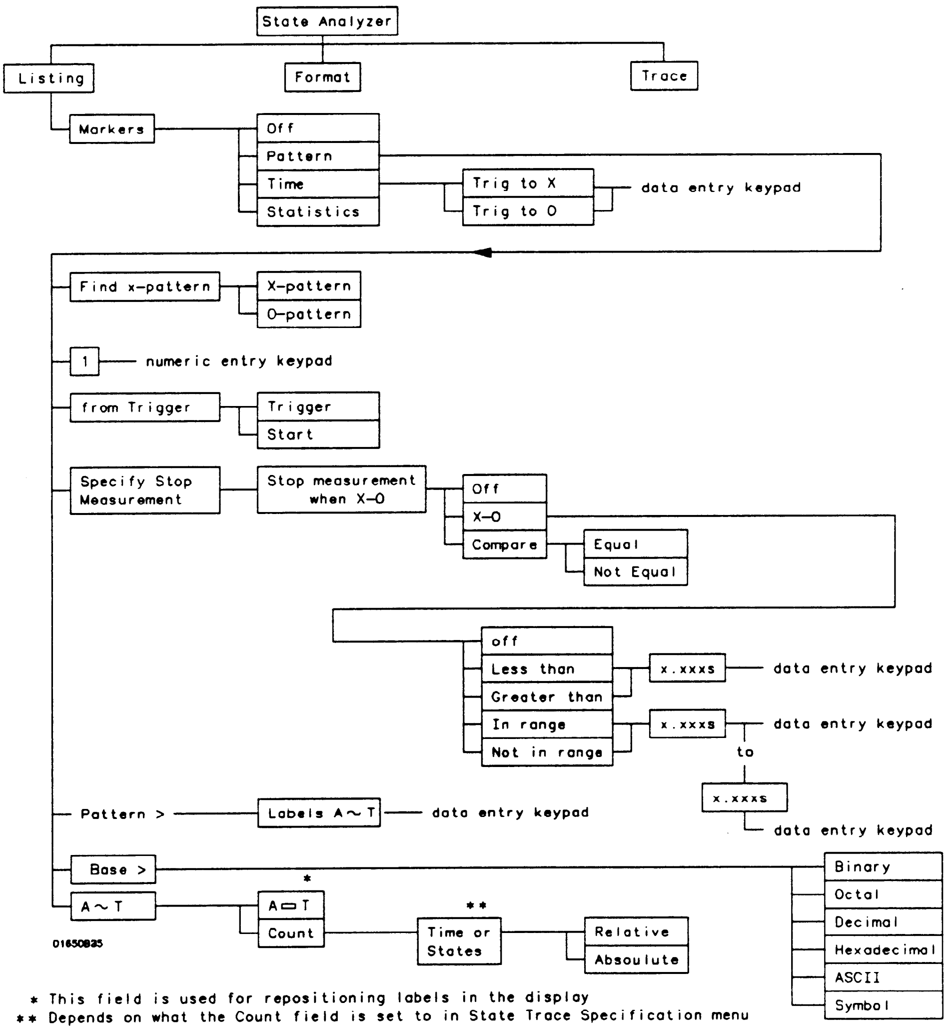


Figure 8-2. State Trace Menu Map (continued)

State Listing Menu Map



* This field is used for repositioning labels in the display

** Depends on what the Count field is set to in State Trace Specification menu

Figure 8-3. State Listing Menu Map

State Compare Menu Map

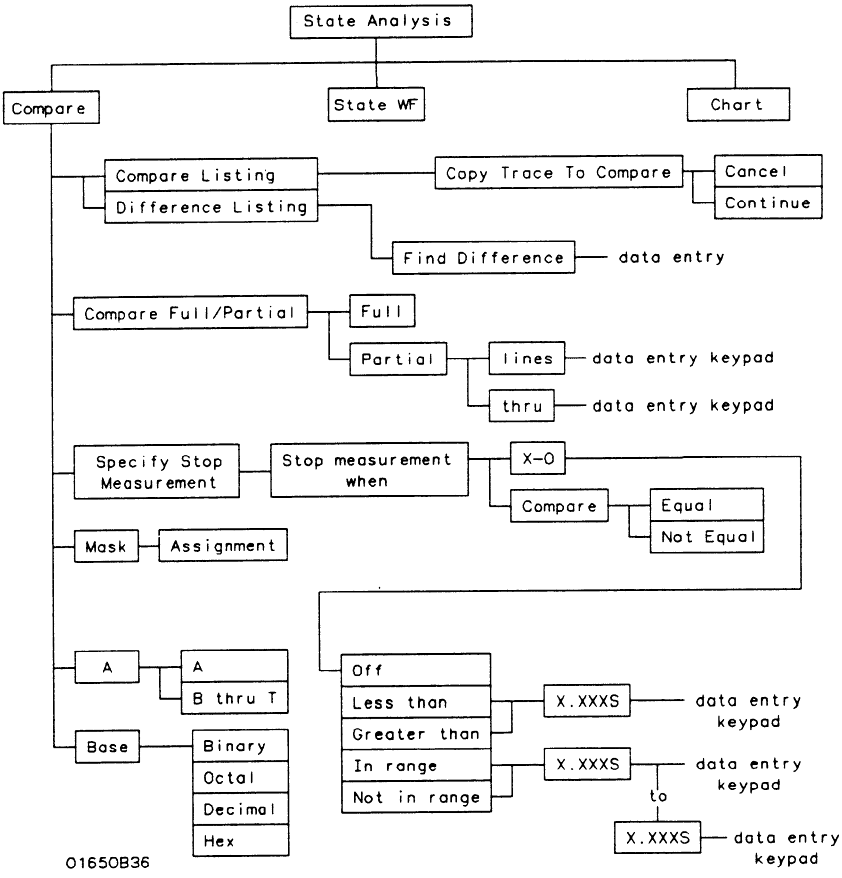


Figure 8-4. State Compare Menu Map

State Waveform Menu Map

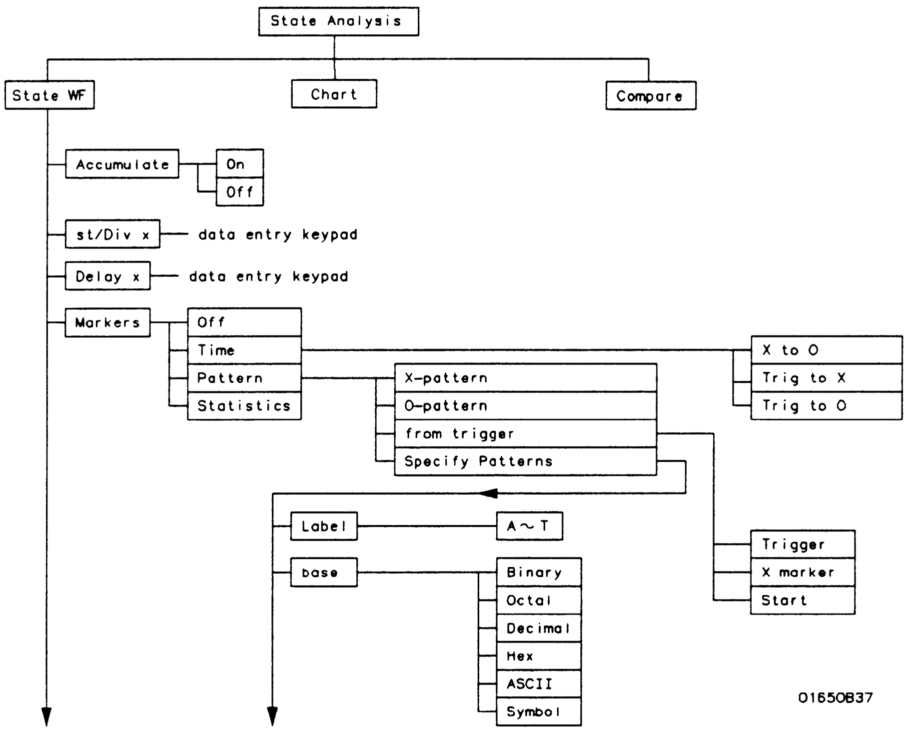


Figure 8-5. State Waveform Menu Map

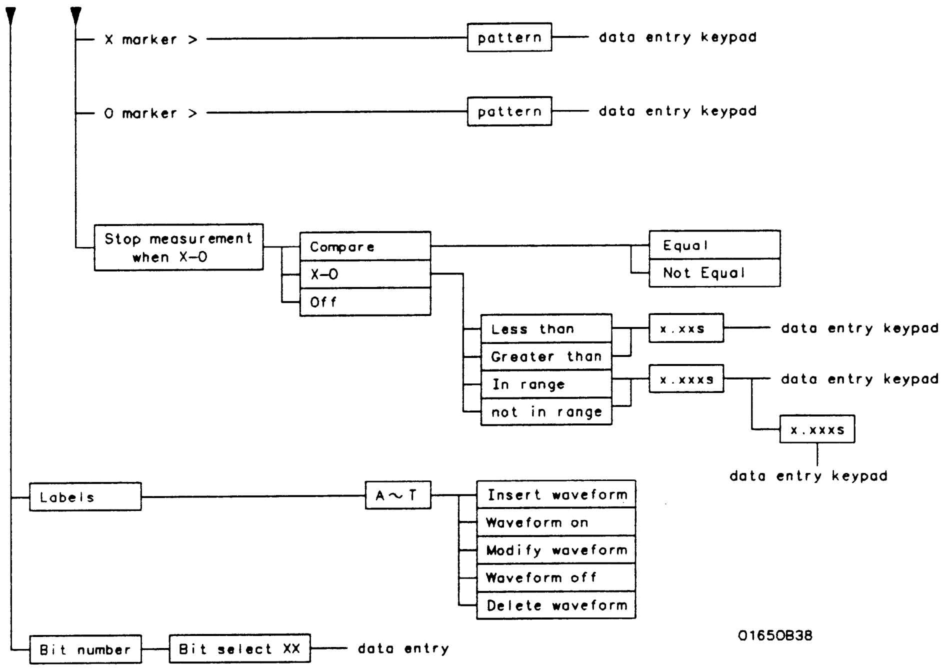


Figure 8-5. State Waveform Menu Map (continued)

State Chart Menu Map

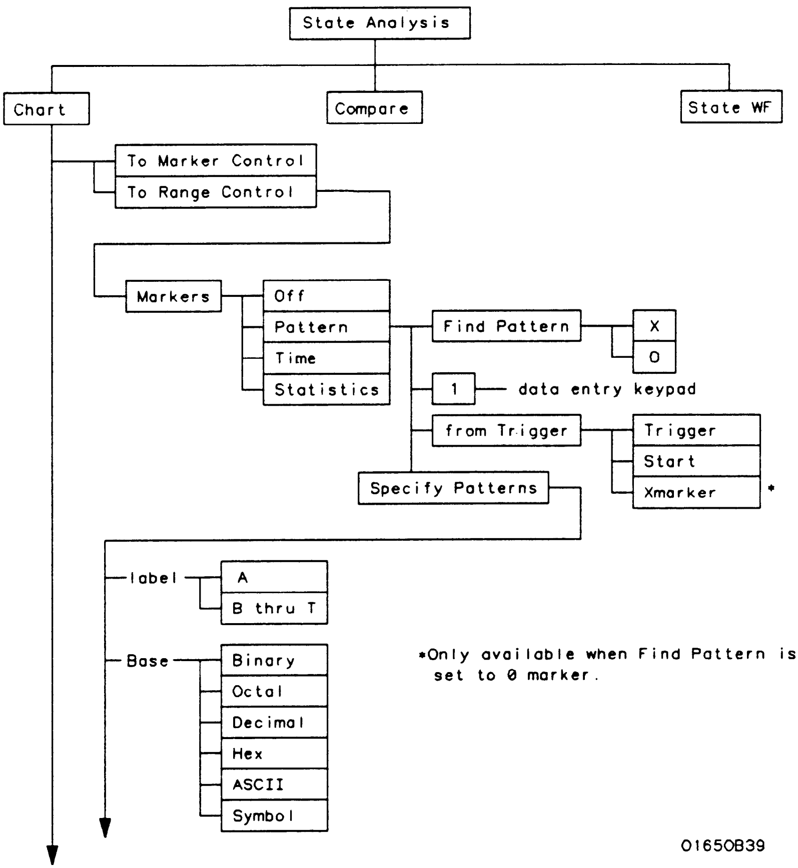


Figure 8-6. State Chart Menu Map

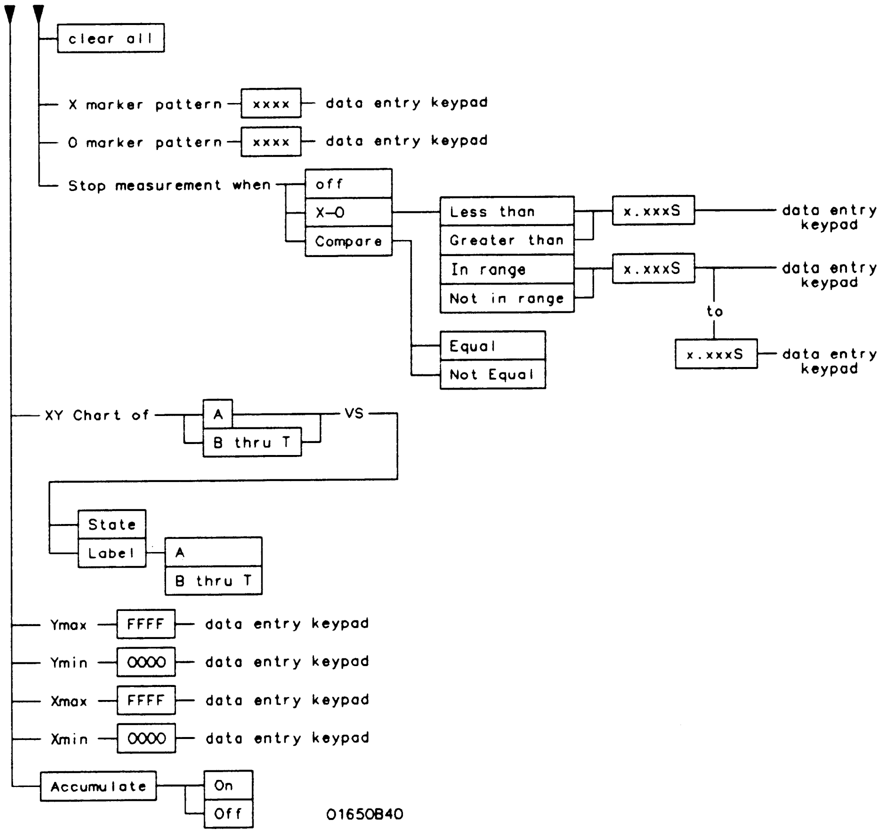


Figure 8-6. State Chart Menu Map (continued)

9

State Format Specification Menu

Introduction

This chapter describes the State Format Specification menu and all pop-up menus that you will use on your state analyzer. The purpose and functions of each menu are explained in detail, and we have included many illustrations and examples to make the explanations clearer.

Accessing the State Format Specification Menu

The State Format Specification menu can be accessed by pressing the FORMAT key on the front panel. If the Timing Format Specification Menu is displayed when you press the FORMAT key, you will have to switch analyzers. This is not a problem, it merely indicates that the last action you performed in the System Configuration Menu was on the timing analyzer.

State Format Specification Menu

The State Format Specification menu lets you configure the logic analyzer to group channels from your microprocessor into labels you assign for your measurements. You can set the threshold levels of the pods assigned to the state analyzer, assign labels and channels, specify symbols, and set clocks for triggering.

At power up, the logic analyzer is configured with a default setting. You can use this default setting to make a test measurement on the system under test. It can give you an idea of where to start your measurement. For an example of setting up configurations for the state analyzer, refer to your *Getting Started Guide* or "State Analyzer Measurement Example" in Chapter 15 of this manual.

At power up the State Format Specification menu looks like that shown below:

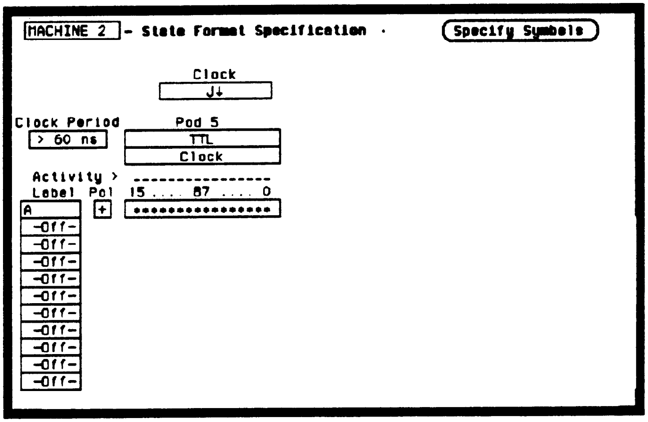


Figure 9-1. State Format Specification Menu

The State Format Specification menu for the HP 1653B is similar to that for the HP 1652B except that Pod 2 appears in the menu instead of Pod 5.

This menu shows only one pod assigned to each analyzer, which is the case at power up. Any number of pods can be assigned to one analyzer, from none to all five for the HP 1652B, and from none to two for the HP 1653B. In the State Format Specification menu, only three pods appear at a time in the display. To view any pods that are off screen, press the left/right ROLL key and rotate the KNOB. The pods are always positioned so that the lowest numbered pod is on the right and the highest numbered pod is on the left.

State Format Specification Menu Fields

Seven types of fields are present in the menus:

- Label
- Polarity (Pol)
- Bit assignments
- Pod threshold
- Specify Symbols
- Clock
- Pod Clock
- Clock Period

A portion of the menu that is not a field is the Activity Indicators display. The indicators appear under the active bits of each pod, next to "Activity > ." When the logic analyzer is connected to your target system and the system is running, you will see ↕ in the Activity Indicators display for each channel that has activity. These tell you that the signals on the channels are transitioning.

The fields in the Format menus are described in the following sections.

Label

The label column contains 20 Label fields that you can define. Of the 20 labels, the state analyzer displays only 11 labels at one time. To view the labels that are off screen, press the up/down ROLL key and rotate the KNOB. The labels scroll up and down. To deactivate the scrolling, press the ROLL key again.

To access one of the Label fields, place the cursor on the field and press SELECT. You will see a pop-up menu like that shown below.

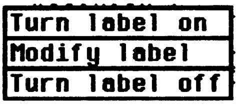


Figure 9-2. Label Pop-Up Menu

Turn Label On

Selecting this option turns the label on and gives it a default letter name. If you turned all the labels on they would be named A through T from top to bottom. When a label is turned on, bit assignment fields for the label appear to the right of the label under the pods.

Modify Label

If you want to change the name of a label, or want to turn a label on and give it a specific name, you would select the Modify label option. When you do, an Alpha Entry pop-up menu appears. You can use the pop-up menu and the keypad on the front panel to name the label. A label name can be a maximum of six characters.

Turn Label Off

Selecting this option turns the label off. When a label is turned off, the bit assignments are saved by the logic analyzer. This gives you the option of turning the label back on and still having the bit assignments if you need them. The waveforms and state listings are also saved.

You can give the same name to a label in the state analyzer as in the timing analyzer without causing an error. The logic analyzer distinguishes between them. An example of this appears in the *Getting Started Guide* and in chapter 15 of this manual.

Polarity (Pol)

Each label has a polarity assigned to it. The default for all the labels is positive (+) polarity. You can change the polarity of a label by placing the cursor on the polarity field and pressing SELECT. This toggles the polarity between positive (+) and negative (–).

In the state analyzer, negative polarity inverts the data.

Bit Assignment

The bit assignment fields allow you to assign bits (channels) to labels. Above each column of bit assignment fields is a line that tells you the bit numbers from 0 to 15, with the left bit numbered 15 and the right bit numbered 0. This line helps you know exactly which bits you are assigning.

The convention for bit assignment is as follows:

* (asterisk) indicates assigned bit
. (period) indicates unassigned bit

At power up the 16 bits of Pod 1 are assigned to the timing analyzer and the 16 bits of Pod 5 are assigned to the state analyzer.

To change a bit assignment configuration, place the cursor on a bit assignment field and press SELECT. You will see the following pop-up menu.

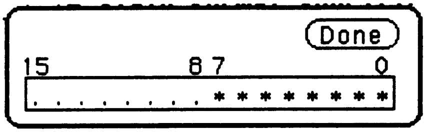


Figure 9-3. Bit Assignment Pop-Up Menu

Use the KNOB to move the cursor to an asterisk or a period and press SELECT. The bit assignment toggles to the opposite state of what it was before. When the bits (channels) are assigned as desired, place the cursor on Done and press SELECT. This closes the pop-up and displays the new bit assignment.

Assigning one channel per label may be handy in some applications. This is illustrated in chapter 8 of the *Getting Started Guide*. Also, you can assign a channel to more than one label, but this usually isn't desired.

Labels may have from 1 to 32 channels assigned to them. If you try to assign more than 32 channels to a label, the logic analyzer will beep, indicating an error, and a message will appear at the top of the screen telling you that 32 channels per label is the maximum.

Channels assigned to a label are numbered from right to left by the logic analyzer. The least significant assigned bit (LSB) on the far right is numbered 0, the next assigned bit is numbered 1, and so on. Since 32 channels can be assigned to one label at most, the highest number that can be given to a channel is 31.

Although labels can contain split fields, assigned channels are always numbered consecutively within a label. The numbering of channels is illustrated with the figure below.

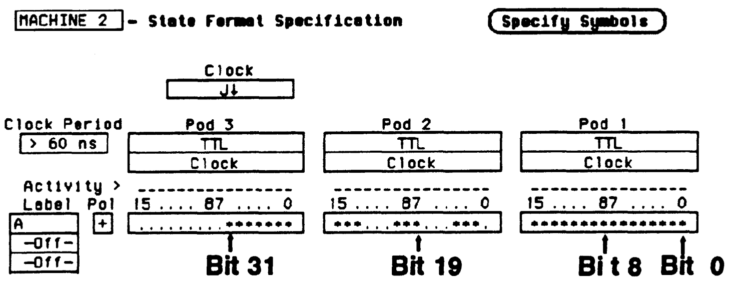


Figure 9-4. Numbering of Assigning Bits

Pod Threshold

Each pod has a threshold level assigned to it. For the HP 1653B Logic Analyzer, threshold levels may be defined for Pods 1 and 2 individually. For the HP 1652B Logic Analyzer, threshold levels may be defined for Pods 1, 2 and 3 individually, and one threshold for Pods 4 and 5. It does not matter if Pods 4 and 5 are assigned to different analyzers. Changing the threshold of one will change the threshold of the other.

If you place the cursor on one of the pod threshold fields and press SELECT, you will see the following pop-up menu.

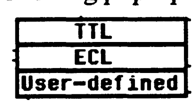


Figure 9-5. Pod Threshold Pop-Up Menu

TTL sets the threshold at + 1.6 volts, and ECL sets the threshold at – 1.3 volts.

The User-defined option lets you set the threshold to a specific voltage between − 9.9 V and + 9.9 V. If you select this option you will see a Numeric Entry pop-up menu as shown.

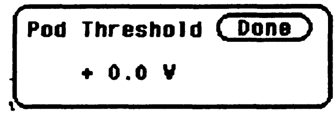


Figure 9-6. User-defined Numeric Entry Pop-Up Menu

You can change the value in the pop-up either with the keypad on the front panel or with the KNOB, which you rotate until you get the desired voltage. When the correct voltage is displayed, press SELECT. The pop-up will close and your new threshold will be placed in the pod threshold field.

The threshold level you specify for the 16 data bits also applies to a pod's clock threshold.

Specify Symbols

This field provides access to the Specify Symbols menu. It differs from the other fields in the State Format Specification menu in that it displays a complete menu instead of a pop-up. The complete description of the Specify Symbols Menu follows the State Format Specification Menu fields later in this chapter.

Clock

The **Clock** field in the Format Specification menu displays the clocks for clocking your system. The display will be referred to as the "clocking arrangement."

The HP 1652B Logic Analyzer has five clock channels, each of which is on a pod. The clocks are connected through the pods simply for convenience. The clock channels are labeled J, K, L, M, and N and are on pods 1 through 5, respectively. The clocking of the state analyzer is synchronous with your system because your analyzer uses the signals present in your system. The signal you use must clock the analyzer when the data you want to acquire is valid.

The HP 1653B Logic Analyzer has two clock channels, each on one of the pods. The J clock is on pod 1 and the K clock is on pod 2.

When you select the **Clock** field, you will see the following pop-up menu with which you specify the clock.

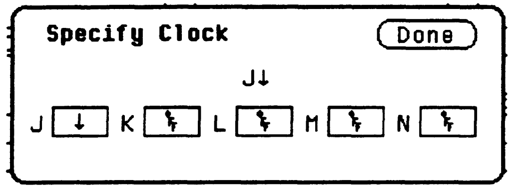


Figure 9-7. Clock Pop-Up Menu

You can use one of the clocks alone or combine them to build one clocking arrangement. If you select a field to the right of one of the clocks in the pop-up you will see another pop-up menu:

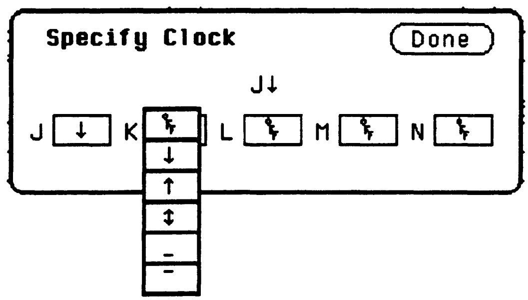


Figure 9-8. Single Clock Pop-Up Menu

You can specify the negative edge of the clock, the positive edge, either edge, a high level, a low level, or the clock to be off.

The clocks are combined by ORing and ANDing them. Clock edges are ORed to clock edges, clock levels are ORed to clock levels, and clock edges are ANDed to clock levels.

For example, if you select ↓ for the J clock, ↑ for the K clock, _ for the M clock, and ‾ for the N clock, the resulting clocking arrangement will appear in the display as:

```
Clock
( J↓ + K↑ ) • ( M_ + N‾ )
```

Figure 9-9. Example of a Clocking Arrangement

With this arrangement, the state analyzer will clock the data when there is a negative edge of the J clock OR a positive edge of the K clock, AND when there is a low level on the M clock OR a high level on the N clock.

You must always specify at least one clock edge. If you try to use only clock levels, the logic analyzer will display a message telling you that at least one edge is required.

Pod Clock

Your logic analyzer has the capability of clocking data in three different ways. The pod **Clock** fields in the State Format Specification menu allow you to specify which of the three ways you want to clock the data.

Each pod assigned to the state analyzer has a pod Clock field associated with it. Selecting one of the pod Clock fields gives you the following pop-up menu:

```
Normal
Demultiplex
Mixed Clocks
```

Figure 9-10. Pod clock Field Pop-Up Menu

Normal

This option specifies that clocking will be done in single phase. That is the clocking arrangement located in the Clock field above the pods in the State Format Specification menu will be used to clock all the pods assigned to this machine.

For example, suppose that the Clock field looks like the following:

```
    Clock
( J↓ + K↑ )
```

Figure 9-11. Example of a Clocking Arrangement

In Normal mode the state analyzer will sample the data on any assigned pods on a negative edge of the J clock OR on a positive edge of the K clock.

Demultiplex

With the HP 1652B/1653B Logic Analyzers, you can clock two different types of data that occur on the same lines. For instance, lines that transfer both address and data information need to be clocked at different times in order to get the right information at the right time.

When you select the Demultiplex option, the pod Clock field changes to "Master | Slave," and two clock fields appear above the pods where just one Clock field used to be. These fields are the Master Clock and Slave Clock, as shown:

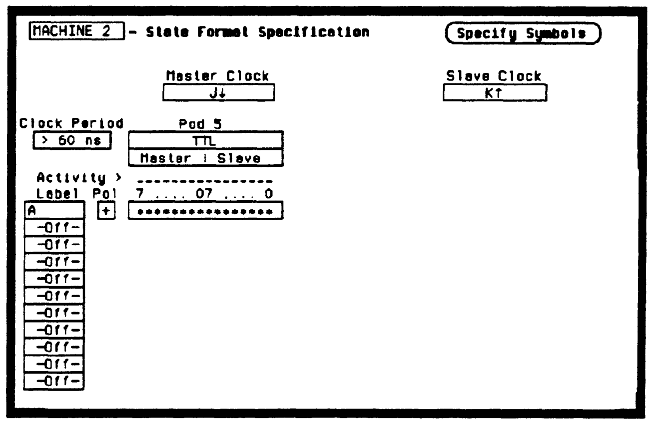


Figure 9-12. Master Clock and Slave Clock

Demultiplexing is done on the data lines of the specified pod to read only the lower eight bits. This is two-phase clocking, with the Master Clock following the Slave Clock. The analyzer first looks for the clocking arrangement that you specify in the Slave Clock. When it sees this arrangement, the analyzer clocks the data present on bits 0-7 of the pod, then waits for the clocking arrangement that you specify in the Master Clock. When it sees this arrangement, it again clocks the data present on bits 0-7 of the pod. The upper eight bits of the pods are ignored and don't need to be connected to your system.

Notice, the bit numbers that appear above the bit assignment field have changed. The bits are now numbered 7....07....0 instead of 15....87....0 . This helps you set up the analyzer to clock the right information at the right time.

The address/data lines AD0-AD7 on the 8085 microprocessor are an example of Demultiplex. During part of the operating time the lines have an address on them, and during other times they have data on them. Hook the lower eight bits of one of the pods to these eight lines and set the Slave and Master Clocks so that they clock the data and the address at the proper time.

In this example, you may choose to assign the bits in the State Format Specification menu similarly to that shown below. In this case you would want to clock the address with the Slave Clock and the data with the Master Clock.

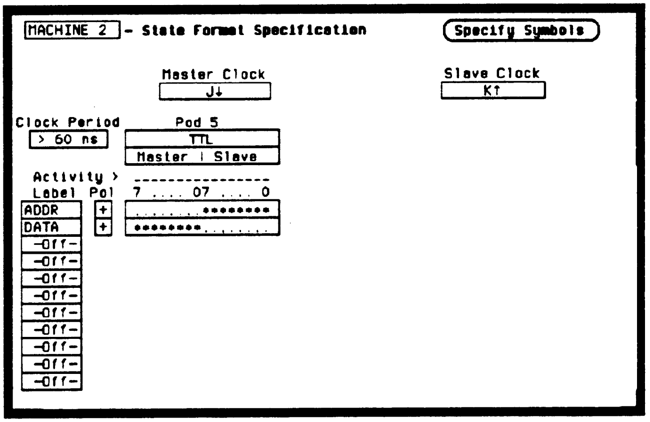


Figure 9-13. Master and Slave Clock Bit Assignments

The Master and Slave Clocks can have the same clocking arrangements. The clocking is still done the same way, with the lower eight bits being clocked first on the Slave Clock, then on the Master Clock.

Mixed Clocks

The Mixed Clocks option allows you to clock the lower eight bits of a pod separately from the upper eight bits. The state analyzer uses Master and Slave Clocks to do this. If you select this option from the pod Clock pop-up, the pod Clock field changes to "Master | Slave," and two Clock fields, Master and Slave, appear above the pods.

As in Demultiplex, the Master Clock follows the Slave Clock. The state analyzer looks for the clocking arrangement given by the Slave Clock and clocks the lower eight bits. Then it looks for the clock arrangement given by the Master Clock and clocks the upper eight bits. Unlike Demultiplex, all 16 bits of a pod are sampled.

The Master and Slave Clocks can have the same clocking arrangements. The clocking is still done the same way, with the lower eight bits clocked on the Slave Clock and the upper eight bits clocked on the Master Clock.

Clock Period

This field provides greater measurement accuracy when your state input clock period is greater than 60 ns. When you select **> 60 ns**, the state analyzer provides greater immunity against noise or ringing in the state input clock signal; also, the logic analyzer provides greater accuracy when triggering another state or timing analyzer or the BNC trigger out.

If your State input clock period is less than 60 ns, you should select **< 60 ns**. This disables the Count field in the State Trace Specification menu because the maximum clock rate when counting is 16.67 MHz (60 ns clock period). This also turns Prestore off.

Specify Symbols Menu

The logic analyzer supplies Timing and State Symbol Tables in which you can define a mnemonic for a specific bit pattern of a label. When measurements are made by the state analyzer, the mnemonic is displayed where the bit pattern occurs if the Symbol base is selected.

It is possible for you to specify up to 200 symbols in the logic analyzer. If you have only one of the internal analyzers on, all 200 symbols can be defined in it. If both analyzers are on, the 200 symbols are split between the two. For example, analyzer 1 may have 150, leaving 50 available for analyzer 2.

To access the Symbol Table in the State Format Specification menu, place the cursor on the **Specify Symbols** field and press SELECT. You will see a new menu as shown. This is the default setting for the Symbol Table in both the timing and state analyzers.

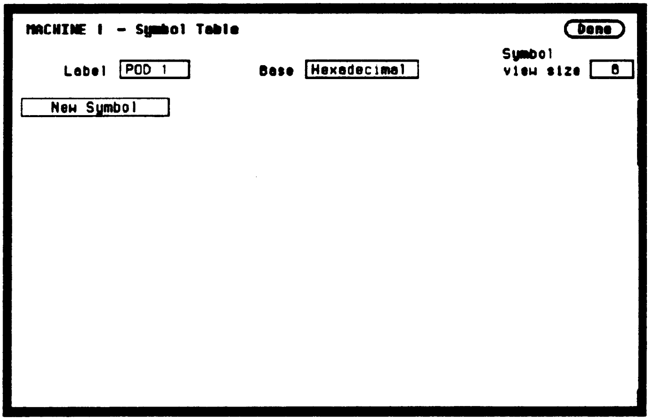


Figure 9-14. Symbol Table Menu

Specify Symbols Menu Fields

There are four fields in the Symbol Table menu. They are:

- Label
- Base
- Symbol view size
- Symbol name

Label

The **Label** field identifies the label for which you are specifying symbols. If you select this field, you will get a pop-up that lists all the labels turned on for that analyzer.

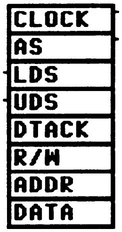


Figure 9-15. Lable Pop-Up Menu

Each label has a separate symbol table. This allows you to give the same name to symbols defined under different labels. In the Label pop-up select the label for which you wish to specify symbols.

Base

The **Base** field tells you the numeric base in which the pattern will be specified. The base you choose here will affect the pattern field of the State Trace Specification menu. This is covered later in this chapter.

To change the base, place the cursor on the field and press SELECT. You will see the following pop-up menu.

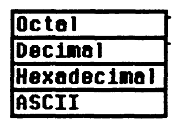


Figure 9-16. Base Pop-Up Menu

If more than 20 channels are assigned to a label, the Binary option is not offered in the pop-up. The reason for this is that when a symbol is specified as a range, there is only enough room for 20 bits to be displayed on the screen.

Decide which base you want to work in and choose that option from the numeric Base pop-up menu.

If you choose the ASCII option, you can see what ASCII characters the patterns and ranges defined by your symbols represent. ASCII characters represented by the decimal numbers 0 to 127 (hex 00 to 7F) are offered on your logic analyzer. Specifying patterns and ranges for symbols is discussed in the next section.

Note

You cannot specify a pattern or range when the base is ASCII. First define the pattern or range in one of the other bases, then switch to ASCII to see the ASCII characters.

Symbol View Size

The **Symbol view size** field lets you specify how many characters of the symbol name will be displayed when the symbol is referenced in the State Trace Specification menu and the State Listing menu. Selecting this field gives you the following pop-up.

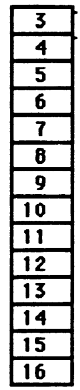


Figure 9-17. Symbol View Size Pop-Up Menu

You can have the logic analyzer display from 3 to all 16 of the characters in the symbol name. For more information see "State Trace Specification Menu" and "State Listing Menu" later in this chapter.

Symbol Name

When you first access the Symbol Table, there are no symbols specified. The symbol name field reads "New Symbol." If you select this field, you will see an Alpha Entry pop-up menu on the display. Use the pop-up menu and the keypad on the front panel to enter the name of your symbol. A maximum of 16 characters can be used in a symbol name.

When you select the Done field in the Alpha Entry pop-up menu the name that appears in the symbol name field is assigned and two more fields appear in the display.

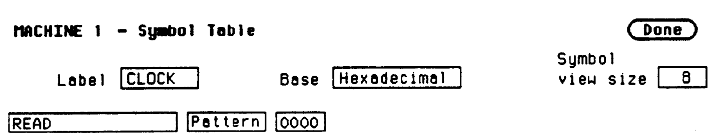


Figure 9-18. Symbol Defined as a Pattern

The first of these fields defines the symbol as either a Pattern or a Range. If you place the cursor on this field and press SELECT, it will toggle between Pattern and Range.

When the symbol is defined as a pattern, one field appears to specify what the pattern is. Selecting this field gives you a pop-up with which you can specify the pattern. Use the keypad and the DON'T CARE key on the front panel to enter the pattern. Be sure to enter the pattern in the numeric base that you specified in the Base field.

Figure 9-19. Specify Pattern Pop-Up Menu

If the symbol is defined as a range, two fields appear in which you specify the upper and lower boundaries of the range.

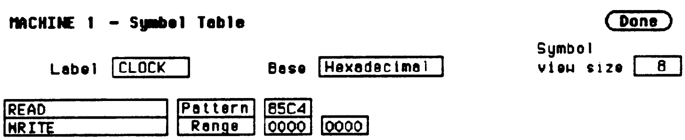


Figure 9-20. Symbol Defined as a Range

Selecting either of these fields gives you a pop-up with which you can specify the boundary of the range.

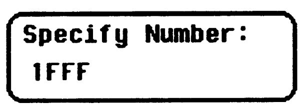


Figure 9-21. Specify Range Pop-Up Menu

You can specify ranges that overlap or are nested within each other. Don't cares are not allowed.

To add more symbols to your symbol table, place the cursor on the last symbol defined and press SELECT. A pop-up menu appears as shown.

Modify symbol
Insert new symbol
Delete symbol

Figure 9-22. Symbol Pop-Up Menu

The first option in the pop-up is Modify symbol. If you select this option, you will see an Alpha Entry pop-up menu with which you can change the name of the symbol.

The second option in the pop-up is Insert new symbol. It allows you to specify another symbol. When you select it, you will see an Alpha Entry pop-up menu. Use the menu and the keypad on the front panel to enter the name of your new symbol. When you select Done, your new symbol will appear in the Symbol Table. The third option in the pop-up is Delete symbol. If you select this option, the symbol will be deleted from the Symbol Table.

Leaving the Symbol Table Menu

When you have specified all your symbols, you can leave the Symbol Table menu in one of two ways. One method is to place the cursor on the Done field and press SELECT. This puts you back in the Format Specification menu that you were in before entering the Symbol Table. The other method is to press the FORMAT, TRACE, or DISPLAY keys on the front panel to get you into the respective menu.

10

State Trace Menu

Introduction

This chapter describes the State Trace menu and the pop-up menus that you will use on your state analyzer. The purpose and functions are described in detail, and we have included many illustrations and examples to make the explanations clearer.

The Trace Specification menu allows you to configure the state analyzer to capture only the data of interest for your measurement. In the state analyzer you can configure the analyzer to trigger on a sequence of states. The default setting is shown in figure 10-1 below.

For an example of setting up a trace configuration for a State analyzer, refer to your *Getting Started Guide* or "State Analyzer Measurement Example" in Chapter 15 of this manual.

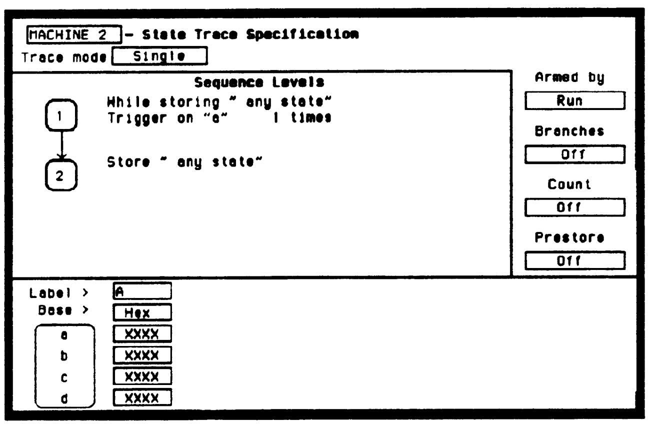


Figure 10-1. State Trace Specification Menu

Accessing the State Trace Menu

The State Trace menu can be accessed by pressing the TRACE key on the front panel. If the Timing Trace Specification menu is displayed when you press the TRACE key, you will have to switch analyzers. This is not a problem, it merely indicates that the last action you performed in the System Configuration Menus was on the timing analyzer.

State Trace Menu Fields

The menu is divided into three sections: the Sequence Levels in the large center box, the acquisition fields at the top and right of the screen, and the qualifier and pattern fields at the bottom of the screen.

Before describing the fields in the menu, we need to define a few terms. These terms will be used in the discussions of the fields, so understanding their meanings is essential.

Pattern Recognizers: a pattern of bits (0, 1, or X) in each label. There are eight recognizers available when one state analyzer is on. Four are available to each analyzer when two state analyzers are on. The pattern recognizers are given the names a through h and are partitioned into groups of four, a-d and e-h.

Range Recognizer: recognizes data which is numerically between or on two specified patterns. One range term is available and is assigned to the first state analyzer created by assigning pods to it or if only one analyzer is on, then the range term is assigned to it.

Qualifier: user-specified term that can be anystate, nostate, a single pattern recognizer, a range recognizer, the complement of a pattern or range recognizer, or a logical combination of pattern and range recognizers. To specify a qualifier, you will use the pop-up shown in figure 10-2. This pop-up appears when accessed through the five different fields encountered when setting qualifiers throughout the State Trace menu.

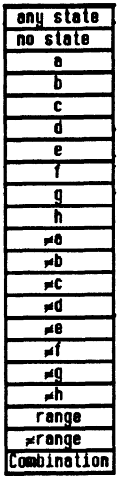


Figure 10-2. Qualifier Pop-Up Menu

If you select the **Combination** option in the pop-up, you will see a pop-up similar to that shown below.

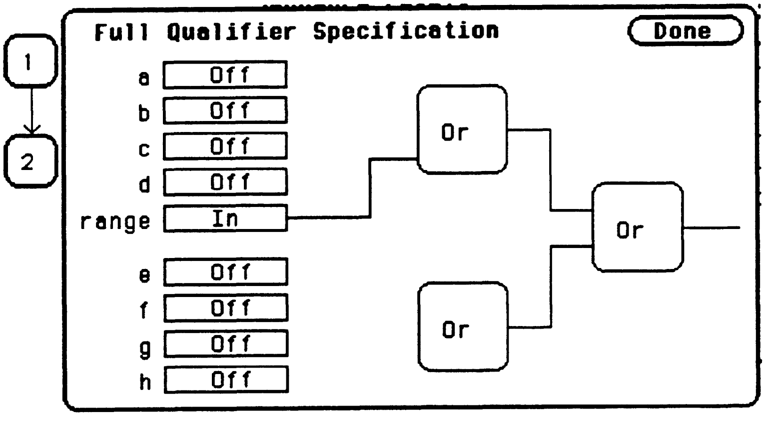


Figure 10-3. Full Qualifier Specification Pop-Up

Note

If two multi-pod state analyzers are on, the qualifier pop-up menu will show that only four pattern recognizers are available to each analyzer. Pattern recognizers a-d and the range recognizer are assigned to the first analyzer created, and pattern recognizers e-h go with the second analyzer. In the Full Qualifier Specification pop-up there will be only one OR gate and one set of pattern recognizers.

With this Full Qualifier Specification pop-up, you specify a logical combination of patterns or ranges as the qualifier. The pattern recognizers are always partitioned into the groups of four shown. Only one operator is allowed between the patterns in a group. Patterns in uncomplimented form (a, b, etc.) can only be ORed.

The complements of patterns (≠ a, ≠ b, etc.) can only be ANDed. For example, if the first OR field (gate) is changed to AND, all the patterns for that gate are complemented, as shown below.

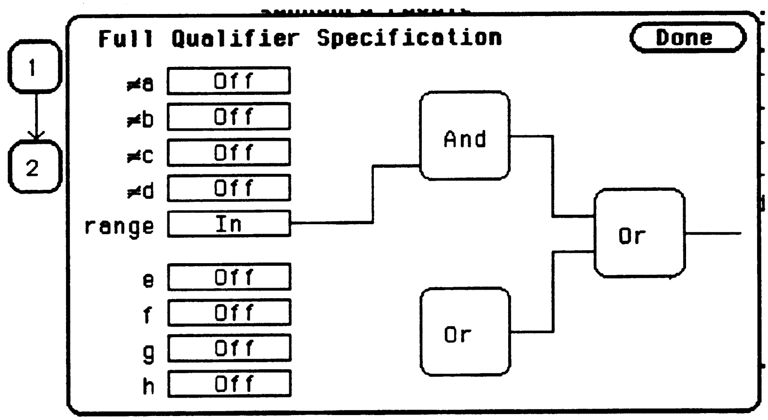


Figure 10-4. Complemented Patterns

To specify a pattern to be used in the combination, place the cursor on the pattern recognizer field and press SELECT. The field toggles from Off to On and a connection is drawn from the pattern field to the gate. In figure 10-5, patterns b, c and d and the range are ORed together, and e and g are ANDed together.

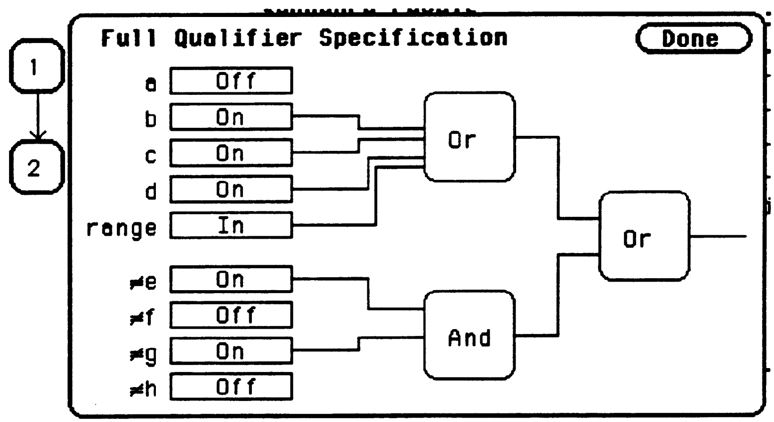


Figure 10-5. Patterns Assigned for Logical Combinations

As shown in the previous figures, the range is included with the first group of patterns (a-d). If you select the range field, you will see the following pop-up menu.

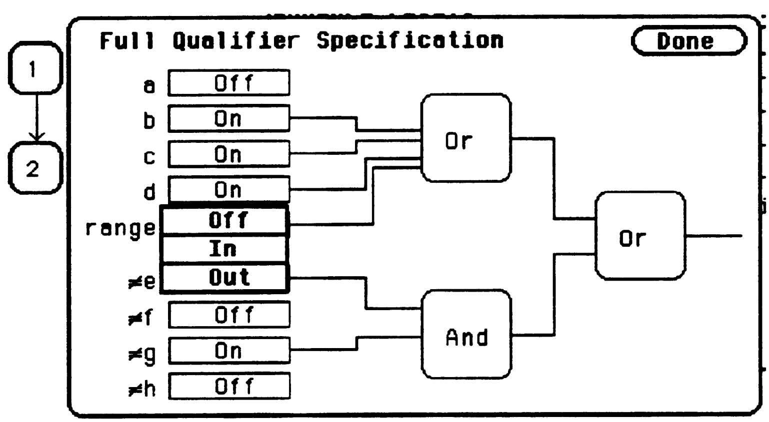


Figure 10-6. Range Specification Pop-Up Menu

Off disconnects the range from the qualifier specification. **In** indicates that the contents of the range are to be in the qualifier specification, and **Out** indicates that the complement of the range is to be in the qualifier specification.

When you have specified your combination qualifier, select Done. The **Full Qualifier Specification** pop-up closes and the Boolean expression for your qualifier appears in the field for which you specified it.

```
While storing [ (b+c+d+range)+(≠e•≠g) ]
```

Figure 10-7. Boolean Expression for Qualifier

Sequence Levels

There are eight trigger sequence levels available in the state analyzer. You can add and delete levels so that you have from two to eight levels at a time.

Only three levels appear in the Sequence Levels display at one time. To display other levels so that they can be accessed, press the up/down ROLL key and rotate the KNOB.

If you select level 1 shown in figure 10-1, you will see the following pop-up menu:

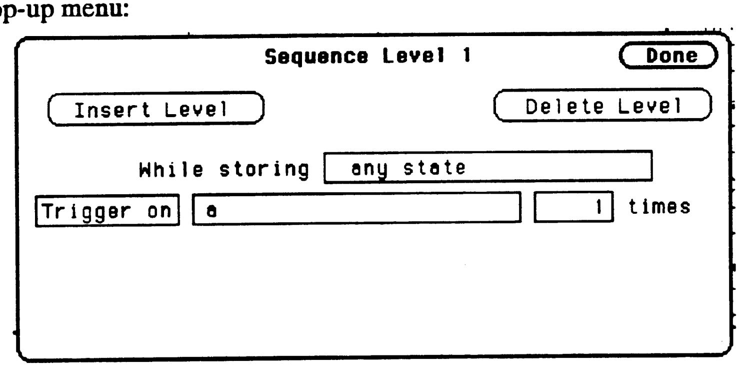


Figure 10-8. Sequence Level Pop-Up Menu

Not all sequence level pop-up menus look like this one. This happens to be the trigger sequence level in which you specify the state on which the analyzer is to trigger. The trigger term can occur in any of the first seven levels, and it is not necessarily a selectable field. The fields in the menu of figure 10-8 are described on the following pages.

Insert Level

To insert a level, place the cursor on the field labeled **Insert Level** and press SELECT. You will see the following pop-up menu.

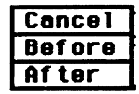


Figure 10-9. Insert Level Pop-Up Menu

Cancel returns you to the sequence level pop-up without inserting a level. **Before** inserts a level before the present level. **After** inserts a level after the present level. If there are eight levels, the **Insert Level** field doesn't appear in the sequence level pop-ups.

Delete Level

If you want to delete the present level, select the field labeled **Delete Level**. You will see a pop-up menu with the choices Cancel and **Execute. Cancel** returns you to the sequence level pop-up without deleting the level. **Execute** deletes the present level and returns you to the State Trace Specification menu.

Note

If there are only two levels, neither field can be deleted even though the Delete Level field still appears in the menu. There will always be a trigger term level and a store term level in Sequence Levels. Therefore, if you try to delete either of these, all terms you have specified in these levels will be set to default terms, and, the trigger and store term levels will remain.

Storage Qualifier

Each sequence level has a storage qualifier. The storage qualifier specifies the states that are to be stored and displayed in the State Listing. Selecting this field gives you the qualifier pop-up menu shown in figure 10-2, with which you specify the qualifier.

As an example, suppose you specify the storage qualifier in a sequence level as shown below.

```
While storing [ a+d ]
```

Figure 10-10. Storage Qualifier Example

The only states that will be stored and displayed are the states given by pattern recognizers a and d.

Branching Qualifier

Every sequence level except the last has a primary branching qualifier. With the branching qualifier, you tell the analyzer to look for a specific state or states. The primary branching qualifier advances the sequencer to the next level if its qualifier is satisfied.

In the example of figure 10-8, the branching qualifier tells the analyzer when to trigger. In other sequence levels, the qualifier may simply specify a state that the analyzer is to look for before continuing to the next level.

Some sequence levels also have a secondary branching qualifier. The secondary branch will, if satisfied, route the sequencer to a level that you define. This is covered in more detail in "Branches" later in this chapter.

Occurrence Counter

The primary branching qualifier has an occurrence counter. With the occurrence counter field you specify the number of times the branching qualifier is to occur before moving to the next level.

To change the value of the occurrence counter, position the cursor on the field and either press SELECT or press a numeric key on the front-panel keypad. You will see a pop-up similar to that shown below.

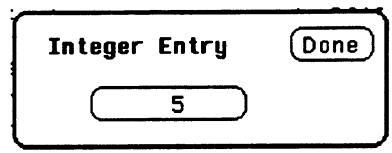


Figure 10-11. Occurrence Counter Pop-Up Menu

You can change the value by either rotating the KNOB or pressing the appropriate numeric keys. The qualifier can be specified to occur from one to 65535 times.

Storage Macro

Your logic analyzer has the capability of post-trigger storage through a storage macro. The storage macro is available only in the second to last level, and it consumes both that level and the last level. The field in figure 10-8 allows you to configure the state analyzer for post-trigger storage. This field does not always say Trigger on. If the sequence level is not a trigger level, the field will say Then find, as shown below.

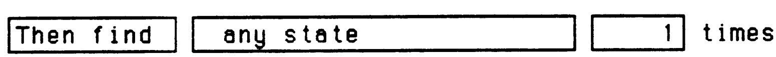


Figure 10-12. Then Find Branching Qualifier

Selecting the field gives you a pop-up with two options. One option is what the field said previously. The other option is Enable on. If you select this option, the Sequence Level pop-up changes to look similar to that shown below.

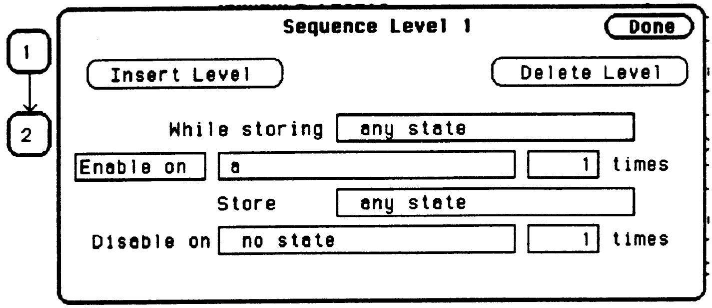


Figure 10-13. Storage Macro Sequence Level Example

Note

Enable on can only be the next to last term, and when on, the last term is combined with the Enable term.

You specify qualifiers for the states on which you want the macro to enable, the states you want to store, and the states on which you want the macro to disable. The storage macro is a loop that keeps repeating itself until memory is full. The loop is repeated when the disable qualifier is satisfied. As an example, suppose you configure the sequence level of figure 10-13 to look like that shown below.

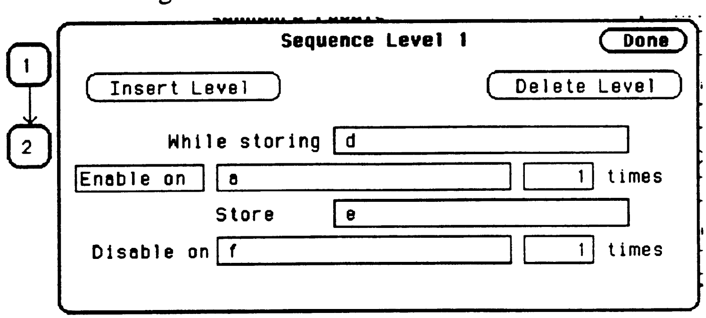


Figure 10-14. Sequence Level Pop-up with Storage Macro

The logic analyzer will store the state given by pattern recognizer **d** until it comes across the state given by **a**. When it sees state **a**, the logic analyzer starts to store the state given by pattern recognizer **e**. It stores that state until it sees the state given by **f**, at which time it disables and starts the process all over again. The analyzer repeats this process until its memory is full.

Reading the Sequence Level Display

Reading the display is fairly straightforward. For example, suppose your display looks like that shown below.

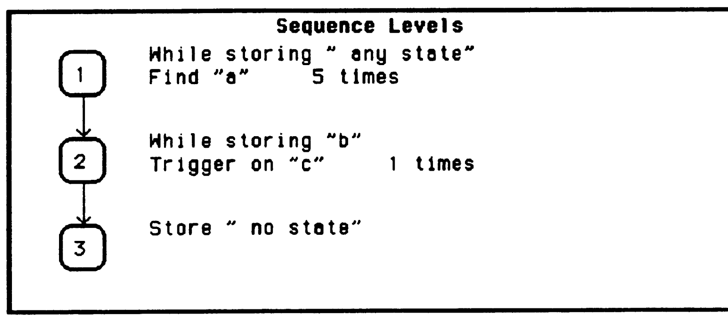


Figure 10-15. Sequence Level Display Example

In level 1 anystate is stored while the logic analyzer searches for five occurrences of the pattern given by pattern recognizer a. When the five occurrences are found, the sequencer moves on to level 2. In level 2 the state given by pattern recognizer b is stored until one occurrence of the pattern given by pattern recognizer c is found and the logic analyzer triggers. In level 3 nostate is stored, so the last state stored is the trigger state.

An example of a state listing for the previous State Trace configuration is shown below. The state patterns specified are:

a = B03C
b = 0000
c = 8930

```
MACHINE 2    -   STATE LISTING

Label   >  A
Base    >   Hex

 -0028      4E75
 -0027      61E6
 -0026      0000
 -0025      88C8
 -0024      B03C
 -0023      00FF
 -0022      6730
 -0021      48E7
 -0020      4E75
 -0019      3000
 -0018      0000
 -0017      8930
 -0016      B03C
 -0015      00FF
 -0014      67F8
 -0013      B03C
 -0012      61FA
 -0011      B03C
 -0010      0000
 -0009      8930
 -0008      4EFA
 -0007      FF9A
 -0006      61E6
 -0005      B03C
 -0004      0000
 -0003      0000
 -0002      0000
 -0001      0000
 +0000      8930
```

Figure 10-16. State Listing Example

Anystate was stored while the analyzer looked for five occurrences of the state B03C. After the fifth occurrence was found, only state 0000 was stored until state 8930 was found, and the analyzer triggered. After the trigger, no states were stored.

Acquisition Fields

The acquisition fields are comprised of the Trace mode, Armed by Branches, Count, and Prestore fields, as shown below.

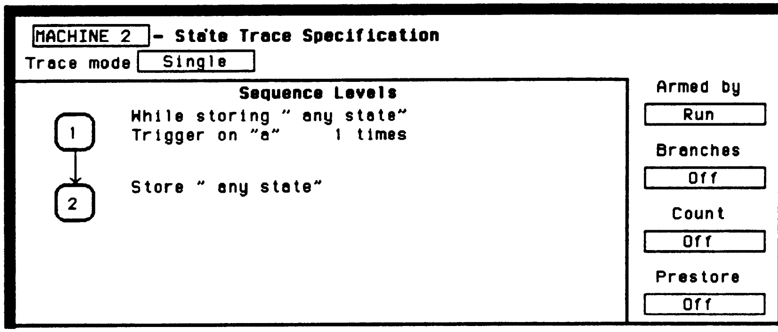


Figure 10-17. State Trace Acquisition Fields

Trace Mode

You specify the mode in which the state analyzer will trace with the **Trace mode** field. You have two choices for trace mode: Single and Repetitive. If you place the cursor on the field and press SELECT the field toggles from one mode to the other.

Single Trace mode acquires data once per trace. Repetitive Trace mode repeats single acquisitions until the STOP key on the front panel is pressed, or if Stop measurement is on, until conditions specified with the X and O markers in the State Listing menu are met.

If both analyzers are on, only one trace mode can be specified. Specifying one trace mode for one analyzer sets the same trace mode for the other analyzer.

Armed By

The **Armed by** field lets you specify how your state analyzer is to be armed. The analyzer can be armed by the RUN key, the other analyzer, the scope or an external instrument through the BNC Input port. Any of these can tell the analyzer when to start capturing data.

When you select the **Armed by** field, a pop-up menu appears like that shown below. The first two options always appear in the pop-up. The third and fourth options will give the name of the other analyzer and scope. If the other analyzer or scope is off, or if the other machine or scope is being armed by this machine, these options will not be available.

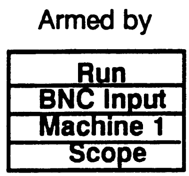


Figure 10-18. Armed By Pop-Up Menu

Branches

The **Branches** field allows you to configure the sequencer of the state analyzer to branch from one sequence level to another with secondary branching qualifiers, or to restart when a certain condition is met. Selecting this field gives you the following pop-up menu.

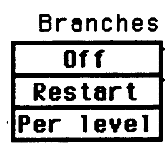


Figure 10-19. Branches Pop-Up Menu

Off

If you select Off, all secondary branching qualifiers are deleted from the sequence levels. Only the primary branches remain.

Restart

The Restart option allows you to start over from sequence level 1 when a specified condition is met. This can be handy if you have code that branches off in several paths and you want the analyzer to follow one certain path. If the analyzer goes off on an undesired path, you would want the analyzer to stop and go back to the beginning and take the correct path.

If you select the Restart option, you will see a qualifier pop-up menu like that shown in figure 10-2. With the pop-up you select the qualifier for the pattern on which you want your analyzer to start over.

When your state analyzer is reading data it proceeds through the sequence. If a term doesn't match the branching qualifier, it is then checked against Restart. If the term matches, the state analyzer jumps back the sequence level 1.

Per Level

Selecting the Per level option allows you to define a secondary branching qualifier for each sequence level. A statement is added in each level so that you can configure the analyzer to move to a different level when a specified condition is met. An example of a sequence level with a secondary branching qualifier is shown in the figure below.

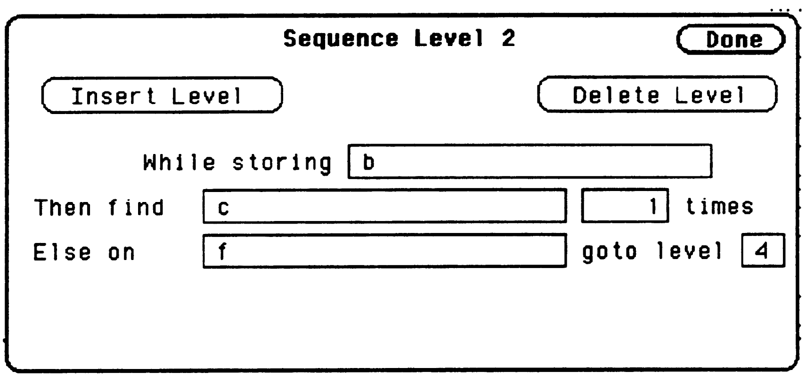


Figure 10-20. Secondary Branching Qualifier

With this configuration, the state analyzer will store b until it finds c. If it finds f before it finds c, it will branch to sequence level 4. If you have specified a storage macro in the next to last sequence level the Else on statement will not appear in that level since a secondary branching qualifier already exists for that level.

In the last sequence level, which only specifies states that are to be stored, the secondary branching qualifier statement looks like that shown below.

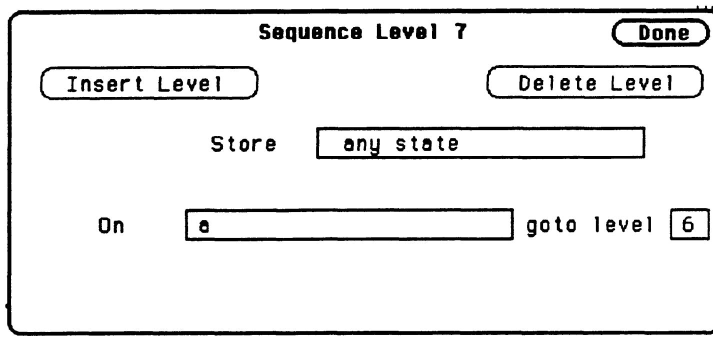


Figure 10-21. Secondary Branch Qualifier in Last Level

In this example, as the state analyzer stores anystate, it will branch to sequence level 6 if it finds the state given by qualifier a.

The trigger sequence level is used as a boundary for branching between levels. This level and the levels that occur before it cannot branch to levels that occur after the trigger level, and vice versa. Therefore, if there are eight sequence levels and level 5 is the trigger sequence level, then levels 1 through 5 can branch to levels 1 through 5 only, and levels 6 through 8 can branch to levels 6 through 8 only.

You can tell if secondary branch qualifiers have been specified by looking at the Sequence Levels display. Figure 10-22 shows how the display looks with the configuration that was given in figure 10-20. An arrow is drawn out of level 2, indicating that branching originates from that level, and an arrow is drawn to level 4 to indicate that a branch is going to that level.

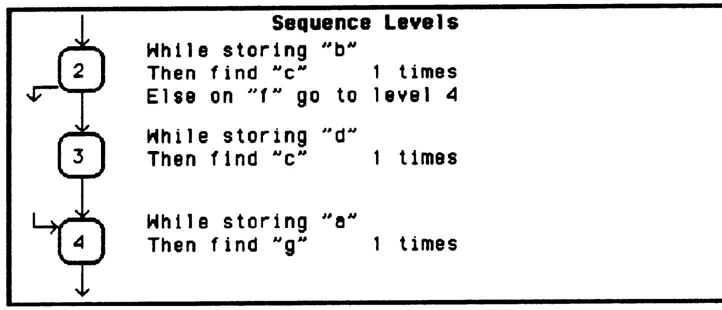


Figure 10-22. Branching Between Sequence Levels

Each sequence level can branch to only one level through a secondary branching qualifier. However, the number of times to which a level can be branched is limited only by the number of levels present. A level can have only one arrow pointing away from it, but it can have two pointing to it if more than one other level is branching to it. An example of this is shown in the figure below. The arrow with two tails indicates that a level above and a level below branch to this level.

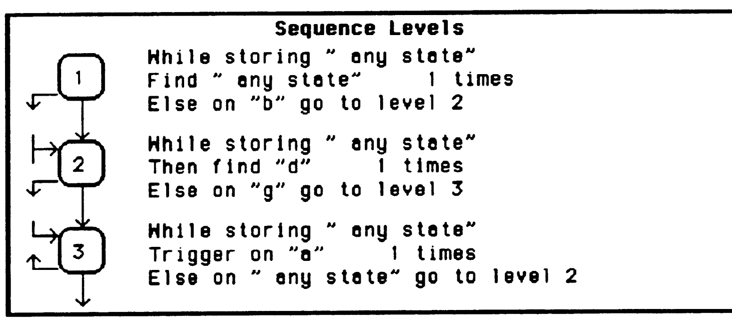


Figure 10-23. Multiple Branching Between Levels

Count

The Count field allows you to place tags on states so you can count them. Counting cuts the acquisition memory in half from 1k to 512 and the maximum clock rate is reduced to 16.67 MHz.

Note

Count (State Trace menu) is turned off when "Clock Period" is set to < 60 ns in the State Format Specification menu since the clock rate is greater than 16.67 MHz. If you select Count, the clock period automatically changes to > 60 ns.

Selecting this field gives you the following pop-up menu.

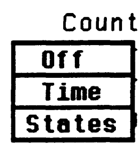


Figure 10-24. Count Pop-Up Menu

Off

If you select Off, the states are not counted in the next measurement.

Time

If you select Time counting, the time between stored states is measured and displayed (after the next run) in the State Listing under the label Time. The time displayed can be either relative to the previous state or to the trigger. The maximum time between states is 48 hours.

An example of a state listing with time tagging relative to the previous state is shown in figure 10-25.

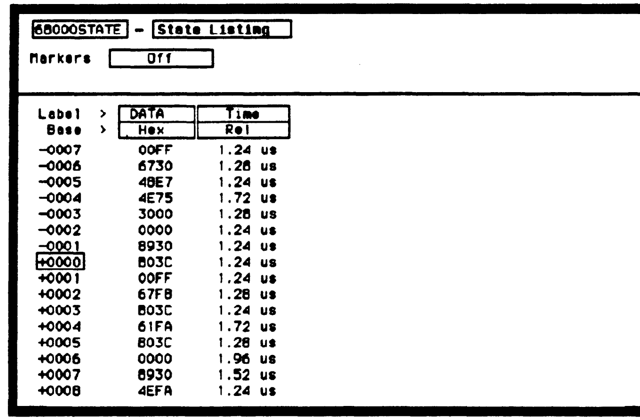


Figure 10-25. Relative Time Tagging

An example of a state listing with time tagging relative to the trigger is shown below.

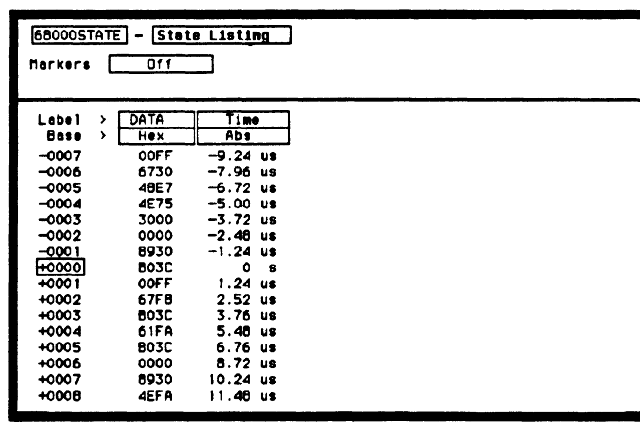


Figure 10-26. Absolute Time Tagging

States

State tagging counts the number of qualified states between each stored state. If you select this option, you will see a qualifier pop-up menu like that shown in figure 10-2. You select the qualifier for the state that you want to count.

In the State Listing, the state count is displayed (after the next run) under the label States . The count can be relative to the previous stored state or to the trigger. The maximum count is 4.4 X 10E12.

An example of a state listing with state tagging relative to the previous state is shown below.

MACHINE 2 - State Listing

Markers Off

Label >	ADDR	States
Base >	Hex	Rel
+0000	0561	
+0001	0564	2
+0002	056E	11
+0003	0570	1
+0004	0576	30
+0005	057B	29
+0006	0566	56352
+0007	0567	0
+0008	0564	56448
+0009	056E	11
+0010	0570	1
+0011	0576	30
+0012	057B	29
+0013	0566	56352
+0014	0567	0
+0015	0564	56448

Figure 10-27. Relative State Tagging

An example of a state listing with state tagging relative to the trigger is shown below.

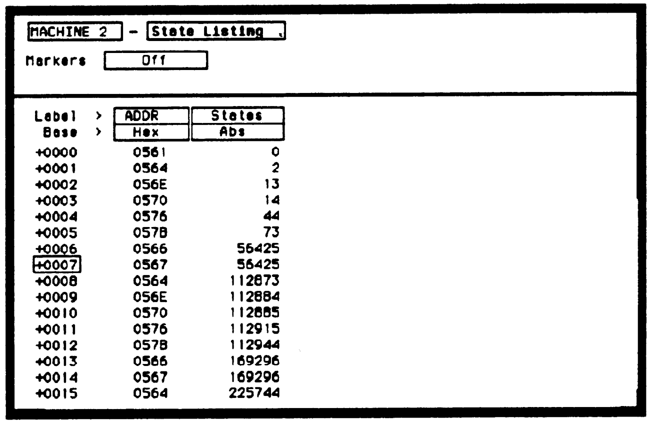


Figure 10-28. Absolute State Tagging

Prestore

Prestore allows you to store two qualified states before each state that is stored. There is only one qualifier that enables prestore for each sequence level. If you select this field, you will see a pop-up with the options **Off** and **On** . Selecting On gives you a qualifier pop-up menu like that in figure 10-2, from which you choose the pattern range or combination of patterns and ranges that you want to prestore.

Note

Prestore is only available when clock period is > 60 ns. If you select Prestore, the clock period automatically changes to > 60ns if it was previously set to < 60 ns.

During a measurement, the state analyzer stores in prestore memory occurrences of the states you specify for prestore. A maximum of two occurrences can be stored. If there are more than two occurrences previous ones are pushed out. When the analyzer finds a state that has been specified for storage, the prestore states are pushed on top of the stored state in memory and are displayed in the State Listing.

Qualifier and Pattern Fields

The qualifier and pattern fields appear at the bottom of the State Trace Specification menu. They allow you to specify patterns for the qualifiers that are used in the sequence levels.

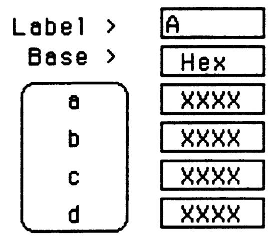


Figure 10-29. Qualifier and Pattern Fields

Label

The **Label** fields display the labels that you specified in the State Format Specification menu. The labels appear in the order that you specified them; however, you can change the order. Select one of the label fields and you will see a pop-up menu with all the labels. Decide which label you want to appear in the label field and select that label. The label that was there previously switches positions with the label you selected from the pop-up.

Base

The **base** fields allow you to specify the numeric base in which you want to define a pattern for a label. The base fields also let you use a symbol that was specified in the State Symbol Table for the pattern. Each label has its own base defined separately from the other labels. If you select one of the base fields, you will see the following pop-up menu. Decide which base you want to define your pattern in and select that option.

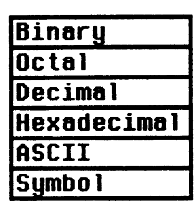


Figure 10-30. Numeric Base Pop-Up Menu

One of the options in the Base pop-up is ASCII. It allows you to see the ASCII characters that are represented by the pattern you specify in the pattern fields.

Note

You cannot define ASCII characters directly. You must first define the pattern in one of the other numeric bases; then you can switch the base to ASCII to see the ASCII characters.

The Symbol option in the Base pop-up allows you to use a symbol that has been specified in the State Symbol Tables as a pattern. In the pattern fields you specify the symbols you want to use.

Qualifier Field

If you select the qualifier field, you will see the following pop-up menu.

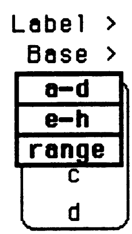


Figure 10-31. Qualifier Field Pop-Up Menu

Patterns

The pattern recognizers are in two groups of four: a-d and e-h. If you select one of these two options, the qualifier field will contain only those pattern recognizers. For instance, the qualifier field in figure 10-29 contains only the recognizers a-d.

Ranges

If you select the range option, the qualifier and pattern fields look similar to that shown below.

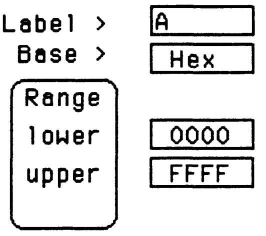


Figure 10-32. Range Qualifier and Pattern Fields

Only one range can be defined, and it can be defined over only one label, hence over only 32 channels. The channels do not have to be adjacent to each other. The logic analyzer selects the label over which the range will be defined by looking at the labels in order and choosing the first one that has channels assigned under only two pods. A label that contains channels from more than two pods cannot be selected for range definition. If all the labels have channels assigned under more than two pods, the range option is not offered in the qualifier field pop-up menu. However, in the HP 1653B, the range option will always be offered since the analyzer has only two pods.

Pattern Fields

The pattern fields allow you to specify the states that you want the state analyzer to search for and store. Each label has its own pattern field that you use to specify a pattern for that label (if you are defining a pattern for a pattern recognizer).

During a run, the state analyzer looks for a specified pattern in the data. When it finds the pattern, it either stores the state or states or it triggers, depending on the step that the sequencer is on.

11

State Listing Menu

Introduction

This chapter describes the State Listing menus and how to interpret it. It also tells you how to use the fields to manipulate the displayed data so you can find your measurement answers. The State Listing menu is the display menu of the state analyzer.

There are two different areas of the state listing display, the menu area and the listing area. The menu area is in the top one-fourth of the screen and the listing area is the bottom three-fourths of the screen.

The listing area displays the data that the state analyzer acquires. The data is displayed in a listing format as shown below.

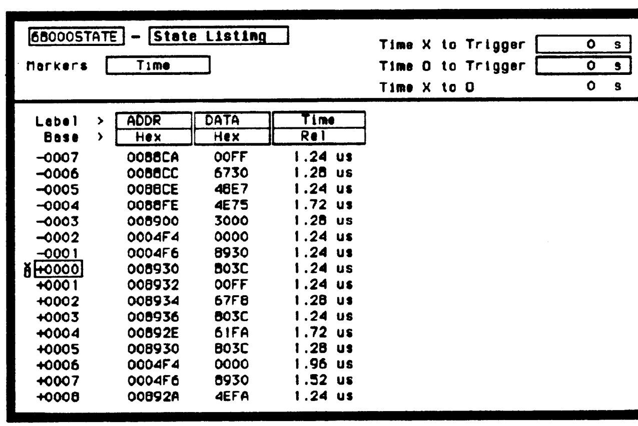


Figure 11-1. State Listing Menu

This listing display shows you 16 of the possible 1024 lines of data at one time. You can use the ROLL keys and the KNOB to roll the listing to the lines of interest.

The column of numbers at the far left represents the location of the acquired data in the state analyzer's memory. The trigger state is always 0000. At the vertical center of this column you will see a box containing a number. The box is used to quickly select another location in the state listing. The rest of the columns (except the Time/States column) represent the data acquired by the state analyzer. The data is grouped by label and displayed in the number base you have selected (hexadecimal is the default base).

When the Time or States option is selected in the Count field (State Trace Specification Menu), the acquired data will be displayed with time or state tags.

The Time column displays either the Rel(ative) time (time from one state to the next) or Abs(olute) time (time from each state to the trigger).

The States column displays the number of qualified states Rel(ative) to the previously stored state or the trigger (absolute).

Accessing the State Listing Menu

The State Listing Menu is accessed by pressing the DISPLAY key on the front panel when the state analyzer is on. It will automatically be displayed when you press RUN. If the Timing Waveforms is displayed when you press the DISPLAY key, you will have to switch analyzers. This is not a problem, it merely indicates that you were in the timing analyzer or you had performed an action to the timing analyzer in the System Configuration Menu.

State Listing Menu Fields

The menu area contains fields that allow you to change the display parameters, place markers, and display listing measurement parameters.

Figure 11-2. State Listing Menu Fields

Markers

The **Markers** field allows you to specify how the X and O markers will be positioned on the state listing. The State Trace Specifications menu options are:

If Count in the State Trace menu is Off, the marker options are:

- Off
- Pattern

If Count in the State Trace menu is set to Time, the marker options are:

- Off
- Pattern
- Time
- Statistics

If Count in the State Trace menu is set to State, the marker options are:

- Off
- Pattern
- State

Markers Off

When the markers are off they are not displayed, but are still placed at the specified points in the data. If Stop measurement is on and the Stop measurement criteria are present in the data, the measurement will stop even though the markers are off.

Markers Patterns

When the markers are set to patterns, you can specify patterns on which the logic analyzer will place the markers. You can also specify how many occurrences of each marker pattern the logic analyzer looks for. This use of the markers allows you to find a specific pattern for each label in the acquired data.

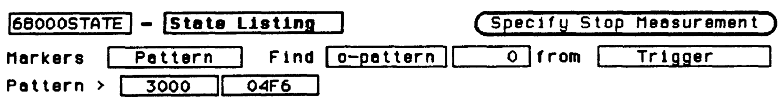


Figure 11-3. Markers Set to Patterns

Patterns for each marker (X and O) can be specified. They can be specified for both markers in each label. The logic analyzer searches for the logical "and" of patterns in all labels.

In the **Find X (O)-pattern 0 from Trigger** field you specify how many occurrences of the marked pattern from a reference point you want the logic analyzer to search for. The reference points are:

- Trigger
- Start (of a trace)
- X Marker (only available in O marker pattern specification)

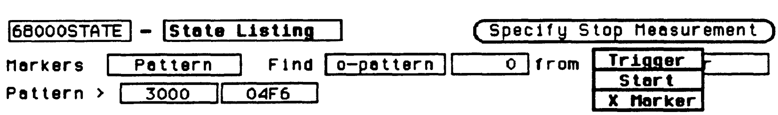


Figure 11-4. Search Reference Pop-Up Menu

Stop Measurement

Another feature of markers set to patterns is Stop Measurement. You can specify either stop measurement when **X-O is** ____ or **Compare is** ____. The options for X-O are: Less than, Greater than, In range, Not in range. The options for Compare are: Equal and Not Equal (see figure 11-5).

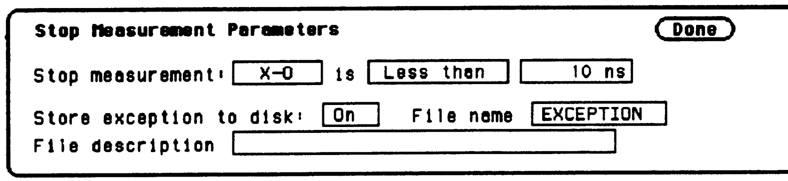


Figure 11-5. Markers Patterns Pop-Up Menu

With this feature you can use the logic analyzer to look for a specified time or range of time between the marked patterns and to stop acquiring data when it finds this time between markers. The X marker must precede the O marker.

Also available is **Store exception to disk** which allows you to specify a file on the disk that exceptions can be stored in. The default filename is EXCEPTION. When the trace mode is repetitive and Store exception to disk is on, the following process takes place: data is acquired until the stop criteria is met, data acquisition will stop, data in the acquisition memory will be stored on the disk, and data acquisition will resume when the data is stored. This process continues until the disk is full. The data is stored in the same file name; however, the last three characters will automatically be replaced with a numerical serial number. For example, EXCEPTION will change to EXCEPT001 the second time memory is stored.

Note

The upper and lower range boundaries must not be the same value. For example, if you want to stop a measurement when the X and O markers are in range of 200 ns, you should set the range values to 190 ns and 210 ns. This eliminates erroneous measurement termination.

Markers Time

When the markers are set to Time, you can place the markers on states in the listing of interest and the logic analyzer will show the following:

- Time X to Trig(ger).
- Time O to Trig(ger).
- Time X to O.

To position the markers, move the cursor to the field of the marker you wish to position and press SELECT. A pop-up will appear showing the current time for that marker. Either rotate the KNOB or enter a numeric value from the keypad to change the position of that marker. Pressing SELECT when you are finished positions the marker and closes the pop-up.

Figure 11-6. Markers Set to Time

The Time X to O field will change according to the position of the X and O markers. It displays the total time between the states marked by the X and O markers.

Markers Statistics

When statistics are specified for markers, the logic analyzer will display the following:

- Number of total runs.
- Number of valid runs (runs where markers were able to be placed on specified patterns).
- Minimum time between the X and O markers * Maximum time between the X and O markers.
- Average time between the X and O markers.

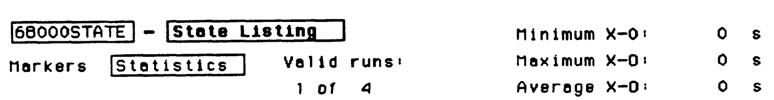


Figure 11-7. Markers Set to Statistics

How the statistics will be updated depends on the state trace mode (repetitive or single).

In repetitive, statistics will be updated each time a valid run occurs until you press STOP. When you press RUN after STOP, the statistics will be cleared and will restart from zero.

In single, each time you press RUN an additional valid run will be added to the data and the statistics will be updated. This will continue unless you change the placement of the X and O markers between runs.

Pattern ___ Field

You use the **Pattern** ___ field to specify the patterns for the X and O markers for each label.

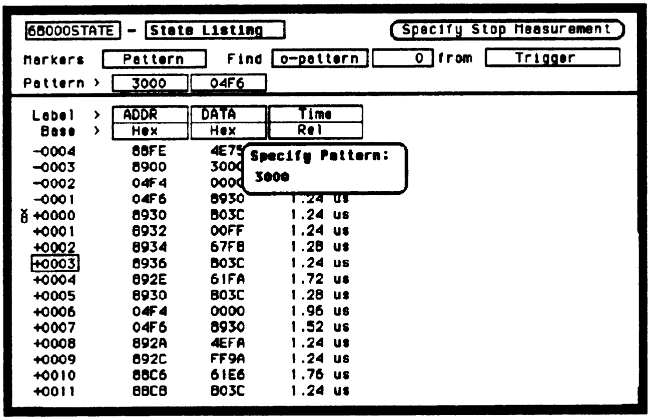


Figure 11-8. Pattern ________ Field Pop-Up Menu

When x-pattern is specified in the **Find** __ **from** __ field, the pop-ups in the Pattern ____ field allow you to specify a pattern for the X marker in each label.

When the O-pattern is specified, the pop-ups in the Pattern _____ field allow you to specify the patterns for the O marker in each label.

12

State Compare Menu

Introduction

State compare is a software post-processing feature that provides the ability to do a bit by bit comparison between the acquired state data listing and a compare data image. You can view the acquired data and the compare image separately. In addition, there is a separate difference listing that highlights the bits in the acquired data that do not match the corresponding bits in the compare image. Each state machine has its own Compare and Difference listings.

You can use the editing capabilities to modify the compare image. Masking capabilities are provided for you to specify the bits that you do not want to compare. "Don't compare" bits can be specified individually for a given label and state row, or specified by channel across all state rows. A range of states can be selected for a comparison. When a range is selected, only the bits in states on or between the specified boundaries are compared.

The comparison between the acquired state listing data and the compare image data is done relative to the trigger points. This means that the two data records are aligned at the trigger points and then compared bit by bit. Any bits in the acquired data that do not match the bits in the compare image are treated as unequal. The don't compare bits in the compare image are ignored for the comparison.

When a logic analyzer configuration is saved to or loaded from a disk, any valid compare data including the data image, etc. is also saved or loaded.

Accessing the Compare Menu

The Compare menu is accessed from the State Listing menu. To access the Compare menu place the cursor on the field **State Listing** and press SELECT. A pop-up appears with the following options:

- State Listing
- State Waveforms
- State Chart
- State Compare

Place the cursor on **State Compare** and press SELECT. The pop-up will close and display the State Compare menu.

The Compare and Difference Listing Displays

Two menus (or displays) in addition to the normal State Listing, are available for making comparison measurements: the Compare Listing and the Difference Listing.

The Compare Listing

The Compare Listing contains the image (or template) that acquired data is compared to during a comparison measurement. The boundaries of the image (or size of the template) can be controlled by using the channel masking and compare range functions described below. Any bits inside the image displayed as "X" have been set to don't compare bits.

The Difference Listing

The Difference Listing highlights the entire row with inverse video, if any, in the acquired data that differs from those in the compare image. In addition, when the base is hexadecimal, octal, or binary, the bit (or digit containing the bit) that differs from the compare image is underlined (see figures 12-2 and 12-3). If the base is inverse assembled symbols, the display does not change; however, the stop measurement functions still function.

To display the Compare Listing or the Difference Listing, place the cursor on the field directly to the right of **Show** in the upper left part of the display and press SELECT. The field will toggle between **Compare Listing** and **Difference Listing.**

The controls that roll the listing in all three menus, the normal State Listing, the Compare Listing, and the Difference Listing are synchronized unless the number of pre-trigger states differ between the Compare listing and the acquired data. This means that when you change the current row position in the Difference Listing, the logic analyzer automatically updates the current row in the acquired State Listing, Compare Listing and vice-versa.

If the three listings are synchronized and you re-acquire data, the Compare Listing may have a different number of pre-trigger states depending on the state trace trigger criteria. The Compare Listing can be resynchronized to the State and Difference Listings (if different) by entering the desired state (acquisition memory) location from the front-panel keypad.

This allows you to view corresponding areas of the two lists, to cross check the alignment, and analyze the bits that do not match.

Since time tags are not required to perform the compare, they do not appear in either the compare image or difference displays. However, correlation is possible since the displays are locked together.

To move between the State Listing and Compare Listing in the HP 1652B/53B, select the field directly to the right of your state machine's label, in the upper left most part of the screen and press SELECT. When this field is selected, a pop-up will appear. Select the State Listing field from this pop-up.

Creating a Compare Image

An initial compare image can be generated by copying acquired data into the compare image buffer. When you place the cursor on the **Copy Trace to Compare** field in the Compare Listing menu a pop-up appears with the options **Cancel and Continue.** If the **Continue** is selected, the contents of the acquisition data structure for the current machine are copied to the compare image buffer. The previous compare image is lost if it has not been saved to a disk. If you select **Cancel** the current compare image remains unchanged.

Bit Editing of the Compare Image

Bit editing allows you to modify the values of individual bits in the compare image or specify them as don't compare bits. The bit editing fields are located in the center of the Compare Listing display to the right of the listing number field (see figure 12-1). A bit editing field exists for every label in the display unless the label's base is ASCII or inverse assembled symbols. You can access any data in the Compare Listing by rolling the desired row vertically until it is located in the bit editing field for that label (column).

When you select one of the bit editing fields a pop-up appears in which you enter your desired pattern or don't compare for each bit.

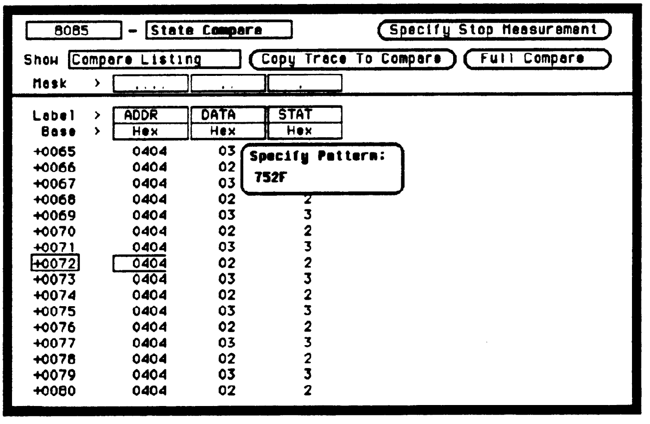


Figure 12-1. Bit Editing Fields

Masking Channels in the Compare Image

The channel masking function allows you to specify a bit, or bits in each label that you do not want compared. This causes the corresponding bits in all states to be ignored in the comparison. The compare data image itself remains unchanged on the display. The Mask fields are directly above the label and base fields at the top of both the Compare and Difference listings (see figure12-2). When you select one of these fields a pop-up appears in which you specify which channels are to be compared and which channels are to be masked. A "." (period) indicates a don't compare mask for that channel and an "*" (asterisk) indicates that channel is to be compared.

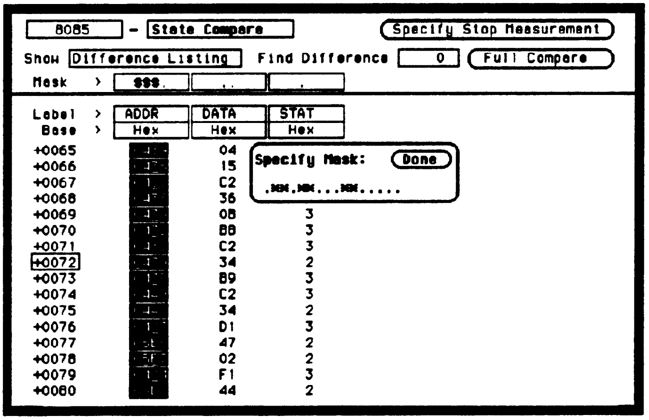


Figure 12-2. Bit Masking Fields

Specifying a Compare Range

The Compare Range function allows you to define a subset of the total number of states in the compare image to be used in the comparison. The range is specified by setting start and stop boundaries. Only bits in states (lines) on or between the boundaries are compared against the acquired data.

The Compare mode is accessed by selecting the **Full Compare/Partial Compare** field in either the Compare or Difference listing menus. When selected, a pop-up appears in which you select either the **Full** or **Partial** option. When you select the Partial option, fields for setting the start state and stop state values appear (see figure 12-3).

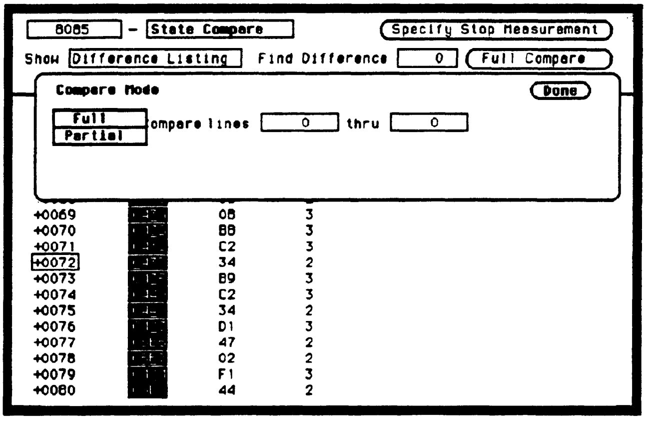


Figure 12-3. Compare Full/Compare Partial Field

Repetitive Comparisons with a Stop Condition

When you do a comparison in the repetitive trace mode, a stop condition may be specified. The stop condition is either **Stop Measurement** when Compare is **Equal** or **Not Equal.** In the case of Equal, bits in the compare image must match the corresponding bits in the acquired data image for the stop condition to be a true. In the case of Not Equal, a mismatch on a single bit will cause the stop condition to be true. When stop conditions are specified in two analyzers, both analyzers stop when the stop condition of either analyzer is satisfied. It is an OR function.

You access the stop measurement function by selecting the **Specify Stop Measurement** field in either the Compare or Difference Listing menus. When you select this field, the Stop Measurement Parameters pop-up appears (see figure 12-4). The first field in this pop-up, just to the right of Stop measurement contains either **Off**, **X-O** or **Compare.**

When this field is selected, a pop-up appears in which you select **Compare**. When you select the Compare option, you can access and select either the **Equal** or **Not Equal** option in the next field to the right.

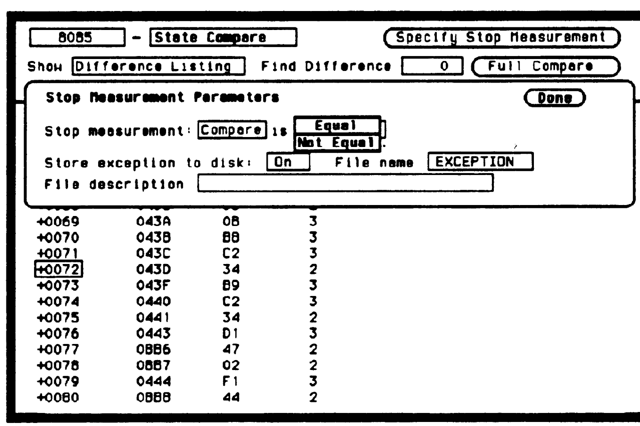


Figure 12-4. Specify Stop Measurement Field

Also available is **Store exception to disk** which allows you to specify a file on the disk that exceptions can be stored in. The default filename is EXCEPTION.

When the trace mode is repetitive and Store exception to disk is on, the following process takes place: data is acquired until the stop criteria is met, data acquistion will stop, data in the acquisition memory will be stored on the disk, and data acquisition will resume when the data is stored. This process continues until the disk is full. The data is stored in the same file name; however, the last three characters will automatically be replaced with a numerical serial number. For example, EXCEPTION will change to EXCEPT001 the second time memory is stored.

Note

You may also specify a stop measurement based on time between the X and O markers in the Compare or Difference Listing menus. This is available only when **Count** is set to **Time** in the State Trace menu. If the Stop Measurement is set to run until Compare Equal or Compare Not Equal in the Compare or Difference Listings, the Stop Measurement on time X to O will change to run until Compare Equal or Compare Not Equal in the other state display menus (i.e. State Listing).

Locating Mismatches in the Difference Listing

The **Find Difference** feature allows you to easily locate any patterns that did not match in the last comparison. Occurrences of differences are found in numerical ascending order from the start of the listing. The first occurrence of an error has the numerical value of one.

This feature is controlled by the **Find Difference**____ field in the Difference Listing menu. When you select this field an Integer Entry pop-up appears in which you enter a number indicating which difference you want to find. The listing is then scanned sequentially until the specified occurrence is found and rolled into view.

Saving Compare Images

When you save a logic analyzer configuration to a disk, the compare images for both state analyzers are saved with it. The compare data is compacted to conserve disk space. Likewise, when you load a configuration from disk, valid compare data will also be loaded.

13

State Chart Menu

Introduction

The State Chart Menu allows you to build X-Y plots of label activity using state data. The Y-axis always represents data values for a specified label. You can select whether the X-axis represents states (ie. rows in the State List) or the data values for another label. You can scale both the axes to selectively view data of interest. An accumulate mode is available that allows the chart display to build up over several runs. When **State** is selected for the X-axis, X & O markers are available which allows the current sample (state or time) relative to trace point and the corresponding Y-axis data value to be displayed. Marker placement is synchronized with the normal State Listing.

Accessing the State Chart Menu

The Chart menu is accessed from the State Listing menu. To access the Chart menu place the cursor on the field **State Listing** and press SELECT. A pop-up appears with the following options:

- State Listing.
- State Waveforms.
- State Chart.
- State Compare.

Place the cursor on State Chart and press SELECT. The pop-up will close and display the State Chart menu.

Selecting the Axes for the Chart

When using the State Chart display, you first select what data you want plotted on each axis. To assign the vertical axis label, position the cursor on the Y-axis **Label** field in the menu. This is the field just to the right of "**XY Chart of Label**". When selected, a pop up appears in which you select one of the labels that were defined in the State Format Specification Menu. The X-axis assignment field is just to the right of "**Versus**", and toggles between **State** and **Label** when selected. When label is selected, a third field appears to the right of **Label** that pops up when selected in which you select one of the defined state labels.

Scaling the Axes

Either axis of the X - Y chart can be scaled by using the associated vertical or horizontal **min** (minimum) or **max** (maximum) value fields. When selected, a **Specify Number** pop up appears in which you specify the actual minimum and maximum values that will be displayed on the chart.

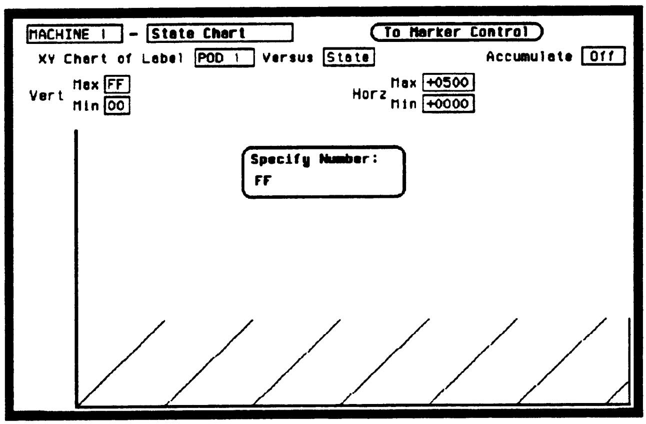


Figure 13-1. Axis Scaling Pop-up Menu

When **State** is selected for the X-axis, state acquisition memory locations are plotted on the X-axis. The minimum and maximum values can range from -1023 to +1023 depending on the trace point location. The minimum and maximum values for labels can range from 00000000H to FFFFFFFFH (0 to 2^{32-1}) regardless of axis, since labels are restricted to 32 bits.

The Label Value vs. States Chart

The Label Value versus State chart is a plot of label activity versus the memory location in which the label data is stored. The label value is plotted against successive analyzer memory locations. For example, in the following figure, label activity of POD 1 is plotted on the Y axis and the memory locations (State) are plotted on the X axis.

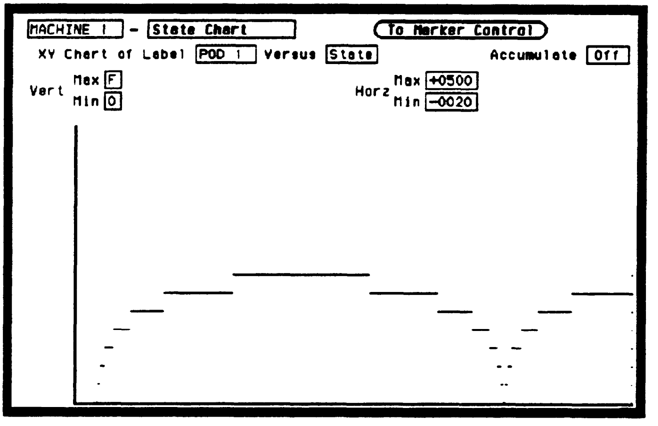


Figure 13-2. Label vs. State Chart

The Label Value vs. Label Value Chart

When labels are assigned to both axis, the chart shows how one label varies in relation to the other for a particular state trace record. Label values are always plotted in ascending order from the bottom to the top of the chart and in ascending order from left to right across the chart. Plotting a label against itself will result in a diagonal line from the lower left to upper right corner. X & O markers are disabled when operating in this mode.

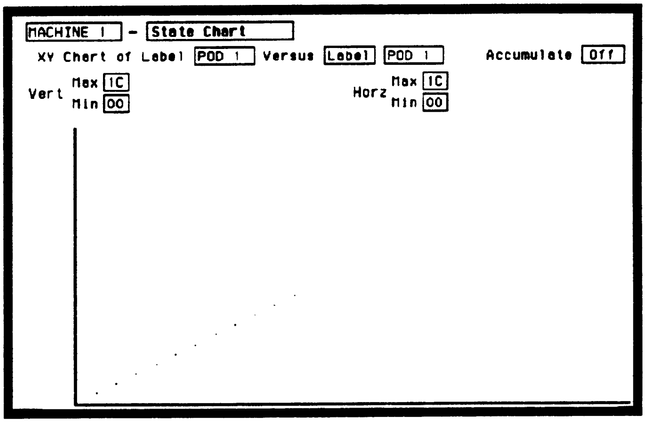


Figure 13-3. Label vs. Label Chart

X & O Markers and Readouts for Chart

When **State** is specified for the X-axis, X & O markers are available which can be moved horizontally. The markers are synchronized with the X and O markers in the normal State Listing.

To select the marker mode for Chart (if it is not presently displayed), place the cursor on the **To Marker Control** field and press SELECT. This field will toggle to **To Range Control** and the marker fields will be displayed (see figure 13-4).

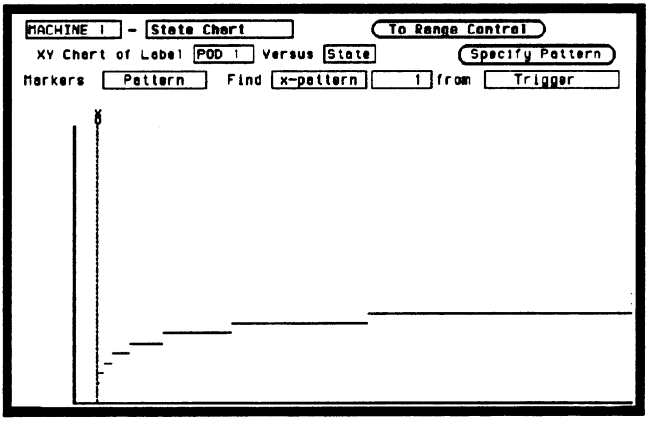


Figure 13-4. Marker Fields

When a marker is positioned in the State Chart menu, it is also positioned in the State Listing menu and vice-versa. The Chart marker operation is identical to the markers in the State Listing menu (see chapter 11).

Marker Options

The marker options in the State Chart menu depend on what **Count** is set to in the State Listing menu.

When Count is set to **Off**, the Chart markers can be set to:

- Off.
- Pattern.

When Count is set to **Time**, the Chart markers can be set to:

- Off.
- Pattern.
- Time.
- Statistics.

When Count is set to **States**, the Chart markers can be set to:

- Pattern.
- States.
- Off.

14

State Waveforms Menu

Introduction

The State Waveforms Menu allows you to view state data in the form of waveforms identified by label name and bit number. Up to 24 waveforms can be displayed simultaneously. Only state data from the current state machine can be displayed as waveforms in the State Waveforms menu.

The presentation and user interface is generally the same as the Timing Waveform menu, except the X-axis of the state waveform display represents only samples, or states instead of time (seconds). This is true regardless of whether **Count** (in the State Trace menu) is set to **Time** or **Off**. As a result, the horizontal axis of the display is scaled by States/Div and Delay in terms of samples from trigger. Marker features are the same as for State List in that Time or States will only be available when **Count** is set to **Time** or **States**. The Sample Rate display is not available in State Waveform even when markers are off.

Accessing the State Waveforms Menu

The State Waveforms menu is accessed from the State Listing menu. To access the State Waveforms menu place the cursor on the State Listing field and press SELECT. A pop-up appears with the following options:

- State Listing.
- State Waveforms.
- State Chart.
- State Compare.

Place the cursor on State Waveforms and press SELECT. The pop-up will close and display the State Waveforms menu.

Selecting a Waveform

You can display up to 24 waveforms on screen at one time. Each waveform is a representation of a predefined label. To select a waveform, place the cursor on a label name on the left side of the display and press SELECT. A pop-up appears in which you:

- Insert waveforms.
- Turn on waveforms.
- Modify waveforms (waveform labels).
- Turn off waveforms.
- Delete waveforms.

Just to the right of each label name is a two-digit number or the word "all." The number indicates which bit of the label the waveform represents; or, all the bits of the label when "all" is displayed (see figure 14-1).

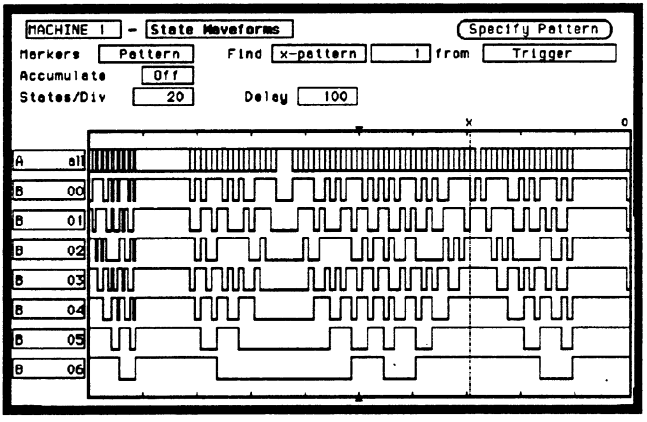


Figure 14-1. State Waveforms Menu

In the above figure, label A has "all" specified displaying all the bits overlaid in a single waveform. Label B however, has seven of its bits displayed individually (bits 0 through 6).

Replacing Waveforms

You can replace a currently displayed waveform (label) with another one of the predefined waveforms (labels). To replace one waveform with another, place the cursor on the waveform you wish to replace and press SELECT. A pop-up appears in which you select **Modify Waveform** as shown in the following figure.

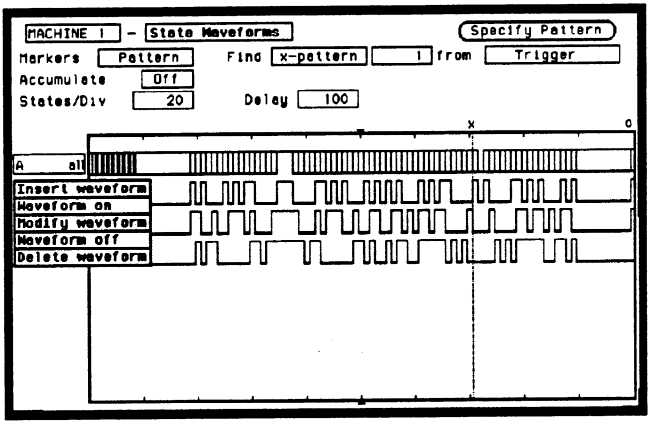


Figure 14-2. Waveform Selection Pop-up Menu

Another pop-up appears in which you select the waveform (label) you wish to display (see figure 14-3). When you place the cursor on the new waveform (label) and press SELECT the new waveform replaces the old waveform.

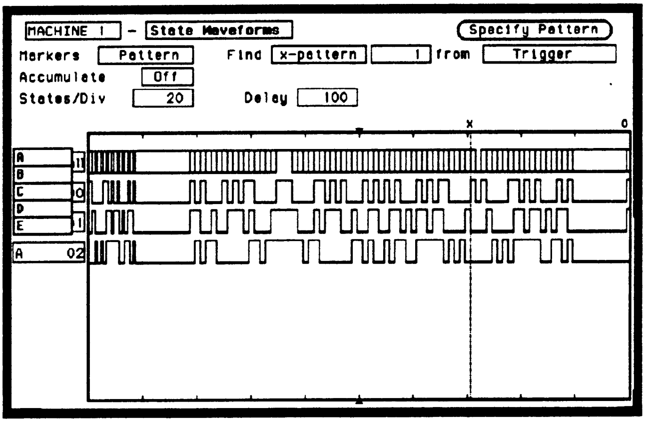


Figure 14-3. Available Waveforms Pop-up Menu

Deleting Waveforms

You can delete any of the currently displayed waveforms by placing the cursor on the waveform you wish to delete and pressing SELECT. When the pop-up appears place the cursor on **Delete waveform** and press SELECT.

Selecting States per Division

You can specify the states per division by placing the cursor on the field just to the right of **States/Div**, pressing SELECT, and either entering the number of states per division with the keypad or the knob. The range is from 1 to 1024 per division.

Delay from Trigger

You can specify the delay from trigger by specifying the number of states from the trigger. The delay will affect only the position of the State Waveforms display. It does not affect data acquisition. The minimum is − 1024 and the maximum is 1024 independent of trace position in the record. Delay is not limited to the window containing data.

State Waveform Display Features

The waveform display features of the State Waveform menu are the same as the Timing Waveform menu with regard to:

- Low levels (below threshold) are represented by darker line.
- Dotted lines representing the X and O markers.
- Inverted triangle representing the trigger point.
- Accumulate Mode.
- Graticule frame with 10 horizontal divisions.

X and O Markers for State Waveform

Markers can be placed on the waveform display by specifying the number of states from trigger in the case of the X marker or number of states from either the trigger or X marker in the case of the O marker.

Markers can be automatically placed on the waveform by searching for specific patterns assigned to each marker.

The X and O marker operation is identical to the marker operation in the Timing Waveform Menu.

15

State Analyzer Measurement Example

Introduction

In this chapter you learn how to use the state analyzer by setting up the logic analyzer to simulate a simple state measurement. Since you may not have the same test circuit available, we will give you the measurement results as actually measured by the logic analyzer,

The exercise in this chapter is organized in a task format. The tasks are in the same order you will most likely use them once you become experienced. The steps in this format are both numbered and lettered. The numbered steps state the step objective. The lettered steps explain how to accomplish each step objective. There is also an example of each menu after it has been properly set up.

How you use the steps depends on how much you remember from chapters 1 through 4 of the *Getting Started Guide*. If you can set up each menu by just looking at the menu picture, go ahead and do so. If you need a reminder of what steps to perform, follow the numbered steps. If you still need more information about "how," use the lettered steps.

To gain confidence using your logic analyzer, we recommend that you configure the menus as you follow the simulated measurement example up to the section "Acquiring the Data." From that section to the end, you will see the measurement results on the State Listing screen as if you had the real test circuit connected, and as if you had selected RUN.

Problem Solving with the State Analyzer

In this example assume you have designed a microprocessor controlled circuit. You have completed the hardware, and the software designer has completed the software and programmed the ROM (read-only memory). When you turn your circuit on for the first time, your circuit doesn't work properly. You have checked the power supply voltages and the system clock and they are working properly.

Since the circuit has never worked before, you and the software engineer aren't sure if it is a hardware or software problem. You need to do some testing to find a solution.

What Am I Going to Measure?

You decide to start where the microprocessor starts when power is applied. We will describe a 68000 microprocessor; however, every processor has similiar start-up routines.

When you power up a 68000 microprocessor, it is held in reset for a specific length of time before it starts doing anything to stabilize the power supplies. The time the microprocessor is held in reset ensures stable levels (states) on all the devices and buses in your circuit. When this reset period has ended, the 68000 performs a specific routine called "fetching the reset vector."

The first thing you check is the time the microprocessor is held in reset. You find the time is correct. The next thing to check is whether the microprocessor fetches the reset vector properly.

The steps of the 68000 reset vector fetch are:

1. Set the stack pointer to a location you specify, which is in ROM at address locations 0 and 2.

2. Find the first address location in memory where the microprocessor fetches its first instruction. This is also specified by you and stored in ROM at address locations 4 and 6.

What you decide to find out is:

1. What ROM address does the microprocessor look at for the location of the stack pointer, and what is the stack pointer location stored in ROM?

2. What ROM address does the microprocessor look at for the address where its first instruction is stored in ROM, and is the instruction correct?

3. Does the microprocessor then go to the address where its first instruction is stored?

4. Is the executable instruction stored in the first instruction location correct?

Your measurement, then, requires verification of the sequential addresses the microprocessor looks at, and of the data in ROM at these addresses. If the reset vector fetch is correct (in this example) you will see the following list of numbers in HEX (default base) when your measurement results are displayed.

```
+ 0000 000000 0000
+ 0001 000002 04FC
+ 0002 000004 0000
+ 0003 000006 8048
+ 0004 008048 3E7C
```

This list of numbers will be explained in detail later in this chapter in "The State Listing."

How Do I Configure the Logic Analyzer?

In order to make this state measurement, you must configure the logic analyzer as a state analyzer. By following these steps you will configure Analyzer 1 as the state analyzer.

If you are in the System Configuration menu you are in the right place to get started and you can start with step 2; otherwise, start with step 1.

1. Using the field in the upper left corner of the display, get the System Configuration menu on screen.

   a. Place the cursor on the field in the upper left corner of the display and press SELECT.

   b. Place the cursor on System and press SELECT.

2. In the System Configuration menu, change the Analyzer 1 type to State. If Analyzer 1 is already a state analyzer, go on to step 3.

Change to a State Analyzer

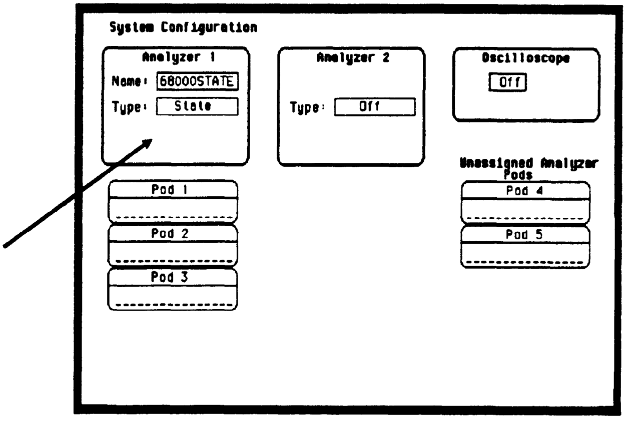


Figure 15-1. System Configuration Menu

   a. Place the cursor on the **Type:** ____ and press SELECT.

   b. Place the cursor on State and press SELECT.

3. Name Analyzer 1 68000STATE (optional).

   a. Place the cursor on the **Name:** ____ field of Analyzer 1 and press SELECT.

   b. With the Alpha Entry pop-up, change the name to 68000STATE.

4. Assign pods 1, 2, and 3 to the state analyzer.

   a. Place the cursor on the Pod 1 field and press SELECT.

   b. In the Pod 1 pop-up, place the cursor on Analyzer 1 and press SELECT.

   c. Repeat steps a and b for pods 2 and 3.

Connecting the Probes

At this point, if you had a target system with a 68000 microprocessor, you would connect the logic analyzer to your system. Since you will be assigning labels ADDR and DATA, you hook the probes to your system accordingly.

- Pod 1 probes 0 through 15 to the data bus lines D0 through D15.
- Pod 2 probes 0 through 15 to the address bus lines A0 through A15.
- Pod 3 probes 0 through 7 to the address bus lines A16 through A23.
- Pod 1, CLK (J clock) to the address strobe (LAS).

Activity Indicators

When the logic analyzer is connected and your target system is running, you will see **Activity Indicators** in the Pod 1, 2, and 3 fields of the System Configuration menu. This indicates which signal lines are transitioning.

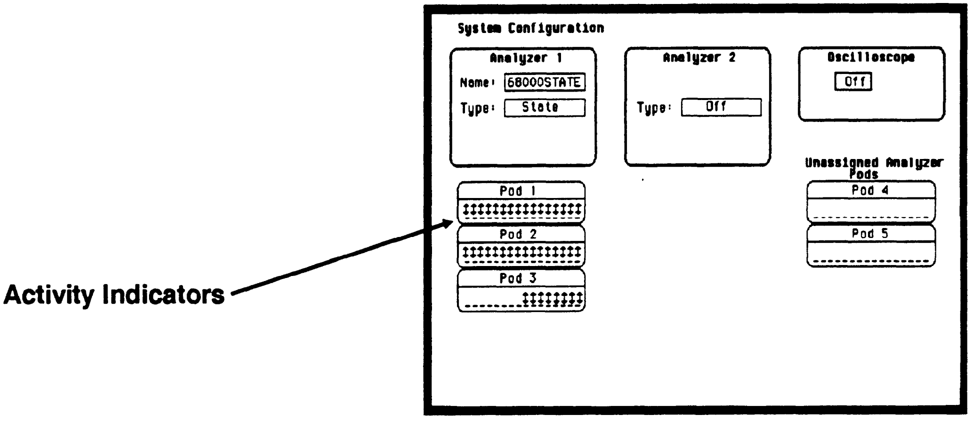


Figure 15-2. Activity Indicators

Configuring the State Analyzer

Now that you have configured the system, you are ready to configure the state analyzer. You will be:

- Creating two names (labels) for the input signals
- Assigning the channels connected to the input signals
- Specifying the State (J) clock
- Specifying a trigger condition

1. Display the State Format Specification menu.

   a. Press the FORMAT key on the front panel.

2. Name two labels, one ADDR and one DATA.

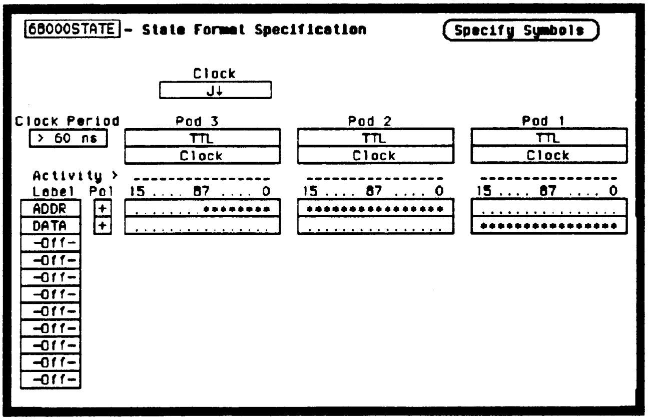


Figure 15-3. State Format Specification Menu

   a. Place the cursor on the top field in the label column and press SELECT.

   b. Place the cursor on Modify label and press SELECT.

c. With the Alpha Entry pop-up, change the name of the label to ADDR.

d. Name the second label DATA by repeating steps a through c.

3. Assign Pod 1 bits 0 through 15 to the label DATA.

a. Place the cursor on the bit assignment field below Pod 1 and to the right of DATA and press SELECT.

b. Any combination of bits may already be assigned to this pod; however, you will want all 16 bits assigned to the DATA label. The easiest way to assign is to press the CLEAR ENTRY key to un-assign any assigned bits before you start.

c. Place the cursor on the period under the 15 in the bit assignment pop-up and press SELECT. This will place an asterisk in the pop-up for bit 15, indicating Pod 1 bit 15 is now assigned to the DATA label. Repeat this procedure until all 16 bits have an asterisk under each bit number. Place the cursor on Done and press SELECT to close the pop-up.

d. Repeat step c for Pod 2 and the ADDR label to assign all 16 bits.

e. Repeat step c except you will assign the lower eight bits (0 - 7) of Pod 3 to the ADDR label.

Specifying the J Clock

If you remember from "What's a State Analyzer" in *Feeling Comfortable With Logic Analyzers*, the state analyzer samples the data under the control of an external clock, which is "synchronous" with your circuit under test. Therefore, you must specify which clock probe you will use for your measurement. In this exercise, you will use the J clock, which is accessible through pod 1.

1. Select the State Format Specification menu by pressing the FORMAT key.

2. Set the J Clock to sample on a negative-going edge.

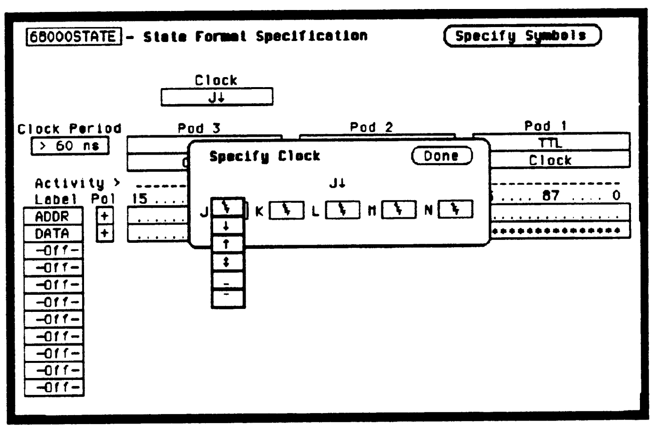


Figure 15-4. Specifying the J Clock

a. Place the cursor on the CLOCK field and press SELECT.

b. Place the cursor on the box just to the right of J in the pop-up (labeled OFF) and press SELECT.

c. Place the cursor on ↓ and press SELECT.

d. Place the cursor on Done and press SELECT.

Specifying a Trigger Condition

To capture the data and place the data of interest in the center of the display of the State Listing menu, you need to tell the state analyzer when to trigger. Since the first event of interest is address 0000, you need to tell the state analyzer to trigger when it detects address 0000 on the address bus.

1. Select the State Trace Specification menu by pressing the TRACE key.

2. Set the trigger so that the state analyzer triggers on address 0000. If the Trigger on option is not already a perform steps a through d. If the option is a skip to step e.

   a. Place the cursor on the 1 in the Sequence Levels field of the menu and press SELECT.

   b. Place the cursor on the field to the right of the Trigger on field

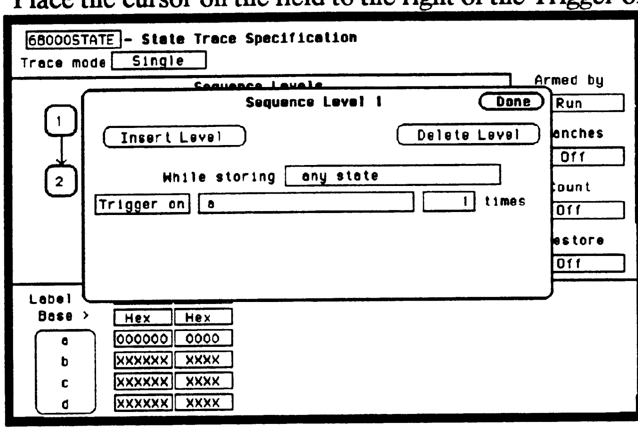


Figure 15-5. State Trace Specification Menu

and press SELECT. Another pop-up appears showing you a list of "trigger on" options. Options *a* through *h* are qualifiers. You can assign them a pattern for the trigger specification.

c. Place the cursor on the a option and press SELECT.

d. Place the cursor on Done in the Sequence Levels pop-up and press SELECT.

e. Place the cursor on the field to the right of the a under the label ADDR and press SELECT.

f. With the keypad, press 0 (zero) until there are all zeros in the Specify Pattern: pop-up and then press SELECT.

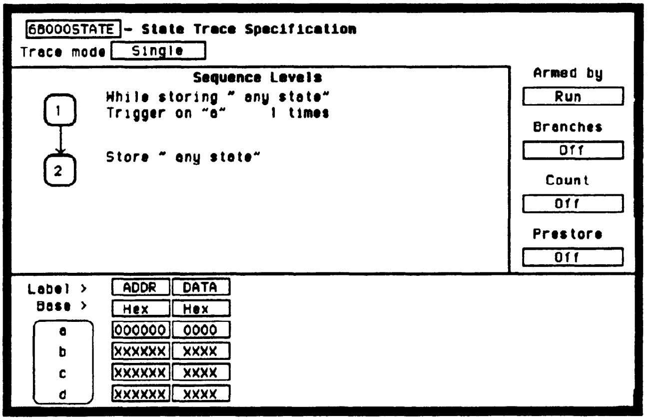


Figure 15-6. State Trace Specification

Your trigger specification now states: "While storing anystate trigger on "a" once and then store anystate."

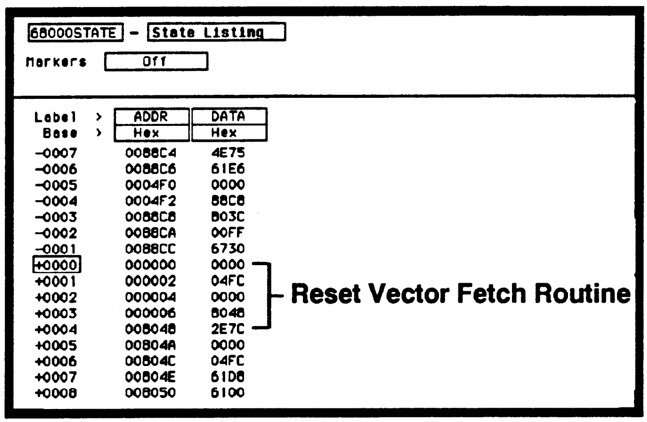
When the state analyzer is connected to your circuit and is acquiring data, it continuously stores until it sees 0000 on the address bus, then it will store anystate until the analyzer memory is filled.

Acquiring the Data

Since you want to capture the data when the microprocessor sends address 0000 on the bus after power-up, you press the RUN key to arm the state analyzer and then force a reset of your circuit. When the reset cycle ends, the microprocessor should send address 0000 trigger the state analyzer and switch the display to the State Listing menu.

Note

From this point in the exercise unto the end, we will give you the measurement results. This way, you will not have to obtain and use an identical circuit.



Label >	ADDR	DATA
Base >	Hex	Hex
-0007	0088C4	4E75
-0006	0088C6	61E6
-0005	0004F0	0000
-0004	0004F2	88C8
-0003	0088C8	803C
-0002	0088CA	00FF
-0001	0088CC	6730
+0000	000000	0000
+0001	000002	04FC
+0002	000004	0000
+0003	000006	8048
+0004	008048	2E7C
+0005	00804A	0000
+0006	00804C	04FC
+0007	00804E	61D8
+0008	008050	6100

Figure 15-7. Reset Vector Fetch Routine

The State Listing

The state listing displays three columns of numbers as shown:

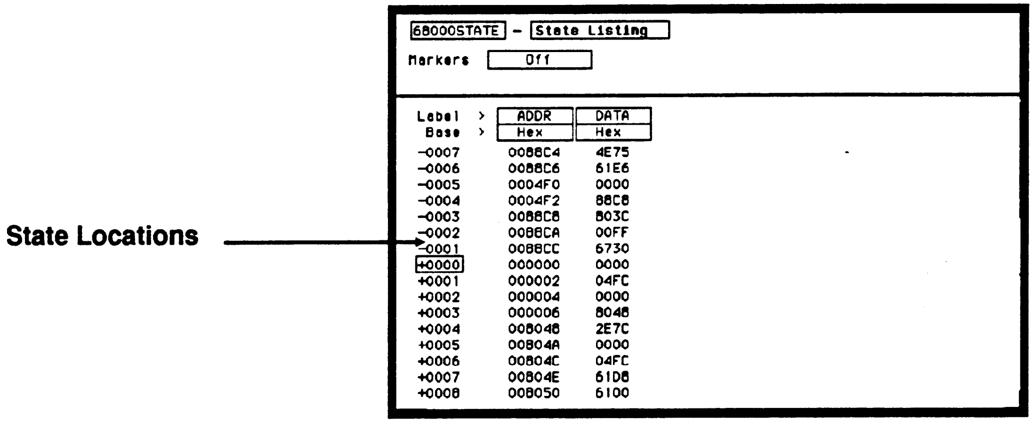


Figure 15-8. State Locations

The first column of numbers are the state line number locations as they relate to the trigger point. The trigger state is on line + 0000 in the vertical center of the list area. The negative numbers indicate states occurring before the trigger and the positive numbers indicate states occurring after the trigger.

The second column of numbers are the states (listed in HEX) the state analyzer sees on the address bus. This column is labeled ADDR.

The third column of numbers are the states (listed in HEX) the state analyzer sees on the data bus. This column is labeled DATA.

Finding the Answer

Your answer is now found in the listing of states + 0000 through + 0004.

The 68000 always reads address locations 0, 2, 4, and 6 to find the stack pointer location and memory location for the instruction it fetches after power-up. The 68000 uses two words for each of the locations that it is looking for, a high word and a low word. When the software designer programs the ROM, he must put the stack pointer location at address locations 0 and 2. 0 is the high word location and 2 is the low word location. Similarly, the high word of the instruction fetch location must be in address location 4 and the low word in location 6.

Since the software design calls for the reset vector to set the stack pointer to 04FC and read memory address location 8048 for its first instruction fetch, you are interested in what is on both the address bus and the data bus in states 0 through 3.

The state listing below lists the codes reset vector search, in states 0 through 3 and the correct first microprocessor instruction in state 4.

+ 0000 000000 0000
+ 0001 000002 04FC
+ 0002 000004 0000
+ 0003 000006 8048
+ 0004 008048 3E7C

You see that states 0 and 1 do contain address locations 0 and 2 under the ADDR label, indicating the microprocessor did look at the correct locations for the stack pointer data. You also see that the data contained in these ROM locations are 0000 and 04FC, which are correct.

You then look at states 2 and 3. You see that the next two address locations are 4 and 6, which is correct, and the data found at these locations is 0000 and 8048, which is also correct.

So far you have verified that the microprocessor has correctly performed the reset vector search. The next thing you must verify is whether the microprocessor addresses the correct location in ROM that it was instructed to address in state 4 and whether the data is correct in this ROM location. From the listing on your machine, you see that the address in state 4 is 008048, which is correct, but the instruction found in this location is 2E7C, which is not correct. You have found your problem: incorrect data stored in ROM for the microprocessor's first instruction.

+ 0000 000000 0000 (high word of stack pointer location)
+ 0001 000002 04FC (low word of stack pointer location)
+ 0002 000004 0000 (high word of instruction fetch location)
+ 0003 000006 8048 (low word of instuction fetch location)
+ 0004 008048 2E7C (first microprocessor instruction)

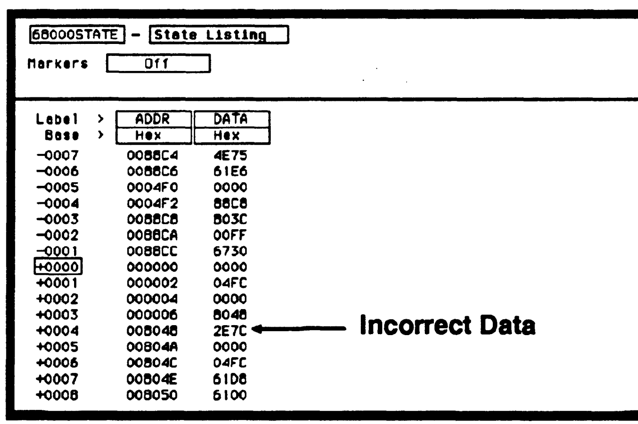


Figure 15-9. Incorrect Data

Summary

You have just learned how to make a simple state measurement with the HP 1652B Logic Analyzer. You have:

- specified a state analyzer
- learned which probes to connect
- assigned pods 1, 2, and 3
- assigned labels
- assigned bits
- specified the J clock
- specified a trigger condition
- acquired the data
- interpreted the state listing

You have seen how easy it is to use the state analyzer to capture the data on the address and data buses. You can use this same technique to capture and display related data on the microprocessor status control, and various strobe lines. You are not limited to using this technique on microprocessors. You can use this technique any time you need to capture data on multiple lines and need to sample the data relative to a system clock.

Chapter 21 shows you how to use the logic analyzer as an interactive timing and state analyzer. You will see a simple measurement that shows you both timing waveforms and state listings and how they are correlated.

If you have an HP 1653B, you do not have enough channels to simultaneously capture all the data for a 68000. But, since you probably aren't working with 16-bit microprocessors, this example is still valuable because it shows you how to make the same kind of measurement on an eight-bit microprocessor.

Index

E

F

Q

R

S

T

U

V

W

X

Your Comments Please

HP 1652B/HP 1653B

Your comments assist us in meeting your needs better. Please complete this questionnaire and return it to us. Feel free to add any additional comments that you might have. All comments and suggestions become the property of Hewlett-Packard. Omit any questions that you feel would be proprietary.

	Yes	No
1. Did you receive your product when expected?	[ ]	[ ]
2. Were you satisfied with the operation of the instrument at turn-on?	[ ]	[ ]

3. Were the proper accessories supplied with your product?

   If not, what was missing?

   Probes [ ]   Manual(s) [ ]   Other ______________________________

4. What measurements will this instrument be used to make?______________________________

   ____

5. How will the instrument be controlled?

   Front Panel [ ]   HP-IB [ ]   RS-232C [ ]   Controller Type______________________________

6. What do you like most about the instrument? ______________________________

   ____

7. What would you like to see changed or improved?______________________________

8. Which manuals have you used?

   [ ] Getting Started Guide
   [ ] Front-Panel Reference
   [ ] Programming Reference
   [ ] Service Manual

9. Please rate the manuals on the following:

   4 = Excellent   3 = Good   2 = Adequate   1 = Poor

   [ ] Breadth and depth of information
   [ ] Ability to easily find information
   [ ] Ability to understand and apply the information provided in the manual

   Please explain:______________________________

10. What is your experience with logic analyzers?

   [ ] No previous experience
   [ ] Less than 1 year experience
   [ ] More than 1 year's experience on one model
   [ ] More than 1 year's experience on several models

Name ______________________________ Company ______________________________

Address ______________________________ Zip Code ______________________________

Phone ______________________________ Instrument Serial # ______________________________

THANK YOU FOR YOUR HELP

NO POSTAGE NECESSARY IF MAILED IN U.S.A.

Your cooperation in completing and returning this form
will be greatly appreciated. Thank you.

FOLD HERE

HEWLETT
PACKARD

NO POSTAGE
NECESSARY
IF MAILED
IN THE
UNITED STATES

BUSINESS REPLY CARD

FIRST CLASS PERMIT NO. 1303 COLORADO SPRINGS, COLORADO

POSTAGE WILL BE PAID BY ADDRESSEE

HEWLETT-PACKARD
COLORADO SPRINGS DIVISION
ATTN: PUBLICATIONS DEPT.
P.O. BOX 2197
COLORADO SPRINGS, COLORADO 80901-9959

HEWLETT
PACKARD

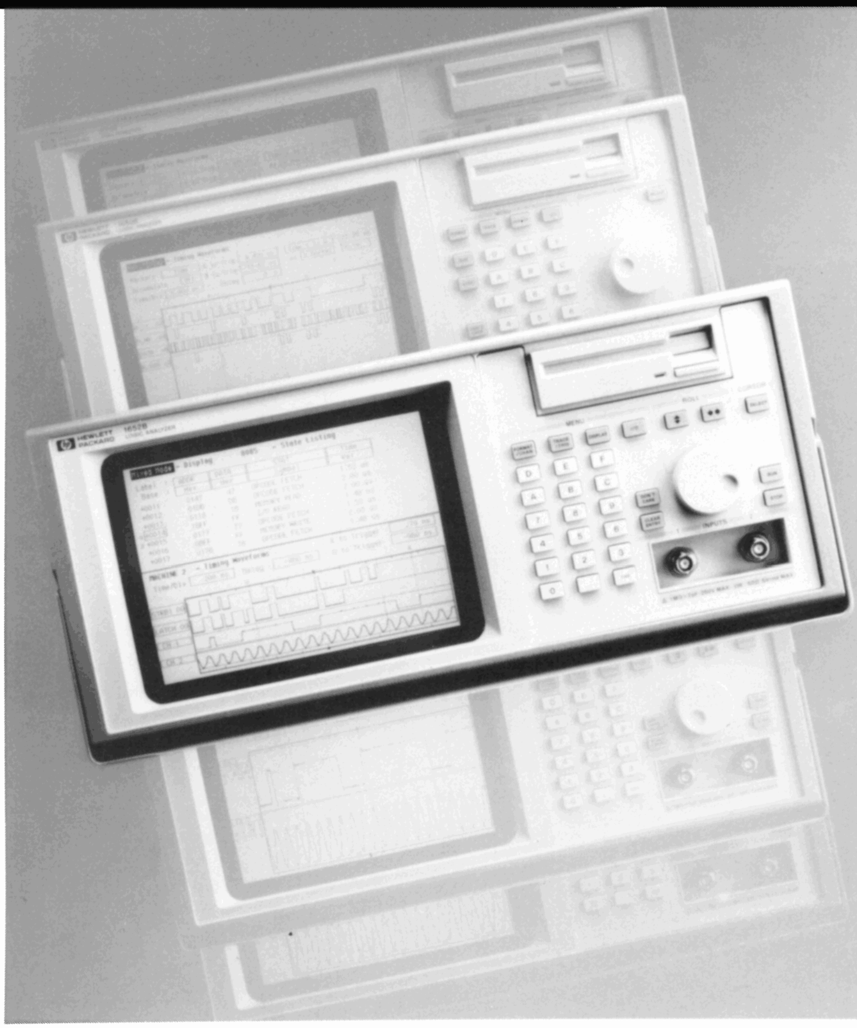
Printed in U.S.A.

HP 1652B/1653B Logic Analyzers
Front-Panel Operation Reference
Volume 1 of 2

HEWLETT
PACKARD

HP 1652B/HP 1653B Logic Analyzers

Front-Panel Operation Reference
Volume 2 of 2



HEWLETT PACKARD

Front-Panel Operation Reference

Volume 2 of 2

HP 1652B/HP 1653B Logic Analyzers

HEWLETT PACKARD

Manual Set Part Number 01652-90902

Printed in the U.S.A. November 1989

Contents Volume 2

Chapter 16: The Timing Analyzer

Chapter 17: Timing Format Specification Menu

Chapter 18: Timing Trace Specification Menu

Chapter 19: Timing Waveforms Menu

Chapter 20: Timing Analyzer Measurement Example

Chapter 21: Timing/State Measurement Example

Chapter 22: The Oscilloscope

Chapter 23: Channel Menu

Chapter 24: Trigger Menu

Chapter 25: Waveforms Menu

Chapter 26: Mixed Mode Displays

Chapter 27: Timing/State/Oscilloscope Measurement Example

Appendix A: Microprocessor Specific Measurements

Appendix B: Automatic Measurement Algorithms

Appendix C: Error Messages

Appendix D: Installation, Maintenance and Calibration

Appendix E: Operator Self Tests

Appendix F: Specifications and Operating Characteristics

Index

16

The Timing Analyzer

Introduction

This chapter introduces the timing analyzer and contains the timing analyzer menu maps.

Chapters 17 through 19 explain each of the Timing Analyzer menus as follows:

- Chapter 17 explains the Timing Format Menu.
- Chapter 18 explains the Timing Trace Menu.
- Chapter 19 explains the Timing Waveforms Menu.
- Chapter 20 gives you a basic Timing Analyzer Measurement example.
- Chapter 21 gives you a basic Timing/State Analyzer Measurement example.

The Timing Analyzer (An Overview)

The timing analyzer acquires data asynchronously using an internal sample clock. This asynchronous data acquisition technique is similar to a digitizing oscilloscope. The acquired data is displayed in the form of one or more waveforms. The timing waveforms differ from a digitizing oscilloscope in that the timing analyzer only stores and displays two levels (one above and one below threshold).

Timing Analyzer Menu Maps

The Timing Analyzer menu maps show you the fields and the available options of each field within the three menus. The menu maps will help you get an overview of each menu as well as provide you with a quick reference of what each menu contains.

Timing Format Menu Map

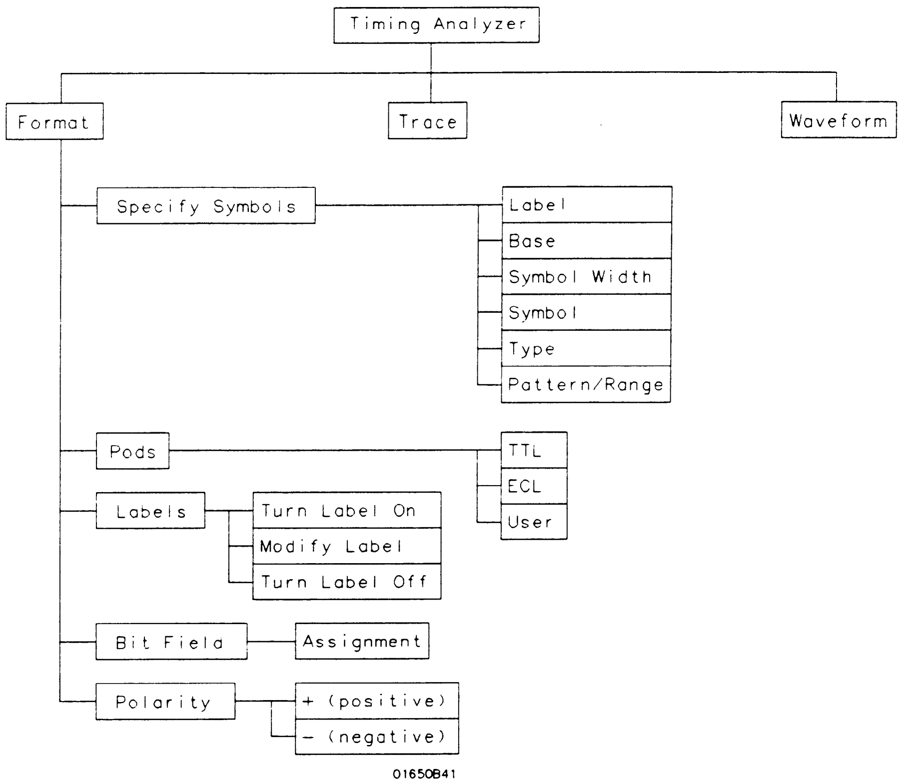


Figure 16-1. Timing Format Menu Map

Timing Trace Menu Map

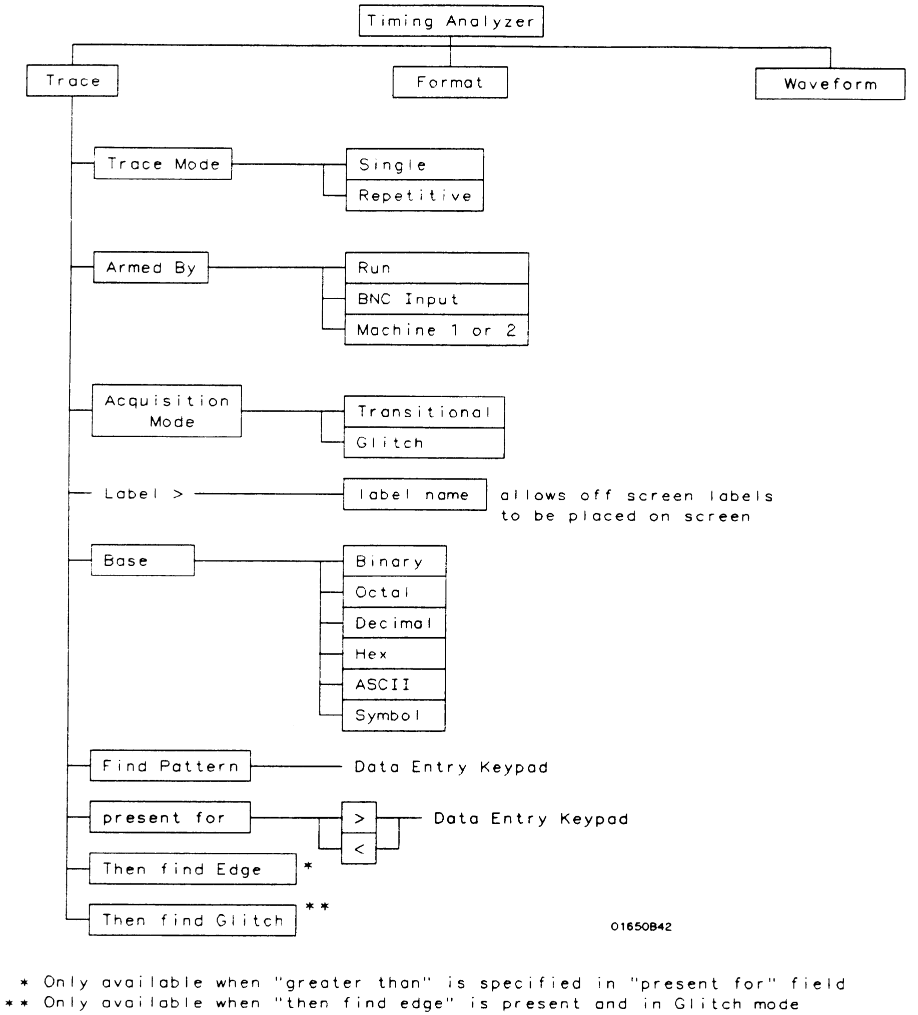


Figure 16-2. Timing Trace Menu Map

Timing Waveform Menu Map

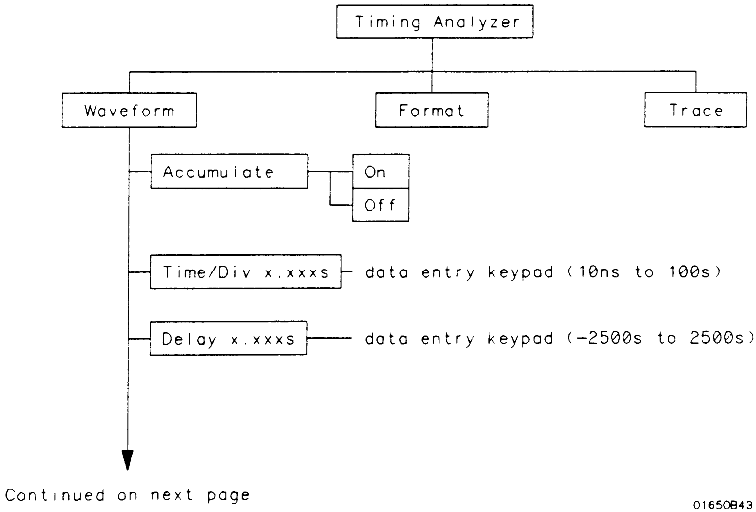


Figure 16-3. Timing Waveform Menu Map

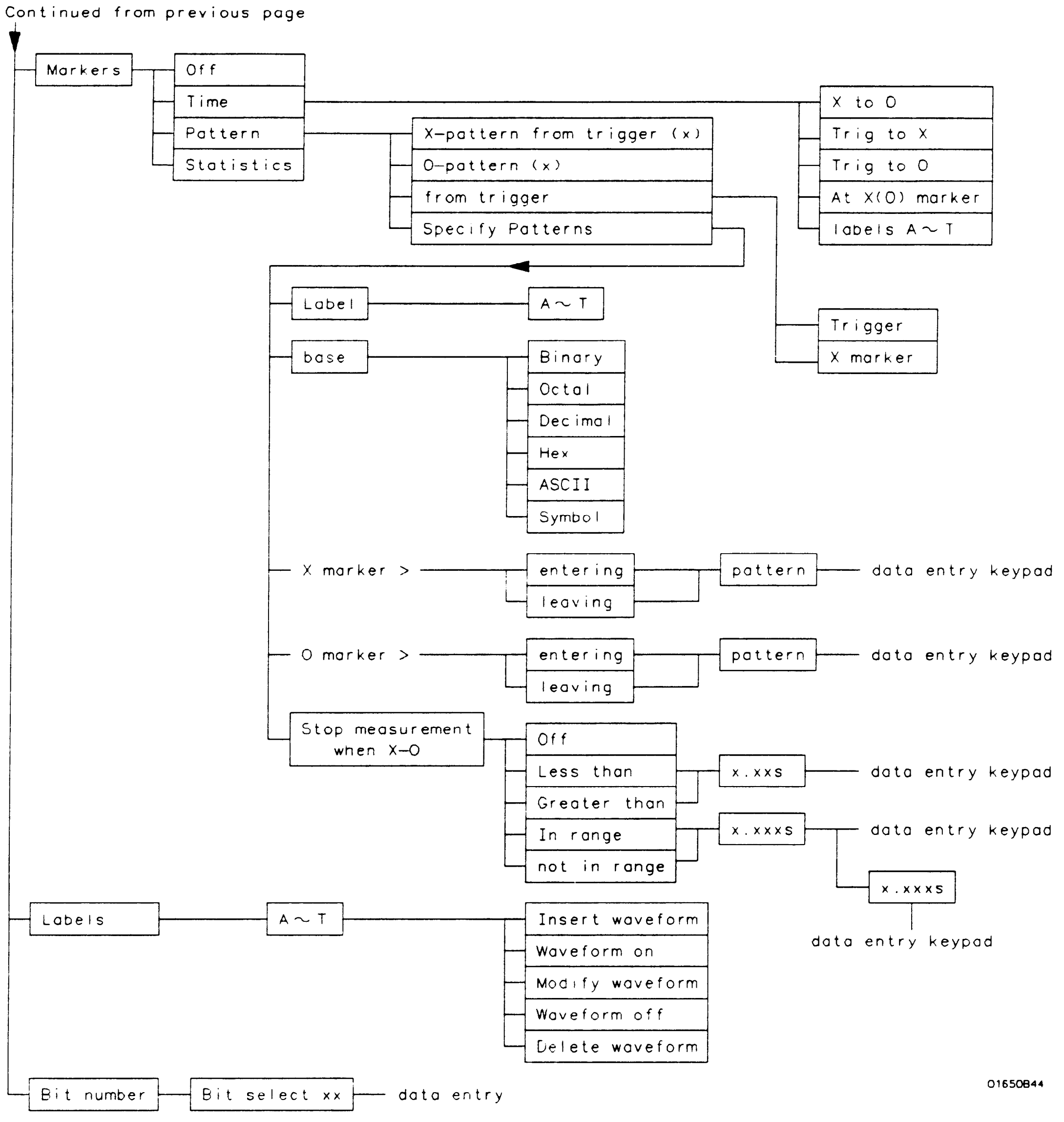


Figure 8-3. Timing Waveform Menu Map (Continued)

17

Timing Format Specification Menu

Introduction

This chapter describes the Timing Format Specification menu and all the pop-up menus that you will use on your timing analyzer. The purpose and function of each pop-up menu is explained in detail, and we have included many illustrations and examples to make the explanations clearer.

Accessing the Timing Format Specification Menu

The Timing Format Specification menu can be accessed by pressing the FORMAT key on the front panel. If the State Format Specification Menu is displayed when you press the FORMAT key, you will have to switch analyzers. This is not a problem, it merely indicates that the last action you performed in the System Configuration Menu was on the state analyzer.

Timing Format Specification Menu

The Timing Format Specification menu lets you configure the timing analyzer to group channels from your microprocessor into labels you assign for your measurements. You can set the threshold levels of the pods assigned to the analyzer, assign labels and channels, and specify symbols.

At power up, the logic analyzer is configured with a default setting. You can use this default setting to make a test measurement on the system under test. It can give you an idea of where to start your measurement. For an example of setting up configurations for the Timing analyzer, refer to the *Getting Started Guide* or "Timing Analyzer Measurement Example" in chapter 20 of this manual.

At power up the Timing Format Specification menu looks like that shown below:

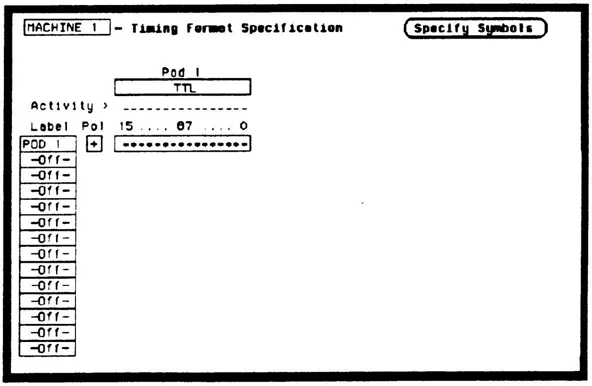


Figure 17-1. Timing Format Specification Menu

The Timing Format Specification menu for the HP 1653B is similar to that for the HP 1652B except that Pod 2 appears in the menu instead of Pod 5.

This menu shows only one pod assigned to each analyzer, which is the case at power up. Any number of pods can be assigned to one analyzer, from none to all five for the HP 1652B, and from none to two for the HP 1653B. In the Timing Format Specification menu, only three pods appear at a time in the display. To view any pods that are off screen, press the left/right ROLL key and rotate the KNOB. The pods are always positioned so that the lowest numbered pod is on the right and the highest numbered pod is on the left.

Timing Format Specification Menu Fields

Five types of fields present in the menu are as follows:

- Label.
- Polarity (Pol).
- Bit assignments.
- Pod threshold.
- Specify Symbols.

A portion of the menu that is not a field is the Activity Indicators display. The indicators appear under the active bits of each pod, next to "Activity." When the logic analyzer is connected to your target system and the system is running, you will see ↕ in the Activity Indicators display for each channel that has activity. These tell you that the signals on the channels are transitioning.

The fields in the Format menus are described in this following sections.

Label

The label column contains 20 Label fields that you can define. Of the 20 labels, the logic analyzer displays only 14 in the Timing Format Specification menu at one time. To view the labels that are off screen, press the up/down ROLL key and rotate the KNOB. The labels scroll up and down. To deactivate the scrolling, press the ROLL key again.

To access one of the Label fields, place the cursor on the field and press SELECT. You will see a pop-up menu like that shown below.

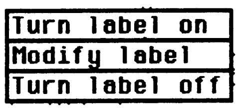


Figure 17-2. Label Pop-Up Menu

Turn Label On

Selecting this option turns the label on and gives it a default letter name. If you turned all the labels on they would be named A through T from top to bottom. When a label is turned on, the bit assignment fields for that label, appear to the right of the label.

Modify Label

If you want to change the name of a label, or want to turn a label on and give it a specific name, you would select the Modify label option. When you do, an Alpha Entry pop-up menu appears. You can use the pop-up menu and the keypad on the front panel to name the label. A label name can be a maximum of six characters.

Turn Label Off

Selecting this option turns the label off. When a label is turned off, the bit assignments are saved by the logic analyzer. This gives you the option of turning the label back on and still having the bit assignments if you need them. The waveforms are also saved.

You can give the same name to a label in the state analyzer as in the timing analyzer without causing an error. The logic analyzer distinguishes between them. An example of this appears in "Using the Timing/State Analyzer" in chapter 7 of the *Getting Started Guide*.

Polarity (Pol)

Each label has a polarity assigned to it. The default for all the labels is positive (+) polarity. You can change the polarity of a label by placing the cursor on the polarity field and pressing SELECT. This toggles the polarity between positive (+) and negative (–).

In the timing analyzer, negative polarity inverts the data.

Bit Assignment

The bit assignment fields allow you to assign bits (channels) to labels. Above each column of bit assignment fields is a line that tells you the bit numbers from 0 to 15, with the left bit numbered 15 and the right bit numbered 0. This line helps you know exactly which bits you are assigning.

The convention for bit assignment is:

* (asterisk) indicates assigned bit
. (period) indicates unassigned bit

At power up the 16 bits of Pod 1 are assigned to the timing analyzer and the 16 bits of Pod 5 are assigned to the state analyzer. To change a bit assignment configuration, place the cursor on a bit assignment field and press SELECT. You will see the following pop-up menu.

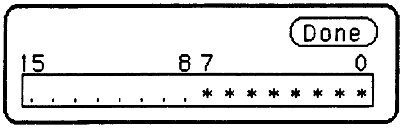


Figure 17-3. Bit Assignment Pop-Up Menu

Use the KNOB to move the cursor to an asterisk or a period and press SELECT. The bit assignment toggles to the opposite state of what it was before. When the bits (channels) are assigned as desired place the cursor on Done and press SELECT. This closes the pop-up and displays the new bit assignment.

Assigning one channel per label may be handy in some applications. This is illustrated in "Using the Timing/State Analyzer" in chapter 7 of the *Getting Started Guide* and chapter 21 of this manual. In addition, you can assign a channel to more than one label.

Labels may have from 1 to 32 channels assigned to them. If you try to assign more than 32 channels to a label, the logic analyzer will beep, indicating an error, and a message will appear at the top of the screen telling you that 32 channels per label is the maximum.

Channels assigned to a label are numbered from right to left by the logic analyzer. The least significant assigned bit (LSB) on the far right is numbered 0, the next assigned bit is numbered 1, and so on. Since 32 channels can be assigned to one label at most, the highest number that can be given to a channel is 31. Although labels can contain split fields, assigned channels are always numbered consecutively within a label as shown in figure 17-4.

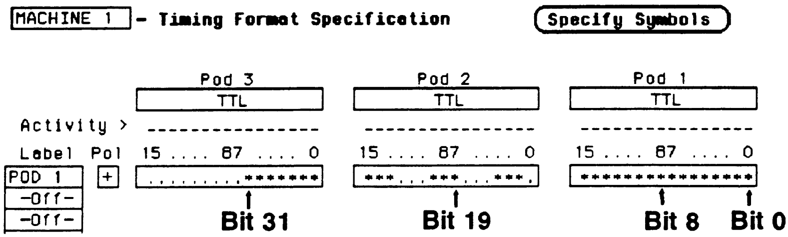


Figure 17-4. Numbering of Assigned Bits

Pod Threshold

Each pod has a threshold level assigned to it. For the HP 1653B Logic Analyzer, threshold levels may be defined for Pods 1 and 2 individually. For the HP 1652B Logic Analyzer, threshold levels may be defined for Pods 1, 2 and 3 individually, and one threshold for Pods 4 and 5. It does not matter if Pods 4 and 5 are assigned to different analyzers. Changing the threshold of one will change the threshold of the other.

If you place the cursor on one of the pod threshold fields and press SELECT, you will see the following pop-up menu.

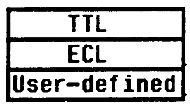


Figure 17-5. Pod Threshold Pop-Up Menu

TTL sets the threshold at + 1.6 volts, and ECL sets the threshold at −1.3 volts.

The User-defined option lets you set the threshold to a specific voltage between −9.9 V and + 9.9 V. If you select this option you will see a Numeric Entry pop-up menu as shown.

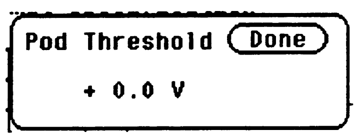


Figure 17-6. User-Defined Numeric Entry Pop-Up Menu

You can change the value in the pop-up either with the keypad on the front panel or with the KNOB, which you rotate until you get the desired voltage. When the correct voltage is displayed, press SELECT. The pop-up will close and your new threshold will be placed in the pod threshold field.

Specify Symbols Menu

The **Specify Symbols** field differs from the other fields in the Timing Format Specification menu in that it displays a complete menu instead of a pop-up.

The logic analyzer supplies Timing and State Symbol Tables in which you can define a mnemonic for a specific bit pattern of a label. When measurements are made by the timing analyzer, the mnemonic is displayed where the bit pattern occurs if the Symbol base is selected.

It is possible for you to specify up to 200 symbols in the logic analyzer. If you have only one of the internal analyzers on, all 200 symbols can be defined in it. If both analyzers are on, the 200 symbols are split between the two. For example, analyzer 1 may have 150, leaving 50 available for analyzer 2.

To access the Symbol Table in the Timing Format Specification menu, place the cursor on the **Specify Symbols** field and press SELECT. You will see a new menu as shown in figure 17-7. This is the default setting for the Symbol Table in both the timing and state analyzers.

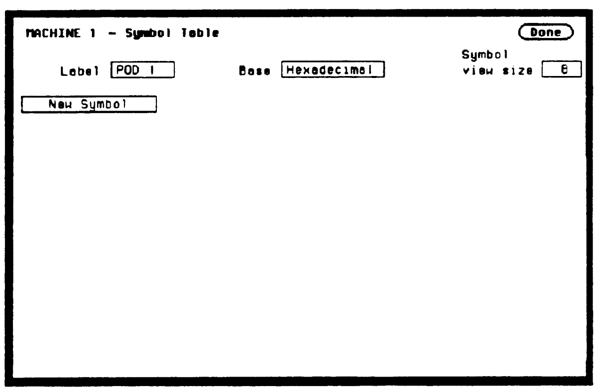


Figure 17-7. Symbol Table Menu

Specify Symbols Menu Fields

There are four fields in the Symbol Table menu. They are:

- Label
- Base
- Symbol view size
- Symbol name

Label

The Label field identifies the label for which you are specifying symbols. If you select this field, you will get a pop-up that lists all the labels turned on for that analyzer.

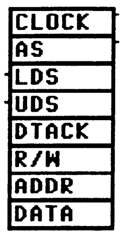


Figure 17-8. Label Pop-Up Menu

Each label has a separate symbol table. This allows you to give the same name to symbols defined under different labels. In the Label pop-up select the label for which you wish to specify symbols.

Base

The Base field tells you the numeric base in which the pattern will be specified. The base you choose here will affect the **Find Pattern** field of the Timing Trace Specification menu. This is covered later in this chapter.

To change the base, place the cursor on the Base field and press SELECT. You will see the following pop-up menu.

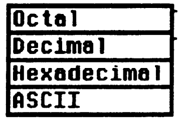


Figure 17-9. Base Pop-Up Menu

If more than 20 channels are assigned to a label, the Binary option is not offered in the pop-up. The reason for this is that when a symbol is specified as a range, there is only enough room for 20 bits to be displayed on the screen.

Decide which base you want to work in and choose that option from the numeric Base pop-up menu.

If you choose the ASCII option, you can see what ASCII characters the patterns and ranges defined by your symbols represent. ASCII characters represented by the decimal numbers 0 to 127 (hex 00 to 7F) are offered on your logic analyzer. Specifying patterns and ranges for symbols is discussed in the next section.

Note

You cannot specify a pattern or range when the base is ASCII. First define the pattern or range in one of the other bases, then switch to ASCII to see the ASCII characters.

Symbol View Size

The Symbol view size field lets you specify how many characters of the symbol name will be displayed when the symbol is referenced in the Timing Trace Specification menu and the Timing Waveforms menu. Selecting this field gives you the following pop-up.

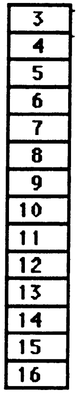


Figure 17-10. Symbol View Size Pop-Up Menu

You can have the logic analyzer display from 3 to all 16 of the characters in the symbol name. For more information see "Timing Trace Specification Menu" in Chapter 18 and the "Timing Waveforms Menu" in Chapter 19.

Symbol Name

When you first access the Symbol Table, there are no symbols specified. The symbol name field reads "New Symbol." If you select this field, you will see an Alpha Entry pop-up menu on the display. Use the pop-up menu and the keypad on the front panel to enter the name of your symbol. A maximum of 16 characters can be used in a symbol name. When you select the Done field in the Alpha Entry pop-up menu the name that appears in the symbol name field is assigned and two more fields appear in the display.

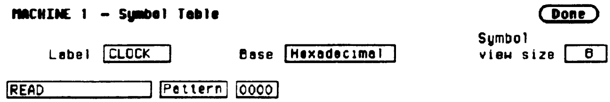


Figure 17-11. Symbol Defined as a Pattern

The first of these fields defines the symbol as either a Pattern or a Range. If you place the cursor on this field and press SELECT, it will toggle between **Pattern** and **Range**.

When the symbol is defined as a pattern, one field appears to specify what the pattern is. Selecting this field gives you a pop-up with which you can specify the pattern. Use the keypad and the DON'T CARE key on the front panel to enter the pattern. Be sure to enter the pattern in the numeric base that you specified in the **Base** field.

Figure 17-12. Specify Pattern Pop-Up

If the symbol is defined as a range, two fields appear in which you specify the upper and lower boundaries of the range.

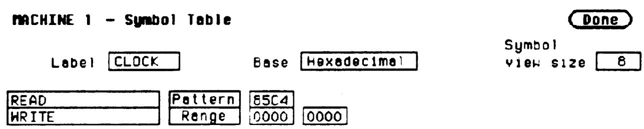


Figure 17-13. Symbol Defined as a Range

Selecting either of these fields gives you a pop-up with which you can specify the boundary of the range.

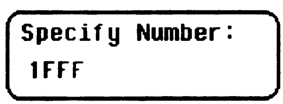


Figure 17-14. Specify Range Pop-Up

You can specify ranges that overlap or are nested within each other. Don't cares are not allowed.

To add more symbols to your symbol table, place the cursor on the last symbol defined and press SELECT. A pop-up menu appears as shown.

Modify symbol
Insert new symbol
Delete symbol

Figure 17-15. Symbol Pop-Up Menu

The first option in the pop-up is Modify symbol. If you select this option, you will see an Alpha Entry pop-up menu with which you can change the name of the symbol.

The second option in the pop-up is Insert new symbol. It allows you to specify another symbol. When you select it, you will see an Alpha Entry pop-up menu. Use the menu and the keypad on the front panel to enter the name of your new symbol. When you select Done, your new symbol will appear in the Symbol Table. The third option in the pop-up is Delete symbol. If you select this option, the symbol will be deleted from the Symbol Table.

Leaving the Symbol Table Menu

When you have specified all your symbols, you can leave the Symbol Table menu in one of two ways. One method is to place the cursor on the Done field and press SELECT. This puts you back in the Format Specification menu that you were in before entering the Symbol Table. The other method is to press the FORMAT, TRACE, or DISPLAY keys on the front panel to get you into the respective menu.

18

Timing Trace Specification Menu

Introduction

This chapter describes Timing Trace Specification menu and all the pop-up menus that you will use on your timing analyzer. The purpose and function of each pop-up menu is explained in detail, and we have included many illustrations and examples to make the explanations clearer.

Accessing the Timing Trace Specification Menu

The Timing Trace Specification menu can be accessed by pressing the TRACE key on the front panel. If the State Trace Specification menu is displayed when you press the TRACE key, you will have to switch analyzers. This is not a problem, it merely indicates that the last action you performed in the System Configuration Menu was on the state analyzer.

Timing Trace Specification Menu

The Trace Specification menus allow you to configure the logic analyzer to capture only the data of interest in your measurement. In the timing analyzer you can configure the analyzer to trigger on specific patterns, edges, or glitches. The Timing Trace Specification menu lets you specify the trigger point for the logic analyzer to start capturing data and the manner in which the analyzer will capture data. You configure the timing analyzer to find a pattern first and then a transition in the signal or signals.

At power up, the logic analyzer is configured with a default setting. You can use this default setting to make a test measurement on the system under test. It can give you an idea of where to start your measurement. For an example on setting up configurations for the Timing analyzer, refer to the *Getting Started Guide* or "Timing Analyzer Measurement Example" in Chapter 20 of this manual.

At power up the Timing Trace menu looks like that shown below.

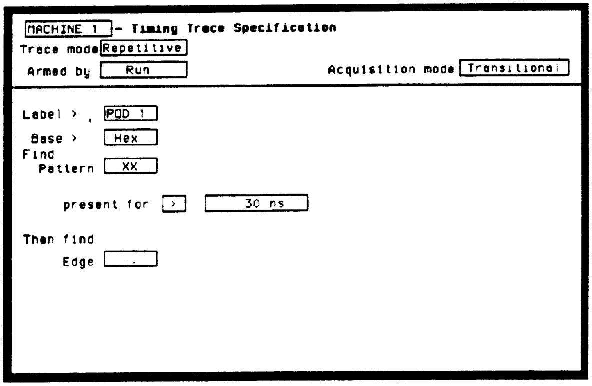


Figure 18-1. Timing Trace Specification Menu

The menu is divided into two sections by a horizontal line. The top section contains the fields that you use to specify the data acquisition. The bottom section contains the fields for setting the trigger point.

Timing Trace Specification Menu Fields

The fields in the Timing Trace Specification menu are as follows:

- Trace mode.
- Armed by.
- Acquisition mode.
- Label.
- Base.
- Find Pattern.
- Pattern Duration (present for ____).
- Then find Edge.

These fields are described in this chapter.

Trace Mode

With the **Trace mode** field you specify the mode in which the timing analyzer will trace. You have two choices for Trace mode: Single and Repetitive. If you place the cursor on the field and press SELECT, the field toggles from one mode to the other.

Single Trace mode acquires data once per trace. Repetitive Trace mode repeats single acquisitions until the STOP key on the front panel is pressed, or if Stop measurement has been selected and the stop measurement condition has been met.

If both analyzers are on, only one Trace mode can be specified. Specifying one trace mode for one analyzer sets the same trace mode for the other analyzer.

Armed By

The **Armed by** field lets you specify how your timing analyzer is to be armed. The analyzer can be armed by the RUN key, the other analyzer, the scope or an external instrument through the BNC Input port.

When you select the **Armed by** field, a pop-up menu appears like that shown below. Use this menu to select the arming option for your analyzer.

Armed by

Run
BNC Input
Machine 1
Scope

Figure18-2. Armed By Pop-Up Menu

Acquisition Mode

The **Acquisition mode** field allows you to specify the mode in which you want the timing analyzer to acquire data. You are given two choices for the mode of acquisition: Transitional and Glitch . If you place the cursor on this field and press SELECT, the field toggles from one mode to the other.

Transitional Acquisition Mode

When the logic analyzer is operating in the Transitional Acquisition mode, it samples the data at regular intervals, but it stores data in memory only on transitions in the signals. A time tag that is stored with each sample allows reconstruction of the samples in the Timing Waveforms display.

Transitional timing always samples at a rate of 100 MHz (10 ns/sample). This provides maximum timing resolution even in records that span long time windows. Time covered by a full memory acquisition varies with the number of pattern changes in the data. If there are many transitions, the data may end prior to the time window desired because the memory is full. However, a prestore qualification in your logic analyzer insures that data will be captured and displayed between the left side of the screen and the trigger point.

Figure 18-3 illustrates Transitional acquisition, comparing it to Traditional acquisition.

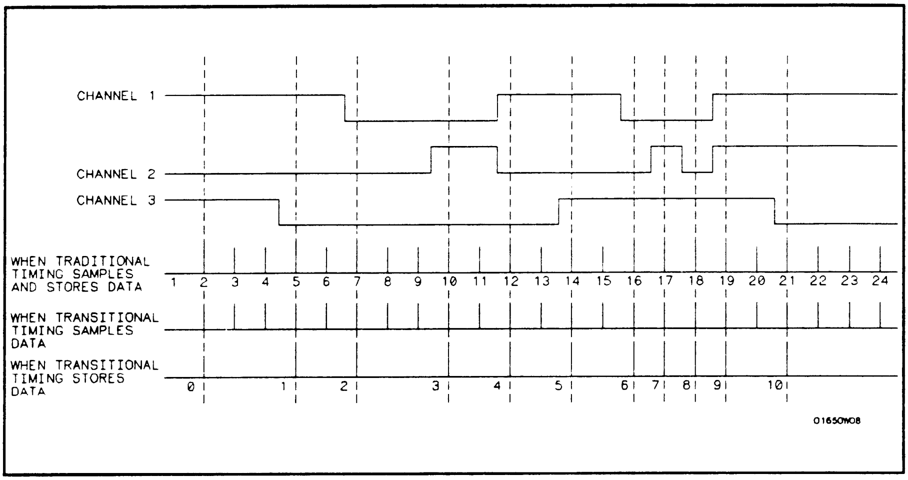


Figure 18-3. Transitional vs. Traditional Acquisition

Traditional timing samples and stores data at regular intervals. Transitional timing samples data at regular intervals but stores a sample only when there has been a transition on one or more of the channels. This makes it possible for Transitional timing to store more information in the same amount of memory.

Glitch Acquisition Mode

A glitch is defined as any transition that crosses logic threshold more than once between samples. It can be caused by capacitive coupling between traces, by power supply ripples, or a number of other events. Since a glitch can cause major problems in your system, you can use the Glitch mode to find it.

Your logic analyzer has the capability of triggering on a glitch and capturing all the data that occurred before it. The glitch must have a width of at least 5 ns at threshold in order for the analyzer to detect it.

If you want your timing analyzer to trigger on a glitch in the data, set the Acquisition mode to Glitch. This causes several changes in the analyzer. One change is that a field for glitch detection in each label is added to the Timing Trace Specification menu, as shown:

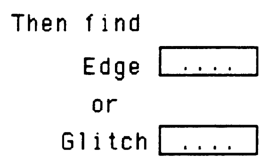


Figure 18-4. Glitch Specification Field

With these glitch detection fields you specify on which channel or channels you want the analyzer to look for a glitch. These fields are discussed in more detail in "Then Find Edge" later in this chapter.

Glitch Acquisition mode causes the storage memory to be cut in half from 1k to 512. Half the memory (512) is allocated for storing the data sample, and the other half for storing the second transition of a glitch in a sample. Every sample is stored.

The sample rate varies from 20 Hz to 50 MHz (50 Ms/sample to 20 ns/sample) and is automatically selected by the timing analyzer to insure complete data in the window of interest.

When your timing analyzer triggers on a glitch and displays the data, the glitch appears in the waveform display as shown below.

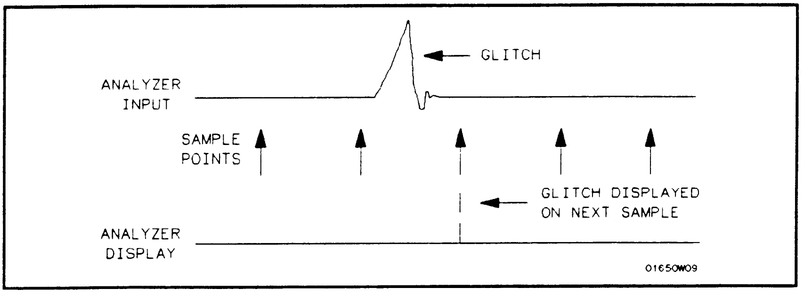


Figure 18-5. Glitch in Timing Waveform

Label

The **Label** fields contain the labels that you define in the Timing Format Specification menu. If there are more labels than can fit on screen, use the left/right ROLL key and the KNOB to view those that are not displayed.

Base

The **Base** fields allow you to specify the numeric base in which you want to define a pattern for a label. The Base fields also let you use a symbol that was specified in the Timing Symbol Table for the pattern. Each label has its own base defined separately from the other labels. If you select one of the Base fields, you will see the following pop-up menu. Decide which base you want to define your pattern in and select that option.

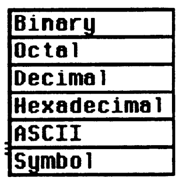


Figure 18-6. Base Pop-up Menu

One of the options in the Base pop-up is ASCII . It allows you to see characters that are represented by the pattern you specified in the **Find Pattern** field.

```
Label >    [ POD 1 ]

 Base >    [ ASCII ]
Find
  Pattern    $    %
```

Figure 18-7. ASCII Defined as Numeric Base

Notice in the figure above that the **Find Pattern** field is no longer a selectable field when the base is ASCII . You cannot specify ASCII characters directly. You must specify a pattern in one of the other bases; then you can switch the base to ASCII and see what characters the pattern represents.

The Symbol option in the Base pop-up allows you to use a symbol that has been specified in the Timing Symbol Tables as a pattern or specify absolute and enter another pattern. You specify the symbol you want to use in the **Find Pattern** field.

Find Pattern

With the **Find Pattern** fields, you configure your timing analyzer to look for a certain pattern in the data. Each label has its own pattern field that you use to specify a pattern for that label.

During a run, the logic analyzer looks for a pattern in your data which is the logical AND of all the labels' patterns. That is, it looks for a simultaneous occurrence of the specified patterns. When it finds the pattern, it triggers at the point that you specified in the **Then find Edge** fields. See "Then Find Edge" later in this chapter for more information about edge triggering.

You select a **Find Pattern** field with one of two methods. The first method is to place the cursor on the Find Pattern field and press SELECT. The second method is to place the cursor on the Find Pattern field and press one of the alphanumeric keys on the front-panel keypad. Both methods give you a pop-up similar to that shown in figure 18-8.

Figure 18-8. Specify Pattern Pop-Up for Find Pattern

The pop-up varies depending on the base you choose and the number of channels you assign to that label. If you press a key on the keypad to open the pop-up, the character on the key is placed in the first location of the pattern.

Enter your pattern in the pop-up and press SELECT. The pattern appears under the label in the **Find Pattern** field.

As mentioned previously in "Base", if you specify ASCII as the base for the label, you won't be able to enter a pattern. You must specify one of the other numeric bases to enter the pattern. Then you can switch the base to ASCII and see what ASCII characters the pattern represents. If you choose Symbols in the Base field, you can use one of the symbols specified in the Timing Symbol Tables as the pattern. The **Find Pattern** field looks similar to that below:

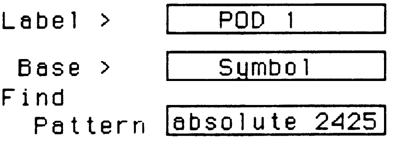


Figure 18-9. Symbol Defined in Base Field

If you select this field you get a pop-up similar to that shown:

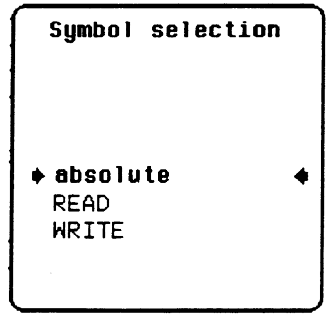


Figure 18-10. Symbol Selection Pop-Up for Find Pattern

The pop-up lists all the symbols defined for that label. It also contains an option "absolute xxxx." Choosing this option gives you another pop-up with which you specify a pattern not given by one of your symbols.

To select an option from the pop-up, use the KNOB to scroll the symbols up and down until the desired symbol is between the two arrows. Press SELECT. The symbol name appears in the Find Pattern field under the label.

When you specify symbols in the Timing Symbol Tables, you also specify the number of characters in the symbol name that are to be displayed. If you specify only three characters of a symbol name in the Symbol menu, only REA of READ and WRI of WRITE would be displayed in the Find Pattern Field. In addition, only the first three letters of "absolute" would be displayed.

Pattern Duration (present for ____)

There are two fields with which you specify the Pattern Duration. They are located next to **present for** ______ in the Timing Trace Specification menu. You use these fields to tell the timing analyzer to trigger before or after the specified pattern has occurred for a given length of time.

The first field can be set to " > " (greater than) or " < " (less than). If you place the cursor on this field and press SELECT, it toggles between > and < . The second field specifies the duration of the pattern. If you select > in the first field, you can set the duration to a value between 30 ns and 10 ms. If you select < in the first field, you can set the duration to a value between 40 ns and 10 ms. If you attempt to set the duration to a value outside the given range, the analyzer will automatically set it to the nearest limit.

To change the value of the pattern duration, place the cursor on the second field and either press SELECT to get a pop-up menu, or just press one of the numeric keys on the front-panel keypad. Both methods give you a Numeric Entry pop-up similar to that shown.

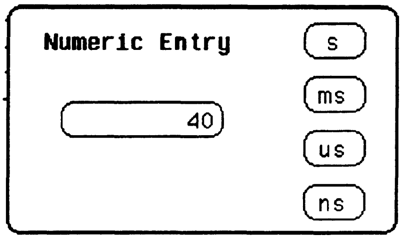


Figure 18-11. Pattern Duration (present for) Pop-Up

With the front-panel keypad, enter the desired pattern duration. Use the KNOB to place the cursor on the correct timing units, then press SELECT. Your value for Pattern Duration will appear in the field.

Note

If you press a key on the keypad to open the pop-up, the number that you pressed will appear in the entry field replacing the previous value. To restore the original value press the CLEAR ENTRY key.

As an example, suppose you configure the **present for ____** field as shown:

```
present for [>] [     50 ns     ]
```

Figure 18-12. Example of Pattern Duration (Greater Than)

This configuration tells the timing analyzer to look for the pattern you specified that occurs for a period of time greater than 50 ns. Once the timing analyzer has found the pattern, it can look for the trigger.

Choosing < (less than) forces glitch and edge triggering off, and the timing analyzer triggers immediately at the end of the pattern that meets the duration requirements. The fields with which you specify edges and glitches do not appear in the menu. For instance, configure the **present for ____** field as shown below.

```
present for [<] [     100 ns     ]
```

Figure 18-13. Example of Pattern Duration (Less Than)

The analyzer triggers when it sees the pattern you specified, and that occurs for a period less than 100 ns. The pattern must also be valid for at least 20 ns.

Then Find Edge

With the **Then find Edge** fields you can specify the edges (transitions) of the data on which your timing analyzer triggers. You can specify a positive edge, a negative edge, or either edge. Each label has its own edge trigger specification field so that you can specify an edge on any channel.

When you specify an edge on more than one channel, the timing analyzer logically ORs them together to look for the trigger point. That is, it triggers when it sees any one of the edges you specified. It also ANDs the edges with the pattern you specified in the Find Pattern fields. The logic analyzer triggers on an edge following the valid duration of the pattern while the pattern is still present. To specify an edge, place the cursor on one of the **Then find Edge** fields and press SELECT. You will see a pop-up similar to that shown in the following figure.

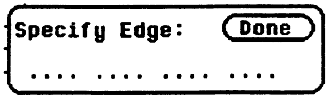


Figure 18-14. Specify Edge Pop-Up for Then Find Edge

Your pop-up may look different than this depending on the number of channels you assigned to the label. Each period in the pop-up indicates that no edge is specified for that channel.

To specify a negative edge, place the cursor on one of the periods in the pop-up and press SELECT once. The period changes to ↓, as shown:

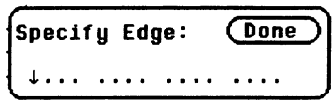


Figure 18-15. Negative Edge Specified

To specify a positive edge, place the cursor on one of the periods and press SELECT twice. The period changes to ↑ , as shown:

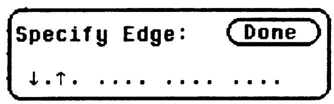


Figure 18-16. Positive Edge Specified

If you want the analyzer to trigger on either a positive or a negative edge, place the cursor on a period and press SELECT three times. The period changes to ↕, as shown:

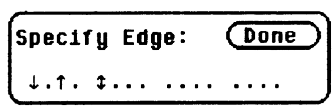


Figure 18-17. Either Edge Specified

If you want to delete an edge specification, place the cursor on the arrow for that channel and press SELECT until you see a period. To clear an entire label, press the CLEAR ENTRY key on the front panel.

When you have finished specifying edges, place the cursor on the Done field and press SELECT to close the pop-up.

Note If you are not in Binary base, you will see dollar signs ($$..) in the Then find Edge field when you close the pop-up. These indicate that edges have been specified; however, the logic analyzer can't display them correctly unless you have selected Binary for the base.

When you set the Acquisition mode on **Glitch**, a glitch detection field, for each label, is added to the screen. These fields allow you to specify glitch triggering on your timing analyzer. Selecting one of these fields displays the following pop-up menu.

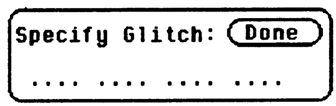


Figure 18-18. Specify Glitch Pop-Up for Then Find Glitch

Your pop-up may look different depending on the number of channels you have assigned to the label. Each period indicates that the channel has not been specified for glitch triggering.

To specify a channel for glitch triggering, place the cursor on one of the periods and press SELECT. The period is replaced with an asterisk, indicating that the logic analyzer will trigger on a glitch on this channel.

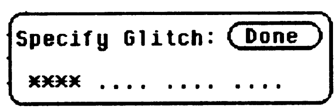


Figure 18-19. Glitches Specified

If you want to delete a glitch specification, place the cursor on the asterisk and press SELECT. The asterisk is replaced with a period.

Note

If you are not in Binary base, you will see dollar signs ($$..) in the Glitch field when you close the pop-up. This indicates that glitches have been specified; however, the logic analyzer can't display them correctly unless you have selected Binary for the base.

When more than one glitch has been specified, the logic analyzer logically ORs them together. In addition, the logic analyzer ORs the glitch specifications with the edge specifications, then ANDs the result with the pattern you specified in the Find Pattern fields in order to find the trigger point. A boolean expression illustrating this is:

(glitch + glitch + edge + edge) * pattern

Note

If you select < (less than) in the **present for___** field, edge and glitch triggering are turned off. The Then find Edge or Glitch field no longer appears on the screen. The logic analyzer then triggers only on the pattern specified in the Find Pattern fields.

19

Timing Waveforms Menu

Introduction

The Timing Waveforms menu is the display menu of the timing analyzer. This chapter describes the Timing Waveforms menu and how to interpret it. It also tells you how to use the fields to manipulate the displayed data so you can find your measurement answers.

There are two different areas of the timing waveforms display: the menu area and the waveforms area. The menu area is in the top one-fourth of the screen and the waveforms area is the bottom three-fourths of the screen.

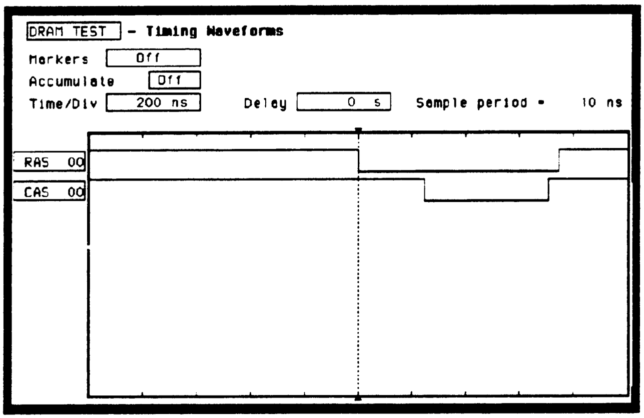


Figure 19-1. Timing Waveforms Menu

The waveforms area displays the data that the timing analyzer acquires. The data is displayed in a format similar to an oscilloscope with the horizontal axis representing time and the vertical axis representing amplitude. The basic differences between an oscilloscope display and the timing waveforms display are: in the timing waveforms display the vertical axis only displays highs (above threshold) and lows (below threshold). Also, the waveform lows are represented by a thicker line for easy differentiation.

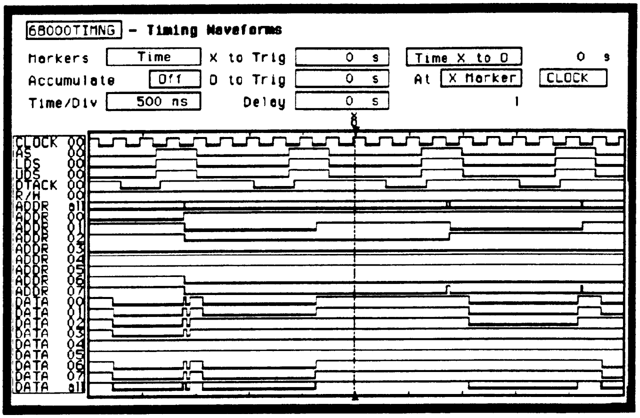


Figure 19-2. Timing Waveforms Menu with 24 Waveforms

Accessing the Timing Waveforms Menu

The Timing Waveforms Menu is accessed by the pressing the DISPLAY key on the front panel when the timing analyzer is on. It will automatically be displayed when you press RUN.

Timing Waveforms Menu Fields

The menu area contains fields that allow you to change the display parameters, place markers, and display waveform measurement parameters.

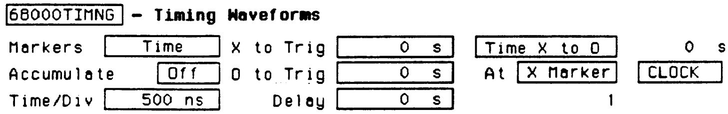


Figure 19-3. Timing Waveforms Menu Fields

Markers

The **Markers** field allows you to specify how the X and O markers will be positioned on the timing data. The options are:

- Off
- Time
- Patterns
- Statistics
- Markers Off/Sample Period

When the markers are off they are not visible and the sample period is displayed. In transitional timing mode, the sample period will always be 10 ns. In Glitch mode, the sample period is controlled by the Time/Div setting and can be monitored by turning the markers off.

Note

The sample period displayed is the sample period of the last acquisition. If you change the Time/Div setting, you must press RUN to initiate another acquisition before the sample period is updated.

Although the markers are off, the logic analyzer still performs statistics, so if you have specified a stop measurement condition the measurement will stop if the pattern specified for the markers is found.

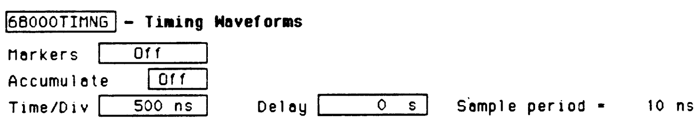


Figure 19-4. Markers Off

Markers Time

When the markers are set to Time, you can place the markers on the waveforms at events of interest and the logic analyzer will tell you:

- Time X to Trig(ger).
- Time O to Trig(ger).
- Time X to O.

To position the markers, move the cursor to the field of the marker you wish to position and press SELECT. A pop-up will appear showing the current time for that marker. Either rotate the KNOB or enter a numeric value from the keypad to change the position of that marker. Pressing SELECT when you are finished positions the marker and closes the pop-up.

When the cursor is on either the **X to Trig** or **O to Trig** fields, you can also enter a value directly from the keypad without pressing SELECT.

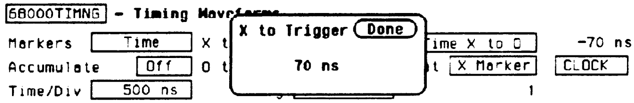


Figure 19-5. Markers Time

The **Time X to O** field will change according to the position of the X and O markers. If you place the cursor on the Time X to O field and press SELECT, another pop-up will appear showing you all three times: X to Trigger, O to Trigger, and Time X to O.

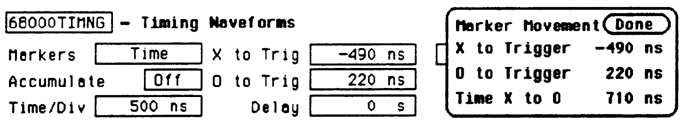


Figure 19-6. Time X to O Pop-up

If you rotate the KNOB while this pop-up is open, both X and O markers will move, but the relative placement between them will not change.

Markers Patterns

When the markers are set to patterns, you can specify the patterns on which the logic analyzer will place the markers. You can also specify how many occurrences of each marker pattern the logic analyzer looks for. This use of the markers allows you to find time between specific patterns in the acquired data.

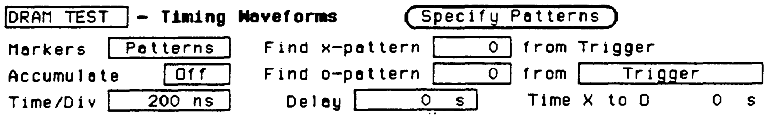


Figure 19-7. Markers Patterns

Patterns for each marker (X and O) can be specified. Patterns can be specified for both markers in each label. The logic analyzer searches for the logical "and" of patterns for all labels even though only one label can be displayed at a time. You can also specify whether the marker is placed on the pattern at the beginning of its occurrence (entering) or at the end of its occurrence (leaving) as shown in figure 19-8.

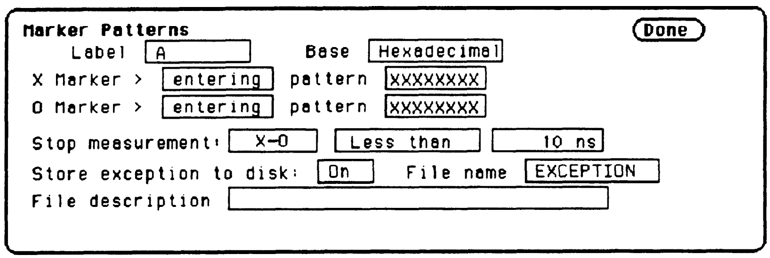


Figure 19-8. Marker Patterns Pop-up menu

Stop Measurement

Another feature of markers set to patterns is the Stop measurement when **Time X-O** ____. The options are: Less than, Greater than, In range, Not in range

With this feature you can use the logic analyzer to look for a specified time or range of time between the marked patterns and have it stop acquiring data when it sees this time between markers. (The X marker must precede the O marker.)

Also available is **Store exception to disk** which allows you to specify a file on the disk that exceptions can be stored in. The default filename is EXCEPTION.

Note

The upper and lower range boundaries must not be the same value. For example, if you want to stop a measurement when the X and O markers are in range of 200 ns, you should set the range values to 190 ns and 210 ns. This eliminates erroneous measurement termination.

Markers Statistics

When statistics are specified for markers, the logic analyzer will display the following:

- Number of total runs.
- Number of valid runs (runs where markers were able to be placed on specified patterns).
- Minimum time between the X and O markers.
- Maximum time between the X and O markers.
- Average time between the X and O markers.

Statistics are based on the time between markers which are placed on specific patterns. If a marker pattern is not specified, the marker will be placed on the trigger point by the logic analyzer. In this case the statistical measurement will be the time from the trigger to the specified marker. How the statistics will be updated depends on the timing trace mode (repetitive or single).

In repetitive, statistics will be updated each time a valid run occurs until you press STOP. When you press RUN after STOP, the statistics will be cleared and will restart from zero.

In single, each time you press RUN an additional valid run will be added to the data and the statistics will be updated. This will continue unless you change the placement of the X and O markers between runs.

Accumulate Mode

Accumulate mode is selected by toggling the Accumulate ON/OFF field in the Timing Waveforms menu. When accumulate is on, the timing analyzer displays the data from a current acquisition on top of the previously acquired data.

When the old data is cleared depends on whether the trace mode is in single or repetitive. In single, new data will be displayed on top of the old each time RUN is selected as long as you stay in the Timing Waveforms menu between runs. Leaving the Timing Waveforms menu always clears the accumulated data. In repetitive mode, data is cleared from the screen only when you start a run after stopping acquisition with the STOP key.

At ____ Marker ____

The **At X (or O) Marker** ______ fields allow you to select either the X or O markers. You can place these markers on the waveforms of any label and have the logic analyzer tell you what the pattern is. For example, in the timing waveforms display (figure 19-9) the number 35 to the right of the **Delay** _____ field is the pattern in hexadecimal that is marked by the O marker. The base of the displayed field is determined by the base of the specified label you selected in the Timing Trace menu.

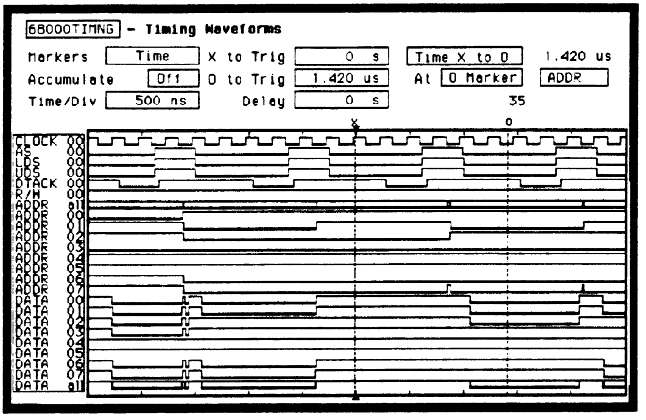


Figure 19-9. At O Marker ADDR fields

This display tells you that 35H is the pattern on the address label lines where the O marker is located.

The next field to the right of the **At ____ Marker** field will pop up when selected and show you all the labels assigned to the timing analyzer as shown below.

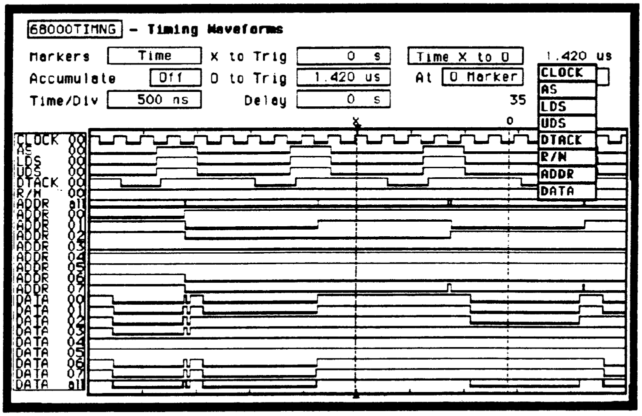


Figure 19-10. Label Option Pop-up

Time/Div (time per division) Field

The time per division field allows you to change the width of the time window of the Timing Waveforms menu.

When the pop-up is open you can change the time per division by rotating the KNOB or entering a numeric value from the keypad. When you rotate the KNOB, the time per division increments or decrements in 1-2-5 sequence from 10 ns/div to 50 ms/div.

Note

Sample period is fixed at 10 ns in the Transitional acquisition mode.

When you enter a value from the keypad, the time per division does not have to be a 1-2-5 sequence.

Note

In Glitch mode, changing the Time/Div setting changes the sample period for the next run. To view the sample period after the next run, turn the markers off if they are on and press RUN.

Delay Field

The **Delay** field allows you to enter a delay. The delay can be either positive or negative. Delay allows you to place the time window (selected by Time/Div) of the acquired data at center screen.

The inverted triangle in the horizontal center of the waveforms area of the display represents trigger + delay. The vertical dotted line represents the trigger point (see figure 19-11).

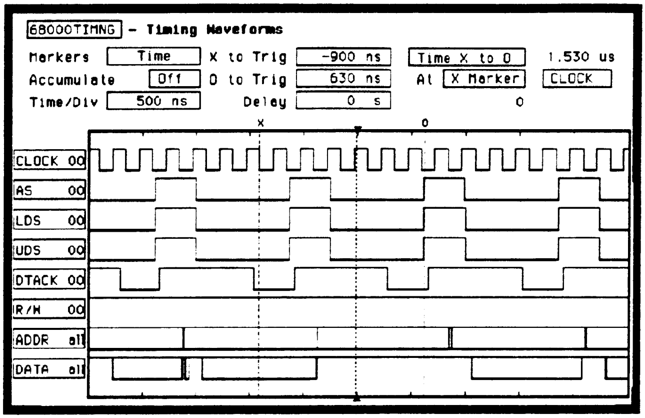


Figure 19-11. Trigger and Trace Points

If you want to trace after the trigger point, enter a positive delay. If you want to trace before the trigger point (similar to negative time) enter a negative delay. The logic analyzer is capable of maximum delays of −2500 seconds to +2500 seconds. In Transitional mode the maximum delay is determined by the number of transitions of the incoming data. Data may not be displayed at all settings of Time/Div and Delay.

In Glitch mode the maximum delay is 25 seconds, which is controlled by memory and sample period (512 x 50ms). The sample rate is also dependent on the delay setting. It is represented by the following formula:

if delay < 20 ns
  Hwdelay = 20 ns (this is an instrument constant)

if delay > 10 ms
  Hwdelay = 10 ms

else Hwdelay = delay (delay setting in timing waveforms menu)

Sample period = larger of:

Time/Div ÷ 25 or

absolute value [(delay − Hwdelay) ÷ 256]

If sample period > 50 ms

Then sample period = 50 ms

20

Timing Analyzer Measurement Example

Introduction

In this chapter you will learn how to use the timing analyzer by setting up the logic analyzer to simulate a simple timing measurement. Since you may not have the same test circuit available, we will give you the measurement results as actually measured by the logic analyzer,

The exercise in this chapter is organized in a task format. The tasks are ordered in the same way you will most likely use them once you become an experienced user. The steps in this format are both numbered and lettered. The numbered steps state the step objective. The lettered steps explain how to accomplish each step objective. There is also an example of each menu after it has been properly set up.

How you use the steps depends on how much you remember from chapters 1 through 4 of the *Getting Started Guide* . If you can set up each menu by just looking at the menu picture, go ahead and do so. If you need a reminder of what steps you need to perform, follow the numbered steps. If you still need more information about "how," use the lettered steps.

To gain confidence using your logic analyzer, we recommend that you configure the menus as you follow the simulated measurement example up to section "Acquiring the Data." From that section unto the end, you will see the measurement results on the Timing Waveforms screen as if you had the real test circuit connected, and as if you had selected RUN.

Problem Solving with the Timing Analyzer

In this exercise, assume you are designing a dynamic RAM memory (DRAM) controller and you must verify the timing of the row address strobe (RAS) and the column address strobe (CAS). You are using a 4116 dynamic RAM and the data book specifies that the minimum time from when LRAS is asserted (goes low) to when LCAS is no longer asserted (goes high) is 250 ns. You could use an oscilloscope but since the timing analyzer will do just fine when you don't need voltage parametrics you decide to go ahead and use the logic analyzer.

What Am I Going to Measure?

After configuring the logic analyzer and hooking it up to your circuit under test, you will be measuring the time (x) from when the RAS goes low to when the CAS goes high, as shown below.

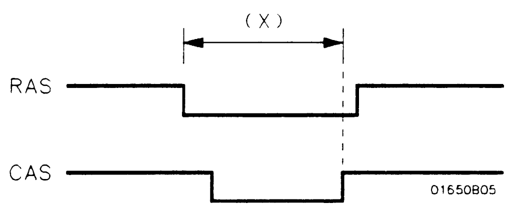


Figure 20-1. RAS and CAS Signals

How Do I Configure the Logic Analyzer?

In order to make this timing measurement, you must configure the logic analyzer as a timing analyzer. By following these steps you will configure Analyzer 1 as the timing analyzer.

If you are in the System Configuration menu you are in the right place to get started and you can start with step 2; otherwise, start with step 1.

1. Using the field in the upper left corner of the display, get the System Configuration menu on screen.

   a. Place the cursor on the field in the upper left corner of the display and press SELECT.

   b. Place the cursor on System and press SELECT.

2. In the System Configuration menu, change Analyzer 1 type to Timing. If analyzer 1 is already a timing analyzer, go on to step 3.

   a. Place the cursor on the **Type:** ____ field and press SELECT.

   b. Place the cursor on Timing and press SELECT.

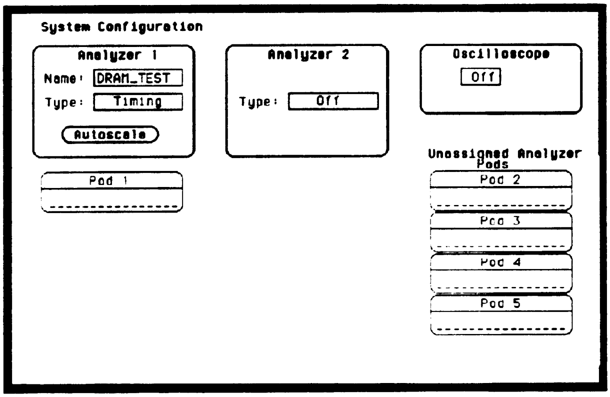


Figure 20-2. System Configuration Menu

3. Name Analyzer 1 "DRAM TEST" (optional)

   a. Place the cursor on the **Name:** ____ field of Analyzer 1 and press SELECT.

   b. With the Alpha Entry pop-up, change the name to "DRAM TEST" (see "How to Enter Alpha Data" in chapter 3 if you need a reminder).

4. Assign pod 1 to the timing analyzer.

   a. Place the cursor on the Pod 1 field and press SELECT.

   b. In the Pod 1 pop-up, place the cursor on Analyzer 1 and press SELECT.

Connecting the Probes

At this point, if you had a target system with a 4116 DRAM memory IC, you would connect the logic analyzer to your system.

Since you will be assigning Pod 1 bit 0 to the RAS label, you hook Pod 1 bit 0 to the memory IC pin connected to the RAS signal. You hook Pod 1 bit 1 to the IC pin connected to the CAS signal.

Activity Indicators

When the logic analyzer is connected and your target system is running, you will see activity indicators, as shown below, at the right-most end (least significant bits) of the Pod 1 field in the System Configuration menu. This indicates the RAS and CAS signals are transitioning.

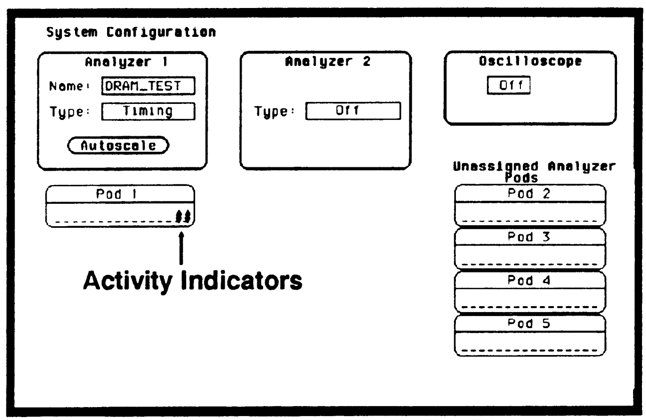


Figure 20-3. Activity Indicators

Configuring the Timing Analyzer

Now that you have configured the system, you are ready to configure the timing analyzer. You will be:

- Creating two names (labels) for the input signals
- Assigning the channels connected to the input signals
- Specifying a trigger condition

1. Display the Timing Format Specification menu.

   a. Press the FORMAT key on the front panel.

2. Name two labels, one RAS and one CAS.

   a. Place the cursor on the top field in the label column and press SELECT.

   b. Place the cursor on Modify label and press SELECT .

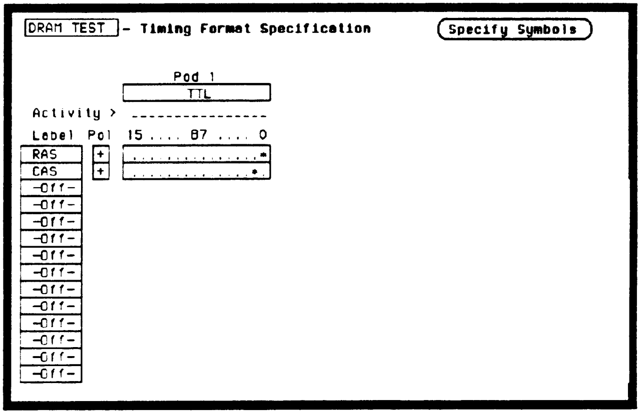


Figure 20-4. Timing Format Specification Menu

   c. With the Alpha Entry pop-up, change the name of the label to RAS.

d. Name the second label CAS by repeating steps a through c.

3. Assign the channels connected to the input signals (Pod 1 bits 0 and 1) to the labels RAS and CAS respectively.

   a. Place the cursor on the bit assignment field below Pod 1 and to the right of RAS and press SELECT.

   b. Any combination of bits may be assigned to this pod; however, you will want only bit 0 assigned to the RAS label. The easiest way to assign bits is to press the CLEAR ENTRY key to un-assign any assigned bits before you start.

   c. Place the cursor on the period under the 0 in the bit assignment pop-up and press SELECT . This will place an asterisk in the pop-up for bit 0 indicating Pod 1 bit 0 is now assigned to the RAS label. Place cursor on Done and press SELECT to close the pop-up.

   d. Assign Pod 1 bit 1 to the CAS label by moving the cursor to bit 1 and pressing SELECT.

Specifying a Trigger Condition

To capture the data and then place the data of interest in the center of the display of the Timing Waveforms menu, you need to tell the logic analyzer when to trigger. Since the first event of interest is when the LRAS is asserted (negative-going edge of RAS), you need to tell the logic analyzer to trigger on a negative-going edge of the RAS signal.

1. Select the Timing Trace menu by pressing the TRACE key.

2. Set the trigger so that the logic analyzer triggers on the negative-going edge of the RAS.

   a. Place the cursor on the Then find Edge field under the label RAS, then press SELECT.

   b. Place the cursor on the . (period) in the pop-up and press SELECT once. Pressing SELECT once in this pop-up changes a period to ↓ which indicates a negative-going edge.

   c. Place the cursor on Done and press SELECT. The pop-up closes and a $ will be located in this field. The $ indicates an edge has been specified even though it can't be shown in the HEX base.

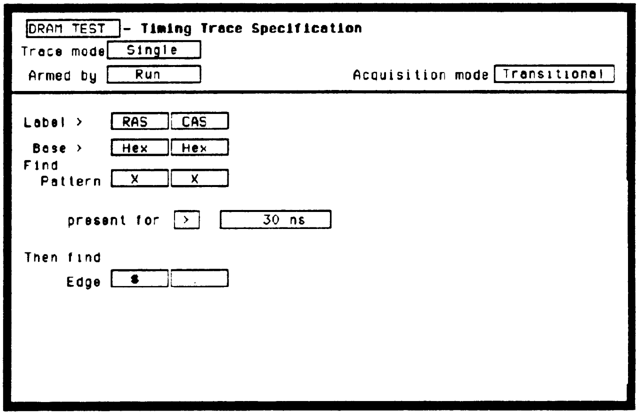


Figure 20-5. Trigger Edge Specified

Acquiring the Data

Now that you have configured and connected the logic analyzer, you acquire the data for your measurement by pressing the RUN key. The logic analyzer will look for a negative edge on the RAS signal and trigger if it sees one. When it triggers, the display switches to the Timing Waveforms menu.

Note

From this point in the exercise unto the end, we will give you the measurement results. This way, you will not have to obtain and use an identical circuit.

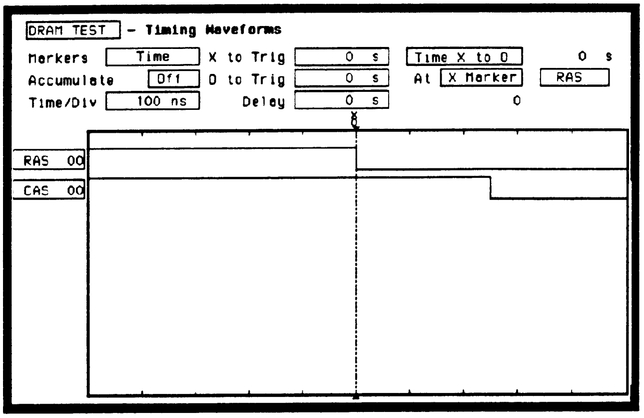


Figure 20-6. Timing Waveforms Menu

The RAS label shows you the RAS signal and the CAS label shows you the CAS signal. Notice the RAS signal goes low at or near the center of the waveform display area (horizontal center).

The Timing Waveforms Menu

The Timing Waveforms menu differs from the other menus you have used so far in this exercise. Besides displaying the acquired data, it has menu fields that you use to change the way the acquired data is displayed and fields that give you timing answers. Before you can use this menu to find answers, you need to know some of the special symbols and their functions. The symbols are:

- The X and O
- The ▼
- The vertical dotted line

The X and O

The X and O are markers you use to find your answer. You place them on the points of interest on your waveforms, and the logic analyzer displays the time between the markers. The X and O markers will be in the center of the display when X to trig (ger) and O to trig (ger) are both 0.000 s (see example below).

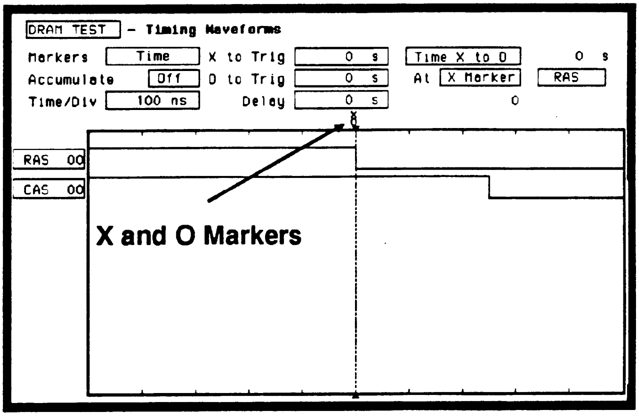


Figure 20-7. X & O Markers

The ▼

The (inverted triangle) indicates the trace point. Remember, trace point = trigger + delay. Since delay in this example is 0.000 s, you will see the negative-going edge of the RAS signal at center screen under the .

The Vertical Dotted Line

The vertical dotted line indicates the trigger point you specified in the Timing Trace Specification menu. The vertical dotted line is at center screen under the inverted triangle and is superimposed on the negative-going edge of the RAS signal.

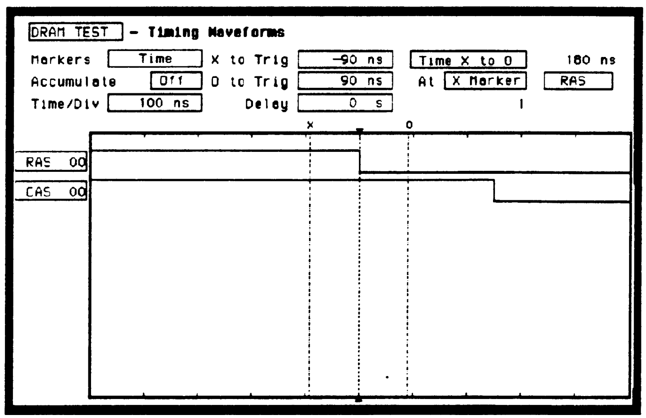


Figure 20-8. Inverted Triangle & Vertical Dotted Line

Configuring the Display

Now that you have acquired the RAS and CAS waveforms, you need to configure the Timing Waveforms menu for best resolution and to obtain your answer.

Display Resolution

You get the best resolution by changing the Time/Div to a value that displays one negative-going edge of both the RAS and CAS waveforms. Set the Time/Div by following these steps.

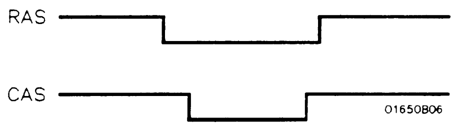


Figure 20-9. RAS and CAS Signals

1. Place the cursor on Time/Div and press SELECT . The Time/Div pop-up appears, showing you the current setting.

2. While the pop-up is present, rotate the KNOB until your waveform shows you only one negative-going edge of the RAS waveform and one positive-going edge of the CAS waveform (see above). In this example 200 ns is best.

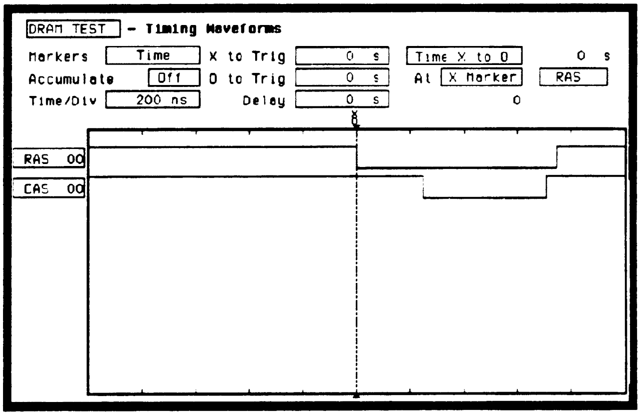


Figure 20-10. Changing Time/Div.

Making the Measurement

What you want to know is how much time elapses between the time RAS goes low and the time CAS goes high again. You will use the X and O markers to quickly find the answer. Remember, you specified the negative-going edge of the RAS to be your trigger point; therefore, the X marker should be on this edge if X to Trig = 0. If not, follow steps 1 and 2.

1. Place the cursor on the X to Trig field and press SELECT . A pop-up will appear showing you the current time from the X marker to the trigger; however, you don't need to worry about this number now.

2. Rotate the KNOB to place the X marker on the negative-going edge of the RAS waveform and press SELECT . The pop-up closes and displays X to Trig = 0.000 s.

3. Place the cursor on O to Trig and press SELECT . Repeat step 2 except place the O marker on the positive-going edge of the CAS waveform and press SELECT. The pop-up closes and displays O to Trig = 710 ns.

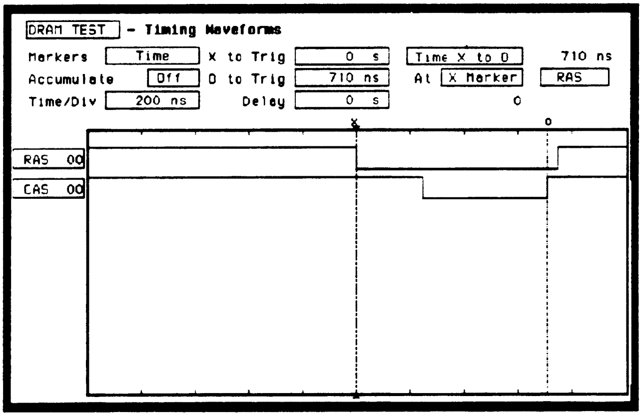


Figure 20-11. Marker Placement

Finding the Answer

Your answer could be calculated by adding the X to Trig and O to Trig times, but you don't need to bother. The logic analyzer has already calculated this answer and displays it in the **Time X to O ___** field.

This example indicates the time is 710 ns. Since the data book specifies a minimum of 250 ns, it appears your DRAM controller circuit is designed properly.

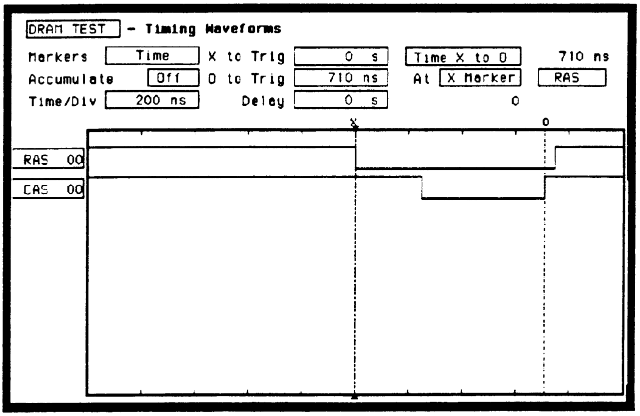


Figure 20-12. Time X to O

Summary

You have just learned how to make a simple timing measurement with the HP 1652B/53B logic analyzer. You have learned to do the following:

- Specified a timing analyzer.
- Assigned pod 1.
- Assigned bits.
- Assigned labels.
- Specifed a trigger condition.
- Learned which probes to connect.
- Acquired the data.
- Configured the display.
- Set the Time/Div for best resolution.
- Positioned the markers for the measurement answer.

You have seen how easy it is to use the timing analyzer to make timing measurements that you could have made with a scope. You can use the timing analyzer for any timing measurement that doesn't require voltage parametrics or doesn't go beyond the accuracy of the timing analyzer.

21

Timing/State Measurement Example

Introduction

In this chapter you will learn how to use the timing and state analyzers interactively by setting up the logic analyzer to simulate a simple timing/state measurement. Since you may not have the same test circuit available, we will give you the measurement results as actually measured by the logic analyzer.

The exercise in this chapter is organized differently than the exercises in the previous chapters. Since you have already set up both the timing and state analyzers, you should be ready to set them up for this simulated measurement by just looking at the menu pictures.

Any new set-ups in this exercise will be explained in task format steps like the previous chapters.

To gain confidence using your logic analyzer, we recommend that you configure the menus as you follow the simulated measurement example up to section "Acquiring the Data." From that section unto the end, you will see the measurement results on the display screens as if you had the real test circuit connected, and as if you had selected RUN.

Problem Solving with the Timing/State Analyzer

In this example assume you have designed a microprocessor-controlled circuit. You have completed the hardware, and the software designer has completed the software and programmed the ROM (read-only memory). When you turn your circuit on for the first time, your circuit doesn't work properly. You have checked the power supply voltages and the system clock, and they are working properly.

Since the circuit has never worked before, you and the software engineer aren't sure if it is a hardware or software problem. You need to do some testing to find a solution.

You also notice the circuit fails intermittently. More specifically, it only fails when the microprocessor attempts to address a routine that starts at address 8930.

What Am I Going to Measure?

To see what might be causing the failure, you decide to start where the microprocessor goes to the routine that starts at address 8930. The first thing you check is whether the microprocessor actually addresses address 8930. The next thing you check is whether the code is correct in all the steps in this routine.

Your measurement, then, requires verification of the following:

- Whether the microprocessor addresses location 8930.
- Whether all the addresses within the routine are correct.
- Whether all the data at the addresses in the routine are correct.

If the routine is correct, the state listing displays the following:

+ 0000 008930 B03C
+ 0001 008932 61FA
+ 0002 008934 67F8
+ 0003 008936 B03C
+ 0004 00892E 61FA

How Do I Configure the Logic Analyzer?

In order to make this measurement, you must configure the logic analyzer as a state analyzer because you want to trigger on a specific state (8930). You also want to verify that the addresses and data are correct in the states of this routine.

Configure the logic analyzer so that Analyzer 1 is a state analyzer as shown below:

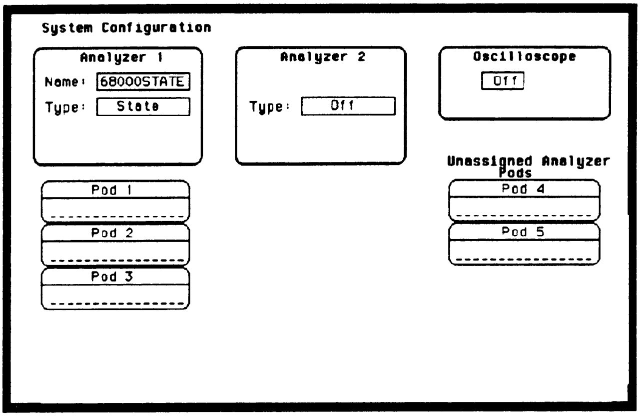


Figure 21-1. System Configuration Menu

Configuring the State Analyzer

Now that you have configured the system, you are ready to configure the state analyzer. Configure the State Format Specification menu as shown:

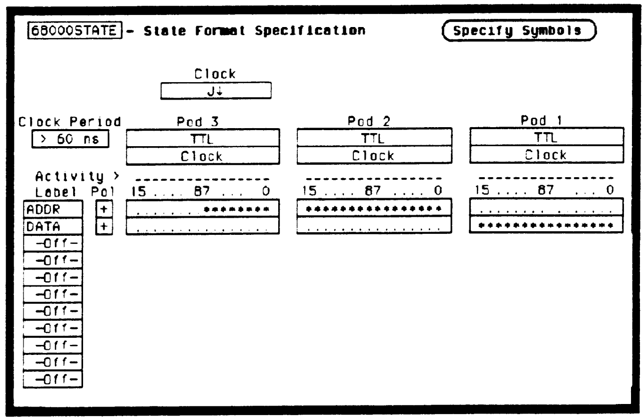


Figure 21-2. State Format Specification Menu

Configure the State Trace Specification menu as shown:

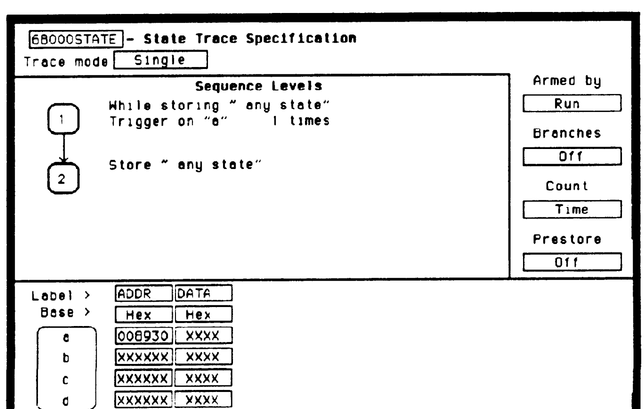


Figure 21-3. State Trace Specification Menu

Connecting the Probes

At this point, if you had a target system with a 68000 microprocessor, you would connect the logic analyzer to your system. Since you will be assigning labels ADDR and DATA, you will hook the probes to your system accordingly:

- Pod 1 probes 0 through 15 to the data bus lines D0 through D15.
- Pod 2 probes 0 through 15 to the address bus lines A0 through A15.
- Pod 3 probes 0 through 7 to the address bus lines A16 through A23.
- Pod 1, CLK (J clock) to the address strobe (LAS).

Acquiring the Data

Since you want to capture the data when the microprocessor sends address 8930 on the bus, you press the RUN key to arm the state analyzer. If the microprocessor sends address 8930, it will trigger the state analyzer and switch the display to the State Listing menu.

Note

From this point in the exercise unto the end, we will give you the measurement results. This way, you will not have to obtain and use an identical circuit.

Finding the Problem

You look at this listing to see what the data is in states + 0000 through + 0004. You know your routine is five states long.

The 68000 does address location 8930, so you know that the routine is addressed. Now you need to compare the state listing with the following correct addresses and data:

+ 0000	008930	B03C
+ 0001	008932	61FA
+ 0002	008934	67F8
+ 0003	008936	B03C
+ 0004	00892E	61FA

As you compare the state listing (shown below) with the above data you notice the data at address 8932 is incorrect. Now you need to find out why.

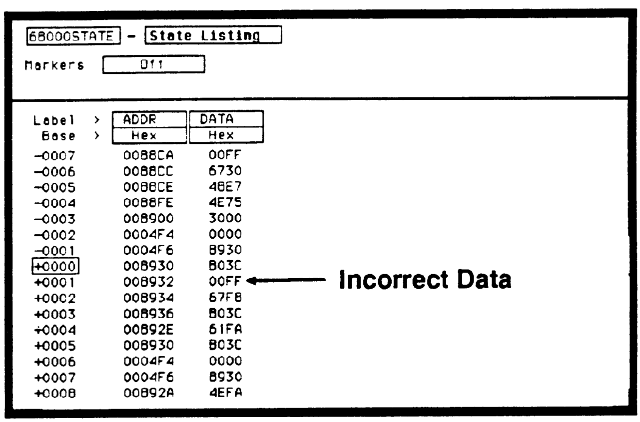


Figure 21-4. Incorrect Data

Your first assumption is that incorrect data is stored to this memory location. Assume this routine is in ROM since it is part of the operating system for your circuit. Since the ROM is programmed by the software designer, you have the software designer verify whether or not the data at address 8932 is correct. The software designer tells you that the data is correct. Now what do you do?

Now it's time to look at the hardware to see if it is causing incorrect data when the microprocessor reads this memory address. You decide you want to see what is happening on the address and data buses during this routine in the time domain.

In order to see the time domain, you need the timing analyzer.

What Additional Measurements Must I Make?

Since the problem exists during the routine that starts at address 8930, you decide you want to see the timing waveforms on the address and data bus when the routine is running. You also want to see the control signals that control the read cycle. You will then compare the waveforms with the timing diagrams in the 68000 data book.

Your measurement, then, requires verification of the following:

- Correct timing of the control signals.
- Stable addresses and data during the memory read.

The control signals you must check are listed below:

- System clock.
- Address strobe (AS).
- Lower and upper data strobes (LDS and UDS).
- Data transfer acknowledge (DTACK).
- Read/write (R/W).

How Do I Re-Configure the Logic Analyzer?

In order to make this measurement, you must re-configure the logic analyzer so Analyzer 2 is a timing analyzer. You leave Analyzer 1 as a state analyzer since you will use the state analyzer to trigger on address 8930.

Configure the logic analyzer so Analyzer 2 is a timing analyzer as shown:

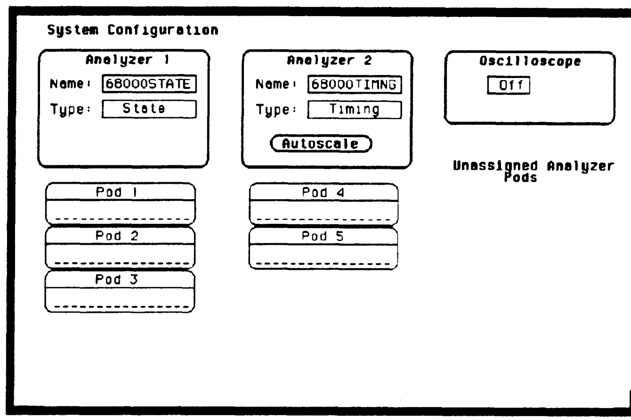


Figure 21-5. System Configuration Menu

Connecting the Timing Analyzer Probes

At this point you would connect the probes of pods 4 and 5 as follows:

- Pod 4 bit 0 to address strobe (AS).
- Pod 4 bit 1 to the system clock.
- Pod 4 bit 2 to low data strobe (LDS).
- Pod 4 bit 3 to upper data strobe (UDS).
- Pod 4 bit 4 to the read/write (R/W).
- Pod 4 bit 5 to data transfer acknowledge (DTACK).
- Pod 5 bits 0 through 7 to address lines A0 through A7.
- Pod 5 bits 8 through 15 to data lines D0 through D7 .

Configuring the Timing Analyzer

Now that you have configured the system, you are ready to configure the timing analyzer. Configure the Timing Format Specification menu as shown:

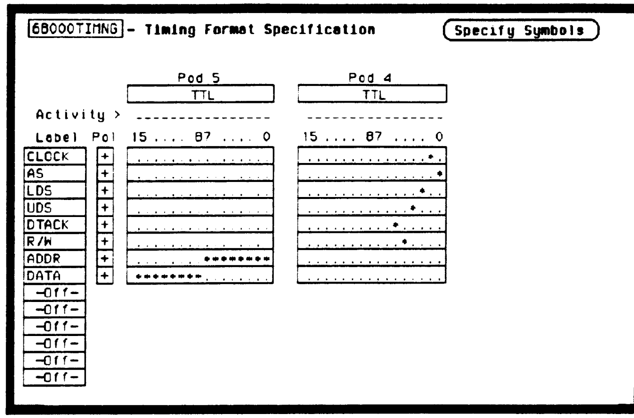


Figure 21-6. Timing Format Specification Menu

Configure the Timing Trace Specification as shown:

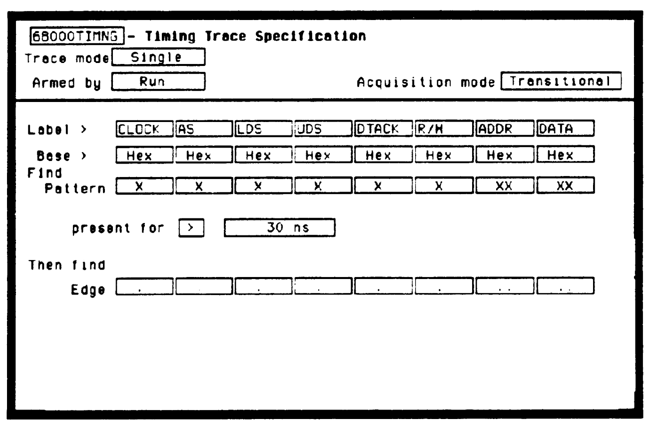


Figure 21-7. Timing Trace Specification Menu

Setting the Timing Analyzer Trigger

Your timing measurement requires the timing analyzer to display the timing waveforms present on the buses when the routine is running. Since you triggered the state analyzer on address 8930, you want to trigger the timing analyzer so the timing waveforms can be time correlated with the state listing.

To set up the logic analyzer so that the state analyzer triggers the timing analyzer, perform these steps:

1. Display the Timing Trace Specification menu.
2. Place the cursor on the **Armed by ____** field and press SELECT.
3. Place the cursor on the 68000STATE option in the pop-up and press SELECT.

Your timing trace specification should match the menu shown:

State Analyzer Arms Timing Analyzer

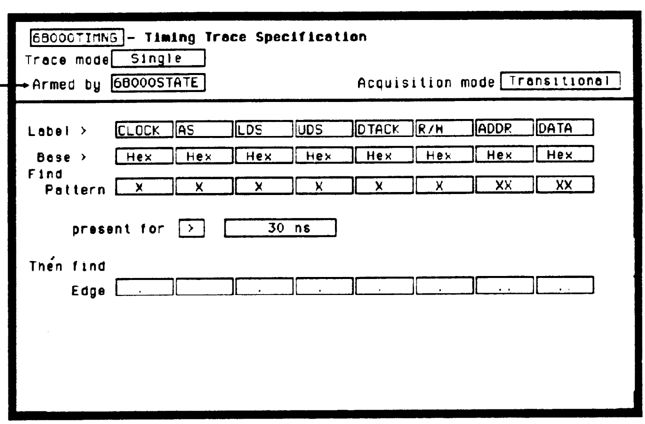


Figure 21-8. Armed by 68000 STATE

Time Correlating the Data

In order to time correlate the data, the logic analyzer must store the timing relationships between states. Since the timing analyzer samples asynchronously and the state analyzer samples synchronously, the logic analyzer must use the stored timing relationship of the data to reconstruct a time correlated display.

To set up the logic analyzer to keep track of these timing relationships, turn on a counter in the State Trace Specification menu. The following steps show you how:

1. Display the State Trace Specification menu.

2. Place the cursor in the field just below Count on the right side of the display and press SELECT.

3. Place the cursor on the Time option and press SELECT. The counter will now be able to keep track of time for the time correlation.

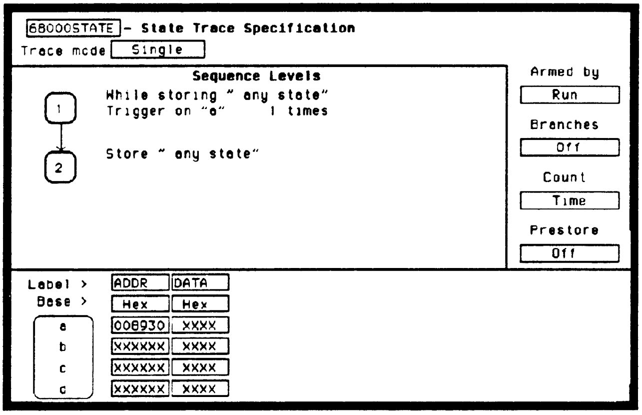


Figure 21-9. Count Set to Time

Re-acquiring the Data

After you connect the probes of pods 4 and 5 to your circuit, all you have to do is press RUN. When the logic analyzer acquires the data it switches the display to the State Listing menu unless you switched one of the other menus to the timing analyzer after reconfiguring the State Trace menu. Regardless of which menu is displayed, change the display to the Mixed Mode.

Mixed Mode Display

The Mixed mode display shows you both the State Listing and Timing Waveforms menus simultaneously. To change the display to the Mixed Mode:

1. Place the cursor on the field in the upper left corner of the display and press SELECT.

2. Place the cursor on Mixed Mode and press SELECT. You will now see the mixed display as shown:

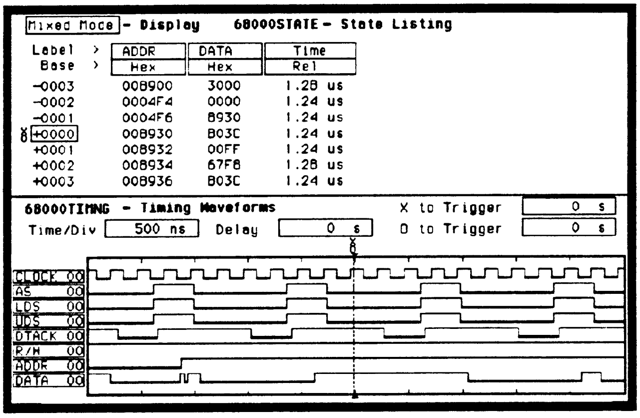


Figure 21-10. Mixed Mode Display

Interpreting the Display

In the Mixed Mode display the state listing is in the top half of the screen and the timing waveforms are in the lower half. The important thing to remember is that you time correlated this display so you could see what is happening in the time domain during the faulty routine.

Notice that the trigger point in both parts of the display is the same as it was when the displays were separate. The trigger in the state listing is in the box containing + 0000 and the trigger of the timing waveform is the vertical dotted line.

As you look at the mixed display, you notice nothing wrong except the data at address 8932 is incorrect. However, you are seeing only one bit each of the address and the data. To see all the data and addresses in the timing waveform part of the display, you must overlap them.

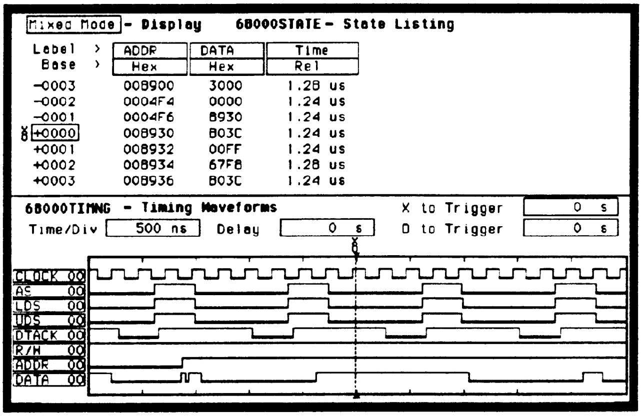


Figure 21-11. Interpreting the Display

Overlapping Timing Waveforms

Since you see nothing wrong with the timing waveforms so far, you think unstable data may be on the data lines during the read cycle. In order to see unstable data, you must be able to see all the data lines during the read and look for transitions. Overlapping the waveforms allows you to do this. To overlap waveforms, follow these steps:

1. Place the cursor on the 00 of the ADDR 00 label and press SELECT. The following pop-up opens in which you specify the bit or bits of the address bus you want to overlap.

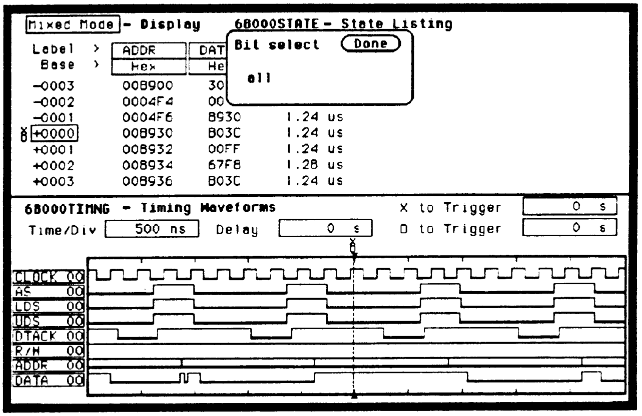


Figure 21-12. Overlapping Timing Waveforms

2. Rotate the KNOB until all is displayed and press SELECT. All the address bits will be overlapped on one line.

3. Repeat step 2 except overlap the data bits.

Finding the Answer

As you look at the overlapping waveforms, you notice there are transitions on the data lines during the read cycle, indicating the data is unstable. You have found the probable cause of the problem in this routine. Additional troubleshooting of the hardware will identify the actual cause.

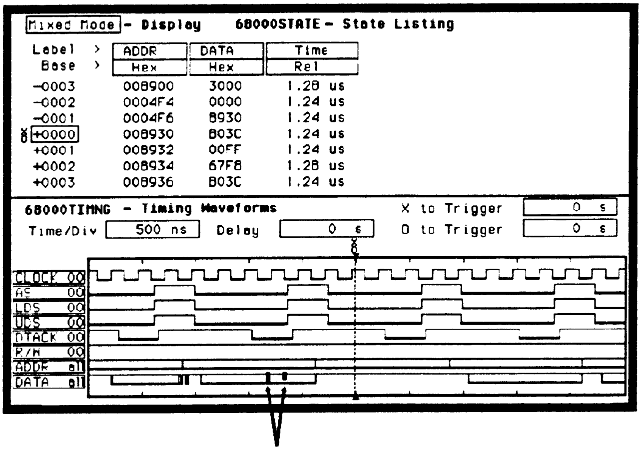


Figure 21-13. Unstable Data

Summary

You have just learned how to use the timing and state analyzers interactively to find a problem that first appeared to be a software problem, but actually was a hardware problem.

You have learned to do the following:

- Trigger one analyzer with the other.
- Time correlate measurement data.
- Interpret the Mixed mode display.
- Overlap timing waveforms.

If you have an HP 1653B, you do not have enough channels to simultaneously capture all the data for a 68000. But, since you probably aren't working with 16-bit microprocessors, this exercise is still valuable because it shows you how to make the same kind of measurement on an eight-bit microprocessor.

22

The Oscilloscope

Introduction

This chapter introduces the oscilloscope and gives an overview of the main menus that you will use on your oscilloscope. Also included are scope menu maps. The purpose and functions of each menu are explained in detail in the following chapters.

- Chapter 23 explains the Channel Menu.
- Chapter 24 explains the Trigger Menu.
- Chapter 25 explains the Waveforms Menu.
- Chapter 26 explains Mixed Mode Displays.
- Chapter 27 gives you a basic mixed mode measurement example.

An actual signal from the compensation signal output is used through out most of these chapters to better illustrate how to use the different oscilloscope menus. If you need an introduction to oscilloscopes, refer to *Feeling Comfortable with Digitizing Oscilloscopes*.

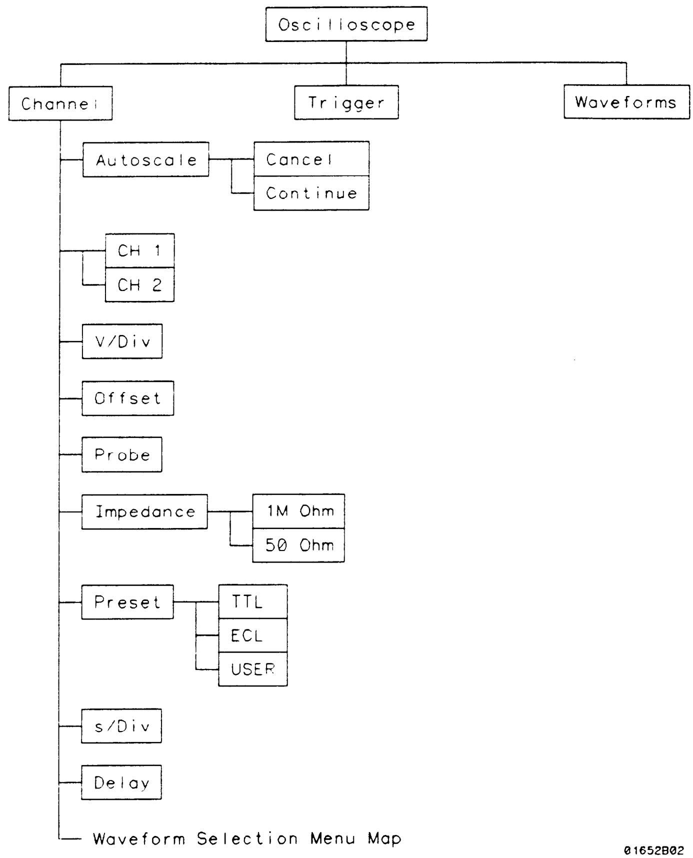
The Scope (An Overview)

The oscilloscope in the HP 1652B/1653B Logic Analyzer is a 2 channel, 400 megasample/second, 100 MHz single-shot and repetitive (repetitive single-shot) bandwidth scope. It is turned on/off from the System Configuration menu with the Channel, Trigger and Waveforms menus accessed from front panel keys. The scope can be armed by the analyzer or an external BNC.

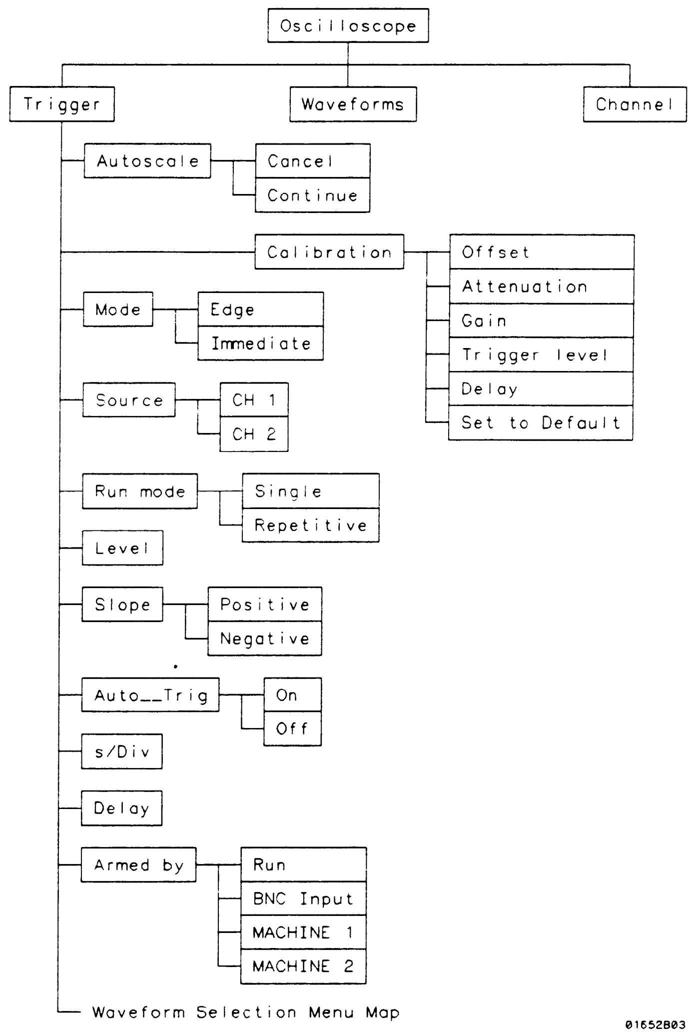
Scope Menu Maps

The scope menu maps show you the fields and the available options of each field within the three menus. The menu maps will help you get an overview of each menu as well as provide a quick reference of what each menu contains. Waveform selection is available in all main menus and is shown in the Waveform Selection Menu Map.

The Channel Menu Map

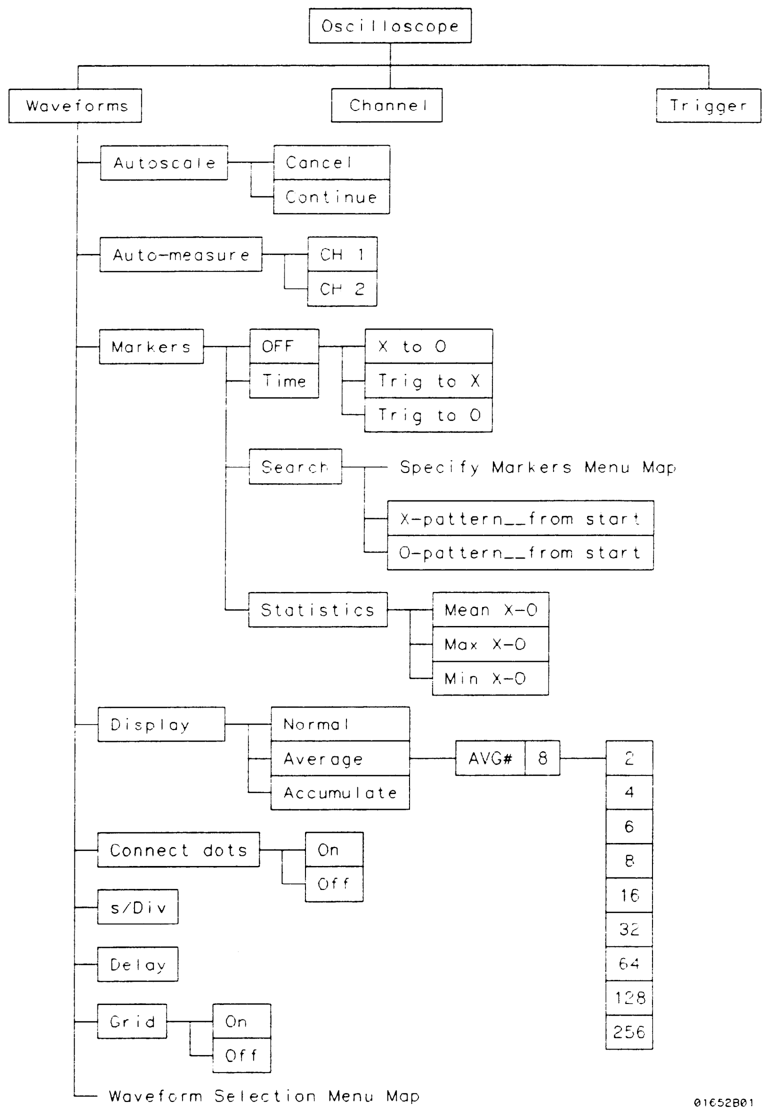


The Trigger Menu Map

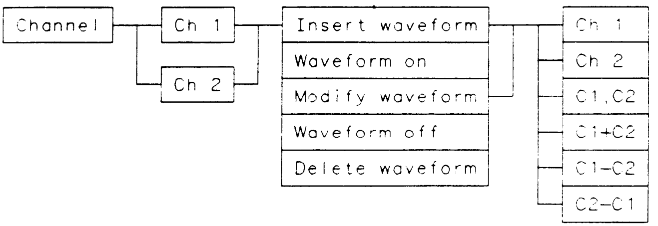


01652B03

The Waveforms Menu Map

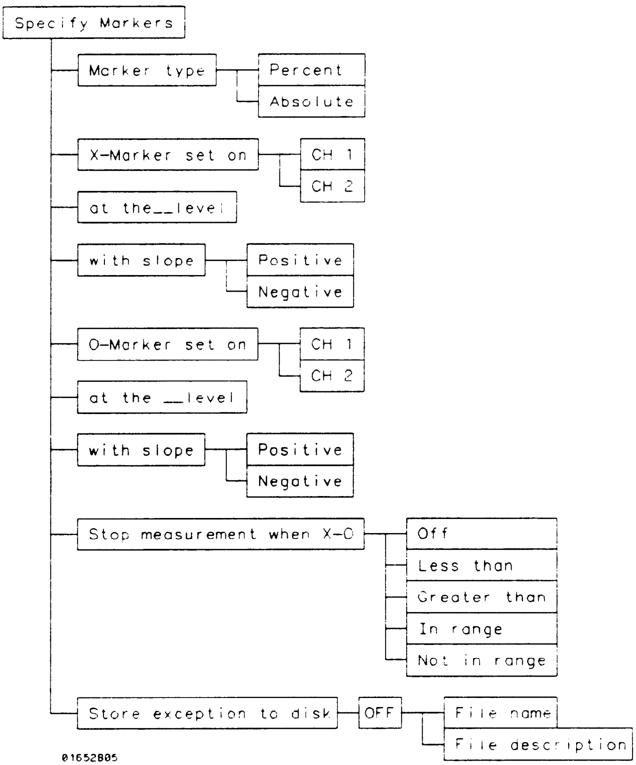


Waveform Selection Menu Map



01652B04

Specify Markers Menu Map



01652B05

Menu Overview

The oscilloscope menus are:

- Channel
  Timebase Function
  Autoscale Function

- Trigger
  Calibration Function

- Waveforms
  Auto-measure Function
  Marker Measurement Function

An illustration for each main menu is given at the beginning of each chapter. As new menu fields and pop-up fields appear, a new illustration will be shown. Use these illustrations to help in identifying the field being discussed.

The **Channel menu** (CHAN key) controls the vertical sensitivity, offset, probe attenuation, and input impedance of both input channels. In addition, the timebase functions of seconds/division (s/Div) and delay are controlled in this menu. The s/Div and delay can be controlled in the other main menus as well, but will be defined in the Channel Menu chapter.

Like the timebase functions, the autoscale function is available in all main menus, but is defined in the Channel Menu chapter.

The **Trigger menu** (TRIG key) controls the selection of trigger modes, input source, level, slope, and auto-trigger. Selection of the arming source and run mode is also controlled in this menu.

Access to the oscilloscope calibration menu is done through this menu. When the calibration field is selected, new menus will appear that guide you through the calibration process. The calibration procedure is found in Appendix D.

The **Waveforms menu** (DISPLAY key) controls the display mode, connect the dots, and grid on/off. Also included in this menu is the Auto-measure function and Marker measurement criterion set-up.

At power up, the System Configuration menu defaults the oscilloscope to on. All main oscilloscope menus can be selected from the front panel keys.

Channel Menu

Introduction

The Channel menu controls the vertical sensitivity, offset, probe attenuation factor, and input impedance of all input channels, as well as the probe attenuation factor. The Channel menu also allows you to preset vertical sensitivity, offset, and trigger level for ECL and TTL logic levels. The default Channel menu is shown below.

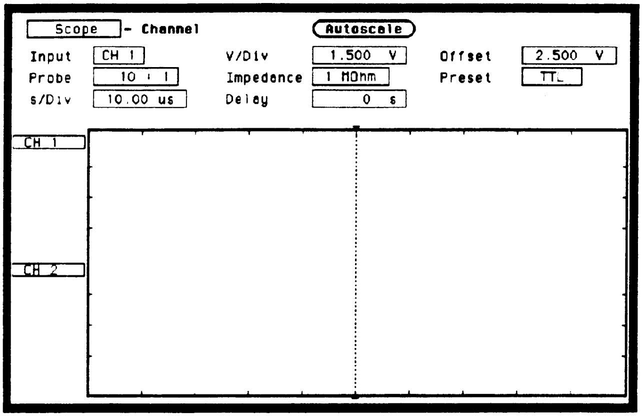


Figure 23-1. Channel Menu

Channel Menu Fields

Input Field

The **Input** field is located on the left side of the top row of fields. It selects the input source for the channel parameters displayed on the Channel Specification menu.

The default Input field selection is channel 1. When you select the Input field, it will toggle from CH 1 to CH 2.

V/Div Field

The V/Div field is located in the middle of the top row of fields. It sets the vertical sensitivity of the channel selected in the Input field. Vertical sensitivity determines the size of a waveform displayed on screen and is measured in volts/division. Each waveform display is divided into four vertical divisions. The divisions are marked by small tick marks at the left and right sides of the waveform display.

When the V/Div field is selected, a pop-up will appear which allows the vertical sensitivity to be changed by turning the knob. See upper pop-up in figure 23-2.

As the vertical sensitivity is changed, the signal expands and compresses in both directions vertically from the center of the display. When probe field is set to 10:1, the vertical sensitivity will change in a 1-2-5 sequence from 150 mV/div to 100 V/div.

Vertical sensitivity can also be entered from the keypad. A Numeric Entry pop-up will appear when the first numeric key is touched. See lower pop-up in figure 23-2.

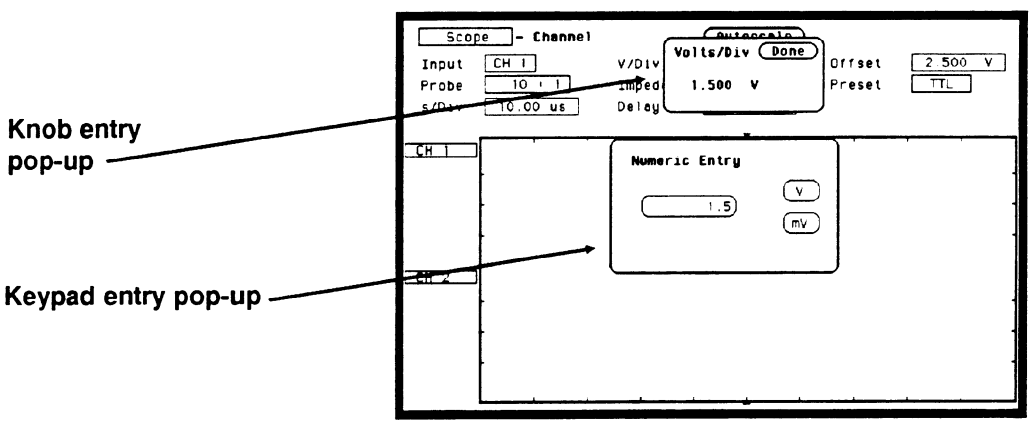


Figure 23-2. V/Div Entry Pop-ups

Any value from 150 mV/div to 100 V/div can be entered from the keypad. The vertical sensitivity value can be set to the two most significant digits. For example, if you entered a value of 154 mV, the value would be rounded to 150 mV.

The default value for the V/Div field is 1.5 V (TTL preset value).

Note

If acquisitions have been stopped, vertical sensitivity changes will not be reflected on the waveform until RUN is touched and the next acquisition is displayed.

Offset Field

The **Offset** field is located on the right side of the top row of fields. Offset is the voltage represented at the center vertical tick mark in the waveform display. Offset is a dc voltage that is added or subtracted from the input signal so that the waveform can be shown centered on the waveform display.

When the Offset field is selected, a pop-up will appear and the offset value of the channel selected in the Input field can be changed by turning the knob. See the upper pop-up in figure 23-3.

As offset is changed, and after a run, the position of the waveform moves up or down on the waveform display. Offset works similar to the vertical position control of an analog oscilloscope, but offset is calibrated.

Valid offset values can also be entered from the keypad. A Numeric Entry pop-up will appear when the first numeric key is touched. See the lower pop-up in figure 23-3.

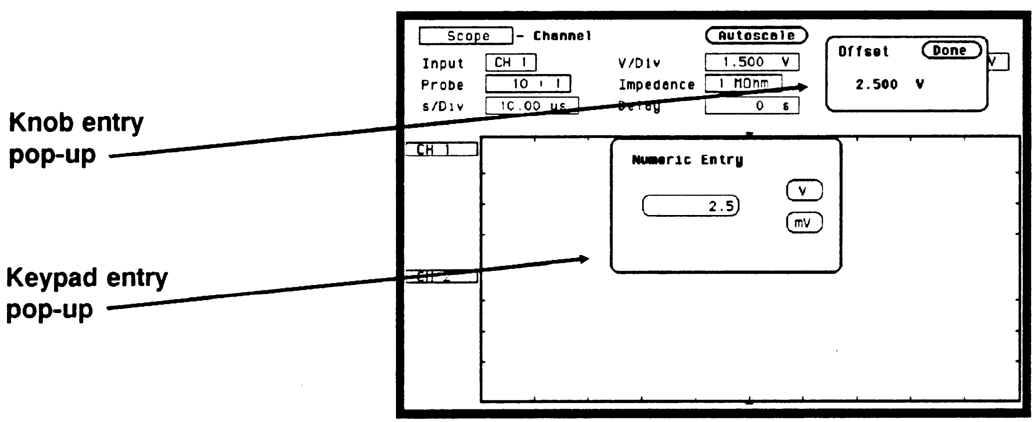


Figure 23-3. Offset Voltage Entry Pop-ups

The default value for the Offset field is 2.5 V (TTL preset value). Offset range and resolution is dependent on vertical sensitivity and input impedance. See table 23-1.

Note

If acquisitions have been stopped, offset changes will not be reflected on the waveform until RUN is touched and the next acquisition is displayed.

Probe Field

The **Probe** field is located on the left side of the middle row of fields. It sets the probe attenuation factor for the channel selected. The probe attenuation factor can be set from 1:1 to 1000:1 in increments of 1. When the Probe field is selected, a pop-up will appear and the probe attenuation factor can be changed by turning the knob. See the upper pop-up in figure 23-4.

Probe attenuation can also be entered by using the keypad. An Integer Entry pop-up will appear when the first key is touched. See the lower pop-up in figure 23-4.

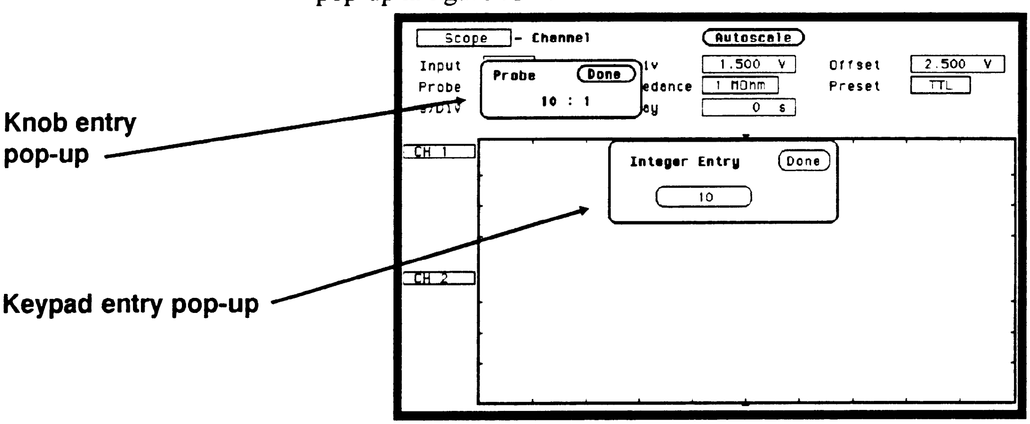


Figure 23-4. Probe Attenuation Entry Pop-ups

When you select a probe attenuation factor, the actual sensitivity at the input does not change; The voltage values used on the display (V/div, offset, marker values, trigger level, automatic measurements) are adjusted to reflect the attenuation factor.

The default value for the Probe field is 10:1 for 10:1 divider probes.

Impedance Field

The **Impedance** field is located in the middle of the middle row of fields. It sets the input impedance for the channel selected. When the Impedance field is selected, the input impedance will toggle between 1 MΩ (dc) and 50 Ω (dc). No pop-up keypad is available for this field. The default value for the Impedance field is 1 MOhm.

Preset Field

The **Preset** field is located on the right side of the middle row of fields. It automatically sets offset, V/div, and trigger level values to properly display TTL and ECL logic levels.

When you select the Preset field, a pop-up will appear as shown in figure 23-5. Rotate the knob until the proper field is highlighted, then touch the select key.

TTL
ECL
USER

Figure 23-5. Preset Field Pop-up

When you select TTL or ECL, the following values are set:

PARAMETER	PRESET VALUE	
	ECL	TTL
V/DIV	500MV	1.5V
OFFSET	-1.0V	2.5V
TRIG LEVEL	-1.300V	1.620V

01652M01

Table 23-1. Preset Value

When any of the values listed in table 23-1 are changed from the preset value, the Preset field will change to User defined. If User is selected from the pop-up, no values will be changed.

The default value for the Preset field is TTL.

Waveform Selection

This section will show you how to insert, modify, and delete input channels on the waveform display and how to perform waveform math and overlay functions. Any of these operations can be performed from any of the oscilloscope main menus.

The channel label fields to the left of the waveform display shows the input channels that are being displayed. Figure 23-6 shows the default setting which displays CH 1 and CH 2 selected.

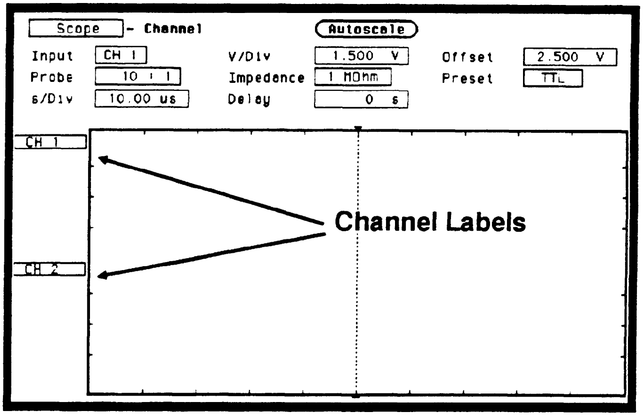


Figure 23-6. Channel Label Fields

Waveform Selection Setup

Set up the oscilloscope as described below. This instrument setup will be used throughout the remainder of this example.

Connecting the Equipment

Connect a BNC tee adapter and BNC cables to the oscilloscope as shown below.

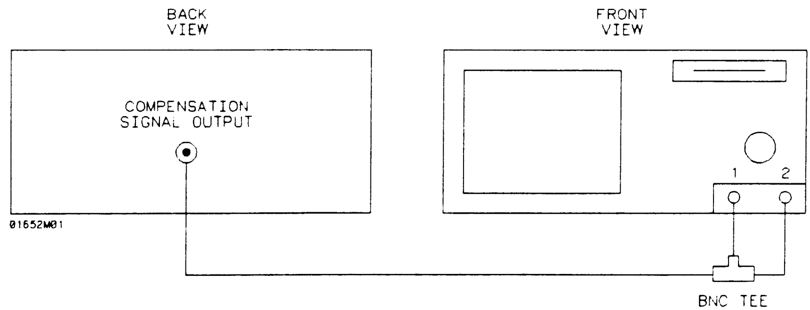


Figure 23-7. Compensation Signal Hookup

Setting Up the Oscilloscope

1. Turn the power on to the instrument.
2. From the System Configuration menu, turn the oscilloscope on and all analyzers off.
3. Touch CHAN key. From the default oscilloscope Channel menu, make the following changes:

   CH 1 Probe field to 1:1

   CH 1 Impedance field to 50 Ohms

   CH 2 Probe field to 1:1

   CH 2 Impedance field to 50 Ohms
4. Select Autoscale and set to Continue.
5. Touch the TRIG key to display the Trigger menu.

6. Select the **Run mode** field and toggle to **Repetitive**..

7. Select the Waveforms menu and toggle **Connect dots** field to **On**. The displayed waveforms should now look like figure 23-8.

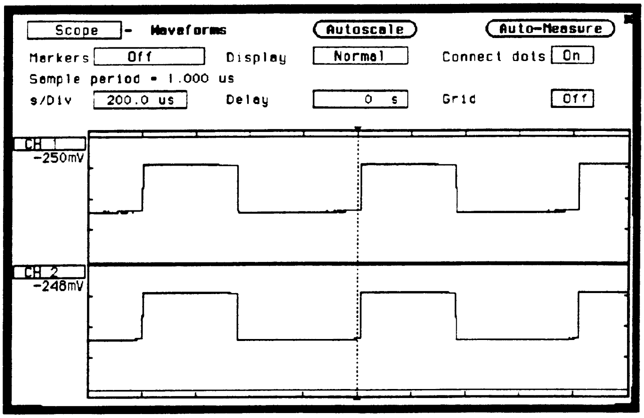


Figure 23-8. Compensation Signal Waveforms

Turning the Waveforms On/Off

The waveform selection defaults to Waveform on for channel 1 and channel 2.

1. Select the CH 1 input label. A waveform selection pop-up appears as shown in figure 23-9.

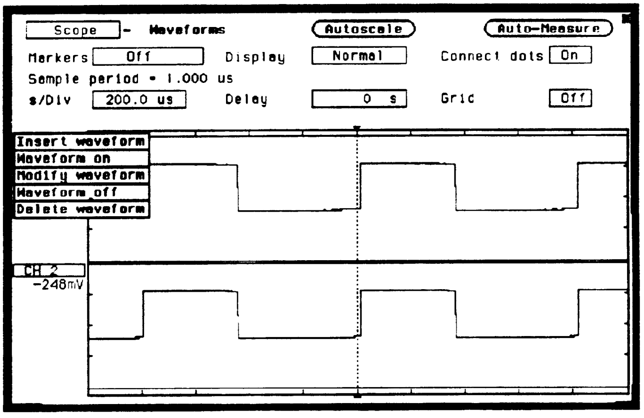


Figure 23-9. Waveform Selection Pop-up menu

2. With Waveform on, the channel 1 signal will be displayed in the waveform display. Select **Waveform off**. Channel 1 signal is now gone, and the channel label has changed to **-off-**. See figure 23-10.

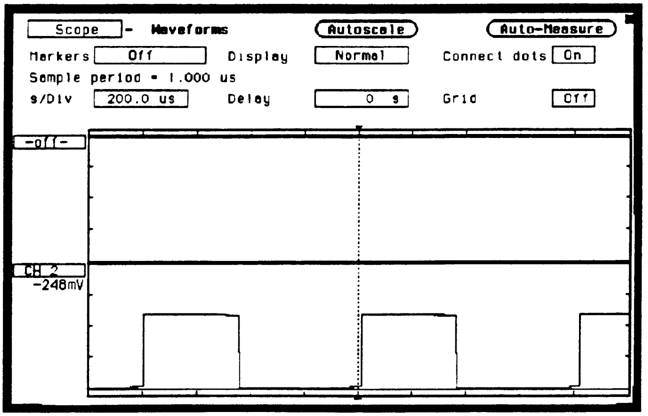


Figure 23-10. Channel 1 Turned Off

3. To turn channel 1 waveform back on, select the channel label field, then select **Waveform on.**

Insert/Delete Waveforms

When a signal is inserted into the waveform display, its label field and waveform will always be displayed directly below the highlighted label and corresponding waveform.

1. Select **CH 1** label field, then select **Insert waveform.** A channel mode pop-up will appear. See figure 23-11.

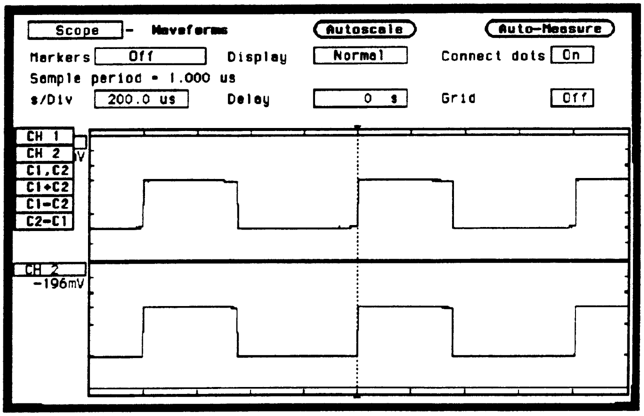


Figure 23-11. Channel Mode Pop-up

2. Select **CH 2**. Notice that the second CH 2 was inserted directly below CH 1.
3. To delete CH 2 from the channel label list, select **CH 2**.
4. Select **Delete waveform**. CH 2 is now removed and you are back to the start.

Modify Waveforms

When you modify a waveform, you select the channel label to be modified and replace it with a selection from the channel mode pop-up.

1. Select **CH 1** label field, then select **Modify waveform**.
2. Select **CH 2** from the channel mode pop-up, then touch the **RUN** key. Notice that CH 1 has been replaced with CH 2. Channel 2 is now being displayed twice.
3. Select **CH 2** label field (the same one just modified), then select **Modify waveform**.
4. Select **CH 1** from the channel mode pop-up, then touch the **RUN** key. Now you are back to the start.

Overlay (C1,C2)

What we have been displaying so far in this section are examples using just single channels. These examples display a single input channel in each waveform display.

Suppose you wanted to take the signal from CH 1 and compare it to the signal from CH 2. The easiest way to do this would be to put both waveforms on the same waveform display, or overlay the waveforms.

1. Select **CH 1**,then select **Modify waveform**. Select **C1,C2,** then touch the **RUN** key. CH 2 is now overlayed on CH 1 in the top waveform display.
2. Select the **s/Div** field and change the sweep speed to 5 ns/div. This will allow us to see the overlayed waveforms easier. The display should now look like the figure below.

Note

To get a better display of the two waveforms overlayed, use an extra long cable on one of the inputs. This will delay one waveform.

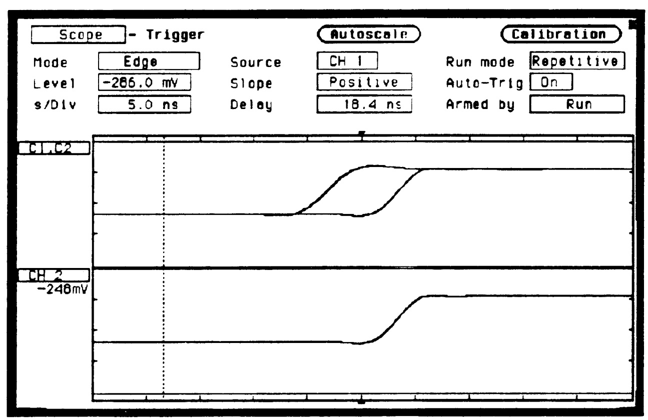


Figure 23-12. Overlay Waveform Display

Waveform Math (C1+C2), (C1-C2)

Suppose you wanted to take the signal from CH 2 and add it to or subtract it from the signal from CH 1. Let's try subtracting CH 2 from CH 1.

1. Select **C1,C2** label field, then select **Modify waveform.**

2. Select **C1-C2** field. With the s/Div still set at 5 ns, the waveform display should look like the figure below.

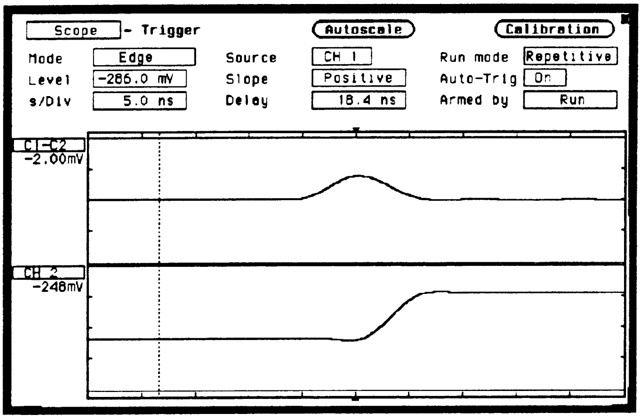


Figure 23-13. C1-C2 Waveform Display

Timebase Functions

The s/Div and Delay timebase functions control the horizontal display on the oscilloscope. The **Delay** and **s/Div** fields are located in the bottom row of fields and are displayed on all oscilloscope main menus.

Instrument Setup

The instrument should already be set up from the previous exercise. If you need to reset the menu fields, refer to that exercise or select the **Autoscale** field and set to **Continue**. Your screen should look like the figure below.

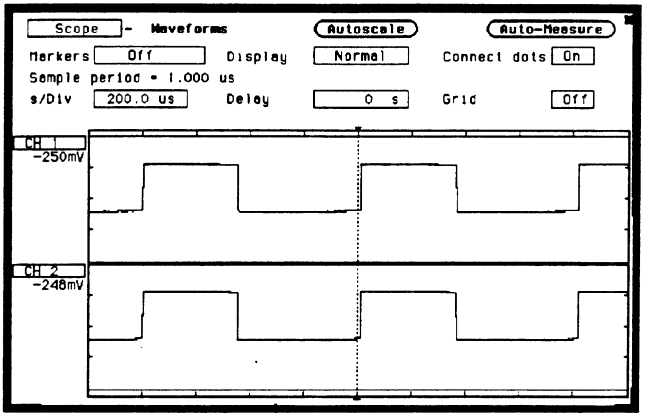


Figure 23-14. Compensation Signal Waveform

s/Div Field

The **s/Div** field sets the sweep speed or time scale on the horizontal axis of the display and is measured in seconds/division. The display is divided into 10 horizontal divisions. The divisions are marked by small tick marks at the top and bottom of the waveform display.

When the s/Div field is selected, a pop-up will appear and the sweep speed of the channel selected in the Input field can be changed by turning the knob. See the upper pop-up in figure 23-15.

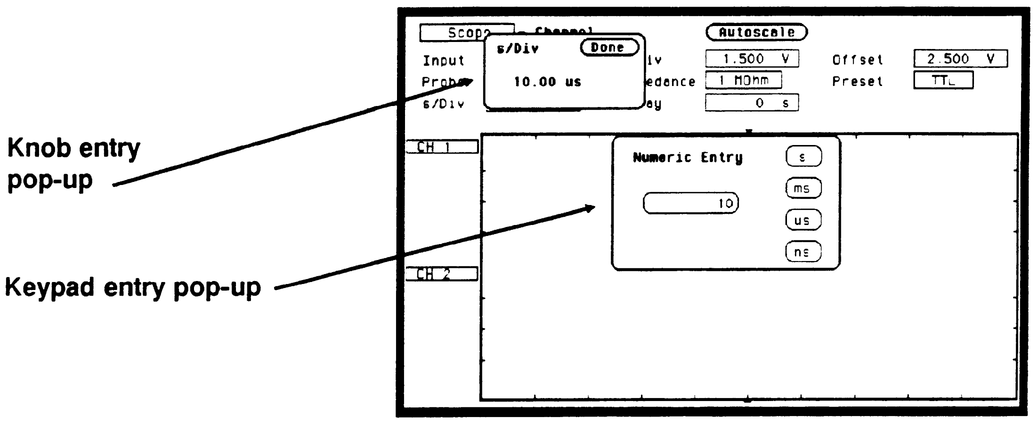


Figure 23-15. s/Div Entry Pop-ups

As the sweep speed is changed, the signal expands and compresses in both directions from the center of the display. As you turn the knob, the sweep speed changes in a 1-2-5 sequence from 5 ns/Div to 5 s/Div.

Sweep speed can also be entered from the Numeric Entry pop-up. The pop-up will appear when the first numeric key is touched. See the lower pop-up in figure 23-15.

Any value from 5 ns/Div to 5 s/Div can be enter from the keypad. Sweep speed can be set to three-digit resolution. For example, if you entered a value of 15.45 ns, the value would be rounded up to 15.5 ns.

At sweep speeds of 100 ms/div and slower, the time to acquire the 2048 sample points for acquisition memory is greater than 1 second. At these sweep speeds the screen will display "Scope waiting for prestore" when acquiring the 2048 sample points prior to the trigger point or "Scope waiting for poststore" when acquiring the 2048 sample points after the trigger point. These advisories let you know the oscilloscope is still actively acquiring data.

The default value for the s/Div field is 10 μs.

Zoom (Acquisition Stopped)

If acquisitions have been stopped, the oscilloscope uses the 2048 sample points stored in acquisition memory to display the new data on screen when the sweep speed is changed. This function would normally be used to zoom in or zoom out on a waveform acquired in Single (single-shot) mode. Zooming either expands or compresses the waveform horizontally and is changed by adjusting the s/Div field.

Zoom Example

Select the **s/Div** field and turn the knob to set the sweep speed to 200 μs/Div, then touch the **Stop** Key to stop acquisitions. Now turn the knob to change the sweep speed and notice how the acquired waveform expands and compresses.

Normally 500 points of the 2k waveform record is displayed on screen. Change the sweep speed to 500 us/div. Now all 2k of the waveform record is compressed and displayed on screen. See figure 23-16.

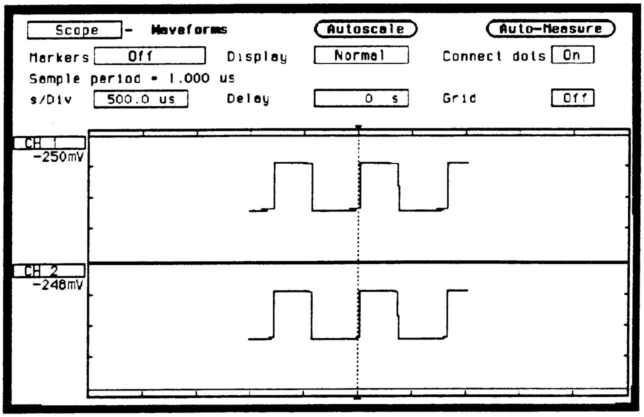


Figure 23-16. Compressed Waveform

Now change the sweep speed to 2 us/Div. At 200 us/Div, 500 points were displayed; at 2 us/Div, only 20 points are displayed. When the waveform is expanded, the oscilloscope uses a reconstruction filter to fill in the waveform points to provide a more useable display. When used in conjunction with scrolling (see "Delay Field" paragraph), zooming is very useful in displaying single-shot waveforms.

Delay Field

Delay time is the time offset before or after the trigger point on the waveform and is always measured from the trigger point to the center of the screen. The dotted line at the center of the display is the trigger point. When delay time is zero, the trigger point is at the center of the screen.

When the Delay field is selected, a pop-up will appear and the delay time can be changed by turning the knob. Remember that the trigger point is always delay time zero and is marked by the dotted line. When the trigger point moves to the right side of the screen, the delay time is negative. This means that what you are viewing at center screen is before the trigger point and is referred to as negative time.

When the trigger point is moved to the left side of the screen, the delay time is positive and what you are viewing at center screen is after the trigger point.

Delay time resolution is equal to 2% of the sweep speed setting when using the knob. When using the pop-up keypad, resolution is 100 ps at sweep speeds of 99.9 ns/Div and faster, and can be set to 4-digit resolution at sweep speeds of 100 ns/div and slower.

Scrolling (Acquisition Stopped)

If acquisitions have been stopped, the Delay field controls the portion of the acquisition memory displayed on screen.

When acquisition has been stopped:

Pre-trigger delay range = delay time setting - (1024 X sample rate)
Post-trigger delay range = delay time setting + (1024 X sample rate)

This means that one-half of data stored in acquisition memory is before the delay time setting and one-half of the data in memory is after the delay time setting.

This function would normally be used to scroll through a waveform acquired in Single (single-shot) mode. Scrolling allows you to view the entire waveform record by adjusting the Delay field.

Scroll Example

Select the **s/Div** field and turn the knob to set the sweep speed to 200 us/div, then touch the Stop key to stop acquisitions. Select the **Delay** field and turn the knob to change delay time to approximately -1 ms. As shown in figure 23-17, you are now looking at the beginning of the waveform record. You can now scroll through the entire 2k waveform record, both before and after the trigger point. When used in conjunction with zooming (see "s/Div Field" paragraph), scrolling is very useful in displaying single-shot waveforms.

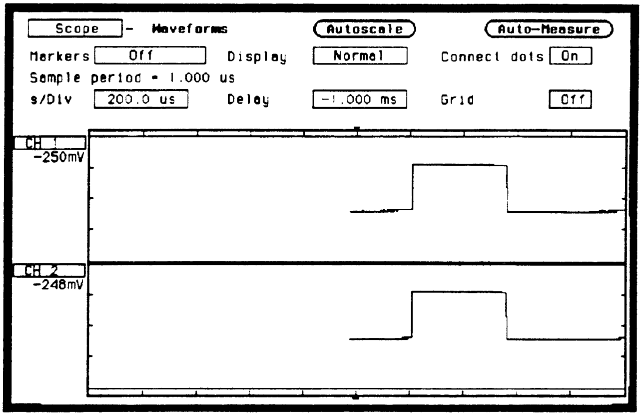


Figure 23-17. Scroll Begining of Waveform

Autoscale Field

The **Autoscale** field is located in the middle of the top row of fields in all scope main menus except when the Markers field is set to Statistics. When the Autoscale field is selected, a pop-up appears allowing you to cancel or continue the autoscale. See figure 23-18.

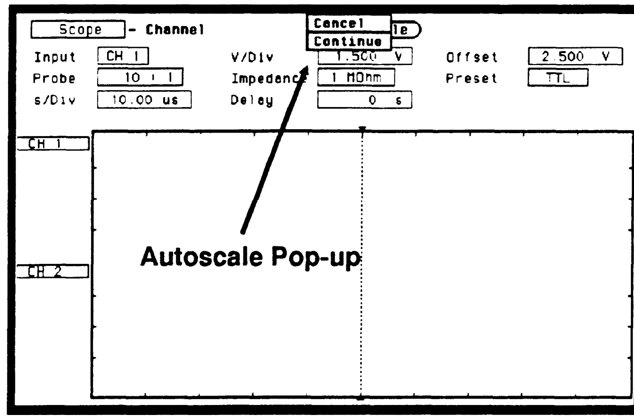


Figure 23-18. Autoscale Pop-up

If you have inadvertently selected autoscale or wish to abort the autoscale, select Cancel. If the Continue field is touched, the autoscale function is started and the advisory Autoscale is in progress is displayed. The oscilloscope automatically sets V/Div (vertical sensitivity), channel Offset, s/Div (sweep speed), and trigger Level so that the input signals are displayed on screen. The oscilloscope checks all vertical inputs and looks for the trigger on channel 1. The following fields are changed when autoscale is complete:

Channel menu

V/Div	scaled
Offset	scaled

Trigger Menu

Mode	Edge
Source	set to lowest number input with signal present
Level	scaled
Slope	positive

Any menu

s/Div
Delay

When a Signal is Found

If a signal is found on any of the vertical inputs, the oscilloscope determines the frequency of the signals and automatically scales the vertical sensitivity, offset, sweep speed, and trigger level to display the waveform on screen. The oscilloscope will normally display between 1 and 3 complete cycles of the waveform.

If a signal is present at more than one input, the trigger source is always assigned to the signal input on channel 1. This input is also used to scale the sweep speed. If only one vertical input has a signal present, that signal is the trigger source.

If No Signal is Found

If no signal is found on any of the vertical inputs, the oscilloscope displays the advisory **No signal found**, then displays **Auto Triggered**, and the oscilloscope is placed in an auto-trigger mode. The auto-trigger mode allows the oscilloscope to auto-sweep and display a baseline anytime a trigger signal is not present.

Trigger Menu

24

Introduction

The Trigger menu controls the selection of trigger modes for the oscilloscope. The Trigger menu has two modes:

- Edge
- Immediate

The default Trigger menu is shown in figure 24-1.

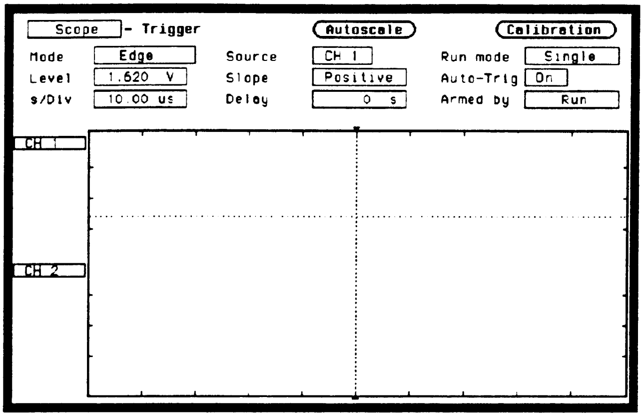


Figure 24-1. Trigger Specification Menu

Calibration

When the Calibration field is selected, a pop-up will appear showing the calibration menu. Information on when and how to calibrate the oscilloscope is found in Appendix D.

Trigger Point Marker

The trigger point marker, is the dotted vertical line at the center of the waveform display. This dotted vertical line, points to the place on the waveform where the trigger source waveform or trigger condition intersects. This point of intersection is where timebase delay is referenced. This point represents a delay time of zero seconds. See figure 24-2.

If delay time is set to greater than 5 times the sweep speed, the trigger marker will be moved off screen.

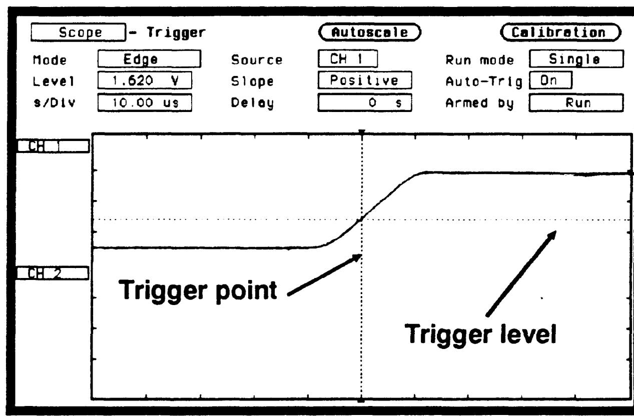


Figure 24-2. Trigger Point

Mode Field

The Mode field is located on the left side of the top row of fields and selects the trigger mode for the oscilloscope. When you select the Mode field, it will toggle between **Edge** and **Immediate.**

The default selection for the Mode field is Edge.

Immediate Trigger Mode

Immediate trigger mode causes the oscilloscope to trigger by itself after the arming requirements are met. This can be used when a logic analyzer arms the scope or another instrument arms the scope via the BNC connector. The default Immediate trigger menu is shown in figure 24-3.

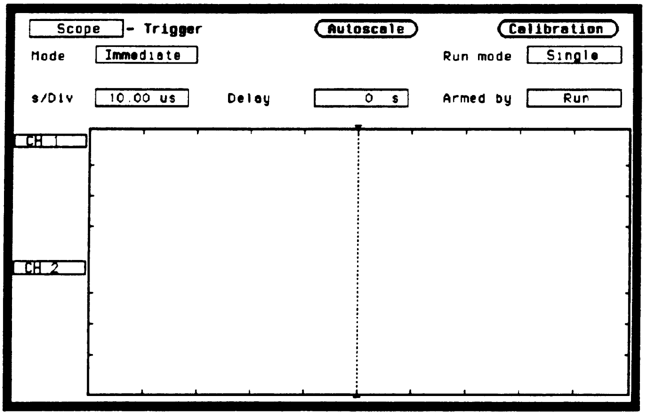


Figure 24-3. Immediate Trigger Mode Default Menu

Armed by Field

The Armed by field is located on the right side of the bottom row of fields. When selected, a pop-up will appear that is used to set any arming requirements. See figure 24-4.

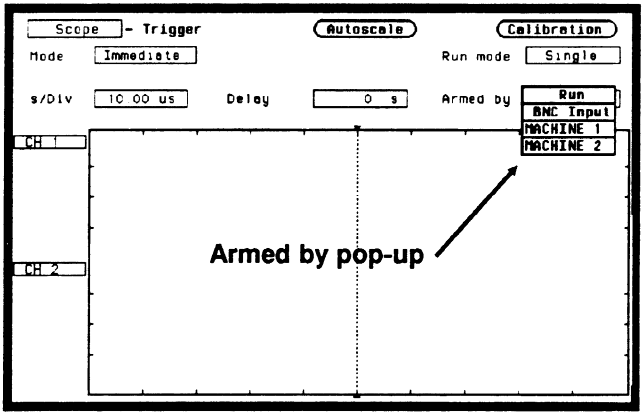


Figure 24-4. Arming Selection Pop-up

Run. If Run is selected in the Armed by field, the oscilloscope will be in the free-run mode and the waveform display will not be synchronized to a trigger point.

BNC Input. If BNC Input is selected, and the oscilloscope is in the trigger Immediate mode, it triggers and synchronizes itself as soon as it is armed by a signal from the External Trigger Input on the rear panel.

Machine 1 and 2. If Machine 1 or Machine 2 is selected, and the oscilloscope is in the trigger Immediate mode, it triggers and synchronizes itself as soon as it is armed by an internal signal from the appropriate analyzer.

The default selection for the Armed by field is Run.

Edge Trigger Mode

Edge trigger is the type of triggering found in all oscilloscopes. In edge trigger mode the oscilloscope triggers at a specified voltage level on a rising or falling edge of one of the input channels.

In this mode you can specify which input is the trigger source, set a trigger level voltage, and specify which edge to trigger on.

Source Field

The trigger Source field is located in the middle of the top row of fields and when selected, will toggle between channels 1 and 2.

The default selection for the Source field is channel 1.

Level Field

The trigger voltage Level field is located on the left side of the middle row of fields and is used to set the voltage level at which the trigger source waveform crosses the trigger marker. When the Level field is selected, a pop-up will appear which allows the trigger level to be changed by turning the knob. See the upper pop-up in figure 24-5.

When the trigger level is changed, the waveform moves on the display to maintain the trigger point (where waveform edge crosses the trigger level line). If the trigger level is set above or below the waveform, trigger is lost and the waveform display will be unsynchronized.

The trigger level, when set with the knob, can be any voltage value contained within the waveform display window in increments of 1% of full scale vertical voltage range (V/Div X 4). For example, if full scale voltage range were 500 mV, trigger level could be set in increments of 20 mV.

Trigger level can also be entered from the keypad. A Numeric Entry pop-up will appear when the first key is touched. See the lower pop-up in figure 24-5.

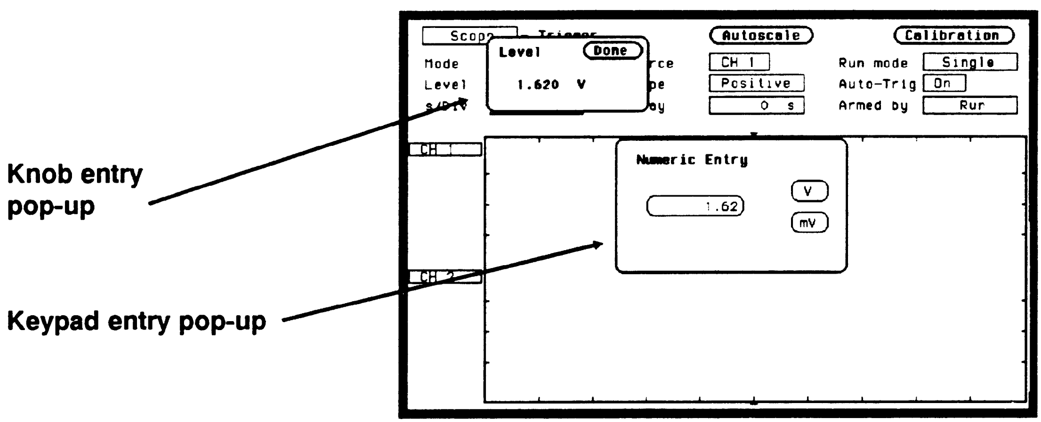


Figure 24-5. Trigger Level Entry Pop-ups

Since the trigger level range is limited by the voltage values set for the waveform window, the voltage level range can be easily determined. Turn the knob in both directions until the Level field reads minimum and maximum voltage. These voltage values are the trigger level limits of the waveform window.

The default value for the Level field is 1.62 V (TTL preset value).

Slope Field

The Slope field is located in the middle of the middle row of fields. It selects which edge of the trigger source waveform the oscilloscope will trigger on. When Slope is selected, it will toggle between positive and negative.

The default selection for the Slope field is Positive.

Auto-Trig Field

The Auto-Trig field is located on the right side in the middle row of fields. It lets you specify whether or not the acquisitions should wait for the specified trigger condition to occur. When the Auto-Trig field is touched, the field will toggle between On and Off.

On. When auto-trigger is set to on, the oscilloscope waits for approximately 1 sec. for a trigger to occur. If a trigger does not occur within that time, whatever is in the acquisition memory is displayed and "Auto triggered" is displayed:

- if no signal is on the input, the oscilloscope will display a baseline.
- if there is a signal but the specified trigger condition has not been met within 1 sec, the waveform display will not be synchronized to a trigger point.

Off. When auto-trigger is set to off, the oscilloscope waits until a trigger is received before the waveform display is updated. If a trigger does not occur, the screen is not updated and "Waiting for Trigger" is displayed.

The default selection for the Auto-Trig field is On.

Run mode Field

The Run mode field is located on the right side of the top row of fields and is displayed only when in the Trigger menu. This field controls whether the oscilloscope performs a single acquisition or multiple acquisitions. Single mode acquires a waveform on a single acquisition and then stops running. Repetitive mode acquires a waveform a multiple number of times and rebuilds the display after each acquisition. Repetitive mode keeps running until the STOP key is pressed. When powered on, the oscilloscope defaults to the Single mode. There is no pop-up for this field. When selected, it simply toggles between Single and Repetitive.

Single Mode Run. After the Run mode field is set to Single, you start the oscilloscope running in the single-acquisition mode by pressing the RUN key on the front panel. In this mode, the oscilloscope makes a single acquisition, displays the results, then waits until the RUN key is touched again before making another acquisition.

Repetitive Mode Run. After the Run mode field is set to Repetitive, you start the oscilloscope running in the Repetitive (repetitive single-shot) mode by pressing the RUN key on the front panel. In this mode, the display is rebuilt each time a new acquisition is made. When you want to stop making repetitive acquisitions, touch the STOP key on the front panel. To resume making repetitive acquisitions, touch the RUN key again.

Note

Before a repetitive run can be executed, all analyzers must either be turned off, or they must have acquired data.

25

Waveforms Menu

Introduction

The Waveforms menu controls how the oscilloscope displays the waveforms. The waveforms may be displayed in normal, averaged, or accumulated mode. This menu also controls the connect-the-dots display feature. The default Display menu is shown in figure 25-1.

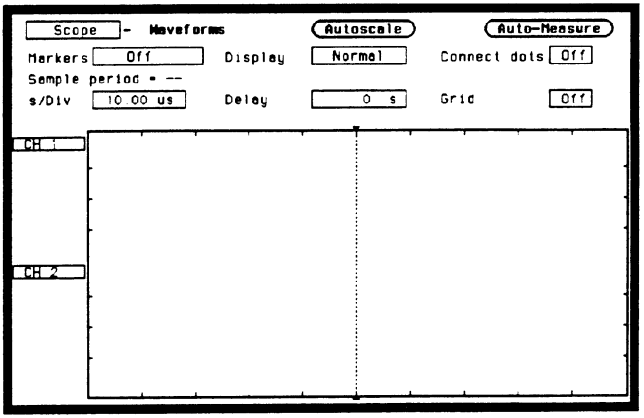


Figure 25-1. Waveforms Display Menu

Scope Field

The Scope field located in the upper left corner of the menu is used for accessing the System Configuration menu, any assigned analyzers, and the mixed mode display operation. More information on mixed mode displays and the system configuration menu is found in the appropriate chapters.

Auto-Measure Field

Automatic parametric measurements are functions built into the digitizing oscilloscope that make parametric measurements on a displayed waveform.

The Auto-measure field is located in the upper right corner of the Waveforms menu. When the Auto-measure field is selected, a pop-up will appear that lists the parameters measured for the channel selected. See Figure 25-2.

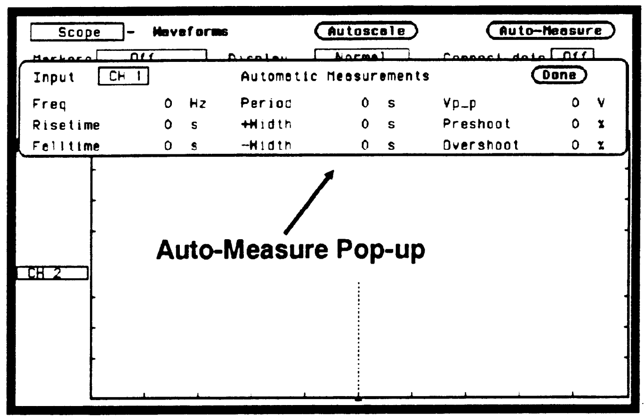


Figure 25-2. Automatic Measurement Pop-up

The channel field , which is located in the upper right corner of the pop-up, will toggle between channel 1 and channel 2 when selected. The default selection for this field is channel 1.

There are nine automatic measurements made from the data that is displayed in the waveform display:

Period
Risetime
Falltime
Frequency
+Width
-Width
Vp_p
Preshoot
Overshoot

Keep the following in mind when making measurements.

- At least one full cycle of the waveform with at least two like edges must be displayed for Period and Freq measurements.

- A complete positive pulse must be displayed to make a +Width measurement, and a complete negative pulse must be displayed to make a -Width measurement.
- Risetime, Falltime, Preshoot, and Overshoot measurements will be more accurate if you expand the edge of the waveform by selecting a faster sweep speed.

Top and Base Voltages

All measurements except Vp-p are calculated using the Vtop (100% voltage) and Vbase (0% voltage) levels of the displayed waveform. The Vtop and Vbase levels are determined from an occurrence density histogram of the data points displayed on screen.

The digitizing oscilloscope displays 6-bit vertical voltage resolution. This means the vertical display is divided up into 2^6 voltage levels. Each of these 64 levels is called a quantization level. Each waveform has a minimum of 500 data points displayed horizontally on screen. Each of these data point sets have one quantization level assigned to it. The histogram is calculated by adding the number of occurrences of each quantization level of the displayed data point sets on the displayed waveform.

The quantization level with the greatest number of occurrences in the top half of the waveform corresponds to the Vtop level. The quantization level with the greatest number of occurrences in the bottom half of the waveform corresponds to the Vbase level.

If Vtop and Vbase do not contain at least 5% of the minimum (500) data points displayed on screen, Vtop defaults to the maximum voltage (Vmaximum) and Vbase defaults to the minimum voltage (Vminimum) found on the display. An example of this case would be measurements made on sine or triangle waves.

From this information the instrument can determined the 10, 50, and 90% points, which are used in most automatic measurements. The Vtop or Vbase of the waveform is not necessarily the maximum or minimum voltage present on the waveform. If a pulse has a slight amount of overshoot, it would be wrong to select the highest peak of the waveform as the top since the waveform normally rests below the perturbation.

Automatic Measurement Example

To demonstrate how to make automatic measurements, set up the oscilloscope as described below. This setup will be used throughout the remainder of this section.

Connecting the Equipment

Connect a BNC tee adapter and BNC cables to the oscilloscope as shown in figure 25-3.

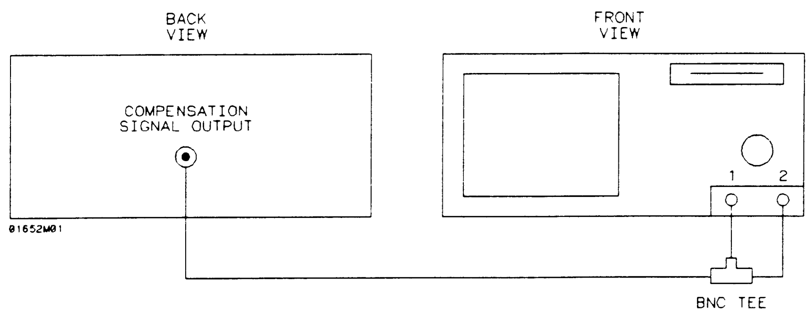


Figure 25-3. Equipment Setup

Setting Up the Oscilloscope

1. Turn the instrument on.
2. From the System Configuration menu, turn the oscilloscope on and all analyzers off.
3. Touch CHAN key and from the default Channel Specification menu, make the following changes:

   Input CH 1 Probe field to 1:1

   Input CH 1 Impedance field to 50 Ohms

   Input CH 2 Probe field to 1:1

   Input CH 2 Impedance field to 50 Ohms

4. Select **Autoscale**, then select **Continue**.

5. Touch the TRIG key and set the Run mode to Repetitive.

6. Select the Waveforms menu and toggle **Connect dots** field to on. The displayed waveform should now look like figure 25-4.

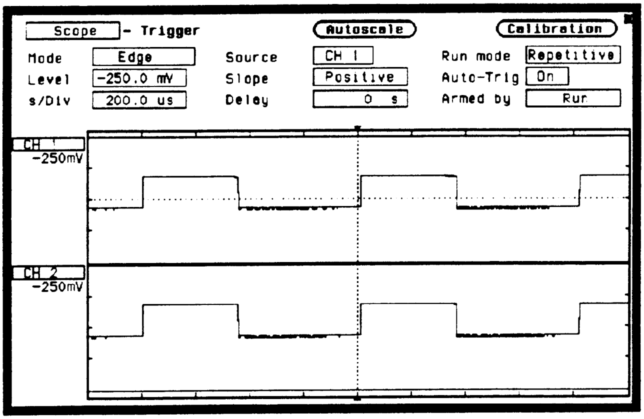


Figure 25-4. Autoscaled Waveform

Rise Time Measurement

Risetime is measured on the positive-going edge of the waveform and is the time it takes the waveform to transition between the 10% voltage point and the 90% voltage point.

1. Select the **s/Div** field and change the sweep speed to 5 ns/div. The expanded waveform should look like the figure below.

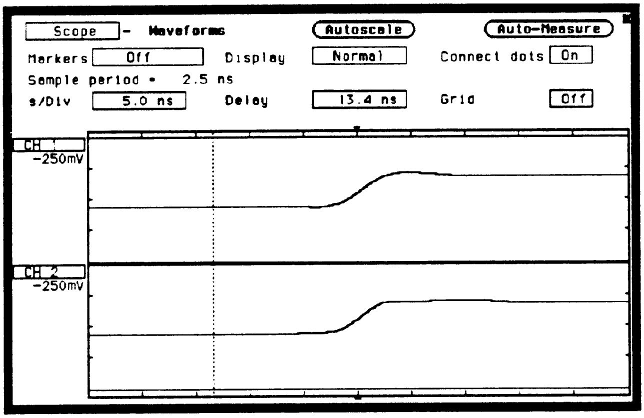


Figure 25-5. Expanded Waveform

Expanding the edge on the waveform will give more accurate results because more data points on the rising edge will be displayed.

2. From the Waveforms menu select the **Auto-Measure** field. The Risetime value is displayed in the Automatic Measurements listing pop-up.

Notice that Period, Falltime, Freq, +Width, and -Width listings are blank. Because only the rising edge of the waveform is displayed, there is insufficient data at this time to make these measurements.

3. Select **Done**. You are now back to the Waveform menu.

Fall Time Measurement

Fall time is measured on the negative-going edge of the waveform and is the time it takes the waveform to transition between the 90% voltage point and 10% voltage point. You are currently displaying the positive-going edge of the waveform, so you need to change it to the negative-going edge.

4. Touch the TRIG key and toggle the **Slope** field in the Trigger Specification menu to Negative. Notice the negative-going edge of the waveform is now displayed as shown in figure 25-6.

5. Touch the Display key and select the **Auto-Measure** field in the Waveforms menu. The Falltime value is now displayed in the Automatic Measurement listing pop-up.

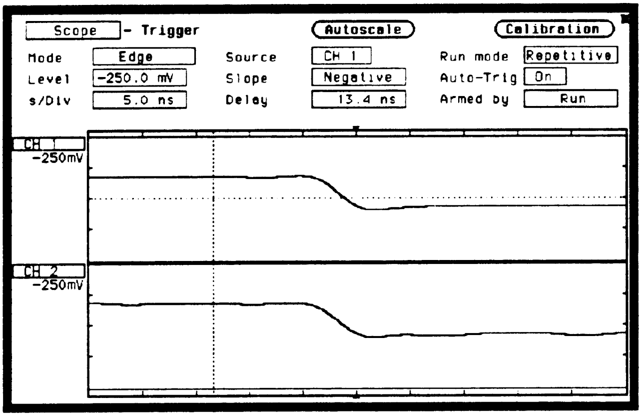


Figure 25-6. Negitive Edge Of Waveform

6. Now measure the falltime on CH 2. Select the **Input** field in the Automatic Measurement listing pop-up and toggle to CH 2.

7. All readings in the Automatic Measurement listing pop-up are now for CH2. Falltime is now being measured on CH 2.

Vp-p Measurement

The peak-to-peak voltage measurement uses the maximum voltage and the minimum voltage values found in the data displayed on screen.

Vp-p = Vmaximum - Vminimum

8. Notice the Vp_p measurement displayed in the Automatic Measurements listing pop-up. This reading is for the channel currently selected.

9. Select **Done** to exit the Automatic Measurements listing pop-up.

Period and Frequency Measurements

Period and Freq (frequency) measurements are made using the first two like edges of an input displayed on screen. At least one full cycle of the waveform must be displayed to make the measurements. If a full cycle is not present, the Period and Freq measurements in the auto-measure field will be blank. Period and Freq are measured using the time (t) at the 50% level of the edges.

If the first edge on the display is rising then:

$$\text{Period} = t_{\text{rising edge 2}} - t_{\text{rising edge 1}}$$
$$\text{Freq} = 1/(t_{\text{rising edge 2}} - t_{\text{rising edge 1}})$$

If the first edge on the display is falling then:

$$\text{Period} = t_{\text{falling edge 2}} - t_{\text{falling edge 1}}$$
$$\text{Freq} = 1/(t_{\text{falling edge 2}} - t_{\text{falling edge 1}})$$

Re-scale the waveform to display at least one full cycle of the waveform, then make a Period and Freq measurement on CH 1.

1. From the Waveform menu, select **s/Div** and change s/Div field to 100 μs to display only one full cycle of the waveform. See figure 25-7.

2. Select **Auto-Measure** and notice the Period and Freq measurements displayed in the Automatic Measurement listing pop-up.

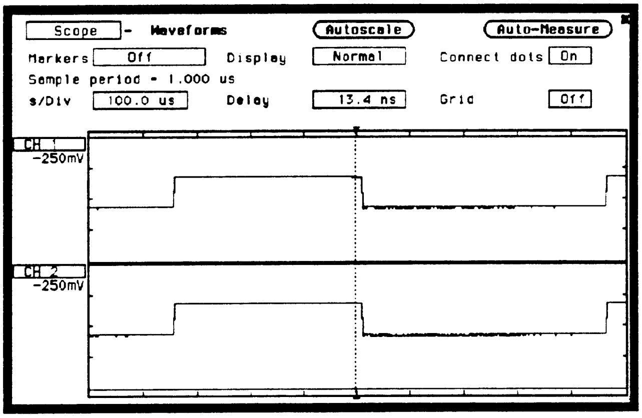


Figure 25-7. One Full Cycle Waveform

+Width and -Width Measurements

+Width (positive pulse width) and -Width (negative pulse width) measurements are made using the time (t) at the 50% level of the waveform edges. At least one positive-going edge followed by a negative-going edge of the waveform must be present to make a +Width measurement and at least one negative-going edge followed by a positive-going edge must be present to make a -Width measurement. If these conditions are not present, the +Width and/or -Width measurements in the auto-measure field will be blank.

If the first edge on the display is rising then:

$$+\text{Width} = t_{\text{falling edge 1}} - t_{\text{rising edge 1}}$$
$$-\text{Width} = t_{\text{rising edge 2}} - t_{\text{falling edge 1}}$$

If the first edge on the display is falling then:

$$+\text{Width} = t_{\text{falling edge 2}} - t_{\text{rising edge 1}}$$
$$-\text{Width} = t_{\text{rising edge 1}} - t_{\text{falling edge 1}}$$

3. Be sure the displayed waveform is at least one full cycle. If not, touch TRIG key, then select **s/Div**. Change s/Div field to 100 μs to display only one full cycle of the waveform. See figure 25-7.

4 Touch Display key, then select **Auto-Measure**. The Period and Freq measurements displayed in the Auto-measure listing pop-up is for the channel selected.

Preshoot and Overshoot Measurements

Preshoot and Overshoot measure the perturbation on a waveform above or below the top and base voltages (see "Top and Base Voltages" section earlier in this chapter). These measurements use all data displayed on screen, therefore it is very important that only the data of interest be displayed. If you want to measure preshoot and overshoot on one edge of a waveform, then only display that edge. If you want to measure the maximum preshoot and overshoot on a waveform, then display several cycles of the waveform.

Preshoot is a perturbation before a rising or a falling edge and is measured as a percentage of the top-base voltage. Overshoot is a perturbation after a rising or a falling edge and is measured as a percentage of the top-base voltage

If the measured edge is rising then:

$$\text{Preshoot} = \left[\frac{\text{Vbase - Vminimum}}{\text{Vtop-base}}\right] \times 100$$

$$\text{Overshoot} = \left[\frac{\text{Vmaximum - Vtop}}{\text{Vtop-base}}\right] \times 100$$

If the measured edge is falling then:

$$\text{Preshoot} = \left[\frac{\text{Vmaximum - Vtop}}{\text{Vtop-base}}\right] \times 100$$

$$\text{Overshoot} = \left[\frac{\text{Vbase - Vminimum}}{\text{Vtop-base}}\right] \times 100$$

Re-scale the waveform to display a rising edge, then make a Preshoot and Overshoot measurement on CH 1.

1. Select **s/Div** and change s/Div field to 5 ns to display a rising edge on the waveform. See figure 25-8.

2. Select **Auto-Measure.** Notice the Preshoot and Overshoot measurements displayed in the Automatic Measurement listing pop-up.

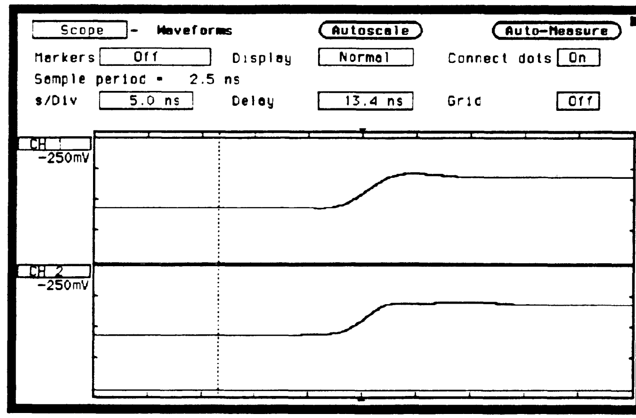


Figure 25-8. Rising Edge on Waveform

Marker Measurements

In addition to automatic parametric measurements, the oscilloscope also has two markers for making time and voltage measurements either manually (Time) or automatically (Search).

To demonstrate how to make marker measurements, connect the oscilloscope as shown in figure 25-3 and reset the oscilloscope as described below.

1. Turn the instrument on.
2. From the System Configuration menu, turn the oscilloscope on and all analyzers off.
3. Touch CHAN key, then from the default Channel Specification menu, make the following changes:

   Input CH 1 Probe field to 1:1

   Input CH 1 Impedance field to 50 Ohms

   Input CH 2 Probe field to 1:1

   Input CH 2 Impedance field to 50 Ohms

4. Select **Autoscale**, then select **Continue**.

5. Touch the TRIG key and set the Run mode to Repetitive.
6. Select the Waveforms menu and toggle **Connect dots** field to on. The displayed waveform should now look like figure 25-9.

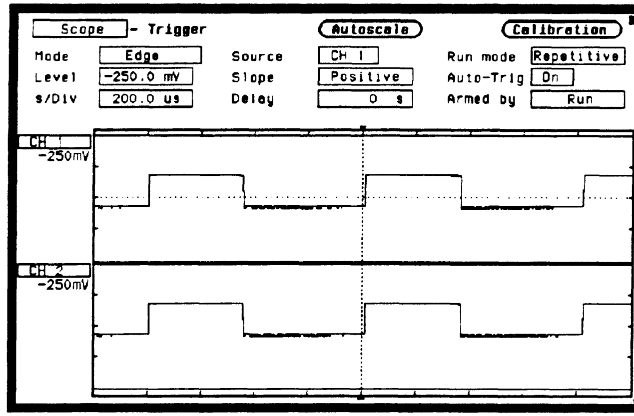


Figure 25-9. Marker Measurement Setup Display

Markers Field

In the Waveforms menu, the **Markers** field is located on the left side of the top row of fields, and can only be accessed from the Oscilloscope's Waveforms menu. When the Markers field is selected, a pop-up appears as shown in figure 25-10.

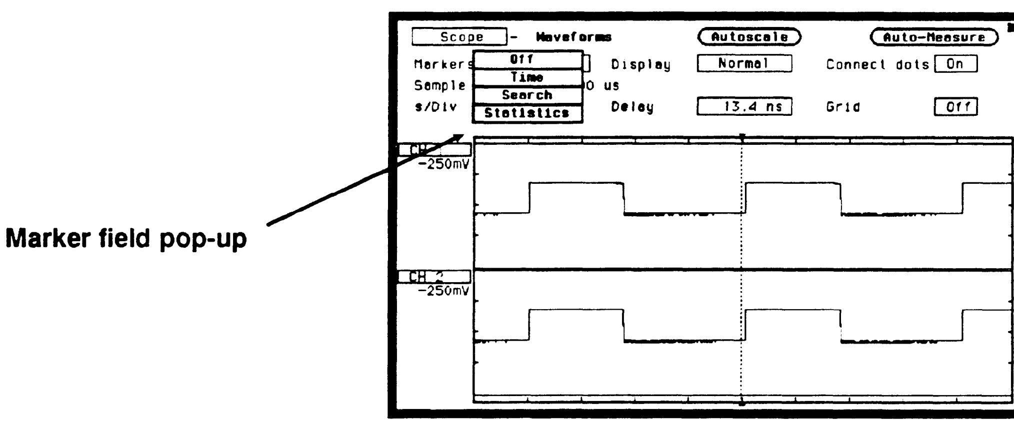


Figure 25-10. Markers Field Pop-up

The default selection for the Markers field is Off.

Sample Period Display

Any time the Markers field is Off, the sample period of the acquired waveform is displayed directly below the markers field. A sample period is the time period between acquired sample points and is the inverse of sample rate (digitizing rate). Sample period is a function of sweep speed and can only be changed by changing the s/Div field.

Time

When the **Time** field is selected from the Markers pop-up, the **Sample period =__** disappears and a new middle row of fields appear in the Waveforms menu. See figure 25-11.

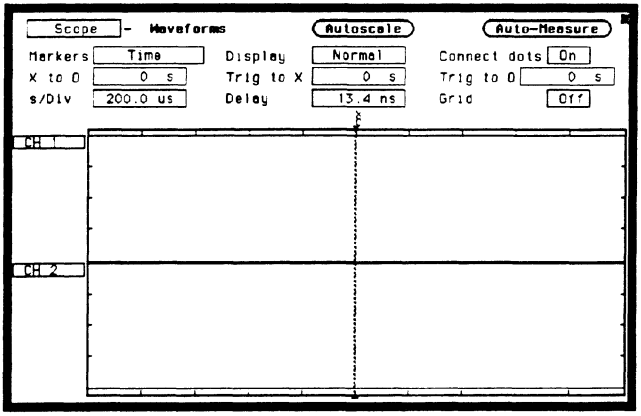


Figure 25-11. Time Markers Field Pop-up

X to O Field

The **X to O** field is located on the left side of the middle row of fields and displays the time (delta time) between the X marker and the O marker. When the X to O field is selected, turning the knob will move both the X and the O marker across the display without changing the value in the X to O field. However, the values in the Trig to X and Trig to O fields will change to reflect the movement of the X and O markers.

The value in the X to O field can be changed by changing the Trig to X or Trig to O values, or by changing the X to O value from the pop-up. The knob entry pop-up will appear when you select the X to O field.

When the time value of X to O is changed using the knob, half the difference of the new value and old value is subtracted from the X marker and half is added to the O marker.

Trig to X Field

The Trig to X field is located in the middle of the middle row of fields. The X marker is shown on the waveform display as a dotted line with an X above it. The time displayed in the Trig to X field is measured from the trigger point to the X marker. The trigger point is marked with a dotted line on the waveform display and is always time 0.

When the Trig to X field is selected, the time value can be changed by turning the knob or by entering a time value from the Numeric Entry pop-up. The Numeric Entry pop-up will appear when any key is touched on the keypad.

Resolution for the Trig to X time values is 2% of the sweep speed setting. The default value for the Trig to X field is 0 s.

Trig to O Field

The Trig to O field is located on the right side of the middle row of fields. The O marker is shown on the waveform display as a dotted line with an O above it. The time displayed in the Trig to O field is measured from the trigger point to the O marker.

When the Trig to O field is selected, the time value can be changed by turning the knob or by entering a time value from the Numeric Entry pop-up. The pop-up will appear when any key is touched on the keypad.

Resolution for Trig to O time values is 2% of the sweep speed setting. The default value for the Trig to O field is 0 s.

Search

When **Search** is selected from the Markers field, a new middle row of fields and the **Specify Search Markers** field will appear. See figure 25-12.

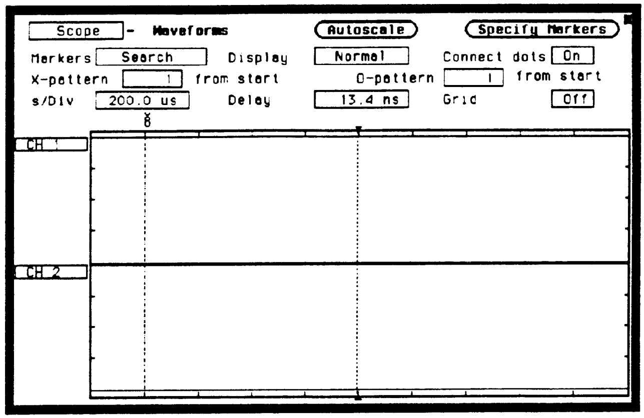


Figure 25-12. Search fields

Specify Search Markers

The **Specify Search Markers** field is located in the upper right corner of the Waveforms menu. When selected, a Search Markers pop-up will appear. After Search Markers criteria is set, the X and O markers will be positioned on the waveform, as specifed, with voltage values displayed below the waveform label. See figure 25-13.

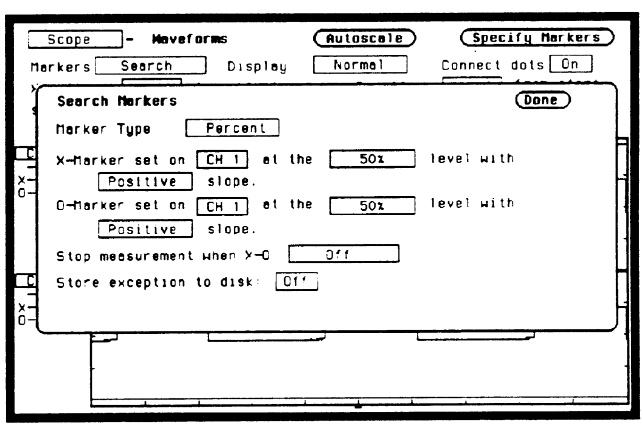


Figure 25-13. Search Markers Pop-up

Type markers

When the **Type markers** field is selected, it will toggle between Percent and Absolute.

The **Percent** type setting is for levels that are a percentage of the top-to-base voltage value of a waveform. The top-to-base voltage value of a square wave is typically not the same as the peak-to-peak value. The oscilloscope determines the top and base voltages by finding the flattest portions of the top and bottom of the waveform. See figure 25-14. The top and base values do not typically include preshoot or overshoot of the waveform. The peak-to-peak voltage is the difference between the minimum and maximum voltage found on the waveform.

The **Absolute** type allows you to set an exact voltage level to the X or O marker.

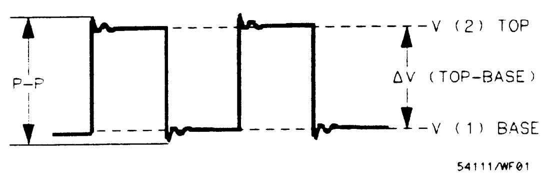


Figure 25-14. Top And Base Levels

X-Marker set on
O-Marker set on

The X-O **Marker set on__** field assigns an input waveform (CH1 or CH2) to the X or O marker. When you select this field, the field will toggle between the waveform sources.

The default selection for the **Marker set on__** field is CH1.

at the ___ level

When the type marker is set to Percent, the **at the___level** field sets the X or O marker to a percentage level (from 10% to 90%) of the top-base voltage. When the type marker is set to Absolute, you can set the marker to an exact voltage level.

The Percentage or Absolute voltage can be changed by turning the knob or by entering a value from the keypad. Percent values from 10% to 90% in increments of 1% can be entered. Absolute voltage values can be entered in increments of 6 mV. The Percent default value for the **at the___level** field is 50%. The Absolute default value for the **at the __level** field is 0 V.

with ___ slope.

The **with___ slope** field sets the X or O marker on either the positive or negative edge of the selected occurrence of a waveform. When the slope field is selected, the slope will toggle between Positive and Negative. The default selection for the Slope field is Positive.

Stop measurement when X-O ___

This field lets you set up a stop condition for the time interval between the X marker and O marker. When this field is selected, a pop-up will appear as shown in figure 25-15.

The default selection for the **Stop measurement when X-O** field is Off.

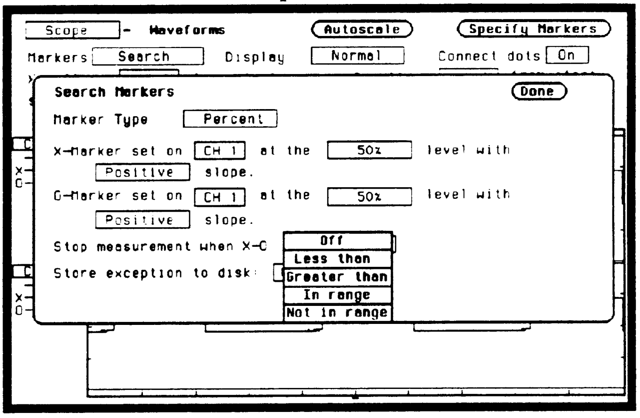


Figure 25-15. Stop Measurement Pop-up

Less than

When the **Less than** field is selected from the pop-up, a time value field appears to the right of the **Less than** field. See figure 25-16.

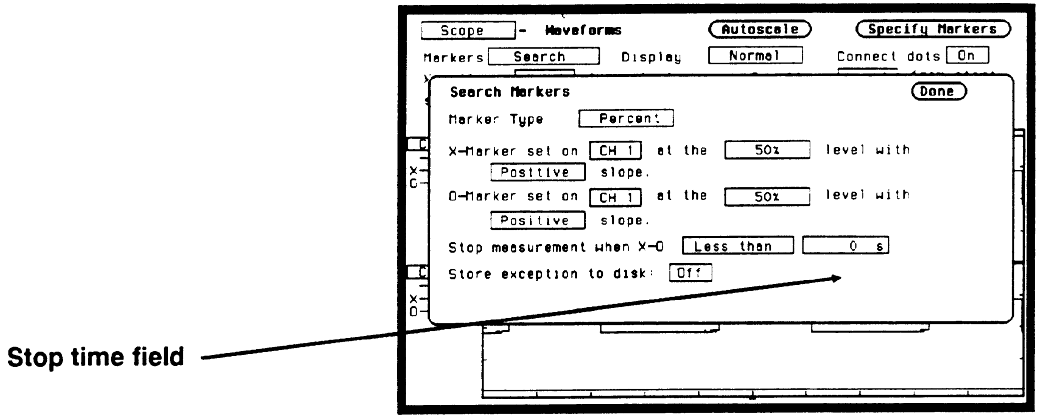


Figure 25-16. Stop Measurement Time Field

When the time value field is selected, the time value can be entered by the keypad. See figure 25-17.

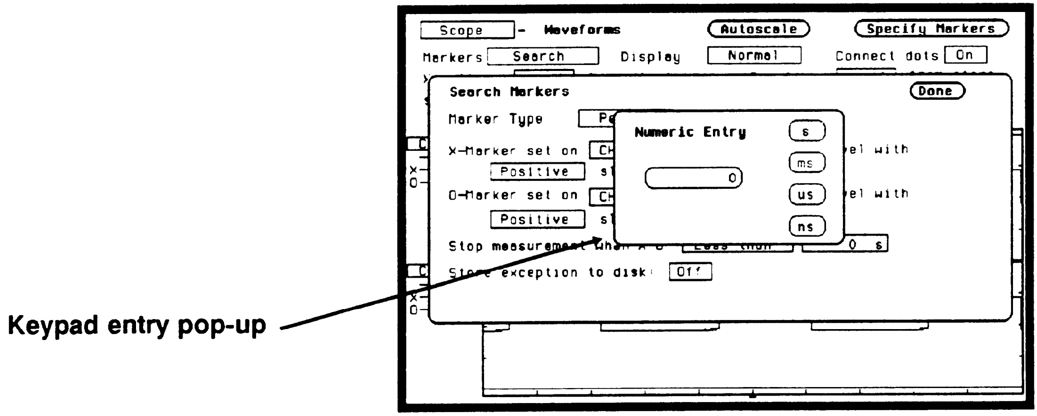


Figure 25-17. Stop Measurement Time Numeric Entry

The keypad will appear when you touch any key. The knob is used to set the scale. When using the keypad, resolution is 10 ns at times up to 99.99 ns and can be set to 5-digit resolution for other times up to 100 Megaseconds. Positive times would be used when the X marker is displayed before the O marker, and negative times would be used when the O marker is displayed before the X marker.

When **Less Than** is selected, the oscilloscope will run until the X-O time interval is less than the value entered for the **Less Than** time field. When the condition is met, the oscilloscope will stop acquisitions and display the message "Stop condition satisfied."

Greater than

When the **Greater than** field is selected from the pop-up, a time value field appears to the right of the **Greater Than** field. When the time value field is selected, the time and scale can be entered the same as for the Less Than field.

When **Greater Than** is selected, the oscilloscope will run until the X-O time interval is greater than the value entered for the **Greater Than** time field. When the condition is met, the oscilloscope will stop acquisitions and display Stop condition satisfied.

In range ___ to___

When the **In range__to__**field is selected from the pop-up, two time value fields appear next to the In range field. See figure 25-18. The time range for the stop condition is entered using the keypad. When either time value field is selected, the time value can be entered the same as for the **Less Than** field. Default values for **In range__to__** is 10 ns.

When **In range__to__** is selected, the oscilloscope will run until the X-O time interval is in the range of the time values entered for the **In range__to__** time fields. When the condition is met, the oscilloscope will stop acquisitions and display Stop condition satisfied.

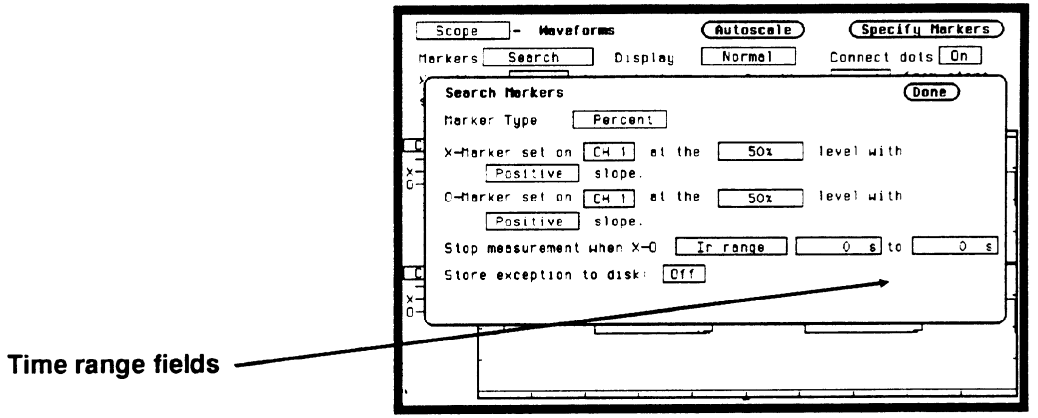


Figure 25-18. Range Fields

Not in range ___ to ___

When the **Not in range__to__** field is selected from the pop-up, two time value fields appear next to the Not In Range field. The time range for the stop condition is entered in these time fields. When either time value field is selected, the time can be entered the same as for the **Less Than** field.

When **Not In range__ to__**is selected, the oscilloscope will run until the X-O time interval is not in the range of the time values entered for the **Not in range__to__** time fields. When the condition is met, the oscilloscope will stop acquisitions and display Stop condition satisfied.

Default values for these fields are 10 ns.

Store exception to disk

When the **Store exception to disk:** field is on, any time the Stop measurement when X-O criterion is met, the measurement is stored to a file on disk. You can designate a **File name** and add a **File description** by selecting those fields and using the Alpha Entry pop-up. After the measurement is stored, the acquisition cycle continues. If the disk is write protected, a notice is displayed and the acquisition cycle is stopped. If the **Stop measurement when X-O** field is off, the Store exception to disk: function is disabled.

X-pattern ___ from start
O-pattern ___ from start

The X and O **pattern__from start** field sets the X or O markers on a specific occurrence of a edge on the waveform. The edge may be the 1st displayed up to the 1024th displayed. The count of edge occurrences is made starting with the first edge displayed on screen, either partial or full.

Note

Auto-marker measurements are made with data that is displayed on screen. Make sure the data of interest is fully displayed on screen. For example, if only part of a positive edge is displayed, the 0% point and 100%-point of the edge is calculated from what is actually displayed on screen. This could cause measurement errors.

When the X or O **pattern__ from start** field is selected, the occurrence can be changed by turning the knob or by entering a new value from the pop-up keypad. The keypad will appear when you make the first entry on the keypad. Any number from 1 to 1024 in increments of 1 can be entered.

The default value for the **pattern__ from start** Occur field is 1.

Statistics

The last field in the Markers pop-up shown in figure 25-10 is **Statistics.** When Statistics is selected from the Markers field, a new middle row of fields appear. See figure 25-19.

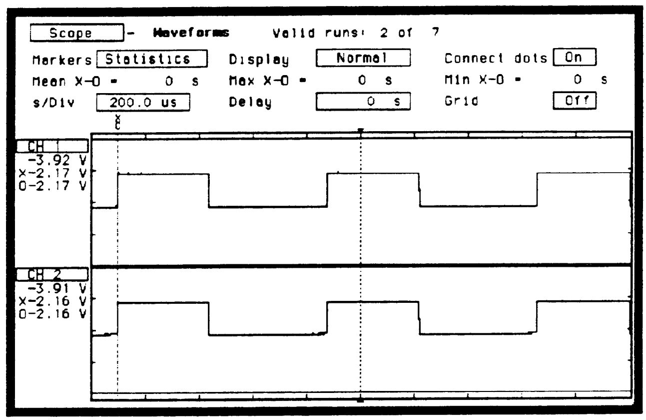


Figure 25-19. Statistics Fields

The Statistics field allows you to make minimum, maximum, and mean time interval measurements from marker X to marker O.

Minimum, maximum, and mean (average) X-O marker time interval data is accumulated and displayed until one of the following happens:

1. Autoscale is executed.
2. Auto-marker parameters are changed.
3. Statistics is set to Off.
4. Repetitive Run mode is stopped.

The default for the Min, Max, and Mean fields is 0 s.

Search Marker Measurement Example

The following example will show how to make an automatic marker measurement using the Search markers. We will set the markers to make an X-O marker measurement on the CH 1 and CH 2 input waveforms. We want to measure the time between the falling edge of the 2nd displayed pulse on CH 1 to the rising edge of the 5th displayed pulse on CH 2. We'll perform the measurement from the 10% point on CH1 to the 90% point on CH 2.

Connecting the Equipment

Connect the equipment as shown in figure 25-20.

Note

An extra long BNC cable is used on channel 2 so the signal is delayed.

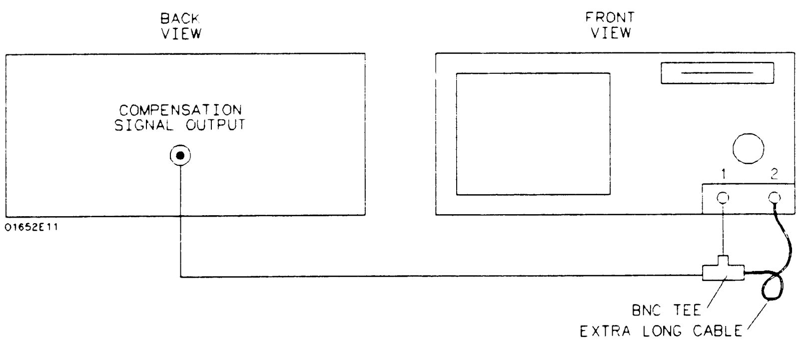


Figure 25-20. Equipment Setup For Search Marker Example

Making the Measurement

1. In the Waveforms menu, select **Markers** field. When the pop-up appears, select the **Search** field. With Markers set to Search, two new fields will appear: **X-pattern___from start**, and **O-pattern from start.** Set the X-pattern to 2, and the O-pattern to 5. Set the s/Div to 500 μs. See figure 25-21.

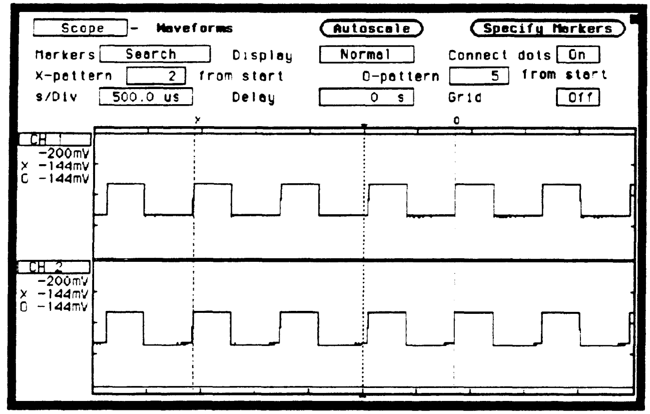


Figure 25-21. Search Markers Menu

2. Select **Specify Search Markers.** Set the Search Markers pop-up as shown in figure 25-22.

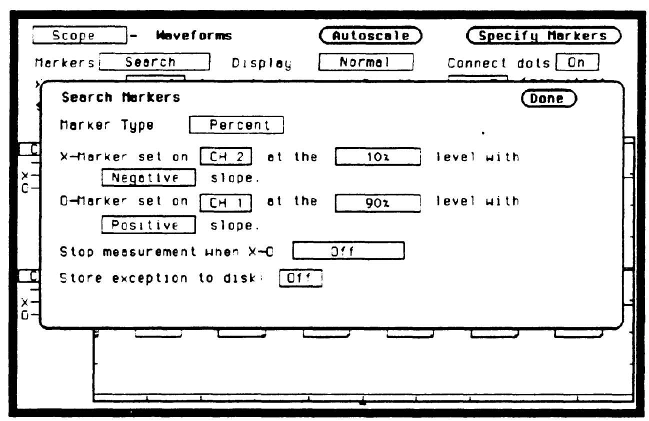


Figure 25-22. Specify Search Markers Pop-up

3. Select the **Done** field to return to the waveform display.

The X marker is on the falling edge of the 2nd displayed pulse and the O marker is on the rising edge of the 5th displayed pulse. See figure 25-23.

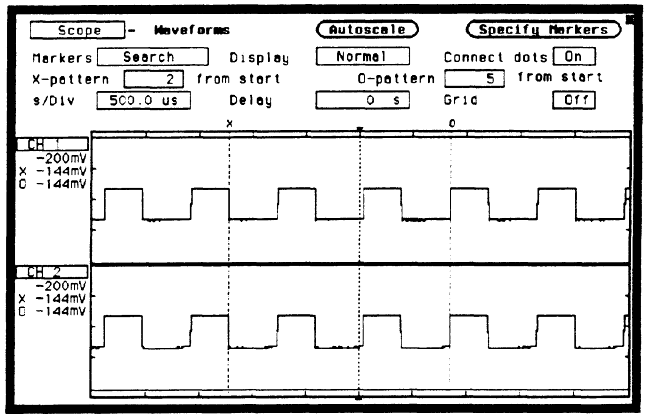


Figure 25-23. Search Marker Measurement Waveform

4. Select the **Markers** field and switch from Search to Statistics. Notice the statistical minimum, maximum, and mean time interval measurements between the X and the O markers are also displayed. The voltage measurements of where the markers intersect the waveforms are displayed under the CH 1 and CH 2 channel labels.

5. Select the **Delay** field, and turn the knob. Notice that as the waveform is moved across the display, the X and O markers also move to the edges specified in the Search Markers pop-up menu.

6. Set the Delay time back to 0 s.

Now let's use the **Stop measurement when X-O** feature for an X-O measurement.

1. Select the **s/Div** field and turn the knob to change the sweep speed to 5 ns.
2. Select the **Markers** field and select Search from the pop-up.
3. Set X-pattern and O-pattern to 1 from start.
4. Select **Specify Search Markers** and set Pop-up as shown in figure 25-24.

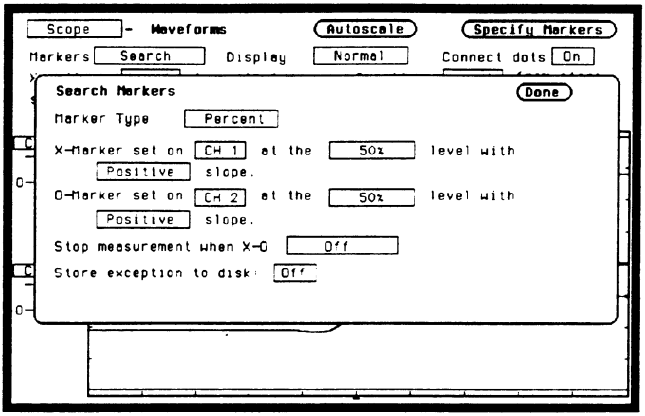


Figure 25-24. Specify Search Markers Pop-up

5. Select **Stop measurement when X-O** and select **Less Than** from the pop-up.
6. Select the time field next to the Less Than field. Set the time field to 10 ns.

7. The Specify Search Markers pop-up now looks like figure 25-25.

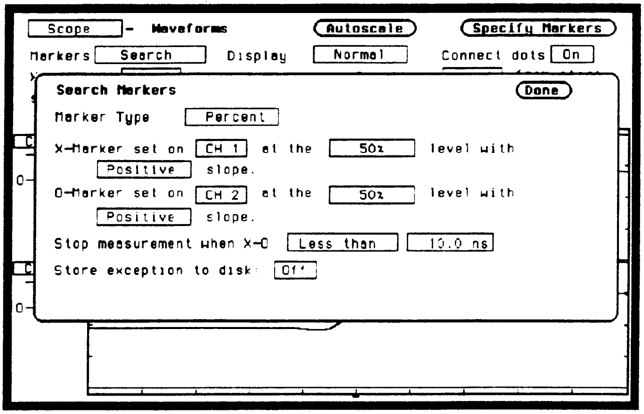


Figure 25-25. Stop Measurement Less than Setup

8. Select the **Done** field to return to the waveform display. The display now looks like figure 25-26.

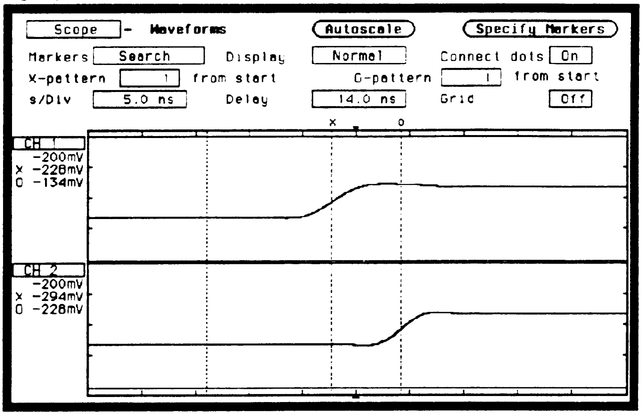


Figure 25-26. Waveform Display

The time interval from X-O in this example is 6.5 ns. Your results may be different because of the different length of BNC cable used to delay channel 2. The oscilloscope was instructed to run until the time interval was less than 5 ns. When the stop condition was satisfied, the oscilloscope stops acquisition and displays the advisory "**Stop run criteria met - run stopped.**"

Display Field

The **Display** field is located in the middle of the top row of fields. When this field is selected, a pop-up appears as shown in figure 25-27. The selections in this pop-up determine how waveform information is displayed.

Display mode pop-up

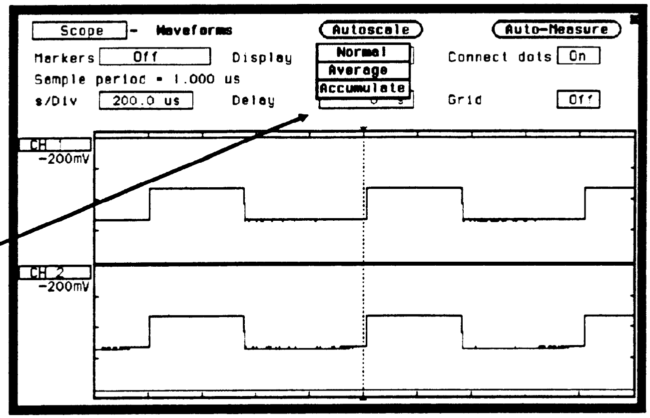


Figure 25-27. Display Field Pop-up

The default selection for the Mode field is Normal.

Normal Mode

In Normal mode, the oscilloscope acquires waveform data and then displays the waveform. When the oscilloscope makes a new acquisition, the previously acquired waveform is erased from the display and replaced by the newly acquired waveform.

Average Mode

In Average mode, the oscilloscope averages the data points on the waveforms with previously acquired data. Averaging helps eliminate random noise from your displayed waveforms. When the Average field is selected from the Display mode pop-up, an integer field appears to the right of the AVG# field. When this Average integer field is selected, an integer selection list pop-up appears, listing all possible average selections.

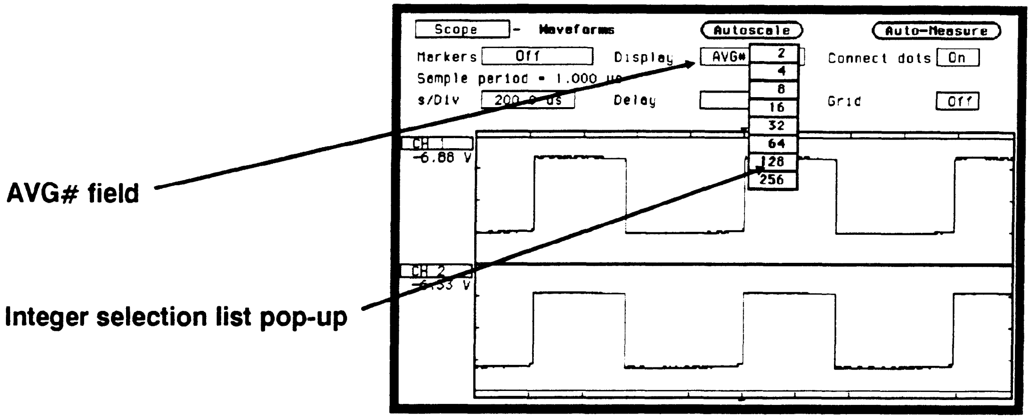


Figure 25-28. Average Integer Entry Pop-up

The number of averages can be changed by turning the knob to position the cursor over the integer you want and pressing the SELECT key.

As an example, assume the Average integer field is set to 16. If the Run mode is set to Repetitive, the oscilloscope will immediately start acquiring waveform data and average them together. When the initial 16 waveforms have been acquired, the oscilloscope will momentarily display the advisory "Number of averages have been met". Once the initial 16 waveforms have been acquired, the oscilloscope will only average the last 16 waveforms acquired; all other data will be discarded.

If the Run mode is set to Single, an acquisition is not made until the Run field has been selected. If Average # is set to 16, as in the previous example, the message "Number of averages have been met" would not be displayed until Run has been selected 16 times.

To exit the Average mode, select the **AVG#** field next to the Average Integer field. The default value for the Average # field is 8.

Accumulate Mode

In Accumulate mode, the oscilloscope accumulates all waveform acquisitions on screen without erasing the previously acquired waveforms. This is similar to infinite persistence on an analog storage oscilloscope. These acquisitions will stay on the display until Mode is changed, or until the waveform is adjusted by a control that causes the display to change, such as s/Div or V/Div.

Connect Dots Field

The **Connect dots** field is located on the right side of the top row of fields. When Connect dots field is selected, it will toggle between on and off.

When Connect dots is On, each displayed dot will be connected to the adjacent dot by a straight line. A waveform with Connect Dots set to On, will be well defined and easier to see.

The default setting for the Connect dots field is Off.

Grid

When the **Grid** field is selected, it will toggle between on and off. When the grid is turned on, the major divisions for both time and voltage will be marked with dotted lines.

26

Mixed Mode Displays

Introduction

This chapter explains mixed mode displays for:

- timing/state
- state/state
- timing/scope
- state/timing/scope

The Mixed Mode waveform display will display all of the above mentioned combinations. The main menu and field definitions for each individual type of display is explained in their respective chapters. Only the unique functions and features of the mixed mode displays are given here.

Mixed Mode field

The Mixed Mode field is located in the same pop-up that is used to access the System Configuration menu, analyzers and the scope. See figure 26-1. The Mixed Mode field will only be available from the Waveform and Listing menus.

Before mixed mode displays can be displayed, the appropriate analyzers and/or scope must be turned on and the appropriate channels or pod bits assigned.

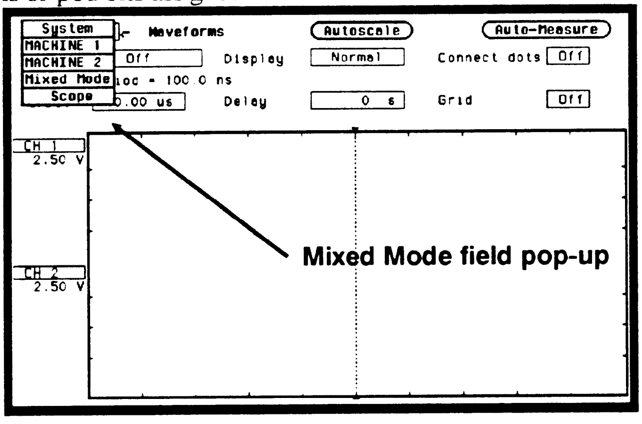


Figure 26-1. Mixed Mode Field

Timing/State Mixed Mode Display

When both timing and state analyzers are on you can display both the State Listing and the Timing Waveforms simultaneously as shown in figure 26-2.

The data in both parts of the display can be time-correlated as long as Count (State Trace menu) is set to Time.

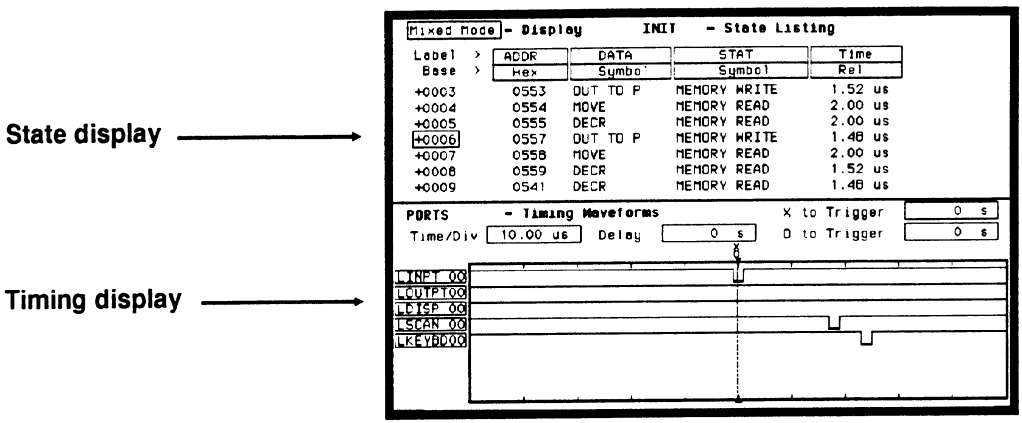
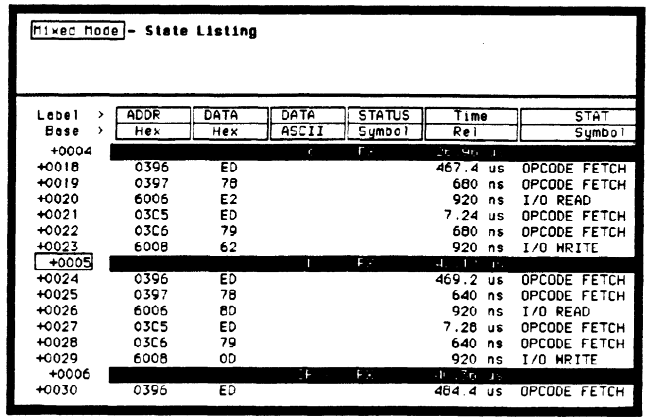


Figure 26-2. Timing/State Mixed Mode Display

The markers for the State Listing and the Timing Waveform in time-correlated Mixed Mode are different from the markers in the individual displays. You will need to place the markers on your points of interest in the time-correlated Mixed Mode even though you have placed them in the individual displays.

State/State Mixed Mode Display

When two state analyzers are on, the logic analyzer will display both state listings as shown below. Data from state machine 1 is the data with the normal memory location columns filled and with normal black on white video. State machine 2 data is interlaced and displayed in inverse video (white on black). Its memory locations are offset to the right in a column.



Mixed Mode - State Listing

Label >	ADDR	DATA	DATA	STATUS	Time	STAT
Base >	Hex	Hex	ASCII	Symbol	Rel	Symbol
+0004			[illegible]	[illegible]	[illegible]	
+0018	0396	ED			467.4 us	OPCODE FETCH
+0019	0397	78			680 ns	OPCODE FETCH
+0020	6006	E2			920 ns	I/O READ
+0021	03C5	ED			7.24 us	OPCODE FETCH
+0022	03C6	79			680 ns	OPCODE FETCH
+0023	6008	62			920 ns	I/O WRITE
+0005			[illegible]	[illegible]	[illegible]	
+0024	0396	ED			469.2 us	OPCODE FETCH
+0025	0397	78			640 ns	OPCODE FETCH
+0026	6006	8D			920 ns	I/O READ
+0027	03C5	ED			7.28 us	OPCODE FETCH
+0028	03C6	79			640 ns	OPCODE FETCH
+0029	6008	0D			920 ns	I/O WRITE
+0006			[illegible]	[illegible]	[illegible]	
+0030	0396	ED			484.4 us	OPCODE FETCH

Figure 26-3. State/State Mixed Mode Display

To time-correlate data from two state machines, you must set the Count (State Trace menu) for both machines to Time.

The markers for a State/State time-correlated Mixed Mode will be the same as the markers placed in each of the individual State Listings.

Timing/Scope Mixed Mode Display

When the timing analyzer and the oscilloscope are both on, you can display both the timing waveform and oscilloscope waveform simultaneously as shown below.

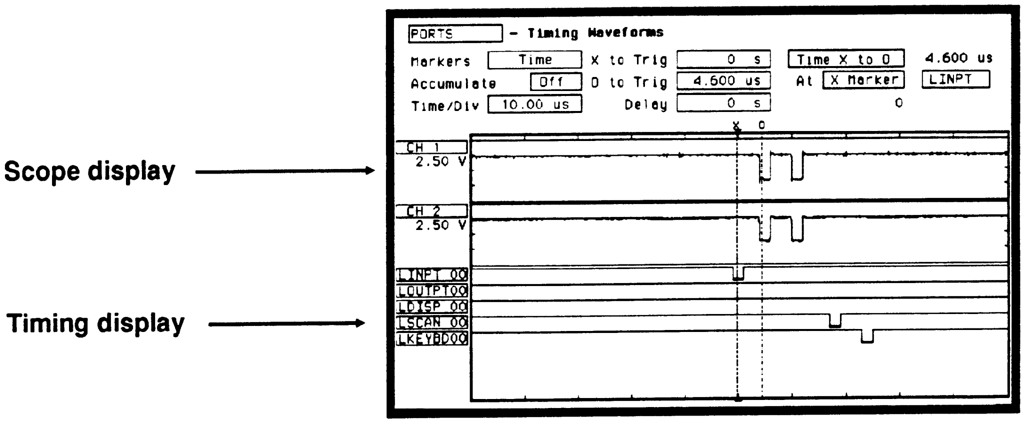


Figure 26-4. Timing/Scope Mixed Mode Display

Arming the Oscilloscope

Both the state and timing analyzers can arm the oscilloscope. However, to display scope and timing waveforms together in the same display, one (either scope or timing) must arm the other. In addition, after proper arming, the scope and timing waveforms can be time-correlated by setting the scope to trigger Immediately.

From the Trigger Specification menu, select the **Armed by** field. A pop-up will appear with all the arming choices. Select the analyzer machine which you have assign as the timing analyzer in the System Configuration menu. For more information see the chapter "Trigger Menu".

Displaying Timing Waveforms

After the oscilloscope has been armed by the timing analyzer, the pods can be assigned and accompanying waveforms displayed in the oscilloscope waveform display. To setup mixed mode display in the oscilloscope waveforms display, follow the steps below:

1. From the waveforms display menu, select a channel label. Remember that the inserted waveform will be placed directly below the label you choose.

2. From the waveform selection pop-up shown in figure 26-5, select **Insert waveform** field.

Waveform selection pop-up

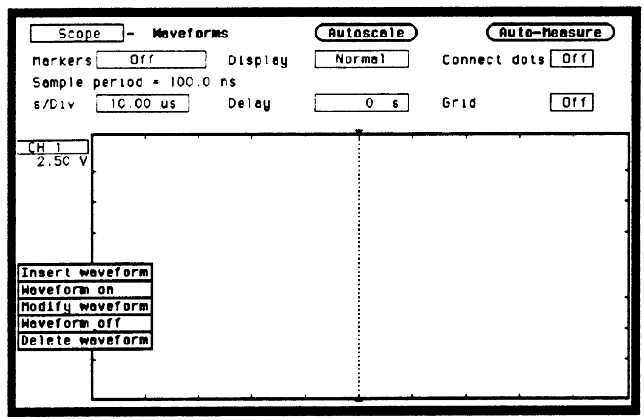


Figure 26-5. Waveform Label Pop-up

3. Since the timing analyzer machine is arming the oscilloscope, another pop-up will appear as shown in figure 26-6 that gives you a choice of Scope or Timing. Select Timing.

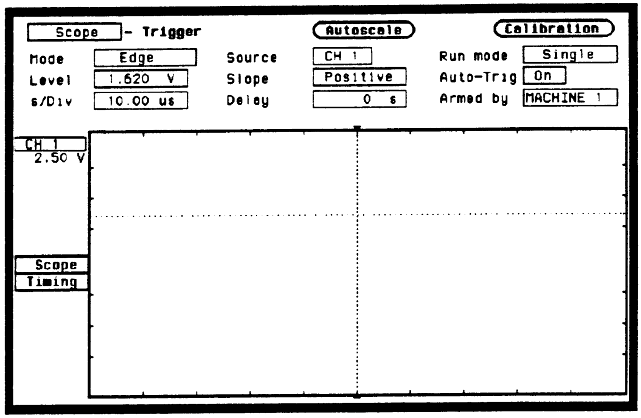


Figure 26-6. Label Type Pop-up

4. A field labeled POD 1 will appear as shown in figure 26-7.

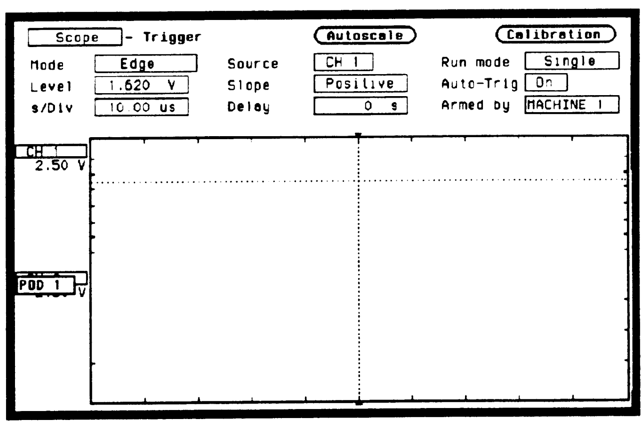


Figure 26-7. Pod Label Field

5. Touch the Select key again and POD 1 will be inserted.

6. Each pod that is inserted, will have the bit number incremented by one, starting from "00". To modify the bit number, select the **Bit select** field which is just to the right of the Pod number. See the Bit selection field in figure 26-8.

7. When the Bit select pop-up appears, enter the bit number you want by rotating the knob or by using the keypad. The keypad will appear when the first key is touched. See figure 26-8.

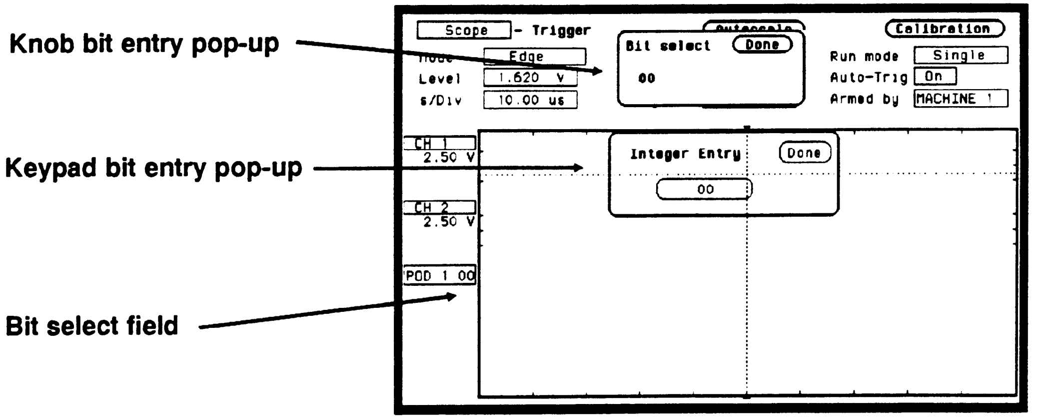


Figure 26-8. Knob And Keypad Entry Pop-ups

The markers for the oscilloscope in time-correlated Mixed Mode are different from the markers in the individual waveforms display. You will need to place the markers on your points of interest in the time-correlated Mixed Mode even though you have placed them in the individual displays.

State/Timing And Scope Mixed Mode Display

When the state analyzer, timing analyzer and the oscilloscope are all on, you can display both analyzer waveforms and the oscilloscope waveforms simultaneously as shown below.

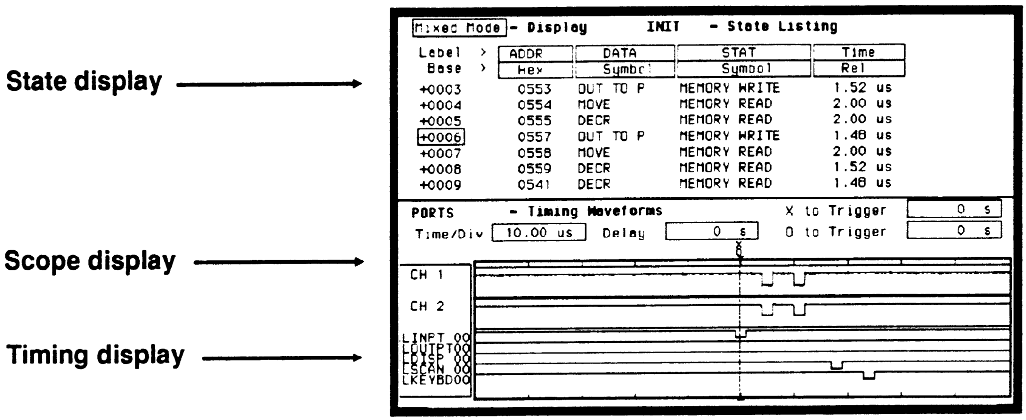


Figure 26-9. State/Timing/Scope Mixed Mode Display

The arming requirements for the oscilloscope are the same as in timing/scope mixed mode displays.

The procedure for inserting waveforms in the timing waveforms display is the same as in timing/scope mixed mode displays.

The markers for the state analyzer, timing analyzer and oscilloscope in time-correlated Mixed Mode are different from the markers in the individual waveforms and listing displays. You will need to place the markers on your points of interest in the time-correlated Mixed Mode even though you have placed them in the individual displays

Time-Correlated Displays

The HP1652B/1653B Logic Analyzers can time-correlate data between the timing analyzer and the state analyzer (see Timing/State Mixed Mode Display) between two state analyzers (see State/State Mixed Mode Display) and between the state analyzer, the timing analyzer and the scope (see State/Timing And Scope Mixed Mode Display).

The logic analyzer uses a counter to keep track of the time between the triggering of one analyzer and the triggering of the second. It uses this count in the mixed mode displays to reconstruct time-correlated data.

To summarize time-correlation between the different displays, remember the following.

To time-correlate the state analyzer display to the timing analyzer display, **arm the timing analyzer** with the state analyzer, then set the state analyzer count mode to **Time**.

Then, to add the scope display, **arm the scope** with the timing analyzer and set the scope to trigger **Immediate.**

27

Timing/State/Oscilloscope Measurement Example

Introduction

In this chapter you will learn how to use the timing analyzer, state analyzer, and oscilloscope interactively by setting up the logic analyzer menus to simulate the process of making a measurement. We will give you the measurement results, as actually measured by the logic analyzer, since you will not have the exact same circuit available.

This measurement example uses an HP 1652B (five pods). The steps for setting up the analyzer menus are ordered in a manner you would natually take if you were actually troubleshooting this problem.

Since you've already had some practice at setting up both the timing and state analyzers in previous examples, you should be able to setup the analyzer menus by looking at the pictures. If you can set up each menu by just looking at the menu pictures, go ahead and do so. If you need a reminder of what steps to perform, follow the numbered steps.

Problem Solving with the Timing/State/Scope Analyzer

In this example assume you have a microprocessor-controlled circuit that sequentially accesses five ports and reports back any that do not respond correctly. After power-up, the system indicates that two of the ports are not responding. The block diagram shown below helps to illustrate the system under test and it's problem.

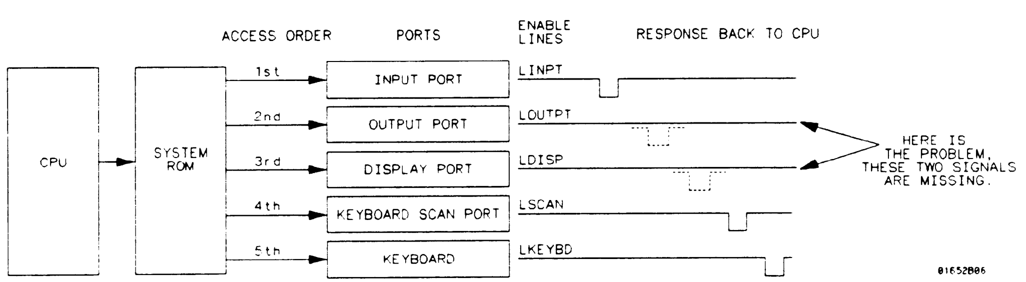


Figure 27-1. System Under Test With Problem

What Am I Going to Measure?

The circuit under test is actually hardware being controlled by firmware. The code stored in the system ROM (Read Only Memory) could be faulty, or there could be a hardware problem, such as a bad IC or shorts/opens on the PC board.

Your measurement should verify the following:

- Whether the code in system ROM is correct.
- Whether the ICs are functioning properly.
- Whether any board shorts/opens exist.

Before you begin your troubleshooting process, you should recognize the strengths of your tools.

1. Your state analyzer can look at the actual code in ROM that controls the circuit operation. In addition, you can use the state analyzer to control (through arming and triggering) the starting point of all other measurements.

2. Your timing analyzer can verify the hardware has correctly translated the code in ROM into the five sequential enable signals with the required timing relationships.

3. Your oscilloscope can then look at any signal lines for closer examination of such thing as noise, spikes, slow pulse transitions, signal amplitude, and any open or shorted conditions.

The measurement sequence should follow the same order. First **verify the system code (state analyzer)**, then look at the **relationships between multiple signals (timing analyzer)**, then take a closer look at any unusual looking signal lines for the **final analysis (oscilloscope)**.

How Do I Configure the Logic Analyzer?

The first part of your measurement is to verify that all 5 ports are initialized by the software in the power-up initialization routine. To do this, you will configure the logic analyzer as a state analyzer.

Configure the System Configuration menu as shown below, so Analyzer 1 is a state analyzer. For this example, the label INIT was chosen for you to discribe the INITialization routine.

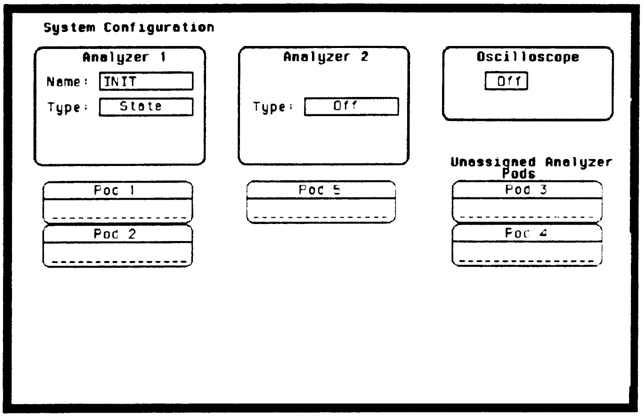


Figure 27-2. Set System Configuration Menu

1. Select **Type** field in Analyzer 1.

2. Select the **State** field.

3. Select **Pod 2** field and assign it to Analyzer 1.

Configuring the State Analyzer

Now that you have configured the system, you are ready to configure the state analyzer. Set the State Format Specification menu as shown in figure 27-3. For this example, the labels ADDR, DATA, and STAT are chosen for you to describe address lines, data lines and CPU status.

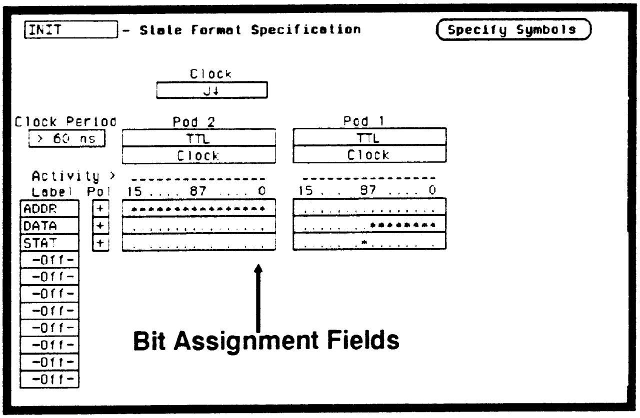


Figure 27-3. Configure the State Format Menu

1. Press FORMAT key on front panel.

2. Select Pod 2 (ADDR) bit assignment field and turn bits 0 through 15 on. These are the address lines.

3. Select Pod 1 (DATA) bit assignment field and turn bits 0 through 7 on. These are the data lines.

4. Select Pod 1 (STAT) bit assignment field and turn bit 8 on. This is the CPU Read/Write line.

You want the state analyzer to trigger and start storing states when it encounters the begining of the initialization routine. This happens at the specific state (0550). To accomplish this, configure the State Trace Specification menu as shown below.

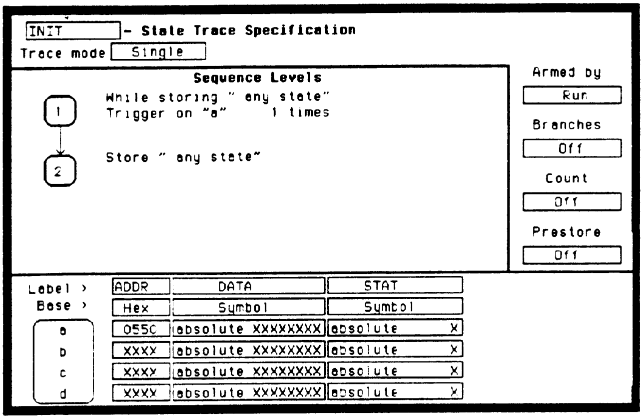


Figure 27-4. Configure the State Trace Menu

1. Press TRACE key on front panel.

2. Select the pattern field and insert the address you want the analyzer to trigger on (0550).

Connecting the Probes

At this point, if you had a target system, you would connect the logic analyzer to your system. Since we have assigned labels ADDR, DATA and STAT, you would connect the probes to your system accordingly.

- Pod 1 probes 0 through 7 to the data bus lines D0 through D7
- Pod 1 probe 9 to the CPU Read/Write control line
- Pod 2 probes 0 through 15 to the address bus lines A0 through A15

Acquiring the Data

You have configured the State Trace Specification menu to start acquiring data when the microprocessor sends address 0550 on the bus. When you press the RUN key, the state analyzer waits until it sees address 0550, then triggers itself, and completes the state data acquisition. The display will then switch automatically to the State Listing menu.

Note

We have assigned symbols for DATA and STAT in the State Listing display for you to better illustrate where the routine executes a memory write to the output ports.

example: 0550   **OUT TO P**   **MEMORY WRITE**

Finding the Problem

You look at the state listing menu to see what the data is in states + 0000 through + 0013. These are the first stored states after trigger. You know your routine has five "OUT TO PORT" memory writes.

The microprocessor, does address the correct memory locations. Now you compare the data from the software engineer as listed below, with what is listed in the State Listing menu in figure 27-5.

0550	OUT TO PORT	MEMORY WRITE
0551	MOVE	MEMORY READ
0552	DECR	MEMORY READ
0553	OUT TO PORT	MEMORY WRITE
0554	MOVE	MEMORY READ
0555	DECR	MEMORY READ
0557	OUT TO PORT	MEMORY WRITE
0558	MOVE	MEMORY READ
0559	DECR	MEMORY READ
0541	DECR	MEMORY READ
0182	OUT TO PORT	MEMORY WRITE
0542	MOVE	MEMORY READ
0543	DECR	MEMORY READ
0544	OUT TO PORT	MEMORY WRITE

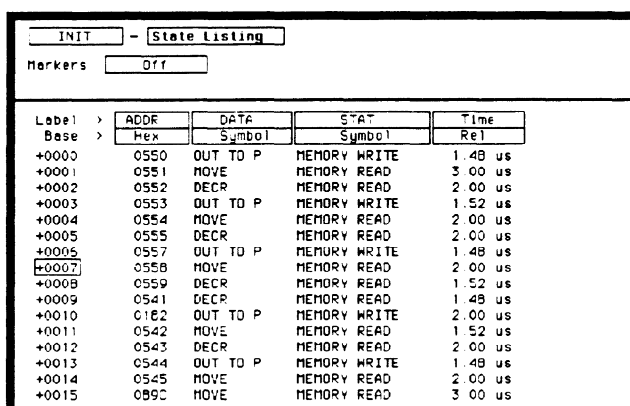


Figure 27-5. State Listing Menu

Since the data stored in memory is correct, it's time to look at the hardware to see if it's causing problems during the initialization routine. You decide to look at the activity on the enable lines during this routine.

In order to see time domain measurements on hardware signals, you need the timing analyzer.

What Additional Measurements Must I Make?

Since the problem exists during the routine that starts at address 0550, you decide to look at the timing waveforms on the enable lines when the routine is running.

Your measurement, then, requires verification of:

- actual response of enable lines from the five ports.
- correct timing of the responding enable lines.

How Do I Re-configure the Logic Analyzer?

In order to make this measurement, you must re-configure the logic analyzer so Analyzer 2 is a timing analyzer. You leave Analyzer 1 as a state analyzer since you will use the state analyzer to trigger the timing analyzer.

Configure the logic analyzer so Analyzer 2 is a timing analyzer as shown below. For this example, the label PORTS was selected for you to describe the output ports.

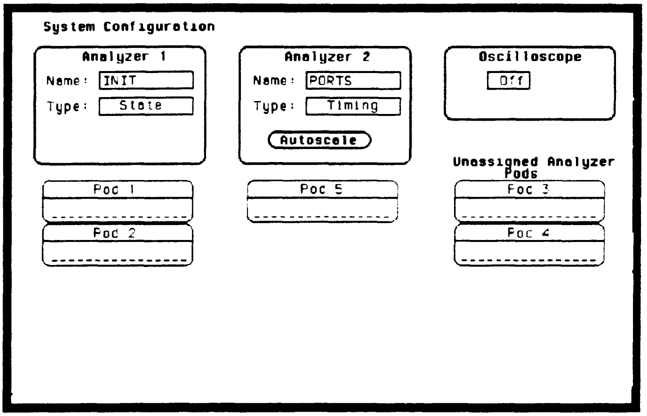


Figure 27-6. Re-configure System Configuration Menu

1. Select **Type** field in Analyzer 2.

2. Select the **Timing** field.

Connecting the Timing Analyzer Probes

At this point you would connect the probes of pod 5 as follows:

- Pod 5 bit 1 to enable line LINPT.
- Pod 5 bit 2 to enable line LOUTPT.
- Pod 5 bit 3 to enable line LDISP.
- Pod 5 bit 4 to enable line LSCAN.
- Pod 5 bit 5 to enable line LKEYBD.

Configuring the Timing Analyzer

Now that you have configured the system, you are ready to configure the timing analyzer. Configure the Timing Format Specification menu as shown below.

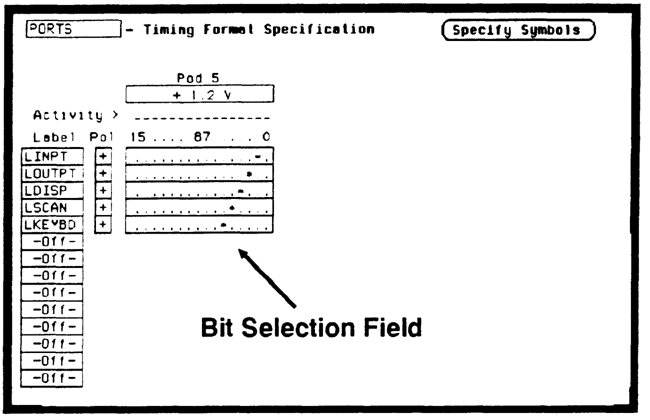


Figure 27-7. Configure the Timing Format Menu

1. Select the bit selection field.

2. Place the cursor on the appropriate bit and turn it on (asterisk *).

Setting the Timing Analyzer Trigger

Your timing measurement requires the timing analyzer to display the timing waveforms present on the enable lines when the routine is running. Since the state analyzer will trigger on address 0550, you will want to arm the timing analyzer with the state analyzer, so the timing waveforms will be captured at the same time.

Configure the Timing Trace Specification menu as shown below.

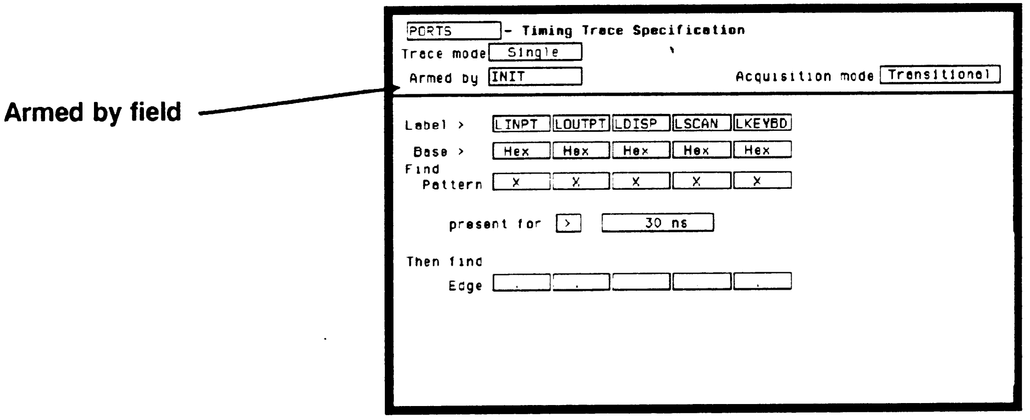


Figure 27-8. Set Armed By Field to INIT

1. Display the Timing Trace Specification menu.
2. Select the **Armed by** field.
3. Select the **INIT** (state analyzer) option in the pop-up.

Time Correlating the Data

In order to time correlate the data, the logic analyzer must store the timing relationships between states. Since the timing analyzer samples asynchronously and the state analyzer samples synchronously, the logic analyzer must use the stored timing relationship of the data to reconstruct a time correlated display.

Configure the State Trace Specification menu as shown below:

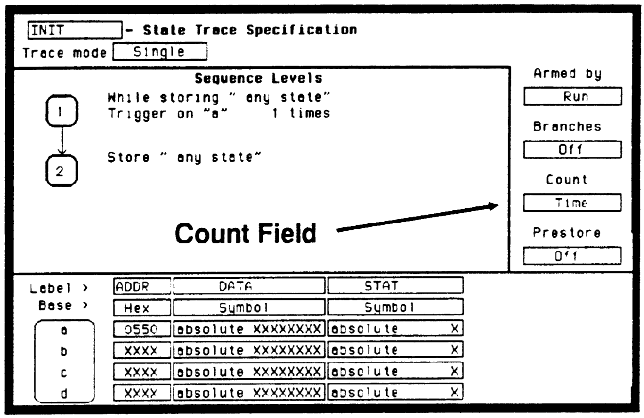


Figure 27-9. Set Count to Time

1. Display the State Trace Specification menu.

2. Select the **Count** field.

3. Select the **Time** field and press the SELECT key. The counter will now be able to keep track of time between states, for the time correlation.

Re-Acquiring the Data

With the timing analyzer configured and the probes of pod 5 connected to the circuit, all you have to do is press RUN. When the logic analyzer acquires the data it switches the display automatically to the Timing Waveforms menu, unless you switched to one of the other menus in the state analyzer after reconfiguring the Timing Trace menu. In that case, you will be in the State Listing menu. Regardless of which analyzer display menu you are in, you should now look at both analyzers together in the Mixed Mode Display.

Mixed Mode Display

The Mixed mode display shows you both the State Listing and Timing Waveforms menus simultaneously. To change the display to the Mixed Mode:

1. Select the field in the upper left corner of the display and press the SELECT key.

2. Select the **Mixed mode** field. You will now see the Mixed Mode display as shown below.

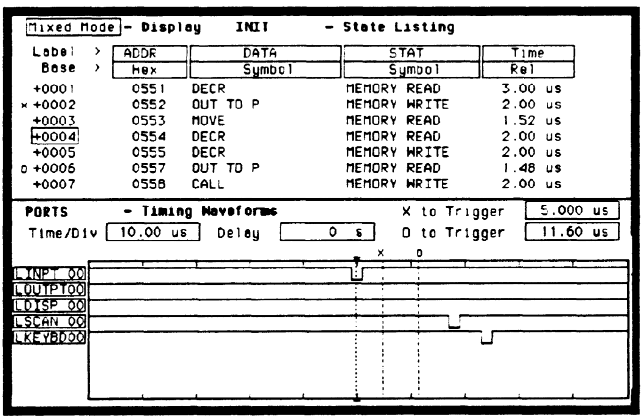


Figure 27-10. Mixed Mode Display

Interpreting the Display

In the Mixed Mode display the state listing is in the top half of the screen and the timing waveforms are in the lower half. The important thing to remember is that you time correlated this display so you could see what is happening in timing during the initialization routine.

Notice that the trigger point in both parts of the display is the same as it was when the displays were separate. The trigger in the state listing is at state + 0000 and the trigger of the timing waveform is the vertical dotted line.

As you look at the Mixed Mode Display, you notice that two of the five sequential enable pulses are missing on the timing waveforms display. This is the problem you are looking for, but you still don't have enough information about what might be causing these two enable lines to be inactive. This is where a closer look with the scope may help.

Re-configure the Analyzer with Scope

The two missing enable signals from the Timing Waveforms display show you where to look next. Before a pulse can be displayed, the voltage level must meet the threshold voltage requirements. To look at these enable lines closer, for a more detailed analysis of their voltage levels or any possible shorts or opens, you will use the oscilloscope.

Your measurement then requires the verification of the following:

- Signal voltage levels.
- Any possible shorted or open conditions.

Go back to the System Configuration menu and turn the Oscilloscope **On** as shown below.

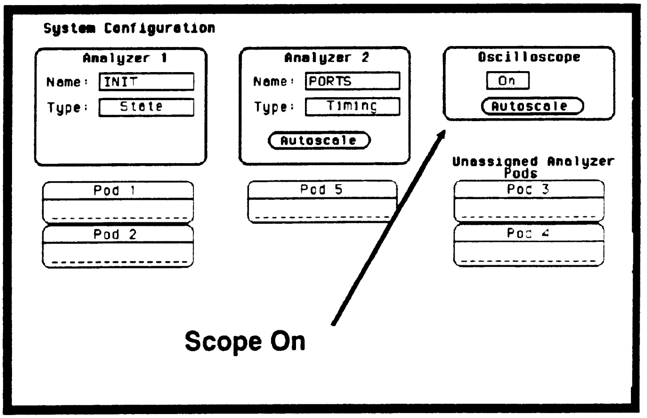


Figure 27-11. Re-configure with Scope

Connecting the Scope Probes

Connect the scope probes to the two enable lines that show missing signals.

Channel one is connected to LOUTPT.
Channel two is connected to LDISP.

Arming the Scope

Before the scope signals can be time-correlated and combined with the Timing Waveforms display, the scope must be armed by the timing analyzer and set to trigger immediately.

Set the **Armed by** field and **Mode** field in the Trigger menu as shown below.

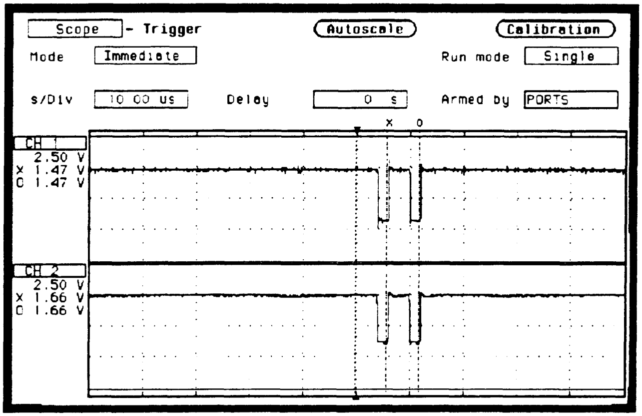


Figure 27-12. Set Armed by Field to PORTS

1. From the System Configuration menu, press the TRIG key. If you don't go to the scope Trigger menu, just select the upper left-most field and from there you can select the **Scope** field.
2. From the scope Trigger menu select the **Armed by** field.
3. From the pop-up select the **PORTS** field.
4. Select the **Mode** field and toggle to **Immediate**.

Making the Scope Measurement

With the scope armed by PORTS (timing analyzer) and the probes connected to LOUTPT and LDISP enable lines, all you have to do is select **RUN**. The scope will automatically switch to the Waveforms display.

The state analyzer cross triggers the timing analyzer, which in turn, triggers the oscilloscope.

Set the s/Div to 10 μs and notice the double pulse and the voltage levels in the figure below.

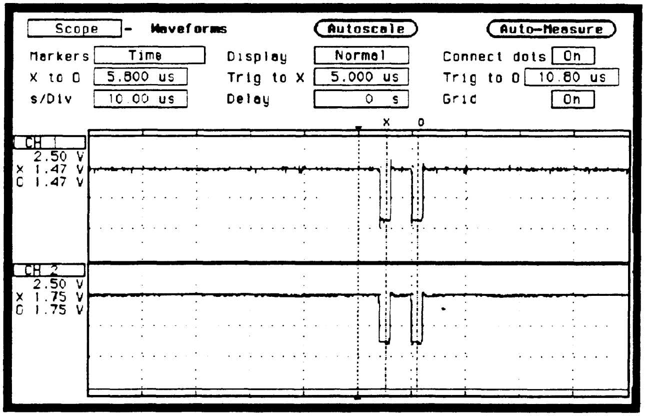


Figure 27-13. Scope Waveforms Display

You can examine the two enable lines in three different displays:

- Scope Waveforms.
- Timing Waveforms (scope channels can be added in).
- Mixed Mode Display.

Mixed Mode Display with Scope

With three different measurements stored in memory, you can now get the total picture of your problem from the Mixed Mode display. As mentioned before, the scope must be armed by the timing analyzer and set to trigger Immediate before the time-correlated scope signals can be inserted into the Timing Waveforms display

Insert the scope waveforms into the Timing Waveforms display (the lower display in Mixed Mode) as shown below.

Insert scope signals here

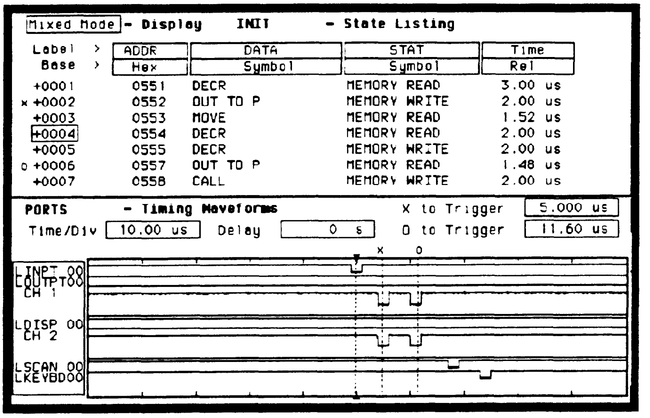


Figure 27-14. Mixed Mode Display

1. Select the upper-left most field and press the SELECT key.

2. From the pop-up select the **Mixed Mode** field.

3. From the Mixed Mode display menu, select the **LOUTPT** field and insert the scope's **CH 1** waveform. The waveform should appear directly below LOUTPT.

4. Select the **LDISP** field and insert the scope's **CH 2** waveform. The waveform should appear directly below LDISP.

Finding the Answer

You notice two double pulses instead of two sequential pulses. Since they are identical, this could mean a short between them. Also, the voltage levels never falls below threshold voltage of the timing analyzer. This is why the pulses were not displayed by the timing analyzer. After futher examination of the pc board, you find a solder bridge shorting the two enable signals together.

Summary

You have just learned how to use the timing and state analyzers interactively with the oscilloscope to find a problem that was not easily determined whether it was a software or hardware problem.

You have learned to do the following:

- Trigger one analyzer with the other.
- Time correlate measurement data.
- Interpret the Mixed mode display.

With three different measurements, time-correlated and displayed side by side, a complete analysis of this problem is done with the HP 1652B.

A

Microprocessor Specific Measurements

Introduction

This appendix contains information about the optional accessories available for microprocessor specific measurements. In depth measurement descriptions are included in the operating notes that come with each of these accessories. The accessories you will be introduced to in this appendix are the preprocessor modules and the HP 10269C General Purpose Probe Interface.

Microprocessor Measurements

A preprocessor module enables you to quickly and easily connect the logic analyzer to your microprocessor under test. Most of the preprocessor modules require the HP 10269C General Purpose Probe Interface. The preprocessor descriptions in the following sections indicate which preprocessors require it.

Included with each preprocessor module is a 3.5-inch disk which contains a configuration file and an inverse assembler file. When you load the configuration file, it configures the logic analyzer for making state measurements on the microprocessor for which the preprocessor is designed. It also loads in the inverse assembler file.

The inverse assembler file is a software routine that will display captured information in a specific microprocessor's mnemonics. The DATA field in the State Listing is replaced with an inverse assembly field (see Figure A-1). The inverse assembler software is designed to provide a display that closely resembles the original assembly language listing of the microprocessor's software. It also identifies the microprocessor bus cycles captured, such as Memory Read, Interrupt Acknowledge, or I/O write.

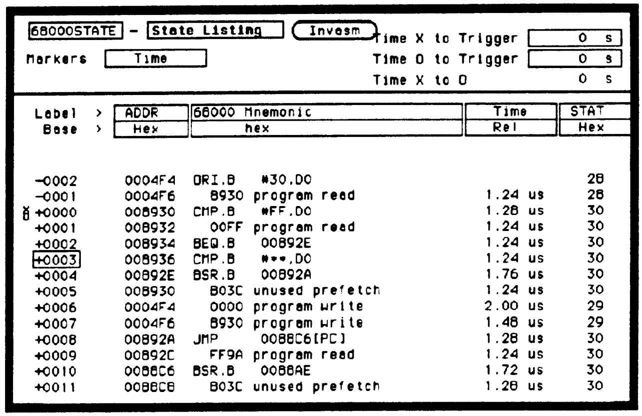


Figure A-1. State Listing with Mnemonics

Microprocessors Supported by Preprocessors

This section lists the microprocessors that are supported by Hewlett-Packard preprocessors and the logic analyzer model that each preprocessor requires. Most of the preprocessors require the HP 10269C General Purpose Probe Interface. The HP 10269C accepts the specific preprocessor PC board and connects it to five connectors on the general purpose interface to which the logic analyzer probe cables connect.

Note

This appendix lists the preprocessors available at the time of printing. However, new preprocessors may become available as new microprocessors are introduced. Check with the nearest Hewlett-Packard sales office periodically for availability of new preprocessors.

Z80

CPU Package: 40-pin DIP

Accessories Required: HP 10300B Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 10 MHz clock input

Signal Line Loading: Maximum of one 74LS TTL load + 35 pF on any line

Microprocessor Cycles Identified: Memory read/write
I/O read/write
Opcode fetch
Interrupt acknowledge
RAM refresh cycles

Maximum Power Required: 0.3 A at + 5 Vdc, supplied by logic analyzer

Logic Analyzer Required: HP 1652B or HP 1653B

Number of Probes Used: Two 16-channel probes

NSC 800

CPU Package: 40-pin DIP

Accessories Required: HP 10303B Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 4 MHz clock input

Signal Line Loading: Maximum of one HCMOS load + 35 pF on any line

Microprocessor Cycles Identified: Memory read/write
I/O read/write
Opcode fetch
Interrupt acknowledge
RAM refresh cycles
DMA cycles

Maximum Power Required: 0.1A at + 5 Vdc, supplied by logic analyzer

Logic Analyzer Required: HP 1652B or HP 1653B

Number of Probes Used: Two 16-channel probes

8085 CPU Package: 40-pin DIP

Accessories Required: HP 10304B Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 6 MHz clock output (12 MHz clock input)

Signal Line Loading: Maximum of one 74LS TTL load + 35 pF on any line

Microprocessor Cycle Identified: Memory read/write
I/O read/write
Opcode fetch
Interrupt acknowledge

Maximum Power Required: 0.8 A at + 5 Vdc, supplied by logic analyzer

Logic Analyzer Required: HP 1652B or HP 1653B

Number of Probes Used: Two 16-channel probes

8086 or 8088

CPU Package: 40-pin DIP

Accessories Required: HP 10305B Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 10 MHz clock input (at CLK)

Signal Line Loading: Maximum of two 74ALS TTL loads + 40 pF on any line

Microprocessor Cycles Identified: Memory read/write
I/O read/write
Code fetch
Interrupt acknowledge
Halt acknowledge
Transfer to 8087 or 8089 co-processors

Additional Capabilities: The 8086 or 8088 can be operating in Minimum or Maximum modes. The logic analyzer can capture all bus cycles (including prefetches) or can capture only executed instructions. To capture only executed instructions, the 8086or 8088 must be operating in the Maximum mode.

Maximum Power Required: 1.0 A at + 5 Vdc, supplied by the logic analyzer

Logic Analyzer Required: HP 1652B

Number of Probes Used: Three 16-channel probes

80186 or 80C186

CPU Package: 68-pin PGA

Accessories Required: HP 10306G Preprocessor

Maximum Clock Speed: 12.5 MHz clock output (25 MHz clock input)

Signal Line Loading: Maximum of 100 kΩ + 18 pF on any line

Microprocessor Cycles Identified: Memory read/write (DMA and non-DMA)
I/O read/write (DMA and non-DMA)
Code fetch
Interrupt acknowledge
Halt acknowledge
Transfer to 8087, 8089, or 82586 co-processors

Additional Capabilities: The 80186 can be operating in Normal or Queue Status modes. The logic analyzer can capture all bus cycles (including prefetches) or can capture only executed instructions.

Maximum Power Required: 0.08 A at + 5 Vdc, supplied system under test.

Logic Analyzer Required: HP 1652B

Number of Probes Used: Four 16-channel probes

80286 CPU Package: 68-contact LCC or 68-pin PGA

Accessories Required: HP 10312D Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 10 MHz clock output (20 MHz clock input)

Signal Line Loading: Maximum of two 74ALS TTL loads + 40 pF on any line

Microprocessor Cycles Identified: Memory read/write
I/O read/write
Code fetch
Interrupt acknowledge
Halt
Hold acknowledge
Lock
Transfer to 80287 co-processor

Additional Capabilities: The logic analyzer captures all bus cycles including prefetches

Maximum Power Required: 0.66 A at + 5 Vdc, supplied by logic analyzer. 80286 operating current from system under test.

Logic Analyzer Required: HP 1652B

Number of Probes Used: Three 16-channel probes

80386

CPU Package: 132-pin PGA

Accessories Required: HP 10314D Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 33 MHz clock output (66 MHz clock input)

Signal Line Loading: Maximum of two 74ALS TTL loads + 35 pF on any line

Microprocessor Cycles Identified: Memory read/write
I/O read/write
Code fetch
Interrupt acknowledge, type 0-255
Halt
Shutdown
Transfer to 8087, 80287, or 80387 co-processors

Additional Capabilities: The logic analyzer captures all bus cycles including prefetches

Maximum Power Required: 1.0 A at + 5 Vdc, supplied by logic analyzer

Logic Analyzer Required: HP 1652B

Number of Probes Used: Five 16-channel probes

6800 or 6802

CPU Package: 40-pin DIP

Accessories Required: HP 10307B Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 2 MHz clock input

Signal Line Loading: Maximum of 1 74LS TTL load + 35 pF on any line

Microprocessor Cycle Identified: Memory read/write
DMA read/write
Opcode fetch/operand
Subroutine enter/exit
System stack push/pull
Halt
Interrupt acknowledge
Interrupt or reset vector

Maximum Power Required: 0.8A at + 5 Vdc, supplied by logic analyzer

Logic Analyzer Required: HP 1652B or HP 1653B

Number of Probes Used: Two 16-channel probes

6809 or 6809E

CPU Package: 40-pin DIP

Accessories Required: HP 10308B Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 2 MHz clock input

Signal Line Loading: Maximum of one 74ALS TTL load + 35 pF on any line

Microprocessor Cycles Identified: Memory read/write
DMA read/write
Opcode fetch/operand
Vector fetch
Halt
Interrupt

Additional Capabilities: The preprocessor can be adapted to 6809/09E systems that use a Memory Management Unit (MMU). This adaptation allows the capture of all address lines on a physical address bus up to 24 bits wide.

Maximum Power Required: 1.0 A at + 5 Vdc, supplied by logic analyzer

Logic Analyzer Required: HP 1652B or HP 1653B

Number of Probes Used: Two 16-channel probes

68008

CPU Package: 40-pin DIP

Accessories Required: HP 10310B Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 10 MHz clock input

Signal Line Loading: Maximum of one 74S TTL load + one 74F TTL load + 35 pF on any line

Microprocessor Cycles Identified: User data read/write
User program read
Supervisor read/write
Supervisor program read
Interrupt acknowledge
Bus grant
6800 cycle

Additional Capabilities: The logic analyzer captures all bus cycles including prefetches

Maximum Power Required: 0.4 A at + 5 Vdc, supplied by logic analyzer

Logic Analyzer Required: HP 1652B

Number of Probes Used: Three 16-channel probes

68000 and 68010 (64-pin DIP)

CPU Package: 64-pin DIP

Accessories Required: HP 10311B Preprocessor
HP 10269C General Purpose Probe Interface

Maximum Clock Speed: 12.5 MHz clock input

Signal Line Loading: Maximum of one 74S TTL load + one 74F TTL load + 35 pF on any line

Microprocessor Cycles Identified: User data read/write
User program read
Supervisor read/write
Supervisor program read
Interrupt acknowledge
Bus Grant
6800 cycle

Additional Capabilities: The logic analyzer captures all bus cycles including prefetches

Maximum Power Required: 0.4 A at + 5 Vdc, supplied by the logic analyzer

Logic Analyzer Required: HP 1652B

Number of Probes Used: Three 16-channel probes

68000 and 68010 (68-pin PGA)

CPU Package: 68-pin PGA

Accessories Required: HP 10311G Preprocessor

Maximum Clock Speed: 12.5 MHz clock input

Signal Line Loading: 100 kΩ + 10 pF on any line

Microprocessor Cycles Identified:
- User data read/write
- User program read
- Supervisor read/write
- Supervisor program read
- Interrupt acknowledge
- Bus Grant
- 6800 cycle

Additional Capabilities: The logic analyzer captures all bus cycles including prefetches.

Maximum Power Required: None

Logic Analyzer Required: HP 1652B

Number of Probes Used: Three 16-channel probes

68020 CPU Package: 114-pin PGA

Accessories Required: HP 10313G

Maximum Clock Speed: 25 MHz clock input

Signal Line Loading: 100 kΩ + 10 pF on any line

Microprocessor Cycles Identified:
- User data read/write
- User program read
- Supervisor read/write
- Supervisor program read
- Bus Grant
- CPU space accesses including:
  - Breakpoint acknowledge
  - Access level control
  - Coprocessor communication
  - Interrupt acknowledge

Additional Capabilities: The logic analyzer captures all bus cycles including prefetches. The 68020 microprocessor must be operating with the internal cache memory disabled for the logic analyzer to provide inverse assembly.

Maximum Power Required: None

Logic Analyzer Required: HP 1652B

Number of Probes Used: Five 16-channel probes

68030 CPU Package: 128-pin PGA

Accessories Required: HP 10316G

Maximum Clock Speed: 25 MHz input

Signal Line Loading: 100KΩ plus 18 pF on all lines except DSACK0 and DSACK1.

Microprocessor Cycles Identified:
- User data read/write
- User program read
- Supervisor program read
- Bus grant
- CPU space accesses including:
  - Breakpoint acknowledge
  - Access level control
  - Coprocessor communication
  - Interrupt acknowledge

Additional Capabilities: The logic analyzer captures all bus cycles, including prefetches. The 68030 microprocessor must be operating with the internal cache memory and MMU disabled for the logic analyzer to provide inverse assembly.

Maximum Power Required: None

Logic Analyzer Required: HP 1652B

Number of Probes Used: Five 16-channel probes

68HC11

CPU Package: 48-pin dual-in-line

Accessories Required: HP 10315G

Maximum Clock Speed: 8.4 MHz input

Signal Line Loading: 100KΩ plus 12 pF on all lines.

Microprocessor Cycles Identified:
- Data read/write
- Opcode/operand fetches
- Index offsets
- Branch offsets
- Irrelevant cycles

Additional Capabilities: The 68HC11 must be operating in the expanded multiplexed mode (addressing external memory and/or peripheral devices) for the logic analyzer to provide inverse assembly.

Maximum Power Required: None

Logic Analyzer Required: HP 1652B or HP1553B

Number of Probes Used: Two 16-channel probes for state analysis and one to four for timing analysis.

Loading Inverse Assembler Files

You load the inverse assembler file by loading the appropriate configuration file. Loading the configuration file automatically loads the inverse assembler file.

Selecting the Correct File

Most inverse assembler disks contain more than one file. Each disk usually contains an inverse assembler file for use with the HP 10269C and preprocessor as well as a file for general purpose probing. Each inverse assembler filename has a suffix which indicates whether it is for the HP 10269C and preprocessor or general purpose probing. For example, filename C68000_I indicates a 68000 inverse assembler file for use with the HP 10269C and the 68000 preprocessor. Filename C68000_P is for general purpose probing. Specific file descriptions and recommended usage are contained in each preprocessor operating note.

Loading the Desired File

To load the inverse assembler file you want, insert the 3.5-inch disk you received with your preprocessor in the disk drive. Select the I/O menu. In the I/O menu, select DISK OPERATIONS. The logic analyzer will read the disk and display the disk directory.

Select the Load option and place the filename you want to load in the "from file" box. Place the cursor on Execute and press SELECT.

Place the cursor on the analyzer you want the file loaded into and press SELECT. An advisory "Loading file from disk" is displayed. When the logic analyzer has finished loading the file, you will see "Load operation complete."

The file is now loaded and the logic analyzer is configured for disassembly of acquired data.

Connecting the Logic Analyzer Probes

The specific preprocessor and inverse assembler you are using determines how you connect the logic analyzer probes. Since the inverse assembler files configure the System Configuration, State Format Specification, and State Trace Specification menus, you must connect the logic analyzer probe cables accordingly so that the acquired data is properly grouped for inverse assembly. Refer to the specific inverse assembler operating note for the proper connections.

How to Display Inverse Assembled Data

The specific preprocessor and inverse assembler you are using determines how the inverse assembled data is displayed. When you press RUN, the logic analyzer acquires data and displays the State Listing menu.

The State Listing menu will display as much information about the captured data as possible. For some microprocessors, the display will show a completely disassembled state listing.

Some of the preprocessors and/or the microprocessors under test do not provide enough status information to disassemble the data correctly. In this case, you will need to specify additional information (i.e., tell the logic analyzer what state contains the first word of an opcode fetch). When this is necessary an additional field (INVASM) will appear in the top center of the State Listing menu (see below). This field allows you to point to the first state of an Op Code fetch.

68000STATE	- State Listing	Invasm	Minimum X-O:	0	s
Markers	Statistics	Valid runs:	Maximum X-O:	0	s
		16 of 16	Average X-O:	0	s

Figure A-2. Inverse Assemble Field

For complete details refer to the Operating Note for the specific preprocessor.

B

Automatic Measurement Algorithms

Introduction

One of the HP 1652B/1653B's primary oscilloscope features is its ability to make parametric measurements. This appendix provides details on how automatic measurements are performed and some tips on how to improve automatic measurement results.

Measurement Setup

Measurements typically should be made at the fastest possible sweep speed to obtain the most measurement accuracy possible. For any measurement to be made, the portion of the waveform required for that measurement must be displayed on the oscilloscope:

- At least one complete cycle must be displayed for period or frequency measurements.
- The entire pulse must be displayed for pulse width measurements.
- The leading (rising) edge of the waveform must be displayed for risetime measurements.
- The trailing (falling) edge of the waveform must be displayed for falltime measurements.

Making Measurements

If more than one waveform, edge, or pulse is displayed, the measurements are made on the first (leftmost) portion of the displayed waveform that can be used. When any of the defined measurements are requested, the oscilloscope first determines the top (100%) and base (0%) voltages of the waveform. From this information, it can determine the other important voltage values (10% voltage, 90% voltage, and 50% voltage) required to make the measurements. The 10% and 90% voltage values are used in the risetime and falltime measurements. The 50% voltage value is used for measuring frequency, period, pulse width, and duty cycle.

Top and Base Voltages

All measurements except Vp_p are calculated using the Vtop (100% voltage) and Vbase (0% voltage) levels of the displayed waveform. The Vtop and Vbase levels are determined from an occurrence density histogram of the data points displayed on screen.

The digitizing oscilloscope displays 6-bit vertical voltage resolution. This means the vertical display is divided up into 64 voltage levels. Each of these 64 levels is called a quantization level. Each waveform has a minimum of 500 data points displayed horizontally on screen. Each of these data point sets have one quantization level assigned to it. The histogram is calculated by adding the number of occurrences of each quantization level of the displayed data point sets on the displayed waveform.

The quantization level with the greatest number of occurrences in the top half of the waveform corresponds to the Vtop level. The quantization level with the greatest number of occurrences in the bottom half of the waveform corresponds to the Vbase level.

If Vtop and Vbase do not contain at least 5% of the minimum (500) data points displayed on screen, Vtop defaults to the maximum voltage (Vmaximum) and Vbase defaults to the minimum voltage (Vminimum) found on the display. An example of this case would be measurements made on sine or triangle waves.

From this information the instrument can determine the 10, 50, and 90% points, which are used in most automatic measurements. The Vtop or Vbase of the waveform is not necessarily the maximum or minimum voltage present on the waveform. If a pulse has a slight amount of overshoot, it would be wrong to select the highest peak of the waveform as the top since the waveform normally rests below the perturbation.

Measurement Algorithms

The following is a condensed explanation of the automatic measurements discussed in chapter 25.

Frequency (Freq)

The frequency of the first complete cycle displayed is measured using the 50% levels.

If the first edge on the display is rising then

$$Freq = 1/(t_{rising\ edge\ 2} - t_{rising\ edge\ 1})$$

If the first edge on the display is falling then

$$Freq = 1/(t_{falling\ edge\ 2} - t_{falling\ edge\ 1})$$

Period

The period is measured at the 50% voltage level of the waveform.

If the first edge on the display is rising then

$$Period = t_{rising\ edge\ 2} - t_{rising\ edge\ 1}$$

If the first edge on the display is falling then

$$Period = t_{falling\ edge\ 2} - t_{falling\ edge\ 1}$$

Peak-to-Peak Voltage (Vp_p)

The maximum and minimum voltages for the selected source are measured.

$$Vp\_p = V_{maximum} - V_{minimum}$$

where $V_{maximum}$ and $V_{minimum}$ are the maximum and minimum voltages present on the selected source.

Positive Pulse width (+Width)

Pulse width is measured at the 50% voltage level. If the first edge on the display is rising then

$$+\text{Width} = t_{\text{falling edge 1}} - t_{\text{rising edge 1}}$$

If the first edge on the display is falling then

$$+\text{Width} = t_{\text{falling edge 2}} - t_{\text{rising edge 1}}$$

Negative Pulse width (-Width)

Negative pulse width is the width of the first negative pulse on screen using the 50% levels.

If the first edge on the display is rising then

$$-\text{Width} = t_{\text{rising edge 2}} - t_{\text{falling edge 1}}$$

If the first edge on the display is falling then

$$-\text{Width} = t_{\text{rising edge 1}} - t_{\text{falling edge 1}}$$

Risetime

The risetime of the first displayed rising edge is measured. To obtain the best possible measurement accuracy, set the sweep speed as fast as possible while leaving the leading edge of the waveform on the display. The risetime is determined by measuring time at the 10% and 90% voltage points on the rising edge.

$$\text{Risetime} = t_{90\%} - t_{10\%}$$

Falltime

Falltime is measured between the 10% and 90% points of the falling edge. To obtain the best possible measurement accuracy, set the sweep speed as fast as possible while leaving the falling edge of the waveform on the display.

$$\text{Falltime} = t_{10\%} - t_{90\%}$$

Preshoot and Overshoot

Preshoot and Overshoot measure the perturbation on a waveform above or below the top and base voltages (see "Top and Base Voltages" section earlier in this appendix). These measurements use all data displayed on screen, therefore it is very important that only the data of interest be displayed. If you want to measure preshoot and overshoot on one edge of a waveform, then only display that edge. If you want to measure the maximum preshoot and overshoot on a waveform, then display several cycles of the waveform.

Preshoot is a perturbation before a rising or a falling edge and is measured as a percentage of the top-base voltage.

Overshoot is a perturbation after a rising or a falling edge and is measured as a percentage of the top-base voltage

If the measured edge is rising then

$$\text{Preshoot} = \left[\frac{\text{Vbase - Vminimum}}{\text{Vtop-base}}\right] \text{X } 100$$

$$\text{Overshoot} = \left[\frac{\text{Vmaximum - Vtop}}{\text{Vtop-base}}\right] \text{X } 100$$

If the measured edge is falling then:

$$\text{Preshoot} = \left[\frac{\text{Vmaximum - Vtop}}{\text{Vtop-base}}\right] \text{X } 100$$

$$\text{Overshoot} = \left[\frac{\text{Vbase - Vminimum}}{\text{Vtop-base}}\right] \text{X } 100$$

Duty Cycle

The positive pulse width and the period of the displayed signal are measured.

duty cycle = (+ pulse width/period) x 100

rms Voltage

The rms voltage is computed over one complete period.

Average Voltage

The average voltage of the first cycle of the displayed signal is measured. If a complete cycle is not present, the instrument will average the data points on screen.

C

Error Messages

Introduction

This appendix lists the error messages that require corrective action to restore proper operation of the logic analyzer. There are several messages that you will see that are merely advisories and are not listed here. For example, "Load operation complete" is one of these advisories.

The messages are listed in alphabetical order and in bold type.

Acquisition aborted. This message is displayed whenever data acquisition is stopped.

At least one edge is required. A state clock specification requires at least one clock edge. This message only occurs if you turn off all edges in the state clock specification.

Autoload file not of proper type. This message is displayed if any file other than an HP 1652B/1653B configuration file is specified for an autoload file and the logic analyzer is powered up.

Autoscale aborted. This message is displayed when the STOP key is pressed or if a signal is not found 15 seconds after the initiation of autoscale.

BNC is being used as an ARM IN and cannot be used as an ARM OUT. This message is displayed when BNC arms machine 1 (or 2), machine 1 (or 2) arms machine 2 (or 1), and the BNC is specified as ARM OUT. It will not occur if BNC arms machine 1 (or 2), and machine 1 (or 2) arms BNC.

Configuration not loaded. Indicates a bad configuration file. Try to reload the file again. If the configuration file will still not load, a new disk and/or configuration file is required.

Copy operation complete. Indicates the copy operation has either successfully completed or has been stopped.

Correlation counter overflow. The correlation counter overflows when the time from when one machine's trigger to the second machine's trigger exceeds the maximum count. It may be possible to add a "dummy" state to the second machine's trigger specification that is closer in time to the trigger of the first machine.

Data can not be correlated-Time count need to be turned on. "Count" must be set to "Time" in both machines to properly correlate the data.

Destination write protected-file not copied. Make sure you are trying to copy to the correct disk. If so, set the write protect tab to the non-protect position and repeat the copy operation.

File not copied to disc-check disc. The HP 1652B/1653B does not support track sparing. If a bad track is found, the disk is considered bad. If the disk has been formatted elsewhere with track sparing, the HP 1652B/1653B will only read up to the first spared track.

Hardware ERROR: trace point in count block. Indicates the data from the last acquisition is not reliable and may have been caused by a hardware problem. Repeat the data acquisition to verify the condition. If this message re-appears, the logic analyzer requires the attention of service personnel.

Insufficient memory to load IAL - load aborted. This message indicates that there is not a block of free memory large enough for the inverse assembler you are attempting to load even though there may be enough memory in several smaller blocks. Try to load the inverse assembler again. If this load is unsuccessful, load the configuration and inverse assembler separately.

Invalid file name. Check the file name. A file name must start with an alpha character and cannot contain spaces or slashes (/).

Inverse assembler not loaded-bad object code. Indicates a bad inverse assembler file on the disc. A new disc or file is required.

Maximum of 32 channels per label. Indicates an attempt to assign more than 32 channels to a label. Reassign channels so that no more than 32 are assigned to a label.

No room on destination-file not copied. Indicates the destination disc doesn't have enough room for the file you are attempting to copy. Try packing the disc and repeating the copy operation. If this is unsuccessful, you will need to use a different disc.

(x) Occurrences Remaining in Sequence. Indicates the logic analyzer is waiting for (x) number of occurrences in a sequence level of the trigger specification before it can go on to the next sequence level.

PRINT has been stopped. This message appears when the print operation has been stopped.

(x) Secs Remaining in Trace. Indicates the amount of time remaining until acquisition is complete in Glitch mode.

Search failed - O pattern not found. Indicates the O pattern does not exist in the acquired data. Check for a correct O marker pattern specification.

Search failed - X pattern not found. Indicates the X pattern does not exist in the acquired data. Check for a correct X marker pattern specification.

Slow Clock or Waiting for Arm. Indicates the state analyzer is waiting for a clock or arm from the other machine. Recheck the state clock or arming specification.

Slow or missing Clock. Indicates the state analyzer has not recognized a clock for 100 ms. Check for a missing clock if the intended clock is faster than 100 ms. If clock is present but is slower than 100 ms, the data will still be acquired when a clock is recognized and should be valid.

Specified inverse assembler not found. Indicates the inverse assembler specified cannot be found on the disk.

State clock violates overdrive specification. Indicates the data from the last acquisition is not reliable due to the state clock signal not being reliable. Check the clock threshold for proper setting and the probes for proper grounding.

States Remaining to Post Store. Indicates the number of states required until memory is filled and acquisition is complete.

Time count need to be turned on. This message appears when the logic analyzer attempts to time correlate data and "Count" is not set to "Time."

Transitions Remaining to Post Store. Indicates the number of transitions required until memory is filled and acquisition is complete.

Unsupported destination format-file not copied. Indicates the disk you have attempted to copy to is either not formatted or formatted in a format not used by the logic analyzer. Format the disk or use a properly formatted disk and repeat the copy operation.

Value out of range. Set to limit. Indicates an attempt to enter a value that is out of range for the specific variable. The logic analyzer will set the value to the limit of the variable range automatically.

Waiting for Arm. Indicates the arming condition has not occurred.

Waiting for Prestore. Indicates the prestore condition has not occurred.

Waiting for Trigger. Indicates the trigger condition has not occurred.

Warning: Chips not successfully running. Indicates the acquisition chips in the logic analyzer are not running properly. Press STOP and then RUN again. If the warning message reappears, refer the logic analyzer to service personnel.

Warning: Chips not successfully stopped. Indicates the acquisition chips in the logic analyzer are not stopping properly. Press RUN and then STOP again. If the warning message reappears, refer the logic analyzer to service personnel.

Warning: Duplicate label name. Indicates an attempt to assign an existing name to a new label.

Warning: Duplicate symbol name. Indicates an attempt to assign an existing name to a new symbol.

Warning: Invalid file type. Indicates an attempt to load an invalid file type. For example, the SYSTEM file can only be loaded on power-up and if you attempt to load it from the I/O menu, this message will appear.

Warning: No clock edge in other clock, add clock edge. This message only occurs in a state analyzer using mixed or demultiplexed clocks. It indicates there is no edge specified in either the master or slave clock. There must be at least one edge in each of the clocks.

Warning: Symbol memory full. Max 200 symbols. Indicates an attempt to store more than 200 symbols.

Warning: Run HALTED due to variable change. Indicates a variable has been changed during data acquisition in the continuous trace mode. The data acquisition will be halted and this message will be displayed when any variable affecting the system configuration, clock thresholds, clock multiplexing, or trace specification menus is changed during data acquisition.

D

Installation, Maintenance and Calibration

Introduction

This appendix contains information and instructions necessary for preparing the HP 1652B/1653B Logic Analyzers for use. Included in this section are inspection procedures, power requirements packaging information, and operating environment. It also tells you how to load the operating system and turn the logic analyzer on. Also included in this appendix is information on calibration and maintenance that you can do as an operator.

Initial Inspection

Inspect the shipping container for damage. If the shipping container or cushioning material is damaged, it should be kept until the contents of the shipment have been checked for completeness and the instrument has been checked mechanically and electrically. Accessories supplied with the instrument are listed under "Accessories Supplied" in chapter 1 of this manual. An overview of the self-test procedure is in Appendix E of this manual. The complete details of the procedure are in Chapter 6 of the Service manual. Electrical performance verification functions are also in Chapter 3 of the Service Manual.

If the contents are incomplete, if there is mechanical damage or defect, or if the instrument does not pass the Self Test Performance Verification, notify the nearest Hewlett-Packard Office. If the shipping container is damaged, or the cushioning materials show signs of stress, notify the carrier as well as the Hewlett-Packard Office. Keep all shipping materials for the carrier's inspection. The Hewlett-Packard office will arrange for repair or replacement at HP option without waiting for claim settlement.

Operating Environment

You may operate your logic analyzer in a normal lab or office type environment without any additional considerations. If you intend to use it in another type of environment, refer to Appendix F for complete operating environment specifications. Note the humidity limitation. Condensation within the instrument cabinet can cause poor operation or malfunction. Protection should be provided against temperature extremes which cause condensation.

Ventilation

You must provide an unrestricted airflow for the fan and ventilation openings in the rear of the logic analyzer. However, you may stack the logic analyzer under, over, or in-between other instruments as long the surfaces of the other instruments are not needed for their ventilation.

Storage and Shipping

This instrument may be stored or shipped in environments within the following limitations:

- Temperature: - 40 ° C to + 75 ° C
- Humidity: Up to 90% at 65 ° C
- Altitude: Up to 15,300 metres (50,000 feet)

Tagging for Service

If the instrument is to be shipped to a Hewlett-Packard office for service or repair, attach a tag to the instrument identifying owner address of owner, complete instrument model and serial numbers and a description of the service required.

Original Packaging

If the original packaging material is unavailable or unserviceable materials identical to those used in factory packaging are available through Hewlett-Packard offices. If the instrument is to be shipped to a Hewlett-Packard office for service, tag the instrument (see "Tagging for Service"). Mark the container FRAGILE to ensure careful handling. In any correspondence, refer to the instrument by model number and full serial number.

Other Packaging

The following general instructions should be followed for repacking with commercially available materials.

a. Wrap the instrument in heavy paper or plastic.

b. Use a strong shipping container. A double-wall carton made of 350 lb. test material is adequate.

c. Use a layer of shock-absorbing material 70 to 100 mm (3 to 4 inches) thick around all sides of the instrument to firmly cushion and prevent movement inside the container. Protect the control panel with cardboard.

d. Seal the shipping container securely.

e. Mark the shipping container FRAGILE to ensure careful handling.

f. In any correspondence, refer to the instrument by model number and full serial number.

Power Requirements

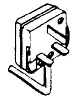
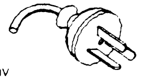
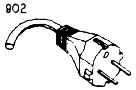
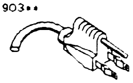
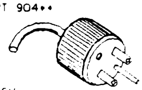
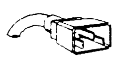
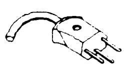
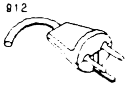
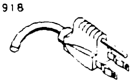
The HP 1652B/53B requires a power source of either 115 or 230 Vac −22% to + 10%; single phase, 48 to 66 Hz; 200 Watts maximum power.

Power Cable

This instrument is provided with a three-wire power cable. When connected to an appropriate AC power outlet, this cable grounds the instrument cabinet. The type of power cable plug shipped with the instrument depends on the country of destination. Refer to Table D-1 for power plugs and HP part numbers for the available plug configurations.

Warning

BEFORE CONNECTING THIS INSTRUMENT, the protective earth terminal of the instrument must be connected to the protective conductor of the (Mains) power cord. The Mains plug must be inserted in a socket outlet provided with a protective earth contact. The protective action must not be negated by the use of an extension cord (power cable) without a protective conductor (grounding). Grounding one conductor of a two conductor outlet does not provide an instrument ground.

PLUG TYPE	CABLE PART NO.	PLUG DESCRIPTION	LENGTH IN/CM	COLOR	COUNTRY
OPT 900  250V	8120-1351 8120-1703	Straight •BS1363A 90°	90/228 90/228	Gray Mint Gray	United Kingdom, Cyprus, Nigeria, Zimbabwe, Singapore
OPT 901  250V	8120-1369 8120-0696	Straight •NZSS198/ASC 90°	79/200 87/221	Gray Mint Gray	Australia New Zealand
OPT 902  250V	8120-1689 8120-1692 8120-2857	Straight •CEE7-Y11 90° Straight (Shielded)	79/200 79/200 79/200	Mint Gray Mint Gray Coco Brown	East and West Europe, Saudi Arabia, So. Africa, India (Unpolarized in many nations)
OPT 903••  125V	8120-1378 8120-1521 8120-1992	Straight •NEMA5-15P 90° Straight (Medical) UL544	90/228 90/228 96/244	Jade Gray Jade Gray Black	United States, Canada, Mexico, Phillipines, Taiwan
OPT 904••  250V	8120-0698	Straight •NEMA6-15P	90/228	Black	United States, Canada
OPT 905  250V	8120-1396 8120-1625	CEE22-V1 (System Cabinet Use) 250V	30/76 96/244	Jade Gray	For interconnecting system components and peripherals. United States and Canada only
OPT 906  250V	8120-2104 8120-2296	Straight •SEV1011 1959-24507 Type 12 90°	79/200 79/200	Mint Gray Mint Gray	Switzerland
OPT 912  220V	8120-2956 8120-2957	Straight •DHCK107 90°	79/200 79/200	Mint Gray Mint Gray	Denmark
OPT 917  250V	8120-4211 8120-4600	Straight SABS164 90°	79/200 79/200	Jade Gray	Republic of South Africa India
OPT 918  100V	8120-4753 8120-4754	Straight Miti 90°	90/230 90/230	Dark Gray	Japan

Rev. 11NOV88 ART00019

•Part number shown for plug is industry identifier for plug only. Number shown for cable is HP part number for complete cable including plug.

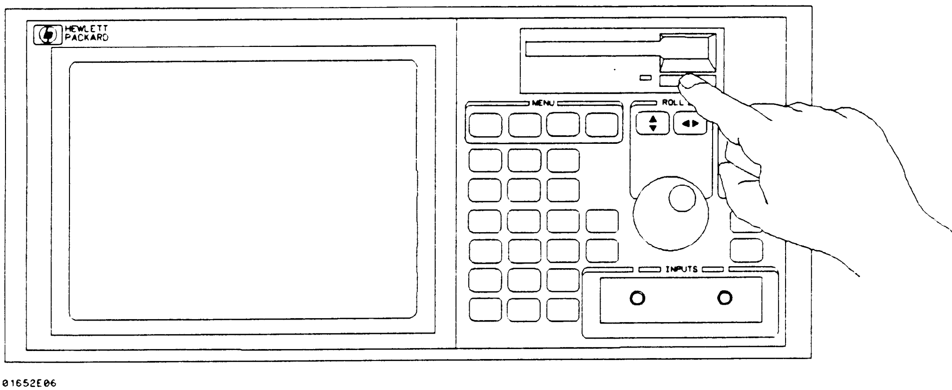
••These cords are included in the CSA certification approval of the equipment.

E=Earth Ground
L=Line
N=Neutral

Table D-1. Power Cord Configurations

Removing Yellow Shipping Disk

Your logic analyzer is shipped with a protective yellow shipping disk in the disk drive. Before you can insert the operating system disk you must remove the yellow shipping disk. Press the disk eject button as shown in figure D-1. The yellow shipping disk will pop out part way so you can pull it out of the disk drive.



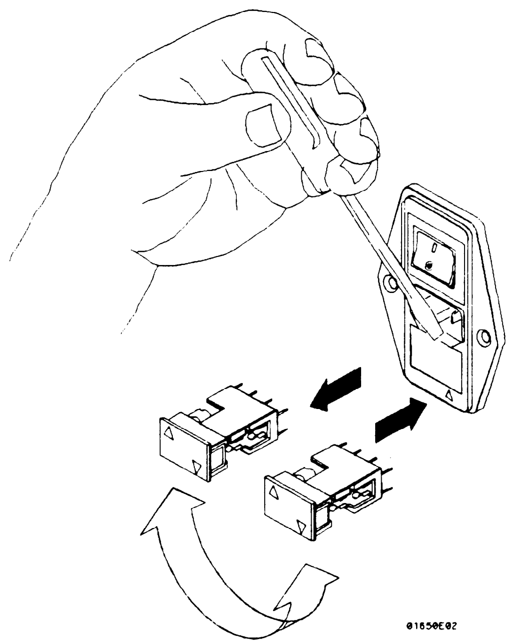
D-1. Removing Yellow Shipping Disk

Selecting the Line Voltage

The line voltage selector has been factory set to the line voltage used in your country. It is a good idea to check the setting of the line voltage selector so you become familiar with what it looks like. If the setting needs to be changed, follow the procedure in the next paragraph.

Caution

You can damage the logic analyzer if the module is not set to the correct position.

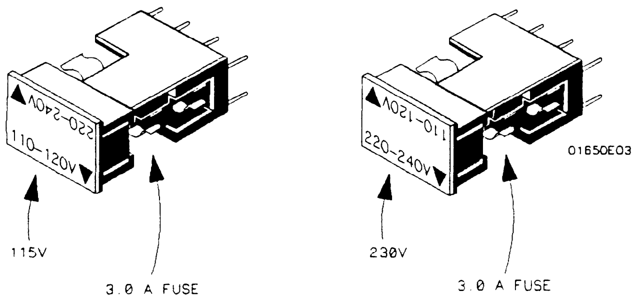


D-2. Selecting The Line Voltage

You change the proper line voltage by pulling the fuse module out and reinserting it with the proper arrows aligned. To remove the fuse module, carefully pry at the top center of the module (as shown) until you can grasp and pull it out by hand.

Checking for the Correct Fuse

If you find it necessary to check or change fuses, remove the fuse module and look at each fuse for its amperage and voltage. Refer to figure D-3 to locate the 115 V and 230 V fuse locations. To remove the fuse module, carefully pry at the top center of the module (see figure D-2) until you can grasp and pull it out by hand.



D-3. Checking For The Correct Fuse

Applying Power

When power is applied to the HP 1652B/1653B, a power-up self test will be performed automatically. For information on the power-up self test, refer to Appendix E and Section 3 of the Service Manual.

Loading the Operating System

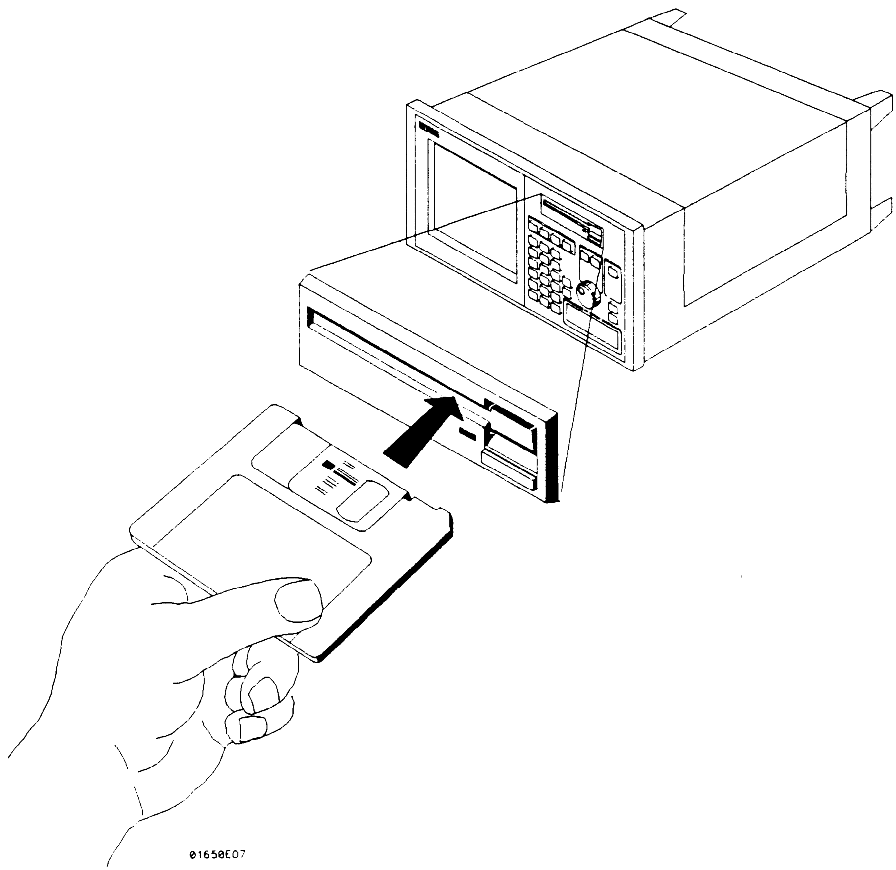
Before you can operate the instrument, you must load the operating system from the operating system disk. You received two identical operating system disk. You should mark one of them Master and store it in a safe place. Mark the other one Work and use only the work copy. This will provide you with a back-up in case your work copy becomes corrupt.

Caution

To prevent damage to your operating system disk, DO NOT remove the disk from the disk drive while it is running. Only remove it after the indicator light has gone out.

Installing the Operating System Disk

To load the logic analyzer's operating system, you must install the disk as shown below before you turn on the power. When the disk snaps into place, the disk eject button pops out and you are ready to turn on the logic analyzer.

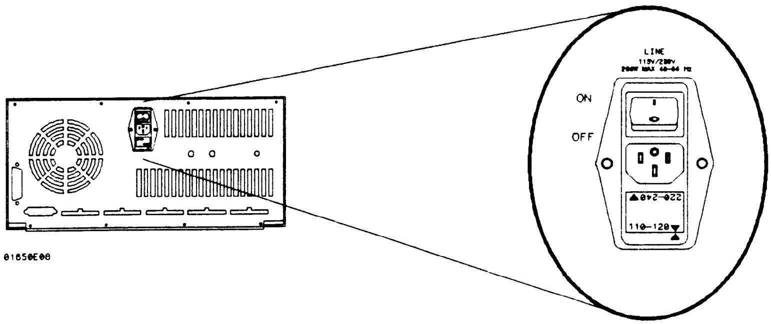


D-4. Installing The Operating System Disk

The logic analyer will read the disk and load the operating system. It will also run self-tests before it is ready for you to operate.

Line Switch

The line switch is located on the rear panel. You turn the instrument on by pressing the 1 on the rocker switch. Make sure the operating system disk is in the disk drive before you turn on the logic analyzer. If you forget the disk, don't worry, you won't harm anything. You will merely have to repeat the turn-on procedure with the disk in the drive.



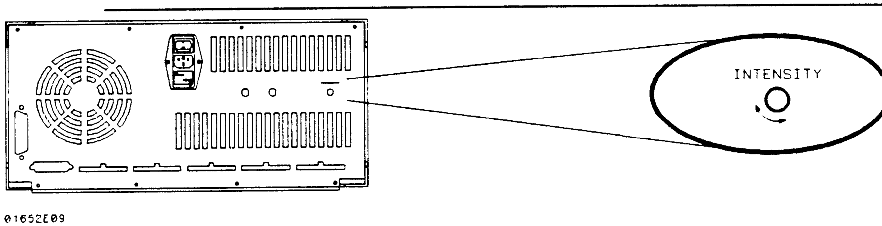
D-5. Line Switch

Intensity Control

Once you have turned the instrument on, you may want to set the display intensity to a level that's more comfortable for you. You do this by turning the INTENSITY control on the rear panel.

Note

A high intensity level setting may shorten the life of the CRT in your instrument.



D-6. Intensity Control

Operator's Maintenance

The only maintenance you need to do is clean the instrument exterior and periodically check the rear panel for air restrictions.

Use only MILD SOAP and WATER to clean the cabinet and front panel. DO NOT use a harsh soap which will damage the water-base paint finish of the instrument.

Calibration

The oscilloscope software calibration is accessed through the oscilloscope Trigger menu. The calibration procedures are performed from the front panel and without need to access internal circuits.

Calibration Interval

Software calibration should be performed if one of the following occurs:

- Oscilloscope board is installed, replaced or repaired.
- Ambient temperature changes more than 10° C.
- 6 months or 1000 hours of operation.

Calibration Integrity

Calibration constants are stored in system memory and not on the Operating System Disk. Therefore, software calibration is not required when a different Operating System Disk is used to boot the instrument on power-up.

Software Calibration Procedures

The following calibration procedures should be performed in their entirety and in the same sequence shown in this procedure. The following test equipment is recommended:

- BNC cable.
- DC Power Supply HP 6114A (± 0.1 % accuracy).
- Digital Voltmeter HP 3478A (± 0.025 % accuracy).
- BNC (female)-to-dual Banana Adapter.
- BNC -to-mini probe adapter

Note

An instrument warm-up of 15 minutes is recommended before starting these procedures. To abort any calibration procedure, use the front-panel knob to select the **Cancel** field, then press the **SELECT** key.

Offset Calibration

1. In the **System Configuration** menu turn both State/Timing analyzers Off, and turn the oscilloscope On.
2. Press the **TRACE/TRIG** key and select the **Calibration** field using the front-panel knob and SELECT key.

Note

Offset calibration should be the default Calibration menu setting. If not, select the Calibration choice field and, when the pop-up appears, select **Offset**.

---

3. Disconnect all signals from the channel 1 and 2 inputs. Select **Start** with the front-panel knob and SELECT key. A message will appear on screen to indicate the calibration is in process.
4. When the calibration is complete, the calibration status screen will appear.

Attenuator Calibration

The attenuator calibration is the only calibration that requires test equipment. If you are not using the recommended equipment listed on the previous page, make sure the substitute equipment meets the critical specifications listed in the *HP 1652B/1653B Service Manual.*

5. Select the Calibration choice field. When the pop-up appears, select **Attenuation**.

6. Connect the test equipment as shown below. The voltmeter monitors the voltage level to the oscilloscope.

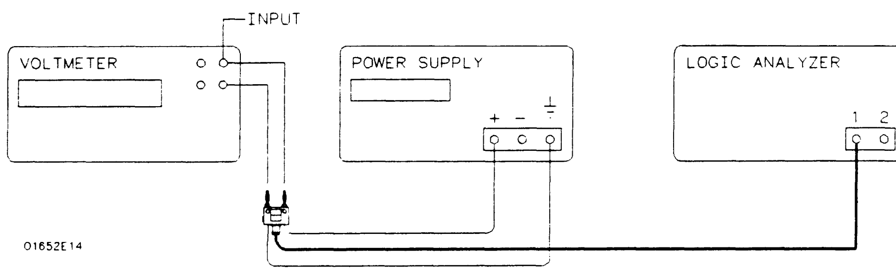


Figure D-7. Attenuator Calibration Equipment Setup

7. Select the **Start** field with the front-panel knob and SELECT key. The calibration screen will prompt you to connect the appropriate channel and set the DC voltage as specified.

8. Adjust the power supply to within ±0.1% of the specified voltage. If the measured voltage displayed on the voltmeter is greater than ±0.1 % from the specified voltage in *step 7*, you will have to compensate the oscilloscope as shown in *step a* below.

   a. Select the **Voltage** field and enter the measured voltage value, then select DONE

9. To proceed with the Attenuator calibration, select the **Continue** field. Repeat steps 8 and 9 for each specified voltage value.

10. When the calibration is complete, the updated calibration status screen will appear.

Gain Calibration

11. Disconnect all test equipment and all inputs to channel 1 and 2 of the oscilloscope.

12. Select the Calibration choice field. When the pop-up appears, select **Gain**, then select **Start** .

13. When the calibration is complete, the updated calibration status screen will appear.

Trigger Calibration

14. Make sure all signals are disconnected from the channel 1 and 2 inputs of the oscilloscope.

15. Select the Calibration choice field. When the pop-up appears, select **Trigger level**, then select **Start**.

16. When the calibration is complete, the updated calibration status screen will appear.

Delay Calibration

17. Select the Calibration choice field. When the pop-up appears, select **Delay**, then select **Start.**

18. Connect a BNC cable from the Probe Compensation output on the rear panel, to the channel 1 input. The instrument will prompt you when you need to switch to channel 2.

Note

You have the option of using the 10:1 scope probe in place of the recommended 1:1 BNC cable. If you use the scope probe, you will have to use the BNC-to-mini probe adapter supplied with the instrument and set attenuation field in *step 19* to 10:1.

---

19. Set the attenuation field in the calibration screen to the appropriate setting.

20. To proceed with the Delay calibration, select **Continue**.

21. When the calibration is complete, the updated calibration status screen will appear.

22. Calibration is now complete. Select **Done** with the front-panel knob and SELECT key to exit the Calibration menu.

Note

Do not execute **Set to Default** after calibrating the instrument. Otherwise, your calibration factors will be replaced by default calibration factors.

E

Operator Self Tests

Introduction

This appendix gives you an overview of the self tests the logic analyzer runs when you turn it on. You can also access the self tests from the I/O menu. This appendix is not intended to provide service information, but to acquaint you with the tests. If service is required, it should be performed by qualified service personnel.

Self Tests

The power-up self test is a set of tests that are automatically performed when you apply power to the logic analyzer. You may perform the self tests individually to have a higher level of confidence that the instrument is operating properly. A message that the instrument has failed a test will appear if any problem is encountered during a test. The individual self tests are listed in the self test menu which is accessed via the I/O menu. The HP 1652B/1653B self tests are on the operating system disk and the disk is required to run the tests.

Power-up Self Test

The power-up self test is automatically initiated at power-up by the HP 1652B/1653B Logic Analyzers. The revision number of the operating system firmware is given in the upper right of the screen during the power-up self test. As each test is completed, either "passed" or "failed" will be displayed before the test name as shown below.

PERFORMING POWER-UP SELF TESTS

passed ROM test
passed RAM test
passed Interrupt test
passed Display test
passed Keyboard test
passed Acquisition test
passed Threshold test
passed Disk test

LOADING SYSTEM FILE

When the power-up self testing is complete, the operating system will be automatically loaded. If the operating system disk is not in the disk drive, the message "SYSTEM DISK NOT FOUND" will be displayed at the bottom of the screen and "NO DISK" will be displayed in front of disk test in place of "passed."

If the "NO DISK" message appears, turn off the instrument, insert the operating system disk into the disk drive, and apply power again.

Selectable Self Tests

The following self tests may be accessed individually in the Self Test menu:

- Analyzer Data Acquisition
- Scope Data Acquisition
- RS-232C
- BNC
- Keyboard
- RAM
- ROM
- Disk Drive
- Cycle through tests

To select a test, place the cursor on the test name and press SELECT. A pop-up menu appears with a description of the test. The self test does not begin until the cursor is placed on Single or Repetitive Test and the SELECT key is pressed.

When the test is complete, either "Passed","Failed", or "Tested" will be displayed in the Self Test menu in front of the test. These tests are also used as troubleshooting aids. If a test fails, refer to Section 6 of the Service manual for information on the individual tests used for troubleshooting.

F

Specifications and Operating Characteristics

Introduction

This appendix lists the specifications, operating characteristics, and supplemental characteristics of the HP 1652B and HP 1653B Logic Analyzers.

Logic Analyzer Specifications

Probes

Minimum Swing: 600 mV peak-to-peak.

Threshold Accuracy:

Voltage Range	Accuracy
-2.0V to +2.0V	± 150 mV
-9.9V to -2.1V	± 300 mV
+2.1V to +9.9V	± 300 mV

Dynamic Range: ± 10 volts about the threshold.

State Mode

Clock Repetition Rate: Single phase is 35 MHz maximum (25 MHz on the HP 1653B). With time or state counting, minimum time between states is 60 ns (16.67 MHz). Both mixed and demultiplexed clocking use master-slave clock timing; master clock must follow slave clock by at least 10 ns and precede the next slave clock by ≥50 ns.

Clock Pulse Width: ≥10 ns at threshold.

Setup Time: Data must be present prior to clock transition, ≥10 ns.

Hold Time: Data must be present after rising clock transition; 0 ns.

Data must be present after falling clock transition, 0 ns (HP 1653B); data must be present after falling L clock transition, 0 ns (HP 1652B); data must be present after falling J, K, M, and N clock transition, 1 ns (HP 1652B).

Timing Mode

Minimum Detectable Glitch: 5 ns wide at the threshold.

Logic Analyzer Operating Characteristics

The following operating characteristics are not specifications, but are typical operating characteristics for the HP 1652B/1653B logic analyzer which are included as additional information for the user.

Probes

Input RC: 100 KΩ ± 2% shunted by approximately 8 pF at the probe tip.

TTL Threshold Preset: +1.6 volts.

ECL Threshold Preset: -1.3 volts.

Threshold Range: -9.9 to +9.9 volts in 0.1 volt increments.

Threshold Setting: Threshold levels may be defined for pods 1 and 2 individually (HP 1653B). Threshold levels may be defined for pods 1, 2, and 3 on an individual basis and one threshold may be defined for pods 4 and 5 (HP 1652B).

Minimum Input Overdrive: 250 mV or 30% of the input amplitude, whichever is greater.

Maximum Voltage: ± 40 volts peak.

Maximum Power Available Through Cables: 600 mA @ 5V per cable; 2 amp @ 5V per HP 1652B/1653B.

Measurement Configurations

Analyzer Configurations:

Analyzer 1	Analyzer 2
Timing	Off
Off	Timing
State	Off
Off	State
Timing	State
State	Timing
State	State
Off	Off

Channel Assignment: Each group of 16 channels (a pod) can be assigned to Analyzer 1, Analyzer 2, or remain unassigned. The HP 1652B contains 5 pods; the HP 1653B contains 2 pods.

State Analysis

Memory

Data Acquisition: 1024 samples/channel.

Trace Specification

Clocks: Five clocks (HP 1652B) or two clocks (HP 1653B) are available and can be used by either one or two state analyzers at any time. Clock edges can be ORed together and operate in single phase, two phase demultiplexing, or two phase mixed mode. Clock edge is selectable as positive, negative, or both edges for each clock.

Clock Qualifier: The high or low level of four ORed clocks (HP 1652B) or one clock (HP1653B) can be ANDed with the clock specification. Setup time: 20 ns; hold time: 5 ns.

Pattern Recognizers: Each recognizer is the AND combination of bit (0, 1, or X) patterns in each label. Eight pattern recognizers are available when one state analyzer is on. Four are available to each analyzer when two state analyzers are on.

Range Recognizers: Recognizes data which is numerically between or on two specified patterns (ANDed combination of zeros and/or ones). One range term is available and is assigned to the first state analyzer turned on. The maximum size is 32 bits and on a maximum of 2 pods.

Qualifier: A user-specified term that can be anystate, nostate, a single pattern recognizer, range recognizer, or logical combination of pattern and range recognizers.

Sequence Levels: There are eight levels available to determine the sequence of events required for trigger. The trigger term can occur anywhere in the first seven sequence levels.

Branching: Each sequence level has a branching qualifier. When satisfied, the analyzer will restart the sequence or branch to another sequence level.

Occurrence Counter: Sequence qualifier may be specified to occur up to 65535 times before advancing to the next level.

Storage Qualification: Each sequence level has a storage qualifier that specifies the states that are to be stored.

Enable/Disable: Defines a window of post-trigger storage. States stored in this window can be qualified.

Prestore: Stores two qualified states that precede states that are stored.

Tagging

State Tagging: Counts the number of qualified states between each stored state. Measurement can be shown relative to the previous state or relative to trigger. Maximum count is 4.4 X (10 to the 12th power).

Time Tagging: Measures the time between stored states, relative to either the previous state or to the trigger. Maximum time between states is 48 hours.

With tagging on, the acquisition memory is halved; minimum time between states is 60 ns.

Symbols

Pattern Symbols: User can define a mnemonic for the specific bit pattern of a label. When data display is SYMBOL, a mnemonic is displayed where the bit pattern occurs. Bit patterns can include zeros, ones, and don't cares.

Range Symbols: User can define a mnemonic covering a range of values. Bit pattern for lower and upper limits must be defined as a pattern of zeros and ones. When data display is SYMBOL, values within the specified range are displayed as mnemonic + offset from base of the range.

Number of Pattern and Range Symbols: 200 per HP 1652B/1653B.

Symbols can be down-loaded over RS-232C and HP-IB.

State Compare Mode

Performs post-processing bit-by-bit comparison of the acquired state data and compare data image.

Compare Image: Created by copying a state acquisition into the compare image buffer. Allows editing of any bit in the compare image to a zero, one, or don't care.

Compare Image Boundaries: Each channel (column) in the compare image can be enabled or disabled via bit masks in the compare image. Upper and lower ranges of states (rows) in the compare image can be specified. Any data bits that do not fall within the enabled channels and the specified range are not compared.

Stop Measurement: Repetitive acquisitions may be halted when the comparison between the current state acquisition and the current compare image is equal or not equal.

Displays: Compare Listing display shows the compare image and bit masks; Difference Listing display highlights differences between the current state acquisition and the current compare image.

State X-Y Chart Display

Plots the value of the specified label (on the y-axis) versus states or another label (on the x-axis). Both axes can be scaled by the user.

Markers: Correlated to state listing, state compare, and state waveform displays. Available as pattern, time, or statistics (with time counting on), and states (with state counting on).

Accumulate: Chart display is not erased between successive acquisitions.

State Waveform Display

Displays a state acquisition in a waveform format.

States/div: 1 to 104 states.

Delay: 0 to 1024 states.

Accumulate: Waveform display is not erased between successive acquisitions.

Overlay Mode: Multiple channels can be displayed on one waveform display line. Primary use is to view a summary of bus activity.

Maximum Number of Displayed Waveforms: 24.

Markers: Correlated to state listing, state compare, and X-Y chart displays. Available as pattern, time, or statistics (with time counting on), and states (with state counting on).

Timing Analysis

Transitional Timing Mode

Sample is stored in acquisition memory only when the data changes. A time tag stored with each sample allows reconstruction of a waveform display. Time covered by a full memory acquisition varies with the number of pattern changes in the data.

Sample Period: 10 ns.

Maximum Time Covered By Data: 5000 seconds.

Minimum Time Covered by Data: 10.24 us.

Glitch Capture Mode

Data sample and glitch information is stored every sample period.

Sample Period: 20 ns to 50 ms in a 1-2-5 sequence dependent on sec/div and delay settings.

Memory Depth: 512 samples/channel.

Time Covered by Data: Sample period X 512

Waveform Display

Sec/div: 10 ns to 100 s; 0.01% resolution.

Screen Delay: -2500 s to 2500 s; presence of data is dependent on the number of transitions in data between trigger and trigger plus delay (transitional timing).

Accumulate: Waveform display is not erased between successive acquisitions.

Hardware Delay: 20 ns to 10 ms.

Overlay Mode: Multiple channels can be displayed on one waveform display line. Primary use is to view a summary of bus activity.

Maximum Number Of Displayed Waveforms: 24

Time Interval Accuracy

Channel to Channel Skew: 4 ns typical.

Sample Period Accuracy: 0.01% of sample period.

Time Interval Accuracy: ± (sample period + channel-to-channel skew + 0.01% of time interval reading).

Trigger Specification

Asynchronous Pattern: Trigger on an asynchronous pattern less than or greater than a specified duration. Pattern is the logical AND of a specified low, high, or don't care for each assigned channel. If pattern is valid but duration is invalid, there is a 20 ns reset time before looking for patterns again.

Greater Than Duration: Minimum duration is 30 ns to 10 ms with 10 ns or 0.01% resolution, whichever is greater. Accuracy is +0 ns to -20 ns. Trigger occurs at pattern + duration.

Less Than Duration: Maximum duration is 40 ns to 10 ms with 10 ns or 0.01% resolution, whichever is greater. Pattern must be valid for at least 20 ns. Accuracy is +20 ns to -0 ns. Trigger occurs at the end of the pattern.

Glitch/Edge Triggering: Trigger on a glitch or edge following a valid duration of an asynchronous pattern while the pattern is still present. Edge can be specified as rising, falling, or either. Less than duration forces glitch and edge triggering off.

Measurement and Display Functions

Autoscale (Timing Analyzer Only)

Autoscale searches for and displays channels with activity on the pods assigned to the timing analyzer.

Acquisition Specifications

Arming: Each analyzer can be armed by the run key, the other analyzer, or the external trigger in port.

Trace Mode: Single mode acquires data once per trace specification; repetitive mode repeats single mode acquisitions until stop is pressed or until the time interval between two specified patterns is less than or greater than a specified value, or within or not within a specified range. There is only one trace mode when two analyzers are on.

Labels

Channels may be grouped together and given a six character name. Up to 20 labels in each analyzer may be assigned with up to 32 channels per label. Primary use is for naming groups of channels such as address, data, and control busses.

Indicators

Activity Indicators: Provided in the Configuration, State Format, and Timing Format menus for identifying high, low, or changing states on the inputs.

Markers: Two markers (X and 0) are shown as dashed lines on the display.

Trigger: Displayed as a vertical dashed line in the timing waveform display and as line 0 in the state listing display.

Marker Functions

Time Interval: The X and 0 markers measure the time interval between one point on a timing waveform and trigger, two points on the same timing waveform, two points on different waveforms, or two states (time tagging on).

Delta States (State Analyzer Only): The X and 0 markers measure the number of tagged states between one state and trigger, or between two states.

Patterns: The X and 0 markers can be used to locate the nth occurrence of a specified pattern before or after trigger, or after the beginning of data. The 0 marker can also find the nth occurrence of a pattern before or after the X marker.

Statistics: X to 0 marker statistics are calculated for repetitive acquisitions. Patterns must be specified for both markers and statistics are kept only when both patterns can be found in an acquisition. Statistics are minimum X to 0 time, maximum X to 0 time, average X to 0 time, and ratio of valid runs to total runs.

Run/Stop Functions

Run: Starts acquisition of data in a specified trace mode.

Stop: In single trace mode or the first run of a repetitive acquisition, STOP halts the acquisition and displays the current acquisition data. For subsequent runs in repetitive mode, STOP halts the acquisition of data and does not change current display.

Data Display/Entry

Display Modes: State listing; timing waveforms; interleaved, time-correlated listing of two state analyzers (time tagging on); time-correlated state listing and timing waveform display (state listing in upper half, timing waveform in lower half, and time tagging on).

Timing Waveform: Pattern readout of timing waveforms at X or 0 marker.

Bases: Binary, Octal, Decimal, Hexadecimal, ASCII (display only), and User-defined symbols.

Oscilloscope Specifications

The following specifications are the performance standards or limits against which the oscilloscope in the HP 1652B/1653B is tested.

Vertical

Bandwidth (-3 dB): dc to 100 MHz (single shot).

DC Gain Accuracy: ± 3% of full scale.

DC Offset Accuracy: ± (2 mV + 2% of the channel offset + 2.5% of full scale).

Voltage Measurement Accuracy (DC): (Gain accuracy + ADC resolution + Offset accuracy).

Horizontal

Time Interval Measurement Accuracy: ±(2% X s/div + 0.01% X delta-t + 500 ps).

Trigger

Sensitivity: 10% of full screen.

Oscilloscope Operating Characteristics

The following operating characteristics are not specifications, but are typical operating characteristics for the oscilloscope in the HP 1652B/1653B. These are included as additional information for the user.

Vertical (at BNC)

Transition Time (10% to 90%): ≤ 3.5 ns.

Number of Channels: 2.

Vertical Sensitivity Range: 15 mV/div to 10 V/div (1:1 probe).

Vertical Sensitivity Resolution: Adjustable 2 digit resolution.

Maximum Sample Rate: 400 MSamples/second.

Analog-to-Digital Conversion: 6 bit real-time.

Analog-to-Digital Resolution: ±1.6% of full scale.

Waveform Record Length: 2048 points.

Input R: 1 MΩ ± 1% or 50 Ω ± 1%.

Input C: Approximately 7 pF.

Input Coupling: dc.

Maximum Safe Input Voltage:

1 MΩ input,	± 250 V [dc + peak ac (< 10 kHz)]
50 Ω input,	± 5 V RMS

DC Offset Range (1:1 Probe):

Vertical Sensitivity	Available Offset
≤50 mV/div	±2.0 V
100 mV/div - 200 mV/div	±10 V
500 mV/div - 1 V/div	±50 V
≥2 V/div	±125 V

±5 V max if input impedance is at 50 Ω.

DC Offset Resolution (1:1 Probe)

Vertical Sensitivity	Resolution
≤50 mV/div	200 uV
100 mV/div - 200 mV/div	1 mV
500 mV/div - 1 V/div	5 mV
≥2 V/div	25 mV or 4 digits of resolution, whichever is greater.

Probe Factors: Any integer ratio from 1:1 to 1000:1.

Channel Isolation:

40 dB: dc to 50 MHz.

30 dB: 50 MHz to 100 MHz (with channels at equal sensitivity).

Horizontal

Timebase Range: 5 ns/div to 5 s/div.

Timebase Resolution:

Time/Division Setting	Resolution
t < 10 ns/div	100 ps
t≥10 ns/div	adjustable with 3-digit resolution

Delay Pre-trigger Range: 5 X (s/div) @ 5 ns ≤ s/div ≤ 500 ns
2.5 μs X (s/div) @ 500 ns ≤ s/div ≤ 5 s

Delay Post-trigger Range:

Time/Division Setting	Available Delay
50 ms - 5 s/div	40 X (s/div)
100 μs - 20 ms/div	1 s
5 ns - 50 μs/div	10,000 X (s/div)

Trigger

Triggering on either input channel, rising or falling edge.

Trigger Level Range: dc Offset ±5 divisions.

Trigger Level Resolution (1:1 Probe):

Trigger Level	Resolution
≤50 mV/div	400 μV
100 mV/ div - 200 mV/div	2 mV
500 mV/div - 1 V/div	10 mV
≥2 V/div	50 mV

Arming: Armed by the Run key, external BNC low input, or by Analyzer 1 or 2.

Trigger Modes

Immediate: Triggers immediately after the arming condition is met.

Edge: Triggers on the rising or falling edge from channel 1 or 2.

Auto-Trigger: Self-triggers if no trigger condition is found within approximately 1 second after arming.

Trigger Out: Arms Analyzer 1 or 2, or triggers the rear panel BNC.

Waveform Display

Display Formats: 1 to 8 oscilloscope waveforms can be displayed.

Display Resolution: 500 points horizontally, 240 points vertical.

Display Modes

Normal: New acquisitions replace old acquisitions on screen.

Accumulate: New acquisitions are added to the screen and displayed with previous acquisitions until a parameter is changed and a new acquisition is made.

Average: New acquisitions are averaged with older acquisitions and displayed. Maximum number of averages is 256.

Overlay: Channel 1 and 2 can be overlayed in the same display area.

Connect-the-dots: Provides a display of the sample points connected by straight lines.

Waveform Reconstruction: A reconstruction filter fills in missing data points when timebase is ≤ 100 ns/Div or when timebase is reduced to a setting where fewer than 500 samples are on screen.

Waveform Math: Display capability of A-B and A + B functions is provided.

Mixed Mode: Oscilloscope plus logic analyzer displays on the same screen.

Measurement Aids

Time Markers: Two vertical markers labeled X and O. Voltage levels are displayed for each marker. Time interval measurements can be made between any two events.

Automatic Search: Searches for a specified absolute or percentage voltage level at a positive or negative edge, count adjustable from 1 to 1024.

Auto Search Statistics: Mean, maximum, and minimum values for elapsed time from X to O markers for multiple runs. Number of valid runs and total number of runs displayed.

Trigger Level Marker: Horizontal trigger level marker displayed in Trace/Trigger menu only.

Automatic Measurements: The following pulse parameter measurements can be performed automatically:

Frequency
Period
V p-p
Rise time
Fall time
Preshoot
Overshoot
+ pulse width
- pulse width

Grid: May be turned on or off.

Setup Aids

Autoscale: Auto sets the vertical and horizontal ranges, offset, and trigger levels to display the input signals. Requires an amplitude above 10 mV peak, and a frequency between 50 Hz and 100 MHz.

Preset: Scales the vertical range, offset, and trigger level to predetermined values for displaying ECL or TTL waveforms.

Calibration: Attenuation, offset, gain, trigger, and delay set to defaults.

Probe Compensation Source: External BNC supplies square wave approximately -400 mV to -900 mV at approximately 1.25 kHz.

Interactive Measurements

Acquisition

Oscilloscope, timing, and state can occur simultaneously or in series.

Mixed Displays

Timing channels and oscilloscope channels can be displayed on the same screen. Multiple state machine listings can be displayed with time tags on the same screen. Timing channels can be displayed with a state listing with Time Tags turned on. State listings with time tags, timing channels, and oscilloscope channels can be displayed on the same screen.

Time Correlation

All modules are time correlated with the exception of when the oscilloscope is being armed by the logic analyzer, and when the oscilloscope is not in trigger immediate mode.

Time Interval Accuracy Between Modules

Equals the sum of channel to channel time interval accuracies of each machine used for a measurement.

General Characteristics

The following general characteristics for the HP 1652B/1653B include the environment operating conditions, shipping weights, and instrument dimensions.

Operating Environment

Temperature

Instrument:

Operating: 0°C to +55°C (32°F to +131°F)
Non-operating: -40°C to +70°C (-40°F to +158°F)

Probes and Cables: 0°C to 65°C (+32°F to +149°F)

Disk Media: 10°C to 50°C (+50°F to +149°F).

Humidity:

Instrument:

Operating: Up to 95% relative humidity (non-condensing) at +40°C (+104°F)
Non-operating: Up to 90% relative humidity at +65°C (+149°F)

Disk Media: 8% to 80% relative humidity at +40°C (+104°F)

Altitude

Operating: Up to 4600 meters (15,000 ft)
Non-operating: Up to 15,300 meters (50,000 ft)

Vibration

Operating: Random vibration 5-500 Hz, 10 minutes per axis, 0.3 g (rms)
Non-operating: Random vibration 5-500 Hz, 10 minutes per axis, 2.41 g (rms);
Resonant search 5-500 Hz swept sine, 1 Octave/minute sweep rate, 0.75 g (0-peak), 5 minute resonant dwell at 4 resonances per axis.

Power Requirements 115/230 Vac, -25% to +15%, 48-66 Hz, 200 W max.

Weight 10.0 kg (22 lbs) net; 18.2 kg (40 lbs) shipping.

Dimensions Refer to the outline drawing below.

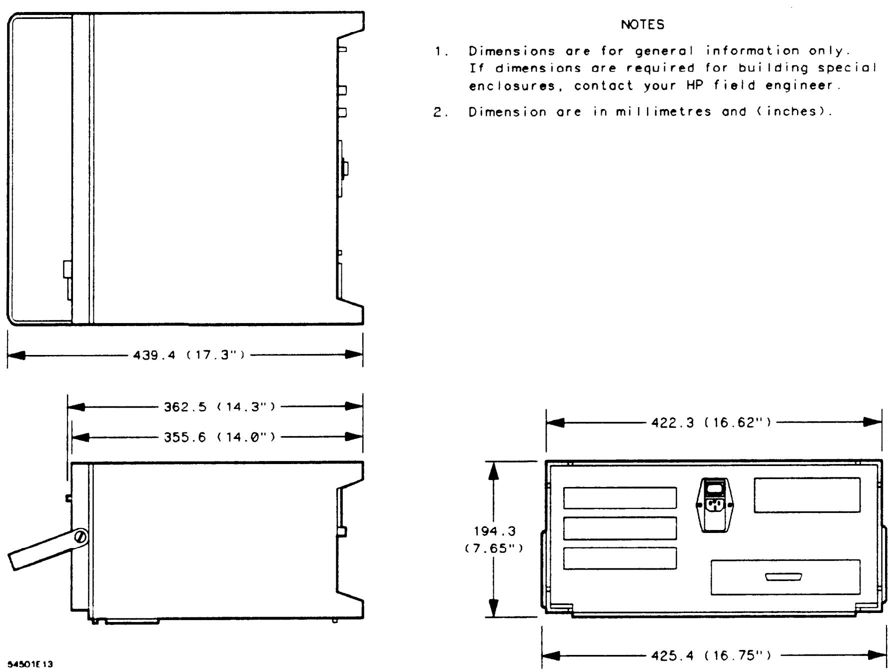


Figure F-1. HP 1652B/1653B Dimensions

Index

E

F

Q

R

S

T

U

V

W

X

HEWLETT
PACKARD

Printed in U.S.A.

HP 1652B/1653B Logic Analyzers
Front-Panel Operation Reference
Volume 2 of 2

HEWLETT
PACKARD

Service Manual

HP 1652B/1653B Logic Analyzers

HEWLETT PACKARD

HEWLETT PACKARD

SERVICE MANUAL

HP 1652B/1653B

Logic Analyzers

SERIAL NUMBERS

This manual applies directly to instruments prefixed with serial number:

2941A/2942A/3011A/3012A

For Additional Information about serial numbers see INSTRUMENTS COVERED BY THIS MANUAL in Section 1.

Manual Part No. 01652-90905
Microfiche Part No. 01652-90805

Printed in U.S.A. February 1990

CERTIFICATION

Hewlett-Packard Company certifies that this product met its published specifications at the time of shipment from the factory. Hewlett-Packard further certifies that its calibration measurements are traceable to the United States National Institute of Standards and Technology, to the extent allowed by the Institute's calibration facility, and to the calibration facilities of other International Standards Organization members.

WARRANTY

This Hewlett-Packard product is warranted against defects in material and workmanship for a period of one year from date of shipment. During the warranty period, Hewlett-Packard Company will, at its option, either repair or replace products which prove to be defective.

For warranty service or repair, this product must be returned to a service facility designated by HP. Buyer shall prepay shipping charges to HP and HP shall pay shipping charges to return the product to Buyer. However, Buyer shall pay all shipping charges, duties, and taxes for products returned to HP from another country.

HP warrants that its software and firmware designated by HP for use with an instrument will execute its programming instructions when properly installed on that instrument. HP does not warrant that the operation of the instrument or software, or firmware will be uninterrupted or error free.

LIMITATION OF WARRANTY

The foregoing warranty shall not apply to defects resulting from improper or inadequate maintenance by Buyer, buyer-supplied software or interfacing, unauthorized modification or misuse, operation outside the environmental specifications for the product, or improper site preparation or maintenance.

NO OTHER WARRANTY IS EXPRESSED OR IMPLIED. HP SPECIFICALLY DISCLAIMS THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

EXCLUSIVE REMEDIES

THE REMEDIES PROVIDED HEREIN ARE BUYER'S SOLE AND EXCLUSIVE REMEDIES. HP SHALL NOT BE LIABLE FOR ANY DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES, WHETHER BASED ON CONTRACT, TORT, OR ANY OTHER LEGAL THEORY.

ASSISTANCE

Product maintenance agreements and other customer assistance agreements are available for Hewlett-Packard products.

For any assistance, contact your nearest Hewlett-Packard Sales and Service Office. Addresses are provided at the back of this manual.

CW3A789

Safety Considerations

General Operation

This is a Safety Class I instrument (provided with terminal for protective earthing). BEFORE APPLYING POWER verify that the power transformer primary is matched to the available line voltage, the correct fuse is installed, and Safety Precautions are taken (see the following warnings). In addition, note the instrument's external markings which are described under "Safety Symbols."

General Warnings and Cautions

- BEFORE SWITCHING ON THE INSTRUMENT, the protective earth terminal of the instrument must be connected to the protective conductor of the (mains) powercord. The mains plug shall only be inserted in a socket outlet provided with a protective earth contact. The protective action must not be negated by the use of an extension cord (power cable) without a protective conductor (grounding). Grounding one conductor of a two-conductor outlet is not sufficient protection.
- Servicing instructions are for use by service-trained personnel. To avoid dangerous electric shock, do not perform any servicing unless qualified to do so.
- If this instrument is to be energized via an auto-transformer (for voltage reduction) make sure the common terminal is connected to the earth terminal of the power source.
- Any interruption of the protective (grounding) conductor (inside or outside the instrument) or disconnecting the protective earth terminal will cause a potential shock hazard that could result in personal injury.
- Whenever it is likely that the protection has been impaired, the instrument must be made inoperative and be secured against any unintended operation.
- Only fuses with the required rated current, voltage, and specified type (normal blow, time delay, etc.) should be used. Do not use repaired fuses or short circuited fuseholders. To do so could cause a shock or fire hazard.
- Do not operate the instrument in the presence of flammable gasses or fumes. Operation of any electrical instrument in such an environment constitutes a definite safety hazard.
- Do not install substitute parts or perform any unauthorized modification to the instrument.
- Adjustments described in the manual are performed with power supplied to the instrument while protective covers are removed. Energy available at many points may, if contacted, result in personal injury.
- Any adjustment, maintenance, and repair of the opened instrument under voltage should be avoided as much as possible, and when inevitable, should be carried out only by a skilled person who is aware of the hazard involved.
- Capacitors inside the instrument may still be charged even if the instrument has been disconnected from its source of supply.

Safety Symbols

Instruction manual symbol. The product will be marked with this symbol when it is necessary for the user to refer to the instruction manual in order to protect against damage to the product.

Indicates Hazardous Voltages

Earth terminal (sometimes used in manual to indicate circuit common connected to grounded chassis).

Warning 

The WARNING sign denotes a hazard. It calls attention to a procedure, practice, or the like, which, if not correctly performed or adhered to, could result in personal injury. Do not proceed beyond a WARNING sign until the indicated conditions are fully understood and met.

Caution 

The CAUTION sign denotes a hazard. It calls attention to an operating procedure, practice, or the like, which, if not correctly performed or adhered to, could result in damage to or destruction of part or all of the product. Do not proceed beyond a CAUTION sign until the indicated conditions are fully understood or met.

Printing History

New editions are complete revisions of the manual. Update packages, which are issued between editions, contain additional and replacement pages to be merged into the manual by the customer. The dates on the title page change only when a new edition is published.

A software and/or firmware code may be printed before the date; this indicates the version level of the software and/or firmware of this product at the time of the manual or update was issued. Many product updates and fixes do not require manual changes and, conversely, manual corrections may be done without accompanying product changes. Therefore, do not expect a one to one correspondence between product updates and manual updates.

Edition 1	February 1990	01652-90905

List of Effective Pages

The List of Effective Pages gives the date of the current edition and of any pages changed in updates to that edition. Within the manual, any page changed since the last edition is indicated by printing the date the changes were made on the bottom of the page. If an update is incorporated when a new edition of the manual is printed, the change dates are removed from the bottom of the pages and the new edition date is listed in Printing History and on the title page.

Pages	**Effective Date**
all	February 1990

Contents

Section 1:

General Information

Section 2:

Installation

Section 3: Performance Tests

Section 4: Adjustments and Calibration

Section 5: Replaceable Parts

Section 6A: Theory of Operation

Section 6B: Self Tests

Section 6C: Troubleshooting

Section 6D: Assembly Removal and Replacement

Contents

Section 1: General Information

General Information 1

Introduction

This Service Manual explains how to test, adjust, and service the Hewlett-Packard 1652B/1653B Logic Analyzer. This manual is divided into six sections:

- 1 - General Information.
- 2 - Installation.
- 3 - Performance Tests.
- 4 - Adjustments and Calibration.
- 5 - Replaceable Parts.
- 6 - Service.

For easier access, the Service section is presented in four sub-sections:

- 6A - Theory of Operation.
- 6B - Self Tests.
- 6C - Troubleshooting.
- 6D - Assembly Removal and Replacement.

Information for operating, programming, and interfacing the HP 1652B/1653B is contained in the HP 1652B/1653B Operating and Programming manual set supplied with each instrument.

Section 1, "General Information," includes a description of the HP 1652B/1653B logic analyzer, including its specifications, options, available accessories, and recommended test equipment for maintaining the instrument.

Listed on the title page of this manual is a microfiche part number. This number can be used to order 4 by 6- inch microfilm transparencies of the manual. Each microfiche contains up to 96 photo-duplicates of the manual pages. The microfiche package also includes the latest Manual Changes supplement and pertinent Service Notes.

Instruments Covered by this Manual

The instrument serial number is located on the rear panel. Hewlett-Packard uses a two part serial number consisting of a four-digit prefix and a five-digit suffix separated by a letter (for example, 0000A00000). The prefix is the same for all identical instruments and changes only when a modification is made that affects parts compatibility. The suffix is assigned and is different for each instrument. This manual applies directly to instruments with the serial prefix shown on the title page.

An instrument manufactured after the printing of this manual may have a serial number prefix that is not listed on the title page. This unlisted serial prefix indicates the instrument is different from those described in this manual. The manual for this newer instrument is accompanied by a Manual Changes supplement. This supplement contains "change information" that explains how to adapt the manual to the newer instrument.

In addition to change information, the supplement may contain information for correcting errors in the manual. To keep this manual as current and accurate as possible, Hewlett-Packard recommends that you periodically request the latest Manual Changes supplement. The supplement for this manual is identified with the manual print date and part number, both of which appear on the manual title page. Complimentary copies of the supplement are available from Hewlett-Packard.

Safety Considerations

This product is a Safety Class 1 instrument (provided with a protective earth terminal). Review the instrument and manual for safety markings and instructions before you begin operating this instrument. Specific warnings, cautions, and instructions are placed wherever applicable throughout the manual. These precautions must be observed during all phases of operation, service, and repair of the instrument. Failure to comply with these precautions, or with specific warnings elsewhere in this manual, violates safety standards of design, manufacture, and intended use of this instrument.

Hewlett-Packard assumes no liability for the customer's failure to comply with these safety requirements.

Product Description

The HP 1652B/1653B logic analyzers are general purpose instruments featuring measurement capabilities in all three domains of interest to the digital system designer: Analog, Timing, and State. Each of these domains is available to the user separately or in an interactive combination.

The HP 1652B includes an 80-channel, 35 MHz state, 100 MHz timing logic analyzer, selectable in 16 channel groupings with a 2-channel, 100 MHz, 400 Msample/s digitizing oscilloscope. The HP 1653B includes an 32-channel, 25 MHz state, 100 MHz timing logic analyzer, also selectable in 16 channel groupings with a 2-channel, 100 MHz, 400 Msample/s digitizing oscilloscope. Both analyzers can be configured as two independent state analyzers or one state and one timing analyzer. Two channels of oscilloscope measurement can be added to any configuration. Some of the main features of the analyzer include the following:

- Simultaneous state/state, or simultaneous state/timing analysis.
- Time interval; number of states; pattern search; minimum, maximum, and average time interval statistics.
- Transitional timing to store data only when there is a transition.
- Clock qualifiers, storage qualification, time and number of state tagging, and prestore.
- Small lightweight probing.

Some of the main features of the digitizing oscilloscope include the following:

- 2 channels of 400 Msamples/s digitizing for 100 MHz bandwidth single-shot analysis.
- 2k memory depth
- Automatic pulse parameters which display time between markers, acquires until capturing specified time between markers, and performs statistical analysis on time between markers.
- Arming by either analyzer or BNC input.
- 60 mV through 40 V full screen resolution.
- Lightweight miniprobes.

Other main features of the HP 1652B/1653B include the following:

- A user interface consisting of a panel keyboard with a Rotary Pulse Generator (RPG) knob.
- Nine-inch white phosphor, high resolution monitor.
- 3.5-inch floppy disk drive.
- HP-IB and RS-232C interfaces for hardcopy output to a printer or controller interface.

Accessories Supplied

The following accessories are supplied with the HP 1652B/1653B Logic Analyzer:

- Woven probe cable (HP part number 01650-61607) with a 40-pin connector on each side, 17 signal lines, 18 return lines, 2 chassis ground lines, and 2 power lines. Each power line supplies +5 volts for preprocessor power. Each cable supplies 600 milliamperes with a maximum power available from the HP 1652B/1653B of 2 amperes. Five probe cables are supplied with the HP 1652B and two are supplied with the HP 1653B.
- Probe Tip Assemblies (HP part number 01650-61608) that provide 16 data channels, 1 clock channel, and 1 ground lead per pod assembly. The probe input specifications are listed in the Logic Analyzer Specifications of this section. Five Probe Tip Assemblies are supplied with the HP 1652B and two are supplied with the HP 1653B.
- Grabbers for the probe tip assemblies are supplied in packages of 20 (HP part number 5959-0288). One-hundred grabbers (5 packages) are supplied with the HP 1652B and 40 grabbers (2 packages) are supplied with the HP 1653B.
- Two HP 10430A 10:1, 1 MΩ, 6.5 pF, 1 m mini-probes.
- Two right angle BNC adapters (HP part number 1250-0076).
- One BNC-to-mini probe adapter (HP part number 1250-1454).
- One Operating System Disk.
- One Performance Verification Disk.
- One 2.3 meter (7.5 feet) Power Cord (see section 2, "Installation," for the available power cords).
- One Operating and Reference Manual Set.
- One Programming Reference Manual.
- One Service Manual.
- One RS-232C Loopback Connector.

Accessories Available

The following accessories are available for the HP 1652B/1653B Logic Analyzer:

- Termination Adapter (HP part number 01650-63201).
- HP Model 10269C General Purpose Probe Interface to connect the logic analyzer directly to microprocessor preprocessors.
- Preprocessors for specific microprocessors and bus systems (for more information see your Hewlett-Packard Sales/Service Offices).
- 10:1, 100:1, 10 MΩ, 10 pf resistive divider probe set, 1.5 m (HP 10020A).
- BNC to BNC cable, 1.2 m (HP 10503A).
- 24-pin IC test clip (HP 10211A).
- BNC-to-BNC ac coupling capacitor (HP 10240B).

10:1 Probes:

- 1 MΩ, 7.5 pF miniprobe, 1 m (HP 10435A).
- 1 MΩ, 10 pF miniprobe, 2 m (HP 10433A).

1:1 Probes:

- 36 pF miniprobe, 1 m (HP 10438A).
- 62 pF miniprobe, 2 m (HP 10439A).
- 50 Ω miniprobe, 2 m (HP 10437A).

100:1 Probes:

- 10 MΩ, 2.5 pF miniprobe, 2 m (HP 10440A).

- Soft Carrying Case (HP part number 1540-1066).
- HP Model 1008A Option 006 Testmobile.
- HP Model 92192A 3.5-inch Microfloppy Disks (box of ten).
- Rackmount Kit (HP part number 5061-6175).

Logic Analyzer Specifications

The following specifications are the performance standards or limits against which the HP 1652B/1653B logic analyzer is tested.

Probes

Minimum Swing: 600 mV peak-to-peak.

Threshold Accuracy:

Voltage Range	Accuracy
-2.0V to +2.0V	± 150 mV
-9.9V to -2.1V	± 300 mV
+2.1V to +9.9V	± 300 mV

State Mode

Clock Repetition Rate: Single phase is 35 MHz maximum (25 MHz on the HP 1653B). With time or state counting, minimum time between states is 60 ns (16.67 MHz). Both mixed and demultiplexed clocking use master-slave clock timing. The master clock must follow the slave clock by at least 10 ns and precede the next slave clock by ≥50 ns.

Clock Pulse Width: ≥10 ns at threshold.

Setup Time: Data must be present prior to the clock transition, ≥10 ns.

Hold Time: Data must be present after the rising clock transition, 0 ns.

Data must be present after the falling clock transition, 0 ns (HP 1653B). Data must be present after the falling L clock transition, 0 ns (HP 1652B). Data must be present after the falling J, K, M, and N clock transition, 1 ns (HP 1652B).

Timing Mode

Minimum Detectable Glitch: 5 ns wide at the threshold.

Logic Analyzer Operating Characteristics

The following operating characteristics are not specifications, but are typical operating characteristics for the HP 1652B/1653B logic analyzer. These characteristics are included as additional information for the user.

Probes

Input RC: 100 KΩ ± 2% shunted by approximately 8 pF at the probe tip.

Dynamic Range: ± 10 volts about the threshold.

TTL Threshold Preset: +1.6 volts.

ECL Threshold Preset: -1.3 volts.

Threshold Range: -9.9 to +9.9 volts in 0.1 volt increments.

Threshold Setting: Threshold levels may be defined for pods 1 and 2 individually (HP 1653B). Threshold levels may be defined for pods 1, 2, and 3 on an individual basis and one threshold may be defined for pods 4 and 5 (HP 1652B).

Minimum Input Overdrive: 250 mV or 30% of the input amplitude, whichever is greater.

Maximum Voltage: ± 40 volts peak.

Maximum Power Available Through Cables: 600 mA at 5V per cable; 2 amp @ 5V per HP 1652B/1653B.

Measurement Configurations

Analyzer Configurations:

Analyzer 1	Analyzer 2
Timing	Off
Off	Timing
State	Off
Off	State
Timing	State
State	Timing
State	State
Off	Off

Channel Assignment: Each group of 16 channels (a pod) can be assigned to Analyzer 1, Analyzer 2, or remain unassigned. The HP 1652B contains 5 pods; the HP 1653B contains 2 pods.

State Analysis

Memory

Data Acquisition: 1024 samples/channel.

Trace Specification

Clocks: Five clocks (HP 1652B) or two clocks (HP 1653B) are available and can be used by either one or two state analyzers at any time. Clock edges can be ORed together and operate in single phase, two phase demultiplexing, or two phase mixed mode. The clock edge is selectable as positive, negative, or both edges for each clock.

Clock Qualifier: The high or low level of four ORed clocks (HP 1652B) or one clock (HP1653B) can be ANDed with the clock specification. Setup time: 20 ns; hold time: 5 ns.

Pattern Recognizers: Each recognizer is the AND combination of bit (0, 1, or X) patterns in each label. Eight pattern recognizers are available when one state analyzer is on. Four are available to each analyzer when two state analyzers are on.

Range Recognizers: Recognizes data which is numerically between or on two specified patterns (ANDed combination of zeros and/or ones). One range term is available and is assigned to the first state analyzer turned on. The maximum size is 32 bits and on a maximum of 2 pods.

Qualifier: A user-specified term that can be anystate, nostate, a single pattern recognizer, range recognizer, or logical combination of pattern and range recognizers.

Sequence Levels: There are eight levels available to determine the sequence of events required for trigger. The trigger term can occur anywhere in the first seven sequence levels.

Branching: Each sequence level has a branching qualifier. When satisfied, the analyzer will restart the sequence or branch to another sequence level.

Occurrence Counter: Sequence qualifier may be specified to occur up to 65535 times before advancing to the next level.

Storage Qualification: Each sequence level has a storage qualifier that specifies the states that are to be stored.

Enable/Disable: Defines a window of post-trigger storage. States stored in this window can be qualified.

Prestore: Stores two qualified states that precede states that are stored.

Tagging

State Tagging: Counts the number of qualified states between each stored state. A measurement can be shown relative to the previous state or relative to trigger. Maximum count is 4.4 X (10 to the 12th power).

Time Tagging: Measures the time between stored states, relative to either the previous state or to the trigger. Maximum time between states is 48 hours.

With tagging on, the acquisition memory is halved; minimum time between states is 60 ns.

Symbols

Pattern Symbols: A mnemonic can be defined for the specific bit pattern of a label. When the data display is SYMBOL, a mnemonic is displayed where the bit pattern occurs. Bit patterns can include zeros, ones, and don't cares.

Range Symbols: A mnemonic can be defined covering a range of values. Bit pattern for lower and upper limits must be defined as a pattern of zeros and ones. When the data display is SYMBOL, values within the specified range are displayed as mnemonic + offset from the base of the range.

Number of Pattern and Range Symbols: 200 per HP 1652B/1653B.

Symbols can be down-loaded over RS-232C and HP-IB.

State Compare Mode

This mode performs a post-processing bit-by-bit comparison of the acquired state data and the compare data image.

Compare Image: This is created by copying a state acquisition into the compare image buffer. It allows editing of any bit in the compare image to a zero, one, or don't care.

Compare Image Boundaries: Each channel (column) in the compare image can be enabled or disabled via bit masks in the compare image. Upper and lower ranges of states (rows) in the compare image can be specified. Any data bits that do not fall within the enabled channels and the specified range are not compared.

Stop Measurement: Repetitive acquisitions may be halted when the comparison between the current state acquisition and the current compare image is equal or not equal.

Displays: Compare Listing display shows the compare image and bit masks. The Difference Listing display highlights differences between the current state acquisition and the current compare image.

State X-Y Chart Display

This function plots the value of the specified label on the y-axis versus states or another label on the x-axis. Both axes can be scaled by the user.

Markers: The markers are correlated to state listing, state compare, and state waveform displays. They are available as pattern, time, or statistics (with time counting on), and states (with state counting on).

Accumulate: Chart display is not erased between successive acquisitions.

State Waveform Display

This function displays a state acquisition in a waveform format.

States/div: 1 to 104 states.

Delay: -1023 to 1024 states.

Accumulate: The waveform display is not erased between successive acquisitions.

Overlay Mode: Multiple channels can be displayed on one waveform display line. The primary use is to view a summary of bus activity.

Maximum Number of Displayed Waveforms: 24.

Markers: The markers are correlated to state listing, state compare, and X-Y chart displays. The markers can be used for pattern, time, or statistics (with time counting on), and states (with state counting on).

Timing Analysis

Transitional Timing Mode

A sample is stored in acquisition memory only when the data changes. A time tag stored with each sample allows reconstruction of a waveform display. Time covered by a full memory acquisition varies with the number of pattern changes in the data.

Sample Period: 10 ns.

Maximum Time Covered By Data: 5,000 seconds.

Minimum Time Covered by Data: 10.24 μs.

Glitch Capture Mode

Data sample and glitch information is stored every sample period.

Sample Period: 20 ns to 50 ms in a 1-2-5 sequence dependent on seconds/division and delay settings.

Memory Depth: 512 samples/channel.

Time Covered by Data: Sample period X 512.

Waveform Display

Sec/div: 10 ns to 100 s; 0.01% resolution.

Screen Delay: -2500 s to 2500 s. The presence of data is dependent on the number of transitions in data between the trigger and trigger plus delay (transitional timing).

Accumulate: The waveform display is not erased between successive acquisitions.

Hardware Delay: $\pm$(20 ns to 10 ms).

Overlay Mode: Multiple channels can be displayed on one waveform display line. The primary use is to view a summary of bus activity.

Maximum Number Of Displayed Waveforms: 24.

Time Interval Accuracy

Channel to Channel Skew: 4 ns typical.

Sample Period Accuracy: 0.01% of sample period.

Time Interval Accuracy: ± (sample period + channel-to-channel skew + 0.01% of time interval reading).

Trigger Specification

Asynchronous Pattern: Trigger on an asynchronous pattern less than or greater than a specified duration. The pattern is the logical AND of a specified low, high, or don't care for each assigned channel. If the pattern is valid but the duration is invalid, there is a 20 ns reset time before the instrument will look for patterns again.

Greater Than Duration: Minimum duration is 30 ns to 10 ms with 10 ns or 0.01% resolution, whichever is greater. Accuracy is +0 ns to -20 ns. Trigger occurs at pattern + duration.

Less Than Duration: Maximum duration is 40 ns to 10 ms with 10 ns or 0.01% resolution, whichever is greater. Pattern must be valid for at least 20 ns. Accuracy is +20 ns to -0 ns. Trigger occurs at the end of the pattern.

Glitch/Edge Triggering: Trigger on a glitch or edge following a valid duration of an asynchronous pattern while the pattern is still present. Edge can be specified as rising, falling, or either. Less than duration forces glitch and edge triggering off.

Measurement and Display Functions

Autoscale (Timing Analyzer Only)

Autoscale searches for and displays channels with activity on the pods assigned to the timing analyzer.

Acquisition Specifications

Arming: Each analyzer can be armed by the run key, the other analyzer, the oscilloscope, or the external trigger in port.

Trace Mode: Single mode acquires data once per trace specification. Repetitive mode repeats single mode acquisitions until stop is pressed or until the time interval between two specified patterns is less than or greater than a specified value, or within or not within a specified range. There is only one trace mode when two analyzers are on.

Labels

Channels may be grouped together and given up to a six character name. Up to 20 labels in each analyzer may be assigned with up to 32 channels per label. The primary use is for naming groups of channels such as address, data, and control busses.

Indicators

Activity Indicators: Provided in the Configuration, State Format, and Timing Format menus for identifying high, low, or changing states on the inputs.

Markers: Two markers (X and 0) are shown as dashed lines on the display.

Trigger: The trigger is displayed as a vertical dashed line in the timing waveform display and as line 0 in the state listing display.

Marker Functions

Time Interval: The X and 0 markers measure the time interval between one point on a timing waveform and trigger, two points on the same timing waveform, two points on different waveforms, or two states (time tagging on).

Delta States (State Analyzer Only): The X and 0 markers measure the number of tagged states between one state and trigger, or between two states.

Patterns: The X and 0 markers can be used to locate the nth occurrence of a specified pattern before or after trigger, or after the beginning of data. The 0 marker can also find the nth occurrence of a pattern before or after the X marker.

Statistics: The X to 0 marker statistics are calculated for repetitive acquisitions. Patterns must be specified for both markers and statistics are kept only when both patterns can be found in an acquisition. Statistics are minimum X to 0 time, maximum X to 0 time, average X to 0 time, and ratio of valid runs to total runs.

Run/Stop Functions

Run: Starts the acquisition of data in a specified trace mode.

Stop: In single trace mode or the first run of a repetitive acquisition, STOP halts the acquisition and displays the current acquisition data. For subsequent runs in repetitive mode, STOP halts the acquisition of data and does not change current display.

Data Display/Entry

Display Modes: State listing; timing waveforms; interleaved, time-correlated listing of two state analyzers (time tagging on); time-correlated state listing and timing waveform display (state listing in upper half, timing waveform in lower half, and time tagging on).

Timing Waveform: Pattern readout of timing waveforms at X or 0 marker.

Bases: Binary, Octal, Decimal, Hexadecimal, ASCII (display only), and User-defined symbols.

Oscilloscope Specifications

The following specifications are the performance standards or limits against which the oscilloscope in the HP 1652B/1653B is tested.

Vertical

Bandwidth (-3 dB): dc to 100 MHz (single shot).

DC Gain Accuracy: ± 3% of full scale.

DC Offset Accuracy: ± (2 mV + 2% of the channel offset + 2.5% of full scale).

Voltage Measurement Accuracy (DC): (Gain accuracy + ADC resolution + Offset accuracy).

Horizontal

Time Interval Measurement Accuracy: ±(2% X s/div + 0.01% X delta-t + 500 ps).

Trigger

Sensitivity: 10% of full screen.

Oscilloscope Operating Characteristics

The following operating characteristics are not specifications, but are typical operating characteristics for the oscilloscope in the HP 1652B/1653B. These are included as additional information for the user.

Vertical (at BNC)

Transition Time (10% to 90%): ≤ 3.5 ns.

Number of Channels: 2.

Vertical Sensitivity Range: 15 mV/div to 10 V/div (1:1 probe).

Vertical Sensitivity Resolution: Adjustable 2 digit resolution.

Maximum Sample Rate: 400 MSamples/second.

Analog-to-Digital Conversion: 6 bit real-time.

Analog-to-Digital Resolution: ±1.6% of full scale.

Waveform Record Length: 2048 points.

Input R: 1 MΩ ± 1% or 50 Ω ± 1%.

Input C: Approximately 7 pF.

Input Coupling: dc.

Maximum Safe Input Voltage:

1 MΩ input	± 250 V [dc + peak ac (< 10 kHz)]
50 Ω input	± 5 V RMS

DC Offset Range (1:1 Probe):

Vertical Sensitivity	Available Offset
≤50 mV/div	±2.0 V
100 mV/div - 200 mV/div	±10 V
500 mV/div - 1 V/div	±50 V
≥2 V/div	±125 V

±5 V max if input impedance is at 50 Ω.

DC Offset Resolution (1:1 Probe):

Vertical Sensitivity	Resolution
≤50 mV/div	200 μV
100 mV/div - 200 mV/div	1 mV
500 mV/div - 1 V/div	5 mV
≥2 V/div	25 mV or 4 digits of resolution, whichever is greater

Probe Factors: Any integer ratio from 1:1 to 1000:1.

Channel Isolation: 40 dB: dc to 50 MHz.

30 dB: 50 MHz to 100 MHz (with channels at equal sensitivity).

Horizontal

Timebase Range: 5 ns/div to 5 s/div.

Timebase Resolution:

Time/Division Setting	Resolution
t < 10 ns/div	100 ps
t≥10 ns/div	adjustable with 3-digit resolution

Delay Pre-trigger Range:

Time/Division Setting	Delay
5 ns ≤ s/div ≤ 500 ns	5 X (sec/div)
500 ns ≤ s/div ≤ 5 s	2.5 μs

Delay Post-trigger Range:

Time/Division Setting	Available Delay
25 ms - 5 s/div	40 X (s/div)
100 μs - 25 ms/div	1 s
5 ns - 100 μs/div	10,000 X (s/div)

Trigger

Triggering on either input channel, rising or falling edge.

Trigger Level Range: dc Offset ±5 divisions.

Trigger Level Resolution (1:1 Probe):

Trigger Level	Resolution
≤50 mV/div	400 μV
100 mV/ div - 200 mV/div	2 mV
500 mV/div - 1 V/div	10 mV
≥2 V/div	50 mV

Arming: Armed by the Run key, external BNC low input, or by Analyzer 1 or 2.

Trigger Modes

Immediate: Triggers immediately after the arming condition is met.

Edge: Triggers on the rising or falling edge from channel 1 or 2.

Auto-Trigger: Self-triggers if no trigger condition is found within approximately 1 second after arming.

Trigger Out: Arms Analyzer 1 or 2, or triggers the rear panel BNC.

Waveform Display

Display Formats: 1 to 8 oscilloscope waveforms can be displayed.

Display Resolution: 500 points horizontally, 240 points vertically.

Display Modes

Normal: New acquisitions replace old acquisitions on screen.

Accumulate: New acquisitions are added to the screen and displayed with the previous acquisitions until a parameter is changed and a new acquisition is made.

Average: New acquisitions are averaged with older acquisitions and displayed. The maximum number of averages is 256.

Overlay: Channels 1 and 2 can be overlayed in the same display area.

Connect-the-dots: Provides a display of the sample points which are connected by straight lines.

Waveform Reconstruction: A reconstruction filter fills in the missing data points when the timebase is set to ≤100 ns/division or when the timebase setting is reduced to a point where there are fewer than 500 data samples on the screen.

Waveform Math: Display capability of A-B, B-A, and A + B functions is provided.

Mixed Mode: Oscilloscope plus logic analyzer displays on the same screen.

Measurement Aids

Time Markers: Two vertical markers labeled X and O. Voltage levels are displayed for each marker. Time interval measurements can be made between any two events.

Automatic Search: Searches for a specified absolute or percentage voltage level at a positive or negative edge with count adjustable from 1 to 1024.

Auto Search Statistics: Displays mean, maximum, and minimum values for elapsed time from X to O markers for multiple runs. The number of valid runs and total number of runs are also displayed.

Trigger Level Marker: A horizontal trigger level marker is displayed in the Trace/Trigger menu only.

Automatic Measurements: The following pulse parameter measurements can be performed automatically:

Frequency
Period
V p-p
Rise time
Fall time
Preshoot
Overshoot
+ pulse width
- pulse width

Grid: Selectable (On/Off).

Setup Aids

Autoscale: Automatically sets the vertical and horizontal ranges, offset, and trigger levels to display the input signals. This requires an amplitude above 10 mV peak, and a frequency between 50 Hz and 100 MHz.

Preset: Scales the vertical range, offset, and trigger level to predetermined values for displaying ECL or TTL waveforms.

Preset	Vertical Range	Offset	Trigger Level
TTL*	1.5 V	2.5 V	1.60 V
ECL*	500 mV	-1.3 V	-1.3 V

* Values when Probe = 10:1.

Calibration: Offset, attenuation, gain, trigger level, delay, and set to defaults.

Probe Compensation Source: The external BNC supplies a square wave signal of approximately -400 mV to -900 mV at approximately 1.25 kHz.

Interactive Measurements

Acquisition

Oscilloscope, timing, and state can occur simultaneously or in series.

Mixed Displays

Timing channels and oscilloscope channels can be displayed on the same screen. Multiple state machine listings can be displayed with time tags on the same screen. Timing channels can be displayed with a state listing with Time Tags turned on. State listings with time tags, timing channels, and oscilloscope channels can be displayed on the same screen.

Time Correlation

All modules are time correlated with the exception of when the oscilloscope is being armed by the logic analyzer, and when the oscilloscope is not in trigger immediate mode.

Time Interval Accuracy between Modules

Equals the sum of channel to channel time interval accuracies of each machine used for a measurement.

General Characteristics

The following general characteristics for the HP 1652B/1653B include the environment operating conditions, shipping weights, and instrument dimensions.

Operating Environment

Temperature

Instrument: **Operating:** 0°C to +55°C (32°F to +131°F).
Non-operating: -40°C to +70°C (-40°F to +158°F).

Probes and Cables: 0°C to 65°C (+32°F to +149°F).

Disk Media: 10°C to 50°C (+50°F to +149°F).

Humidity

Instrument: **Operating:** Up to 95% relative humidity (non-condensing) at +40°C (+104°F).
Non-operating: Up to 90% relative humidity at +65°C (+149°F).

Disk Media: 8% to 80% relative humidity at +40°C (+104°F).

Altitude

Operating: Up to 4600 meters (15,000 ft).
Non-operating: Up to 15,300 meters (50,000 ft).

Vibration

Operating: Random vibration 5 to 500 Hz, 10 minutes per axis, 0.3 g (rms)
Non-operating: Random vibration 5 to 500 Hz, 10 minutes per axis, 2.41 g (rms); Resonant search 5 to 500 Hz swept sine, 1 Octave/minute sweep rate, 0.75 g (0-peak), 5 minute resonant dwell at 4 resonances per axis.

Power Requirements

115/230 Vac, -25% to +15%, 48 to 66 Hz, 200 W max.

Weight

10.0 kg (22 lbs) net weight; 18.6 kg (41 lbs) shipping weight.

Dimensions

Refer to the outline drawing below.

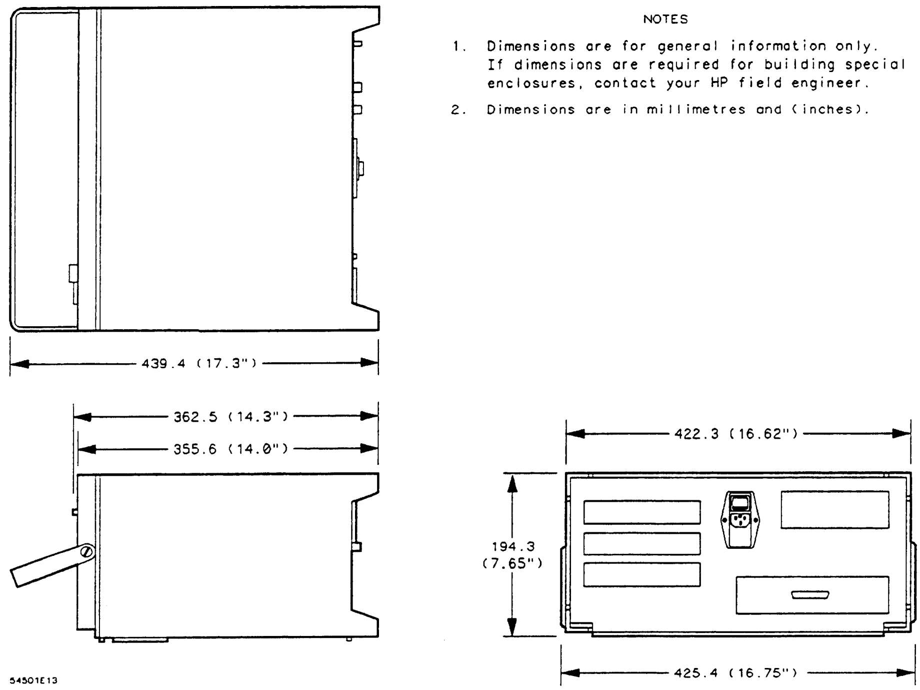


Figure 1-1. HP 1652B/1653B Dimensions

Recommended Test Equipment

Table 1-1 lists the test equipment required to test performance, make adjustments, and troubleshoot the HP 1652B/1653B Logic Analyzer. The table includes the critical specifications of the test equipment and lists each procedure in which the equipment is required. Other equipment may be substituted if it meets or exceeds the critical specifications listed in the table.

Table 1-1. Recommended Test Equipment

Instrument	Critical Specifications	Recommended Model	Use*
Oscilloscope	dual channel dc to 300 MHz	HP 54502A	P
Pulse Generator	5 ns pulse width 20 ns period 1.3 ns risetime double pulse 100 kHz Repetition Rate Overshoot: 5% of Amp.	HP 8161A/020	P
Pulse Generator	Risetime ≤300 ps	Picosecond Pulse Labs 2700C	A
Signal Generator	Frequency: 100 kHz to 300 MHz Output Accuracy: ±1 dB	HP 8656B	P
Power Supply	±10.2 V output current: 0 to 0.4 amperes	HP 6216C	P
DC Power Supply	Range: ±100 mV to ±5 V Accuracy: ±0.1%	HP 6114A	P, A
Digital Voltmeter	5.5 digit resolution Accuracy: ±0.025%	HP 3478A	P, A, T
Power Meter/ Power Sensor	1 to 500 MHz, -70 dBm to 0 dBm, ±1.2%	HP 436A/ HP 8482A	P
Power Splitter	50 ohms type N, outputs differ by <0.15 dB	HP 11667A	P
Adapter	Type N male to BNC female (qty. 2)	HP Part Number 1250-0780	P

* P = Performance Tests   A = Adjustments   T = Troubleshooting

Table 1-1. Recommended Test Equipment (Continued)

Instrument	Critical Specifications	Recommended Model	Use*
Adapter	Type N male to BNC male	HP Part Number 1250-0082	P
Adapter	BNC(female)-to-Dual Banana	HP Part Number 1251-2277	P, A
Adapter	50 ohm feedthrough (Qty 2)	HP 10100C	P
Power Supply Cable	No Substitute	54503-61604	A
BNC Cable	(male-to-male) 48-inch (Qty 2)	HP 10503A	P, A
Cable	Banana (male)-to-Banana (male) (Qty 2)	HP 11000-60001	P, A
Cable	Type N (male) 24-inch	HP 11500B	P
BNC Tee	1M,2F (Qty 2)	HP Part Number 1250-0781	P
Coupler	BNC male-to-male (Qty 2)	HP 1250-0216	P
Resistor	2 Ohms, 25 Watts	HP Part Number 0811-1390	T

* P = Performance Tests   A = Adjustments   T = Troubleshooting

Contents

Section 2:

Installation

Installation

2

Introduction

This section of the manual contains information and instructions necessary for setting up the HP 1652B/1653B Logic Analyzer. This includes inspection procedures, power requirements, hardware connections and configurations, and packaging information.

Safety Considerations

The safety symbols used with Hewlett-Packard instruments are illustrated in the front of this manual. WARNING and CAUTION symbols and instructions should be reviewed before operating the instrument. These warnings and cautions must be followed for your own protection and to avoid damaging the instrument.

Initial Inspection

Inspect the shipping container for damage. If the shipping container or cushioning material is damaged, keep it until the contents of the shipment have been checked for completeness and the instrument has been checked mechanically and electrically. The contents of the shipment are listed under "Accessories Supplied" in Section 1. If the contents are incomplete, if there is mechanical damage or defect, or if the instrument does not operate properly, notify the nearest Hewlett-Packard office. If the shipping container is damaged, or the cushioning materials show signs of stress, notify the carrier as well as the Hewlett-Packard office. Keep the shipping materials for the carrier's inspection. The Hewlett-Packard office will arrange for repair or replacement at HP option without waiting for claim settlement.

Operating Disk Installation

The instrument is shipped with a yellow protective disk in the disk drive. Before applying power to the instrument, remove the protective disk from the disk drive and install the operating system disk. Reinstall the protective disk whenever the instrument is to be transported.

Power Requirements

The HP 1652B/1653B Logic Analyzer requires a power source of either 115 Vac or 230 Vac, -22% to +10%, single phase, 48 to 66 Hz, 200 Watts maximum power.

Caution

BEFORE CONNECTING POWER TO THIS INSTRUMENT, be sure the Line Voltage Select switch on the rear panel of the instrument is set properly and the correct fuse is installed.

Line Voltage Selection

When shipped from the factory, the line voltage selector is set and an appropriate fuse is installed for operating the instrument in the country of destination.

To operate the instrument from a power source other than the one set at the factory:

1. Turn the rear power switch to the OFF position and remove the power cord from the instrument.
2. Remove the fuse module by carefully prying at the top center of the module until you can grasp it and pull it out by hand (see figure 2-1).

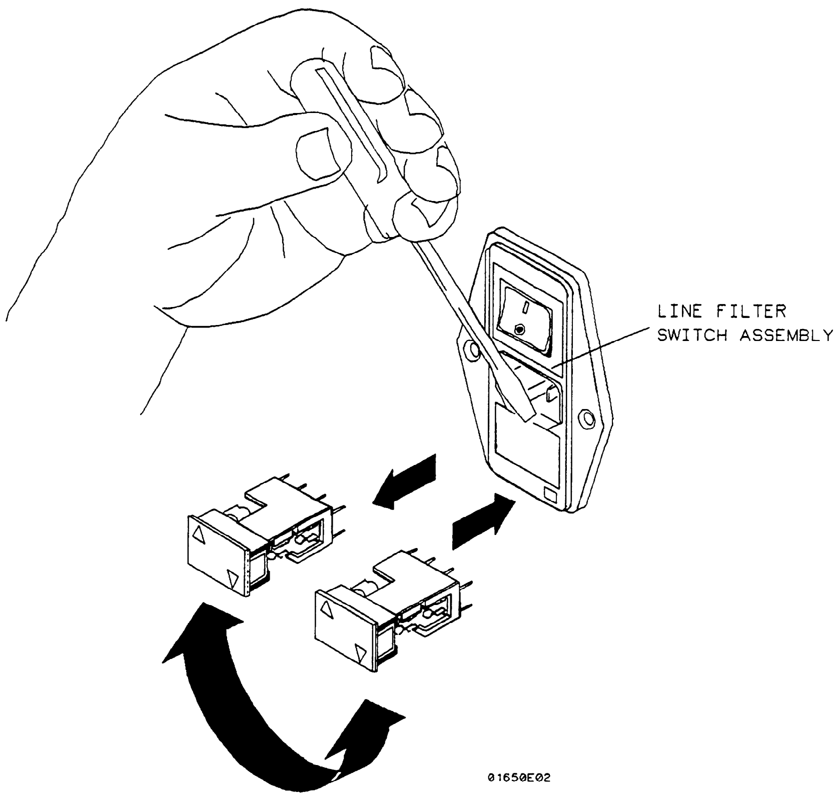


Figure 2-1. Removing the Fuse Module

3. Reinsert the fuse module with the arrow for the appropriate line voltage aligned with the bar on the line filter assembly switch (see figure 2-2).

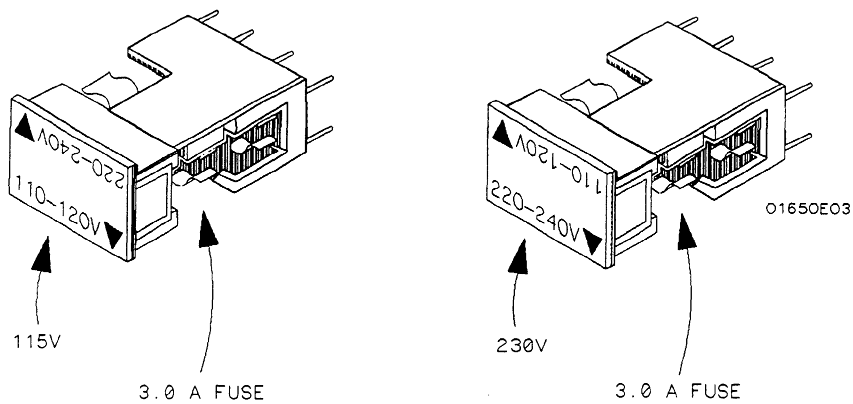


Figure 2-2. Fuse Module Settings

4. Reconnect the power cord, turn the rear power switch to the ON position, and continue normal operation.

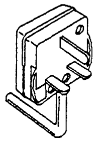
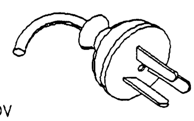
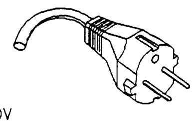
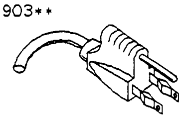
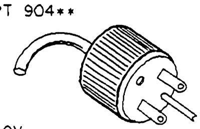
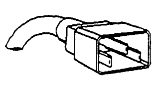
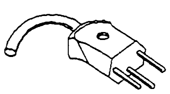
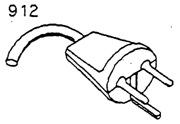
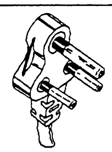
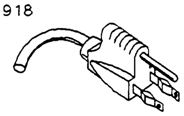
Power Cable

This instrument is equipped with a three-wire power cable. When connected to an appropriate AC power outlet, this cable grounds the instrument cabinet. The type of power cable plug shipped with the instrument depends on the country of destination. See Table 2-1 for the option numbers of available power cables and plug configurations.

Applying Power

When power is applied to the HP 1652B/1653B, a power-up self test is automatically performed. For information on the power-up self test, refer to section 3.

Table 2-1. Power Plug Cord Configurations

PLUG TYPE	CABLE PART NO.	PLUG DESCRIPTION	LENGTH IN/CM	COLOR	COUNTRY
OPT 900  250V	8120-1351 8120-1703	Straight *BS1363A 90°	90/228 90/228	Gray Mint Gray	United Kingdom, Cyprus, Nigeria, Zimbabwe, Singapore
OPT 901  250V	8120-1369 8120-0696	Straight *NZSS198/ASC 90°	79/200 87/221	Gray Mint Gray	Australia New Zealand
OPT 902  250V	8120-1689 8120-1692 8120-2857	Straight *CEE7-Y11 90° Straight (Shielded)	79/200 79/200 79/200	Mint Gray Mint Gray Coco Brown	East and West Europe, Saudi Arabia, So. Africa, India (Unpolarized in many nations)
OPT 903**  125V	8120-1378 8120-1521 8120-1992	Straight *NEMA5-15P 90° Straight (Medical) UL544	90/228 90/228 96/244	Jade Gray Jade Gray Black	United States, Canada, Mexico, Phillipines, Taiwan
OPT 904**  250V	8120-0698	Straight *NEMA6-15P	90/228	Black	United States, Canada
OPT 905  250V	8120-1396 8120-1625	CEE22-V1 (System Cabinet Use) 250V	30/76 96/244	Jade Gray	For interconnecting system components and peripherals. United States and Canada only
OPT 906  250V	8120-2104 8120-2296	Straight *SEV1011 1959-24507 Type 12 90°	79/200 79/200	Mint Gray Mint Gray	Switzerland
OPT 912  220V	8120-2956 8120-2957	Straight *DHCK107 90°	79/200 79/200	Mint Gray Mint Gray	Denmark
OPT 917  250V	8120-4211 8120-4600	Straight SABS164 90°	79/200 79/200	Jade Gray	Republic of South Africa India
OPT 918  100V	8120-4753 8120-4754	Straight Miti 90°	90/230 90/230	Dark Gray	Japan

Rev. 11NOV88 ART00019

*Part number shown for plug is industry identifier for plug only. Number shown for cable is HP part number for complete cable including plug.
**These cords are included in the CSA certification approval of the equipment.

E=Earth Ground
L=Line
N=Neutral

User Interface

The front-panel user interface of the HP 1652B/1653B consists of the front-panel keys, the KNOB, and the display. The interface allows you to configure the logic analyzer, each analyzer (machine) within the logic analyzer, and the oscilloscope in the logic analyzer. It also displays acquired data and measurement results.

Using the front-panel interface is a process of:

- selecting the desired menu with menu keys.
- placing the cursor on the desired field within the menu by rotating the KNOB.
- displaying the field options or current data by pressing the SELECT key.
- selecting the desired option by rotating the KNOB or entering new data by using the KNOB or the keypad.
- Starting and stopping data acquisition by using the RUN and STOP keys.

For additional information on the user interface refer to the *HP 1652B/1653B Front-Panel Operation Reference* manual.

HP-IB Interfacing

The Hewlett-Packard Interface Bus (HP-IB) is Hewlett-Packard's implementation of IEEE Standard 488-1978, "Standard Digital Interface for Programming Instrumentation." HP-IB is a carefully defined interface that simplifies the integration of various instruments and computers into systems. The interface makes it possible to transfer messages between two or more HP-IB compatible devices. HP-IB is a parallel bus of 16 active signal lines divided into three functional groups according to function.

Eight signal lines, called data lines, are in the first functional group. The data lines are used to transmit data in coded messages. These messages are used to program the instrument function, transfer measurement data, and coordinate instrument operation. Input and output of all messages, in bit parallel-byte serial form, are also transferred on the data lines. A 7-bit ASCII code normally represents each piece of data.

Data is transferred by means of an interlocking "Handshake" technique which permits data transfer (asynchronously) at the rate of the slowest active device used in that transfer. The data byte control lines coordinate the handshaking and form the second functional group.

The remaining five general interface management lines (third functional group) are used to manage the devices connected to the HP-IB. This includes activating all connected devices at once, clearing the interface, and other operations.

The connections to the HP-IB connector on the rear panel are shown in figure 2-3.

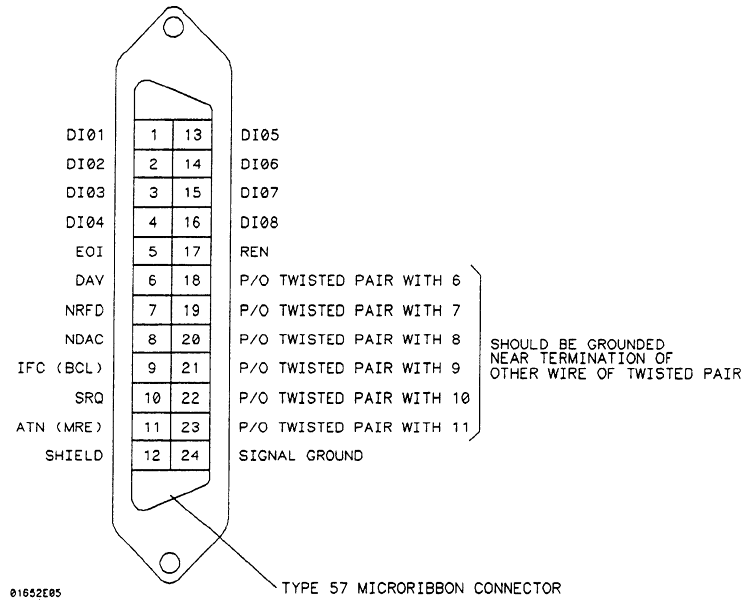


Figure 2-3. HP-IB Interface Connector

HP-IB Address Selection

Each instrument connected to the HP-IB interface bus requires a unique address. The address provides a method for the system controller to select individual instruments on the bus. The address of the HP 1652B/1653B defaults at power up to decimal "07." To change the address of the HP 1652B/1653B proceed as follows:

1. Press the I/O key on the front-panel keypad and the I/O menu will appear on screen.
2. Rotate the KNOB until "I/O Port Configuration" is highlighted.
3. Touch the SELECT key and the External I/O Port Configuration menu will appear on screen.

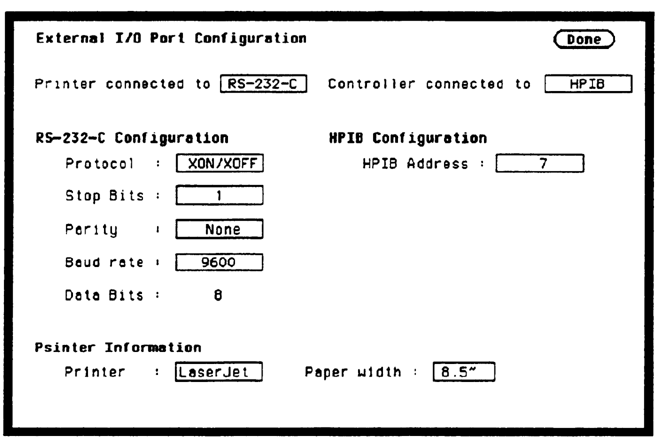


Figure 2-4. External I/O Port Configuration Menu

4. Select the HP-IB Address field with the KNOB and press the SELECT key.
5. When the pop-up field appears on screen, rotate the KNOB to select the desired HP-IB address.

6. Touch the SELECT key to enter the new address.

7. Select the DONE field in the upper-right corner of the menu using the KNOB and SELECT key to exit the External I/O Port Configuration menu.

RS-232-C Interface

The HP 1652B/1653B interfaces with RS-232-C communication lines through a standard 25 pin D connector. The HP 1652B/1653B is compatible with RS-232-C protocol. When a hardwire handshake method is used, the Data Terminal Ready (DTR) line (pin 20 on the Computer/Modem connector) is used to signal whether space is available for more data in the logical I/O buffer. Pin outs of the RS-232-C connectors are listed in table 2-2.

RS-232-C Configuration

At power up, the RS-232-C interface is configured as shown in figure 2-5. To change the RS-232-C configuration:

1. Press the I/O key on the front-panel keypad and the I/O menu will appear on screen.

2. Rotate the KNOB until "I/O Port Configuration" is highlighted.

3. Touch the SELECT key and the External I/O Port Configuration menu will appear on screen.

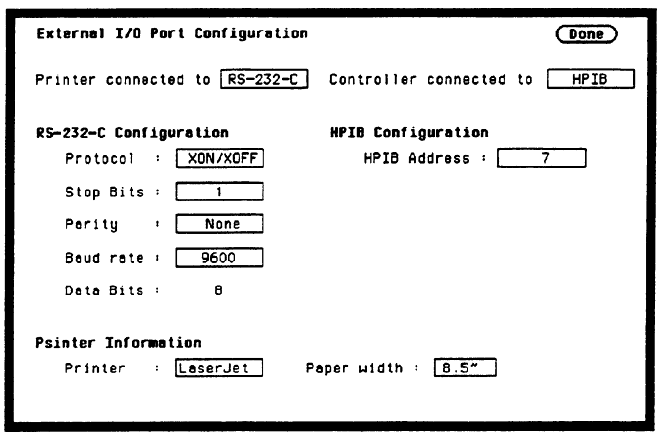


Figure 2-5. External I/O Port Configuration Menu

4. Using the KNOB and SELECT key, configure the RS-232-C interface as desired.

5. Select the DONE field in the upper-right corner of the menu using the KNOB and SELECT key to exit the External I/O Port Configuration menu.

Table 2-2. RS-232-C Signal Definitions

Pin No.	Function	RS-232-C Standard	Signal Direction and Level
1	Protective Ground	AA	Not applicable
2	Transmitted Data (TD)	BA	Data from Mainframe High = Space = "0" = +12 V Low = Mark = "1" = -12 V
3	Received Data (RD)	BB	Data to Mainframe High = Space = "0" = +3 V to +25 V Low = Mark = "1" = -3 V to -25 V
4	Request to Send (RTS)	CA	Signal from Mainframe High = ON = +12 V Low = OFF = -12 V
5	Clear to Send (CTS)	CB	Signal to Mainframe High = ON = +3 V to +12 V Low = OFF = -3 V to -25 V
6	Data Set Ready (DSR)	CC	Signal to Mainframe High = ON = +3 V to +25 V Low = OFF = -3 V to -25 V
7	Signal Ground	AB	Not applicable
8	Data Carrier Detect (DCD)	CF	Signal to Mainframe High = ON = +3 V to +25 V Low = OFF = -3 V to -25 V
20	Data Terminal Ready (DTR)	CD	Signal from Mainframe High = ON = +12 V Low = OFF = -12 V
23	Data Signal Rate Selector	CH/CI	Signal from Mainframe Always High = ON = +12 V

Degaussing the Display

If the instrument has been subjected to strong magnetic fields, the CRT may become magnetized and display data may become distorted. To correct this condition, it may be necessary to degauss the CRT with a conventional external television type degaussing coil.

Operating Environment

The operating environment for the HP 1652B/1653B is described in section 1 of this manual. Note the non-condensing humidity limitation. Condensation within the instrument cabinet can cause poor operation or malfunction. Protection should be provided against temperature extremes which cause condensation within the instrument.

The HP 1652B/1653B will operate at all specifications within the temperature and humidity range given in section 1 of this manual.

Storage and Shipment

The instrument may be stored or shipped in environments within the following limits:

Temperature:	-40° C to +75° C.
Humidity:	Up to 90% at 65° C.
Altitude:	Up to 15,300 meters (50,000 feet).

Tagging for Service

If the instrument is to be shipped to a Hewlett-Packard office for service or repair, attach a tag to the instrument identifying the owner, address of the owner, complete instrument model and serial numbers, and a description of the service required.

Original Packaging

If the original packaging material is unavailable or unserviceable, materials identical to those used in factory packaging are available through Hewlett-Packard offices. If the instrument is to be shipped to a Hewlett-Packard office for service, attach a tag identifying the owner, address of the owner, complete instrument model and serial numbers, and a description of the service required. Mark the container FRAGILE to ensure careful handling. In any correspondence, refer to the instrument by model number and full serial number.

Other Packaging

The following general instructions should be followed for repacking the instrument with commercially available materials.

- Remove the disk from disk drive and install a yellow shipping disk.
- Wrap the instrument in heavy paper or plastic.
- Use a strong shipping container. A double-wall carton made of 350 lb. test material is adequate.
- Use a layer of shock-absorbing material 70 to 100 mm (3 to 4 inches) thick around all sides of the instrument to provide firm cushioning and prevent movement inside the container. Protect the control panel with cardboard.
- Seal the shipping container securely.
- Mark the shipping container FRAGILE to ensure careful handling.
- In any correspondence, refer to the instrument by model number and serial number.

Cleaning Requirements

Use MILD SOAP AND WATER to clean the HP 1652B/1653B cabinet and front panel. Care must be taken to not use a harsh soap which may damage the water-base paint finish of the instrument.

Contents

Section 3:

Performance Tests

3

Performance Tests

Introduction

The procedures in this section test the instrument's electrical performance by using the specifications listed in section 1 as the performance standards. All tests may be performed without access to the interior of the instrument.

Recommended Test Equipment

Equipment required for the performance tests in this section are listed in the Recommended Test Equipment table in section 1. Any equipment that satisfies the critical specification listed in the table may be substituted for the recommended model.

Test Record

The results of the performance tests may be tabulated on the Performance Test Record provided at the end of this section. The Performance Test Record lists the performance tests and provides an area to mark whether the test passed or failed. The results recorded in the table at incoming inspection may be used for later comparisons of the tests during periodic maintenance, troubleshooting, and after repairs or adjustments.

Self Tests

The power-up self test is automatically performed upon applying power to the logic analyzer. Self tests do not require test equipment and may be performed individually to provide a higher level of confidence that the instrument is operating properly. A message that the instrument has failed the test will appear if any problem is encountered during the test. The individual self tests may be performed for functions listed in the self test menu which is invoked via the I/O menu. Since the HP 1652B/1653B self test is located on the Performance Verification disk, you must have the Performance Verification disk installed to run the tests.

Power-up Self Test

The power-up self test is automatically invoked at power-up of the HP 1652B/1653B Logic Analyzer. The revision number of the operating system firmware is given in the upper right of the screen during the power-up self test. As each test is completed, either "passed" or "failed" will be printed in front of the name of the test in this manner:

PERFORMING POWER-UP SELF TESTS

passed ROM test
passed RAM test
passed Interrupt test
passed Display test
passed Keyboard test
passed Acquisition test
passed Threshold test
passed Disk test

LOADING SYSTEM FILE

As indicated by the last message, the HP 1652B/1653B will automatically load from the operating system disk in the disk drive. If the operating system disk is not in the disk drive, the message "SYSTEM DISK NOT FOUND" will be displayed at the bottom of the screen and "NO DISK" will be displayed in front of disk test in place of "passed".

If the message "SYSTEM DISK NOT FOUND" appears on screen, insert the operating system disk into the disk drive, and press any front-panel key.

Selectable Self Tests

Eight self tests may be invoked individually via the Self Test menu. The eight selectable self tests are:

- Analyzer Data Acquisition
- Scope Data Acquisition
- RS-232-C
- BNC
- Keyboard
- RAM
- ROM
- Disk Drive
- Cycle through tests

After entering the I/O Self Tests menu, the required test is selected by moving the cursor to the test and pressing the front panel SELECT key. A pop-up menu appears with a description of the test to be performed. The self test does not begin until the cursor is placed on Execute, Single test, or Repetitive test and the front panel SELECT key is pressed.

After the test has been completed, either "Passed", "Failed", or "Tested" will be displayed on the Self Test menu in front of the test. These self tests are used as troubleshooting aids. For more information, refer to section 6.

Performance Test Interval

Periodic performance verification of the HP 1652B/1653B is required at two year intervals. The instrument's performance should be verified after it has been serviced, or if improper operation is suspected. Calibration should be performed before any performance verification tests. Further checks requiring access to the interior of the instrument are included in the adjustment section, but are not required for the performance verification.

Performance Test Procedures

All performance tests should be performed at the instrument's environmental operating temperature and after a 15 minute warm up. The performance tests for the HP 1652B/1653B are separated into two sections. The first section contains the performance verification tests for the logic analyzer portion of the HP 1652B/1653B and the second section contains the performance verification tests for the oscilloscope portion. Procedures are based on the model or part number for the recommended equipment.

Logic Analyzer Performance Tests

These procedures test the electrical performance of the logic analyzer by using the specifications in section 1 as the performance standards. All tests may be performed without access to the interior of the instrument. Results of performance tests may be tabulated in the Performance Test Record at the end of this section.

Test Connector

The logic analyzer performance tests and adjustments require connecting the pulse generator outputs to probe pod inputs. Figure 3-1 is a test connector that may be built to allow testing of multiple channels (up to eight at one time). The test connector consists of a BNC connector and a length of wire. Connecting more than eight channels to the test connector at a time will induce loading of the circuit and true signal representation will degrade. Test results may not be accurate if more than eight channels are connected to the test connector.

The Hewlett-Packard part number for the BNC connector in figure 3-1 is 1250-1032. An equivalent part may be used in place of the Hewlett-Packard part.

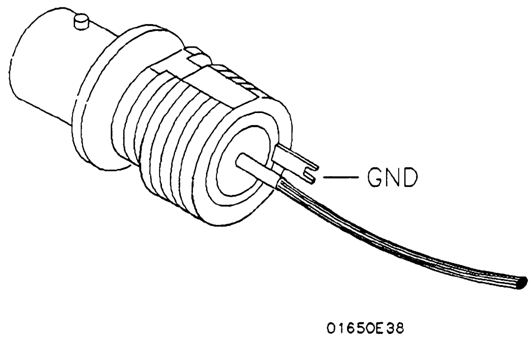


Figure 3-1. Test Connector

For quicker connection without the use of grabbers, a test connector may be built as shown in figure 3-2 to allow testing of multiple channels (up to eight at one time). The test connector consists of a BNC connector and a 2-by-8 Berg connector. The Hewlett-Packard part number for the BNC connector in figure 3-2 is 1250-1032 and the Hewlett-Packard part number for the 2-by-8 Berg connector is 1252-1816. Equivalent parts may be used in place of the Hewlett-Packard parts.

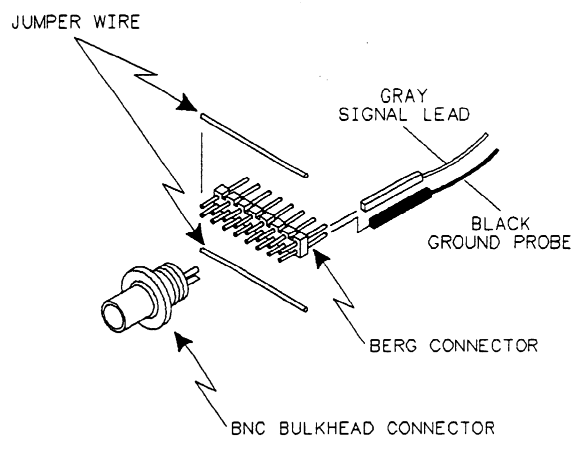


Figure 3-2. Test Connector Using Berg Connector

Clock, Qualifier, and Data Inputs Test 1

Description:

This test verifies the setup and hold times for the falling edge of the HP 1652B L clock specification, and the falling edge of the HP 1653B J and K clock specification. This test also verifies the maximum clock rate with counting mode on.

Specification:

Clock repetition rate: With time or state counting mode on, minimum time between states is 60 ns.

Setup time: Data must be present prior to the clock transition, ≥ 10 ns.

HP 1652B hold time: Data must be present after the falling edge of the L clock transition, 0 ns.

HP 1653B hold time: Data must be present after the falling edge of the J and K clock transition, 0 ns.

Equipment Required:

Pulse Generator	HP 8161A/020
Oscilloscope	HP 54502A
50 Ohm Feedthrough (2)	HP 10100C
Test Connector (2)	see figure 3-1 and 3-2
BNC m-m Coupler (2)	HP 1250-0216
BNC Cable (2)	HP 10503A
BNC Tee m-f-f (2)	HP 1250-0781

Procedure:

1. Connect the HP 1652B/1653B and test equipment as in figure 3-3.

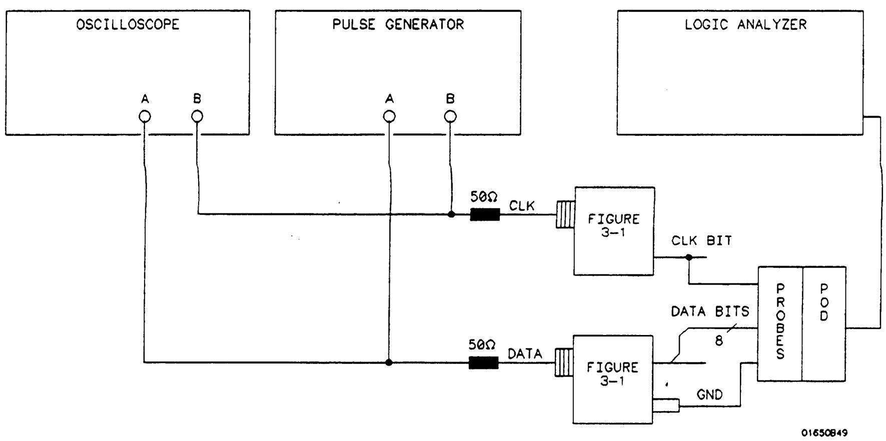


Figure 3-3. Setup for Data Test 1

Note In this setup, only eight channels are tested at one time to minimize loading. The ground lead must be connected to ensure accurate test results.

2. Adjust the pulse generator for the output in figure 3-4.

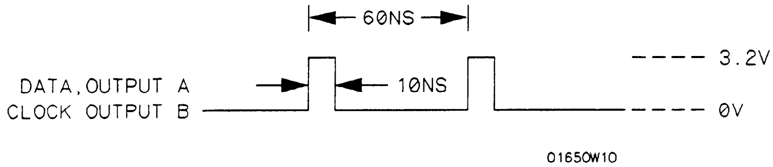


Figure 3-4. Waveform for Data Test 1

Setting for HP 8161A:

Parameter	Output A	Output B
Input Mode	Norm	---
Period (PER)	60 ns	---
Width (WID)	10 ns	10 ns
Leading Edge (LEE)	1 ns	1 ns
Trailing Edge (TRE)	1 ns	1 ns
High Level (HIL)	3.2 V	3.2 V
Low Level (LOL)	0 V	0 V
Delay (DEL)	0 ns	0 ns
Output Mode	ENABLE	ENABLE

3. Assign the pod under test to **Analyzer 1** in the **System Configuration** menu as in figure 3-5. Refer to steps a through c if you are unfamiliar with menus.

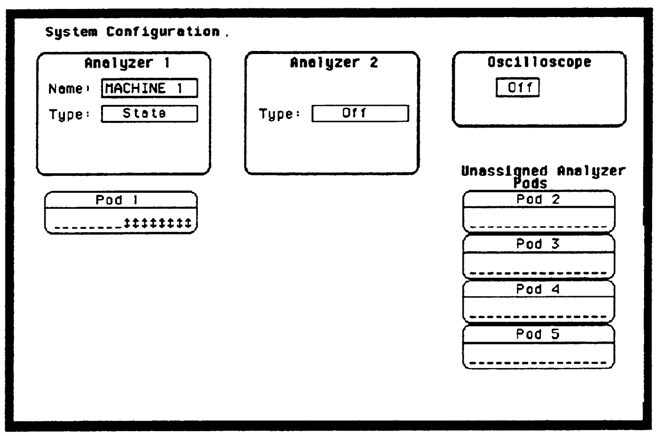


Figure 3-5. System Configuration for Data Test 1

a. Move the cursor to the **Type** field of **Analyzer 1** and press SELECT.

b. Set the analyzer **Type** to **State** using the cursor and the SELECT key.

c. Move the cursor to the **Pod** to be tested and assign it to **Machine 1**.

4. In the **State Format Specification** assign the **Clock Period** to **>60 ns**. Also assign the lower 8 channels of the pod under test to a label as shown in figure 3-6. If you are testing the HP 1652B, assign the falling edge of the L clock for all pods. If you are testing the HP 1653B, assign the falling edge of the J clock for all pods. Refer to steps a through c if you are unfamiliar with the menus.

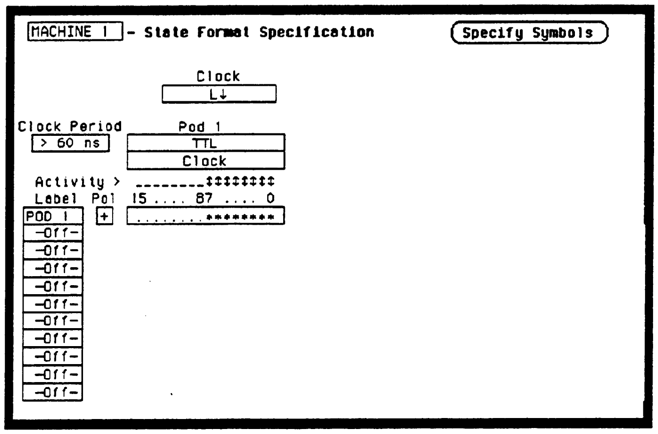


Figure 3-6. Format Specification for Data Test 1

a. Press the front-panel FORMAT/CHAN key.

b. Move the cursor to **Clock** field. Then use the cursor and SELECT key to assign the falling edge of the appropriate clock as in figure 3-7.

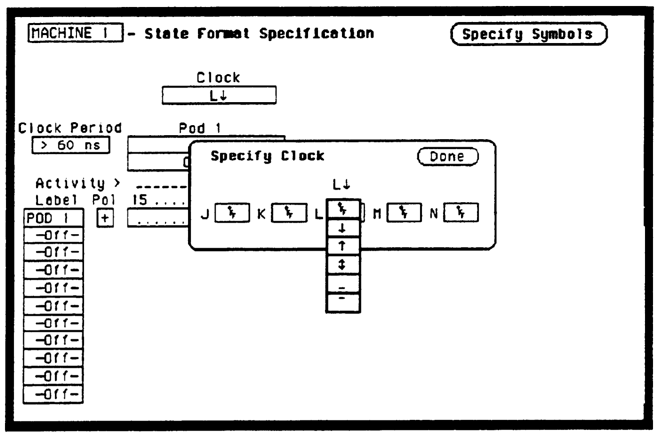


Figure 3-7. Clock Assignment for Data Test 1

c. Move the cursor to the bit assignment field and turn on the appropriate eight bits to be tested (* = on; . = off) as in figure 3-8.

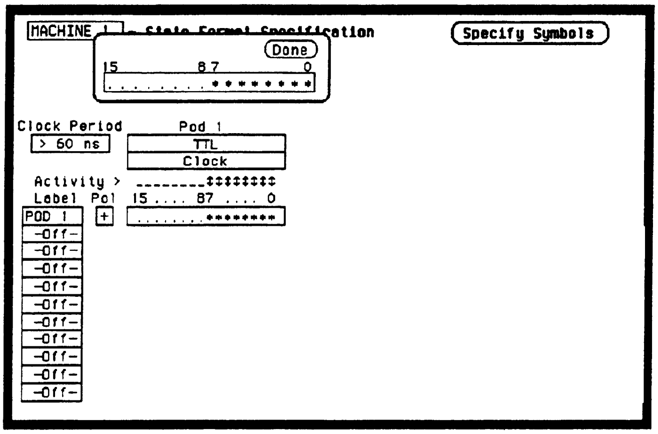


Figure 3-8. Bit Assignment for Data Test 1

5. Set up the **State Trace Specification** without sequencing levels and set **Count States** as in figure 3-9. Refer to steps a through c if you are unfamiliar with menus.

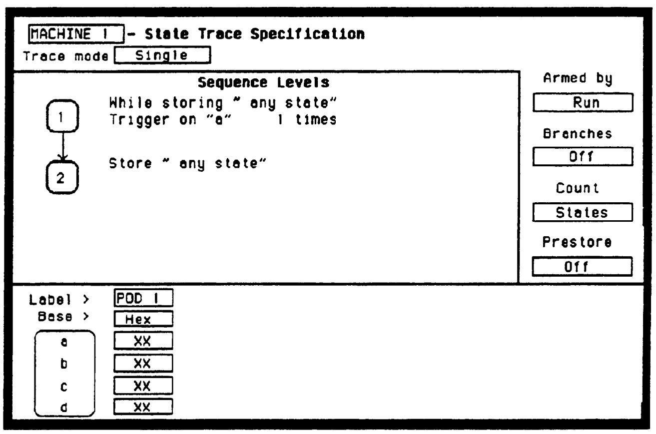


Figure 3-9. Trace Specification for Data Test 1

a. Press the front-panel TRACE/TRIG key.

b. Move the cursor to **Count** and press SELECT.

c. Move the cursor to **States**, press SELECT, and set it to **any state** by pressing SELECT again.

6. Press RUN. The **State Listing** is displayed and shows Fs for the channels under test as in figure 3-10.

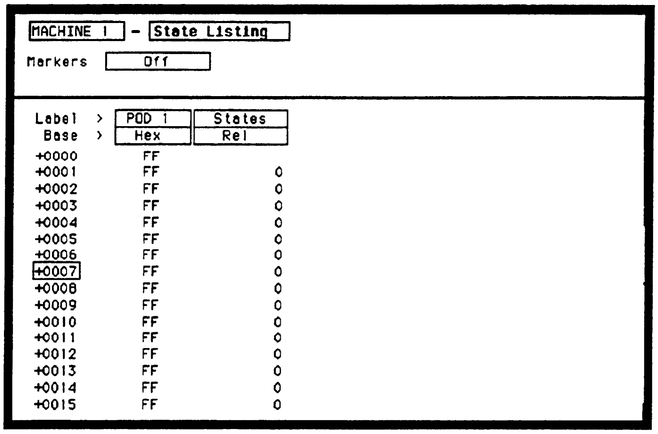


Figure 3-10. State Listing for Data Test 1

Note To ensure a consistent pattern of Fs in the listing, use the front-panel ROLL field and knob to scroll through the **State Listing**.

---

7. If you are testing the HP 1653B, connect the K clock of Pod 2 to the test connector and repeat steps 4 and 6 for the falling edge of the K clock.

8. Remove the probe tip assembly from the logic analyzer probe cable and attach it to the next logic analyzer probe cable to be tested. Take care not to dislodge grabbers from the test connector. If you are testing the HP 1653B, reassign the falling edge of the J clock.

9. Repeat steps 3, 4, 6 and 7 until all of the pods have been tested.

10. Disconnect the lower eight bits (bits 0 through 7) from the test connector and attach the upper eight bits (bits 8 through 15) to the test connector.

11. Repeat steps 3, 4, 6, 7 and 8 until the upper bits of all pods have been tested.

Clock, Qualifier, and Data Inputs Test 2

Description:

This test verifies the setup and hold time specification for the rising edge transition of all of the clocks on the HP 1652B/1653B.

Specification:

Setup time: Data must be present prior to the clock transition, ≥ 10 ns.

Hold time: Data must be present after the rising clock transition, 0 ns.

Equipment Required:

Pulse Generator	HP 8161A/020
Oscilloscope	HP 54502A
50 Ohm Feedthrough (2)	HP 10100C
Test Connector (2)	see figure 3-1 and 3-2
BNC m-m Coupler (2)	HP 1250-0216
BNC Cable (2)	HP 10503A
BNC Tee m-f-f (2)	HP 1250-0781

Procedure:

1. Connect the HP 1652B/1653B and test equipment as in figure 3-11.

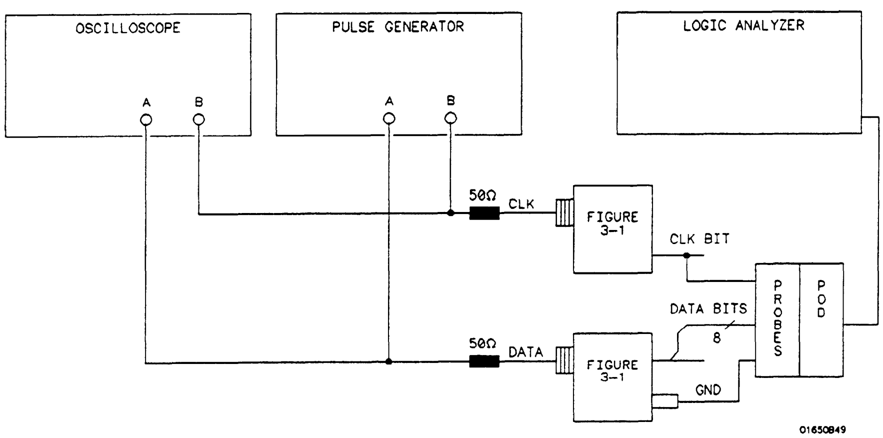


Figure 3-11. Test Setup for Data Test 2

Note

In this setup, only eight channels are tested at one time to minimize loading. The ground lead must be connected to ensure accurate test results.

---

2. Adjust the pulse generator for the output in figure 3-12.

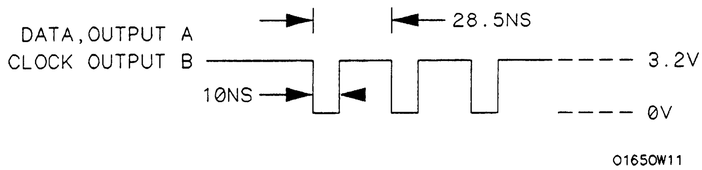


Figure 3-12. Waveform for Data Test 2

Setting for HP 8161A:

Parameter	**Output A**	**Output B**
Input Mode	Norm	---
Period (PER)		
HP 1652B	28.5 ns	---
HP 1653B	40.0 ns	---
Width (WID)	30 ns	30 ns
Leading Edge (LEE)	1 ns	1 ns
Trailing Edge (TRE)	1 ns	1 ns
High Level (HIL)	3.2 V	3.2 V
Low Level (LOL)	0 V	0 V
Delay (DEL)	0 ns	0 ns
Output Mode	ENABLE	ENABLE

3. Assign the pod under test to **Analyzer 1** in the **System Configuration** as in the previous test figure 3-5.

4. In the **State Format Specification** assign the **Clock Period** to **< 60 ns**, and assign the rising edge of the J clock to the **Clock** field. Also, assign the lower 8 channels of the pod under test to a label as in figure 3-13.

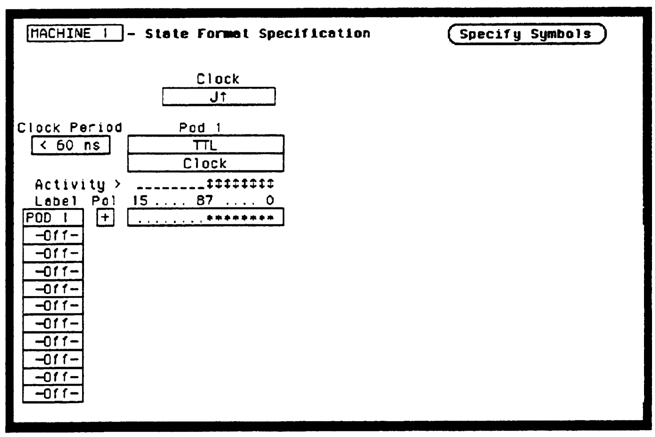


Figure 3-13. Format Specification for Data Test 2

5. Set the **State Trace Specification** without sequencing levels and set **Count** to **Off** as in figure 3-14.

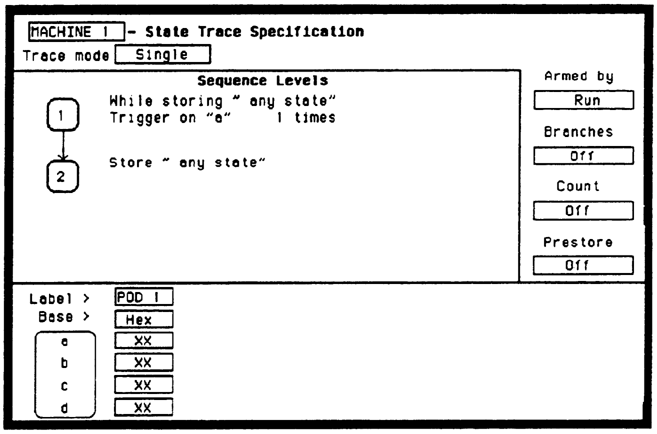


Figure 3-14. Trace Specification for Data Test 2

6. Press RUN. The **State Listing** is displayed and lists all 0s for the channels under test as in figure 3-15.

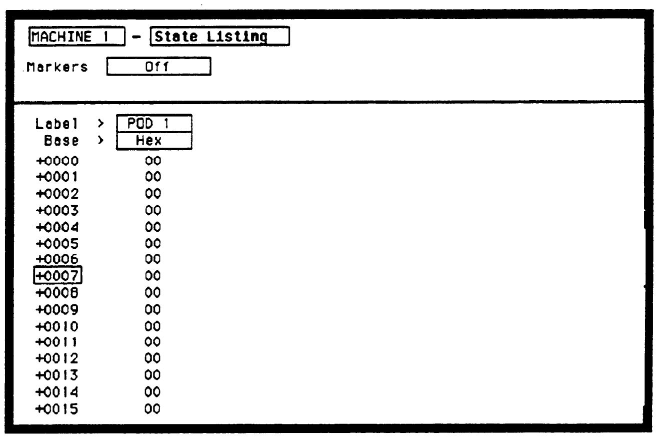


Figure 3-15. State Listing for Data Test 2

Note

To ensure a consistent pattern of 0s in the listing, use the front-panel ROLL field and knob to scroll through the **State Listing**.

7. Connect the next clock to the test connector and repeat steps 4 and 6 for the appropriate clock. Repeat these steps until all clocks have been tested (clocks J, K, L, M and N).

8. Remove the probe tip assembly from the logic analyzer probe cable and attach it to the next logic analyzer probe cable to be tested. Take care not to dislodge grabbers from the test connector.

9. Repeat steps 3, 4, 6, and 7 until the lower bits of all pods (pods 1 through 5) have been tested with all clocks.

10. Disconnect the lower eight bits (bits 0 through 7) from the test connector. Attach the upper eight bits (bits 8 through 15) to the test connector and repeat steps 3, 4, 6, 7, and 8 until the upper bits of all pods (pods 1 through 5) have been tested with all clocks.

Clock, Qualifier, and Data Inputs Test 3 (HP 1652B Only)

Description:

This performance test verifies the hold time specification for the falling clock transitions of the J, K, M, and N clock on the HP 1652B.

Specification:

HP 1652B Hold time: Data must be present after the falling J, K, M, and N clock transition, 1 ns.

Equipment Required:

Pulse Generator	HP 8161A/020
Oscilloscope	HP 54502A
50 Ohm Feedthrough (2)	HP 10100C
Test Connector (2)	see figure 3-1 and 3-2
BNC m-m Coupler (2)	HP 1250-0216
BNC Cable (2)	HP 10503A
BNC Tee m-f-f (2)	HP 1250-0781

Procedure:

1. Connect the HP 1652B and test equipment as in figure 3-16.

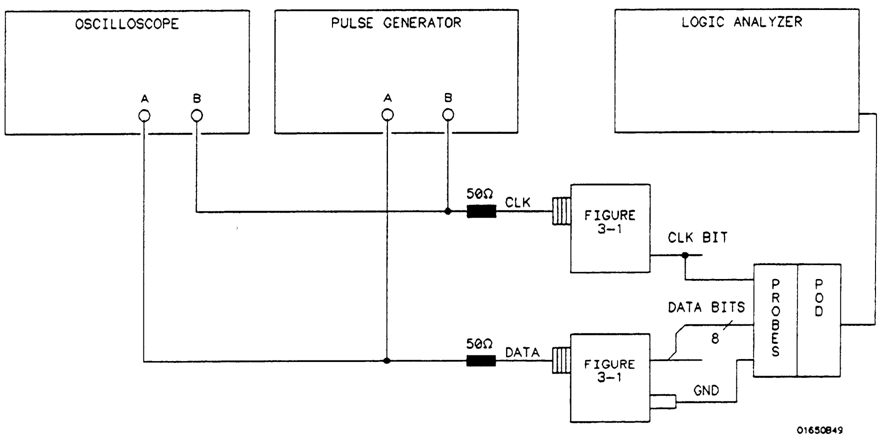


Figure 3-16. Setup for Data Test 3

Note In this setup, only eight channels are tested at one time to minimize loading. The ground lead must be connected to ensure accurate test results.

2. Adjust the pulse generator for the outputs in figure 3-17.

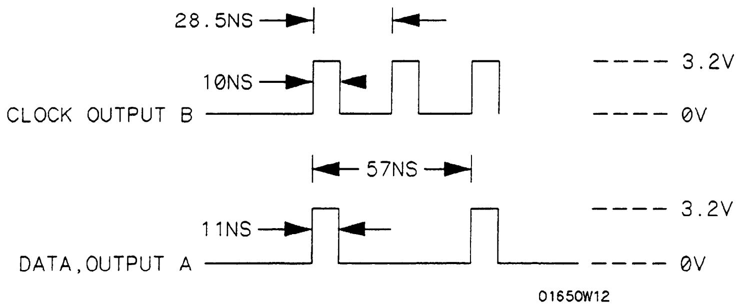


Figure 3-17. Waveform for Data Test 3

Setting for HP 8161A:

Parameter	**Output A**	**Output B**
Input Mode	Norm	---
Period (PER)	57 ns	---
Width (WID)	11 ns	10 ns
Leading Edge (LEE)	1 ns	1 ns
Trailing Edge (TRE)	1 ns	1 ns
High Level (HIL)	3.2 V	3.2 V
Low Level (LOL)	0 V	0 V
Delay (DEL)	0 ns	0 ns
Double Pulse (DBL)	---	28.5 ns
Output Mode	ENABLE	ENABLE

3. Assign the pod under test to **Analyzer 1** in the **System Configuration** as in the previous figure 3-5.

4. In the **State Format Specification** menu assign the **Clock Period** to **< 60 ns**, and the falling edge of J clock to the **Clock** field. Also, assign the lower 8 channels of the pod under test to a label as in the previous test figure 3-13.

5. Set the **State Trace Specification** without sequencing levels and set **Count** to **Off** as in the previous test figure 3-14.

6. Press RUN. The **State Listing** is displayed and lists alternating Fs and 0s as in figure 3-18.

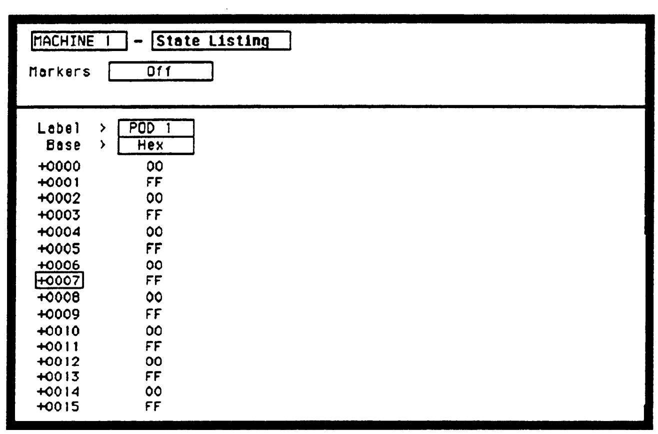


Figure 3-18. State Listing for Data Test 3

Note

To ensure a consistent pattern of alternating Fs and 0s, use the front-panel ROLL field and knob to scroll through the **State Listing**.

7. Connect the next clock to the test connector and repeat steps 4 and 6 for the appropriate clock. Repeat these steps until the J, K, M, and N clocks have been tested.

8. Remove the probe tip assembly from the logic analyzer probe cable and attach it to the next logic analyzer probe cable to be tested. Take care not to dislodge grabbers from the test connector. Repeat steps 3, 4, 6, and 7 until all pods have been tested (pods 1 through 5).

9. Disconnect the lower eight bits (bits 0 through 7) from the test connector. Attach the upper eight bits (bits 8 through 15) to the test connector. Then repeat steps 3, 4, 6, 7, and 8 until the upper bits of all pods have been tested (pods 1 through 5).

Clock, Qualifier, and Data Inputs Test 4

Description:

This test verifies the minimum swing voltages of the input probes and the maximum clock rate of the HP 1652B/1653B when it is in the single phase mode.

Specification:

Minimum swing: 600 mV peak-to-peak.

Clock repetition rate: Single phase is 35 MHz maximum (25 MHz maximum for the HP 1653B).

Clock pulse width: ≥ 10 ns at threshold.

Equipment Required:

Pulse Generator	HP 8161A/020
Oscilloscope	HP 54502A
50 Ohm Feedthrough (2)	HP 10100C
Test Connector (2)	see figure 3-1 and 3-2
BNC m-m Coupler (2)	HP 1250-0216
BNC Cable (2)	HP 10503A
BNC Tee m-f-f (2)	HP 1250-0781

Procedure:

1. Connect the HP 1652B/1653B and test equipment as in figure 3-19. In order to most accurately measure the amplitude of the test signals from the pulse generator, high impedance scope probes should be used to look at the signal levels at the output of the pulse generator.

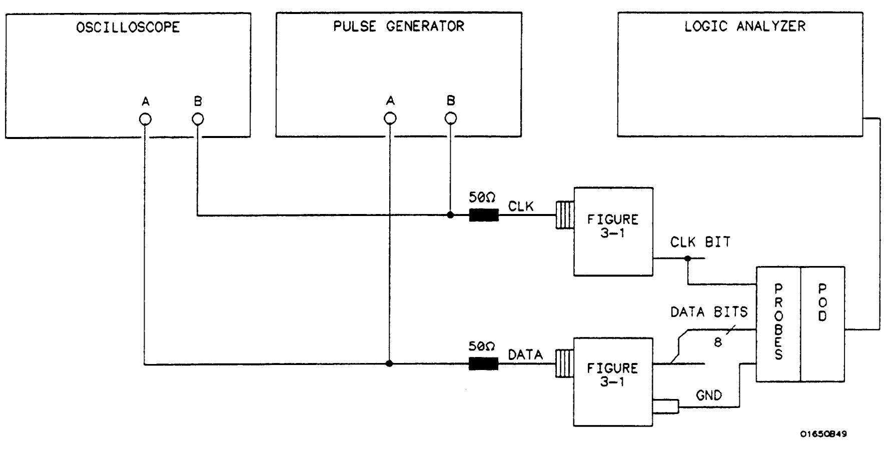


Figure 3-19. Setup for Data Test 4

Note In this setup, only eight channels are tested at one time to minimize loading. The ground lead must be connected to ensure accurate test results. It is recommended that all eight channel grounds be connected.

---

2. Adjust the pulse generator for the output in figure 3-20.

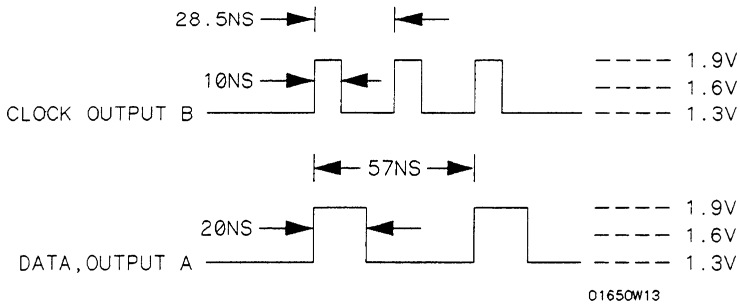


Figure 3-20. Waveform for Data Test 4

Setting for HP 8161A:

Parameter	**Output A**	**Output B**
Input Mode	Norm	---
Period (PER)		
HP 1652B	57 ns	---
HP 1653B	80 ns	---
Width (WID)	20 ns	10 ns
Leading Edge (LEE)	1 ns	1 ns
Trailing Edge (TRE)	1 ns	1 ns
High Level (HIL)	1.9V	1.9V (see Note)
Low Level (LOL)	1.3V	1.3V (see Note)
Delay (DEL)		
HP 1652B	18.5 ns	0 ns
HP 1653B	30 ns	0 ns
Double Pulse (DBL)		
HP 1652B	---	28.5 ns
HP 1653B	---	40 ns
Output Mode	ENABLE	ENABLE

Note The voltage levels of the waveforms must have the correct amplitude at the logic analyzer probe tips. The pulse generator output may have to be increased slightly to compensate for loading by the logic analyzer.

3. Assign the pod under test to **Analyzer 1** in the **System Configuration** as in the previous figure 3-5.

4. In the **State Format Specification** assign the **Clock Period** to **< 60 ns**, and the rising edge of the J clock to the **Clock** field. Also, assign the lower 8 channels of the pod under test to a label as in the previous figure 3-13.

5. Set the **State Trace Specification** without sequencing levels and set the **Count** to **Off** as in the previous figure 3-14.

6. Press RUN. The **State Listing** is displayed and shows alternating Fs and 0s for the channels under test as in figure 3-21.

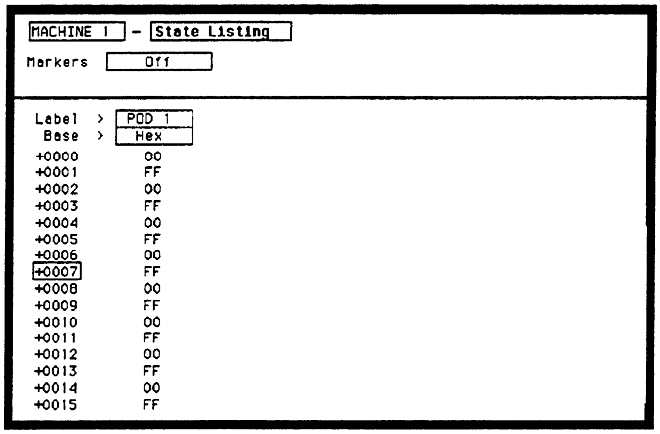


Figure 3-21. State Listing for Data Test 4

Note To ensure a consistent pattern of alternating Fs and 0s, use the front-panel ROLL field and knob to scroll through the **State Listing**.

---

7. Connect the next clock to the test connector and repeat steps 4 and 6 until all clocks have been tested (clocks J, K, L, M, and N).

8. Remove the probe tip assembly from the logic analyzer probe cable and attach it to the next logic analyzer probe cable to be tested. Take care not to dislodge grabbers from the test connector.

9. Repeat steps 3, 4, 6 and 7 until the lower bits of all pods have been tested (pods 1 through 5).

10. Disconnect the lower eight bits (bits 0 through 7) from the test connector and attach the upper eight bits (bits 8 through 15) to the test connector.

11. Repeat steps 3, 4, 6, 7, and 8 until the upper bits of all pods (pods 1 through 5) have been tested with all clocks.

Clock, Qualifier, and Data Inputs Test 5

Description:

This performance test verifies the maximum clock rate for mixed mode clocking during a state operation.

Specification:

Clock repetition rate: Single phase is 35 MHz maximum (25 MHz maximum for the HP 1653B). With time or state counting, minimum time between states is 60 ns (16.7 MHz maximum). Both mixed and demultiplexed clocking use master-slave clock timing. The master clock must follow the slave clock by at least 10 ns and precede the next slave clock by 50 ns.

Equipment Required:

Pulse Generator	HP 8161A/020
Oscilloscope	HP 54502A
50 Ohm Feedthrough (2)	HP 10100C
Test Connector (2)	see figure 3-1 and 3-2
BNC m-m Coupler (2)	HP 1250-0216
BNC Cable (2)	HP 10503A
BNC Tee m-f-f (2)	HP 1250-0781

Procedure:

1. Connect the HP 1652B/1653B and test equipment as in figure 3-22 by connecting channels 0-3 and 8-11 of the pod under test to the test connector. On the slave clock transition, the four bits of the lower byte are transferred to the logic analyzer. On the master clock transition, the four bits of the upper byte are transferred to the logic analyzer.

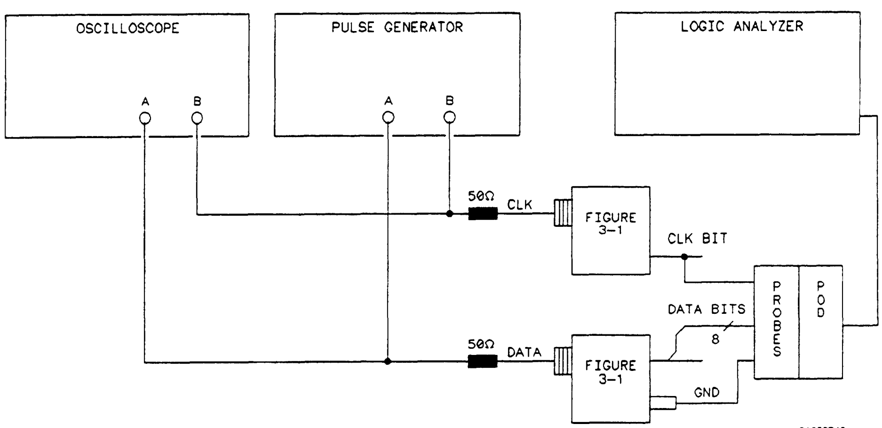


Figure 3-22. Setup for Data Test 5

Note

In this setup, only eight channels are tested at one time to minimize loading. The ground lead must be connected to ensure accurate test results.

---

2. Adjust the pulse generator for the output in figure 3-23.

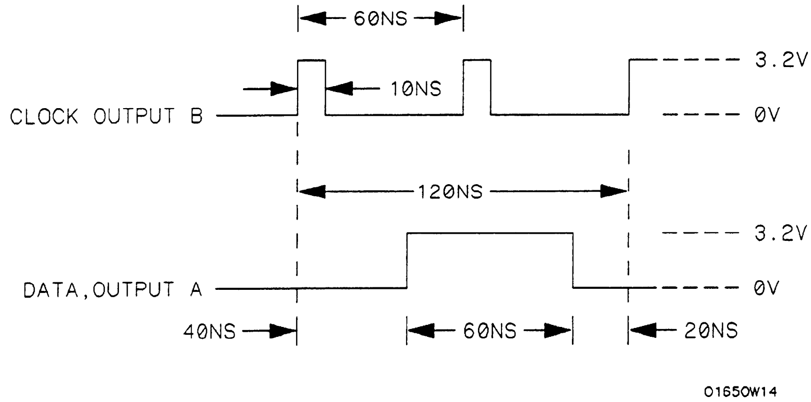


Figure 3-23. Waveform for Data Test 5

Setting for HP 8161A:

Parameter	Output A	Output B
Input Mode	Norm	---
Period (PER)	120 ns	---
Width (WID)	60 ns	10 ns
Leading Edge (LEE)	1 ns	1 ns
Trailing Edge (TRE)	1 ns	1 ns
High Level HIL)	3.2 V	3.2 V
Low Level (LOL)	0 V	0 V
Delay (DEL)	40 ns	0 ns
Double Pulse (DBL)	---	60 ns
Output Mode	ENABLE	ENABLE

3. Assign the pod under test to **Analyzer 1** in the **System Configuration** as in the previous figure 3-5.

4. Set up the **State Format Specification** as in figure 3-24. Assign the falling J clock to the **Master Clock** and the rising J clock to the **Slave Clock**. Refer to steps a through d after figure 3-24 if you are unfamiliar with menus.

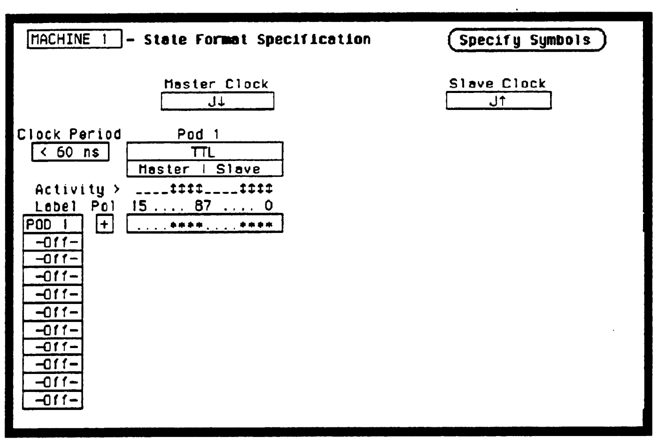


Figure 3-24. Format Specification for Data Test 5

a. Move the cursor to the **Pod Clock** field and press SELECT. Then assign **Mixed Clocks**.

b. Move the cursor to the clock fields and assign the falling transition of the J clock to the **Master Clock** and the rising transition of the J clock to the **Slave Clock.**

c. Move the cursor to the appropriate bit assignment field and turn on channels 0-3 and 8-11 of the pod under test.

d. Move the cursor to the **Clock Period** and set it to **< 60 ns**.

5. Set the **State Trace Specification** without sequencing levels and **Count Off** as in the previous figure 3-14.

6. Press RUN. The **State Listing** displays alternating Fs and 0s for the channels under test as in figure 3-25.

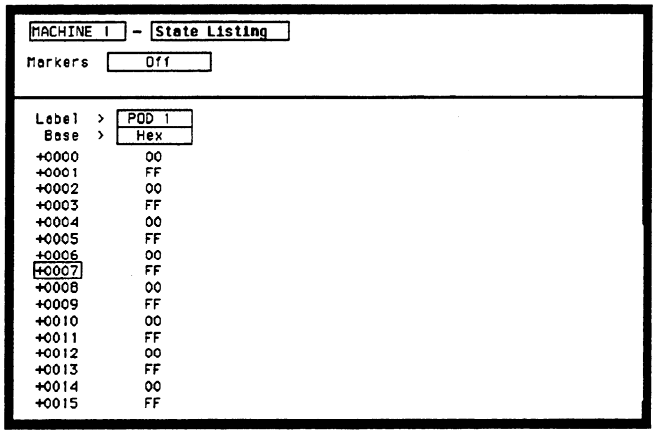


Figure 3-25. State Listing for Data Test 5

Note To ensure a consistent pattern of alternating Fs and 0s, use the front-panel ROLL field and knob to scroll through the **State Listing**.

7. Connect the next clock to the test connector and repeat steps 4 and 6. Repeat these steps until all clocks have been tested (clocks J, K, L, M, and N).

8. Remove the probe tip assembly from the logic analyzer probe cable and attach it to the next logic analyzer probe cable to be tested. Take care not to dislodge grabbers from the test connector.

9. Repeat steps 3, 4, 6, and 7 until channels 0 - 3 and 8 - 11 of all pods have been tested (pods 1 through 5). Start with the falling edge of the J clock as the Master clock and rising edge of the J clock as the Slave clock.

10. Disconnect bits 0-3 and bits 8-11 from the test connector and attach bits 4-7 and bits 12-15 to the test connector. Repeat steps 3, 4, 6, 7, and 8 until all pods have been tested (pods 1 through 5) with all clocks.

Clock, Qualifier, and Data Inputs Test 6

Description:

This performance test verifies the maximum clock rate for demultiplexed clocking during a state operation.

Specification:

Clock repetition rate: Single phase 35 MHz maximum (25 MHz maximum for the HP 1653B). With time or state counting, minimum time between states is 60 ns (16.7 MHz maximum). Both mixed and demultiplexed clocking use master-slave clock timing; the master clock must follow the slave clock by at least 10 ns and precede the next slave clock by 50 ns.

Equipment Required:

Pulse Generator	HP 8161A/020
Oscilloscope	HP 54502A
50 Ohm Feedthrough (2)	HP 10100C
Test Connector (2)	see figure 3-1 and 3-2
BNC m-m Coupler (2)	HP 1250-0216
BNC Cable (2)	HP 10503A
BNC Tee m-f-f (2)	HP 1250-0781

Procedure:

1. Connect the HP 1652B/1653B and test equipment as in figure 3-26 by connecting channels 0-7 of the pod under test to the test connector. During demultiplexed clocking only the lower eight bits of each pod are used.

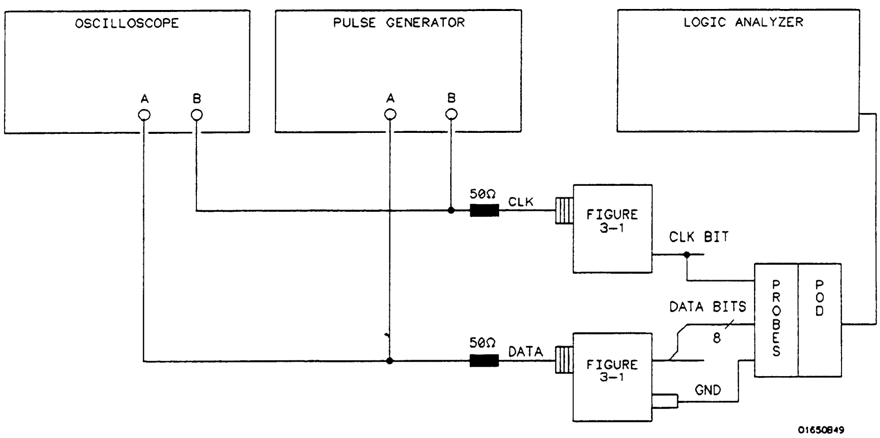


Figure 3-26. Setup for Data Test 6

2. Adjust the pulse generator for the output in figure 3-27.

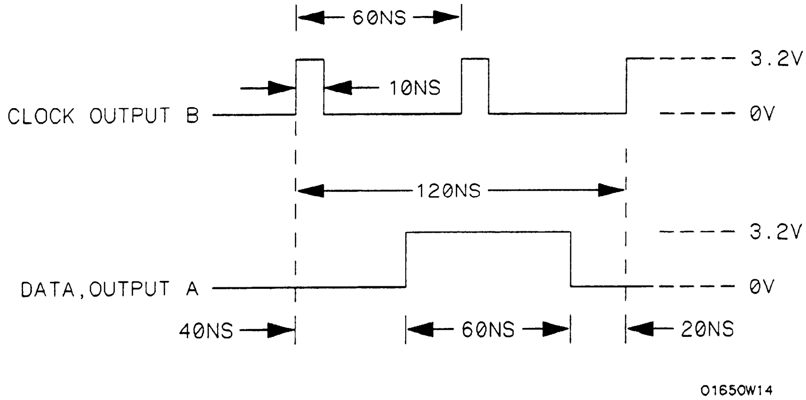


Figure 3-27. Waveform for Data Test 6

Setting for HP 8161A:

Parameter	**Output A**	**Output B**
Input Mode	Norm	---
Period (PER)	120 ns	---
Width (WID)	60 ns	10 ns
Leading Edge (LEE)	1 ns	1 ns
Trailing Edge (TRE)	1 ns	1 ns
High Level (HIL)	3.2 V	3.2 V
Low Level (LOL)	0 V	0 V
Delay (DEL)	40 ns	ns
Double Pulse (DBL)	---	60 ns
Output Mode	ENABLE	ENABLE

3. Assign the pod under test to **Analyzer 1** in the **System Configuration** as in the previous figure 3-5.

4. Set up the **State Format Specification** as in figure 3-28. Assign the falling J clock as the **Master Clock** and the rising J clock as the **Slave Clock.** Refer to steps a through d if you are unfamiliar with the menus.

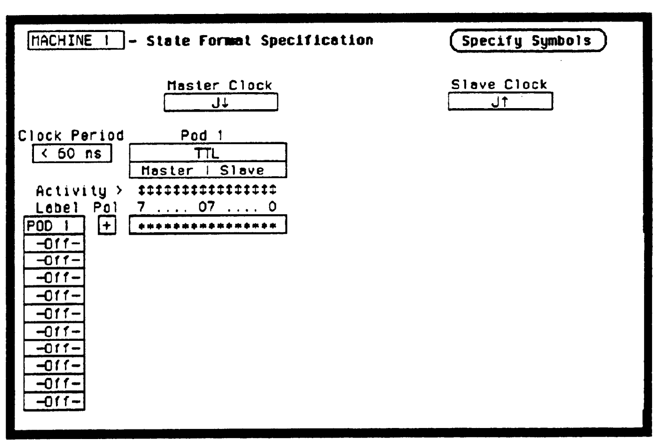


Figure 3-28. Format Specification for Data Test 6

a. Move the cursor to the **Pod Clock** field, press SELECT, and assign **Demultiplex.**

b. Move the cursor to the clock fields and assign the falling clock transition of the J clock to the **Master Clock** and the rising J clock transition to the **Slave Clock.**

c. Move the cursor to the appropriate bit field and assign ALL channels to the pod under test (only bits 0 through 7 are available for assignment).

d. Move the cursor to the **Clock Period** and set it to **<60 ns**.

5. Set the **State Trace Specification** without sequencing levels and set **Count Off** as in the previous figure 3-14.

6. Press RUN. The **State Listing** shows alternating Fs and 0s for the pod under test as in figure 3-29.

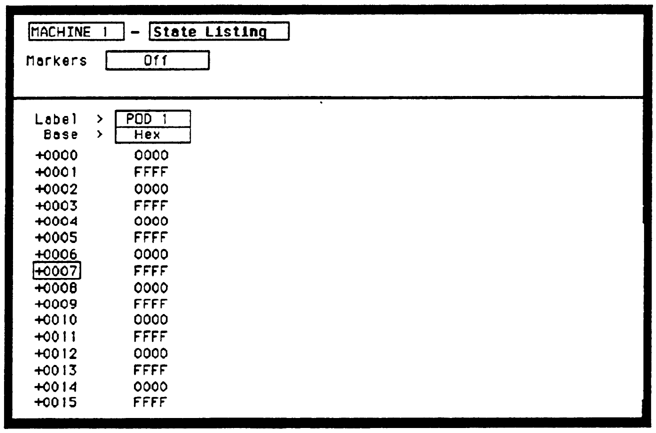


Figure 3-29. State Listing for Data Test 6

Note To ensure a consistent pattern of alternating Fs and 0s, use the front-panel ROLL field and knob to scroll through the **State Listing.**

---

7. Connect the next clock to the test connector and repeat steps 4 and 6.

8. Repeat steps 4, 6, and 7 until all clocks have been tested (clocks J, K, L, M and N).

9. Remove the probe tip assembly from the logic analyzer probe cable and attach it to the next logic analyzer probe cable to be tested. Take care not to dislodge grabbers from the test connector.

10. Repeat steps 3, 4, 6, 7, 8, and 9 until all pods have been tested (pods 1 through 5). Start with the falling edge of the J clock as the **Master Clock** and rising edge of the J clock as the **Slave Clock.**

Glitch Test

Description:

This performance test verifies the glitch detection specification of the HP 1652B/1653B.

Specification:

Minimum detectable glitch: 5 ns wide at the threshold.

Equipment Required:

Pulse Generator	HP 8161A/020
Oscilloscope	HP 54502A
50 Ohm Feedthrough	HP 10100C
Test Connector	see figure 3-1 and 3-2
BNC m-m Coupler	HP 1250-0216
BNC Cable	HP 10503A
BNC Tee m-f-f	HP 1250-0781

Procedure:

1. Connect the test equipment as in figure 3-30. The clock inputs are not used for the glitch test since glitch detection is part of timing analysis. Use the oscilloscope to make sure pulses are 5 ns wide at the threshold (1.6 V).

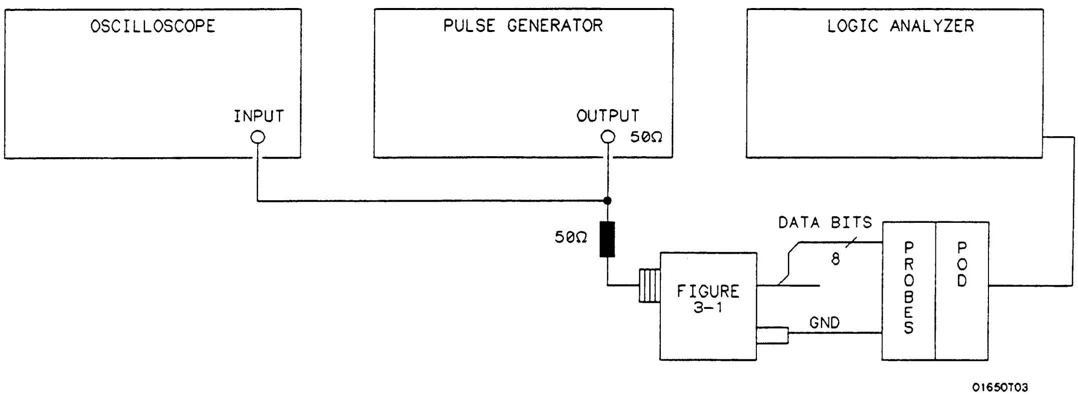


Figure 3-30. Setup for Glitch Test

Note

In this setup, only eight channels are tested at one time to minimize loading. The ground lead must be connected to ensure accurate test results.

2. Adjust the pulse generator for the output in figure 3-31.

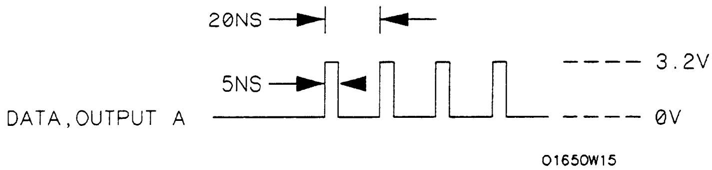


Figure 3-31. Waveform for Glitch Test

Setting for HP 8161A:

Parameter	**Output A**	**Output B**
Input Mode	Norm	---
Period (PER)	20 ns	---
Width (WID)	5 ns	---
Leading Edge (LEE)	1 ns	---
Trailing Edge (TRE)	1 ns	---
High Level (HIL)	3.2 V	---
Low Level (LOL)	0 V	---
Delay (DEL)	0 ns	---
Output Mode	ENABLE	---

3. Assign the pod under test to **Analyzer 1** in the **System Configuration** as in figure 3-32. Refer to steps a through c if you are unfamiliar with menus.

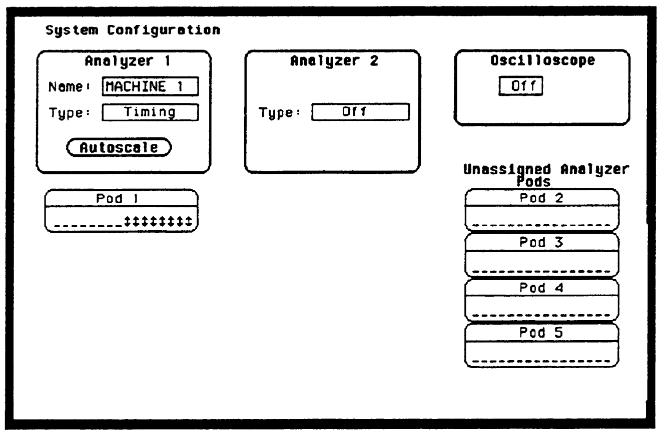


Figure 3-32. System Configuration for Glitch Test

a. Move the cursor to the **Type** field of **Analyzer 1** and press SELECT.

b. Set the analyzer **Type** to **Timing** using the cursor and SELECT key.

c. Move the cursor to the pod to be tested and assign it to **Machine 1** (Analyzer 1).

4. In **State Format Specification** assign the lower eight bits of the pod under test to a label as shown in figure 3-33. Make sure the appropriate eight bits in the bit assignment field are turned on.

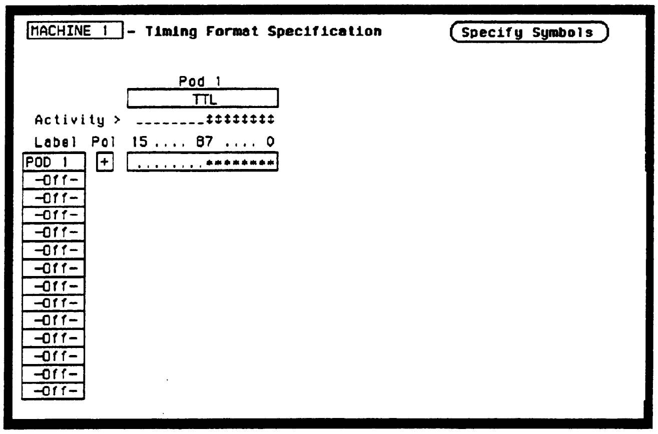


Figure 3-33. Glitch Test Timing Format Specification

5. Set **Timing Trace Specification** as in figure 3-34. Follow steps a through d if you are unfamiliar with menus.

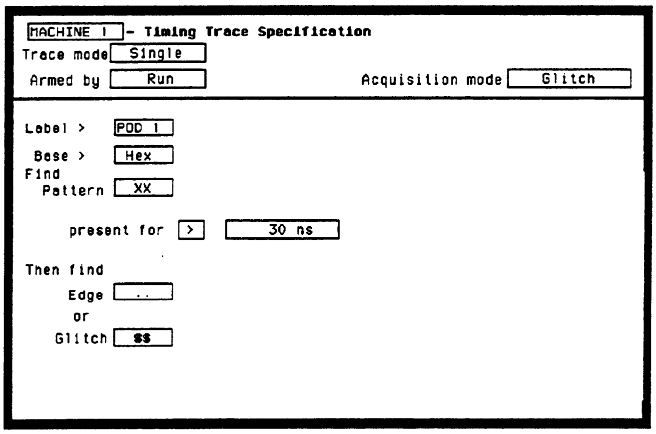


Figure 3-34. Glitch Test Timing Trace Specification

a. Move the cursor to the **Acquisition mode** field and select the **Glitch** mode.

b. Move the cursor to the **Find Pattern** field and press SELECT. Assign all Don't Cares (all Xs) and press SELECT.

c. Set the **Present for** field to **>30 ns.**

d. Set **Then find Glitch** "on" for all channels (* = on; . = off).

6. Press RUN. The timing analyzer acquires data and shows glitches for channels under test as in figure 3-35. Select the **Delay** field and rotate the knob to assure consistent glitch detection.

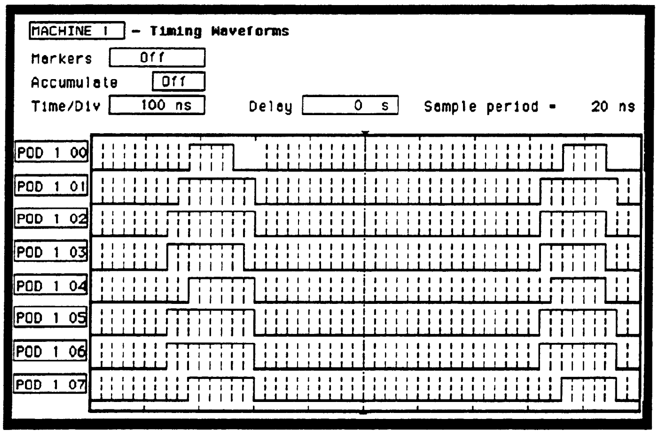


Figure 3-35. Glitch Test Timing Waveforms

Note

If the sample clock and data synchronize, glitches may be displayed on the timing screen as valid data transitions.

---

7. Remove the probe tip assembly from the logic analyzer probe cable and attach it to the next logic analyzer probe cable to be tested. Take care not to dislodge grabbers from the test connector.

8. Repeat steps 3, 4, and 6 until all pods have been tested (pods 1 through 5). Make sure to assign the correct pod to be tested in the **System Configuration** menu.

9. Disconnect the lower eight bits (bits 0 through 7) from the test connector and attach the upper eight bits (bits 8 through 15) to the test connector.

10. Repeat steps 3, 4, 6, and 7 until the upper bits of all pods have been tested (pods 1 through 5).

Threshold Accuracy Test

Description:

This performance test verifies the threshold accuracy within the ranges stated in the specification.

Specification:

Threshold accuracy: 150 mV accuracy over the range -2.0 to +2.0 volts; 300 mV accuracy over the ranges -9.9 to -2.1 volts and +2.1 to +9.9 volts.

Equipment Required:

Power Supply	HP 6216C
Test Connector	see figure 3-1 and 3-2
BNC (f)-to-Dual Banana (m) Adapter	HP 1251-2277
BNC Cable	HP 10503A

Procedure:

1. Connect the test equipment as in figure 3-36.

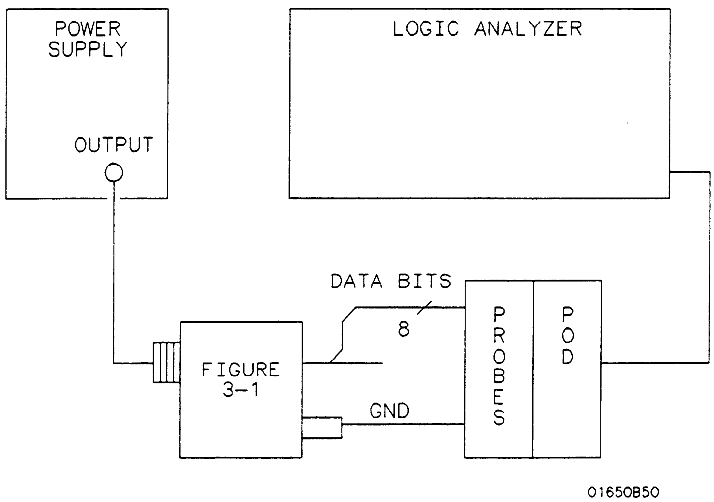


Figure 3-36. Threshold Accuracy Test Setup

Note In this setup, only eight channels are tested at one time to minimize loading. The ground lead must be grounded to ensure accurate test results.

2. Assign the pod under test to **Analyzer 1** in the **System Configuration** as in the previous figure 3-32.

3. Configure the **Timing Format Specification** for a **User Defined** pod threshold of **0.0 V** for the pod under test and assign the lower eight bits in the bit assignment field as in figure 3-37. Refer to steps a through c if you are unfamiliar with menus.

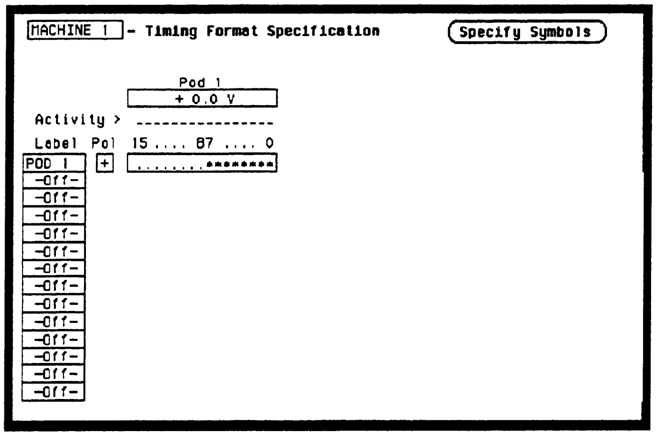


Figure 3-37. Threshold Accuracy Format Specification

a. Move the cursor to the Pod Threshold field and press SELECT.

b. Move the cursor to **User-defined** and press SELECT. Then enter the appropriate voltage threshold.

c. Move the cursor to the bit assignment field and turn on the appropriate eight bits to be tested (* = on; . = off).

4. Set the **Timing Trace Specification** as in figure 3-38. Follow steps a through d if you are unfamiliar with menus.

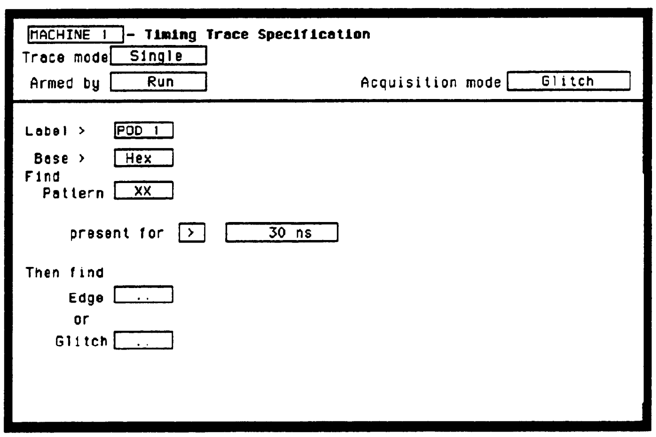


Figure 3-38. Threshold Accuracy Trace Specification

a. Move the cursor to the **Acquisition mode** and select the **Glitch** mode.

b. Move the cursor to **Find Pattern** and press SELECT. Then assign Don't Cares (all X s) and press SELECT.

c. Set **present for** to **>30 ns**.

d. Set **Then find Glitch** to all Don't Cares (all periods ".").

5. Adjust the power supply output for +150 mV.

6. Press RUN. Data displayed on the **Timing Waveforms** display is all high for the pod and channels under test as in figure 3-39.

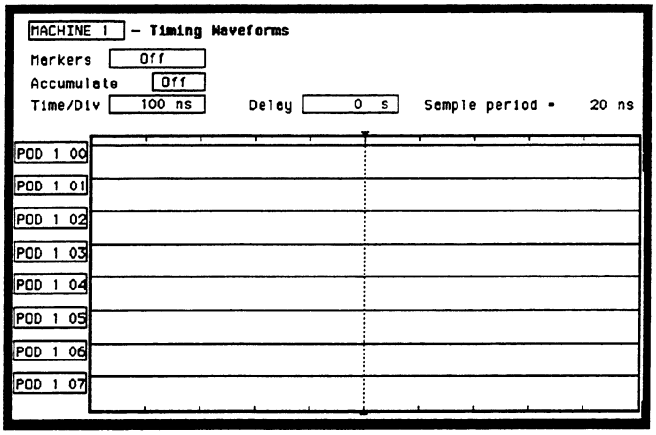


Figure 3-39. Threshold Accuracy Timing Waveforms 1

7. Adjust the power supply output for -150 mV.

8. Press RUN. Data displayed on the **Timing Waveforms** display is all low for the channels under test as in figure 3-40.

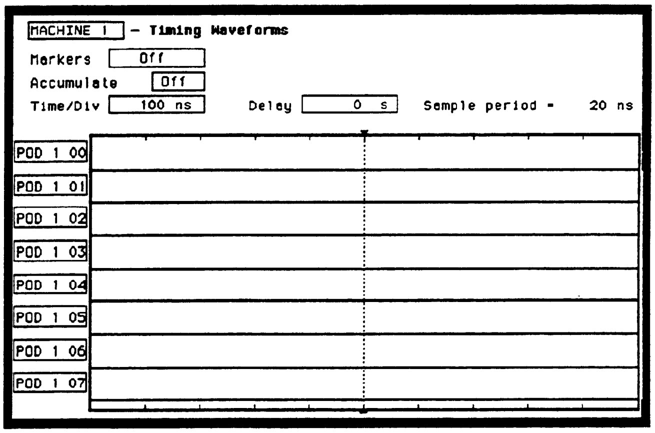


Figure 3-40. Threshold Accuracy Timing Waveforms 2

9. Return to the **Timing Format Specification** and change the **User Defined** pod threshold to **+9.9 V**.

10. Adjust the power supply output for +10.2 V.

11. Press RUN. Data displayed on **Timing Waveforms** display is all high for the pod and channels under test as in the previous figure 3-39.

12. Adjust the power supply output for +9.6 V.

13. Press RUN. Data displayed on **Timing Waveforms** display is all low for the pod and channels under test as in the previous figure 3-40.

14. Return to the **Timing Format Specification** and change the **User Defined** pod threshold to **-9.9** V.

15. Adjust the power supply output for -9.6 V.

16. Press RUN. Data displayed on **Timing Waveforms** display is all high for the pod and channels under test as in the previous figure 3-39.

17. Adjust the power supply output for -10.2 V.

18. Press RUN. Data displayed on **Timing Waveforms** display is all low for the pod and channels under test as in the previous figure 3-40.

19. Remove the probe tip assembly from the logic analyzer probe cable and attach it to the next logic analyzer probe cable to be tested. Take care not to dislodge grabbers from the test connector.

20. Repeat steps 2 through 18 until all pods have been tested (pods 1 through 5).

21. Disconnect the lower eight bits (bits 0 through 7) from test connector and attach the upper eight bits (bits 8 through 15) to the test connector.

22. Repeat steps 2 through 19 until the upper bits of all pods have been tested (pods 1 through 5).

Oscilloscope Performance Tests

These procedures test the HP 1652B/1653B oscilloscope module's electrical performance using the specifications listed in section 1 as the performance standards. All tests can be performed without access to the interior of the instrument. Results of performance tests may be tabulated in the Performance Test Record at the end of this section.

Input Resistance Test

Description:

This test checks the input resistance of the vertical inputs. A four-wire measurement is used for accuracy at 50 Ω. Input resistance is not a specification, but this test is provided for the convenience of the user.

Note

The Input Resistance Test is optional. The input resistance is not specified in the instrument performance specifications. The values given are typical. Results are not recorded in the test record.

Characteristic:

1 MΩ ±1% and 50 Ω ±1%

Equipment Required:

Digital Multimeter	HP 3478A
BNC Cable (2)	HP 10503A
BNC Tee m-f-f	HP 1250-0781
BNC (f)-to-Banana (m) Adapter (2)	HP 1251-2277

Procedure:

1. Set up the multimeter to make a four-wire resistance measurement.
2. Use the BNC-to-banana adapters to connect one end of each BNC cable to the four-wire resistance connections on the multimeter. Then connect the free ends of the cables to the BNC tee as in figure 3-41.

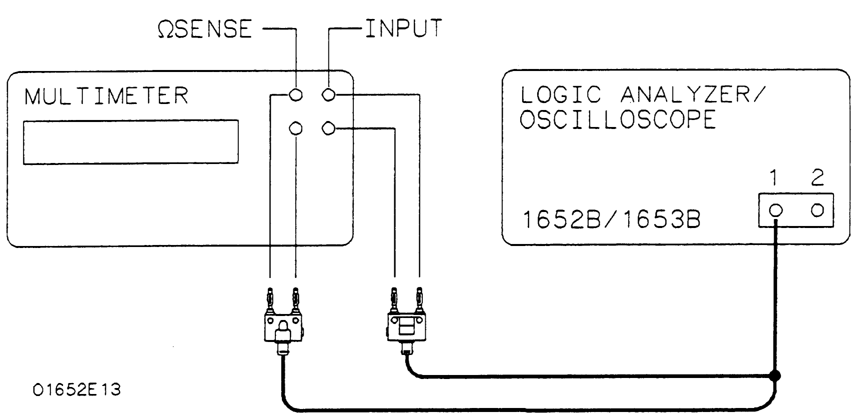


Figure 3-41. Setup for Input Resistance Test

3. Connect the male end of the BNC tee to the channel 1 input of the HP 1652B/1653B oscilloscope.

4. In the **System Configuration** menu, turn both State/Timing Analyzers off and turn the oscilloscope on as in figure 3-42. Refer to steps a through d if you are unfamiliar with menus.

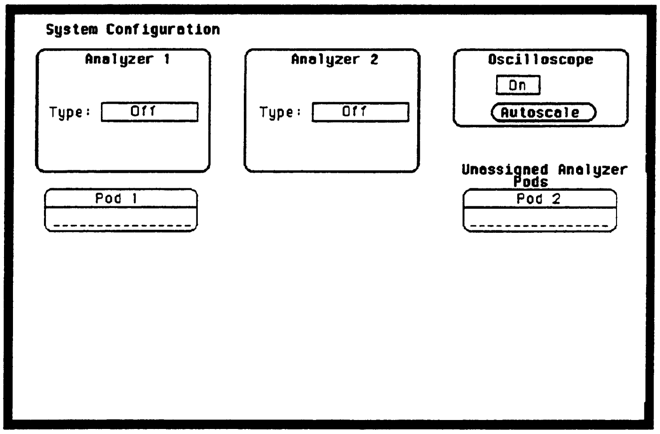


Figure 3-42. System Configuration for Input Resistance

a. Move the cursor to the **Type** field of **Analyzer 1** and press the SELECT key.

b. Set the analyzer **Type** to **Off** using the cursor and SELECT key.

c. Repeat steps a and b for **Analyzer 2**.

d. Move the cursor to the **On/Off** field of the **Oscilloscope** and press the SELECT key to turn the oscilloscope **On**.

5. Press the TRACE/TRIG key and use the cursor and SELECT key to set the **Run Mode** to **Single.**

6. Press the FORMAT/CHAN key and use the cursor and SELECT key to set **Input** to **CH 1**.

7. Set the **Impedance** to **1 MOhm** and press RUN. The multimeter should read 1 MΩ ±10 kΩ .

8. Set the **Impedance** to **50 Ohms** and press RUN. The multimeter should read 50 Ω ±0.5 Ω.

9. Repeat steps 3, 6, 7, and 8 for channel 2.

Note Failure of this test indicates a faulty attenuator if resistance is out of specifications. The oscilloscope assembly also may be at fault if input resistance cannot be changed. See troubleshooting in section 6C for more information.

Voltage Measurement Accuracy Test

Description:

This test verifies the voltage measurement accuracy of the instrument.

Specification:

± (Gain Accuracy + Offset Accuracy + ADC Resolution)

Equipment Required:

Power Supply	HP 6114A
Digital Multimeter	HP 3478A
BNC Cable	HP 10503A
BNC (f)-to-Banana (m) Adapter	HP 1251-2277
Banana (m)-to-Banana (m) Cable (2)	HP 11000-60001

Procedure:

1. Connect the HP 1652B/1653B and test equipment as in figure 3-43.

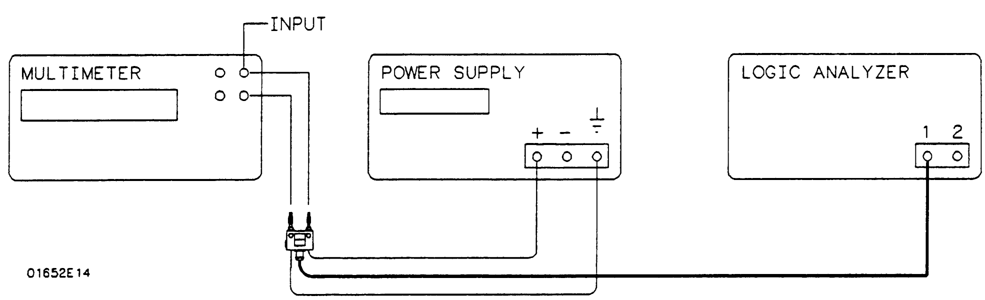


Figure 3-43. Setup for Voltage Measurement Accuracy

2. Connect the power supply to Channel 1 of the HP 1652B/1653B oscilloscope.
3. In the **System Configuration** menu, turn both State/Timing Analyzers off and turn the oscilloscope on as shown in the previous test figure 3-42.

4. Unassign all of the pods from the analyzers as shown in figure 3-44. Refer to steps a through c if you are unfamiliar with menus.

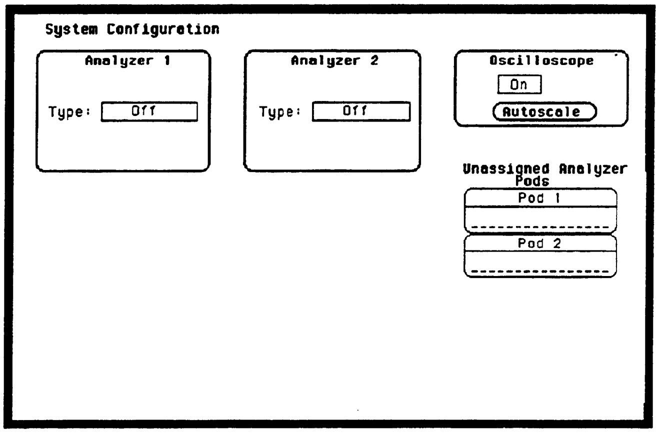


Figure 3-44. System Configuration

a. Move the cursor to an assigned pod and press SELECT.

b. Move the cursor to **Unassigned** and press SELECT.

c. Repeat steps a and b for all assigned pods.

5. Press the FORMAT/CHAN key and turn off channel 2 by deleting the channel 2 waveform as in figure 3-45. Refer to steps a and b if you are unfamiliar with menus.

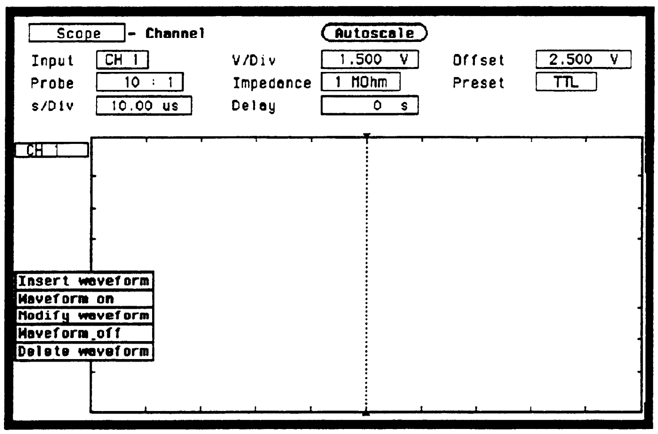


Figure 3-45. Deleting Channel 2

a. Move the cursor to **CH 2** at the left side of the display and press SELECT.

b. Move the cursor to **Delete waveform** and press SELECT.

6. Using the knob and SELECT key, set **Input** to **CH 1**, **Probe** to **1:1**, and **Impedance** to **1 MOhm** as in figure 3-46.

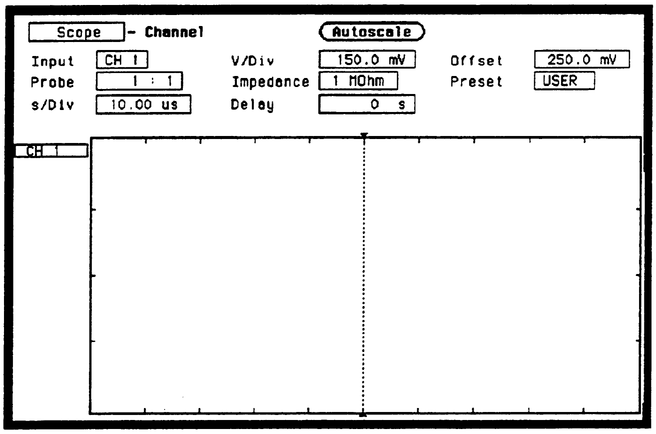


Figure 3-46. Channel Menu Configuration

7. Press the TRACE/TRIG key and set the **Mode** to **Immediate** and **Run mode** to **Repetitive** as in figure 3-47.

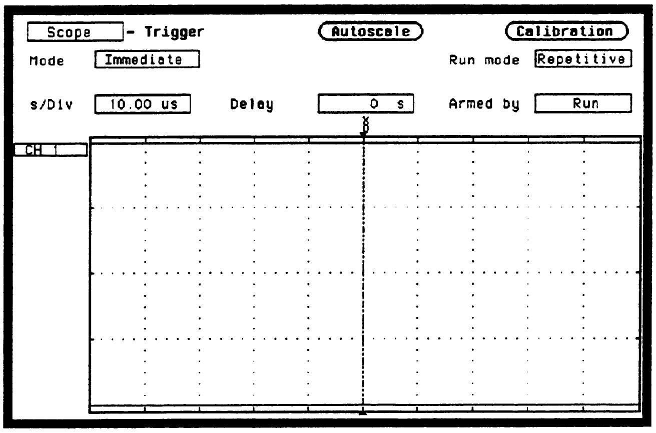


Figure 3-47. Trigger Menu Configuration

8. Press DISPLAY and set **Markers** to **Time**, **Display** to **AVG# 32**, **Connect dots** to **On**, and **Grid** to **On** as in figure 3-48.

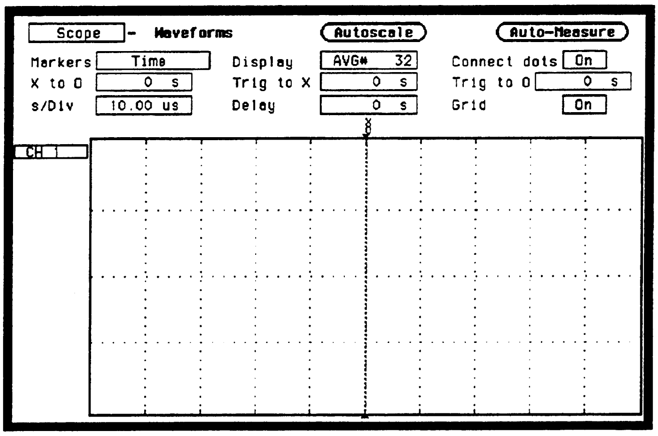


Figure 3-48. Waveforms Menu Configuration

9. Press the FORMAT/CHAN key and set **V/Div** and **Offset** according to the first line of the following table.

V/Div	Offset	Power Supply	Test Tolerance	X and O Result
10.0 V	20.00 V	35.0 V	±3.242 V	31.758 - 38.242 V
5.0 V	10.00 V	17.5 V	±1.622 V	15.878 - 19.122 V
2.0 V	4.00 V	7.0 V	±0.650 V	6.935 - 7.065 V
1.0 V	2.00 V	3.5 V	±0.326 V	3.174 - 3.826 V
500 mV	1.00 V	1.75 V	±0.164 V	1.586 - 1.914 V
200 mV	400 mV	700 mV	±66.8 mV	633.2 - 766.8 mV
100 mV	200 mV	350 mV	±34.4 mV	315.6 - 384.4 mV
50 mV	100 mV	175 mV	±18.2 mV	156.8 - 193.2 mV
20 mV	40 mV	70 mV	±8.48 mV	61.52 - 78.48 mV
15 mV	30 mV	50 mV	±6.86 mV	43.14 - 56.86 mV

10. Set the power supply to the voltage listed on the first line of the previous table.

11. Press the TRACE/TRIG key, then press RUN. The Trigger level cursor will appear as in figure 3-49.

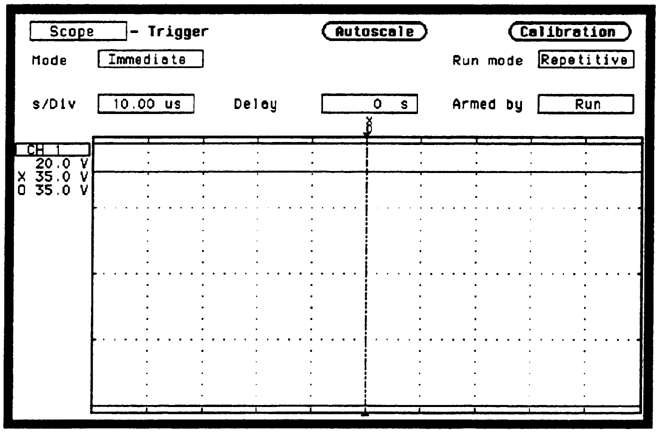


Figure 3-49. Trigger Menu

12. After the display has time to settle, observe the X and O cursor voltage display. Verify that these voltages are within the limits in the previous table and press STOP.

13. Repeat steps 9 through 12 for each line of the table.

14. Press the FORMAT/CHAN key and turn on channel 2 by inserting a waveform on the display. Refer to steps a through c if you are unfamiliar with menus.

    a. Move the cursor to **CH 1** at the left side of the display and press SELECT.

    b. Move the cursor to **Insert waveform** and press SELECT.

    c. Move the cursor to **CH 2** and press SELECT.

15. Turn off channel 1 by deleting the channel 1 waveform.

16. Set **Input** to **CH 2** and connect the power supply to Channel 2.

17. Repeat steps 6 through 13 for channel 2.

Note Voltage measurement errors can be caused by the need for self-calibration. Perform the **Offset Calibration** and **Gain Calibration** (see Adjustments, section 4) before troubleshooting the instrument. If self-calibration fails to correct the problem, the cause may be the attenuator or the oscilloscope assembly.

DC Offset Accuracy Test

Description:

This test verifies the DC offset accuracy of the instrument.

Specification:

±(2 mV + 2% of channel offset + 2.5% of full scale)

Equipment Required:

Digital Voltmeter	HP 3478A
Power Supply	HP 6114A
BNC Cable	HP 10503A
BNC (f)-to-Banana (m) Adapter	HP 1251-2277
Banana (m)-to-Banana (m) Cable	HP 11000-60001

Procedure:

1. Connect the HP 1652B/1653B and test equipment as in figure 3-50.

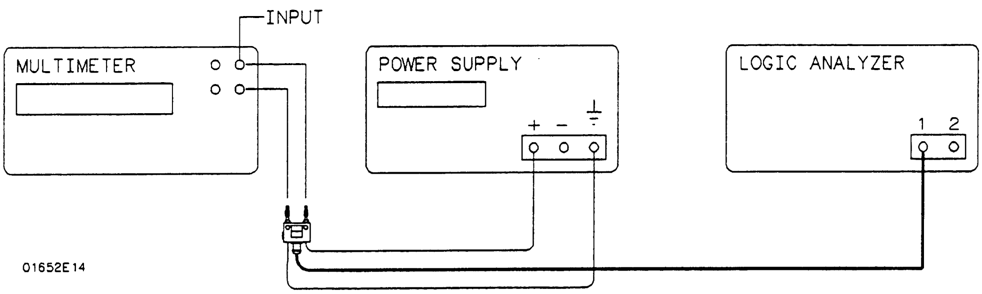


Figure 3-50. Setup for DC Offset Accuracy

2. In the **System Configuration** menu, turn both State/Timing Analyzers off, unassign all of the pods from the analyzers, and turn the oscilloscope on as in the previous test figure 3-44.

3. Press the FORMAT/CHAN key and turn on channel 1 by inserting the channel 1 waveform. Then turn off channel 2 by deleting the channel 2 waveform.

4. Using the knob and SELECT key, set **Input** to **CH 1**, **Probe** to **1:1**, and **Impedance** to **1 MOhm** as in the previous test figure 3-46.

5. Press the TRACE/TRIG key and set the **Mode** to **Immediate** and the **Run mode** to **Repetitive** as in figure 3-51.

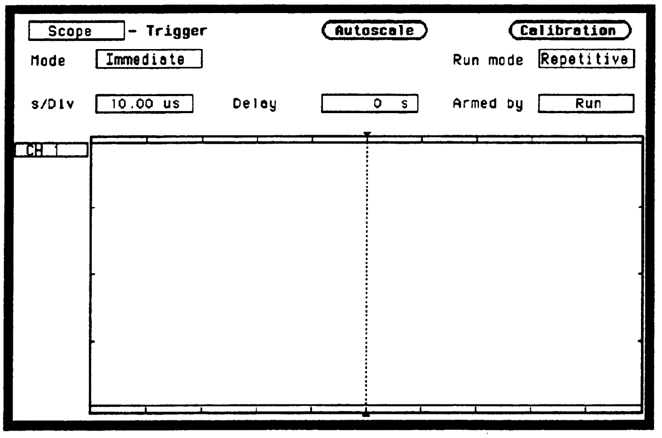


Figure 3-51. Trigger Menu

6. Press DISPLAY and set the **Markers** to **Time, Display** to **AVG# 32, Connect dots On**, and **Grid On** as in figure 3-52.

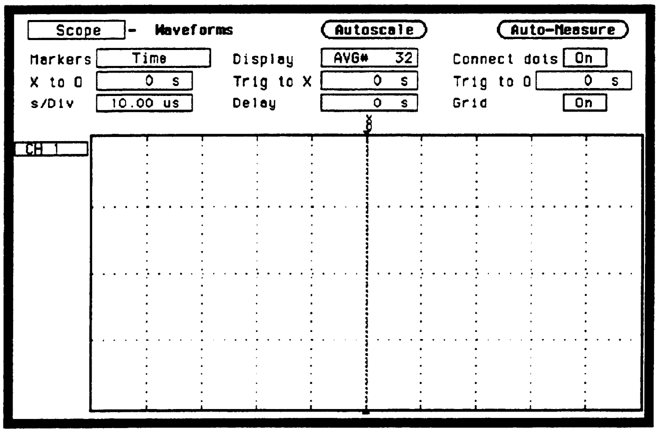


Figure 3-52. Waveforms Display Menu

7. Press FORMAT/CHAN and set **V/Div** and **Offset** according to the first line of the following table.

V/Div	Offset	Power Supply	Test Tolerance	Offset Result
1.0 V	20.0 V	20.0 V	±0.502 V	19.50 - 20.50 V
500 mV	10.0 V	10.0 V	±0.242 V	9.758 - 10.242 V
200 mV	5.0 V	5.0 V	±0.122 V	4.878 - 5.122 V
100 mV	2.0 V	2.0 V	±0.052 V	1.948 - 2.052 V

8. Set the power supply to the voltage listed on the first line of the previous table.

9. Press RUN and readjust **Offset** so the trace is as close to the center horizontal line of the graticule as possible after it has settled (averaging complete) as in figure 3-53.

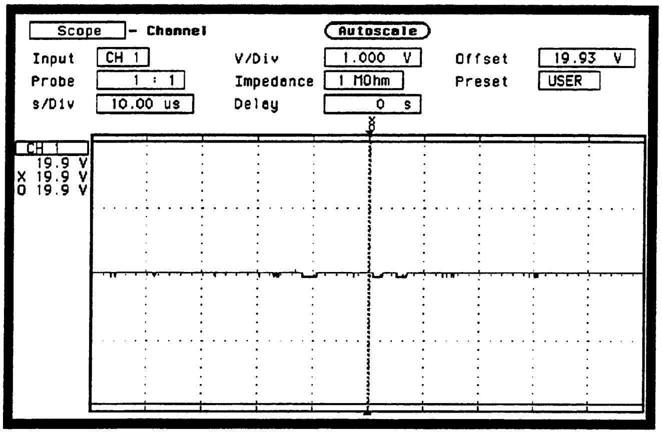


Figure 3-53. Channel Menu

10. Verify that the **Offset** voltage is within the limits specified in the previous table. Then press STOP.

11. Repeat steps 7 through 10 for each line of the table.

12. Turn on channel 2 by inserting a waveform on the display. Refer to steps a through c if you are unfamiliar with menus.

    a. Move the cursor to **CH 1** at the left side of the display and press SELECT.

    b. Move the cursor to **Insert waveform** and press SELECT.

    c. Move the cursor to **CH 2** and press SELECT.

13. Turn off channel 1 by deleting the channel 1 waveform.

14. Set **Input** to **CH 2** and connect the power supply to Channel 2.

17. Repeat steps 7 through 11 for channel 2.

Note Offset errors can be caused by the need for self-calibration. Perform the **Offset Calibration** (see Adjustments) before troubleshooting the instrument. If self-calibration fails to correct the problem, the cause may be the attenuator or oscilloscope assembly.

Bandwidth Test

Description:

This test checks the bandwidth of the oscilloscope in the HP 1652B/1653B.

Note Before doing the Bandwidth test, verify that the **Attenuator Calibration** is valid (performed within the last six months or 1000 hours).

Specification:

Bandwidth: dc to 100 MHz

Equipment Required:

Signal Generator	HP 8656B
Power Meter	HP 436A
Power Sensor	HP 8482A
Power Splitter	HP 11667B
Type N (m) 24 inch cable	HP 11500B
Type N (m) to BNC (m) Adapter	HP 1250-0082

Procedure:

1. With the N cable, connect the signal generator to the power splitter input. Connect the power sensor to one output of the power splitter as in figure 3-54.

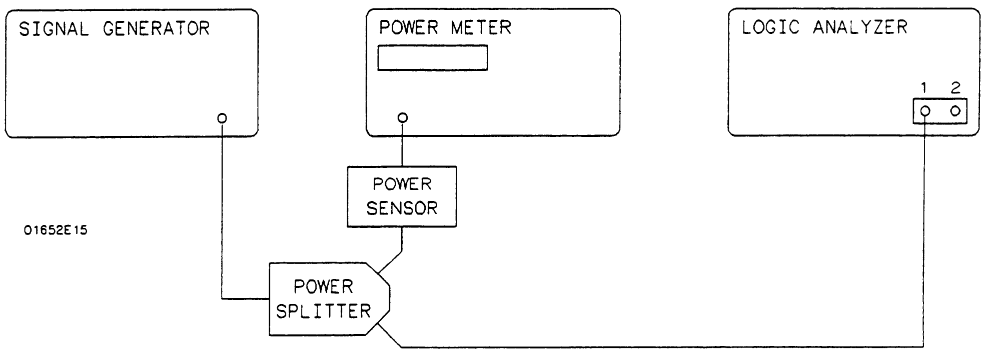


Figure 3-54. Setup for Bandwidth Test

2. Using an N-to-BNC adapter, connect the other power splitter output to the channel 1 input of the HP 1652B/1653B oscilloscope.

3. In the **System Configuration** menu, turn both State/Timing Analyzers off, unassign all of the pods from the analyzers, and turn the oscilloscope on as in the previous figure 3-44.

4. Press FORMAT/CHAN and turn on channel 1 by inserting the channel 1 waveform. Then turn off channel 2 by deleting the channel 2 waveform.

5. Set the **Input** to **CH 1, V/Div** to **100 mV, Offset to 0 V, Probe** to **1:1, Impedance** to **50 Ohms**, and **s/Div** to **5.0 us** as in figure 3-55.

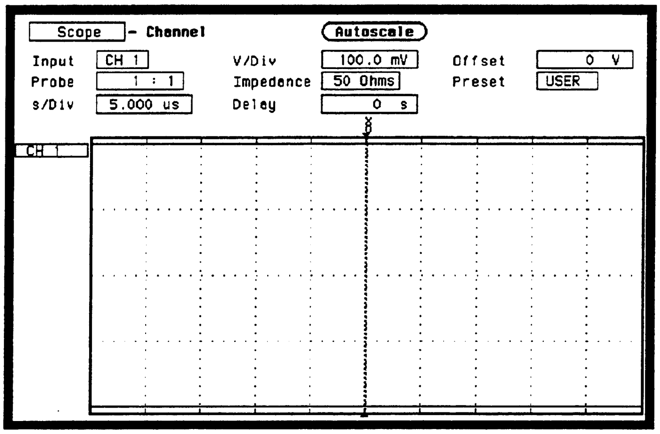


Figure 3-55. Channel Menu Configuration

6. Press TRACE/TRIG and set the **Run mode** to **Repetitive** and trigger **Level** to **0 V** as in the previous figure 3-51.

7. Press DISPLAY and set **Markers** to **Time, Display** to **Normal, Connect dots On**, and **Grid On** as in figure 3-56.

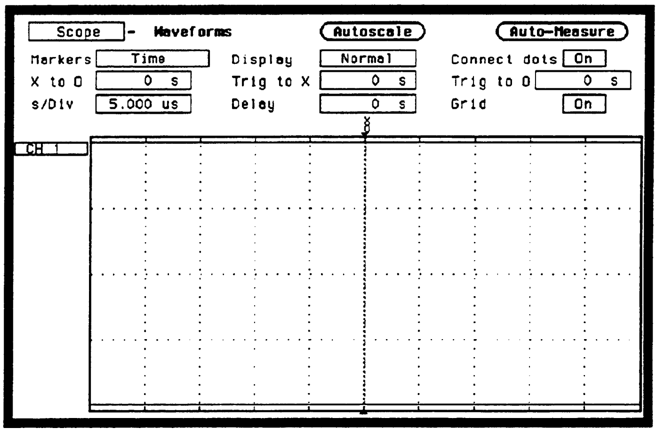


Figure 3-56. Waveforms Display Menu Configuration

8. Set the signal generator for 100 kHz at -4.5 dBm and press RUN on the HP 1652B/1653B. The signal on screen should be five cycles at two divisions amplitude.

9. Press TRACE/TRIG and adjust the trigger **Level** for a stable display. The signal on screen should be five cycles at approximately 2 divisions amplitude.

10. Press DISPLAY and set **Display** to **AVG# 16.**

11. After the measurement settles (averaging complete, about 10 seconds) use **Auto-Measure** to obtain a peak-to-peak voltage measurement as in figure 3-57.

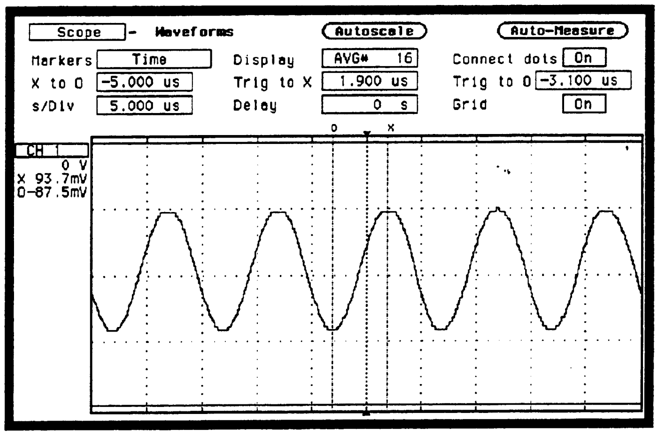


Figure 3-57. Waveforms Display Menu

12. Set the power meter Cal Factor % to the 100 kHz value from the cal chart on the power sensor probe, then press dB[REF] to set a 0 dB reference.

13. In the Waveforms Display menu, set **Display** to **Normal** and **s/Div** to **5 ns.**

14. Change the signal generator to 100 MHz and set the power meter Cal Factor to the 100 MHz % value from chart.

15. Adjust the signal generator amplitude for a power reading as close as possible to 0.0 dB(REL).

16. Set the oscilloscope **Display** to **AVG# 16.**

17. After the measurement settles (averaging complete), use **Auto-Measure** to obtain a peak-to-peak voltage as in step 11. Note this value.

18. Calculate the response using the formula:

$$\text{Response (dB)} = 20 \log_{10} \frac{V_{100MHz}}{V_{100kHz}}$$

19. Correct the result from step 18 with any difference in the power meter from step 15. Observe signs. For example:

Result from step 18	=	-2.3 dB
Power meter reading	=	-0.2 dB(REL)
true response	=	(-2.3)-(-0.2) = -2.1 dB

20. Turn on channel 2 by inserting a waveform on the display.

21. Turn off channel 1 by deleting the channel 1 waveform.

22. Connect the power splitter to channel 2 and repeat steps 5 through 19 for channel 2.

Note Failure of the bandwidth test can be caused by faulty attenuator or oscilloscope assembly, or the need for high-frequency pulse response adjustment.

Time Measurement Accuracy Test

Description:

This test uses a precise frequency source to check the accuracy of the time measurement functions.

Specification:

± (500 ps + 2% X s/Div + 0.01% X delta-t)

Equipment Required:

Signal Generator	HP 8656B
BNC Cable	HP 10503A
Type N (m) to BNC (f) Adapter	HP 1250-0780

Procedure:

1. Use a Type N to BNC adapter to connect the signal generator to the channel 1 input of the HP 1652B/1653B oscilloscope as in figure 3-58.

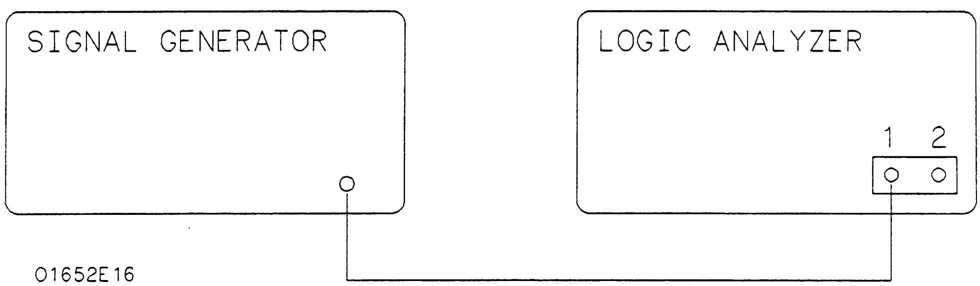


Figure 3-58. Setup for Time Measurement Accuracy

2. In the **System Configuration** menu, turn both State/Timing Analyzers off, unassign all of the pods from the analyzers, and turn the oscilloscope on as in the previous figure 3-44.

3. Press FORMAT/CHAN and turn on channel 1 by inserting the channel 1 waveform. Then turn off channel 2 by deleting the channel 2 waveform.

4. Set **Input** to **CH 1, V/Div** to **100 mV, Offset** to **0 V, Probe** to **1:1, Impedance** to **50 Ohms**, and **s/Div** to **5.0 ns** as in figure 3-59.

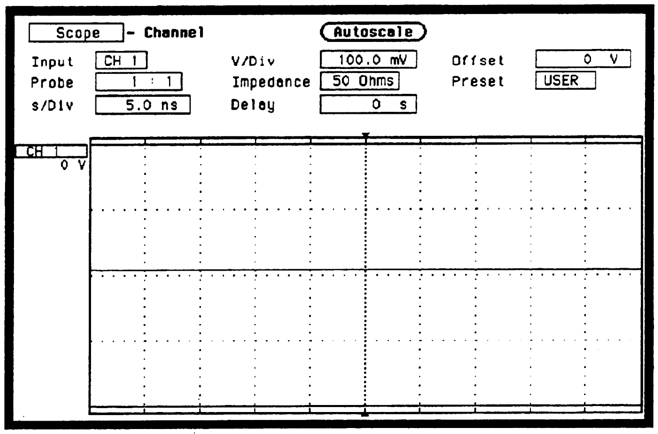


Figure 3-59. Channel Menu Configuration

5. Press DISPLAY and set **Display** to **Normal, Connect dots On**, and **Grid On.**

6. Press TRACE/TRIG and set the **Run mode** to **Repetitive** and trigger **Level** to **0 V** as in the previous figure 3-51.

7. Set the signal generator to 100 MHz at approximately 150 mV.

8. Press RUN and use the **Level** and **Delay** to center the rising edge of the waveform as close as possible to center screen as in figure 3-60. It may be necessary to use **Offset** to center the signal symmetrically about the horizontal axis.

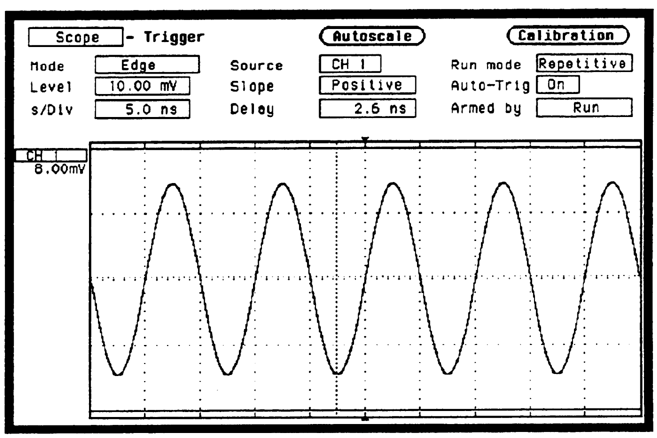


Figure 3-60. Centering the Waveform

9. Press STOP, then press DISPLAY. Set **Display** to **AVG# 16.**

10. Press RUN and when the waveform has stabilized at center screen, press STOP.

11. Use the following table for the next steps.

Delay	Tolerance	Limits
10 ns	±0.601 ns	9.399 to 10.601 ns
100 ns	±0.610 ns	99.39 to 100.61 ns
500 ns	±0.650 ns	499.35 to 500.65 ns

12. Select Delay and enter the delay value listed on the first line of the previous table using the keypad.

13. Select Delay again and use the knob to move the rising edge of the waveform directly over the center reference as in figure 3-61. Verify that the delay is within the limits specified in the table.

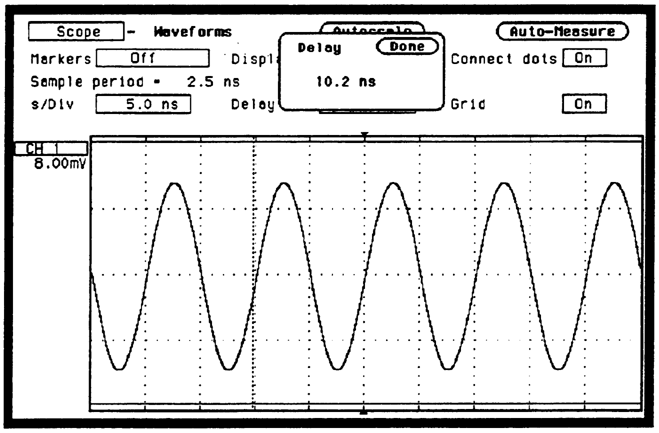


Figure 3-61. Waveforms Display

14. Repeat steps 12 and 13 for the other delays in the table.

15. Set the signal generator to 1 MHz.

16. Press DISPLAY and set Display to Normal.

17. Press FORMAT/CHAN and set **s/Div** to **500 ns** and **Offset** to **0 V.**

18. Repeat steps 8 through 14 using the values in the following table.

Delay	Tolerance	Limits
1 us	±10.600 ns	989.4 to 1.010 us
2 us	±10.700 ns	1.989 to 2.010 us

19. Turn on channel 2 by inserting a waveform on the display.

20. Turn off channel 1 by deleting the channel 1 waveform.

21. Connect the signal generator to channel 2 and repeat steps 4 through 18 for channel 2.

Note Time Measurement Accuracy failure is caused by a defective oscilloscope assembly.

Trigger Sensitivity Test

Description:

This test checks the channel trigger for sensitivity at the rated bandwidth.

Note Before doing the Trigger Sensitivity test, verify that the **Trigger Calibration** is valid (performed in the last 6 months or 1000 hours).

Specification:

10.0% of full scale.

Equipment Required:

Signal Generator	HP 8656B
BNC Cable	HP 10503A
Type N (m)-to-BNC (f)	HP 1250-0780

Procedure:

1. Use the Type N-to-BNC adapter to connect the signal generator to channel 1 of the HP 1652B/1653B as in figure 3-62.

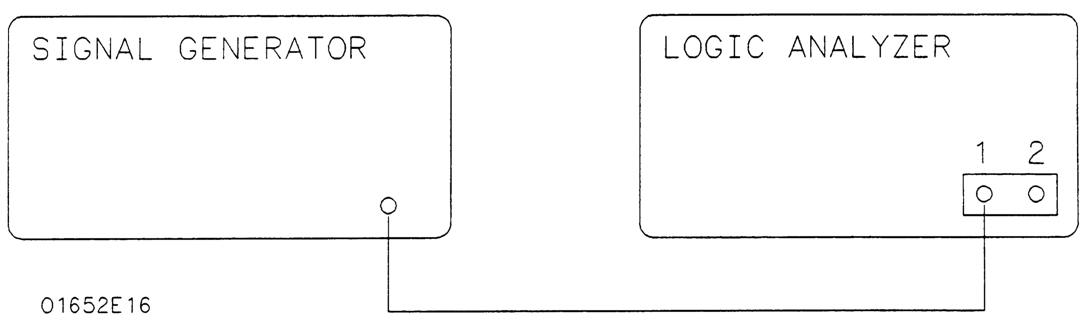


Figure 3-62. Setup for Trigger Sensitivity

2. In the **System Configuration** menu, turn both State/Timing Analyzers off, unassign all of the pods from the analyzers, and turn the oscilloscope on as in the previous figure 3-44.

3. Press FORMAT/CHAN and turn on channel 1 by inserting the channel 1 waveform. Then turn off channel 2 by deleting the channel 2 waveform.

4. Set **Input** to **CH 1**, **V/Div** to **2 V**, **Offset** to **0 V**, **Probe** to **1:1**, **Impedance** to 50 **Ohms**, and **s/Div** to **2.0 us** as in figure 3-63.

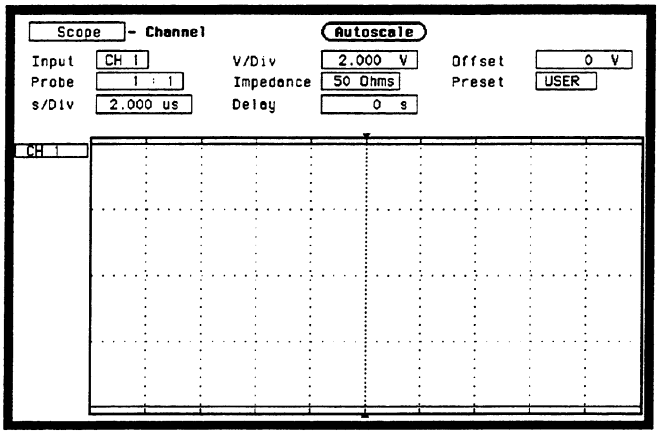


Figure 3-63. Channel Menu Configuration

5. Press TRACE/TRIG and set the **Run mode** to **Repetitive** and trigger **Level** to **0 V.**

6. Press DISPLAY and set **Display** to **Normal**, **Connect dots On**, and **Grid On** as in figure 3-64.

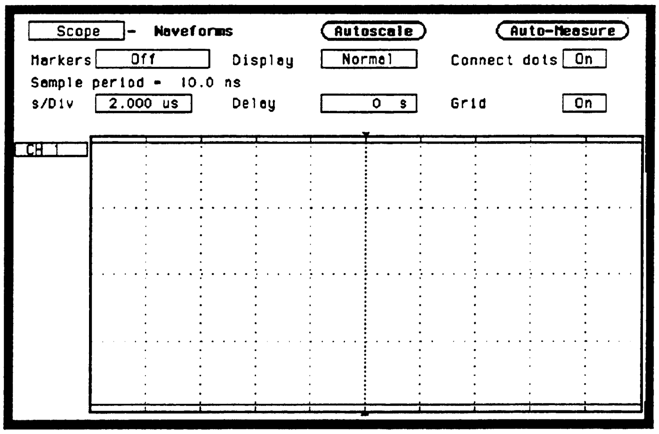


Figure 3-64. Waveforms Display Menu

7. Set the signal generator to 1 MHz and press RUN on the HP 1652B/1653B.

8. Adjust the signal generator output level for 0.4 divisions of vertical deflection (approximately +3 dBm).

9. Press TRACE/TRIG and adjust the trigger **Level** for a stable display (Auto-triggered message does not appear on screen). The test passes if triggering is stable, as shown in figure 3-65.

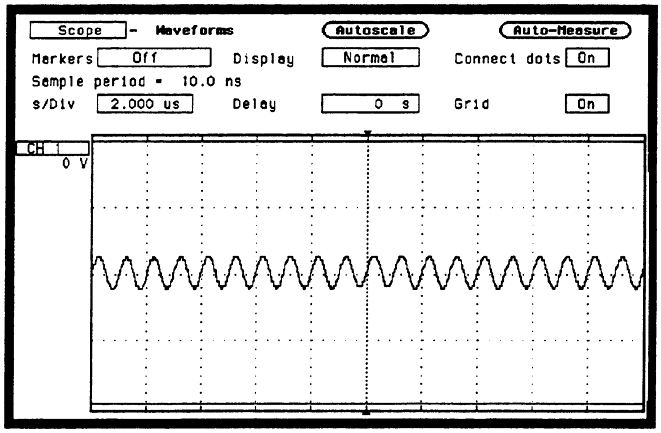


Figure 3-65. Waveforms Display Menu

10. Press STOP and set the **s/Div** to **5 ns**.
11. Set the signal generator to 100 MHz and press RUN on the HP 1652B/1653B.
12. Adjust the signal generator output level for 0.4 divisions of vertical deflection (approximately +3 dBm).
13. Press TRACE/TRIG and adjust the trigger level for a stable display (Auto-triggered message does not appear on screen). The test passes if triggering is stable.
14. Press FORMAT/CHAN and set V/Div to 200 mV and repeat steps 4 through 13. The signal generator output should be reduced to approximately -17 dBm.
15. Press FORMAT/CHAN and set V/Div to 20 mV and repeat steps 4 through 13. The signal generator output should be set to approximately -37 dBm.
16. Turn on channel 2 by inserting a waveform on the display.
17. Turn off channel 1 by deleting the channel 1 waveform.
18. Connect the signal generator to channel 2 and repeat steps 4 through 15 for channel 2.

Note Trigger sensitivity test failure is caused by a defective attenuator or oscilloscope assembly.

Table 3-1. Performance Test Record

Hewlett-Packard Model 1652B/1653B Logic Analyzer Serial Number ____________	Tested by ____________ Work Order No. ________ Date Tested ____________

Recommended Calibration
Interval _24_ Months

Test	Results		
Clock, Qualifier, and Data Inputs Test 1		Passed	Failed
	Pod1	______	______
	Pod2	______	______
	Pod3	______	______
	Pod4	______	______
	Pod5	______	______
Clock, Qualifier, and Data Inputs Test 2		Passed	Failed
	Pod1	______	______
	Pod2	______	______
	Pod3	______	______
	Pod4	______	______
	Pod5	______	______
Clock, Qualifier, and Data Inputs Test 3 (HP 1652B Only)		Passed	Failed
	Pod1	______	______
	Pod2	______	______
	Pod3	______	______
	Pod4	______	______
	Pod5	______	______
Clock, Qualifier, and Data Inputs Test 4		Passed	Failed
	Pod1	______	______
	Pod2	______	______
	Pod3	______	______
	Pod4	______	______
	Pod5	______	______
Clock, Qualifier, and Data Inputs Test 5		Passed	Failed
	Pod1	______	______
	Pod2	______	______
	Pod3	______	______
	Pod4	______	______
	Pod5	______	______

Table 3-1. Performance Test Record (continued)

Test	Results		
Clock, Qualifier, and Data Inputs Test 6		Passed	Failed
	Pod1	______	______
	Pod2	______	______
	Pod3	______	______
	Pod4	______	______
	Pod5	______	______
Glitch Test		Passed	Failed
	Pod1	______	______
	Pod2	______	______
	Pod3	______	______
	Pod4	______	______
	Pod5	______	______
Threshold Accuracy Test		Passed	Failed
	Pod1	______	______
	Pod2	______	______
	Pod3	______	______
	Pod4	______	______
	Pod5	______	______

Table 3-1. Performance Test Record (continued)

Test	Limits		Results	
Voltage Measurement Accuracy Test	Range		Chan 1	Chan2
	10.0 V	31.758 - 38.242 V	_____	_____
	5.0 V	15.878 - 19.122 V	_____	_____
	2.0 V	6.935 - 7.065 V	_____	_____
	1.0 V	3.174 - 3.826 V	_____	_____
	500 mV	1.586 - 1.914 V	_____	_____
	200 mV	633.2 - 766.8 mV	_____	_____
	100 mV	315.6 - 384.4 mV	_____	_____
	50 mV	156.8 - 193.2 mV	_____	_____
	20 mV	61.52 - 78.48 mV	_____	_____
	15 mV	43.14 - 56.86 mV	_____	_____
DC Offset Accuracy Test	Range		Chan 1	Chan 2
	1.0 V	19.50 - 20.50 V	_____	_____
	500 mV	9.758 - 10.242 V	_____	_____
	200 mV	4.878 - 5.122 V	_____	_____
	100 mV	1.948 - 2.052 V	_____	_____
Bandwidth Test			Chan 1	Chan 2
		Down <3dB at 100 MHz	_____	_____
Time Measurement Accuracy Test			Chan 1	Chan 2
	10 ns	9.4 to 10.6 ns	_____	_____
	100 ns	99.4 to 100.6 ns	_____	_____
	500 ns	499.4 to 500.6 ns	_____	_____
	1 us	989.5 to 1.010 us	_____	_____
	2 us	1.990 to 2.010 us	_____	_____
Trigger Sensitivity Test			Chan 1	Chan 2
	2 V/div	0.4 div at 1 MHz	_____	_____
		0.4 div at 100 MHz	_____	_____
	200 mV/div	0.4 div at 1 MHz	_____	_____
		0.4 div at 100 MHz	_____	_____
	20 mV/div	0.4 div at 1 MHz	_____	_____
		0.4 div at 100 MHz	_____	_____

Contents

Section 4:

Adjustments and Calibration

4

Adjustments and Calibration

Introduction

This section provides the adjustment procedures for the HP 1652B/1653B. The primary adjustments groups are:

- Power Supply Assembly Adjustment.
- CRT Monitor Assembly Adjustment.
- System Board Assembly Threshold Adjustment.
- Oscilloscope Assembly High-Frequency Pulse Adjustment.

This section also contains the software calibration procedures for the oscilloscope assembly.

Caution

The effects of ELECTROSTATIC DISCHARGE can damage electronic components. Use grounded wriststraps and mats when performing any kind of service to this instrument.

Equipment Required

The equipment required for the adjustments and calibration procedures in this section are listed in the Recommended Equipment table in section 1 of this manual. Any equipment that satisfies the critical specifications listed in this table may be substituted for the recommended model. Equipment for individual procedures is listed with the procedure.

Adjustments and Calibration Interval

The recommended adjustment interval for the HP 1652B/1653B is two years. The adjustments are set at the factory on assemblies when they are tested. However, adjustments may be necessary after repairs have been made to the instrument. Usually the only assembly that may require adjustments is the assembly that has been replaced.

Warning

Read the Safety Summary at the beginning of this manual before performing any adjustment procedures.

Software calibration should be done on the HP 1652B/1653B oscilloscope under any of the following conditions:

- At six month intervals or every 1,000 hours.
- If the ambient temperature changes more than 10° C from the temperature at the last software calibration.
- To optimize measurement accuracy.

Safety Requirements

Specific warnings, cautions, and instructions are placed wherever applicable throughout the manual. These must be observed during all phases of operation, service, and repair of the instrument. Failure to comply with them violates safety standards of design, manufacture, and intended use of this instrument. Hewlett-Packard assumes no liability for the failure of the customer to comply with these safety requirements.

Instrument Warm-up

Adjust and calibrate the instrument at it's environmental ambient temperature and after a 15 minute warm-up.

Adjustments

Unless specified elsewhere, each adjustment procedure must be followed in its entirety and in the same sequence shown.

Caution

The adjustment procedures are performed with the top cover of the instrument removed. Take care to avoid shorting or damaging internal parts of the instrument.

Warning

Read the Safety Summary at the beginning of this manual before performing any adjustment procedures.

Calibration

The calibration procedures in this section should be followed in their entirety and in the same sequence shown in this section. The steps in each succeeding procedure assumes that all the previous procedures have been completed in the proper order.

Note

Calibration constants are stored in system memory and not on the Operating System Disk. Therefore, software calibration is not required when a different Operating System Disk is used to boot the instrument on power-up.

Power Supply Assembly Adjustment

Equipment Required:

Digital Voltmeter .... HP 3478A

Procedure:

Note

The Power Supply Adjustment should be performed prior to the other adjustment and calibration procedures.

1. Disconnect the power cord from HP 1652B/1653B and remove the top cover. Then refer to figure 4-1 for the testpoint and adjustment locations.

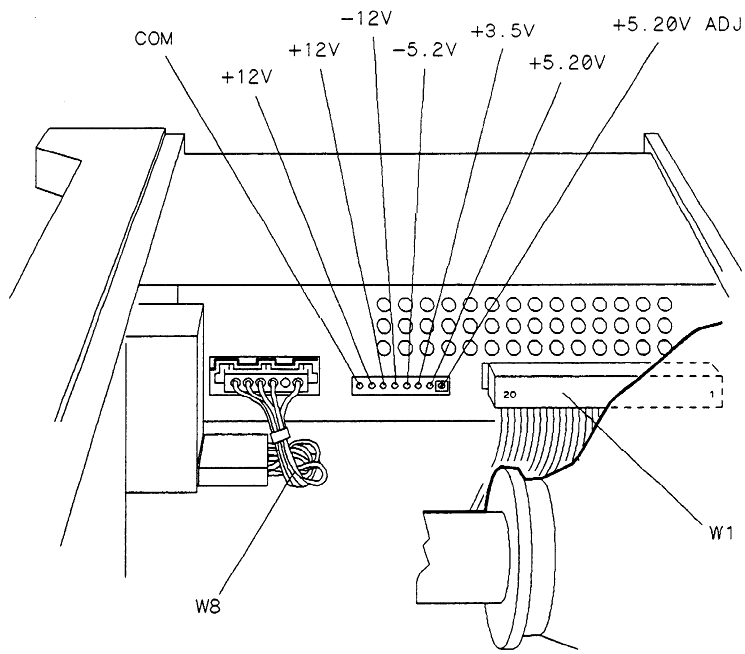


Figure 4-1. Power Supply Adjustments

2. Connect the negative lead of the voltmeter to the COM test point on the Power Supply Assembly.
3. Connect the positive lead of the voltmeter to +5V on the Power Supply Assembly.
4. Connect the power cord to the HP 1652B/1653B and turn the instrument on.
5. The voltmeter reading should be within the range of +5.180 V to +5.220 V. If the voltmeter reading is out of this range, adjust the +5.20V ADJ on the Power Supply Assembly to +5.200 V ±0.020 V (+5.180 V to +5.220 V).

Caution

High voltages exist on the sweep board that can cause personal injury. Avoid contact with the CRT monitor sweep board when adjusting the +5.20V.

CRT Monitor Assembly Adjustments

The CRT Monitor Assembly Adjustments optimize the characters of the CRT display. Set up the instrument for these adjustments as follows:

1. Turn off the HP 1652B/1653B and disconnect the power cord. Then remove the top cover.

Caution

The adjustment procedures are performed with the top cover of the instrument removed. Take care to avoid shorting or damaging internal parts of the instrument.

2. Connect the power cord to the HP 1652B/1653B and turn on the instrument.
3. In the **System Configuration** menu, select the Type field for Analyzer 1 (MACHINE 1) and, when the pop-up appears, select **Timing**.

Intensity, Sub-Bright, and Contrast Adjustment

1. Press the DISPLAY key to place the **Timing Waveforms** menu on the screen of the HP 1652B/1653B.

Note

This menu is used because it has characters throughout the screen which are watched during the procedures. Any other menu may be used, however, the adjustments may not be as accurate if characters and/or lines are not displayed throughout the screen.

2. Set the rear-panel INTENSITY control to its minimum setting.
3. Refer to figure 4-2 for the adjustment locations.

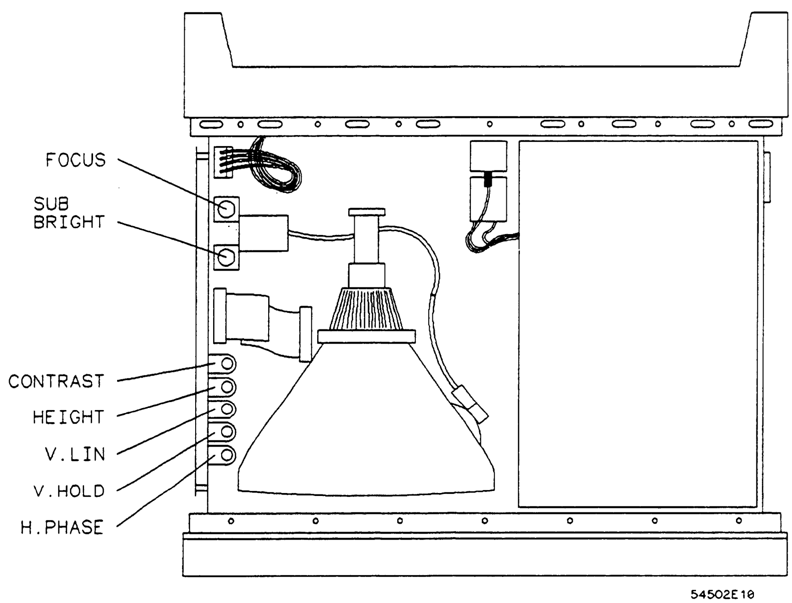


Figure 4-2. CRT Adjustment Locations

4. Adjust the sweep board SUB BRIGHT control to the lowest setting of brightness where the menu is still visible on the CRT screen.

Caution

High voltages exist on the sweep board. Avoid contact with the sweep board when making CRT adjustments.

5. Adjust the rear-panel INTENSITY control to bring up the intensity level on screen. Screen intensity should be at a comfortable viewing level and the position of both adjustments should be close to mid-range.

Note

Setting the intensity level excessively high may shorten the life of the CRT. For optimum usage, set the intensity as low as possible while maintaining a comfortable viewing level.

6. Press RUN and then STOP.
7. Adjust the sweep board CONTRAST control so that the error message is easily seen.

Focus Adjustment

1. Refer to the previous figure 4-2 for the adjustment locations.
2. Press the DISPLAY key to place the **Timing Waveforms** menu on the screen of the HP 1652B/1653B.
3. Adjust the sweep board FOCUS control for sharp pixels in the center of the screen menu. Then note the FOCUS control position.
4. Adjust the sweep board FOCUS for sharp pixels at the corners of the screen. Then note the FOCUS control position.
5. Set the sweep board FOCUS control for mid-position between the two positions noted in steps 3 and 4 for best over-all pixel focus.

Horizontal Phase, Vertical Linearity, and Height Adjustments

1. Refer to the previous figure 4-2 for adjustment locations.
2. Press DISPLAY to place the **Timing Waveforms** menu on the screen of the HP 1652B/1653B.

Note

This menu is used because it has characters and lines throughout the menu which are watched during the procedures. Any other menu may be used, however, the adjustments may not be as accurate.

3. Adjust the sweep board H. PHASE control to center the menu horizontally on the CRT screen.
4. Adjust the sweep board V. LIN control so that the top and bottom rows of text are equal in height. Text height should be approximately 1mm.
5. Adjust the sweep board HEIGHT control so that the screen menu top and bottom borders are equal in width to the side borders of the menu.

6. Readjust steps 4 and 5 as necessary for a uniform display of the screen menu.

Note

The V.LIN and HEIGHT adjustments interact with each other and may need to be repeated for best results.

System Board Assembly Threshold Adjustment

Equipment Required:

Digital Voltmeter ....HP 3478A
Power Supply Cable 54503-61604

Procedure:

Note

The threshold adjustment A1R95 is located beneath the oscilloscope board and is not accessible without dismantling part of the instrument. Consequently, it is advisable to see if the threshold requires adjustment before dismantling the instrument. Perform the **Threshold Accuracy Test** in section 3 to verify if adjustment is required before executing this procedure.

1. Disconnect the power cord from HP 1652B/1653B and remove the top cover.
2. Connect the negative (-) lead of the voltmeter to TP GND. Refer to figure 4-3 for testpoint and adjustment locations.

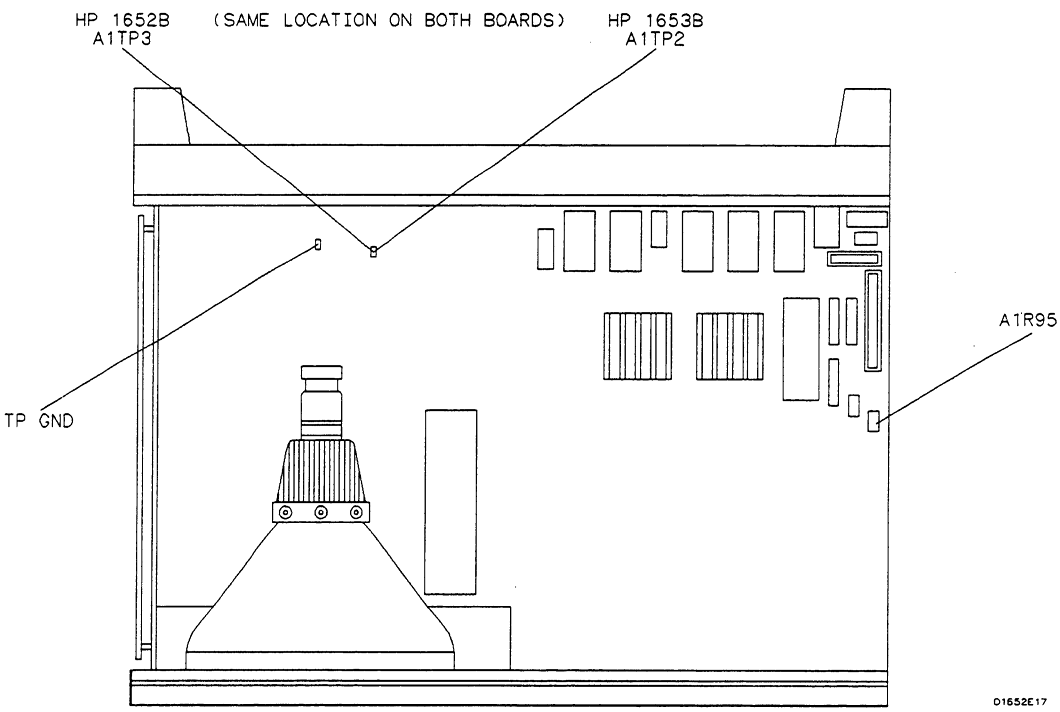


Figure 4-3. System Board Testpoints and Adjustments

3. Connect the positive (+) lead of the voltmeter to A1TP3 on the HP 1652B, or A1TP2 on the HP 1653B, System Board Assembly.

4. Connect the power cord to HP 1652B/1653B and turn on the instrument.

5. Assign pod 3 of the HP 1652B, or pod 2 of the HP 1653B, to a machine in the **System Configuration** menu by using front-panel knob and SELECT key.

6. Press the CHAN/FORMAT key and set the **User-defined** pod threshold of the pod assigned in the previous step to -9.9 V. Refer to the following steps if you are unfamiliar with menus.

   a. Move cursor to the pod threshold field (TTL) with the front-panel knob and press SELECT.

   b. Move the cursor to **User-defined** with the front-panel knob and press SELECT.

   c. Use the front-panel keys to enter the value -9.9 V. Then move the cursor to **Done** and press SELECT.

7. The voltmeter readout should indicate a voltage value within the range of -.975 V to -1.005 V (-.990 V ±0.015 V).

8. Set the **User-defined** pod threshold of the pod assigned in step 6 to +9.9 V.

9. Note voltmeter readout. The voltage reading should be within the range of +.975 V to +1.0005 V (+ .990 V ±0.015 V).

10. If either voltage reading is not within the given range, use the following procedure to adjust the threshold level.

    a. Turn off the instrument and disconnect the power cable.

Caution Never attempt to remove or install any assembly with the instrument ON or the power cable connected. This can result in component damage.

---

   b. Remove the oscilloscope assembly by following the procedure "Removal and Replacement of the Oscilloscope Assembly" in section 6D.

   c. Loosen the two screws that hold the rear bracket on the oscilloscope assembly support panel until the bracket moves freely.

   d. Remove the support panel by carefully tilting the rear of the panel up and lifting the panel out through the top of the instrument cabinet. Make sure the metal tabs on the front of the support panel clear the front panel.

   e. Reconnect the power supply using the power supply cable (HP part number 54503-61604).

   f. Reconnect the disk drive assembly cable to the disk drive.

   g. Reinstall the oscilloscope assembly without the support panel to allow unabstracted access to A1R95.

   h. Reconnect the oscilloscope assembly to the system assembly with the appropriate cable.

i. Reconnect the line filter assembly to the power supply.

j. Reconnect the power cord and turn on the instrument.

k. Repeat steps 5 and 6.

l. Set the **User-defined** pod threshold of the pod assigned to -9.9 V.

m. With the digital voltmeter connected, adjust A1R95 for reading of -.9900 V ±0.0005 V. Refer to figure 4-3 for adjustment locations.

n. Set the **User-defined** pod threshold to +9.9 V.

o. Note the difference between this reading and +0.9900 V.

p. Adjust A1R95 so the difference in step d is halved, ±0.0005 V.

q. When the adjustment is complete, turn off the instrument and remove the power cord. Then reassemble the instrument. Refer to the section "Removal and Replacement of the Oscilloscope Assembly" for addition information on reassembling the instrument.

EXAMPLES

If the reading is + .9952 V, the difference is .0052 V. Adjust A1R95 for +.9926 V ±0.0005 V.

If the reading is + .9834 V, the difference is .0066 V. Adjust A1R95 for +.9867 V ±0.0005 V.

Oscilloscope Assembly High-Frequency Pulse Adjustment

This procedure optimizes the pulse response so that the instrument will meet its bandwidth specification.

Note

This procedure should not be performed as part of the routine adjustments. Typically, it needs to be done only when the instrument fails the bandwidth performance test, an attenuator has been changed, or the oscilloscope assembly has been changed (new combination of attenuators and PC board). Only the channel(s) involved with the failure or repair should be adjusted.

Equipment Required:

Pulse Generator .... Picosecond Pulse Labs 2700C
BNC Cable .... HP 10503A

Procedures:

1. Turn off the HP 1652B/1653B and disconnect the power cord. Then remove the top cover.
2. Connect the power cord to the HP 1652B/1653B and turn on the instrument.
3. Set the pulse generator for pulse output.

Caution

Attenuate the signals from the Picosecond Pulse Labs generator by at least 20 dB. Setting the attenuation from 0 dB to 20 dB may result in damage to the HP 1652B/1653B attenuators.

4. In the **System Configuration** menu, turn both State/Timing analyzers Off, and turn the oscilloscope On.
5. Press FORMAT/CHAN and set **V/Div** to **100 mV**, **Offset** to **150 mV**, **Probe** to **1:1**, **Impedance** to **50 Ohms**, and **s/Div** to **5 ns**.
6. Press TRACE/TRIG and set the **Run mode** to **Repetitive**.
7. Press DISPLAY and set **Display** to **Normal**, **Connect dots On**, and **Grid On**.
8. Connect the pulse generator to the channel 1 input of the HP 1652B/1653B oscilloscope and press RUN.
9. Press TRACE/TRIG and adjust the trigger **Level** for a stable display.
10. Press FORMAT/CHAN and adjust **V/Div** and **Offset** to obtain a waveform as shown in figure 4-4.

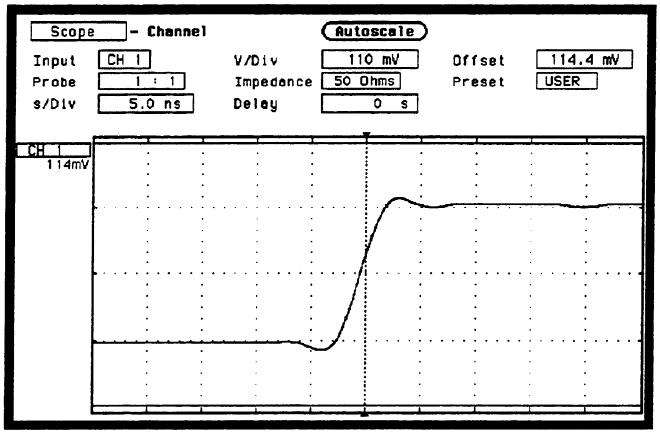


Figure 4-4. Channel Menu

11. Press Display and set **Display** to **AVG# 32.**

12. Select **Auto-Measure** and verify the overshoot and rise time as shown in figure 4-5. Use the SELECT key to toggle the **Auto-Measure** display between CH 1 and CH 2 to update the information.

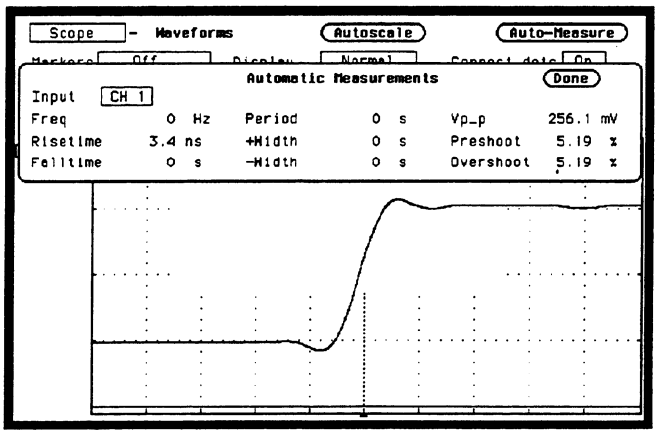


Figure 4-5. Waveforms Display Menu

13. The rise time should be 3.5 nS and overshoot should be < 10%. If either of these is out of specification, adjust the appropriate capacitor. The capacitor locations are shown in figure 4-6.

    - Capacitor C119 is for Channel 1.
    - Capacitor C161 is for Channel 2.

    These capacitors are located on the oscilloscope assembly board. They can be accessed through the right side of the instrument, just below the power supply assembly. The optimum rise time is approximately 3.2 nS.

Note Increase overshoot slightly if the instrument fails the bandwidth test. However, keep the overshoot within the specification.

---

14. Repeat steps 3 through 13 for channel 2.

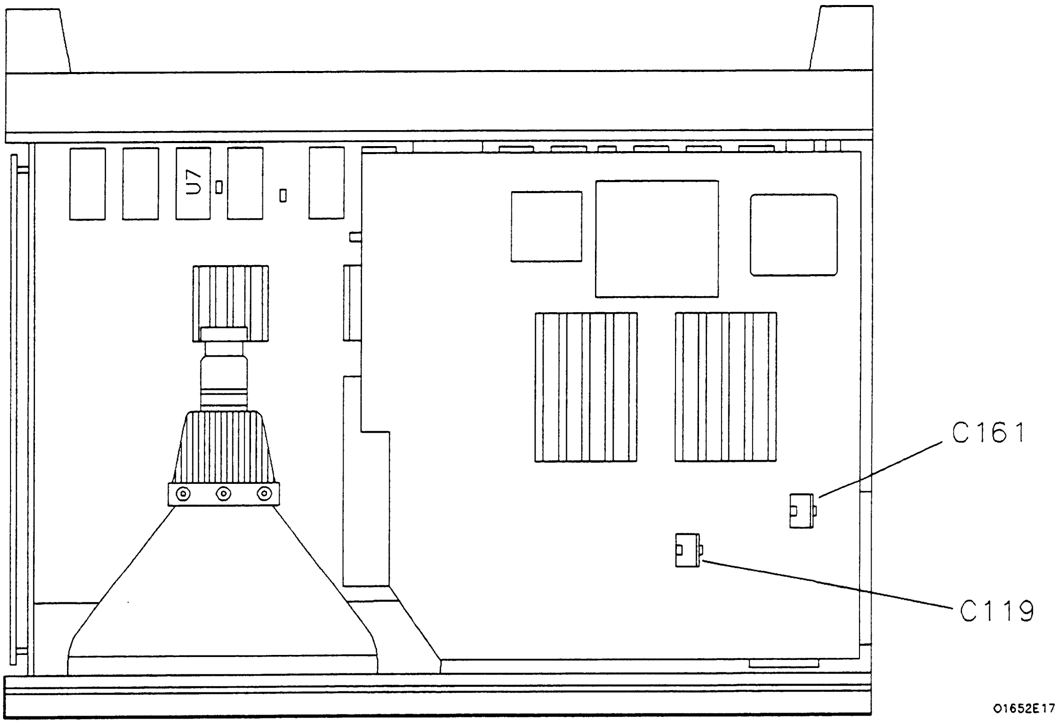


Figure 4-6. High-Frequency Pulse Adjustments

Software Calibration

Software Calibration is accessed through the Trigger menu of the oscilloscope. The calibration procedures in this section should be followed in their entirety and in the same sequence shown.

Note

An instrument warm-up of 15 minutes is recommended before starting these procedures.

Offset Calibration

1. In the **System Configuration** menu turn both State/Timing analyzers Off, and turn the oscilloscope On.

2. Press TRACE/TRIG and select **Calibration** using the front-panel knob and SELECT key.

Note

Offset should be listed as the default Calibration choice. If not, select the Calibration choice field and, when the pop-up appears, select **Offset.**

---

3. Select **Start** with the front-panel knob and SELECT key.

Note

To abort the Offset calibration, select **Cancel** using the front-panel knob and SELECT key.

---

4. Disconnect all signals from the channel 1 and 2 inputs of the HP 1652B/1653B oscilloscope. Then select **Continue** using the front-panel knob and SELECT key to proceed with the calibration. A message will appear on screen to indicate the instrument is performing the calibration.

5. When the calibration is complete, the updated calibration status appears on screen and the instrument remains in the Calibration menu.

Attenuator Calibration

Equipment Required:

DC Power Supply .... HP 6114A
Digital Voltmeter .... HP 3478A

6. Select the Calibration choice field and, when the pop-up appears, select **Attenuation.**

7. Connect the power supply to the HP 1652B/1653B oscilloscope and monitor the supply with the digital voltmeter as shown in figure 4-7.

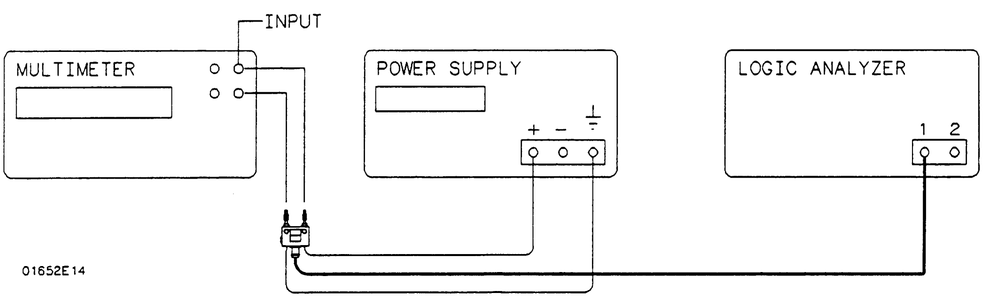


Figure 4-7. Attenuator Calibration Setup

8. Select **Start** with the front-panel knob and SELECT key. The instrument will display the appropriate DC voltages required and prompt you to connect the power supply to the appropriate channel.

Note

To abort the Attenuator calibration, select **Cancel** using the front-panel knob and SELECT key.

---

9. Adjust the power supply to within 0.1% of the specified voltage. If the measured DC source varies more than 0.1%, select the voltage field with the front-panel knob and SELECT key. Then enter the source level and select DONE.

10. To proceed with the calibration, select **Continue** using the front-panel knob and SELECT key.

11. Repeat steps 9 and 10 for each specified voltage and channel.

12. When the calibration is complete, the updated calibration status appears on screen and the instrument remains in the Calibration menu.

13. Disconnect the power supply from the HP 1652B/1653B inputs.

Gain Calibration

14. Select the Calibration choice field and, when the pop-up appears, select **Gain**.

15. Select **Start** with the front-panel knob and SELECT key.

Note

To abort the Gain calibration, select **Cancel** using the front-panel knob and SELECT key.

---

16. Disconnect all signals from the channel 1 and 2 inputs of the HP 1652B/1653B oscilloscope. Then select **Continue** using the front-panel knob and SELECT key to proceed with the calibration. A message will appear on screen to indicate the instrument is performing the calibration.

17. When the calibration is complete, the updated calibration status appears on screen and the instrument remains in the Calibration menu.

Trigger Calibration

18. Select the Calibration choice field and, when the pop-up appears, select **Trigger level.**

19. Select **Start** with the front-panel knob and SELECT key.

Note

To abort the Trigger calibration, select **Cancel** using the front-panel knob and SELECT key.

---

20. Disconnect all signals from the channel 1 and 2 inputs of the HP 1652B/1653B oscilloscope. Then select **Continue** using the front-panel knob and SELECT key to proceed with the calibration. A message will appear on screen to indicate the instrument is performing the calibration.

21. When the calibration is complete, the updated calibration status appears on screen and the instrument remains in the Calibration menu.

Delay Calibration

22. Select the Calibration choice field and, when the pop-up appears, select **Delay.**

23. Select **Start** with the front-panel knob and SELECT key.

Note

To abort the Delay calibration, select **Cancel** using the front-panel knob and SELECT key.

---

24. Connect a BNC cable from the Probe Compensation output on the rear panel to the channel 1 input of the HP 1652B/1653B oscilloscope. The instrument will prompt you when you need to switch to the channel 2 input.

Note

If you use a 10:1 probe in place of the recommended 1:1 BNC cable, use the BNC-to-mini probe adapter supplied with the instrument. Then set the attenuation field in step 25 to 10:1.

---

25. Set the attenuation field in the calibration menu to the appropriate setting.

26. To proceed with the calibration, select **Continue** using the front-panel knob and SELECT key.

27. When the calibration is complete, the updated calibration status appears on screen and the instrument remains in the Calibration menu.

Caution

Do not execute **Set to Default** after calibrating the instrument. Otherwise, your calibration factors will be replaced by default calibration factors.

---

28. Select **Done** with the front-panel knob and SELECT key to exit the Calibration menu.

Contents

Section 5:

Replaceable Parts

5

Replaceable Parts

Introduction

This section contains information for ordering parts. Since service support for this instrument is down to the assembly level, the replaceable parts list only includes assemblies and chassis parts. Figure 5-1 shows an exploded view of the HP 1652B/1653B Logic Analyzer.

Abbreviations

Table 5-1 lists the abbreviations used in the parts list and throughout this manual. In some cases two forms of the abbreviations are used: one in all capital letters, and one in partial or no capital letters. However, elsewhere in the manual, other abbreviation forms may be used with both lowercase and uppercase letters.

Replaceable Parts

Table 5-2 is a list of replaceable parts and is organized as follows:

1. Exchange assemblies in alphanumerical order by reference designation.
2. Electrical assemblies in alphanumerical order by reference designation.
3. Chassis-mounted parts in alphanumerical order by reference designation.

The information given for each part consists of the following:

- Reference designation.
- HP part number.
- Part number Check Digit (CD).
- Total quantity (Qty) used in the instrument or on an assembly. The total quantity is given once at the first appearance of the part number in the list.
- Description of the part.
- Typical manufacturer of the part in an identifying five-digit code. All parts in this list (except hardware) are manufactured by or for Hewlett-Packard, code 28480. No list of manufacturers is provided.

Exchange Assemblies

Some parts used in this instrument have been set up for an exchange program. This program allows the customer to exchange a faulty assembly with one that has been repaired, calibrated, and performance-verified by the factory. The cost is significantly less than that of a new part. The exchange parts have a part number in the form XXXXX-695XX.

After receiving the repaired exchange part from Hewlett-Packard, a United States customer has 30 days to return the faulty assembly. For orders not originating in the United States, contact the local HP service organization. If the faulty assembly is not returned within the warranty time limit, the customer will be charged an additional amount. The additional amount will be the difference in price between a new assembly and that of an exchange assembly.

Ordering Information

To order a part in the material list, quote the HP part number, indicate the quantity desired, and address the order to the nearest HP Sales/Service Office.

To order a part not listed in the material list, include the instrument part number, instrument serial number, a description of the part (including its function), and the number of parts required. Address the order to the nearest HP Sales and Service Office.

Direct Mail Order System

Within the USA, Hewlett-Packard can supply parts through a direct mail order system. There are several advantages to this system:

- Direct ordering and shipment from the HP Parts Center in California, USA.
- No maximum or minimum on any mail order (there is a minimum amount for parts ordered through a local HP office when the orders require billing and invoicing).
- Prepaid transportation (there is a small handling charge for each order).
- No invoices.

In order for Hewlett-Packard to provide these advantages, a check or money order must accompany each order.

Mail order forms and specific ordering information are available through your local HP office. Addresses and telephone numbers are in a separate document included with this manual.

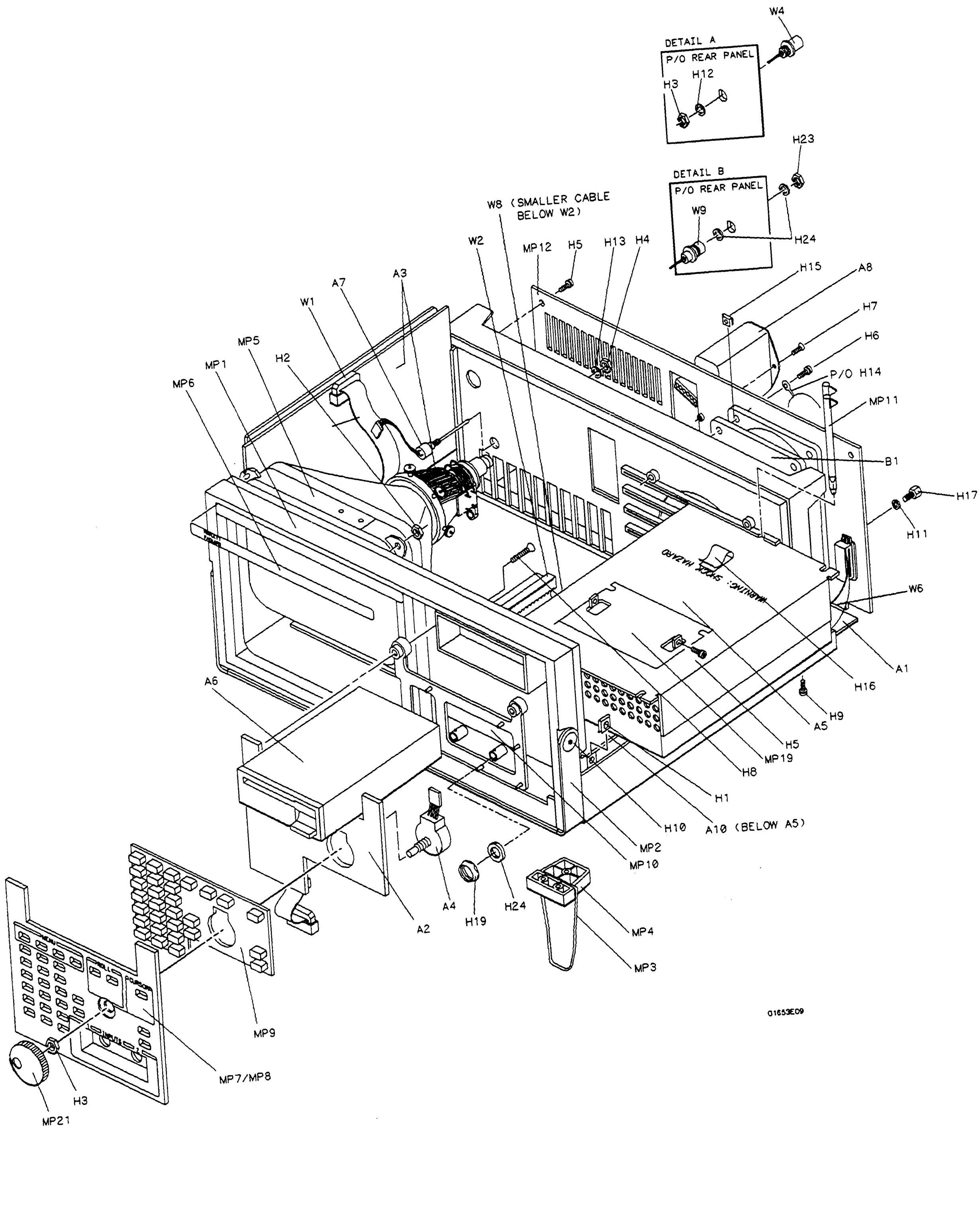


Figure 5-1. HP 1652B/1653B Exploded View

Table 5-1. Reference Designator and Abbreviations

REFERENCE DESIGNATOR

A	= assembly
B	= fan;motor
BT	= battery
C	= capacitor
CR	= diode;diode thyristor; varactor
DL	= delay line
DS	= annunciator;lamp;LED
E	= misc. electrical part
F	= fuse
FL	= filter
H	= hardware
J	= electrical connector (stationary portion);jack
L	= coil;inductor
MP	= misc. mechanical part
P	= electrical connector (moveable portion);plug
Q	= transistor;SCR; triode thyristor
R	= resistor
RT	= thermistor
S	= switch;jumper
T	= transformer
TB	= terminal board
TP	= test point
U	= integrated circuit; microcircuit
V	= electron tube; glow lamp
VR	= voltage regulator; breakdown diode
W	= cable
X	= socket
Y	= crystal unit(piezo-electric or quartz)

ABBREVIATIONS

A	= amperes
A/D	= analog-to-digital
AC	= alternating current
ADJ	= adjust(ment)
AL	= aluminum
AMPL	= amplifier
ANLG	= analog
ANSI	= American National Standards Institute
ASSY	= assembly
ASTIG	= astigmatism
ASYNCHRO	= asynchronous
ATTEN	= attenuator
AWG	= American wire gauge
BAL	= balance
BCD	= binary-code decimal
BD	= board
BFR	= buffer
BIN	= binary
BRDG	= bridge
BSHG	= bushing
BW	= bandwidth
C	= ceramic;cermet (resistor)
CAL	= calibrate;calibration
CC	= carbon composition
CCW	= counterclockwise
CER	= ceramic
CFM	= cubic feet/minute
CH	= choke
CHAM	= chamfered
CHAN	= channel
CHAR	= character
CM	= centimeter
CMOS	= complementary metal-oxide-semiconductor
CMR	= common mode rejection
CNDCT	= conductor
CNTR	= counter
CON	= connector
CONT	= contact
CRT	= cathode-ray tube
CW	= clockwise
D	= diameter
D/A	= digital-to-analog
DAC	= digital-to-analog converter
DARL	= darlington
DAT	= data
DBL	= double
DBM	= decibel referenced to 1mW
DC	= direct current
DCDR	= decoder
DEG	= degree
DEMUX	= demultiplexer
DET	= detector
DIA	= diameter
DIP	= dual in-line package
DIV	= division
DMA	= direct memory access
DPDT	= double-pole, double-throw
DRC	= DAC refresh controller
DRVR	= driver
DWL	= dowel
ECL	= emitter coupled logic
ELAS	= elastomeric
EXT	= external
F	= farads;metal film (resistor)
FC	= carbon film/ composition
FD	= feed
FEM	= female
FF	= flip-flop
FL	= flat
FM	= foam;from
FR	= front
FT	= gain bandwidth product
FW	= full wave
FXD	= fixed
GEN	= generator
GND	= ground(ed)
GP	= general purpose
GRAT	= graticule
GRV	= groove
H	= henries;high
HD	= hardware
HDND	= hardened
HG	= mercury
HGT	= height
HLCL	= helical
HORIZ	= horizontal
HP	= Hewlett-Packard
HP-IB	= Hewlett-Packard Interface Bus
HR	= hour(s)
HV	= high voltage
HZ	= Hertz
I/O	= input/output
IC	= integrated circuit
ID	= inside diameter
IN	= inch
INCL	= include(s)
INCAND	= incandescent
INP	= input
INTEN	= intensity
INTL	= internal
INV	= inverter
JFET	= junction field-effect transistor
JKT	= jacket
K	= kilo(103)
L	= low
LB	= pound
LCH	= latch
LCL	= local
LED	= light-emitting diode
LG	= long
LI	= lithium
LK	= lock
LKWR	= lockwasher
LS	= low power Schottky
LV	= low voltage
M	= mega(106);megohms; meter(distance)
MACH	= machine
MAX	= maximum
MFR	= manufacturer
MICPROC	= microprocessor
MINTR	= miniature
MISC	= miscellaneous
MLD	= molded
MM	= millimeter
MO	= metal oxide
MTG	= mounting
MTLC	= metallic
MUX	= multiplexer
MW	= milliwatt
N	= nano(10-9)
NC	= no connection
NMOS	= n-channel metal-oxide-semiconductor
NPN	= negative-positive-negative
NPRN	= neoprene
NRFR	= not recommended for field replacement
NSR	= not separately replaceable
NUM	= numeric
OBD	= order by description
OCTL	= octal
OD	= outside diameter
OP AMP	= operational amplifier
OSC	= oscillator
P	= plastic
P/O	= part of
PC	= printed circuit
PCB	= printed circuit board
PD	= power dissipation
PF	= picofarads
PI	= plug in
PL	= plate(d)
PLA	= programmable logic array
PLST	= plastic
PNP	= positive-negative-positive
POLYE	= polyester
POS	= positive;position
POT	= potentiometer
POZI	= pozidrive
PP	= peak-to-peak
PPM	= parts per million
PRCN	= precision
PREAMP	= preamplifier
PRGMBL	= programmable
PRL	= parallel
PROG	= programmable
PSTN	= position
PT	= point
PW	= potted wirewound
PWR	= power
R-S	= reset-set
RAM	= random-access memory
RECT	= rectifier
RET	= retainer
RF	= radio frequency
RGLTR	= regulator
RGTR	= register
RK	= rack
RMS	= root-mean-square
RND	= Round
ROM	= read-only memory
RPG	= rotary pulse generator
RX	= receiver
S	= Schottky-clamped; seconds(time)
SCR	= screw;silicon controlled rectifier
SEC	= second(time);secondary
SEG	= segment
SEL	= selector
SGL	= single
SHF	= shift
SI	= silicon
SIP	= single in-line package
SKT	= skirt
SL	= slide
SLDR	= solder
SLT	= slot(ted)
SOLD	= solenoid
SPCL	= special
SQ	= square
SREG	= shift register
SRQ	= service request
STAT	= static
STD	= standard
SYNCHRO	= synchronous
TA	= tantalum
TBAX	= tubeaxial
TC	= temperature coefficient
TD	= time delay
THD	= thread(ed)
THK	= thick
THRU	= through
TP	= test point
TPG	= tapping
TPL	= triple
TRANS	= transformer
TRIG	= trigger(ed)
TRMR	= trimmmer
TRN	= turn(s)
TTL	= transistor-transistor
TX	= transmitter
U	= micro(10-6)
UL	= Underwriters Laboratory
UNREG	= unregulated
VA	= voltampere
VAC	= volt,ac
VAR	= variable
VCO	= voltage-controlled oscillator
VDC	= volt,dc
VERT	= vertical
VF	= voltage,filtered
VS	= versus
W	= watts
W/	= with
W/O	= without
WW	= wirewound
XSTR	= transistor
ZNR	= zener
oC	= degree Celsius (Centigrade)
oF	= degree Fahrenheit
oK	= degree Kelvin

Table 5-2. Replaceable Parts List

Reference Designator	HP Part Number	CD	Qty	Description	Mfr Code	Mfr Part Number
A1	01652-66501	0	1	SYSTEM BOARD ASSEMBLY – 80 CHANNEL (1652B)	28480	01652-66501
A1	01653-66501	1	1	SYSTEM BOARD ASSEMBLY – 32 CHANNEL (1653B)	28480	01653-66501
A2	01652-66503	2	1	KEYBOARD CIRCUIT BOARD ASSEMBLY	28480	01652-66503
A3	2090-0204	9	1	MONITOR ASSEMBLY	28480	2090-0204
A4	0960-0753	6	1	ROTARY PULSE GENERATOR	28480	0960-0753
A5	0950-1879	8	1	POWER SUPPLY ASSEMBLY	28480	0950-1879
A6	0950-1798	0	1	DISK DRIVE ASSEMBLY	28480	0950-1798
A7	01650-61614	4	1	INTENSITY ADJUSTMENT ASSEMBLY	28480	01650-61614
A8	9135-0325	8	1	LINE FILTER SWITCH ASSEMBLY	28480	9135-0325
A9	01650-61608	6	5	PROBE TIP ASSEMBLY (1652B)	28480	01650-61608
A9	01650-61608	6	2	PROBE TIP ASSEMBLY (1653B)	28480	01650-61608
A10	01652-66502	1	1	OSCILLOSCOPE BOARD ASSEMBLY - 2 CHANNEL	28480	01652-66502
A11	54503-63401	4	2	ATTENUATOR ASSEMBLY	28480	54503-63401
A12	10430A	8	2	PROBE 500 MHZ 1M 10:1	28480	10430A
B1	3160-0429	0	1	FAN-TUBEAXIAL 100-CFM 12VDC	28480	3160-0429
E1	5959-9333	8	0	REPLACEMENT PROBE LEADS (PKG OF 5)	28480	5959-9333
E2	5959-9334	9	0	REPLACEMENT PROBE GROUNDS (PGK OF 5)	28480	5959-9334
E3	5959-9335	0	0	REPLACEMENT POD GROUNDS (PKG OF 5)	28480	5959-9335
E4	5959-0288	4	5	GRABBER ASSEMBLY SET – 20 (1652B)	28480	5959-0288
E4	5959-0288	4	2	GRABBER ASSEMBLY SET-20 (1653B)	28480	5959-0288
F1	2110-0003	0	1	FUSE 3A 250V NTD FE UL	28480	2110-0003
H1	0535-0113	1	10	NUT "U"-TP M3 X 0.500.3MM-THK (TOP COVER)	28480	0535-0113
H2	0535-0056	1	4	NUT-HEX PRVLG-TRQ M4 X 0.7 5MM-THK (CRT)	28480	0535-0056
H3	2950-0001	8	3	NUT-DBL-CHAM 3/8-32-THD0.094-IN-THK (RPG, BNC)	28480	2950-0001
H4	2950-0072	3	1	NUT-DBL-CHAM 1/4-32-THD0.062-IN-THK (INTEN ADJ)	28480	2950-0072
H5	0515-0372	2	12	SCREW M3 X 0.5 8MM-LG (DISK DRIVE, REAR PANEL)	28480	0515-0372
H6	0515-0821	6	4	SCREW- M3.5 X 0.6 (FAN)	28480	0515-0821
H7	0515-1035	6	22	SCREW-M3 X 0.5 8MM- (FEET, LINE FIL, TOP COVER)	28480	0515-1035
H8	0515-1135	7	4	SCREW- M3 X 0.5 25MM-LG (KEYPAD)	28480	0515-1135
H9	0515-1951	5	8	SCREW-TAPPING M4.2 (SYSTEM BOARD)	28480	0515-1951
H10	01650-82401	1	2	M5 SHOULDER SCREW (HANDLE)	28480	01650-82401
H11	2190-0009	4	2	WASHER-LK INTL T NO. 80.168-IN-ID (HP-IB CABLE)	28480	2190-0009
H12	2190-0016	3	2	WASHER-LK INTL T 3/8 IN0.377-IN-ID (BNC)	28480	2190-0016
H13	2190-0027	6	1	WASHER-LK INTL T 1/4 IN0.256-IN-ID (INTEN ADJ)	28480	2190-0027
H14	3160-0092	3	1	FAN GUARD	28480	3160-0092
H15	0590-1868	1	4	FAN MOUNTING CLIP	28480	0590-1868
H16	1400-0611	0	1	CLAMP-FL-CA 1-WD (DISK DRIVE CABLE)	28480	1400-0611
H17	0380-1482	5	2	HEX STANDOFF .0340 (HP-IB CABLE)	28480	0380-1482
H18	01650-00203	3	2	NUT PLATE (HANDLE)	28480	01650-00203
H19	2950-0054	1	2	NUT 1/2 - 28 .125 (ATTENUATOR BNC)	28480	2950-0054
H20	3050-0893	9	2	WASHER - FLAT (PC BOARD BRACKET)	28480	3050-0893

Table 5-2. Replaceable Parts List (continued)

Reference Designator	HP Part Number	CD	Qty	Description	Mfr Code	Mfr Part Number
H21	0515-0374	4	6	SCREW-MACH M3.0 X 0.50 (OSCILLOSCOPE BOARD)	28480	0515-0374
H22	0515-1246	1	4	SCREW-MACH M3 X 0.5 (ATTENUATOR)	28480	0515-1246
H23	2950-0035	8	1	NUT .468-32 .078 (PROBE COMPENSATION BNC)	28480	2950-0035
H24	2190-0068	5	3	WASHER-LK .505 .630 .02 (ATTENUATOR)	28480	2190-0068
MP1	01650-45207	7	1	CABINET MOLDED PLASTIC	28480	01650-45207
MP2	01650-04901	2	1	BALE HANDLE	28480	01650-04901
MP3	1460-1345	5	2	TILT STAND SST	28480	1460-1345
MP4	01650-47701	0	2	MOLDED FOOT	28480	01650-47701
MP5	01650-01202	0	1	GROUND BRACKET	28480	01650-01202
MP6	01652-94302	8	1	IDENTIFICATION LABEL (1652B)	28480	01652-94302
MP6	01653-94302	9	1	IDENTIFICATION LABEL (1653B)	28480	01653-94302
MP7	01652-94301	7	1	KEYBOARD LABEL	28480	01652-94301
MP8	01652-40501	6	1	KEYBOARD HOUSING	28480	01652-40501
MP9	01652-41901	2	1	ELASTOMERIC KEYPAD	28480	01652-41901
MP10	01652-40502	7	1	KEYBOARD SPACER	28480	01652-40502
MP11	01650-46101	2	2	LOCKING PIN PCB	28480	01650-46101
MP12	01650-00205	1	1	REAR PANEL (1652B)	28480	01650-00205
MP12	01651-00203	0	1	REAR PANEL (1653B)	28480	01651-00203
MP13	7120-4835	0	1	CSA CERTIFICATION LABEL	28480	7120-4835
MP14	01650-04101	4	1	TOP COVER	28480	01650-04101
MP15	01650-84501	6	1	ACCESSORY POUCH	28480	01650-84501
MP16	01650-94303	7	1	PROBE LABELS	28480	01650-94303
MP17	01650-29101	6	5	GROUND SPRING (SYSTEM BOARD 1652B)	28480	01650-29101
MP17	01650-29101	6	2	GROUND SPRING (SYSTEM BOARD 1653B)	28480	01650-29101
MP18	01650-29102	7	1	CLIP RS-232 ESD	28480	01650-29102
MP19	01650-25401	1	1	DISK INSULATOR	28480	01650-25401
MP20	01650-63202	0	1	RS-232 LOOPBACK CONNECTOR	28480	01650-63202
MP21	01650-47401	7	1	RPG KNOB	28480	01650-47401
MP22	01652-01201	1	1	BRACKET - PC BD (OSCILLOSCOPE BOARD)	28480	01652-01201
MP23	01652-01202	2	1	PLATE - PC BD (OSCILLOSCOPE BOARD)	28480	01652-01202
MP24	01652-94303	9	1	PROBE COMPENSATION LABEL	28480	01652-94303
W1	01650-61601	9	1	SWEEP CABLE	28480	01650-61601
W2	54503-61606	7	1	POWER SUPPLY CABLE	28480	54503-61606
W3	01650-61604	2	1	DISK CABLE	28480	01650-61604
W4	01650-61605	3	2	BNC CABLE	28480	01650-61605
W5	01650-61607	5	5	PROBE CABLE (1652B)	28480	01650-61607
W5	01650-61607	5	2	PROBE CABLE (1653B)	28480	01650-61607
W6	01650-61613	3	1	HP-IB CABLE	28480	01650-61613
W7	01650-61616	6	1	FAN CABLE	28480	01650-61616
W8	01652-61602	2	1	OSCILLOSCOPE CABLE ASSY 60 COND	28480	01652-61602
W9	54100-61610	6	1	PROBE COMPENSATION BNC	28480	54100-61610

Contents

Section 6A:

Theory of Operation

6A

Theory of Operation

Introduction

This section provides the theory of operation of the HP 1652B/1653B Logic Analyzer. The theory of operation is included for information only and is not intended for troubleshooting purposes.

Safety

Read the Safety Summary at the front of this manual before servicing the instrument. Before performing any procedure, review it for cautions and warnings.

Warning

Maintenance should be performed by trained service personnel aware of the hazards involved (for example, fire and electric shock). When maintenance can be performed without power applied, the power cord should be removed from the instrument.

Block Level Theory

The HP 1652B is an 80 channel state and timing logic analyzer with a 2 channel, 100 MHz, 400 Msample/s digitizing oscilloscope. The HP 1653B is a 32 channel state and timing logic analyzer with a 2 channel, 100 MHz, 400 Msample/s digitizing oscilloscope. The human interface is a front-panel keypad and knob for instrument control and a 9-inch (diagonal) white phosphor CRT for information display. Available on the rear panel are RS-232-C and HP-IB ports for communication to a printer or from a controller. Also on the rear panel are two BNCs for input or output of an external trigger and a BNC for oscilloscope probe compensation.

The instrument is built around the 68000 microprocessor and powerful data acquisition ICs that probe, shape, store, and analyze data from a target system. An acquisition interface to the 68000 makes the data acquisition system fully compatible with the architecture of the 68000 microprocessor. The System Assembly Board contains the necessary circuitry to interface the keypad, CRT monitor, disk drive, RS-232-C, and HP-IB ports.

Figures 6A-1 and 6A-2 show a simplified block diagram of the instrument.

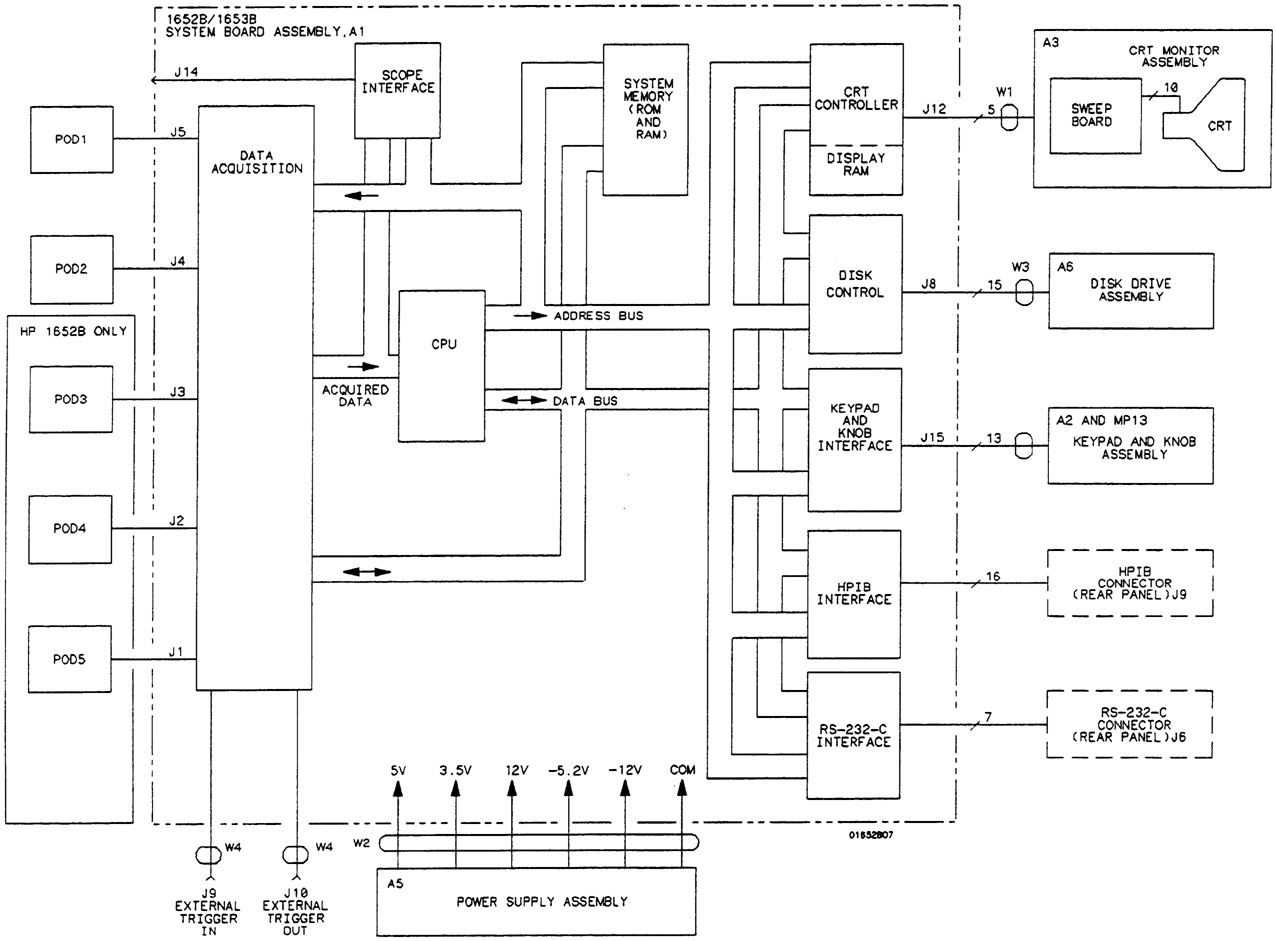


Figure 6A-1. System Board Assembly Block Diagram

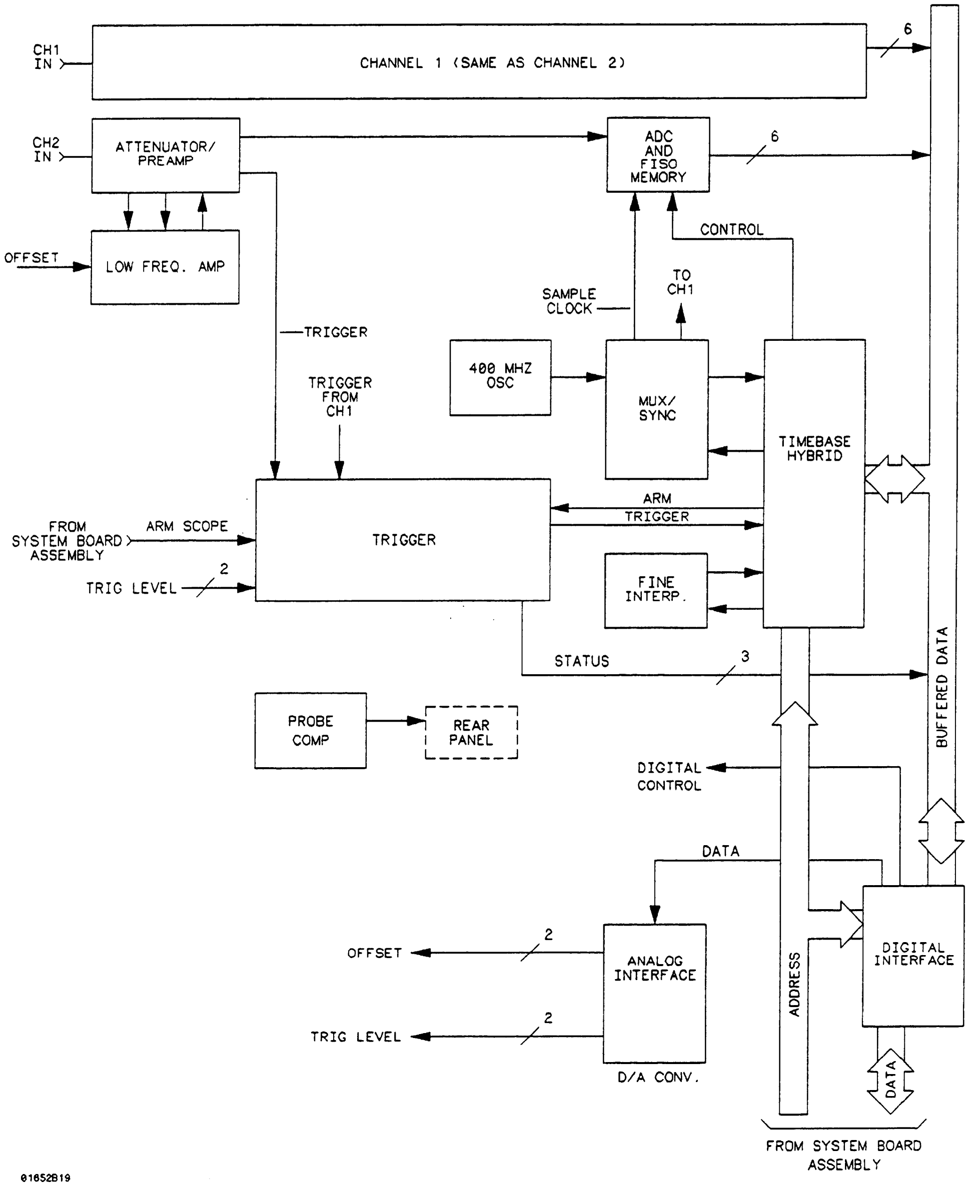


Figure 6A-2. Oscilloscope Assembly Block Diagram

Power Supply Assembly

The switching power supply provides 120 W (200 W maximum) for the instrument. The ac input to the power supply is 115V or 230 V, -25% to +15%. Maximum input power is 350 VA maximum. The ac input frequency is 48 to 66 Hz.

All voltages necessary to operate the instrument are applied first to the Main Assembly. Unfiltered voltages of + 15V, +12V, -12V, +5.15V, -5.2V, and +3.5V are supplied to the board where they are then filtered and distributed throughout the main assembly board, oscilloscope board, and to the CRT Monitor Assembly. Filtered voltages of approximately +5 V and +12 V are routed through the Main Assembly to the CRT Monitor Assembly. The +5.15 V supply is adjustable on the supply.

CRT Monitor Assembly

The CRT Monitor Assembly consists of the sweep board circuitry, a 9-inch white phosphor CRT, and the CRT yoke. The assembly requires + 5 V and +12 V from the power supply via the Main Assembly.

The non-interlacing raster display is controlled by the CPU portion of the Main Assembly. System control provides synchronization and pixel information.

Main Assembly

The Main Assembly contains the logic analyzer acquisition system and system control circuitry. It also provides interfaces for the Power Supply Assembly, CRT Monitor Assembly, keyboard, RS-232C, and HP-IB. The input to the Main Assembly is from any or all of the data acquisition pods, which exit the rear panel. The user interface is from the front-panel keyboard or with a controller via the HP-IB or RS-232C connector on the rear panel. A more detailed theory of the logic analyzer circuitry follows block level theory.

Central Processing Unit (CPU)

The CPU is a 68000 P10 microprocessor with addressing capability of 16 megabytes (23 address lines/16 data lines). The CPU receives its clock (10 MHz) from the TCL (Timing Control Logic). The TCL circuitry consists of programmable array logic (PALs), various logic gates, and miscellaneous circuitry for arbitrating between display and refresh requests of display and system RAM. The PALs and arbitration circuitry are synchronized with a 20 MHz clock. The rest of the circuitry is asynchronous. The signals generated by the TCL provide all timing for the CPU. The CPU drives the read/write line and the address and data strobes.

The CPU supplies a 2.5 MHz enable clock for synchronization with the CRT Controller (CRTC).

Oscilloscope Assembly

The Oscilloscope Assembly contains the oscilloscope acquisition system. The analog input to the Oscilloscope Assembly is from either or both of two channels, located at the front-panel BNCs. The user interface is from the front-panel keyboard or with a controller via the HP-IB or RS-232C connector on the rear panel. A more detailed theory of the Oscilloscope Assembly follows the block level theory.

Keypad and Knob Assembly

The front-panel keypad is elastomeric and has 29 keys. The keyboard rows are continually scanned at a frequency of 60 Hz. When a key is pressed the signal is sent as data to the 68000 which determines the key pressed and its function. The Rotary Pulse Generator (RPG) is connected to the front-panel knob and supplies pulses to the 68000 microprocessor when the knob is turned. The RPG is used for cursor movement and value entry.

Disk Controller

The disk controller performs the necessary functions for reading or writing data to the built-in disk drive of the logic analyzer. The disk controller interface to the 68000 is an 8-bit bidirectional bus for data, status, and control word transfers.

The built-in disk drive is a 3.5-inch double-sided Sony disk drive. The main features of the disk drive are low power consumption, low height, and high reliability with simple mechanism and electronic circuitry.

RS-232-C Interface

The controlling IC of the RS-232-C Interface is a Signetics SCN2661 Enhanced Programmable Communications Interface (EPCI) which is a universal synchronous/asynchronous receiver/transmitter (USART) data communications IC.

The SCN2661 serializes parallel data from the 68000 for transmission. At the same time, it also receives serial data and converts it to parallel data characters for the 68000.

The SCN2661 IC contains a baud rate generator which can be programmed from the logic analyzer I/O menu for one of eight baud rates. Protocol, word length, stop bits length, and parity are also programmed via the I/O menu.

Two additional ICs, the DS14C88 and DS14C89, are line drivers/receivers used by the SCN2661 IC for interface of terminal equipment with data communications equipment. Slew rate control is provided on the ICs, eliminating the need for external capacitors.

HP-IB Interface

The HP-IB controller provides an interface between the microprocessor system and the HP-IB in accordance with IEEE 488 standards. An 8-bit data buffer and 8-bit control line buffer interface the HP-IB controller to the HP-IB bus.

The HP-IB is a 24 conductor shielded cable carrying 8 data lines, 8 control lines, 7 system grounds, and 1 chassis ground.

Logic Analyzer Theory of Operation

The HP 1652B/1653B logic analyzer operation is based around a 68000 microprocessor and proprietary acquisition ICs. Input data is captured by passive probing, reshaped, and stored into memory.

Data Acquisition

The data acquisition for the logic analyzer consists of the data acquisition pods, acquisition ICs, and the interface to the 68000. The interface to the target system is through any of the data acquisition pods. There are five pods available on the HP 1652B (80 channels) and two pods available on the HP 1653B (32 channels). Each pod contains 16 input data probes and one external clock input for state mode measurements. The data probes can be used for state or timing measurements.

Each pod consists of a probe tip assembly and a 4.5 foot woven cable. A probe tip assembly includes 17 twelve-inch probes and a common ground return. This is connected to one end of the cable. The other end of the woven cable terminates at the rear panel of the logic analyzer. The woven cable consists of 17 signal lines, 17 signal return lines, 2 chassis grounds, and 2 power supply lines. All are woven together with polyarmid yarn.

Each probe input has an input impedance of 100k ohms in parallel with approximately 8 pF. The probes can be grounded in two ways: with a common pod ground for state measurements, or with a probe tip ground for higher frequency measurements.

The input signals are attenuated by a factor of 10 in the passive probe. The signals are applied to a comparator where they are compared against a voltage threshold to determine if the voltage level is above or below the threshold level. The comparator then shapes the single-ended signal and outputs it at an ECL level to the acquisition IC. The input data is then stored at the acquisition IC.

Arming Control

The two BNCs on the rear panel are used for arming control of the logic analyzer acquisition ICs. An arm signal may be output from the ICs to the rear panel EXTERNAL TRIGGER OUT (J10), or input to the ICs from EXTERNAL TRIGGER IN (J9).

Memory

The memory of the logic analyzer consists of three separate memories: one ROM and two RAMs. The system (EP)ROM is 32 K long by 16 wide and is used primarily for booting-up the system and self-test storage. The system (D)RAM is 512 K long by 16 wide and contains the operating system and the acquired data from the target system. Since the RAM is a volatile memory, the operating system is loaded at each power-up of the instrument via the built-in disk drive and a mini floppy disk.

The display (D)RAM is 64K long by 4 wide and is cycle shared between the 68000 and the display refresh circuitry. This is why the display bus is separate from the local bus. The two buses are separated by a set of address multiplexers and data buffers.

Oscilloscope Theory of Operation

The oscilloscope circuitry provides the conditioning, sampling, digitizing, and storage of the signals at the channel input connectors. The channels are identical. The trigger circuitry input can be selected between the oscilloscope channels and the logic analyzer. A 400 MHz oscillator, with the time base and mux/sync (multiplexer synchronizer), provides the sample clocking. After conditioning, the signals are digitized and stored in a hybrid IC containing both the ADC and memory. The signal data is then transferred over the data bus where it is processed for display.

Attenuator/ Preamps

The channel signals are conditioned by the attenuator/preamps, thick film hybrids containing passive attenuators, impedance converters, and a programmable amplifier. The channel sensitivity defaults to the standard 1-2-5 sequence (other sensitivities can be set also). However, the firmware uses passive attenuation of 1, 5, 25, and 125, with the programmable preamp, to cover the entire sensitivity range.

The input has a selectable 1 MΩ or 50 Ω input impedance. Compensation for the passive attenuators is laser trimmed and not adjustable. After the passive attenuators, the signal is split into high-frequency and low-frequency components. Low frequency components are amplified on the PC board where they are combined with the offset voltage.

The high- and low-frequency components of the signal are recombined and applied to the input FET of the preamp. The FET provides a high input impedance for the preamp. The programmable preamp adjusts the gain to suit the required sensitivity and provides two output signals. One signal is the same polarity as the input and goes to the trigger circuitry. The other is of the opposite polarity and is sent to the post amplifier.

Post Amplifier

The post amplifier conditions the signal for the ADC. It has a gain of approximately 2.5 and it has one compensation capacitor adjustment per channel. This adjustment effects the transition rise time and overshoot.

ADC and FISO Memory

A single hybrid digitizes and stores the channel signal. Digitization is done by a set of comparators in a flash converter. A precision voltage divider within the ADC provides a separate reference for each comparator. This voltage divider is controlled by a reference supply and amplifier on the PC board.

The FISO (fast in, slow out) memory is 2 k by 6-bit bytes. Sample clocks are provided by the time base circuitry. At 500 ns/div and faster, the sample clock is 400 MHz. At sweep speeds of 1 us and slower, the sample clocks range from 200 MHz to 25 Hz. The FISO data is buffered onto the CPU data bus for further processing.

Triggering

The trigger circuitry accepts inputs from both oscilloscope channels, the logic analyzer, and the time base. Only one of these signals is multiplexed into the trigger circuitry, depending on the trigger mode. For example, the input from the time base is used while the instrument is in the "trigger immediate" mode.

When in the "edge trigger" mode, the preamp outputs are fed through a high speed voltage comparator using a reference voltage from the DAC.

The trigger circuitry output drives the time base and the logic analyzer arming input. This output and internal status signals are interfaced to the data bus for software processing purposes.

Time Base

The time base provides the sample clocks and timing necessary for data acquisition. It consists of the 400 MHz reference oscillator, mux/sync hybrid (multiplexer/synchronizer), and time base IC.

The mux/sync hybrid provides sample clocks to the ADC. At sample rates of 400 MHz and 200 MHz, this sample clock is derived from the 400 MHz reference oscillator. At 100 MHz and slower, the sample clock comes from the time base IC. The mux/sync hybrid synchronizes the gating of the sample clock to provide only full sample clocks.

The time base hybrid has programmable dividers to provide the rest of the sample frequencies appropriate for the time range selected. It uses the time-stretched output of the fine interpolator to time-reference the sampling to the trigger point. It has counters to control how much data is taken before (pre-trigger data) and after (post-trigger data) the trigger event. After the desired number of pre-trigger samples has occurred, the time base hybrid sends a signal to the Logic Trigger (trigger arm) indicating it is ready for the trigger event. When the trigger condition is satisfied, the Logic Trigger sends a signal back to the time base hybrid. The time base hybrid then starts the post-trigger delay counter. When the countdown reaches zero, the sample clocks are stopped and the CPU is signaled that the acquisition is complete.

Fine Interpolator

The Fine Interpolator is a dual-slope integrator that acts as a time-interval stretcher. When the trigger circuitry receives a signal that meets the programmed triggering requirements, it signals the time base. The time base then sends a pulse to the fine interpolator. The pulse is equal in width to the time between the trigger and the next sample clock. The fine interpolator stretches this time by a factor of approximately 375. Meanwhile, the time base hybrid runs a counter with a clock derived from the sample rate oscillator. When the interpolator indicates the stretch is complete, the counter is stopped. The count represents, with much higher accuracy, the time between the trigger and the first sample clock. The count is stored and used to place the recently acquired data in relationship with previous data.

Probe Compensation

An oscillator generates a 1.25 kHz square wave with fast edges for oscilloscope probe compensation. The oscillator's levels range from approximately -400 mV to -900 mV.

Digital Interface

The Digital Interface provides control and interface between the system control and digital functions in the acquisition circuitry.

Analog Interface

The Analog Interface provides control of analog functions in the acquisition circuitry. It is primarily a 16 channel DAC with an accurate reference, and filters on the outputs. It controls channel offsets and trigger levels.

Contents

Section 6B:

Self Tests

6B

Self Tests

Introduction

This section provides information on the power-up self tests and extended self tests of the HP 1652B/1653B. All of these self tests may be performed without access to the interior of the instrument.

Power-Up Self Tests

The power-up self tests are automatically performed upon applying power to the instrument. The revision number of the operating system is given in the upper right corner of the screen during the power-up self tests. As each test is completed, either "passed" or "failed" is printed in front of the name of each test in the following manner:

PERFORMING POWER-UP SELF-TESTS

passed	**ROM test**
passed	**RAM test**
passed	**Interrupt test**
passed	**Display test**
passed	**Keyboard test**
passed	**Acquisition test**
passed	**Threshold test**
passed	**Disk test**

LOADING SYSTEM FILE

As indicated by the last message, the HP 1652B/1653B logic analyzer will automatically load from the operating system disk in the disk drive. If the operating system disk is not in the disk drive, the message "SYSTEM DISK NOT FOUND" will be displayed at the bottom of the screen and "NO DISK" will be displayed in front of the disk test in place of "passed."

Note

If the message "SYSTEM DISK NOT FOUND" appears on screen, insert the operating system disk into the disk drive and press any front panel key.

Selectable Self Tests

Selectable self tests are used as troubleshooting aids. Eight self tests may be invoked via the Self Tests menu:

- Analyzer Data Acquisition
- Scope Data Acquisition
- RS-232-C
- BNC
- Keyboard
- RAM
- ROM
- Disk Drive
- Cycle through tests

The required test is selected by moving the cursor to the test and pressing the front panel SELECT key. A pop-up menu will appear with a description of the test to be performed. The self test does not begin until the cursor is placed on Single test, Repetitive test, or Execute and the front panel SELECT key is pressed.

The repetitive self tests display the number of "runs" and "failures" for the selected test. Press STOP to discontinue the test.

After the test is completed, either "Passed,""Failed," or "Tested" will be displayed on the Self Tests menu in front of the test.

Selecting the Self Tests Menu

The self tests may be invoked from any menu by pressing the front panel I/O key. The pop-up I/O menu appears on screen with the following choices:

- Done
- Print Screen
- Print All
- Disk Operations
- I/O Port Configuration
- External BNC Configuration
- Self Tests

1. Move the cursor to **Self Tests** with the front panel knob and press SELECT.

Caution

The self tests are loaded from the Performance Verification disk. The process of running the self tests destroys the current configuration and data.

---

2. Insert the Performance Verification disk (or copy of it) into the disk drive.

3. Move the cursor to the **Start self test** field with the front panel knob and press SELECT. After loading the self tests, the HP 1652B/1653B Self Tests menu will display the following:

Untested	*	**Analyzer Data Acquisition**
Untested	*	**Scope Data Acquisition**
Untested	*	**RS-232-C**
Untested	*	**BNC**
Untested	*	**Keyboard**
Untested	*	**RAM**
Untested	*	**ROM**
Untested	*	**Disk Drive**
	*	**Cycle through tests**

4. To select a self test, move the cursor to the appropriate test with the front panel knob and press SELECT. To leave the HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT. The HP 1652B/1653B will reload the operating system from the disk and display the default System Configuration menu.

Note

The operating system disk (or copy of it) must be in disk drive to reload the operating system after leaving the Self Tests menu.

Analyzer Data Acquisition Self Test

The Analyzer Data Acquisition self test verifies the functionality of key elements of the internal acquisition system.

1. In HP 1652B/1653B Self Tests menu, move the cursor to **Analyzer Data Acquisition** and press SELECT. A menu will appear with a description of the test, the number of "runs" and "failures" for the selected test, and fields to select Single test, Repetitive test, or Done.

2. Move the cursor to **Single test** or **Repetitive test** and press SELECT. The message "Running Data Acquisition Test" appears on screen while the instrument is performing the test. When the test is finished, the message "Data Acquisition Test complete" will appear on screen.

3. If you are running repetitive tests, press the front-panel STOP key when you want to discontinue the test. The number of runs and failures will be displayed in the menu.

4. To return to HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT.

Scope Data Acquisition Self Test

The Scope Data Acquisition self test verifies the functionality of key elements of the internal acquisition system. These key elements include the following:

- Scope Memory
- Scope Pretrigger Delay
- Scope Trigger
- Scope Sample Rate
- Scope Preamp
- Scope Interpolator

1. In HP 1652B/1653B Self Tests menu, move the cursor to **Scope Data Acquisition** and press SELECT. A menu will appear with a description of the test, the number of "runs" and "failures" for the selected test, and fields to select Single test, Repetitive test, or Done.
2. Move the cursor to **Single test** or **Repetitive test** and press SELECT. The message "Running Scope Data Acquisition Test" appears on screen while the instrument is performing the test. When the test is finished, the message "Scope Data Acquisition Test complete" will appear on screen.
3. If you are running repetitive tests, press the front-panel STOP key when you want to discontinue the test. The number of runs and failures will be displayed in the menu.
4. To return to HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT.

RS-232-C Self Test

The RS-232-C self test verifies the functionality of the RS-232-C driver and continuity of the RS-232-C data paths.

Equipment Required:

RS-232-C Loopback Connector 01650-63202

Procedure:

1. In HP 1652B/1653B Self Tests menu, move the cursor to **RS-232-C** and press SELECT. A menu will appear with a description of the test, the number of "runs" and "failures" for the selected test, and fields to select Single test, Repetitive test, or Done.
2. Connect the RS-232-C loopback connector to the rear-panel RS-232-C receptacle. The message "Running RS-232C Test" appears on screen while the instrument is performing the test. When the test is finished, the message "RS-232C Test complete" will appear on screen.

Note

The RS-232-C loopback connector is an accessory supplied with the HP 1652B/1653B.

---

3. Move the cursor to **Single test** or **Repetitive test** and press SELECT.
4. If you are running repetitive tests, press the front-panel STOP key when you want to discontinue the test. The number of runs and failures will be displayed in the menu.

5. To return to HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT.

BNC Self Test

The BNC self test verifies the functionality of the internal BNC trigger circuitry.

1. In HP 1652B/1653B Self Tests menu, move the cursor to **BNC** and press SELECT. A menu will appear with a description of the test, the number of "runs" and "failures" for the selected test, and fields to select Single test, Repetitive test, or Done.
2. Move the cursor to **Single test** or **Repetitive test** and press SELECT.The message "Running BNC Test" appears on screen while the instrument is performing the test. When the test is finished, the message "BNC Test complete" will appear on screen.
3. If you are running repetitive tests, press the front-panel STOP key when you want to discontinue the test. The number of runs and failures will be displayed in the menu.
4. To return to HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT.

Keyboard Self Test

The Keyboard self test verifies the key closures and knob operation on the front panel system.

1. In HP 1652B/1653B Self Tests menu, move the cursor to **Keyboard** and press SELECT. A menu will appear with a description of the test and fields to Execute the test or exit the menu (Done).
2. Move the cursor to **Execute** and press SELECT. A menu displaying the front-panel keys will appear on screen.
3. Press all of the keys on the front panel keypad and rotate the front panel RPG knob to verify their operation.
4. Press the front-panel STOP key twice to return to the Keyboard Self Test menu.
5. To return to HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT.

RAM Self Test

The RAM self test verifies the operation of system RAM and display RAM.

1. In HP 1652B/1653B Self Tests menu, move the cursor to **RAM** and press SELECT. A menu will appear with a description of the test, the number of "runs" and "failures" for the selected test, and fields to select Single test, Repetitive test, or Done.
2. Move the cursor to **Single test** or **Repetitive test** and press SELECT.The message "Running RAM Test" appears on screen while the instrument is performing the test. When the test is finished, the message "RAM Test complete" will appear on screen.
3. If you are running repetitive tests, press the front-panel STOP key when you want to discontinue the test. The number of runs and failures will be displayed in the menu.

4. To return to HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT.

ROM Self Test

The ROM self test verifies the operation of system ROM.

1. In HP 1652B/1653B Self Tests menu, move the cursor to **ROM** and press SELECT. A menu will appear with a description of the test, the number of "runs" and "failures" for the selected test, and fields to select Single test, Repetitive test, or Done.

2. Move the cursor to **Single test** or **Repetitive test** and press SELECT.The message "Running ROM Test" appears on screen while the instrument is performing the test. When the test is finished, the message "ROM Test complete" will appear on screen.

3. If you are running repetitive tests, press the front-panel STOP key when you want to discontinue the test. The number of runs and failures will be displayed in the menu.

4. To return to HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT.

Disk Drive Self Test

The Disk Drive self test verifies the functionality of the key elements of the internal disk drive system.

1. In HP 1652B/1653B Self Tests menu, move the cursor to **Disk Drive** and press SELECT. A menu will appear with a description of the test, the number of "runs" and "failures" for the selected test, and fields to select Single test, Repetitive test, or Done.

Note

There must be a formatted disk in the disk drive to successfully run the Disk Drive self tests.

2. Move the cursor to **Single test** or **Repetitive test** and press SELECT.The message "Running Disk Test" appears on screen while the instrument is performing the test. When the test is finished, the message "Disk Test complete" will appear on screen.

3. If you are running repetitive tests, press the front-panel STOP key when you want to discontinue the test. The number of runs and failures will be displayed in the menu.

4. To return to HP 1652B/1653B Self Tests menu, move the cursor to **Done** and press SELECT.

Cycle Through Tests

Cycle through tests allows you to cycle through the following tests:

- Analyzer Data Acquisition
- Scope Data Acquisition
- BNC
- RAM
- ROM
- Disk Drive

1. In HP 1652B/1653B Self Tests menu, move the cursor to **Cycle through tests** and press SELECT.

   The tests listed above will run consecutively and continually until the front-panel STOP key is pressed.

2. Press the front-panel STOP key to end the continuous tests.

3. To see the results of the tests for individual tests, move the cursor to the appropriate test and press SELECT. The number of runs and failures of the continuous test will be displayed on the individual self test menu.

4. Move the cursor to **Done** and press SELECT to return to the HP 1652B/1653B Self Tests menu.

Contents

Section 6C:

Troubleshooting

6C

Troubleshooting

Introduction

This section provides troubleshooting information for the HP 1652B/1653B Logic Analyzer. Troubleshooting consists of flowcharts, and signal level tables. The troubleshooting information is provided to isolate a faulty assembly. When a faulty assembly has been located, the disassembly/assembly procedures in section 6D help direct replacement of the assembly.

Self-test descriptions and instructions are provided in section 6B.

Caution

The effects of ELECTROSTATIC DISCHARGE can damage electronic components. Use grounded wriststraps and mats when performing any kind of service to this instrument.

Safety

Read the Safety Summary at the front of this manual before servicing the instrument. Before performing any procedure, review it for cautions and warnings.

Warning

Maintenance should be performed by trained service personnel aware of the hazards involved (for example, fire and electric shock). When maintenance can be performed without power applied, the power cord should be removed from the instrument.

Trouble Isolation Flowcharts

The trouble isolation flowcharts are the troubleshooting guides. Start there when repairing a defective instrument.

The flowcharts refer to other tests, tables, and procedures to help isolate troubles. Disassembly procedures are included in section 6D to direct you in replacing faulty assemblies. The circled numbers on the charts indicate the next chart to use for isolating a problem.

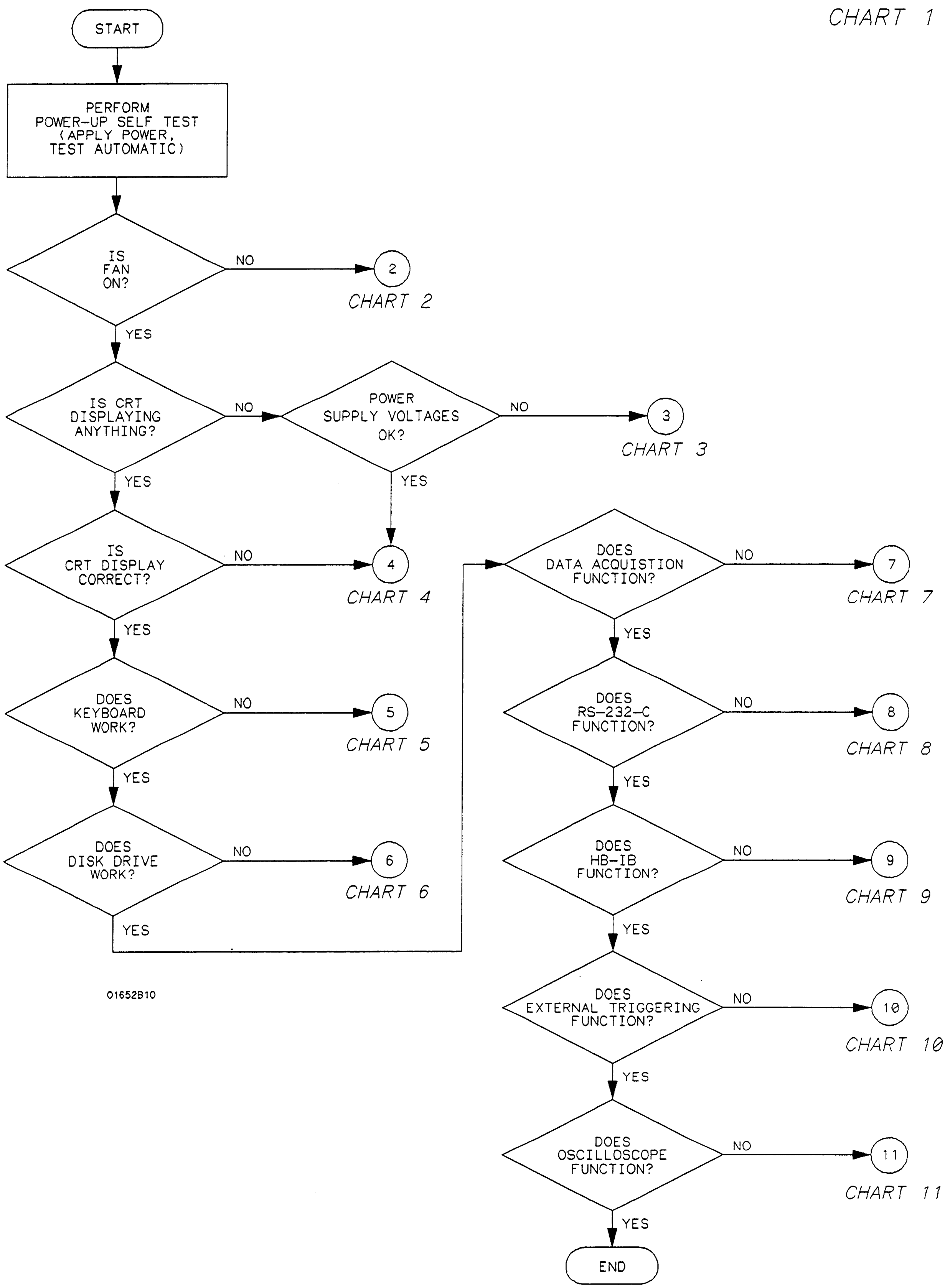


Figure 6C-1. Primary Troubleshooting Flowchart

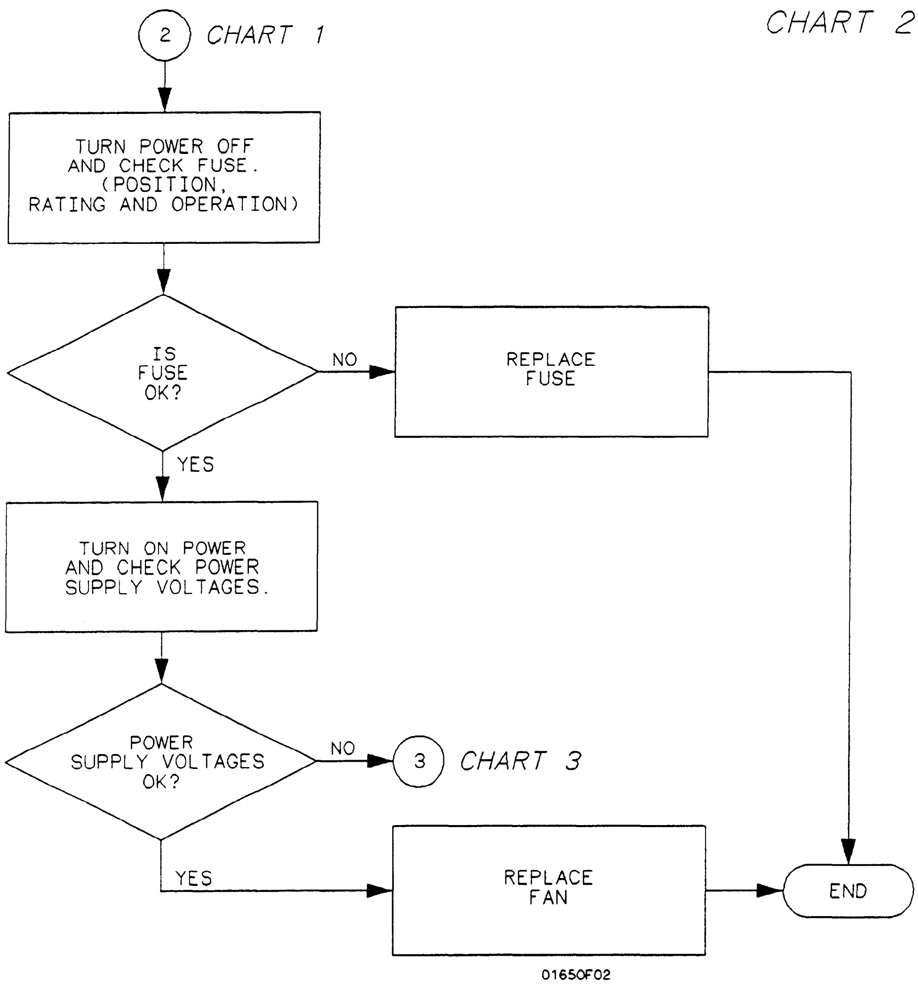


Figure 6C-2. Trouble Isolation Flowchart for Fan/Fuse

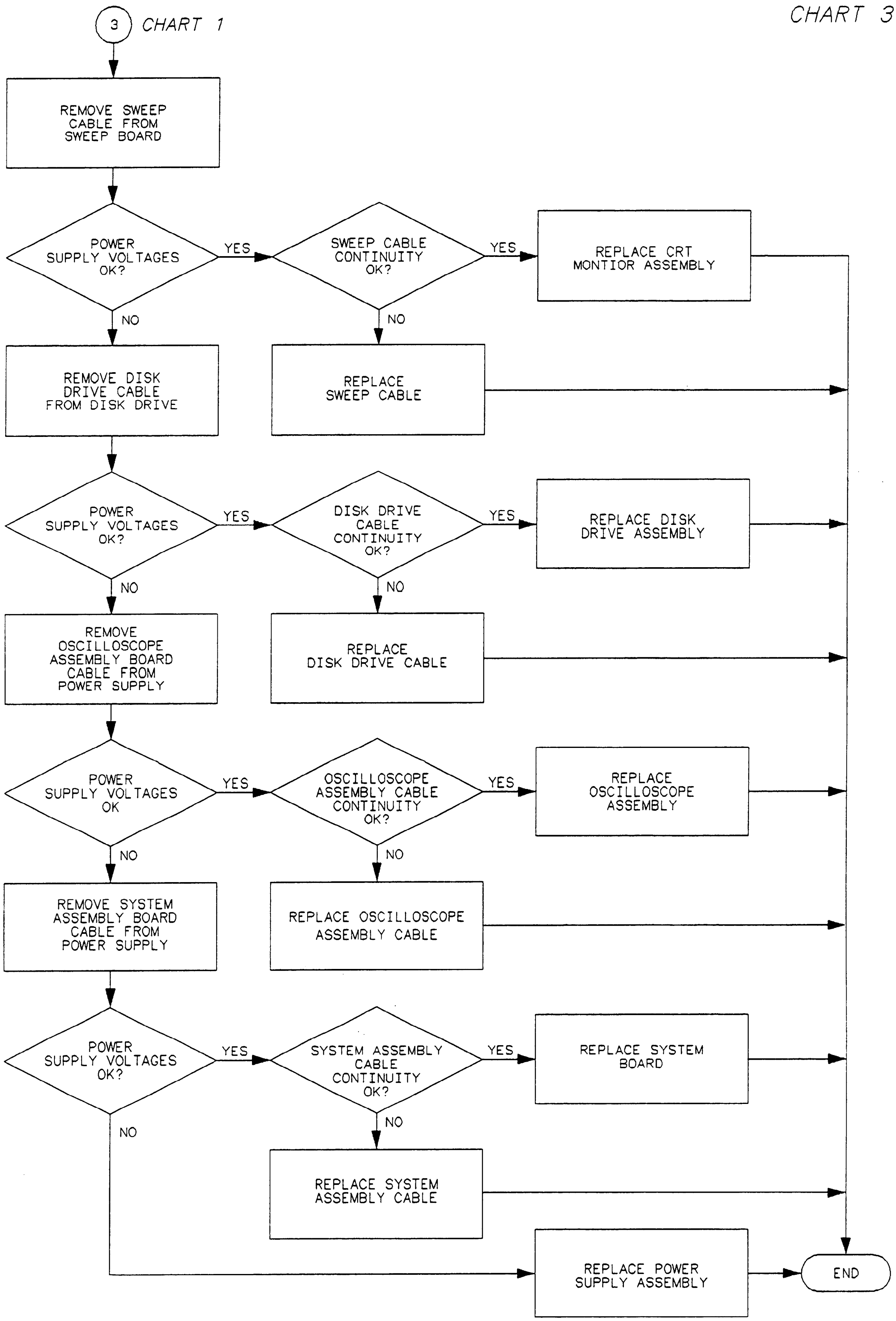


Figure 6C-3. Trouble Isolation for Power Supply

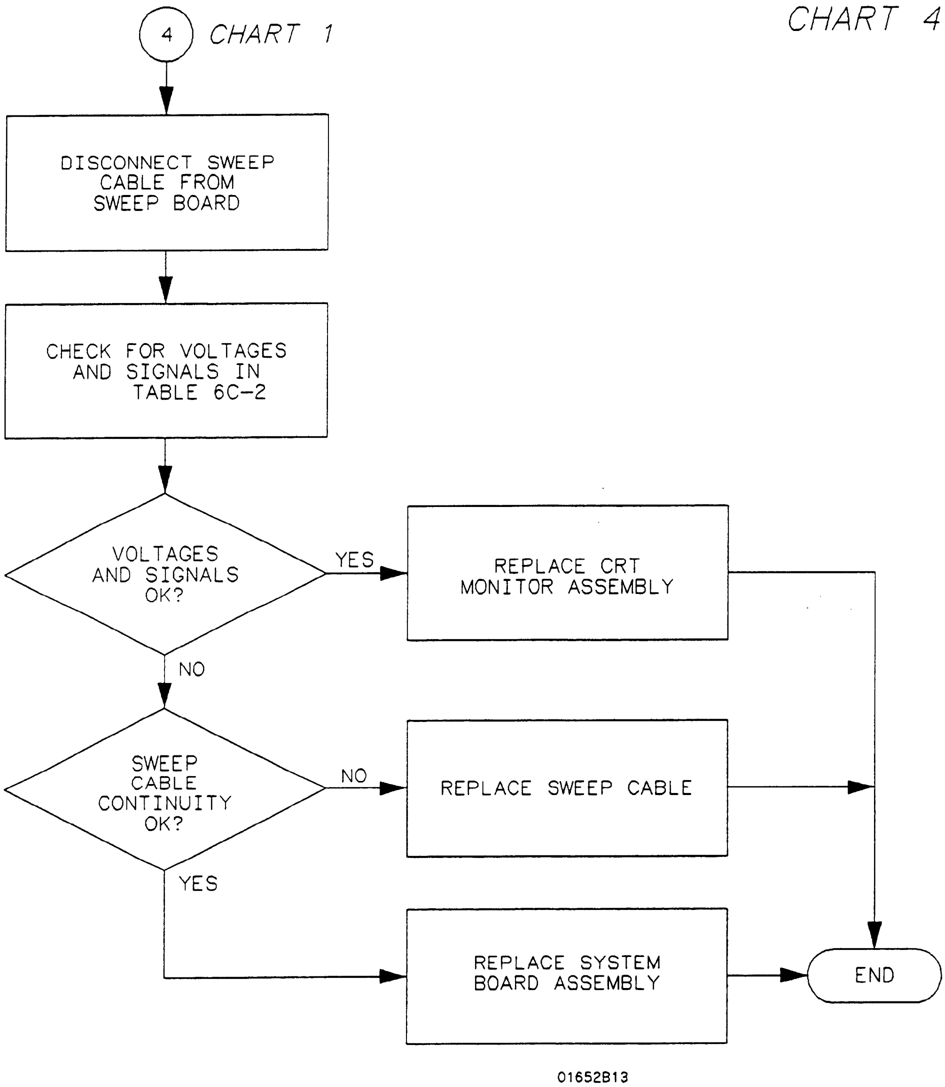


Figure 6C-4. Trouble Isolation for CRT Monitor

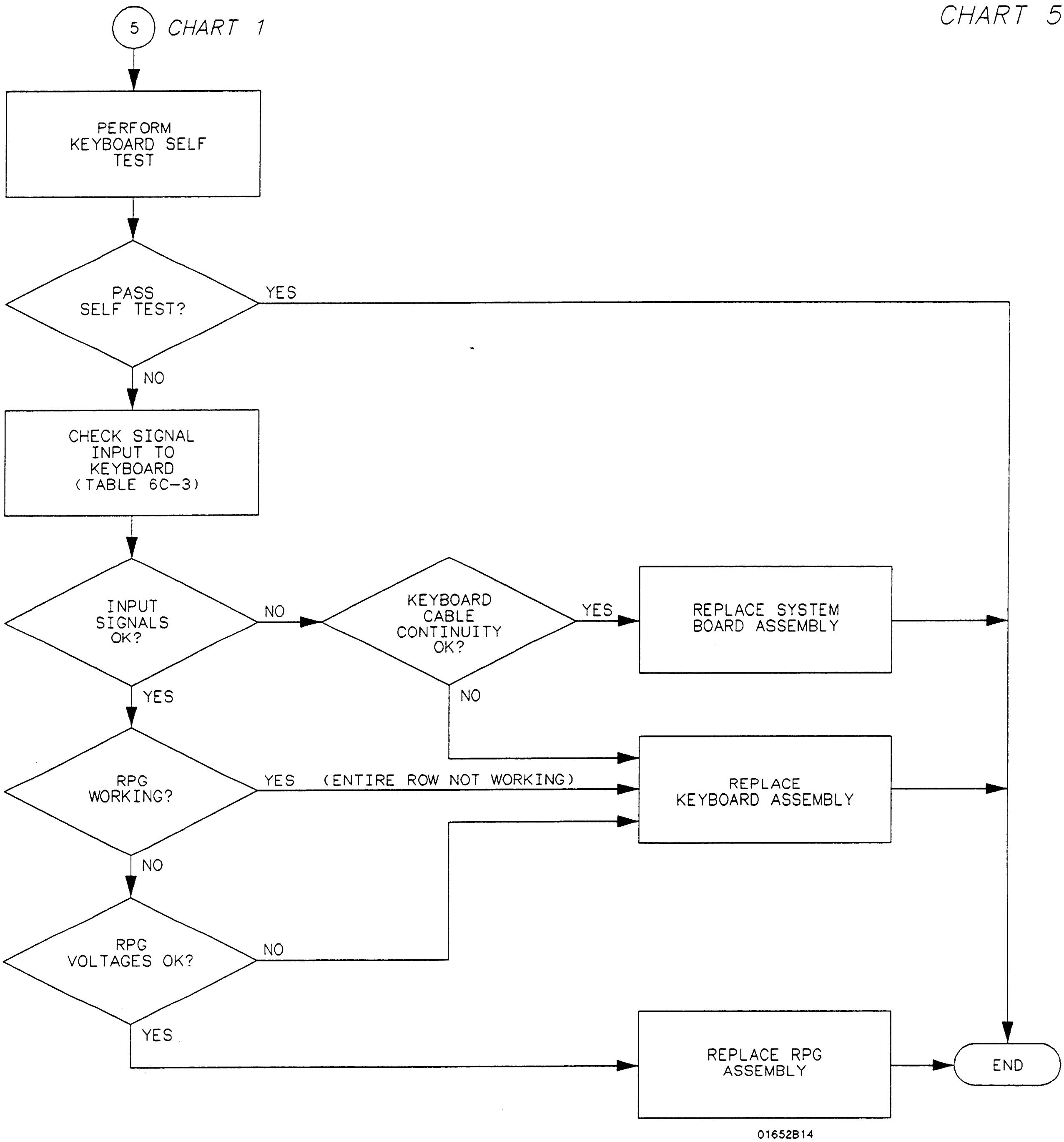


Figure 6C-5. Trouble Isolation for Keyboard

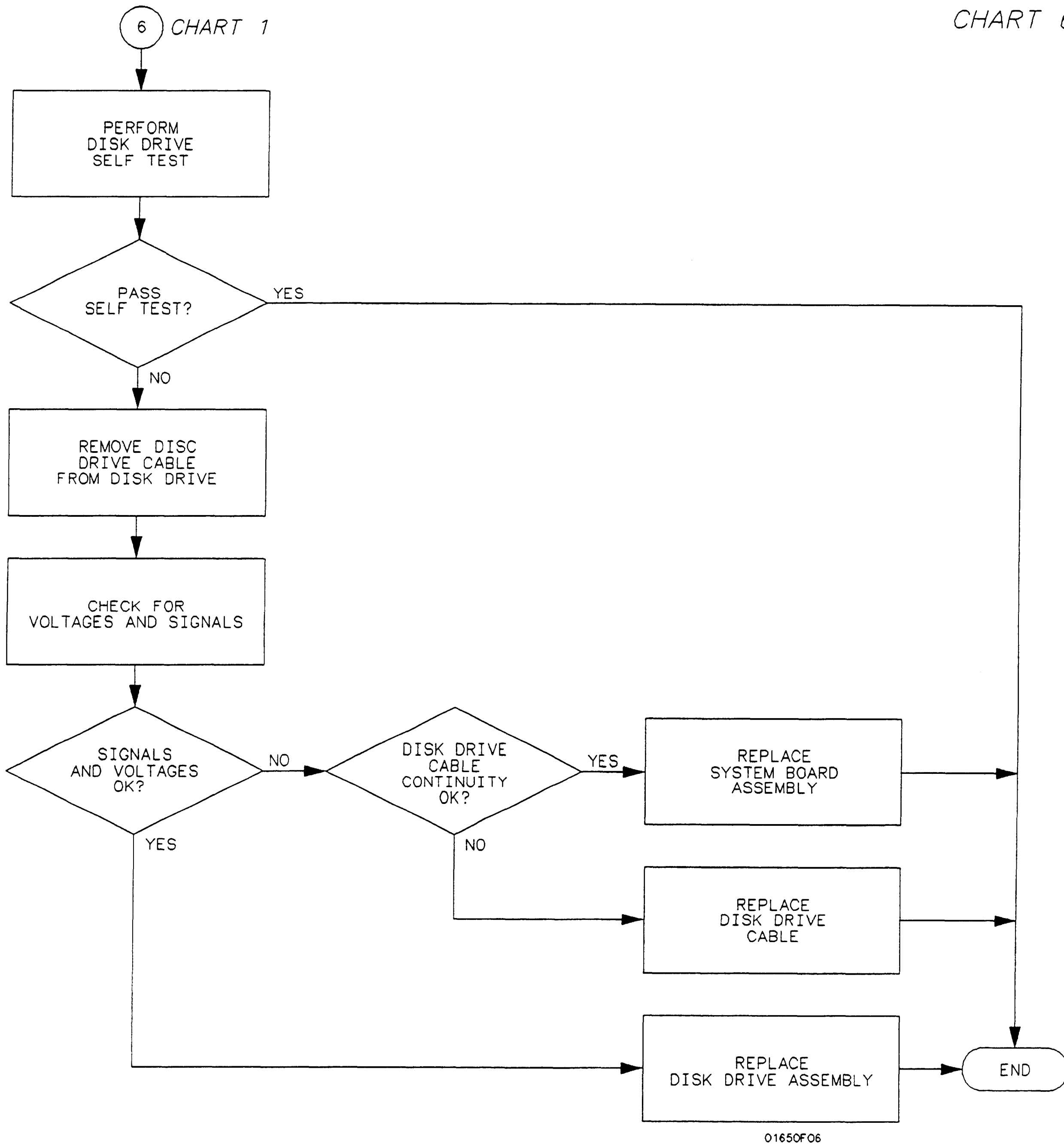


Figure 6C-6. Trouble Isolation for Disk Drive

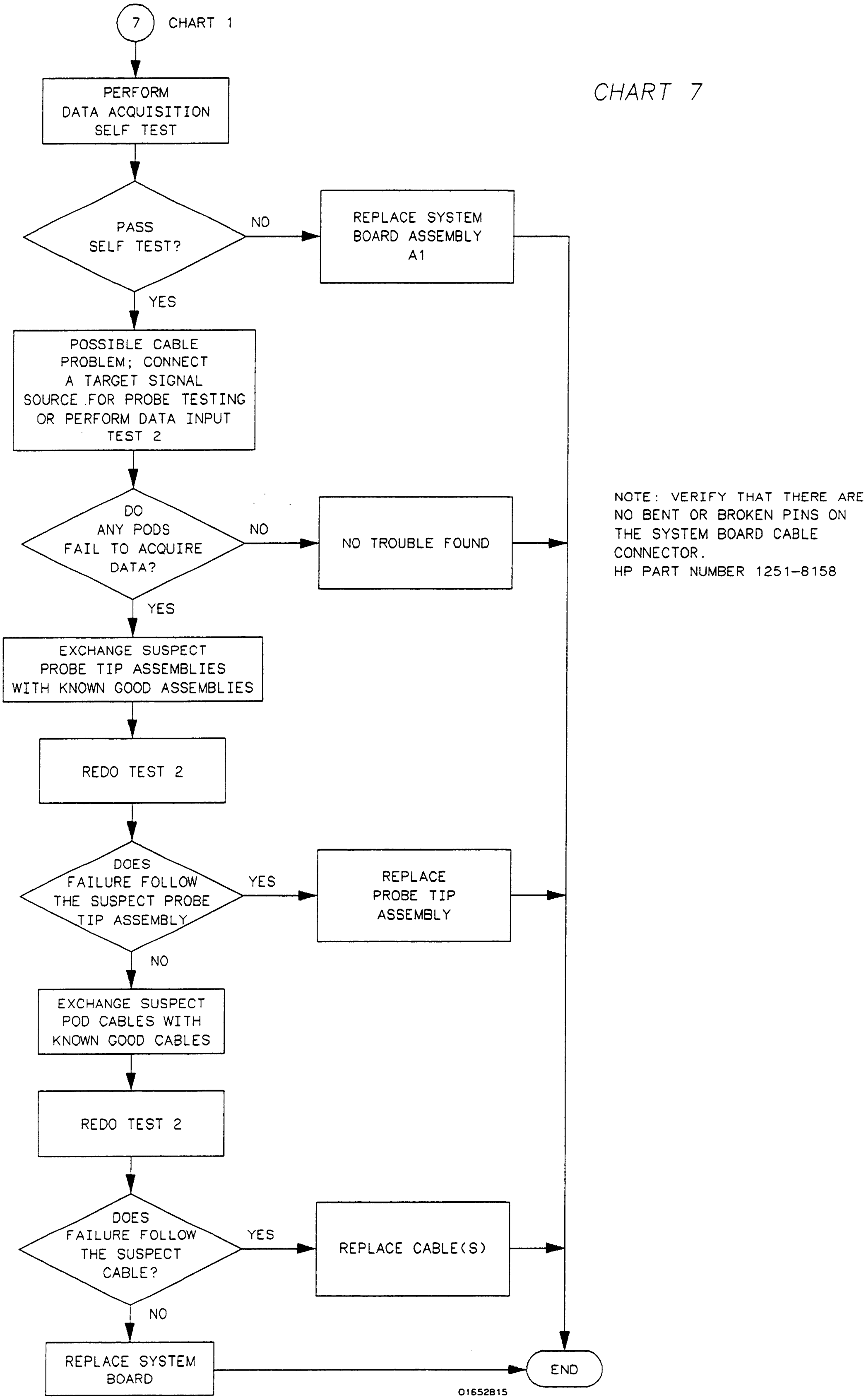


Figure 6C-7. Trouble Isolation for Data Acquisition

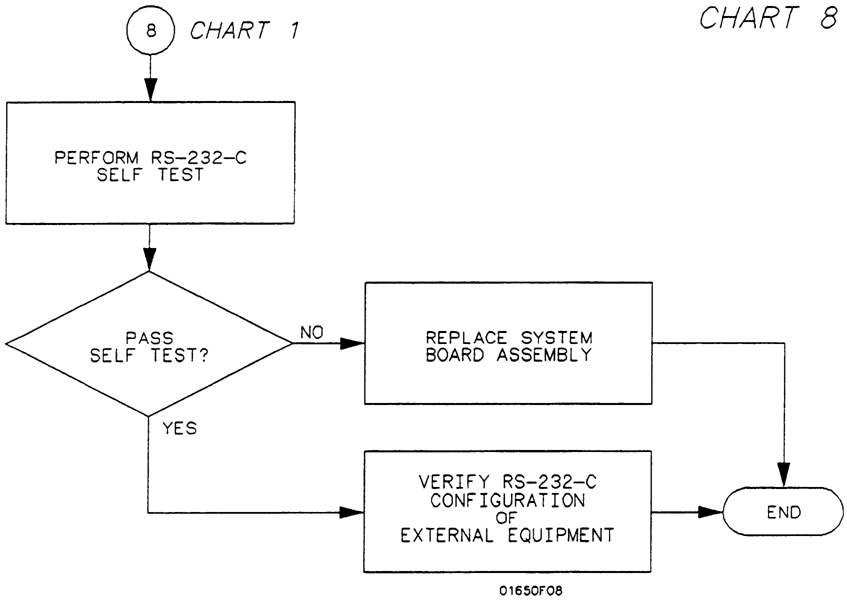


Figure 6C-8. Trouble Isolation for RS-232C

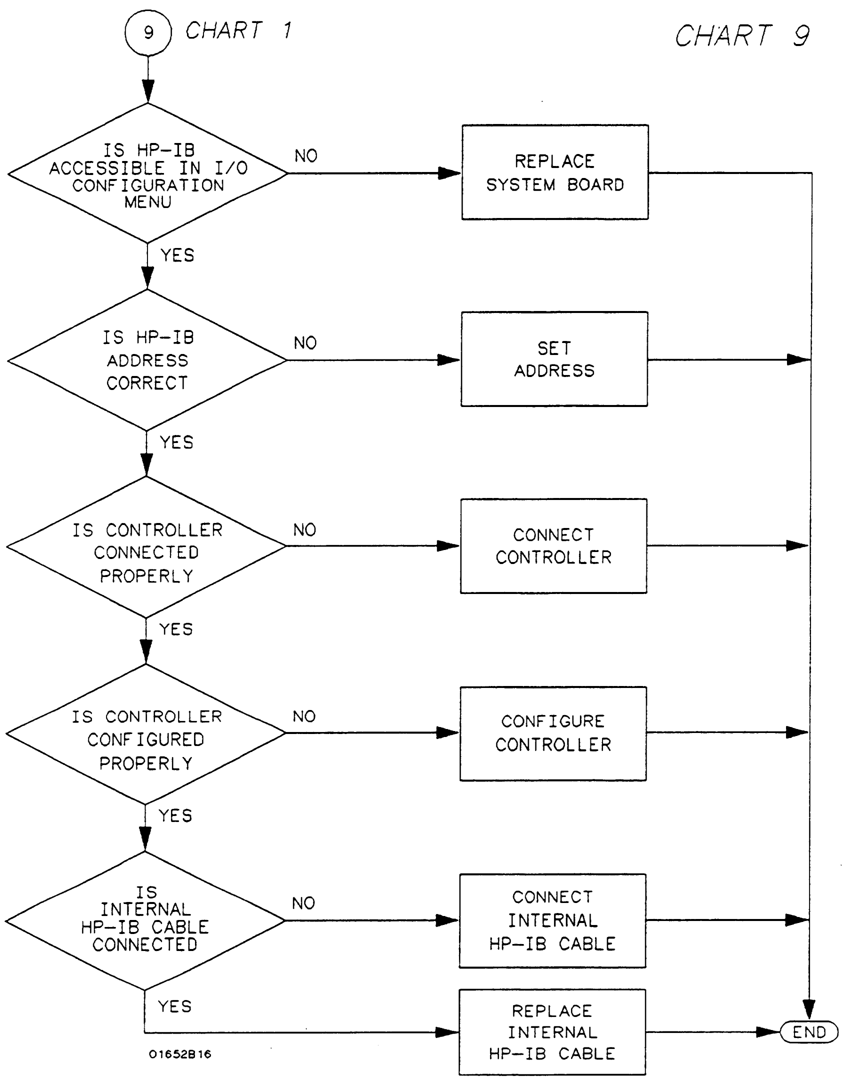


Figure 6C-9. Trouble Isolation for HP-IB

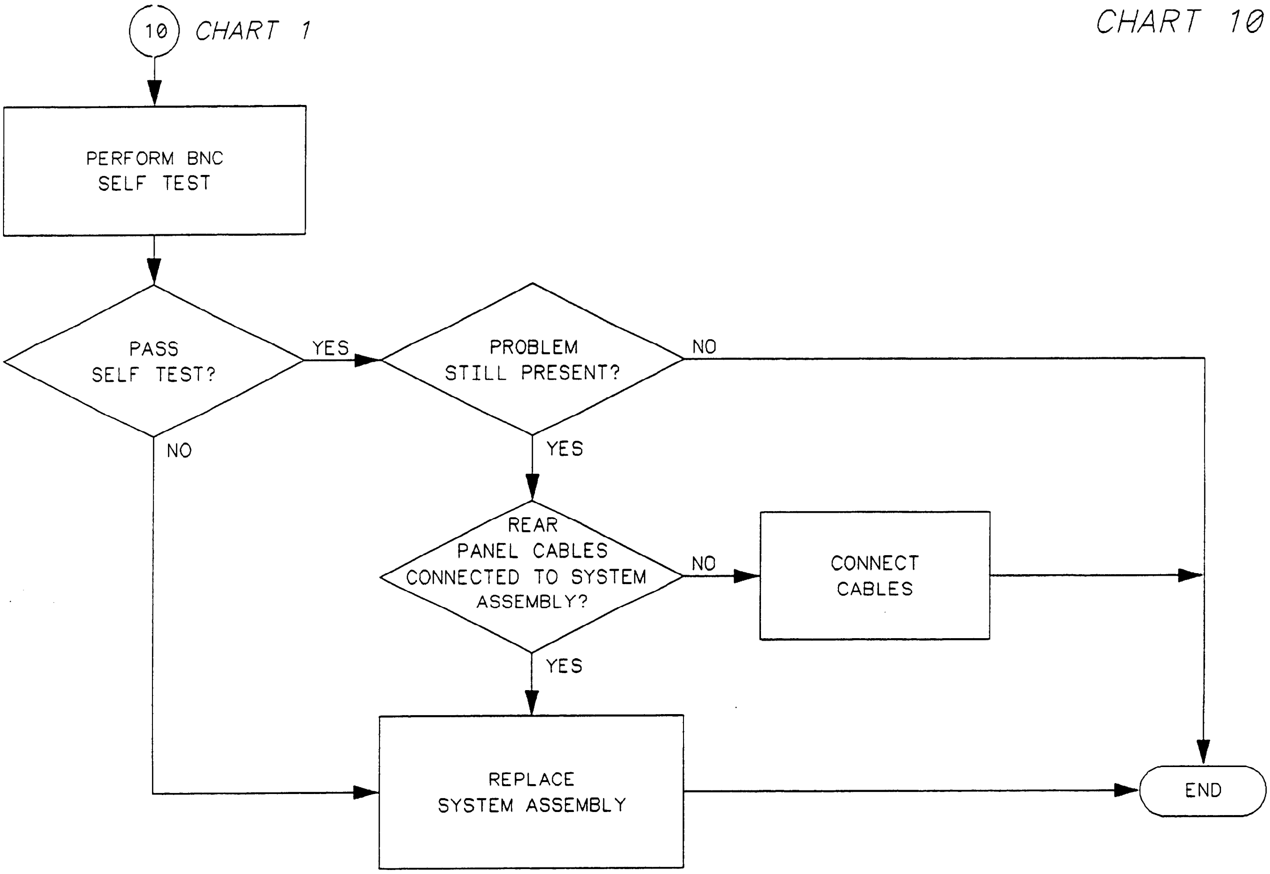


Figure 6C-10. Trouble Isolation for BNC

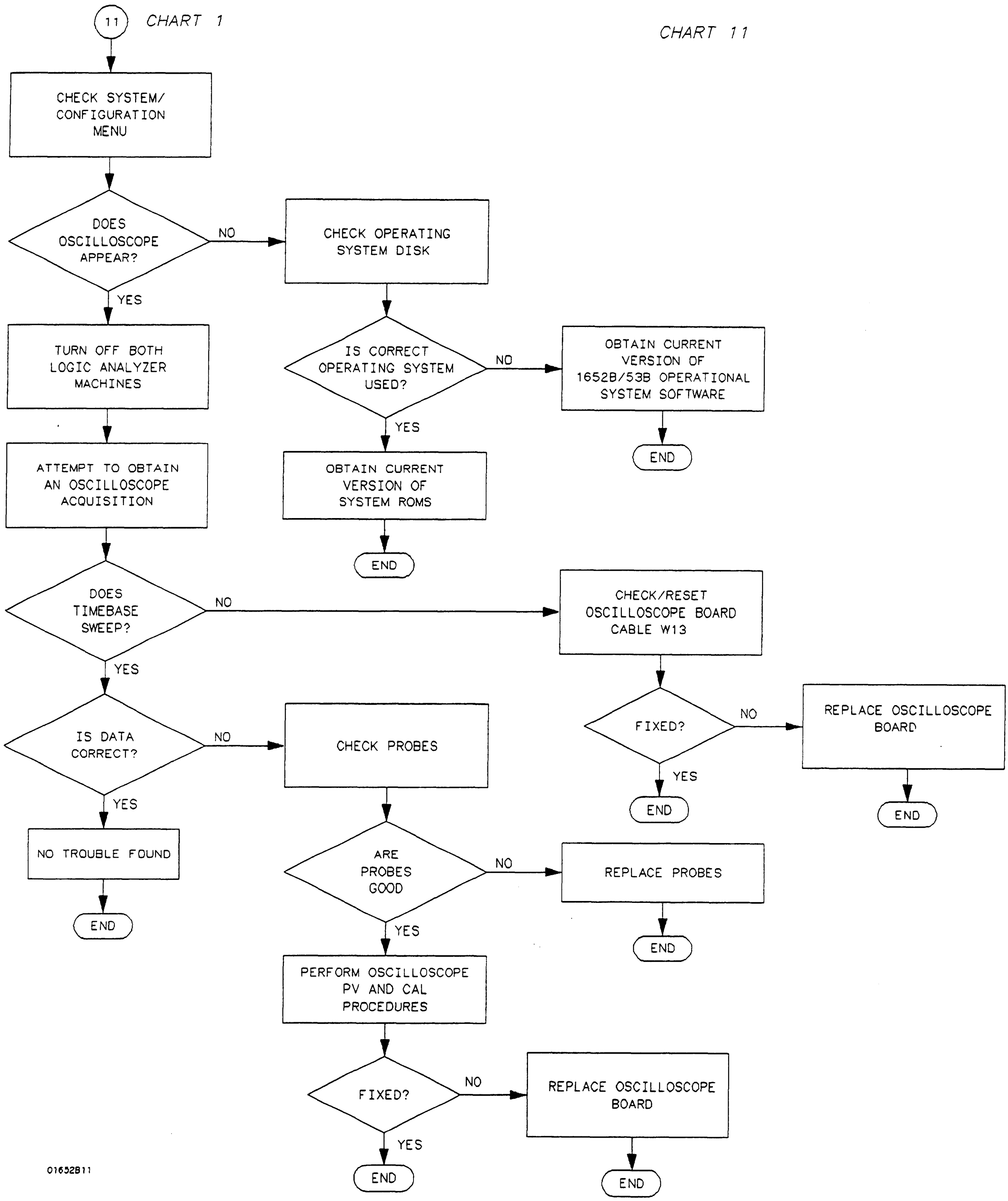


Figure 6C-11. Trouble Isolation for Oscilloscope

Power Supply Voltages Check

The power supply must be loaded by either the System Assembly Board or with an added resistor to check the voltages.

Warning

This procedure is to be performed only by service-trained personnel aware of the hazards involved (such as fire and electrical shock).

Power Supply Loaded by System Assembly

1. Remove the instrument top cover.
2. Using the figure below, check for the voltages indicated at the testpoints.

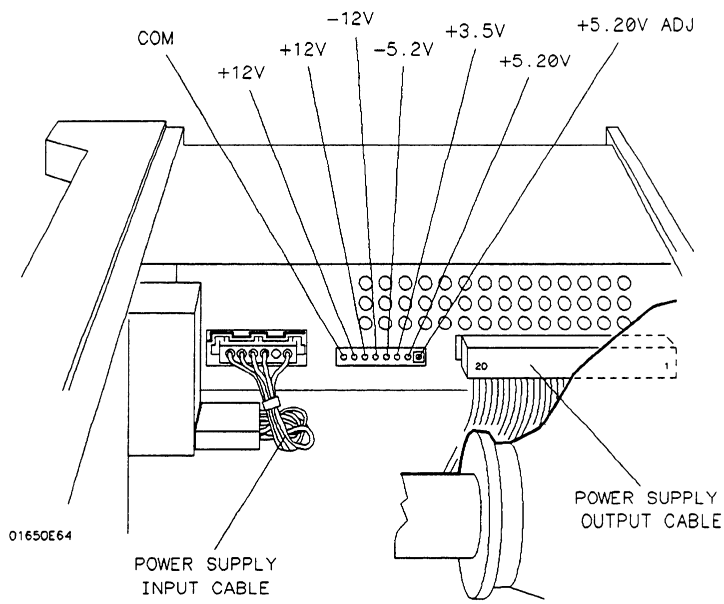


Figure 6C-12. Power Supply Test Points

Power Supply Isolated

Isolate and check the supply with the following steps. Use the figure above for reference.

1. Turn off the instrument and remove the power cable.
2. Disconnect the supply output cable at the supply (see figure above).
3. Load the +5.20 V supply with a 2 ohm 25 watt resistor.
   a. With a jumper wire, connect one end of the resistor to one of the 5.20 V pins (pins 1 through 4) on the supply output.
   b. With another jumper wire, connect the other end of the resistor to one of the ground pins (pins 5 through 8) on the supply output.

4. Reconnect the power cable and turn on the instrument. Then check for the voltages at the supply output using the values in the following table.

Table 6C-2. Power Supply/Main Assembly Voltages

PIN	SIGNAL	PIN	SIGNAL
1	+5.20 V	11	-5.2 V
2	+5.20 V	12	GROUND
3	+5.20 V	13	+12 V
4	+5.20 V	14	GROUND
5	GROUND (Display)	15	-12 V
6	GROUND (Digital)	16	GROUND
7	GROUND (Digital)	17	+12 V (Display)
8	GROUND (Display)	18	-5.2 V
9	+3.5 V	19	+15.5 V (Fan)
10	GROUND	20	GROUND (Fan)

Note

The ground planes (digital, fan, and display) are at the same potential on the power supply, but when you are measuring them on the main assembly, the supplies must be measured with reference to the respective ground.

CRT Monitor Signals Check

1. Remove the instrument top cover.
2. Check the CRT Monitor input cable for the signals and supplies listed in the table below. The cable is the wide ribbon cable connecting the monitor assembly to the System Assembly Board.
3. Dynamic video signals FB (Full-bright) and HB (Half-bright) are TTL inputs. Check for activity on these lines. The table includes a truth table for these signals.

Table 6C-1. CRT Monitor Input Cable Pin Assignments

PIN	SIGNAL	PIN	SIGNAL
1	+5 V (Digital)	2	+12 V (Display)
3	GROUND (Display)	4	GROUND (Display)
5	+12 V (Display)	6	GROUND (Display)
7	+12 V (Display)	8	GROUND (Display)
9	+12 V (Display)	10	HSYNC
11	VSYNC	12	+12 V (Display)
13	GROUND (Digital)	14	GROUND (Digital)
15	GROUND (Display)	16	FB (Full-bright)
17	GROUND (Display)	18	HB (Half-bright)
19	GROUND (Display)	20	+5 V (Digital)

FB	HB	VIDEO
0	0	OFF
0	1	HALF
1	0	FULL
1	1	FULL

Keyboard Signals Check

Isolate a faulty elastomeric keypad or keyboard when the random key(s) are not operating by performing the following steps.

1. Turn off the instrument and remove the power cable.
2. Without disconnecting the keyboard cable, follow the keyboard removal procedure to loosen the keyboard. Leave the keyboard in place in front of the instrument.
3. Reconnect the power cable and turn on the instrument.
4. Run the Keyboard Self Test and press all of the keys.
5. Allow the keyboard assembly to fall forward from the front panel. Separate the elastomeric keypad and keyboard panel from the PC board.
6. Short the PC board trace (with a paper clip or screwdriver) of the non-operating key and look for an appropriate response on the display.
7. If the display responds as if the key were pressed, replace the elastomeric keypad.
8. If the display does not respond as if the key were pressed, replace the keyboard.

The RPG connector has a TTL pulse on pins 1 and 3, when the knob is being turned. Pin 5 of the connector is + 5 V.

The ROW (scan) signal is a low duty-cycle pulse at approximately 60 Hz. It is continually present on pins 14 through 20 of the keyboard cable. Because of the resistance of the keypad contacts, the signal does not appear the same on the COLUMN (data) pins when keys are pressed. Refer to the following table for signals going to and from the keyboard.

Table 6C-3. Keyboard Connector Voltages and Signals

PIN	SIGNAL	PIN	SIGNAL
1	GROUND	2	GROUND
3	COLUMN 6 (Data)	4	+5 V (DIGITAL)
5	GROUND	6	RPG (CLICKS)
7	RPG (DIRECTION)	8	N/C
9	COLUMN 5 (Data)	10	COLUMN 4 (Data)
11	COLUMN 3 (DATA)	12	COLUMN 2 (DATA)
13	COLUMN 1 (DATA)	14	ROW 4 (Scan)
15	ROW 5 (Scan)	16	ROW 2 (Scan)
17	ROW 3 (Scan)	18	ROW 1 (Scan)
19	ROW 0 (SCAN)	20	ROW 6 (SCAN)

Disk Drive Voltages Check

Use the following steps to check the disk drive voltages.

1. Remove the top cover of the instrument.
2. Run the repetitive Disk Drive Self Test.
3. Remove the disk drive cable from the disk drive.
4. Check the disk drive cable for the voltages listed in the following table.

PIN	SIGNAL DESCRIPTION	PIN	SIGNAL DESCRIPTION
1	CHANGE RESET (+5 V)	2	DISK CHANGE
3	+5 V	4	IN USE
5	+5 V	6	DRIVE SELECT3 (+5 V)
7	+5 V	8	INDEX
9	+5 V	10	DRIVE SELECT0
11	+5 V	12	DRIVE SELECT1
13	GROUND	14	DRIVE SELECT2 (+5 V)
15	GROUND	16	MOTOR ON
17	GROUND	18	DIRECTION
19	GROUND	20	STEP
21	GROUND	22	WRITE DATA
23	GROUND	24	WRITE GATE
25	GROUND	26	TRACK 00
27	GROUND	28	WRITE PROTECT
29	+12 V	30	READ DATA
31	+12 V	32	HEAD SELECT
33	+12 V	34	READY

Caution

Do not match the arrows of the cable and connector when connecting the disk drive cable to the disk drive. The connector of the disk drive is marked with an arrow at pin 34 of the connector. The end of the disk drive cable is marked at pin 1 of the cable. Matching the arrows will damage the disk drive.

Troubleshooting Auxiliary Power

The +5 volt auxiliary power line is protected by a current limiting circuit. If the current on pins 1 and 39 exceeds 2.3 amperes, the circuit will open. When the short is removed, the circuit will reset in approximately 20 ms. If you suspect a problem with this circuit, remove all loads from pins 1 and 39 and measure the voltage between these pins and ground (pins 2 and 40) with a voltmeter. There should be +5 volts at pins 1 and 39 after the 20 ms reset time.

If the +5 volts does not appear on one or both of these pins (pins 1 and 39), replace the analyzer cable. If the +5 volts still does not appear on these pins, refer to chart 3 in figure 6C-3.

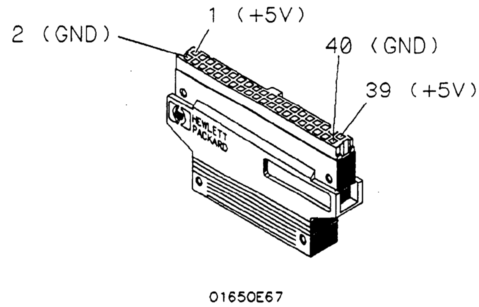


Figure 6C-13. Cable Power and Ground

Contents

Section 6D:

Assembly Removal and Replacement

6D

Assembly Removal and Replacement

Introduction

This section contains the procedures for removal and installation of major assemblies of the HP 1652B/1653B Logic Analyzer. Read the Safety Summary at the front of this manual before servicing the instrument. The relative location of the replaceable components are shown in figure 6D-1. The part numbers and descriptions for these components are listed in section 5.

Warning

Hazardous voltages exist on the power supply, the CRT, and the display sweep board. To avoid electrical shock, adhere closely to the following procedures. After disconnecting the power cable, wait at least three minutes for the capacitors on the power supply and sweep boards to discharge before servicing this instrument.

Caution

Never attempt to remove or install any assembly with the instrument ON or the power cable connected. This can result in component damage.

Caution

The effects of ELECTROSTATIC DISCHARGE can damage electronic components. Use grounded wriststraps and mats when performing any kind of service to this instrument.

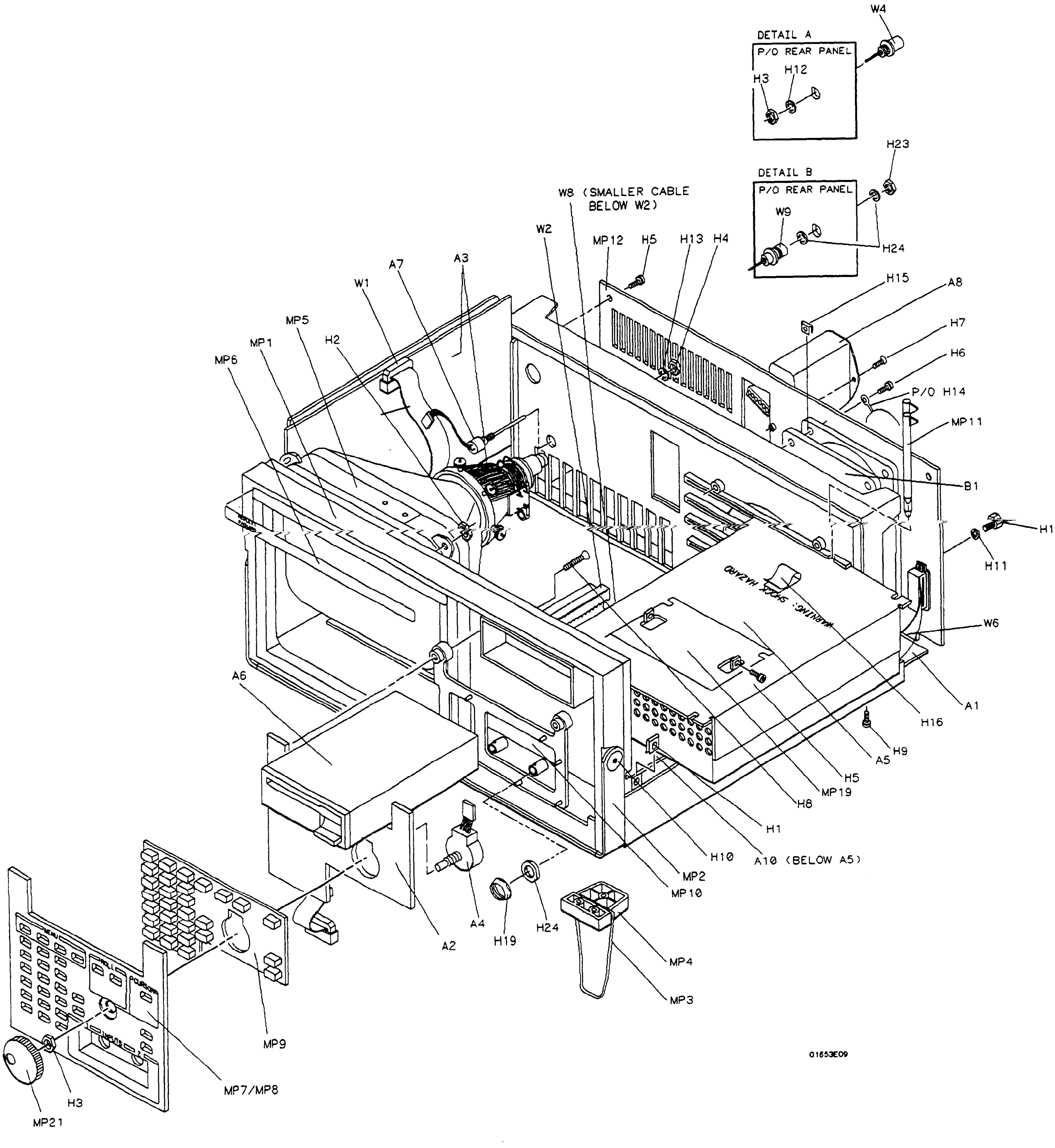


Figure 6D-1. HP 1652B/1653B Exploded View

Removal and Replacement of the Rear Panel Assembly

1. Turn off the instrument and disconnect the power cable.
2. Disconnect the logic analyzer cables from the rear panel of the instrument.
3. Remove the six screws from the top and the two screws from each side of the instrument's top cover.
4. Lift off the top cover.
5. Disconnect the line filter cable from the power supply.
6. Disconnect the ground cable of the line filter from the oscilloscope assembly.
7. Disconnect the Intensity adjust cable from the rear of the high voltage sweep board.
8. Remove the eight screws at the edges of the rear panel.
9. Pull the rear panel straight out from the instrument about three inches.

Note

An ESD ground spring clip is installed on the RS-232C connector behind the rear panel. This ground spring clip is not mechanically secured to the instrument. Make sure the ground spring clip does not fall off during disassembly.

10. Remove the two screws holding the HP-IB ribbon cable connector to the rear panel.
11. Disconnect the External Trigger Input cable from connector J9 on the Main Assembly.
12. Disconnect the External Trigger Output cable from connector J10 on the Main Assembly.
13. Disconnect the Probe Compensation cable from the oscilloscope board.
14. Disconnect the fan cable from the Main Assembly.
15. Separate the rear panel from the instrument cabinet.
16. Replace the rear panel by reversing this procedure.

Note

When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

Removal and Replacement of the Disk Drive

1. Turn off the instrument and disconnect the power cable.
2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.
3. Lift off the top cover.
4. Remove the two screws securing the disk drive to the power supply panel.
5. Disconnect the disk drive cable assembly (W3) from the disk drive.

Caution

Do not match the arrows of the cable and connector when reconnecting the disk drive cable to the disk drive. The connector of the disk drive is marked with an arrow at pin 34 of the connector. The end of the disk drive cable is marked at pin 1 of the cable. Matching the arrows will result in damaging the disk drive.

6. Slide the disk drive through the front panel of the instrument cabinet as in figure 6D-2.
7. Replace the disk drive by reversing this procedure.

Note

When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

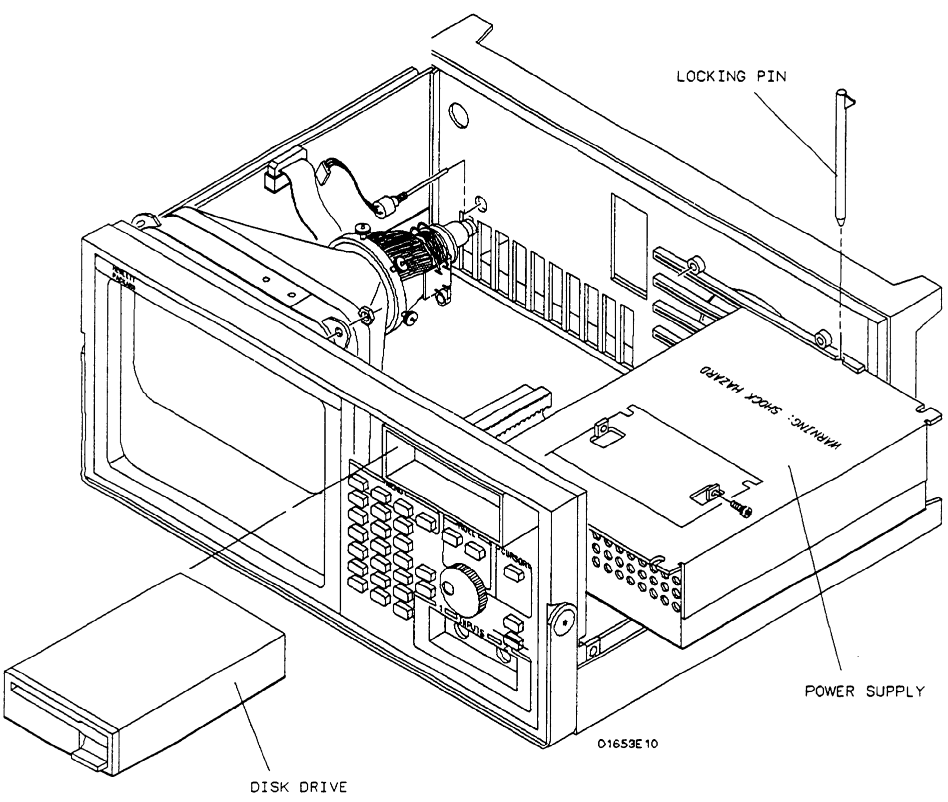


Figure 6D-2. Disk Drive and Power Supply Removal

Removal and Replacement of the Power Supply Assembly

When necessary, refer to other removal procedures.

1. Turn off the instrument and disconnect the power cable.
2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.
3. Lift off the top cover.
4. Remove the Disk Drive
5. Remove the disk drive cable assembly (W3) from the disk drive panel and let it lay off the side of the instrument.
6. Remove the cable (W2) that connects the Power Supply to the Main Assembly.
7. Disconnect the line filter cable from the Power Supply.
8. Remove the two locking pins that secure the Power Supply at the right front and rear corners of the instrument cabinet. Pull these pins up and out of the instrument.
9. Slide the power supply through the side of the cabinet as in the previous figure 6D-2.
10. Replace the power supply by reversing this procedure.

Note When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

Removal and Replacement of the Oscilloscope Assembly

When necessary, refer to other removal procedures.

1. Turn off the instrument and disconnect the power cable.
2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.
3. Lift off the top cover.
4. Remove the Disk Drive and Power Supply.
5. Remove the Line Filter Switch Assembly from the rear panel.
6. Disconnect the probe compensation cable from the oscilloscope assembly.
7. Disconnect the ground cable of the line filter from the oscilloscope assembly.
8. Disconnect the cable assembly W8 from connector J2 on the oscilloscope assembly.

9. Remove the six screws securing the oscilloscope assembly to the support panel.

Note Do not remove the two screws at the front of the oscilloscope board that hold the two attenuators in place.

---

10. Remove the two nuts (H19) and two washers (H20) that secure the attenuator BNCs to the front panel.
11. Slide the oscilloscope assembly toward the rear panel to allow the BNCs to clear the front panel.
12. Remove the oscilloscope assembly by tilting the rear of the assembly up and lifting the assembly out through the top of the instrument cabinet. Make sure that the BNCs clear the front panel.
13. Replace the oscilloscope assembly by reversing this procedure.

Note When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

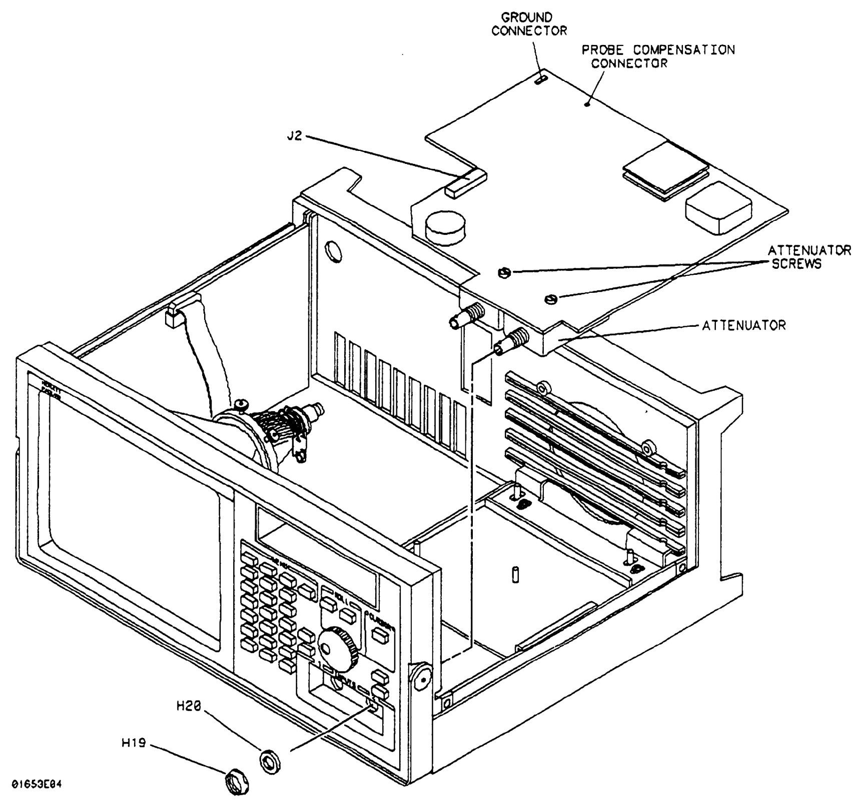


Figure 6D-3. Oscilloscope Assembly Removal

Removal and Replacement of the Attenuators

Attenuators are not part of the oscilloscope board. If the oscilloscope board is replaced, the attenuators must be moved to the replacement board.

Caution

ELECTROSTATIC DISCHARGE can damage electronic components. Use grounded wriststraps and mats when servicing attenuators.

When necessary, refer to other removal procedures.

1. Turn off the instrument and disconnect the power cable.
2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.
3. Lift off the top cover.
4. Remove the Disk Drive, Power Supply, and Oscilloscope Assembly.
5. From the component side of the Oscilloscope Assembly, remove the two screws that secure the Attenuator.
6. A 24-pin connector, located at the rear of the inside of the attenuator, connects the attenuator to the PC board. With a gentle rocking or prying motion, lift the attenuator from the PC board.

Note

Prying at the rear of the attenuator with a small flat-blade screwdriver, between the attenuator and the PC board, will help control the attenuator removal.

7. Replace the attenuator by reversing this procedure.

Note

When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

Removal and Replacement of the Keyboard Assembly

When necessary, refer to other removal procedures.

1. Turn off the instrument and disconnect the power cable.
2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.
3. Lift off the top cover.
4. Remove the Disk Drive, Power Supply, and Oscilloscope Assembly.
5. Loosen the two screws that hold the rear bracket on the oscilloscope assembly support panel until the bracket moves freely.
6. Remove the support panel by carefully tilting the rear of the panel up and lifting the panel out through the top of the instrument cabinet. Make sure the metal tabs on the front of the support panel clear the front panel.

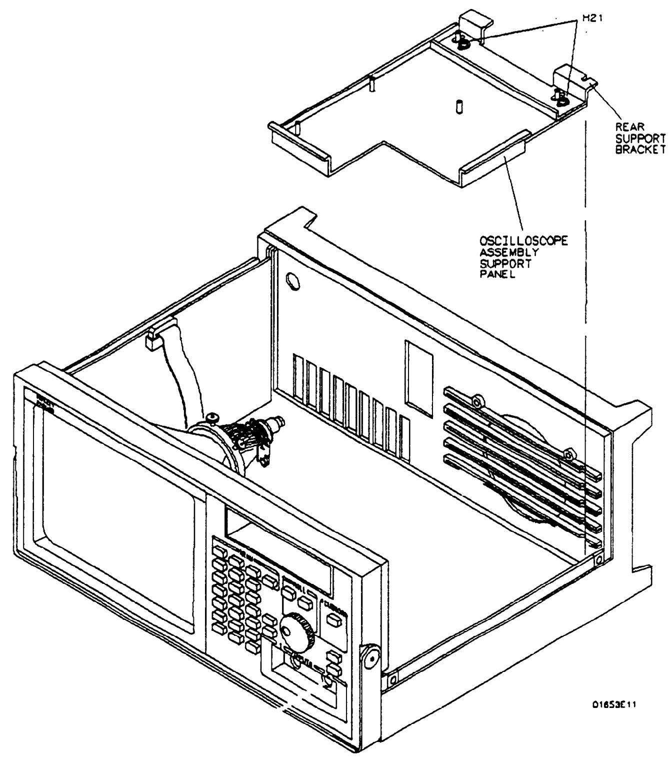


Figure 6D-4. Support Panel Removal

7. From the back side of the front panel, remove the four screws securing the keyboard assembly to the front of the instrument cabinet.
8. Disconnect the keyboard assembly ribbon cable from the Main Assembly.
9. Pull on the knob to remove the keyboard assembly (label, keyboard panel, keypad, PC board, RPG, and knob) from the front panel as one unit.

Disassembling the Keyboard Assembly

Use the following steps to disassemble the keyboard assembly.

10. Disconnect the Rotary Pulse Generator (RPG) cable from the PC board on the keyboard assembly.
11. Separate the PC board, keypad, and keyboard panel/label.

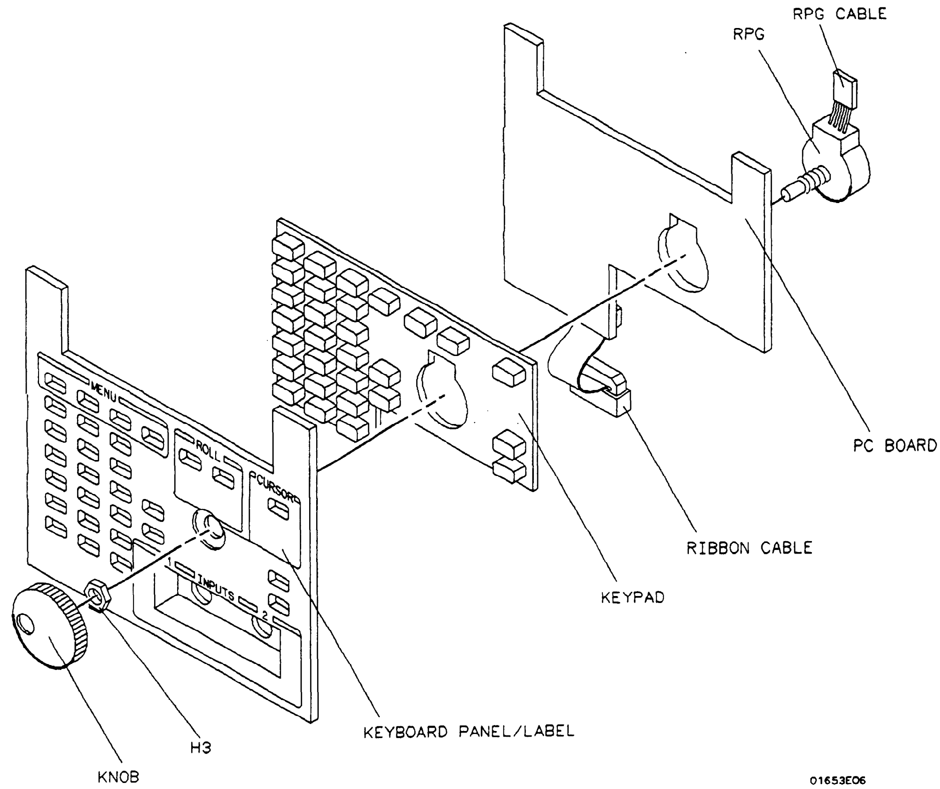


Figure 6D-5. Exploded view of the Keyboard Assembly

12. The knob is force fitted on the RPG shaft. To remove the knob, pull it straight off.
13. Remove the 3/8-inch nut from the RPG.
14. Remove the RPG from the keyboard panel.
15. The keyboard label uses a self-stick adhesive. If the label must be removed, carefully peel it off.
16. Replace the keyboard assembly by reversing this procedure.

Note

When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

Removal and Replacement of the Fan

When necessary, refer to other removal procedures.

1. Turn off the instrument and disconnect the power cable.
2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.
3. Lift off the top cover.
4. Remove the Rear Panel Assembly.
5. Pull the rear panel out until the fan clears the instrument cabinet. It is not necessary to completely remove the rear panel.
6. For reassembly, note the orientation of the fan cable. Remove the fan by removing the four screws securing the fan to the rear panel.
7. Replace the fan by reversing this procedure.

Note When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

Removal and Replacement of the Main Assembly

Caution ELECTROSTATIC DISCHARGE can damage electronic components. Use grounded wriststraps and mats when servicing the main assembly.

When necessary, refer to other removal procedures.

1. Turn off the instrument and disconnect the power cable.
2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.
3. Lift off the top cover.
4. Remove the Disk Drive, Power Supply, and Oscilloscope Assembly.
5. Loosen the two screws that hold the rear bracket on the oscilloscope assembly support panel until the bracket moves freely.
6. Remove the support panel by tilting the rear of the panel up and lifting the panel out through the top of the instrument cabinet as in the previous figure 6D-4. Make sure the metal tabs on the support panel clear the front panel.
7. Remove the Rear Panel Assembly.

8. Remove the following cables from the main assembly board:

   - Disk drive cable.
   - Oscilloscope board cable.
   - Power supply cable.
   - CRT sweep cable.
   - HP-IB cable.

9. Disconnect the keyboard assembly ribbon cable from the Main Assembly.

10. Carefully place the instrument on its side.

11. From the bottom of the instrument, remove the eight screws that secure the Main Assembly to the instrument cabinet.

12. Set the instrument in the normal position.

13. Slide the main assembly out of the rear of the instrument cabinet.

14. Replace the Main Assembly by reversing this procedure.

Note

When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

Removal and Replacement of the CRT Monitor Assembly

The sweep board, CRT, and CRT yoke are all parts of one HP part number. They have been adjusted as a unit and should be replaced as a unit, rather than individually. Do not remove the yoke from the CRT.

When necessary, refer to other removal procedures.

1. Turn off the instrument and disconnect the power cable.

2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.

3. Lift off the top cover.

4. Remove the Rear Panel, Power Supply, and Main Assembly.

Caution

Discharge the post accelerator lead to the CRT monitoring band only. Components will be damaged if the post accelerator is discharged to other areas.

5. Connect a jumper lead between the mounting band of the CRT and the shaft of a screwdriver.

6. Discharge the CRT by placing the grounded screwdriver under the protective rubber cap of the post accelerator lead and momentarily touching the screwdriver to the metal clip of the post accelerator.

Caution

The CRT may charge up by itself even while disconnected. Discharge the CRT before handling. Use a jumper lead to short the CRT post accelerator terminal to the CRT mounting band.

7. Disconnect the post accelerator lead from the CRT by firmly squeezing the rubber cap until the metal clip disengages from the CRT.

8. Disconnect the following cables at the sweep board or CRT:

   - Intensity cable.
   - CRT Monitor ribbon cable.
   - Two CRT yoke cables.
   - CRT base cable.

9. Slide the sweep board up and out of the cabinet slot. When installing the sweep board, it may be necessary to press on the center of the outer shield of the sweep board to allow the board to clear the cabinet support rib.

10. Carefully place the instrument with the front panel facing down.

11. Remove the four nuts securing the CRT to the front panel.

12. Remove the sweep board guide.

13. Remove the CRT. When reinstalling the CRT, place it with the post accelerator terminal toward the inside of the instrument, away from the sweep board.

14. Replace the CRT Monitor Assembly by reversing this procedure.

Note If necessary, after replacing the CRT Monitor Assembly perform the CRT Monitor Assembly Adjustment procedures in section 4 of this manual.

Note When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

Removal and Replacement of the Feet/Tilt Stand

When necessary, refer to other removal procedures.

1. Turn off the instrument and disconnect the power cable.

2. Remove the six screws from the top and the two screws from each side of the instrument's top cover.

3. Lift off the top cover.

4. Remove the Rear Panel, Power Supply, Main Assembly, and CRT Monitor Assembly.

5. Remove the three screws securing each foot/tilt stand to the bottom of the instrument cabinet.

6. Replace the feet/tilt stand by reversing this procedure.

Note When you reinstall the top cover, insert the four screws on the sides of the cover first while making sure the cover fits into the grooves of the instrument cabinet. Then insert the six screws in the top of the cover.

HP
1
6
5
2
B
/
1
6
5
3
B

HP 1652B/1653B
Logic Analyzers
Service Manual

Please send directory corrections to:
Test & Measurement Catalog, 54-AF
Hewlett-Packard Company
5301 Stevens Creek Blvd.
P.O. Box 58059
Santa Clara, CA 95052-8059
Tel: (408) 553-7271
Fax: (408) 983-1006

HEADQUARTERS OFFICES

If no sales office is listed for your country or area, please contact one of these headquarters offices.

AUSTRALIA/ NEW ZEALAND
Hewlett-Packard Ltd.
31 - 41 Joseph St.
BLACKBURN, Victoria 3130
Tel: (61/3) 895-2895
Fax: (61/3) 898-7831
Telex: 31024MELB

FAR EAST
Hewlett-Packard Asia Ltd.
22/F Bond Centre, West Tower
89 Queensway, Central
G.P.O. Box 863
HONG KONG
Tel: (852/5) 848-7777
Fax: (852/5) 868-4997
Telex: 76793 HPA HX
Cable: HPASIAL TD

JAPAN
Yokogawa-Hewlett-Packard Ltd.
3-29-21, Takaido-Higashi
Suginami-Ku, **TOKYO** 168
Tel: (81/3) 331-6111
Fax: (81/3) 335-1478

UNITED KINGDOM
Hewlett-Packard Ltd.
Nine Mile Ride
WOKINGHAM
Berkshire, RG11 3LL
Tel: (44/34) 477-3100
Fax: (44/34) 476-3526

EASTERN EUROPE
Hewlett-Packard GmbH
Lieblgasse 1
P.O. Box 72
1222 **VIENNA**, Austria
Tel: (43/222) 2500-0
Fax: (43/222) 2500-444
Telex: 134 425

NORTHERN EUROPE
Hewlett-Packard S.A.
Van der Hooplaan 241
P.O. Box 999
1185 LN **AMSTELVEEN**
The Netherlands
Tel: (31/20) 547 9999
Fax: (31/20) 43 1120
Telex: 18919 hpner nl

SOUTH EAST EUROPE
Hewlett-Packard GmbH
Lelbigasse 1
P.O. Box 72
1222 **VIENNA**, Austria
Tel: (43/222) 2500-0
Fax: (43/222) 2500-444
Telex: 134 425 hpner nl

EUROPEAN MULTICOUNTRY REGION
Hewlett-Packard S.A.
150, Route du Nant d'Avril
1217 Meyrin 2
GENEVA, Switzerland
Tel: (41/22) 780 8111
Fax: (41/22) 780 8609

MIDDLE EAST AND AFRICA
Hewlett-Packard S.A.
International Sales Branch
Middle East/Africa
7, rue du Bois-du-Lan
P.O. Box 364
1217 Meyrin 1
GENEVA, Switzerland
Tel: (41/22) 780 7111
Fax: (41/22) 783 7535

CANADA
Hewlett-Packard (Canada) Ltd.
6877 Goreway Drive
MISSISSAUGA, Ontario L4V 1M8
Tel: (416) 678-9430
Fax: (416) 673-7253

LATIN AMERICA
Hewlett-Packard Company
Latin American Region, Miami Office
7208 N.W. 31st St.
MIAMI, FL 33122
United States
(305) 590-1400/1401
Fax: (305) 590-1427/1428
Telex: 441603HPMIAMI

UNITED STATES OF AMERICA
Customer Information Center
(800) 752-0900
6:00 AM to 5:00 PM Pacific Time

U.S. GOVERNMENT SPECIALISTS
ARMY
Tel: (201) 562-6107
Fax: (201) 562-6246
NAVY
Tel: (301) 258-2311
Fax: (301) 948-5986
AIR FORCE
Tel: (512) 491-1223
Fax: (512) 491-1299
FEDERAL PROGRAMS
Tel: (301) 362-7702
Fax: (301) 362-7650

EASTERN USA
Hewlett-Packard Co.
4 Choke Cherry Road
ROCKVILLE, MD 20850
Tel: (301) 670-4300

MIDWESTERN USA
Hewlett-Packard Co.
5201 Tollview Drive
ROLLING MEADOWS, IL 60008
Tel: (312) 255-9800

SOUTHERN USA
Hewlett-Packard Co.
2015 South Park Place
ATLANTA, GA 30339
Tel: (404) 955-1500

WESTERN USA
Hewlett-Packard Co.
5161 Lankershim Blvd.
NORTH HOLLYWOOD, CA 91601
Tel: (818) 505-5600

ALGERIA
Hewlett-Packard Trading S.A.
Bureau de Liaison
71, Lot. Benachour Abdelkader
CHERAGA
Tel: (213) 281 0287
Fax: (213) 281 0287
Telex: 0408 63161

ARGENTINA
Hewlett-Packard Argentina S.A.
Montaneses 2140/50
1428 **BUENOS AIRES**
Tel: (54/1) 781 4069
(54/1) 781-4090

AUSTRALIA

Brisbane, Queensland Office
Hewlett-Packard Australia Ltd.
10 Payne Road
THE GAP, Queensland 4061
Tel: (61/7) 300-4133
Fax: (61/7) 300-5592
Telex: 42133
Cable: HEWPARD Brisbane

Canberra, Australia Capital Territory Office
Hewlett-Packard Australia Ltd.
Thynne Street, Fern Hill Park
BRUCE, A.C.T. 2617
Tel: (61/6) 251-6999
Fax: (61/6) 251-6948
Telex: 62650
Cable: HEWPARD Canberra

Melbourne, Victoria Office
Hewlett-Packard Australia Ltd.
126-142 Trennery Crescent
Abbotsford, VIC 3067
Tel: (61/3) 895-2895
Fax: (61/3) 898-7831
Cable: HEWPARD Melbourne

Parkside, South Australia Office
Hewlett-Packard Australia Ltd.
153 Greenhill Road
PARKSIDE, S.A. 5063
Tel: (61/8) 272-5911
Fax: (61/8) 373-1398

Perth, Western Australia Office
Hewlett-Packard Australia Ltd.
Herdsman Business Park
Cnr. Hasler & Gould St.
OSBORNE PARK, W.A. 6017
Tel: (61/9) 441-8000
Fax: (61/9) 242-1682
Cable: HEWPARD Perth

Sydney, New South Wales Office
Hewlett-Packard Australia Ltd.
17-23 Talavera Road
P.O. Box 308
NORTH RYDE, N.S.W. 2113
Tel: (61/2) 888-4444
Fax: (61/2) 888-9072
Cable: HEWPARD Sydney

AUSTRIA
Hewlett-Packard GmbH
Verkaufsbüro Graz
Grottenhofstrasse 94
8052 **GRAZ**
Tel: (43/316) 28 3066
Fax: (43/316) 28 1756
Telex: 312375

Hewlett-Packard GmbH
Lieblgasse 1
P.O. Box 72
1222 **VIENNA**
Tel: (43/222) 2500-0
Fax: (43/222) 2500 - 444

BAHRAIN
Wael Pharmacy and Drugstore
P.O. Box 648
MANAMA
Tel: (973) 25 61 23
Fax: (973) 23 36 38
Telex: 8550 WAEL BN

SALES OFFICES
Arranged alphabetically by country (cont'd)

BARBADOS
Computers & Controls (Barbados) Ltd.
Suite 5, 1st Floor
Wildey Shopping Plaza
ST. MICHAEL
Tel: (809) 429-4103
Fax: (809) 429-4103

BELGIUM
Hewlett-Packard Belgium S.A./N.V.
Blvd de la Woluwe, 100
Woluwedal
1200 **BRUSSELS**
Tel: (32/2) 761 31 11
Fax: (32/2) 763 06 13

BENIN
Engineering Business Concept (E.B.C)
21, Route du Canal
Zone 3
04 B.P. 1357
ABIDJAN
Ivory Coast
Tel: (225) 21 50 24
Fax: (225) 35 37 90

BERMUDA
Applied Computer Technologies
17 Reid St.
HAMILTON 5
Tel: (809) 295-1616
Fax: (809) 292-7967
Telex: 380 3589/ACT BA

BOLIVIA
Siser Ltda. (Sistemas de Importacion y Servicios Ltda.)
Gabriel Gozalvez 221
Casilla 4084
LA PAZ
Tel: (591/2) 340962/ 363365/343245
Fax: (591/2) 359268

BRAZIL
Edisa Informatica S.A.
Alameda Rio Negro, 750-Alphaville
06454-**BARUERI**-SP
Tel: (55/11) 709-1444
Fax: (55/11) 709-1244
Telex: 1171351

Edisa Informatica S.A.
Rua dos Andradas, 1001-10 andar
90020-**PORTO ALEGRE**-RS
Tel: (55/51) 225-7166
Fax: (55/51) 225-4060
Telex: 515663

Edisa Informatica S.A.
Av. Eng. Abdias de Carvalho, 1111-5 andar
50751-**RECIFE**-PE
Tel: (55/81) 227-2722
Fax: (55/81) 228-3793
Telex: 812904

Edisa Informatica S.A.
Praia de Botafogo, 228-4 andar
22250-**RIO DE JANEIRO**-RJ
Tel: (55/21) 552-0222
Fax: (55/21) 551-1449
Telex: 2121905

BRUNEI
Komputer Wisman Sdn Bhd
Room 1, 3rd Floor
Jalan Tutong
BS Begawan
BRUNEI DARUSSALEM
Tel: (673/2) 23918
Telex: 0809 2447

BURKINA FASSO
Engineering Business Concept (E.B.C.)
21, Route du Canal
04 B.P. 1357
ABIDJAN
Ivory Coast
Tel: (225) 21 50 24
Fax: (225) 35 37 90

CANADA

Alberta
Hewlett-Packard (Canada) Ltd.
150 - 6th Avenue S.W.
CALGARY, Alberta T2P 3Y7
Tel: (403) 262-0777
Fax: (403) 237-9309

Hewlett-Packard (Canada) Ltd.
11120-178th Street
EDMONTON, Alberta T5S 1P2
Tel: (403) 486-6666
Fax: (403) 489-8764

British Columbia
Hewlett-Packard (Canada) Ltd.
10691 Shellbridge Way
RICHMOND,
British Columbia V6X 2W8
Tel: (604) 270-2277
Fax: (604) 270-0859

Manitoba
Hewlett-Packard (Canada) Ltd.
1825 Inkster Blvd.
WINNIPEG, Manitoba R2X 1R3
Tel: (204) 694-2777
Fax: (204) 694-3901

Nova Scotia
Hewlett-Packard (Canada) Ltd.
201 Brownlow Avenue
DARTMOUTH, Nova Scotia B3B 1W2
Tel: (902) 468-4725
Fax: (902) 468-2817

Hewlett-Packard (Canada) Ltd.
475 Hood Rd., Unit #2
MARKHAM, Ontario L3R 8H1
Tel: (416) 479-1770
Fax: (416) 479-3105

Ontario
Hewlett-Packard (Canada) Ltd.
552 Newbold Street
LONDON, Ontario N6E 2S5
Tel: (519) 686-9181
Fax: (519) 686-9145

Hewlett-Packard (Canada) Ltd.
6877 Goreway Drive
MISSISSAUGA, Ontario L4V 1M8
Tel: (416) 678-9430
Fax: (416) 678-9421

Hewlett-Packard (Canada) Ltd.
2670 Queensview Dr.
OTTAWA, Ontario K2B 8K1
Tel: (613) 820-6483
Fax: (613) 820-0377

Quebec
Hewlett-Packard (Canada) Ltd.
17500 Trans Canada Highway
South Service Road
KIRKLAND, Quebec H9J 2X8
Tel: (514) 697-4232
Fax: (514) 697-6941

Saskatchewan
Hewlett-Packard (Canada) Ltd.
#1, 2175 Airport Rd.
SASKATOON, Saskatchewan S7L 7E1
Tel: (306) 242-3702

CHILE
Avanzados Sistemas de Conocimientos S. A. (ASC)
Austria 2041
SANTIAGO
Tel: (56/2) 223-5946/6148
Fax: (56/2) 223-1912

CHINA, PEOPLE'S REPUBLIC OF
China Hewlett-Packard Co., Ltd.
38 Bei San Huan Xi Road
Shuang Yu Shu, Hai Dian District
P.O. Box 9610
BEIJING
Tel: (86/1) 256-6888
Fax: (86/1) 256-3207

China Hewlett-Packard Co., Ltd.
28/F Shanghai Union Building
100 Yan An Dong Road
SHANGHAI 200002
Tel: (86/21) 320-3240
Fax: (86/21) 320-2149

COLOMBIA
Carbajal S.A.
Calle 29 Norte No. 6 A 40
Apartado Aereo 46
CALI
Tel: (57/23) 675 011
Fax: (57/23) 688 466

COSTA RICA
I.S. de Costa Rica S.A.
Calle 25, Avs. 6 y 8 No.648
SAN JOSE
Tel: (50/6) 33 3722
Fax: (50/6) 55 3528

CYPRUS
Telerexa Ltd.
P.O. Box 1152
Valentine House
8 Stassandrou St.
NICOSIA
Tel: (357/2) 445 628
Telex: 0605 5845 tlrx cy

DENMARK
Hewlett-Packard A/S
Birkerød Kongevejen 25
3460 **BIRKERØD**
Tel: (45/42) 81 66 40
Fax: (45/42) 81 58 10

Hewlett-Packard A/S
Voldbjergvej 16
8240 **RISSKOV,** Aarhus
Tel: (45/06) 17 60 00
Fax: (45/06) 17 60 58
Telex: 37409 hpas dk

DOMINICAN REPUBLIC
Esacomp S.A.
Edificio Mercantil del Caribe
Ave. John F. Kennedy No. 11
P.O. Box 1496
SANTO DOMINGO
Tel: (809) 567 3241
Fax: (809) 566 9774
Telex: ITT 346 0439

ECUADOR
CYEDE Cia. Ltda.
Avenida Eloy Alfaro 1749
y Belgica
Casilla 6423 CCI
QUITO
Tel: (593) 245-0075
Fax: (593) 224-4223
Telex: 39322548 CYEDE ED

EGYPT
International Engineering Associates
6 El Gamea Street
AGUZA/CAIRO
Tel: (20/2) 71 21 68
Telex: 93830 IEA UN
Cable: INTEGASSO

EL SALVADOR
IPESA de El Salvador S.A.
29 Avenida Norte 1223
SAN SALVADOR
Tel: (503) 266 858
Telex: 301 20539 IPESA SAL

FINLAND
Hewlett-Packard Oy
Piispankalliontie 17
02200 **ESPOO (Helsinki)**
Tel: (358/0) 887 21
Fax: (358/0) 887 22 77

Hewlett-Packard Oy
Välinönkatu 9 C
40100 **JYVÄSKYLÄ**
Tel: (358/41) 21 85 11

Hewlett-Packard Oy
Valtatie 57
90500 **OULU**
Tel: (358/81) 340 144
Fax: (358/81) 340 145

FRANCE

Aix-En-Provence
Hewlett-Packard France
ZI Mercure B
Rue Berthelot
13763 **LES MILLES** Cédex
Tel: (33) 42 24 32 43
Fax: (33) 42 59 48 72
Telex: 410 770

Bordeaux
Hewlett-Packard France
Domaine de Pélus
5, avenue de Pythagore
33700 **MERIGNAC**
Tel: (33) 56 34 00 84
Fax: (33) 56 34 80 84
Telex: 550 105

Brest
Hewlett-Packard France
ZAC Kergaradec
8, rue Fernand Forest
29239 **GOUESNOU**
Tel: (33) 98 41 87 90
Fax: (33) 98 41 74 77

Grenoble
Hewlett-Packard France
57, chemin du Vieux Chêne
ZIRST
38240 **MEYLAN**
Tel: (33) 76 90 38 40
Fax: (33) 76 41 05 36

Lille
Hewlett-Packard France
Parc d'activités des Prés
1, rue Papin
59658 **VILLENEUVE D'ASCQ**
Tel: (33) 20 47 78 78
Fax: (33) 20 33 36 77
Telex: 160 124

Lyon
Hewlett-Packard France
Chemin des Mouilles
BP 162
69131 **ECULLY** Cédex
Tel: (33) 72 29 32 93
Fax: (33) 78 33 49 82
Telex: 310 617

Nice
Hewlett-Packard France
Les Cardoulines - Bât. 2
Route des Dolines
Sophia Antipolis
06560 **VALBONNE**
Tel: (33) 93 65 39 40
Fax: (33) 93 65 31 34

Orléans
Hewlett-Packard France
Parc Tertiaire Héliopolis
Route de Micy
45380 **LA CHAPELLE ST MESMIN**
Tel: (33) 38 43 94 56
Fax: (33) 38 88 22 81
Telex: 783 497

Orsay
Hewlett-Packard France
Zone Industrielle de Courtaboeuf
1, avenue du Canada
91947 **LES ULIS** Cédex
Tel: (33/1) 69 82 60 60
Fax: (33/1) 69 82 60 61
Telex: 600 048

Rennes
Hewlett-Packard France
Parc d'activités de la Poterie
Rue Louis Kerautret-Botmel
35000 **RENNES**
Tel: (33) 99 51 42 44
Fax: (33) 99 32 29 19
Telex: 740 912

Rouen
Hewlett-Packard France
PAT Lavatine
3, rue Jacques Monod
BP 228
76136 **MONT-ST-AIGNAN**
Tel: (33) 35 59 19 20
Fax: (33) 35 59 85 11
Telex: 770 035

Strasbourg
Hewlett-Packard France
4, rue de la Faisanderie
Parc Club des Tânneries
BP 40
67381 **LINGOLSHEIM** Cédex
Tel: (33) 88 76 15 00
Fax: (33) 88 78 17 63
Telex: 890 141

Toulouse
Hewlett-Packard France
Innoparc
BP 167 Voie n° 7
31328 **LABEGE** Cédex
Tel: (33) 61 39 11 40
Fax: (33) 61 39 10 97
Telex: 531 639

GERMAN FEDERAL REPUBLIC

HEADQUARTERS
Hewlett-Packard GmbH
Herrenberger Strasse 130
7030 **BÖBLINGEN**
Tel: (49/7031) 14-0
Fax: (49/7031) 14-2999

Hewlett-Packard GmbH
Bad Homburg
Hewlett-Packard-Strasse
6380 **BAD HOMBURG**
Tel: (49) 6172 16-0
Fax: (49) 6172 16-1309
Telex: 410 844 hpbhg

Berlin
Hewlett-Packard GmbH
Lützowplatz 15
1000 **BERLIN** 30
Tel: (49/30) 2500 02-0
Fax: (49/30) 2500 02-62
Telex: 018 3405 hpbln d

Böblingen
Hewlett-Packard GmbH
Schickardstrasse 2
7030 **BÖBLINGEN**
Tel: (49/7031) 645-0
Fax: (49/7031) 645-429

Bonn
Hewlett-Packard GmbH
Friedrich-Ebert-Allee 26
5300 **BONN** 1
Tel: (49/228) 23400-1
Fax: (49/228) 235 315

Dortmund
Hewlett-Packard GmbH
Schleefstrasse 28
4600 **DORTMUND**-Aplerbeck
Tel: (49/231) 45001-0
Fax: (49/231) 45001-37
Telex: 822858 hepdod

Frankfurt
Hewlett-Packard GmbH
Reparaturzentrum Frankfurt
Berner Strasse 117
6000 **FRANKFURT 56**
Tel: (49/69) 5000 60
Fax: (49/69) 5000 6-200
Telex: 412 815 hpffm

Hamburg
Hewlett-Packard GmbH
Kapstadtring 5
2000 **HAMBURG** 60
Tel: (49/40) 63804-0
Fax: (49/40) 63804-327
Telex: 2163032

Hannover
Hewlett-Packard GmbH
Heidering 37-39
3000 **HANNOVER** 61
Tel: (49/511) 5706-0
Fax: (49/511) 5706-126

Mannheim
Hewlett-Packard GmbH
Rosslauer Weg 2-4
6800 **MANNHEIM** 31
Tel: (49/621) 7005-0
Fax: (49/621) 7005-200
Telex: 462105

Munchen (Munich)
Hewlett-Packard GmbH
Eschenstrasse 5
8028 **TAUFKIRCHEN**
Tel: (49/89) 61207-0
Fax: (49/89) 61207-300
Telex: 0524985 hpmch

Nürnberg
Hewlett-Packard GmbH
Emmericher Strasse 13
8500 **NÜRNBERG** 10
Tel: (49/911) 5205-0
Fax: (49/911) 5205-140
Telex: 623860

Ratingen
Hewlett-Packard GmbH
Berliner Strasse 111
4030 **RATINGEN** 1
Tel: (49/2102) 494-0
Fax: (49/2102) 494-300
Telex: 8589070

Ulm
Hewlett-Packard GmbH
Messerschmittstrasse 7
7910 **NEU ULM**
Tel: (49/731) 7073-0
Fax: (49/731) 7076-66
Telex: 712816

Waldbronn
Hewlett-Packard GmbH
Ermlis-Allee
7517 **WALDBRONN** 2 (Karlsruhe)
Tel: (49/7243) 602-0
Fax: (49/7243) 602-512
Telex: 7265743

GHANA
Engineering Business Concept (E.B.C.)
21, Route du Canal
Zone 3
04 B.P. 1357
ABIDJAN
Ivory Coast
Tel: (225) 31 50 24
Fax: (225) 35 37 90

GREAT BRITAIN
See United Kingdom

GREECE
Hewlett-Packard Hellas
32, Kiffissias Avenue
15125 Amaroussion
ATHENS
Greece
Tel: (30/1) 682 88 11
Fax: (30/1) 683 29 78
Telex: 216588 hpat gr

GUATEMALA
Ipesa de Guatemala
Avenida Reforma 3-48, Zona 9
GUATEMALA CITY
Tel: (502) 231-7853
Fax: (502) 231-6627
Telex: 3055765 IPESA GU

HONG KONG
Hewlett-Packard Asia, Ltd.
22/F, Bond Centre, West Tower
89 Queensway Central
HONG KONG
Tel: (852/5) 848-7777
Fax: (852/5) 868-4997
Cable: HEWPACK HONG KONG

SALES OFFICES
Arranged alphabetically by country (cont'd)

HUNGARY
Hewlett-Packard
Accredite Office Hungary
Radvany u. 7
1118 **BUDAPEST**
Tel: (36/1) 185 23 68
Fax: (36/1) 165 10 85
Telex: 861 227 632

ICELAND
Hewlett-Packard Iceland
Høfdabakka 9
110 **REYKJAVIK**
Tel: (354/1) 67-1000
Fax: (354/1) 67-3031
Telex: 37409

INDIA

Bangalore
Hewlett-Packard India Pvt. Ltd.
29 Cunningham Road
BANGALORE 560 052
Tel: (91/812) 261075
Fax: (91/812) 261554

Bombay
Hewlett-Packard India Pvt. Ltd.
Sahas
414/2 Veer Savarkar Marg
Prabhadevi
BOMBAY 400 025
Tel: (91/22) 4306155
Fax: (91/22) 4307078

Calcutta
Hewlett-Packard India Pvt. Ltd.
DBS Executive Centre
8 Acharya J. C. Bose Road
CALCUTTA 700 017
Tel: (91/33) 444990
Fax: (91/33) 444614

Hyderabad
Hewlett-Packard India Pvt. Ltd.
9-5-13 Taramandal Complex
9th Floor
Saifabad
HYDERABAD 500 004
Tel: (91/842) 231756
Fax: (91/842) 8313444

New Delhi
Hewlett-Packard India Pvt. Ltd.
B-8 Jangpura
Mathura Road
NEW DELHI 110 014
Tel: (91/11) 690329
Fax: (91/11) 353315

INDONESIA
BERCA Indonesia P.T.
Wisma Dharmala Sakti
10/F, J1, Jendral Sudirman K.av 32
P.O. Box 41/JKPDS
JAKARTA 10001
Tel: (62/21) 578-0005
Fax: (62/21) 570-1287
Telex: 62065 BERCAMIA

IRAQ
Hewlett-Packard Trading S.A.
(Service Operation)
Al Mansoor City 609/10/7
BAGHDAD
Tel: (964/1) 541 49 73
Fax: (964/1) 541 49 73
Telex: 212455 hepairagik

IRELAND
Hewlett-Packard Ireland Ltd.
Temple House, Temple Road
Blackrock, Co. **DUBLIN**
Tel: (353/1) 883399
Fax: (353/1) 883742
Telex: 30439

ISRAEL
Computation and Measurement
Systems (CMS) Ltd.
11, Hashlosha Street
TEL-AVIV 67060
Tel: (972) 3 5380-333
Fax: (972) 3 5375-055
Telex: 371234 HPCMS

ITALY

Anzola Emilia-Bologna
Hewlett-Packard Italiana S.p.A.
Via Emilia, 51/C
40011 **ANZOLA EMILIA-BOLOGNA**
Tel: (39/51) 73 10 61
Fax: (39/51) 73 48 30
Telex: 51 16 30

Bari
Hewlett-Packard Italiana S.p.A.
Via Vitantonio di Cagno, 34
70124 **BARI**
Tel: (39/80) 41 07 44
Fax: (39/80) 41 78 51

Catania
Hewlett-Packard Italiana S.p.A.
Via Principe Nicola, 43 G/C
95126 **CATANIA**
Tel: (39/95) 37 10 87
Fax: (39/95) 38 85 69

Corsico
Hewlett-Packard Italiana S.p.A.
Via G. di Vittorio, 10
20094 **CORSICO** (MI)
Tel: (39/2) 440 83 51
Fax: (39/2) 440 95 64
Telex: 440 9564

Firenze
Hewlett-Packard Italiana S.p.A.
Via Sacco e Vanzetti, 1
50145 **FIRENZE**
Tel: (39/55) 31 85 53
Fax: (39/55) 37 39 65

Genova
Hewlett-Packard Italiana S.p.A.
Viale Brigata Bisagno, 2
16129 **GENOVA**
Tel: (39/10) 54 11 41
Fax: (39/10) 59 17 33
Telex: 28 52 38

Limito
Hewlett-Packard Italiana S.p.A.
Via Nuova Rivoltana, 95
20090 **LIMITO** (MI)
Tel: (39/2) 757 61
Fax: (39/2) 757 6230
Telex: 32 31 16

Napoli
Hewlett-Packard Italiana S.p.A.
Via Orazio, 16
80122 **NAPOLI**
Tel: (39/81) 761 14 44
Fax: (39/81) 68 01 64
Telex: 71 06 98

Padova
Hewlett-Packard Italiana S.p.A.
Via Pellizzo, 15
35128 **PADOVA**
Tel: (39/49) 807 01 66
Fax: (39/49) 77 30 97
Telex: 43 03 15

Roma-Eur
Hewlett-Packard Italiana S.p.A.
Via del Tintoretto, 200
00142 **ROMA-EUR**
Tel: (39/6) 5 48 31
Fax: (39/6) 540 87 10
Telex: 62 65 24

Torino
Hewlett-Packard Italiana S.p.A.
C.so Svizzera, 185
10149 **TORINO**
Tel: (39/11) 74 40 44
Fax: (39/11) 771 08 15
Telex: 22 10 79

IVORY COAST
Engineering Business Concept (E.B.C.)
Angle Avenue J. Anoma et Bd.
République
08 B.P. 323 **ABIDJAN** 08
Tel: (225) 32 50 24/41 48 70
Fax: (225) 35 37 90

JAPAN

Akita
Yokogawa-Hewlett-Packard Ltd.
Nihonseimei Akita Chudori Bldg.
4-2-7 Nakadori
AKITA 010
Tel: (81/188) 36-5021
Fax: (81/188) 36-5099

Atsugi
Yokogawa-Hewlett-Packard Ltd.
Keno Kikaku Kogyo No. 2 Bldg.
9-32 Tamuracho
ATSUGI 243
Tel: (81/462) 25-0031
Fax: (81/462) 25-0064

Chiba
Yokogawa-Hewlett-Packard Ltd.
Fujimoto Daiichi Seimei Bldg.
3-3-1 Chuo
CHIBA 280
Tel: (81/472) 25-7701
Fax: (81/472) 21-0382

Fukuoka
Yokogawa-Hewlett-Packard Ltd.
Daisan Hakata-Kaisei Bldg.
1-3-6 Hakataekiminami Hakata-ku
FUKUOKA 812
Tel: (81/92) 472-8731
Fax: (81/92) 473-4745

Hiroshima
Yokogawa-Hewlett-Packard Ltd.
Yasuda-semei Hiroshima Bldg.
6-11 Hondori Naka-ku
HIROSHIMA 730
Tel: (81/82) 241-0611
Fax: (81/82) 241-0619

Kobe
Yokogawa-Hewlett-Packard Ltd.
Towa Building
2-2-3 Kaigandori, Chuo-ku
KOBE 650
Tel: (81/78) 392-4791
Fax: (81/78) 392-4839

Koriyama
Yokogawa-Hewlett-Packard Ltd.
Nihon-Dantai-seimei Bldg.
21-10 Toramaru-machi
KORIYAMA 963
Tel: (81/249) 39-7111
Fax: (81/249) 39-7234

Kumagaya
Yokogawa-Hewlett-Packard Ltd.
Kumagaya Asahi 82 Bldg.
3-4 Tsukuba
KUMAGAYA 360
Tel: (81/485) 24-6563
Fax: (81/485) 24-9050

Kyoto
Yokogawa-Hewlett-Packard Ltd.
Shin-Kyoto Center Bldg.
614 Higashi-Shokoji-cho
Karasuma-nishi-iru, Shiokoji-dori
Shimogyo-ku
KYOTO 600
Tel: (81/75) 343-0921
Fax: (81/75) 343-4356

Mito
Yokogawa-Hewlett-Packard Ltd.
Mito Mitsui Bldg.
1-4-73 Sannomaru
MITO 310
Tel: (81/292) 25-7470
Fax: (81/292) 31-6589

Miyazaki
Yokogawa-Hewlett-Packard Ltd.
Nomura Shoken Bldg.
4-1-2 Tachibana-dori
Miyazaki-C.
MIYAZAKI 880
Tel: (81/985) 23-7280
Fax: (81/985) 23-7864

Nagano
Yokogawa-Hewlett-Packard Ltd.
Nagano-Toiyo-kaijo Bldg.
1081, Minamiagatamachi
NAGANO 380
Tel: (81/262) 24-8012
Fax: (81/262) 24-8016

Nagoya
Yokogawa-Hewlett-Packard Ltd.
Nagoya Kokusai Center Bldg.
1-47-1 Nakono, Nakamura-ku
NAGOYA 450
Tel: (81/52) 571-2262
Fax: (81/52) 565-0896

Nara
Yokogawa-Hewlett-Packard Ltd.
Nissei Nara Ekimae Bldg.
1-1-15 Omiya-cho
NARA 630
Tel: (81/742) 22-8235
Fax: (81/742) 22-8219

Oomiya
Yokogawa-Hewlett-Packard Ltd.
Saikyoren Bldg.
1-2 Dotemachi
OOMIYA 330
Tel: (81/48) 645-8031
Fax: (81/48) 645-8001

Osaka
Yokogawa-Hewlett-Packard Ltd.
Chuo Bldg.
5-4-20 Nishinakajima
Yodogawa-ku
OSAKA 532
Tel: (81/6) 304-6021
Fax: (81/6) 304-0216

Sapporo
Yokogawa-Hewlett-Packard Ltd.
Sumitomo-seimei Sapporo Chu Bldg.
1-1-14 Nanjo-higashi Chuo-ku
SAPPORO 060
Tel: (81/11) 251-1011
Fax: (81/11) 222-3239

Sendai
Yokogawa-Hewlett-Packard Ltd.
Yamaguchi Bldg.
1-1-31 Ichibancho Aoba-ku
SENDAI 980
Tel: (81/22) 225-1011
Fax: (81/22) 225-1616

Suwa-gun
Yokogawa-Hewlett-Packard Ltd.
Daimon Bldg.
139-1 Yaginishi Shimosuwa-cho
SUWA-GUN 393
Tel: (81/266) 28-8851
Fax: (81/266) 28-8873

Tokyo
Yokogawa-Hewlett-Packard Ltd.
Shinjuku Daiichi-seimei Bldg.
2-7-1 Nishi-shinjuku
Shinjuku-ku
TOKYO 160
Tel: (81/3) 342-2734
Fax: (81/3) 348-7969

Yokogawa-Hewlett-Packard Ltd.
9-1 Takakura-cho
Hachioji-shi
TOKYO 192
Tel: (81/426) 42-1231
Fax: (81/426) 45-2631

Yokogawa-Hewlett-Packard Ltd.
Tokyo-Nissan Bldg.
1-6-34 Konan, Minato-ku
TOKYO, 108
Tel: (81/3) 458-5411
Fax: (81/3) 458-5400

Yokogawa-Hewlett-Packard Ltd.
3-29-21 Takaido-higashi
Suginami-ku
TOKYO 168
Tel: (81/3) 331-6111
Telex: 232 2024 YHPTOK

Yokogawa-Hewlett-Packard Ltd.
Tokyu Sakuragaokacho Bldg.
31-2 Sakuragaoka Shibuya-ku
TOKYO 150
Tel: (81/3) 780-5511
Fax: (81/3) 780-5510

Toyota
Yokogawa-Hewlett-Packard Ltd.
Toyota-Tokyo-Kaijo Bldg.
1-179 Miyuki-honmachi
TOYOTA 473
Tel: (81/565) 27-5611
Fax: (81/565) 27-5400

Tsukuba
Yokogawa-Hewlett-Packard Ltd.
Issei Bldg.
2-3-17 Takezono
TSUKUBA 305
Tel: (81/298) 51-5141
Fax: (81/298) 51-5381

Utsunomiya
Yokogawa-Hewlett-Packard Ltd.
Chiyodaseimei-Utsunomiya Bldg.
2-3-1 Odori
UTSUNOMIYA 320
Tel: (81/286) 33-1153
Fax: (81/286) 33-1175

Yokohama
Yokogawa-Hewlett-Packard Ltd.
No. 2 Yasuda Bldg.
2-32-12 Tsuruyacho
Kanagawa-ku
YOKOHAMA-SHI KANAGAWA
Tel: (81/45) 313-1352
Fax: (81/45) 312-1846

JORDAN
Scientific & Medical Supplies Co.
P.O. Box 1387
AMMAN
Tel: (962/6) 62 49 07
Fax: (962/6) 62 82 58
Telex: 0493 21456 sabco jo

KENYA
ADCOM Ltd., Inc., Kenya
P.O. Box 30070
NAIROBI
Tel: (254/2) 33 1955
Telex: 22639

KOREA
Samsung Hewlett-Packard Co. Ltd.
Dongbang Building, 12-16/F
36-1 Yeoeuido-dong
Youngdeungpo-Ku
SEOUL, 150
Tel: (82/2) 769-0114
Fax: (82/2) 784-7084
Telex: SAM HP K 25166

Samsung Hewlett-Packard Co. Ltd.
13/F Taegu Bank Bldg.
118, 2-Ka
Suseong-Dong
Suseong-Ku
TAEGU
Tel: (82/53) 754-2666
Fax: (82/53) 752-4696

Samsung Hewlett-Packard Co. Ltd.
8/F, Daehankyoyuk Insurance Bldg.
382-1 Sun Hwa-Dong
Chung-Gu
TEEJON 301-050
Tel: (82/42) 256-6725
Fax: (82/42) 256-6727

KUWAIT
Al-Khaldiya Trading & Contracting
P.O. Box 830
SAFAT 13009
Tel: (965) 242 49 10, 241 17 26
Fax: (965) 245 20 67
Telex: 22481 AREEG KT
Cable: VISCOUNT

LEBANON
Computer Information Systems S.A.L.
Chammas Building
P.O. Box 11-6274
DORA-BEIRUT
Tel: (961/1) 89 31 13
Fax: (961/1) 58 18 34
Telex: 42309 chacis le

LUXEMBOURG
Hewlett-Packard Belgium S.A./N.V.
Blvd de la Woluwe, 100
Woluwedal
1200 **BRUSSELS**
Tel: (32/2) 761-3111
Fax: (32/2) 763-0613
Telex: 23 494 palobe b

MALAWI
Field Consolidated (Private) Ltd.
System Division
Manhattan Court
61 Second Street
P.O. Box 3458
HARARE
Zimbabwe
Tel: (263/4) 73 98 81
Fax: (263/4) 70 20 08
Telex: 0907 26241

MALAYSIA
Hewlett-Packard Sales (M) SDN BHD
9/F, Chung Khiaw Bank Building
Section 46, Jalan Raja Laut
GPO Box 11119
50736 **KUALA LUMPUR**
Tel: (60/3) 298-6555
Fax: (60/3) 291-5495

Hewlett-Packard Sales (M) SDN BHD
6/F, Wisma Penang Garden
42, Jalan Sultan Ahmad Shah
10050 **PENANG**
Tel: (60/4) 376-532
Fax: (60/4) 371-027

MEXICO
Hewlett-Packard de Mexico,
S.A. de C.V.
Rio Nio No. 4049 Desp. 12
Fracc. Cordoba
CIUDAD JUAREZ, Mexico
Tel: (52/161) 3 15 62

Hewlett-Packard de Mexico,
S.A. de C.V.
Condominio Kadereyta
Circuito del Mezon No. 186 Desp. 6
Col. Del Prado
76030 **QUERETARO**, QRO
Tel: (52/4) 636-0271

Hewlett-Packard de Mexico,
S.A. de C.V.
Czda. del Valle
409 Ote. 4th Piso
Colonia del Valle
Municipio de Garcia
66220 **GARZA GARCIA**, N.L.
Tel: (52/8) 378-4240
Telex: 382410 HPMY

Hewlett-Packard de Mexico,
S.A. de C.V.
Monti Morelos No. 299
Fraccionamiento Loma Bonita 45060
GUADALAJARA, Jalisco
Tel: (52/3) 631-4600
Telex: 0684 186 ECOME

Hewlett-Packard de Mexico,
S.A. de C.V.
Monte Pelvoux No. 111
Lomas de Chapultepec
11000 **MEXICO**, D.F.
Tel: (52/5) 202-0155
Fax: (52/5) 540-4208

Hewlett-Packard de Mexico,
S.A. de C.V.
Blvd. Independencia
No. 2000 Ote. Piso 3
Col. Estrella
27010 **TORREON**, COA.
Tel: (52/1) 718-2201

MOROCCO
SICOTEL
Complexe des Habous
Tour C, avenue des Far
CASABLANCA 01
Tel: (212) 31 22 70
Telex: 0407 27604

NETHERLANDS
Hewlett-Packard Nederland B.V.
Startbaan 16
P.O. Box 667
1180 NL **AMSTELVEEN**
Tel: (31/20) 547 69 11
Fax: (31/20) 471 825
Telex: 13 216 HEPA NL

SALES OFFICES
Arranged alphabetically by country (cont'd)

NETHERLANDS (cont'd)
Hewlett-Packard Nederland B.V.
Boschdijk 131
P.O. Box 2342
5612 HB **EINDHOVEN**
Tel: (31/40) 32 69 11
Fax: (31/40) 44 65 46
Telex: 51484 hepae nl

NEW ZEALAND
Hewlett-Packard (N.Z.) Ltd.
5 Owens Road
Epsom, **AUCKLAND** 3
Tel: (64/9) 687-159
Fax: (64/9) 600-507

Hewlett-Packard (N.Z.) Ltd.
186-190 Willis Street
WELLINGTON 1
Tel: (64/4) 820-400
Fax: (64/4) 843-380

NIGER
Engineering Business Concept (E.B.C.)
21, Route du Canal
Zone 3
04 B.P. 1357
ABIDJAN
Ivory Coast
Tel: (225) 21 50 24
Fax: (225) 35 37 90

NIGERIA
Management Information Systems Co., Ltd.
3, Gerrard Road, Ikoyi
LAGOS
Tel: (234/1) 68 08 87
Fax: (234/1) 68 54 87
Telex: 23582

NORTHERN IRELAND
See United Kingdom

NORWAY
Hewlett-Packard Norge A/S
Oesterndalen 16-18
P.O. Box 34
N-1345 **OEŠTERAAS**
Tel: (47/2) 24-60 90
Fax: (47/2) 24 66 96, 24 30 78
Telex: 76621 HPNAS N

Hewlett-Packard Norge A/S
Boemergt. 42
Box 2470
N-5037 **SOLHEIMSVIK**
Tel: (47/5) 29 00 90
Fax: (47/5) 29 10 72

OMAN
Imtac LLC
P.O. Box 9196
MINA AL FAHAL
Tel: (968) 70-77-27, 70-77-23
Fax: (968) 79 66 39
Telex: 0498 3865

PAKISTAN
Mushko Electronics (PVT) Ltd.
68-W Sama Plaza
Blue Area, F-7
ISLAMABAD
Tel: (92/51) 814 492
Fax: (92/51) 821 201
Telex: 54001 Muski Pk

Mushko Electronics (PVT) Ltd.
Victoria Chambers
Abdullah Haroon Road
KARACHI
Tel: (92/21) 524 131, 524 132
Fax: (92/21) 512 298
Telex: 2894 MUSKO PK

PANAMA
Electronico Balboa, S.A.
Calle Samuel Lewis, Ed. Alfa
Apartado 4929
PANAMA CITY
Tel: (507) 636 613
Fax: (507) 361 820
Telex: 368 3483 ELECTRON PG

PERU
Cía Electro Médica, S.A.
Los Flamencos No. 145, Ofna. 301 & 302
San Isidro, **LIMA** 1
Tel: (51/14) 41-4325, 41-3703
Fax: (51/14) 42-3572
Telex: 39425257 PE PB SIS

PHILIPPINES
The Online Advanced Systems Corp.
2nd Floor, Electra House
115-117 Esteban Street
Legaspi Village, Makati
Metro **MANILA**
Tel: (63/2) 818-4394
Fax: (63/2) 816-0590
Telex: 63274 ONLINE PN

PORTUGAL
Hewlett-Packard Portugal S.A.
Torre de Santo Antonio
Rua Gregorio Lopes, Lote 1732B
Restelo
1400 **LISBON**
Tel: (351) 617 343/44/46
Fax: (351) 617345
Telex: 43126

Hewlett-Packard Portugal S.A.
Edificio Tranquilidade
Rua D. Manuel, 296
4000 **PORTO**
Tel: (351) 493 122
Telex: 26054
Fax: (351) 488 721

PUERTO RICO
Hewlett-Packard Puerto Rico
P.O. Box 4048
AGUADILLA 00605
Tel: (809) 890-6000

Hewlett-Packard Puerto Rico
101 Munoz Rivera Avenue
Esu. Calle Ochoa
HATO REY, 00918
Tel: (809) 754-7800

QATAR
Qatar Datamation Systems
P.O. Box 350
DOHA
Tel: (974) 44 19 16
Fax: (974) 44 12 19
Telex: 4833

SAUDI ARABIA
Modern Electronic Establishment
P.O. Box 281
Thouqbah
AL-KHOBAR 31952
Tel: (966/3) 895-1760, 895-1764
Telex: 0495 671 106 HPMEEK SJ
Cable: ELECTA AL-KHOBAR

Modern Electronic Establishment
P.O. Box 1228
Redec Plaza, 6th Floor
JEDDAH 21431
Tel: (966/2) 644 96 28
Fax: (966/2) 644 18 33
Cable: ELECTA JEDDAH

Modern Electronic Establishment
P.O.Box 22015
RIYADH 11495
Tel: (966/1) 476 30 30
Fax: (966/1) 476 76 49
Telex: 402049 MEERYD SJ

SCOTLAND
See United Kingdom

SINGAPORE
Hewlett-Packard Singapore Ltd.
150 Beach Road
#29-00 Gateway West
PO Box 2, Bras Basah Post Office
SINGAPORE, 9118
Tel: (65) 291 9088
Fax: (65) 292 7089
Telex: RS 34209

SOUTH AFRICA
Hi Performance Systems (Pty.) Ltd.
P.O. Box 120, Howard Place
CAPE TOWN 7450
Tel: (27/21) 53-7954
Fax: (27/21) 53-5119

Hi Performance Systems (Pty.) Ltd.
Private Bag Wendywood
SANDTON 2144
Tel: (27/11) 802-5111
Fax: (27/11) 802-6332

SPAIN

Barcelona
Hewlett-Packard Española S.A.
Avda. Diagonal, 605
08028 **BARCELONA**
Tel: (34/3) 401 91 00
Fax: (34/3) 230 84 68
Telex: 52603 hpbee

Bilbao
Hewlett-Packard Española, S.A.
Avda. de Zugazarte, 8
(Edificio El Abra 4)
48930 **LAS ARENAS-GUECHO**
Tel: (34/4) 464 32 55
Fax: (34/4) 464 90 83
Telex: 33032

Madrid
Hewlett-Packard Española S.A.
Crta. de la Coruña, km 16,500
28230 Las Rozas
MADRID
Tel: (34/1) 637 00 11
Fax: (34/1) 637 55 62
Telex: 23515 HPE

Sevilla
Hewlett-Packard Española S.A.
c/Luis de Morales, S/N
Edificio Forum
41005 **SEVILLA**
Tel: (34/5) 458 17 00
Fax: (34/5) 457 57 20
Telex: 72933

Valencia
Hewlett-Packard Española S.A.
Isabel La Católica, 8
46004 **VALENCIA**
Tel: (34/6) 351 59 44
Telex: 63435
Fax: (34/6) 351 49 06

SWEDEN

Göteborg
Hewlett-Packard Sverige AB
Topasgatan 1A
Box 266
42123 **VÄSTRA-FRÖLUNDA**
Tel: (46/31) 89 10 00
Fax: (46/31) 49 50 99

Malmö
Hewlett-Packard Sverige AB
Östra Tullgatan 3
20011 **MALMÖ**
Box 6132
Tel: (46/40) 702 70
Fax: (46/40) 97 74 18

Örebro
Hewlett-Packard Sverige AB
Elementvägen 16
70227 **ÖREBRO**
Tel: (46/19) 10 48 80
Fax: (46/19) 27 01 87

Stockholm
Hewlett-Packard Sverige AB
Skalholtsgatan 9, Kista
P.O. Box 19
16493 **KISTA**
Tel: (46/8) 750 20 00
Fax: (46/8) 752 77 81
Telex: (854) 17886

SWITZERLAND

Basel
Hewlett-Packard (Schweiz) AG
Clarastrasse 12
4058 **BÂLE**
Tel: (41/61) 681 59 20
Fax: (41/61) 681 98 59
Telex: 964 719

Geneva/Meyrin
Hewlett-Packard (Schweiz) AG
39, rue de Veyrot
1217 **MEYRIN** 1
Tel: (41/22) 780 41 11
Fax: (41/22) 782 13 68
Telex:27333 HPAG CH

Widen
Hewlett-Packard (Schweiz) AG
Allmend 2
8967 **WIDEN**
Tel: (41/57) 32 11 11
Telex: 828 010 hpag ch
Fax: (41/57) 31 73 74

TAIWAN
Hewlett-Packard Taiwan Ltd.
11/F, 456, Chung Hsiao 1st Road
KAOHSIUNG
Tel: (886/7) 241-2318

Hewlett-Packard Taiwan Ltd.
THM Office
20, Kao-Shuang Road
PIN CHEN, Taoyuan
Tel: (886/3) 492-9660
Fax: (886/3) 492-8087

Hewlett-Packard Taiwan Ltd.
126-1, Sec. 2
Lu-Shung Road
TAICHUNG
Tel: (886/4) 237-1425

Hewlett-Packard Taiwan Ltd.
8/F, Hewlett-Packard Building
337 Fu Hsing North Road
TAIPEI
Tel: (886/2) 712-0404
Fax: (886/2) 715-0851

TANZANIA
Adcom Ltd, Inc. Kenya
P.O. Box 30070
NAIROBI
Kenya
Tel: (254/2) 33 19 55
Telex: 0987 22639

THAILAND
Hewlett-Packard Thailand Ltd.
10th floor
Pacific Place
140 Sukhumvit Road
BANGKOK 10110
Tel: (66/2) 254-6720
Fax: (66/2) 254-6731
Telex: 086 84439 Simonco TH
Cable: UNIMESA Bangkok

TOGO
Engineering Business Concept (E.B.C)
21, Route du Canal
Zone 3
04 B.P. 1357
ABIDJAN
Ivory Coast
Tel: (225) 21 50 24
Fax: (225) 25 27 90

TRINIDAD & TOBAGO
Computer and Controls Ltd.
80-82 Edwards Street
P.O. Box 51
PORT-OF-SPAIN, Trinidad
Tel: (809) 624-2388/2389
Fax: (809) 625-6547

TUNISIA
Precision Electronique
5, rue de Chypre
Mutuelleville
1002 **TUNIS BELVEDERE**
Tunisia
Tel: (216/1) 78 50 37
Telex: 0409 13238

TURKEY
Hewlett-Packard Bilgisayar Ve Olcum Sistemleri A.S.
Paris Caddesi No 3
Diare 9
06670 **ANKARA**
Tel: (90/4) 125 83 13
Fax: (90/4) 125 47 45
Telex: 0607 46180

Hewlett-Packard Bilgisayar Ve Olcum Sistemleri A.S.
Mesrutiyet Mah, 19 Mayis Cad.
Nova-Baran Plaza Kat: 11-12
SISLI / ISTANBUL
Tel: (901) 175 29 70
Fax: (901) 175 29 92
Telex: 0607 39150

UGANDA
Adcom Ltd, Inc. Kenya
P O Box 30070
NAIROBI
Kenya
Tel: (254/2) 33 19 55
Telex: 0987 22639

UNION OF SOVIET SOCIALIST REPUBLICS
Hewlett-Packard Representative Office
Pokrovski Blvd. 4/17 KV12
10100 **MOSCOW**
Tel: (007/095) 923 5001
Fax: (007/095) 230 2611

UNITED ARAB EMIRATES
Emitac Ltd.
P.O. Box 2711
ABU DHABI
Tel: (971/2) 770419
Fax: (971/2) 72 30 58

Emitac Ltd.
Block "B" Arenco Bldg.
Zabel Road
P.O. Box 8391
DUBAI
Tel: (971/4) 37 75 91
Fax: (971/4) 37 08 99

Emitac Ltd.
P.O. Box 1641
SHARJAH
Tel: (971/6) 591181
Telex: 0893 48710 EMITAC EM
Cable: EMITAC SHARJAH

UNITED KINGDOM

ENGLAND
Birmingham
Hewlett-Packard Limited
Avon House
435 Stratford Road
Shirley, **SOLIHULL**
West Midlands B90 4BL
Tel: (44/21) 745-8800
Fax: (44/21) 733-5104
Telex: 339 105

Bracknell
Hewlett-Packard Limited
Coin Road
BRACKNELL
Berkshire RG 12 IHN
Tel: (44/344) 36 0000
Fax: (44/344) 36 3344
Telex: 849 555

Central London
Hewlett-Packard Limited
Bridewell House
9 Bridewell Place
LONDON EC4V 6BS
Tel: (44/71) 583-6565
Fax: (44/71) 583-6565
Ext. 57713/57803
Telex: 298163

Gatwick
Hewlett-Packard Limited
Churchill Court
Gatewick Road
CRAWLEY
West Sussex RH10 2PN
Tel: (44/293) 562955
Fax: (44/293) 578574
Telex: 878862

Manchester
Hewlett-Packard Limited
Heathside Park Road
Cheadle Heath
STOCKPORT
Cheshire SK3 ORB
Tel: (44/61) 428-0828
Fax: (44/61) 495-5009
Telex: 668 068

Uxbridge
Hewlett-Packard Limited
Harman House
No. 1 George Street
UXBRIDGE
Middlesex UB8 1YH
Tel: (44/895) 72020
Fax: (44/895) 75260
Telex: 893 135

Winnersh
Hewlett-Packard Limited
King Street Lane
WINNERSH Wokingham
Berkshire RG 11 5AR
Tel: (44/734) 78 4774
Fax: (44/734) 77 7285
Telex: 847 178

NORTHERN IRELAND
Hewlett-Packard (Ireland) Ltd.
Carrickfergus Industrial Centre
75 Belfast Road, **CARRICKFERGUS**
Co. Antrim BT38 8PM
Tel: (44/960) 367333

SCOTLAND
Hewlett-Packard Ltd.
1/3 Springburn Place
College Milton North
EAST KILBRIDE, G74 5NU
Tel: (44/35) 524 92 61
Fax: (44/35) 523 59 29
Telex: 779615

Hewlett-Packard Ltd.
SOUTH QUEENSFERRY
West Lothian, EH30 9TG
Tel: (44/31) 331 11 88
Fax: (44/31) 331

UNITED STATES
Hewlett-Packard Co.
Customer Information Center
Tel: (800) 752-0900
Hours: 6:00 AM to 5:00 PM
Pacific Time

Alabama
Hewlett-Packard Co.
2100 Riverchase Center
Building 100 - Suite 118
BIRMINGHAM, AL 35244
Tel: (205) 988-0547
Fax: (205) 988-5308

Hewlett-Packard Co.
620 Discovery Dr.
HUNTSVILLE, AL 35806
Tel: (205) 830-2000
Fax: (205) 830-1427

Alaska
Hewlett-Packard Co.
4000 Old Seward Highway
Suite 101
ANCHORAGE, AK 99503
Tel: (907) 563-8855
Fax: (907) 561-7409

SALES OFFICES
Arranged alphabetically by country (cont'd)

UNITED STATES (cont'd)

Arizona
Hewlett-Packard Co.
8080 Pointe Parkway West
PHOENIX, AZ 85044
Tel: (602) 273-8000
Fax: (602) 273-8080

Hewlett-Packard Co.
3400 East Britannia Dr.
Bldg. C, Suite 124
TUCSON, AZ 85706
Tel: (602) 573-7400
Fax: (602) 573-7429

Arkansas
Hewlett-Packard Co.
10816 Executive Center Dr
Conway Bldg. Suite 116
LITTLE ROCK, AR 72211
Tel: (501) 225-7178
Fax: (501) 221-3614

California
Hewlett-Packard Co.
26701 W. Argoura Rd.
CALABASAS, CA 91302
Tel: (818) 880-3400
Fax: (818) 880-3437

Hewlett-Packard Co.
353 Lakeside Dr
FOSTER CITY, CA 94404
Tel: (415) 378-8400
Fax: (415) 378-8405

Hewlett-Packard Co.
1907 North Gateway Blvd.
FRESNO, CA 93727
Tel: (209) 252-9652
Fax: (209) 456-9302

Hewlett-Packard Co.
1421 S. Manhattan Av.
FULLERTON, CA 92631
Tel: (714) 999-6700
Fax: (714) 778-3033

Hewlett-Packard Co.
7408 Hollister Ave. #A
GOLETA, CA 93117
Tel: (805) 685-6100
Fax: (805) 685-6163

Hewlett-Packard Co.
9800 Muirlando Ave.
IRVINE, CA 92718
Tel: (714) 472-3000
Fax: (714) 581-3607 (Direct Dial only)

Hewlett-Packard Co.
2525 Grand Avenue
LONG BEACH, CA 90815
Tel: (213) 498-1111
Fax: (213) 494-1986

Hewlett-Packard Co.
5651 West Manchester Ave.
LOS ANGELES, CA 90045
Tel: (213) 337-8000
Fax: (213) 337-8338

Hewlett-Packard Co.
351 E. Evelyn Ave.
Bldg. 33
MOUNTAIN VIEW, CA 94039
Tel: (415) 694-2000
Fax: (415) 694-0600

Hewlett-Packard Co.
5161 Lankershim Blvd.
NORTH HOLLYWOOD, CA 91601
Tel: (818) 505-5600
Fax: (818) 505-5875

Hewlett-Packard Co.
5725 W. Las Positas Blvd.
PLEASANTON, CA 94566
Tel: (415) 460-0282
Fax: (415) 460-0713

Hewlett-Packard Co.
4244 So. Market Court, Suite A
SACRAMENTO, CA 95834
Tel: (916) 929-7222
Fax: (916) 927-7152

Hewlett-Packard Co.
9606 Aero Drive
SAN DIEGO, CA 92123
Tel: (619) 279-3200
Fax: (619) 268-8487

Hewlett-Packard Co.
50 Fremont St. Suite 200
SAN FRANCISCO, CA 94105
Tel: (415) 882-6800
Fax: (415) 882-6805

SANTA BARBARA (see **GOLETA**)

Hewlett-Packard Co.
3003 Scott Boulevard
SANTA CLARA, CA 95054
Tel: (408) 988-7000
Fax: (408) 988-7103

Hewlett-Packard Co.
5280 Valentine Rd. Suite 205
VENTURA, CA 93003
Tel: (805) 658-6898
Fax: (805) 650-0721

Colorado
Hewlett-Packard Co.
3005 Center Green Drive #205
Suite A
BOULDER, CO 80301
Tel: (303) 530-3940
Fax: (303) 938-3025

Hewlett-Packard Co.
24 Inverness Place, East
ENGLEWOOD, CO 80112
Tel: (303) 649-5000
Fax: (303) 649-5787

Connecticut
Hewlett-Packard Co.
3 Parkland Dr.
DARIEN, CT 06820
Tel: (203) 656-0040
Fax: (203) 656-5563

Hewlett-Packard Co.
115 Glastonbury Blvd
GLASTONBURY, CT 06033
Tel: (203) 633-8100
Fax: (203) 659-6087

District of Columbia
(see Rockville, MD)

Florida
Hewlett-Packard Co.
5900 N. Andrews, Suite 100
FORT LAUDERDALE, FL 33309
Tel: (305) 938-9800
Fax: (305) 938-2293

Hewlett-Packard Co.
6800 South Point Parkway
Suite 301
JACKSONVILLE, FL 32216
Tel: (904) 636-9955
Fax: (904) 636-9955

Hewlett-Packard Co.
255 East Drive, Suite B
MELBOURNE, FL 32901
Tel: (407) 729-0704
Fax: (407) 723-4557

Hewlett-Packard Co.
6177 Lake Ellenor Drive
ORLANDO, FL 32809
Tel: (407) 859-2900
Fax: (407) 826-9309

Hewlett-Packard Co.
4700 Bayou Blvd.
Building 5
PENSACOLA, FL 32503
Tel: (904) 476-8422
Fax: (904) 476-4116

Hewlett-Packard Co.
5550 Idlewild, #150
TAMPA, FL 33634
Tel: (813) 884-3282
Fax: (813) 889-4445

Hewlett-Packard Co.
2015 South Park Place
ATLANTA, GA 30339
Tel: (404) 955-1500
Fax: (404) 980-7669

Hewlett-Packard Co.
3607 Parkway Lane
Suite 300
NORCROSS, GA 30092
Tel: (404) 448-1894
Fax: (404) 246-5206

Hawaii
Hewlett-Packard Co.
Pacific Tower
1001 Bishop St.
Suite 2400
HONOLULU, HI 96813
Tel: (808) 526-1555
Fax: (808) 536-7873

Idaho
Hewlett-Packard Co.
11309 Chinden Blvd.
BOISE, ID 83714
Tel: (208) 323-2700
Fax: (208) 323-2528

Illinois
Hewlett-Packard Co.
2205 E. Empire St.
BLOOMINGTON, IL 61704
Tel: (309) 662-9411
Fax: (309) 662-0351

Hewlett-Packard Co.
525 W. Monroe St., Suite 1308
CHICAGO, IL 60606
Tel: (312) 930-0010
Fax: (312) 930-0986

Hewlett-Packard Co.
1200 East Diehl Road
NAPERVILLE, IL 60566
Tel: (708) 505-8800
Fax: (708) 505-7876

Hewlett-Packard Co.
5201 Tollview Drive
ROLLING MEADOWS, IL 60008
Tel: (708) 255-9800
Fax: (708) 259-5878

Indiana
Hewlett-Packard Co.
11911 N. Meridian St.
CARMEL, IN 46032
Tel: (317) 844-4100
Fax: (317) 843-1291

Hewlett-Packard Co.
111 E. Ludwig Road
Suite 108
FT. WAYNE, IN 46825
Tel: (219) 482-4283
Fax: (219) 482-9907

INDIANAPOLIS (see **CARMEL**)

Iowa
Hewlett-Packard Co.
4050 River Center Court
CEDAR RAPIDS, IA 52402
Tel: (319) 393-0606
Fax: (319) 378-1024

Hewlett-Packard Co.
4201 Corporate Dr.
WEST DES MOINES, IA 50265
Tel: (515) 224-1435
Fax: (515) 224-1870

Kansas
Hewlett-Packard Co.
North Rock Business Park
3450 N. Rock Rd.
Suite 300
WICHITA, KS 67226
Tel: (316) 636-4040
Fax: (316) 636-4504

Kentucky
Hewlett-Packard Co.
305 N. Hurstbourne Lane,
Suite 100
LOUISVILLE, KY 40222
Tel: (502) 426-0100
Fax: (502) 426-0322

Louisiana
Hewlett-Packard Co.
160 James Drive East
ST. ROSE, LA 70087
Tel: (504) 467-4100
Fax: (504) 467-4100 x 291

Maryland

Hewlett-Packard Co.
3701 Koppers Street
BALTIMORE, MD 21227
Tel: (301) 644-5800
Fax: (301) 362-7650

Hewlett-Packard Co.
2 Choke Cherry Road
ROCKVILLE, MD 20850
Tel: (301) 258-2000
Fax: (301) 258-5986

Massachusetts

Hewlett-Packard Co.
1775 Minuteman Road
ANDOVER, MA 01810
Tel: (508) 682-1500
Fax: (508) 794-2619

Hewlett-Packard Co.
29 Burlington Mall Rd.
BURLINGTON, MA 01803-4514
Tel: (617) 270-7000
Fax: (617) 221-5240

Michigan

Hewlett-Packard Co.
3033 Orchard Vista S.E.
GRAND RAPIDS, MI 49546
Tel: (616) 957-1970
Fax: (616) 956-9022

Hewlett-Packard Co.
39550 Orchard Hill Place Drive
NOVI, MI 48050
Tel: (313) 349-9200
Fax: (313) 349-9240

Hewlett-Packard Co.
560 Kirts Rd.
Suite 101
TROY, MI 48084
Tel: (313) 362-5180
Fax: (313) 362-3028

Minnesota

Hewlett-Packard Co.
2025 W. Larpenteur Ave.
ST. PAUL, MN 55113
Tel: (612) 644-1100
Fax: (612) 641-9787

Mississippi

Hewlett-Packard Co.
800 Woodland Parkway, Suite 101
RIDGELAND, MS 39157
Tel: (601) 957-0730
Fax: (601) 957-2515

Missouri

Hewlett-Packard Co.
6601 Winchester Ave.
KANSAS CITY, MO 64133
Tel: (816) 737-0071
Fax: (816) 737-4690

Hewlett-Packard Co.
530 Maryville Centre Drive
ST. LOUIS, MO 63140
Tel: (314) 542-1500
Fax: (314) 542-1586

Nebraska

Hewlett-Packard
11626 Nicholas St.
OMAHA, NE 68154
Tel: (402) 493-0300
Fax: (402) 493-4334

New Jersey

Hewlett-Packard Co.
120 W. Century Road
PARAMUS, NJ 07653
Tel: (201) 599-5000
Fax: (201) 599-5382

Hewlett-Packard Co.
10 Sylvan Way
PARSIPPANY, NJ 07054
Tel: (201) 682-4000
Fax: (201) 682-4031

Hewlett-Packard Co.
20 New England Ave. West
PISCATAWAY, NJ 08854
Tel: (201) 562-6100
Fax: (201) 562-6246

New Mexico

Hewlett-Packard Co.
7801 Jefferson N.E.
ALBUQUERQUE, NM 87109
Tel: (505) 823-6100
Fax: (505) 823-1243

Hewlett-Packard Co.
1362-C Trinity Dr.
LOS ALAMOS, NM 87544
Tel: (505) 662-6700
Fax: (505) 662-4312

New York

Hewlett-Packard Co.
5 Computer Drive South
ALBANY, NY 12205
Tel: (518) 458-1550
Fax: (518) 458-1550 x 0393

Hewlett-Packard Co.
130 John Muir Dr.
AMHERST, NY 14228
Tel: (716) 689-3003
Fax: (716) 636-7034

BUFFALO (see AMHERST)

Hewlett-Packard Co.
200 Cross Keys Office Park
FAIRPORT, NY 14450
Tel: (716) 223-9950
Fax: (716) 223-6331

Hewlett-Packard Co.
7641 Henry Clay Blvd.
LIVERPOOL, NY 13088
Tel: (315) 451-1820
Fax: (315) 451-1820 x 255

Hewlett-Packard Co.
No. 1 Pennsylvania Plaza
55th Floor
34th Street & 7th Avenue
MANHATTAN NY 10119
Tel: (212) 971-0800
Fax: (212) 330-6967

Hewlett-Packard Co.
7 Old Sod Farm Road
MELVILLE, NY 11714
Tel: (516) 753-0555
Fax: (516) 753-3469

Hewlett-Packard Co.
2975 Westchester Ave
PURCHASE, NY 10577
Tel: (914) 935-6300
Fax: (914) 935-6497

ROCHESTER (see FAIRPORT)

SYRACUSE (see LIVERPOOL)

Hewlett-Packard Co.
Executive Square Office Bldg.
66 Middlebush Rd.
WAPPINGERS FALLS, NY 12590
Tel: (914) 298-9125
Fax: (914) 298-9154

North Carolina

Hewlett-Packard Co.
305 Gregson Dr.
CARY, NC 27511
Tel: (919) 467-6600
Fax: (919) 460-2296
(919) 460-2297

Hewlett-Packard Co.
P.O. Box 240318
CHARLOTTE, NC 28224
Tel: (704) 527-8780
Fax: (704) 523-7857

Hewlett-Packard Co.
7029 Albert Pick Rd. #100
GREENSBORO, NC 27409
Tel: (919) 665-1800
Fax: (919) 668-1797
Mailing Address
PO Box 26500
Greensboro, NC 27426

RALEIGH (see CARY)

Ohio

AKRON (see COPLEY)

Hewlett-Packard Co.
4501 Erskine Road
CINCINNATI, OH 45242
Tel: (513) 891-9870
Fax: (513) 891-0033

CLEVELAND (see STRONGSVILLE)

Hewlett-Packard Co.
Montrose West Ave.
COPLEY, OH 44321
Tel: (216) 666-7711
Fax: (216) 666-6054

Hewlett-Packard Co.
7887 Washington Village Dr.
DAYTON, OH 45459
Tel: (513) 433-2223
Fax: (513) 433-8633

Hewlett-Packard Co.
15885 Sprague Road
STRONGSVILLE, OH 44136
Tel: (216) 243-7300
Fax: (216) 234-7230

Hewlett-Packard Co.
One Maritime Plaza, 5th Floor
720 Water Street
TOLEDO, OH 43604
Tel: (419) 242-2200
Fax: (419) 241-7655

Hewlett-Packard Co.
675 Brooksedge Blvd.
WESTERVILLE, OH 43081
Tel: (614) 891-3344
Fax: (614) 891-1476

Oklahoma

Hewlett-Packard Co.
3525 N.W. 56th St.
Suite C-100
OKLAHOMA CITY, OK 73112
Tel: (405) 946-9499
Fax: (405) 942-2127

Hewlett-Packard Co.
6655 South Lewis,
Suite 105
TULSA, OK 74136
Tel: (918) 481-6700
Fax: (918) 481-2250

Oregon

EUGENE (see WILSONVILLE)

Hewlett-Packard Co.
9255 S. W. Pioneer Court
WILSONVILLE, OR 97070
Tel: (503) 484-4085
Fax: (503) 682-8155

Pennsylvania

Hewlett-Packard Co.
Heatherwood Industrial Park
50 Dorchester Rd.
P.O. Box 6080
HARRISBURG, PA 17112
Tel: (717) 657-5900
Fax: (717) 657-5946

Hewlett-Packard Co.
111 Zeta Drive
PITTSBURGH, PA 15238
Tel: (412) 782-0400
Fax: (412) 963-1300

Hewlett-Packard Co.
2750 Monroe Boulevard
VALLEY FORGE, PA 19482
Tel: (215) 666-9000
Fax: (215) 666-2034

South Carolina

Hewlett-Packard Co.
Brookside Park, Suite 122
1 Harbison Way
COLUMBIA, SC 29212
Tel: (803) 732-0400
Fax: (803) 732-4567

Hewlett-Packard Co.
545 N. Pleasantburg Dr.
Suite 100
GREENVILLE, SC 29607
Tel: (803) 232-8002
Fax: (803) 232-8739

Tennessee

Hewlett-Packard Co.
One Energy Center Suite 200
Pellissippi Pkwy.
KNOXVILLE, TN 37932
Tel: (615) 966-4747
Fax: (615) 966-8147

SALES OFFICES
Arranged alphabetically by country (cont'd)

UNITED STATES (cont'd)

Tennessee (cont'd)
Hewlett-Packard Co.
889 Ridge Lake Blvd.,
Suite 100
MEMPHIS, TN 38119
Tel: (901) 763-4747
Fax: (901) 762-9723

Hewlett-Packard Co.
44 Vantage Way,
Suite 160
NASHVILLE, TN 37228
Tel: (615) 255-1271
Fax: (615) 726-2310

Texas
Hewlett-Packard Co.
9050 Capital of Texas Highway, North
#290
AUSTIN, TX 78759
Tel: (512) 346-3855
Fax: (512) 338-7201
Mailing Address
PO Box 9431
Austin, TX 78766-9430

DALLAS (see RICHARDSON)

Hewlett-Packard Co.
5700 Cromo Dr
EL PASO, TX 79912
Tel: (915) 833-4400
Fax: (915) 581-8097

Hewlett-Packard Co.
10535 Harwin Drive
HOUSTON, TX 77036
Tel: (713) 776-6400
Fax: (713) 776-6495

Hewlett-Packard Co.
3301 West Royal Lane
IRVING, TX 75063
Tel: (214) 869-3377
Fax: (214) 830-8951

Hewlett-Packard Co.
109 E. Toronto, Suite 100
McALLEN, TX 78503
Tel: (512) 630-3030
Fax: (512) 630-1355

Hewlett-Packard Co.
930 E. Campbell Rd.
RICHARDSON, TX 75081
Tel: (214) 231-6101
Fax: (214) 699-4337

Hewlett-Packard Co.
14100 San Pedro Ave., Suite 100
SAN ANTONIO, TX 78232
Tel: (512) 494-9336
Fax: (512) 491-1299

Utah
Hewlett-Packard Co.
3530 W. 2100 South
SALT LAKE CITY, UT 84119
Tel: (801) 974-1700
Fax: (801) 974-1780

Virginia
Hewlett-Packard Co.
840 Greenbrier Circle
Suite 101
CHESAPEAKE, VA 23320
Tel: (804) 424-7105
Fax: (804) 424-1494

Hewlett-Packard Co.
4401 Water Front Dr.
GLEN ALLEN, VA 23060
Tel: (804) 747-7750
Fax: (804) 965-9297

NORFOLK (see CHESAPEAKE)

RICHMOND (see GLEN ALLEN)

Hewlett-Packard Co.
2800 Electric Road Suite 100
ROANOKE, VA 24018
Tel: (703) 774-3444
Fax: (703) 989-8049

Washington
Hewlett-Packard Co.
15815 S.E. 37th Street
BELLEVUE, WA 98006
Tel: (206) 643-4000
Fax: (206) 643-8748

Hewlett-Packard Co.
N. 1225 Argonne Rd
SPOKANE, WA 99212-2657
Tel: (509) 922-7000
Fax: (509) 927-4236

Washington, D.C.
(see Rockville, MD)

West Virginia
Hewlett-Packard Co.
501 56th Street
CHARLESTON, WV 25304
Tel: (304) 925-0492
Fax: (304) 925-1910

Wisconsin
Hewlett-Packard Co.
275 N. Corporate Dr.
BROOKFIELD, WI 53005
Tel: (414) 792-8800
Fax: (414) 792-0218

Hewlett-Packard Co.
333 Main St.
GREEN BAY, WI 54301
Tel: (414) 436-2780
Fax: (414) 436-2786

MILWAUKEE (see BROOKFIELD)

URUGUAY
Pablo Ferrando S.A.C. e I.
Avenida Italia 2877
Casilla de Correo 370
MONTEVIDEO
Tel: (598/2) 802 586
Telex: 398802586

VENEZUELA
Hewlett-Packard de Venezuela, C.A.
Tercera Transversal
Los Ruices Norte
Edificio Segre, **CARACAS**
Tel: (58/2) 239 4133/4244
Fax: (58/2) 239 3080

YUGOSLAVIA
Do Hermes
General Zdanova 4
YU-11000 **BEOGRAD**
Tel: (38/11) 342 641
Telex: 11433

Do Hermes
Celovska 73
YU-61000 **LJUBLJANA**
Tel: (38/61) 553 170
Telex: 31583

ZAMBIA
R.J. Tilbury (Zambia) Ltd.
P.O. Box 32792
LUSAKA
Tel: (260/1) 21 55 80
Telex: 0902 40128

ZIMBABWE
Field Consolidated (Private) Ltd.
Systron Division
Manhattan Court
61 Second Street
P.O. Box 3458
HARARE
Tel: (263/4) 73 98 81
Fax: (263/4) 70 20 08
Telex: 26241

Please send directory corrections to:
Test & Measurement Catalog, 54AF
Hewlett-Packard Company
5301 Stevens Creek Blvd.
P.O. Box 58059
Santa Clara, CA 95052-8059
Tel: (408) 553-7271
Fax: (408) 983-1006

September 1990

HEWLETT PACKARD

5959-6031

Printed in U.S.A.

HP 1652B/HP 1653B Logic Analyzers

Programming Reference

HEWLETT PACKARD

Your Comments Please

HP 1650B/1651B Programming

Your comments assist us in meeting your needs better. Please complete this questionnaire and return it to us. Feel free to add any additional comments that you might have. All comments and suggestions become the property of Hewlett-Packard. Omit any questions that you feel would be proprietary.

	Yes	No
1. Were you satisfied with the operation of the instrument over the bus?	[ ]	[ ]

2. What measurements will this instrument be used to make over the bus?

____

3. What type of controller are you using?____________________

4. What programming language are you using?____________________

5. What do you like most about programming the instrument?____________________

____

6. What would you like to see changed or improved?____________________

____

7. Which sections of the manual have you used?

[ ] Introductory chapters 1 through 4
[ ] Command List chapters 5 through 27
[ ] Appendix A
[ ] Programming Examples
[ ] Index

8. Please rate the manual on the following:

4 = Excellent   3 = Good   2 = Adequate   1 = Poor

[ ] Breadth and depth of information
[ ] Ability to easily find information
[ ] Ability to understand and apply the information provided in the manual

Please explain:____

9. What is your experience with programming instruments over the bus?

[ ] No previous experience
[ ] Less than 1 year experience
[ ] More than 1 year's experience on one model
[ ] More than 1 year's experience on several models

Name ____________________ Company ____________________
Address ____________________ Zip Code ____________________
Phone ____________________ Instrument Serial # ____________________

THANK YOU FOR YOUR HELP

NO POSTAGE NECESSARY IF MAILED IN U.S.A.

Your cooperation in completing and returning this form will be greatly appreciated. Thank you.

FOLD HERE

HEWLETT PACKARD

NO POSTAGE NECESSARY IF MAILED IN THE UNITED STATES

BUSINESS REPLY CARD

FIRST CLASS PERMIT NO. 1303 COLORADO SPRINGS, COLORADO

POSTAGE WILL BE PAID BY ADDRESSEE

HEWLETT-PACKARD
COLORADO SPRINGS DIVISION
ATTN: PUBLICATIONS DEPT.
P.O. BOX 2197
COLORADO SPRINGS, COLORADO 80901-9959

Programming Reference

HP 1652B/HP 1653B Logic Analyzers

HEWLETT
PACKARD

Manual Number 01652-90903

Printed in the U.S.A. December 1989

Product Warranty

This Hewlett-Packard product has a warranty against defects in material and workmanship for a period of three years from date of shipment. During warranty period, Hewlett-Packard Company will, at its option, either repair or replace products that prove to be defective.

For warranty service or repair, this product must be returned to a service facility designated by Hewlett-Packard. However, warranty service for products installed by Hewlett-Packard and certain other products designated by Hewlett-Packard will be performed at the Buyer's facility at no charge within the Hewlett-Packard service travel area. Outside Hewlett-Packard service travel areas, warranty service will be performed at the Buyer's facility only upon Hewlett-Packard's prior agreement and the Buyer shall pay Hewlett-Packard's round trip travel expenses.

For products returned to Hewlett-Packard for warranty service, the Buyer shall prepay shipping charges to Hewlett-Packard and Hewlett-Packard shall pay shipping charges to return the product to the Buyer. However, the Buyer shall pay all shipping charges, duties, and taxes for products returned to Hewlett-Packard from another country.

Hewlett-Packard warrants that its software and firmware designated by Hewlett-Packard for use with an instrument will execute its programming instructions when properly installed on that instrument. Hewlett-Packard does not warrant that the operation of the instrument software, or firmware will be uninterrupted or error free.

Limitation of Warranty

The foregoing warranty shall not apply to defects resulting from improper or inadequate maintenance by the Buyer, Buyer-supplied software or interfacing, unauthorized modification or misuse, operation outside of the environmental specifications for the product, or improper site preparation or maintenance.

NO OTHER WARRANTY IS EXPRESSED OR IMPLIED. HEWLETT-PACKARD SPECIFICALLY DISCLAIMS THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

Exclusive Remedies THE REMEDIES PROVIDED HEREIN ARE THE BUYER'S SOLE AND EXCLUSIVE REMEDIES. HEWLETT-PACKARD SHALL NOT BE LIABLE FOR ANY DIRECT, INDIRECT, SPECIAL INCIDENTAL, OR CONSEQUENTIAL DAMAGES, WHETHER BASED ON CONTRACT, TORT, OR ANY OTHER LEGAL THEORY.

Assistance Product maintenance agreements and other customer assistance agreements are available for Hewlett-Packard products.

For any assistance, contact your nearest Hewlett-Packard Sales and Service Office.

Certification Hewlett-Packard Company certifies that this product met its published specifications at the time of shipment from the factory. Hewlett-Packard further certifies that its calibration measurements are traceable to the United States National Bureau of Standards, to the extent allowed by the Bureau's calibration facility, and to the calibration facilities of other International Standards Organization members.

Safety This product has been designed and tested according to International Safety Requirements. To ensure safe operation and to keep the product safe, the information, cautions, and warnings in this manual must be heeded.

Printing History

New editions are complete revisions of the manual. Update packages, which are issued between editions, contain additional and replacement pages to be merged into the manual by the customer. The dates on the title page change only when a new edition or a new update is published. No information is incorporated into a reprinting unless it appears as a prior update; the edition does not change when an update is incorporated.

A software code may be printed before the date; this indicates the version level of the software product at the time of the manual or update was issued. Many product updates and fixes do not require manual changes and, conversely, manual corrections may be done without accompanying product changes. Therefore, do not expect a one to one correspondence between product updates and manual updates.

Edition 1	December 1989	01652-90903

List of Effective Pages

The List of Effective Pages gives the data of the current edition and of any pages changed in updates to that edition. Within the manual, any page changed since the last edition will have the date the changes were made printed on the bottom of the page. If an update is incorporated when a new edition of the manual is printed, the change dates are removed from the bottom of the pages and the new edition date is listed in Printing History and on the title page.

Pages	**Effective Date**
All	December 1989

Contents

Chapter 1

Chapter 2

Programming Over HP-IB

Chapter 3

Programming Over RS-232C

Chapter 4

Programming and Documentation Conventions

Chapter 5

Common Commands

Chapter 6

System Commands

Chapter 7

MMEMory Subsystem

Chapter 8

DLISt Subsystem

Chapter 9

WLISt Subsystem

Chapter 10

MACHine Subsystem

Chapter 11

SFORmat Subsystem

Chapter 12

STRace Subsystem

Chapter 13

SLISt Subsystem

Chapter 14

SWAVeform Subsystem

Chapter 15

SCHart Subsystem

Chapter 16

COMPare Subsystem

Chapter 17

TFORmat Subsystem

Chapter 18

TTRace Subsystem

Chapter 19

TWAVeform Subsystem

Chapter 20

SYMBol Subsystem

Chapter 21

SCOPe Subsystem

Chapter 22

CHANnel Subsystem

Chapter 23

TRIGger Subsystem

Chapter 24

ACQuire Subsystem

Chapter 25

TIMebase Subsystem

Chapter 26

WAVeform Subsystem

Chapter 27

MEASure Subsystem

Appendix A

Message Communication and System Functions

Appendix B

Status Reporting

Appendix C

Error Messages

Index

1 - Introduction to Programming an Instrument

Introduction to Programming an Instrument

1

Introduction

This chapter introduces you to the basics of remote programming. The programming instructions explained in this book conform to the IEEE 488.2 Standard Digital Interface for Programmable Instrumentation. These programming instructions provide a means of remotely controlling the HP 1652B/53B. There are three general categories of use. You can:

- Set up the instrument and start measurements
- Retrieve setup information and measurement results
- Send measurement data to the instrument

The instructions listed in this manual give you access to the measurements and front panel features of the HP 1652B/53B. The complexity of your programs and the tasks they accomplish are limited only by your imagination. This programming reference is designed to provide a concise description of each instruction.

About This Manual

This manual is organized in 27 chapters. Chapter 1 is divided into two sections. The first section (pages 2 through 9) concentrates on program syntax, and the second section (pages 10 through 17) discusses programming an instrument. Read either chapter 2, "Programming Over HP-IB," or chapter 3, "Programming Over RS-232C" for information concerning the physical connection between the HP 1652B/53B and your controller. Chapter 4, "Programming and Documentation Conventions," gives an overview of all instructions and also explains the notation conventions used in our syntax definitions and examples. The remaining chapters 5 through 27 are used to explain each group of instructions.

Programming Syntax

Talking to the Instrument

In general, computers acting as controllers communicate with the instrument by sending and receiving messages over a remote interface, such as HP-IB or RS-232C. Instructions for programming the HP 1652B/53B will normally appear as ASCII character strings embedded inside the output statements of a "host" language available on your controller. The host language's input statements are used to read in responses from the HP 1652B/53B.

For example, HP 9000 Series 200/300 BASIC uses the OUTPUT statement for sending commands and queries to the HP 1652B/53B. After a query is sent, the response is usually read in using the ENTER statement. All programming examples in this manual are presented in BASIC. The following BASIC statement sends a command which causes the HP 1652B/53B's machine 1 to be a state analyzer:

```
OUTPUT XXX;":MACHINE1:TYPE STATE"  <terminator>
```

Each part of the above statement is explained in the following pages.

Instruction Syntax

To program the instrument remotely, you must have an understanding of the command format and structure expected by the instrument. The IEEE 488.2 syntax rules govern how individual elements such as headers, separators, parameters and terminators may be grouped together to form complete instructions. Syntax definitions are also given to show how query responses will be formatted. Figure 1-1 shows the main syntactical parts of a typical program statement.

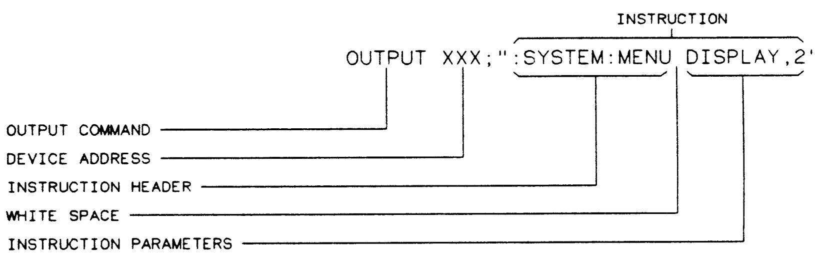


Figure 1-1. Program Message Syntax

Output Command

The output command is entirely dependant on the language you choose to use. Throughout this manual HP 9000 Series 200/300 BASIC 4.0 is used in the programming examples. People using another language will need to find the equivalents of BASIC commands like OUTPUT, ENTER and CLEAR in order to convert the examples. The instructions for the HP 1652B/53B are always shown between the double-quotes.

Device Address

The location where the device address must be specified is also dependent on the host language which you are using. In some languages, this could be specified outside the output command. In BASIC, this is always specified after the keyword OUTPUT. The examples in this manual use a generic address of XXX. When writing programs, the number you use will depend on the cable you use in addition to the actual address. If you are using an HP-IB, see chapter 2. RS-232C users should refer to chapter 3, "Programming Over RS-232C."

Instructions

Instructions (both commands and queries) normally appear as a string embedded in a statement of your host language, such as BASIC, Pascal or C. The only time a parameter is not meant to be expressed as a string is when the instruction's syntax definition specifies <block data>. There are only five instructions which use block data.

Instructions are composed of two main parts: The header, which specifies the command or query to be sent; and the parameters, which provide additional data needed to clarify the meaning of the instruction.

Instruction Header

The instruction header is one or more keywords separated by colons (:). The command tree in figure 4-1 illustrates how all the keywords can be joined together to form a complete header (see chapter 4, "Programming and Documentation Conventions").

The example in figure 1-1 shows a command. Queries are indicated by adding a question mark (?) to the end of the header. Many instructions can be used as either commands or queries, depending on whether or not you have included the question mark. The command and query forms of an instruction usually have different parameters. Many queries do not use any parameters.

When you look up a query in this programming reference, you'll find a paragraph labeled "Returned Format" under the one labeled "Query Syntax." The syntax definition by "Returned format" will always show the instruction header in square brackets, like [:SYSTem:MENU]. What this really means is that the text between the brackets is optional, but it's also a quick way to see what the header looks like.

White Space White space is used to separate the instruction header from the instruction parameters. If the instruction does not use any parameters, you do not need to include any white space. White space is defined as one or more spaces. ASCII defines a space to be character 32 (in decimal). Tabs can be used only if your controller first converts them to space characters before sending the string to the instrument.

Instruction Parameters Instruction parameters are used to clarify the meaning of the command or query. They provide necessary data, such as whether a function should be on or off, which waveform is to be displayed, or which pattern is to be looked for. Each instruction's syntax definition shows the parameters, as well as the values they accept. This chapter's "Parameter Syntax Rules" section has all of the general rules about acceptable values.

When there is more than one parameter they are separated by commas (,). You are allowed to add spaces around the commas.

Header Types There are three types of headers: Simple Command; Compound Command; and Common Command.

Simple Command Header. Simple command headers contain a single keyword. START and STOP are examples of simple command headers typically used in this instrument. The syntax is:

<function> <terminator>

When parameters (indicated by <data>) must be included with the simple command header (for example, :RMODE SINGLE) the syntax is:

<function> <white space> <data> <terminator>

Compound Command Header. Compound command headers are a combination of two or more program keywords. The first keyword selects the subsystem, and the last keyword selects the function within that subsystem. Sometimes you may need to list more than one subsystem before being allowed to specify the function. The keywords within the compound header are separated by colons. For example:

To execute a single function within a subsystem, use the following:

:<subsystem>:<function> <white space> <data> <terminator>

(For example :SYSTEM:LONGFORM ON)

To traverse down a level of a subsystem to execute a subsystem within that subsystem:

:<subsystem>:<subsystem>:<function> <white space> <data> <terminator>

(For example :MMEMORY:LOAD:CONFIG "FILE_")

Common Command Header. Common command headers control IEEE 488.2 functions within the instrument (such as clear status, etc.). Their syntax is:

*<command header> <terminator>

No space or separator is allowed between the asterisk and the command header. *CLS is an example of a common command header.

Combining Commands from the Same Subsystem

To execute more than one function within the same subsystem a semi-colon (;) is used to separate the functions:

:<subsystem>:<function> <white space> <data>;
<function> <white space> <data> <terminator>

(For example :SYSTEM:LONGFORM ON;HEADER ON)

Duplicate Keywords

Identical function keywords can be used for more than one subsystem. For example, the function keyword MMODE may be used to specify the marker mode in the subsystem for state listing or the timing waveforms:

:SLIST:MMODE PATTERN - sets the marker mode to pattern in the state listing.

:TWAVEFORM:MMODE TIME - sets the marker mode to time in the timing waveforms.

SLIST and TWAVEFORM are subsystem selectors and determine which marker mode is being modified.

Query Usage Command headers immediately followed by a question mark (?) are queries. After receiving a query, the instrument interrogates the requested function and places the response in its output queue. The output message remains in the queue until it is read or another command is issued. When read, the message is transmitted across the bus to the designated listener (typically a controller). For example, the logic analyzer query :MACHINE1:TWAVEFORM:RANGE? places the current seconds per division full scale range for machine 1 in the output queue. In BASIC, the input statement

```
ENTER XXX; Range
```

passes the value across the bus to the controller and places it in the variable Range.

Query commands are used to find out how the instrument is currently configured. They are also used to get results of measurements made by the instrument. For example, the command

```
:MACHINE1:TWAVEFORM:XOTIME?
```

instructs the instrument to place the X to O time in the output queue.

Note

The output queue must be read before the next program message is sent. For example, when you send the query :TWAVEFORM:XOTIME? you must follow that with an input statement. In BASIC, this is usually done with an ENTER statement.

Sending another command before reading the result of the query will cause the output buffer to be cleared and the current response to be lost. This will also generate a "QUERY UNTERMINATED" error in the error queue.

Program Header Options

Program headers can be sent using any combination of uppercase or lowercase ASCII characters. Instrument responses, however, are always returned in uppercase.

Both program command and query headers may be sent in either longform (complete spelling), shortform (abbreviated spelling), or any combination of longform and shortform. Either of the following examples turns on the headers and longform.

```
OUTPUT XXX;":SYSTEM:HEADER ON;LONGFORM ON"    - longform
OUTPUT XXX;":SYST:HEAD ON;LONG ON"            - shortform
```

Programs written in longform are easily read and are almost self-documenting. The shortform syntax conserves the amount of controller memory needed for program storage and reduces the amount of I/O activity.

Note

The rules for shortform syntax are shown in chapter 4 "Programming and Documentation Conventions."

Parameter Syntax Rules

There are three main types of data which are used in parameters. They are numeric, string, and keyword. A fourth type, block data, is used only for five instructions: the DATA and SETup instructions in the SYSTem subsystem (see chapter 6); the CATalog, UPLoad, and DOWNload instructions in the MMEMory subsystem (see chapter 7). These syntax rules also show how data may be formatted when sent back from the HP 1652B/53B as a response.

The parameter list always follows the instruction header and is separated from it by white space. When more than one parameter is used, they are separated by commas. You are allowed to include one or more spaces around the commas, but it is not mandatory.

Numeric data. For numeric data, you have the option of using exponential notation or using suffixes to indicate which unit is being used. Tables A-1 and A-2 in appendix A list all available suffixes. Do not combine an exponent with a unit. The following numbers are all equal: 28 = 0.28E2 = 280e-1 = 28000m = 0.028K.

The base of a number is shown with a prefix. The available bases are binary (#B), octal (#Q), hexadecimal (#H) and decimal (default). For example, #B11100 = #Q34 = #H1C = 28. You may not specify a base in conjunction with either exponents or unit suffixes. Additionally, negative numbers must be expressed in decimal.

When a syntax definition specifies that a number is an integer, that means that the number should be whole. Any fractional part would be ignored, truncating the number. Numeric parameters which accept fractional values are called real numbers.

All numbers are expected to be strings of ASCII characters. Thus, when sending the number 9, you would send a byte representing the ASCII code for the character "9" (which is 57, or 0011 1001 in binary). A three-digit number like 102 would take up three bytes (ASCII codes 49, 48 and 50). This is taken care of automatically when you include the entire instruction in a string.

String data. String data may be delimited with either single (') or double (") quotes. String parameters representing labels are case-sensitive. For instance, the labels "Bus A" and "bus a" are unique and should not be used indiscriminately. Also pay attention to the presence of spaces, since they act as legal characters just like any other. So the labels "In" and "   In" are also two separate labels.

Keyword data. In many cases a parameter must be a keyword. The available keywords are always included with the instruction's syntax definition. When sending commands, either the longform or shortform (if one exists) may be used. Upper-case and lower-case letters may be mixed freely. When receiving responses, upper-case letters will be used exclusively. The use of longform or shortform in a response depends on the setting you last specified via the SYSTem:LONGform command (see chapter 6).

Instruction Terminator

An instruction is executed after the instruction terminator is received. The terminator is the NL (New Line) character. The NL character is an ASCII linefeed character (decimal 10).

Note

The NL (New Line) terminator has the same function as an EOS (End Of String) and EOT (End Of Text) terminator.

Selecting Multiple Subsystems

You can send multiple program commands and program queries for different subsystems on the same line by separating each command with a semicolon. The colon following the semicolon enables you to enter a new subsystem. For example:

<instruction header> <data>;: <instruction header> <data> <terminator>

:MACHINE1:ASSIGN2;:SYSTEM:HEADERS ON

Note

Multiple commands may be any combination of simple, compound and common commands.

Programming an Instrument

Initialization To make sure the bus and all appropriate interfaces are in a known state, begin every program with an initialization statement. BASIC provides a CLEAR command which clears the interface buffer. If you're using HP-IB, CLEAR will also reset the HP 1652B/53B's parser. The parser is the program which reads in the instructions which you send it.

After clearing the interface, load a predefined configuration file from the disk to preset the instrument to a known state. For example:

```
OUTPUT XXX;":MMEMORY:LOAD:CONFIG 'DEFAULT__'"
```

This BASIC statement would load the configuration file "DEFAULT__" (if it exists) into the HP 1652B/53B. Refer to the chapter "MMEMory Subsystem" for more information on the LOAD command.

Note Refer to your controller manual and programming language reference manual for information on initializing the interface.

Example Program

This program demonstrates the basic command structure used to program the HP 1652B/53B.

```
10 CLEAR XXX                                   !Initialize instrument interface
20 OUTPUT XXX;":SYSTEM:HEADER ON"                           !Turn headers on
30 OUTPUT XXX;":SYSTEM:LONGFORM ON"                        !Turn longform on
40 OUTPUT XXX;":MMEM:LOAD:CONFIG 'TEST_E'"      !Load configuration file
50 OUTPUT XXX;":MENU FORMAT,1"            !Select Format menu for machine 1
60 OUTPUT XXX;":RMODE SINGLE"                               !Select run mode
70 OUTPUT XXX;":START"                                  !Run the measurement
```

Program Overview

Line 10 initializes the instrument interface to a known state
Lines 20 and **30** turn the headers and longform on.
Line 40 loads the configuration file "TEST_E" from the disc drive.
Line 50 displays the Format menu for machine 1.
Lines 60 and **70** tell the analyzer to run the measurement configured by the file "TEST_E" one time.

Receiving Information from the Instrument

After receiving a query (command header followed by a question mark), the instrument interrogates the requested function and places the answer in its output queue. The answer remains in the output queue until it is read or another command is issued. When read, the message is transmitted across the bus to the designated listener (typically a controller). The input statement for receiving a response message from an instrument's output queue typically has two parameters;the device address and a format specification for handling the response message. For example, to read the result of the query command :SYSTEM:LONGFORM? you could execute the BASIC statement:

```
ENTER XXX; Setting
```

where XXX represents the address of your device. This would enter the current setting for the longform command in the numeric variable *Setting*.

Note

All results for queries sent in a program message must be read before another program message is sent. For example, when you send the query :MACHINE1:ASSIGN?, you must follow that query with an input statement. In BASIC, this is usually done with an ENTER statement.

The format specification for handling the response messages is dependent on both the controller and the programming language.

Response Header Options

The format of the returned ASCII string depends on the current settings of the SYSTEM HEADER and LONGFORM commands. The general format is:

<instruction header> <space> <data> <terminator>

The header identifies the data that follows (the parameters) and is controlled by issuing a :SYSTEM:HEADER ON/OFF command. If the state of the header command is OFF, only the data is returned by the query.

The format of the header is controlled by the :SYSTEM:LONGFORM ON/OFF command. If longform is OFF , the header will be in its shortform and the header will vary in length depending on the particular query. The separator between the header and the data always consists of one space.

The following examples show some possible responses for a :MACHINE1:SFORMAT:THRESHOLD2? query:

- with HEADER OFF:
  <data> <terminator>
- with HEADER ON and LONGFORM OFF:
  :MACH1:SFOR:THR2 <space> <data> <terminator>
- with HEADER ON and LONGFORM ON:
  :MACHINE1:SFORMAT:THRESHOLD2 <space> <data> <terminator>

Note

A command or query may be sent in either longform or shortform, or in any combination of longform and shortform. The HEADER and LONGFORM commands only control the format of the returned data and have no effect on the way commands are sent.

Refer to the chapter "System Commands" for information on turning the HEADER and LONGFORM commands on and off.

Response Data Formats

Both numbers and strings are returned as a series of ASCII characters, as described in the following sections. Keywords in the data are returned in the same format as the header, as specified by the LONGform command. Like the headers, the keywords will always be in upper-case.

The following are possible responses to the "MACHINE1: TFORMAT: LAB? 'ADDR' " query.

MACHINE1:TFORMAT:LABEL "ADDR ",19,POSITIVE<terminator> (Header on; Longform on)

MACH1:TFOR:LAB "ADDR ",19,POS<terminator> (Header on; Longform off)

"ADDR ",19,POSITIVE<terminator> (Header off; Longform on)

"ADDR ",19,POS<terminator> (Header off; Longform off)

Note

Refer to the individual commands in this manual for information on the format (alpha or numeric) of the data returned from each query.

String Variables

Since there are so many ways to code numbers, the HP 1652B/53B handles almost all data as ASCII strings. Depending on your host language, you may be able to use other types when reading in responses.

Sometimes it is helpful to use string variables in place of constants to send instructions to the HP 1652B/53B. The example below combines variables and constants in order to make it easier to switch from MACHINE1 to MACHINE2. In BASIC, the & operator is used for string concatenation.

```
10 LET Machine$ = ":MACHINE2"   !Send all instructions to machine 2
20 OUTPUT XXX; Machine$ & ":TYPE STATE"   !Make machine a state analyzer
30 ! Assign all labels to be positive
40 OUTPUT XXX; Machine$ & ":SFORMAT:LABEL 'CHAN 1', POS"
50 OUTPUT XXX; Machine$ & ":SFORMAT:LABEL 'CHAN 2', POS"
60 OUTPUT XXX; Machine$ & ":SFORMAT:LABEL 'OUT', POS"
99 END
```

If you want to observe the headers for queries, you must bring the returned data into a string variable. Reading queries into string variables requires little attention to formatting. For example:

```
ENTER XXX;Result$
```

places the output of the query in the string variable Result$.

Note

In the language used for this book (HP BASIC 4.0), string variables are case sensitive and must be expressed exactly the same each time they are used.

The output of the instrument may be numeric or character data depending on what is queried. Refer to the specific commands for the formats and types of data returned from queries.

The following example shows logic analyzer data being returned to a string variable with headers off:

```
10 OUTPUT XXX;":SYSTEM:HEADER OFF"
20 DIM Rang$[30]
30 OUTPUT XXX;":MACHINE1:TWAVEFORM:RANGE?"
40 ENTER XXX;Rang$
50 PRINT Rang$
60 END
```

After running this program, the controller displays:

+1.00000E-05

Numeric Base

Most numeric data will be returned in the same base as shown on screen. When the prefix #B precedes the returned data, the value is in the binary base. Likewise, #Q is the octal base and #H is the hexadecimal base. If no prefix precedes the returned numeric data, then the value is in the decimal base.

Numeric Variables

If your host language can convert from ASCII to a numeric format, then you can use numeric variables. Turning off the response headers will help you avoid accidently trying to convert the header into a number.

The following example shows logic analyzer data being returned to a numeric variable.

```
10 OUTPUT XXX;":SYSTEM:HEADER OFF"
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:RANGE?"
30 ENTER XXX;Rang
40 PRINT Rang
50 END
```

This time the format of the number (such as whether or not exponential notation is used) is dependant upon your host language. In BASIC, the output would look like:

1.E-5

Definite-Length Block Response Data

Definite-length block response data allows any type of device-dependent data to be transmitted over the system interface as a series of 8-bit binary data bytes. This is particularly useful for sending large quantities of data or 8-bit extended ASCII codes. The syntax is a pound sign (#) followed by a non-zero digit representing the number of digits in the decimal integer. After the non-zero digit is the decimal integer that states the number of 8-bit data bytes being sent. This is followed by the actual data.

For example, for transmitting 80 bytes of data, the syntax would be:

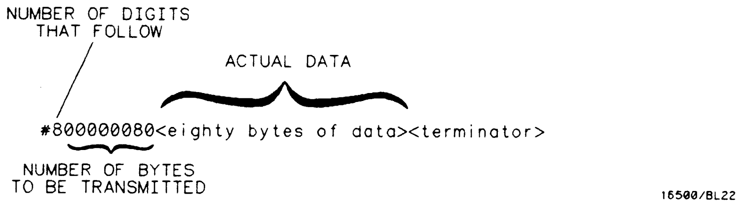


Figure 1-2. Definite-length Block Response Data

The "8" states the number of digits that follow, and "00000080" states the number of bytes to be transmitted.

Note Indefinite-length block data is not supported on the HP.1652B/53B.

Multiple Queries You can send multiple queries to the instrument within a single program message, but you must also read them back within a single program message. This can be accomplished by either reading them back into a string variable or into multiple numeric variables. For example, you could read the result of the query :SYSTEM:HEADER?;LONGFORM? into the string variable Results$ with the command:

```
ENTER XXX; Results$
```

When you read the result of multiple queries into string variables, each response is separated by a semicolon. For example, the response of the query :SYSTEM:HEADER?:LONGFORM? with HEADER and LONGFORM on would be:

```
:SYSTEM:HEADER 1;:SYSTEM:LONGFORM 1
```

If you do not need to see the headers when the numeric values are returned, then you could use following program message to read the query :SYSTEM:HEADERS?;LONGFORM? into multiple numeric variables:

```
ENTER XXX; Result1, Result2
```

Note When you are receiving numeric data into numeric variables, the headers should be turned off. Otherwise the headers may cause misinterpretation of returned data.

Instrument Status Status registers track the current status of the instrument. By checking the instrument status, you can find out whether an operation has been completed, whether the instrument is receiving triggers, and more. Appendix B, "Status Reporting," explains how to check the status of the instrument.

2 - Programming Over HP-IB

Programming Over HP-IB 2

Introduction

This section describes the interface functions and some general concepts of the HP-IB. In general, these functions are defined by IEEE 488.1 (HP-IB bus standard). They deal with general bus management issues, as well as messages which can be sent over the bus as bus commands.

Interface Capabilities

The interface capabilities of the HP 1652B/53B, as defined by IEEE 488.1 are SH1, AH1, T5, TE0, L3, LE0, SR1, RL1, PP0, DC1, DT1, C0, and E2.

Command and Data Concepts

The HP-IB has two modes of operation: command mode and data mode. The bus is in command mode when the ATN line is true. The command mode is used to send talk and listen addresses and various bus commands, such as a group execute trigger (GET). The bus is in the data mode when the ATN line is false. The data mode is used to convey device-dependent messages across the bus. These device-dependent messages include all of the instrument commands and responses found in chapters 5 through 27 of this manual.

Addressing

By using the front-panel I/O and SELECT keys, the HP-IB interface can be placed in either talk only mode "Printer connected to HP-IB" or addressed talk/listen mode "Controller connected to HP-IB" (see "I/O Port Configuration" in Chapter 5 of the *HP 1652B/HP 1653B Front-Panel Reference* manual. Talk only mode must be used when you want the instrument to talk directly to a printer without the aid of a controller. Addressed talk/listen mode is used when the instrument will operate in conjunction with a controller. When the instrument is in the addressed talk/listen mode, the following is true:

- Each device on the HP-IB resides at a particular address ranging from 0 to 30.
- The active controller specifies which devices will talk, and which will listen.
- An instrument, therefore, may be talk addressed, listen addressed, or unaddressed by the controller.

If the controller addresses the instrument to talk, it will remain configured to talk until it receives an interface clear message (IFC), another instrument's talk address (OTA), its own listen address (MLA), or a universal untalk (UNT) command.

If the controller addresses the instrument to listen, it will remain configured to listen until it receives an interface clear message (IFC) its own talk address (MTA), or a universal unlisten (UNL) command.

Communicating Over the HP-IB Bus (HP 9000 Series 200/300 Controller)

Since HP-IB can address multiple devices through the same interface card, the device address passed with the program message must include not only the correct instrument address, but also the correct interface code.

Interface Select Code (Selects Interface). Each interface card has its own interface select code. This code is used by the controller to direct commands and communications to the proper interface. The default is always "7" for HP-IB controllers.

Instrument Address (Selects Instrument). Each instrument on the HP-IB port must have a unique instrument address between decimal 0 and 30. The device address passed with the program message must include not only the correct instrument address, but also the correct interface select code.

DEVICE ADDRESS = (Interface Select Code) X 100 + (Instrument Address)

For example, if the instrument address for the HP 1652B/53B is 4 and the interface select code is 7, when the program message is passed, the routine performs its function on the instrument at device address 704.

Local, Remote, and Local Lockout

The local, remote, and remote with local lockout modes may be used for various degrees of front-panel control while a program is running. The instrument will accept and execute bus commands while in local mode, and the front panel will also be entirely active. If the HP 1652B/53B is in remote mode, the instrument will go from remote to local with any front panel activity. In remote with local lockout mode, all controls (except the power switch) are entirely locked out. Local control can only be restored by the controller.

Note

Cycling the power will also restore local control, but this will also reset certain HP-IB states.

The instrument is placed in remote mode by setting the REN (Remote Enable) bus control line true, and then addressing the instrument to listen. The instrument can be placed in local lockout mode by sending the local lockout (LLO) command (see SYSTem:LOCKout in chapter 6). The instrument can be returned to local mode by either setting the REN line false, or sending the instrument the go to local (GTL) command.

Bus Commands

The following commands are IEEE 488.1 bus commands (ATN true). IEEE 488.2 defines many of the actions which are taken when these commands are received by an instrument.

Device Clear

The device clear (DCL) or selected device clear (SDC) commands clear the input and output buffers, reset the parser, clear any pending commands, and clear the Request-OPC flag.

Group Execute Trigger (GET)

The group execute trigger command will cause the same action as the START command for Group Run: the instrument will acquire data for the active waveform and listing display(s).

Interface Clear (IFC)

This command halts all bus activity. This includes unaddressing all listeners and the talker, disabling serial poll on all devices, and returning control to the system controller.

3 - Programming
Over RS-232C

Programming Over RS-232C 3

Introduction

This section describes the interface functions and some general concepts of the RS-232C. The RS-232C interface on this instrument is Hewlett-Packard's implementation of EIA Recommended Standard RS-232C, "Interface Between Data Terminal Equipment and Data Communications Equipment Employing Serial Binary Data Interchange." With this interface, data is sent one bit at a time and characters are not synchronized with preceding or subsequent data characters. Each character is sent as a complete entity without relationship to other events.

Interface Operation

The HP 1652B/53B can be programmed with a controller over RS-232C using either a minimum three-wire or extended hardwire interface. The operation and exact connections for these interfaces are described in more detail in the following sections. When you are programming an HP 1652B/53B over RS-232C with a controller, you are normally operating directly between two DTE (Data Terminal Equipment) devices as compared to operating between a DTE device and a DCE (Data Communications Equipment) device.

When operating directly between two DTE devices, certain considerations must be taken into account. For three-wire operation, XON/XOFF must be used to handle protocol between the devices. For extended hardwire operation, protocol may be handled either with XON/XOFF or by manipulating the CTS and RTS lines of the RS-232C link. For both three-wire and extended hardwire operation, the DCD and DSR inputs to the HP 1652B/53B must remain high for proper operation.

With extended hardwire operation, a high on the CTS input allows the HP 1652B/53B to send data and a low on this line disables the HP 1652B/53B data transmission. Likewise, a high on the RTS line allows the controller to send data and a low on this line signals a request for the controller to disable data transmission. Since three-wire operation has no control over the CTS input, internal pull-up resistors in the HP 1652B/53B assure that this line remains high for proper three-wire operation.

Cables

Selecting a cable for the RS-232C interface is dependent on your specific application. The following paragraphs describe which lines of the HP 1652B/53B are used to control the operation of the RS-232C relative to the HP 1652B/53B. To locate the proper cable for your application, refer to the reference manual for your controller. This manual should address the exact method your controller uses to operate over the RS-232C bus.

Minimum Three-Wire Interface with Software Protocol

With a three-wire interface, the software (as compared to interface hardware) controls the data flow between the HP 1652B/53B and the controller. This provides a much simpler connection between devices since you can ignore hardware handshake requirements. The HP 1652B/53B uses the following connections on its RS-232C interface for three-wire communication:

- Pin 7 SGND (Signal Ground)
- Pin 2 TD (Transmit Data from HP 1652B/53B)
- Pin 3 RD (Receive Data into HP 1652B/53B)

The TD (Transmit Data) line from the HP 1652B/53B must connect to the RD (Receive Data) line on the controller. Likewise, the RD line from the HP 1652B/53B must connect to the TD line on the controller. Internal pull-up resistors in the HP 1652B/53B assure the DCD, DSR, and CTS lines remain high when you are using a three-wire interface.

Note

The three-wire interface provides no hardware means to control data flow between the controller and the HP 1652B/53B. XON/OFF protocol is the only means to control this data flow.

Extended Interface with Hardware Handshake

With the extended interface , both the software and the hardware can control the data flow between the HP 1652B/53B and the controller. This allows you to have more control of data flow between devices. The HP 1652B/53B uses the following connections on its RS-232C interface for extended interface communication:

- Pin 7 SGND (Signal Ground)
- Pin 2 TD (Transmit Data from HP 1652B/53B)
- Pin 3 RD (Receive Data into HP 1652B/53B)

The additional lines you use depends on your controller's implementation of the extended hardwire interface.

- Pin 4 RTS (Request To Send) is an output from the HP 1652B/53B which can be used to control incoming data flow.
- Pin 5 CTS (Clear To Send) is an input to the HP 1652B/53B which controls data flow from the HP 1652B/53B.
- Pin 6 DSR (Data Set Ready) is an input to the HP 1652B/53B which controls data flow from the HP 1652B/53B within two bytes.
- Pin 8 DCD (Data Carrier Detect) is an input to the HP 1652B/53B which controls data flow from the HP 1652B/53B within two bytes.
- Pin 20 DTR (Data Terminal Ready) is an output from the HP 1652B/53B which is enabled as long as the HP 1652B/53B is turned on.

The TD (Transmit Data) line from the HP 1652B/53B must connect to the RD (Receive Data) line on the controller. Likewise, the RD line from the HP 1652B/53B must connect to the TD line on the controller.

The RTS (Request To Send), is an output from the HP 1652B/53B which can be used to control incoming data flow. A true on the RTS line allows the controller to send data and a false on this line signals a request for the controller to disable data transmission.

The CTS (Clear To Send), DSR (Data Set Ready), and DCD (Data Carrier Detect) lines are inputs to the HP 1652B/53B which control data flow from the HP 1652B/53B (Pin 2). Internal pull-up resistors in the HP 1652B/53B assure the DCD and DSR lines remain high when they are not connected. If DCD or DSR are connected to the controller, the controller must keep these lines and the CTS line high to enable the HP 1652B/53B to send data to the controller. A low on any one of these lines will disable the HP 1652B/53B data transmission. Dropping the CTS line low during data transmission will stop HP 1652B/53B data transmission immediately. Dropping either the DSR or DCD line low during data transmission will stop HP 1652B/53B data transmission, but as many as two additional bytes may be transmitted from the HP 1652B/53B.

Cable Example

Figure 3-1 is an example of how to connect the HP 1652B/53B to the HP 98628A Interface card of an HP 9000 series 200/300 controller. For more information on cabling, refer to the reference manual for your specific controller.

Note

Since this example does not have the correct connections for hardware handshake, XON/XOFF protocol must be used when connecting the HP 1652B/53B as shown in figure 3-1

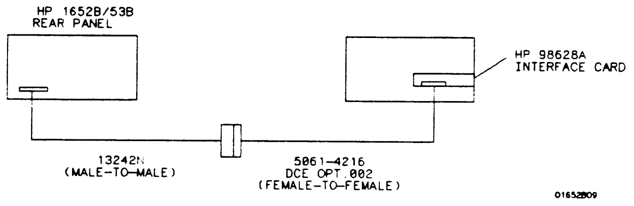


Figure 3-1. Cable Example

Configuring the Instrument Interface

The front-panel I/O menu key allows you access to the RS-232C Configuration menu where the RS-232C interface is configured.

If you are not familiar with how to configure the RS-232C interface, refer to the *HP 1652B/53B Front-panel Reference* manual.

Interface Capabilities

The baud rate, stop bits, parity, protocol, and data bits must be configured exactly the same for both the controller and the HP 1652B/53B to properly communicate over the RS-232C bus. The HP 1652B/53B RS-232C interface capabilities are listed below:

- Baud Rate: 110, 300, 600, 1200, 2400, 4800, 9600, or 19.2 k
- Stop Bits: 1, 1.5, or 2
- Parity: None, Odd, or Even
- Protocol: None or XON/XOFF
- Data Bits: 8

Protocol

NONE. With a three-wire interface, selecting NONE for the protocol does not allow the sending or receiving device to control data flow. No control over the data flow increases the possibility of missing data or transferring incomplete data.

With an extended hardwire interface, selecting NONE allows a hardware handshake to occur. With hardware handshake, hardware signals control data flow.

XON/XOFF. XON/XOFF stands for Transmit On/Transmit Off. With this mode the receiver (controller or HP 1652B/53B) controls data flow and can request that the sender (HP 1652B/53B or controller) stop data flow. By sending XOFF (ASCII 19) over its transmit data line, the receiver requests that the sender disables data transmission. A subsequent XON (ASCII 17) allows the sending device to resume data transmission.

Data Bits Data bits are the number of bits sent and received per character that represent the binary code of that character. Characters consist of either 7 or 8 bits, depending on the application. The HP 1652B/53B supports 8 bit only.

8 Bit Mode. Information is usually stored in bytes (8 bits at a time). With 8-bit mode, you can send and receive data just as it is stored, without the need to convert the data.

Note The controller and the HP 1652B/53B must be in the same bit mode to properly communicate over the RS-232C. This means that both the controller and the HP 1652B/53B must have the capability to send and receive 8 bit data.

For more information on the RS-232C interface, refer to the *HP 1652B/HP 1653B Front-Panel Reference* Manual. For information on RS-232C voltage levels and connector pinouts, refer to the *HP 1652B/53B Service Manual.*

Communicating Over the RS-232C Bus (HP 9000 Series 200/300 Controller)

Each RS-232C interface card has its own interface select code. This code is used by the controller to direct commands and communications to the proper interface by specifying the correct interface code for the device address.

Generally, the interface select code can be any decimal value between 0 and 31, except for those interface codes which are reserved by the controller for internal peripherals and other internal interfaces. This value can be selected through switches on the interface card. For more information, refer to the reference manual for your interface card or controller.

For example, if your RS-232C interface select code is 9, the device address required to communicate over the RS-232C bus is 9.

Lockout Command

To lockout the front panel controls use the SYSTem command LOCKout. When this function is on, all controls (except the power switch) are entirely locked out. Local control can only be restored by sending the command :LOCKout OFF. For more information on this command see the chapter "System Commands" in this manual.

Note

Cycling the power will also restore local control, but this will also reset certain RS-232C states.

4 - Programming and
Documentation Conventions

Programming and Documentation Conventions

4

Introduction

This section covers the programming conventions used in programming the instrument, as well as the documentations conventions used in this manual. This chapter also contains a detailed description of the command tree and command tree traversal.

Truncation Rule

The truncation rule for the keywords used in headers and parameters is:

If the longform has four or fewer characters, there is no change in the shortform. When the longform has more than four characters the shortform is just the first four characters, unless the fourth character is a vowel. In that case only the first three characters are used.

Note

There are some commands that do not conform to the truncation rule by design. These will be noted in their respective description pages.

Some examples of how the truncation rule is applied to various commands are shown in table 4-1.

Longform	Shortform
OFF	OFF
DATA	DATA
START	STAR
LONGFORM	LONG
DELAY	DEL
ACCUMULATE	ACC

Table 4-1. Keyword Truncation

Infinity Representation

The representation of infinity is 9.9E + 37 for real numbers and 32767 for integers. This is also the value returned when a measurement cannot be made.

Sequential and Overlapped Commands

IEEE 488.2 makes the distinction between sequential and overlapped commands. Sequential commands finish their task before the execution of the next command starts. Overlapped commands run concurrently, and therefore the command following an overlapped command may be started before the overlapped command is completed. The overlapped commands for the HP 1652B/53B are STARt, STOP, and AUToscale.

Response Generation

IEEE 488.2 defines two times at which query responses may be buffered. The first is when the query is parsed by the instrument and the second is when the controller addresses the instrument to talk so that it may read the response. The HP 1652B/53B will buffer responses to a query when it is parsed.

Syntax Diagrams

At the beginning of each of the following chapters are syntax diagrams showing the proper syntax for each command. All characters contained in a circle or oblong are literals, and must be entered exactly as shown. Words and phrases contained in rectangles are names of items used with the command and are described in the accompanying text of each command. Each line can only be entered from one direction as indicated by the arrow on the entry line. Any combination of commands and arguments that can be generated by following the lines in the proper direction is syntactically correct. An argument is optional if there is a path around it. When there is a rectangle which contains the word "space," a white space character must be entered. White space is optional in many other places.

Notation Conventions and Definitions

The following conventions are used in this manual when describing programming rules and examples:

< > — Angular brackets enclose words or characters that are used to symbolize a program code parameter or a bus command.

::= — "is defined as." For example, A ::= B indicates that A can be replaced by B in any statement containing A .

| — "or": indicates a choice of one element from a list. For example, A | B indicates A or B, but not both.

... — An ellipsis (trailing dots) is used to indicate that the preceding element may be repeated one or more times.

[ ] — Square brackets indicate that the enclosed items are optional.

{ } — When several items are enclosed by braces and separated by |s, one, and only one of these elements must be selected.

XXX — Three Xs after an ENTER or OUTPUT statement represent the device address required by your controller.

In addition, the following definition is used:

<NL> ::= Linefeed (ASCII decimal 10).

The Command Tree

The command tree (figure 4-1) shows all commands in the HP 1652B/53B logic analyzers and the relationship of the commands to each other. Parameters are not shown in this figure. The command tree allows you to see what the HP 1652B/53B's parser expects to receive. All legal headers can be created by traversing down the tree, adding keywords until the end of a branch has been reached.

Command Types

As shown in chapter 1's "Header Types" section, there are three types of headers. Each header has a corresponding command type. This section shows how they relate to the command tree.

System Commands. The system commands reside at the top level of the command tree. These commands are always parsable if they occur at the beginning of a program message, or are preceded by a colon. START and STOP are examples of system commands.

Subsystem Commands. Subsystem commands are grouped together under a common node of the tree, such as the MMEMORY commands.

Common Commands. Common commands are independent of the tree, and do not affect the position of the parser within the tree. *CLS and *RST are examples of common commands.

Tree Traversal Rules

Command headers are created by traversing down the command tree. For each group of keywords not separated by a branch, one keyword must be selected. As shown on the tree, branches are always preceded by colons. Do not add spaces around the colons. The following two rules apply to traversing the tree:

A leading colon (the first character of a header) or a <terminator> places the parser at the root of the command tree.

Executing a subsystem command places you in that subsystem (until a leading colon or a <terminator> is found). The parser will stay at the colon above the keyword where the last header terminated. Any command below that point can be sent within the current program message without sending the keywords(s) which appear above them.

Examples The following examples are written using HP BASIC 4.0 on a HP 9000 Series 200/300 Controller. The quoted string is placed on the bus, followed by a carriage return and linefeed (CRLF).

The three Xs (XXX) shown in this manual after an ENTER or OUTPUT statement represents the device address required by your controller.

Example 1

```
OUTPUT XXX;":SYSTEM:HEADER ON;LONGFORM ON"
```

In example 1, the colon between SYSTEM and HEADER is necessary since SYSTEM:HEADER is a compound command. The semicolon between the HEADER command and the LONGFORM command is the required <program message unit separator>. The LONGFORM command does not need SYSTEM preceding it, since the SYSTEM:HEADER command sets the parser to the SYSTEM node in the tree.

Example 2

```
OUTPUT XXX;":MMEMORY:INITIALIZE;STORE 'FILE__','FILE DESCRIPTION'"
```

or

```
OUTPUT XXX;":MMEMORY:INITIALIZE"
OUTPUT XXX;":MMEMORY:STORE 'FILE__','FILE DESCRIPTION'"
```

In the first line of example 2, the "subsystem selector" is implied for the STORE command in the compound command. The STORE command must be in the same program message as the INITIALIZE command, since the <program message terminator> will place the parser back at the root of the command tree.

A second way to send these commands is by placing "MMEMORY:" before the STORE command as shown in the fourth line of example 2.

Example 3

```
OUTPUT XXX;":MMEM:CATALOG?;:SYSTEM:PRINT ALL"
```

In example 3, the leading colon before SYSTEM tells the parser to go back to the root of the command tree. The parser can then see the SYSTEM:PRINT command.

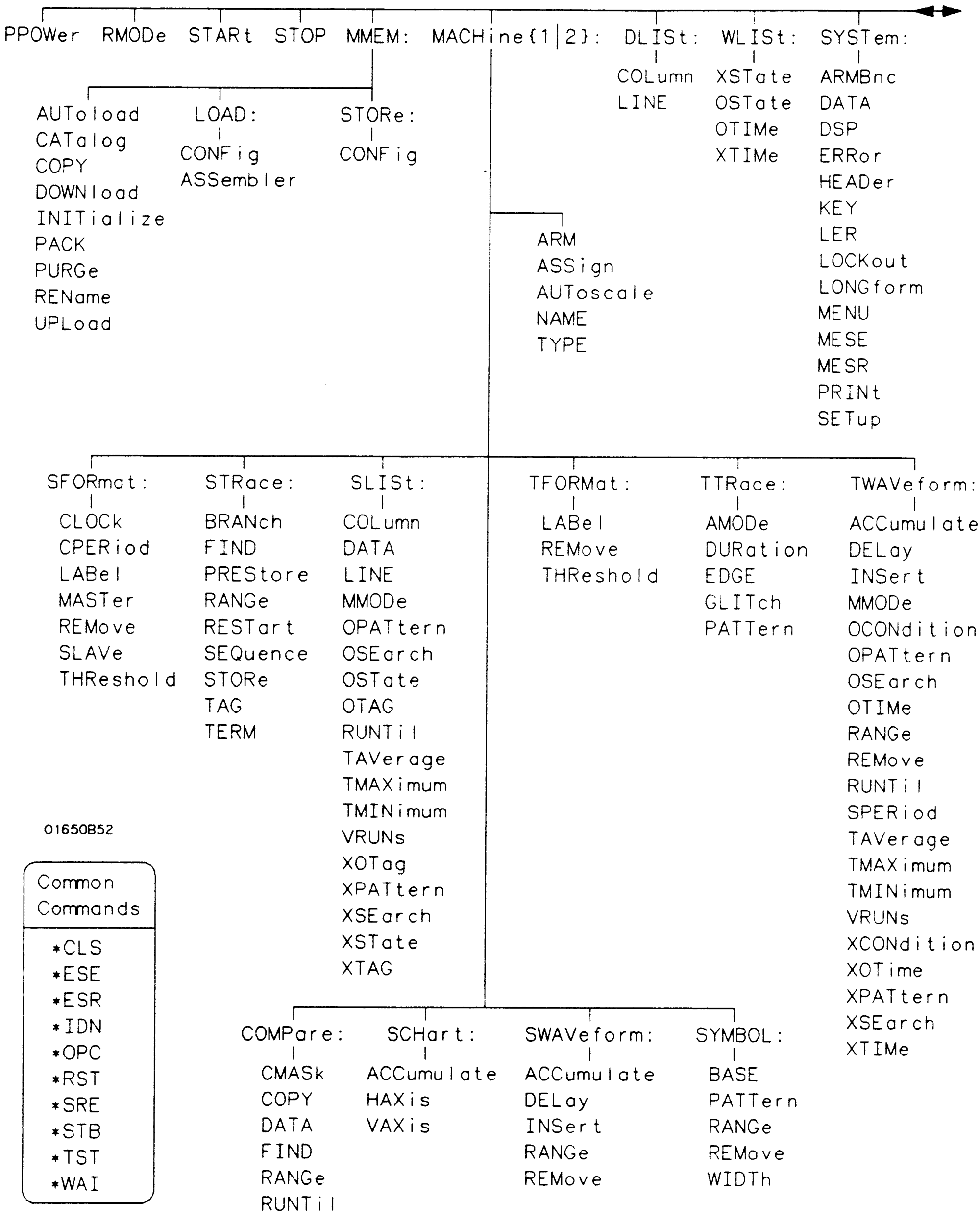


Figure 4-1. HP 1652B/53B Command Tree

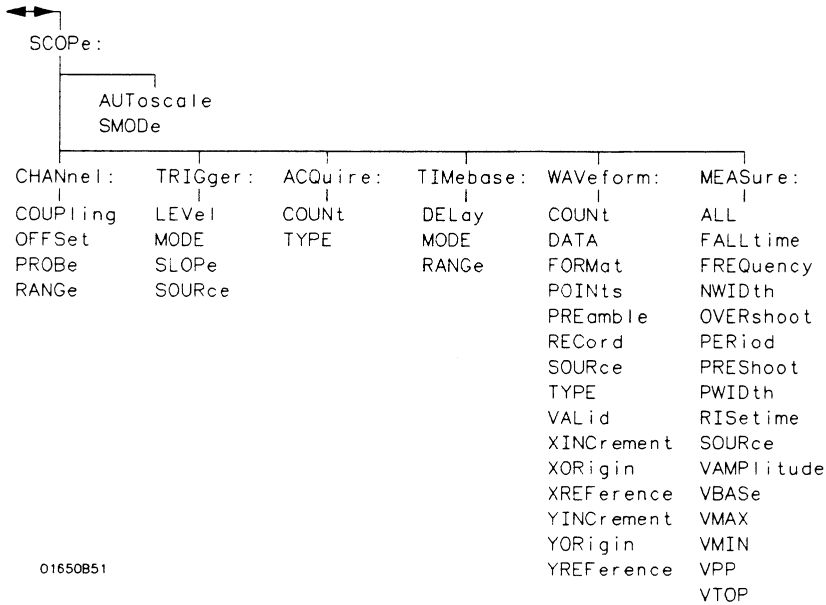


Figure 4-1. HP 1652B/53B Command Tree (continued)

Table 4-2. Alphabetic Command Cross-Reference

Command	Where used	Command	Where used
ACCumulate	SCHart, SWAVeform, TWAVeform	GLITch	TTRace
		HAXis	SCHart
ALL	MEASure	HEADer	System
AMODe	TTRace	INITialize	MMEMory
ARM	MACHine	INSert	SWAVeform, TWAVeform
ARMBnc	System	KEY	System
ASSign	MACHine	LABel	SFORmat, TFORmat
AUToload	MMEMory	LER	System
AUToscale	MACHine, SCOPe	LEVel	TRIGger
BASE	SYMBol	LINE	DLISt, SLISt
BRANch	STRace	LOAD	MMEMory
CATalog	MMEMory	LOCKout	System
CLOCk	SFORmat	LONGform	System
CMASk	COMPare	MASTer	SFORmat
COLumn	DLISt, SLISt	MENU	System
COPY	COMPare, MMEMory	MESE	System
COUNt	ACQuire, WAVeform	MESR	System
COUPling	CHANnel	MMODe	SLISt
CPERiod	SFORmat	MODE	TIMebase, TRIGger
DATA	COMPare, SLISt, System, WAVEform	NAME	MACHine
		NWIDth	MEASure
DELay	SWAVeform, TIMebase, TWAVeform	OCONdition	TWAVeform
		OFFSet	CHANnel
DOWNload	MMEMory	OPATtern	SLISt
DSP	System	MMODe	TWAVeform
DURation	TTRace	OPATtern	TWAVeform
EDGE	TTRace	OSEarch	SLISt, TWAVeform
ERRor	System	OSTate	SLISt, WLISt
FALLtime	MEASure	OTAG	SLISt
FIND	COMPare, STRace	OTIMe	TWAVeform, WLISt
FORMat	WAVeform	OVERshoot	MEASure
FREQuency	MEASure	PACK	MMEMory

Table 4-2. Alphabetic Command Cross-Reference (continued)

Command	Where used	Command	Where used
PATTern	SYMBol, TRace	STORe	MMEMory, STRace
PERiod	MEASure	TAG	STRace
POINts	WAVeform	TAVerage	SLISt, TWAVeform
PPOWer	System	TERM	STRace
PREamble	WAVeform	THReshold	SFORmat, TFORmat
PREShoot	MEASure	TMAXimum	SLISt, TWAVeform
PREStore	STRace	TMINimum	SLISt, TWAVeform
PRINt	System	TYPE	ACQuire, MACHine, WAVeform
PROBe	CHANnel	UPLoad	MMEMory
PURGe	MMEMory	VALid	WAVeform
PWIDth	MEASure	VAMPlitude	MEASure
RANGe	CHANnel, COMPare, STRace, SWAVeform, SYMBol, TIMebase, TWAVeform	VAXis	SCHart
RECord	WAVeform	VBASe	MEASure
REMove	SFORmat, SWAVeform, Symbol, TFORmat, TWAVeform	VMAX	MEASure
REName	MMEMory	VMIN	MEASure
RESTart	STRace	VPP	MEASure
RISetime	MEASure	VRUNs	SLISt, TWAVeform
RMODe	System	VTOP	MEASure
RUNTil	COMPare, SLISt, WAVeform	WIDTh	SYMBol
SEQuence	STRace	XCONdition	TWAVeform
SETup	System	XINCrement	WAVeform
SLAVe	SFORmat	XORigin	WAVeform
SLOPe	TRIGger	XOTag	SLISt
SMODe	SCOPe	XOTime	TWAVeform
SOURce	MEASure, TRIGger, WAVeform	XPATtern	SLISt, TWAVeform
SPERiod	TWAVeform	XREFerence	WAVeform
STARt	System	XSEarch	SLISt, TWAVeform
STOP	System	XSTate	SLISt, WLISt
		XTAG	SLISt
		XTIMe	TWAVeform, WLISt
		YINCrement	WAVeform
		YORigin	WAVeform
		YREFerence	WAVeform

Command Set Organization

The command set for the HP 1652B/53B logic analyzer is divided into 24 separate groups: common commands, system commands and 22 sets of subsystem commands. Each of the 24 groups of commands is described in the following chapters. Each of the chapters contain a brief description of the subsystem, a set of syntax diagrams for those commands, and finally, the commands for that subsystem in alphabetical order. The commands are shown in the longform and shortform using upper and lowercase letters. As an example AUToload indicates that the longform of the command is AUTOLOAD and the shortform of the command is AUT. Each of the commands contain a description of the command and its arguments, the command syntax, and a programming example.

Subsystems

There are 19 subsystems in this instrument. In the command tree (figure 4-1) they are shown as branches, with the node above showing the name of the subsystem. Only one subsystem may be selected at a time. At power on, the command parser is set to the root of the command tree, and therefore no subsystem is selected. The 22 subsystems in the HP 1652B/53B are:

- SYSTem - controls some basic functions of the instrument.
- MMEMory - provides access to the internal disk drive.
- DLISt - allows access to the dual listing function of two state analyzers.
- WLISt - allows access to the mixed (timing/state) functions.
- MACHine - provides access to analyzer functions and subsystems.
- SFORmat - allows access to the state format functions.
- STRace - allows access to the state trace functions.
- SLISt - allows access to the state listing functions.
- SWAVeform - allows access to the state waveforms functions.
- SCHart - allows access to the state chart functions.
- COMPare - allows access to the compare functions.
- TFORmat - allows access to the timing format functions.
- TTRace - allows access to the timing trace functions.
- TWAVeform - allows access to the timing waveforms functions.
- SYMBol - allows access to the symbol specification functions.
- SCOPe - provides access to oscilloscope functions and subsystems.
- CHANnel - provides access to the vertical axis of the oscilloscope
- TRIGger - allows control of the trigger conditions
- ACQuire - allows control of how the oscilloscope data is acquired.

- TIMebase - allows control of the timebase (horizontal axis) of the oscilloscope.
- WAVeform - allows access to data transfer commands.
- MEASure - allows you to control automated measurements.

Program Examples

The program examples given for each command in the following chapters and appendices were written on an HP 9000 Series 200/300 controller using the HP BASIC 4.0 language. The programs always assume a generic address for the HP 1652/53B of XXX.

In the following examples, special attention should be paid to the ways in which the command and/or query can be sent. Keywords can be sent using either the longform or shortform (if one exists for that word). With the exception of some string parameters, the parser is not case-sensitive. Upper-case (capital) and lower-case (small) letters may be mixed freely. System commands like HEADer and LONGform allow you to dictate what forms the responses take, but have no affect on how you must structure your commands and queries.

The following commands all set Timing Waveform Delay to 100 ms.

- keywords in longform, numbers using the decimal format.

```
OUTPUT XXX;":MACHINE1:TWAVEFORM:DELAY .1"
```

- keywords in shortform, numbers using an exponential format.

```
OUTPUT XXX;":MACH1:TWAV:DEL 1E-1"
```

- keywords in shortform using lower-case letters, numbers using a suffix.

```
OUTPUT XXX;":mach1:twav:del 100ms"
```

Note

In these examples, the colon shown as the first character of the command is optional on the HP 1652B/53B.

The space between DELay and the argument is required.

Common Commands 5

Introduction

The common commands are defined by the IEEE 488.2 standard. These commands will be common to all instruments that comply with this standard.

The common commands control some of the basic instrument functions, such as instrument identification and reset, how status is read and cleared, and how commands and queries are received and processed by the instrument.

Common commands can be received and processed by the HP 1652B/53B whether they are sent over the bus by themselves or as part of a multiple-command string. If an instrument subsystem has been selected and a common command is received by the instrument, the instrument will remain in the selected subsystem. For example, if the instruction

```
":MMEMORY:INITIALIZE;*CLS; STORE 'FILE_','DESCRIPTION'"
```

is received by the instrument, the instrument will initialize the disk and store the file; and clear the status information. This would not be the case if some other type of command were received within the program message. For example, the program message

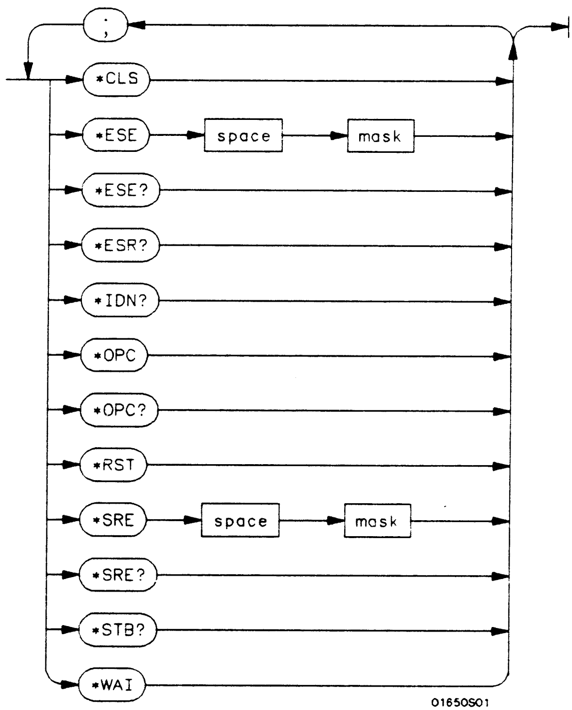
```
":MMEMORY:INITIALIZE;:SYSTEM:HEADERS ON:MMEMORY
:STORE 'FILE_','DESCRIPTION'"
```

would initialize the disk, turn headers on, then store the file. In this example :MMEMORY must be sent again in order to reenter the mmemory subsystem and store the file.

Each status register has an associated status enable (mask) register. By setting the bits in the mask value you can select the status information you wish to use. Any status bits that have not been masked (enabled in the enable register) will not be used to report status summary information to bits in other status registers.

Refer to appendix B, "Status Reporting," for a complete discussion of how to read the status registers and how to use the status information available from this instrument.

Refer to figure 5-1 for the common commands syntax diagram.



mask = *An integer, 0 through 255. This number is the sum of all the bits in the mask corresponding to conditions that are enabled. Refer to the *ESE and *SRE commands for bit definitions in the enable registers.*

Figure 5-1. Common Commands Syntax Diagram

*CLS (Clear Status) command

The *CLS common command clears the status data structures, including the device defined error queue. If the *CLS command immediately follows a <terminator>, the output queue and the MAV (Message Available) bit will be cleared.

Command Syntax: *CLS

Example:

```
OUTPUT XXX;"*CLS"
```

Note Refer to appendix B, "Status Reporting," for a complete discussion of status.

*ESE (Event Status Enable) command/query

The *ESE command sets the Standard Event Status Enable Register bits. The Standard Event Status Enable Register contains a mask value for the bits to be enabled in the Standard Event Status Register. A one in the Standard Event Status Enable Register will enable the corresponding bit in the Standard Event Status Register. A zero will disable the bit. Refer to table 4-1 for information about the Standard Event Status Enable Register bits, bit weights, and what each bit masks.

The *ESE query returns the current contents of the enable register.

Note Refer to appendix B, "Status Reporting," for a complete discussion of status.

Command Syntax: *ESE <mask>

where:

<mask> ::= integer from 0 to 255

Example:

```
OUTPUT XXX;"*ESE 32"
```

In this example, the *ESE 32 command will enable CME (Command Error), bit 5 of the Standard Event Status Enable Register. Therefore, when a command error occurs, the event summary bit (ESB) in the Status Byte Register will also be set.

Query Syntax: *ESE?

Returned Format: <mask> <NL>

Example:

```
10 DIM Event$[100]
20 OUTPUT XXX;"*ESE?"
30 ENTER XXX;Event$
40 PRINT Event$
50 END
```

Table 5-1. Standard Event Status Enable Register

Bit	Weight	Enables
7	128	PON - Power On
6	64	URQ - User Request
5	32	CME - Command Error
4	16	EXE - Execution Error
3	8	DDE - Device Dependent Error
2	4	QYE - Query Error
1	2	RQC - Request Control
0	1	OPC - Operation Complete

High - enables the ESR bit

*ESR (Event Status Register) query

The *ESR query returns the contents of the Standard Event Status Register. Reading the register clears the Standard Event Status Register.

Note The bits in this register must be set by sending the *ESE command before sending the *ESR query (see "*ESE command/query" on page 5-4).

Query Syntax: *ESR?

Returned Format: <status> <NL>

where:

<status> ::= integer from 0 to 255

Example:

```
10 DIM Esr_event$[100]
20 OUTPUT XXX;"*ESR?"
30 ENTER XXX;Esr_event$
40 PRINT Esr_event$
50 END
```

With the example, if a command error has occurred the variable "Esr_event" will have bit 5 (the CME bit) set.

Table 4-2 shows the Standard Event Status Register. The table shows each bit in the Standard Event Status Register, and the bit weight. When you read Standard Event Status Register, the value returned is the total bit weights of all bits that are high at the time you read the byte.

Table 5-2. The Standard Event Status Register.

BIT	BIT WEIGHT	BIT NAME	CONDITION
7	128	PON	0 = Register read - not in power up mode 1 = Power up
6	64	URQ	0 = user request - not used - always zero
5	32	CME	0 = no command errors 1 = a command error has been detected
4	16	EXE	0 = no execution errors 1 = an execution error has been detected
3	8	DDE	0 = no device dependent errors 1 = a device dependent error has been detected
2	4	QYE	0 = no query errors 1 = a query error has been detected
1	2	RQC	0 = request control - NOT used - always 0
0	1	OPC	0 = operation is not complete 1 = operation is complete

0 = False = Low
1 = True = High

*IDN (Identification Number) query

The *IDN? query allows the instrument to identify itself. It returns the string:

"HEWLETT-PACKARD,1652B,0,REV <revision code>"

An *IDN? query must be the last query in a message. Any queries after the *IDN? in the program message will be ignored.

Query Syntax: *IDN?

Returned Format: HEWLETT-PACKARD,1652B,0,REV <revision code>

where:

<revision code> ::= four-digit code representing ROM revision

Example:

```
10 DIM Id$[100]
20 OUTPUT XXX;"*IDN?"
30 ENTER XXX;Id$
40 PRINT Id$
50 END
```

*OPC (Operation Complete) command/query

The *OPC command will cause the instrument to set the operation complete bit in the Standard Event Status Register when all pending device operations have finished. The commands which affect this bit are the Overlapped Commands. An Overlapped Command is a command that allows execution of subsequent commands while the device operations initiated by the Overlapped Command are still in progress. The overlapped commands for the HP 1652B/53B are:

STARt
STOP
AUToscale

The *OPC query places an ASCII "1" in the output queue when all pending device operations have been completed.

Command Syntax: *OPC

Example: `OUTPUT XXX;"*OPC"`

Query Syntax: *OPC?

Returned Format: 1<NL>

Example:

```
10 DIM Status$[100]
20 OUTPUT XXX;"*OPC?"
30 ENTER XXX;Status$
40 PRINT Status$
50 END
```

*RST (Reset) command

The *RST command (488.2) sets the HP 1652B/53B to the power-up default settings as if no autoload file was present.

Command Syntax: `*RST`

Example: `OUTPUT XXX;"*RST"`

*SRE (Service Request Enable) command/query

The *SRE command sets the Service Request Enable Register bits. The Service Request Enable Register contains a mask value for the bits to be enabled in the Status Byte Register. A one in the Service Request Enable Register will enable the corresponding bit in the Status Byte Register. A zero will disable the bit. Refer to table 5-3 for the bits in the Service Request Enable Register and what they mask.

The *SRE query returns the current value.

Note Refer to appendix B, "Status Reporting," for a complete discussion of status.

Command Syntax: *SRE <mask>

where:

<mask> ::= integer from 0 to 255

Example:

```
OUTPUT XXX;"*SRE 16"
```

This example forces the MAV bit high (see table 5-3).

Query Syntax: *SRE?

Returned Format: <mask> <NL>

where:

<mask> ::= sum of all bits that are set - 0 through 255

Example:

```
10 DIM Sre_value$[100]
20 OUTPUT XXX;"*SRE?"
30 ENTER XXX;Sre_value$
40 PRINT Sre_value$
50 END
```

Table 5-3. HP 1652B/53B Service Request Enable Register

Bit	Weight	Enables
15-8		not used
7	128	not used
6	64	MSS - Master Summary Status
5	32	ESB - Event Status
4	16	MAV - Message Available
3	8	not used
2	4	not used
1	2	LCL - Local
0	1	MSB - Module Summary

*STB (Status Byte) query

The *STB query returns the current value of the instrument's status byte. The MSS (Master Summary Status) bit and not RQS (Request Service) bit is reported on bit 6. The MSS indicates whether or not the device has at least one reason for requesting service. Refer to table 5-4 for the meaning of the bits in the status byte.

Note Refer to appendix B, "Status Reporting," for a complete discussion of status.

Query Syntax: *STB?

Returned Format: <value><NL>

where:

<value> ::= integer from 0 to 255

Example:

```
10 DIM Stb_value$[100]
20 OUTPUT XXX;"*STB?"
30 ENTER XXX;Stb_value$
40 PRINT Stb_value$
50 END
```

Table 5-4. The Status Byte Register

BIT	BIT WEIGHT	BIT NAME	CONDITION
7	128	---	0 = not used
6	64	MSS	0 = instrument has no reason for service 1 = instrument is requesting service
5	32	ESB	0 = no event status conditions have occurred 1 = an enabled event status condition has occured
4	16	MAV	0 = no output messages are ready 1 = an output message is ready
3	8		not used
2	4	---	not used
1	2	LCL	0 = a remote-to-local transition has not occurred 1 = a remote-to-local transition has occurred
0	1	MSB	0 = HP 1652B/1653B has activity to report 1 = no activity to report

0 = False = Low
1 = True = High

*WAI (Wait) command

The *WAI command causes the device to wait until the completion of all overlapped commands before executing any further commands or queries. An overlapped command is a command that allows execution of subsequent commands while the device operations initiated by the overlapped command are still in progress. The overlapped commands for the HP 1652B/53B are:

STARt
STOP
AUToscale

Command Syntax: *WAI

Example: `OUTPUT XXX;"*WAI"`

6 - System Commands

System Commands 6

Introduction

System commands control the basic operation of the instrument including formatting query responses and enabling reading and writing to the advisory line of the instrument's display. They can be called at anytime. The HP 1652B/53B System commands are:

- ARMBnc
- DATA
- DSP (display)
- ERRor
- HEADer
- KEY
- LER (Local Event Register)
- LOCKout
- LONGform
- MENU
- MESE
- MESR
- PRINt
- SETup

In addition to the system commands, there is are three run control commands and a preprocessor power supply condition query. These commands are:

- PPOWer
- RMODe
- STARt
- STOP

The run control commands can be called at anytime and also control the basic operation of the logic analyzer. These commands are at the same level in the command tree as SYSTem; therefore they are not preceded by the :SYSTem header.

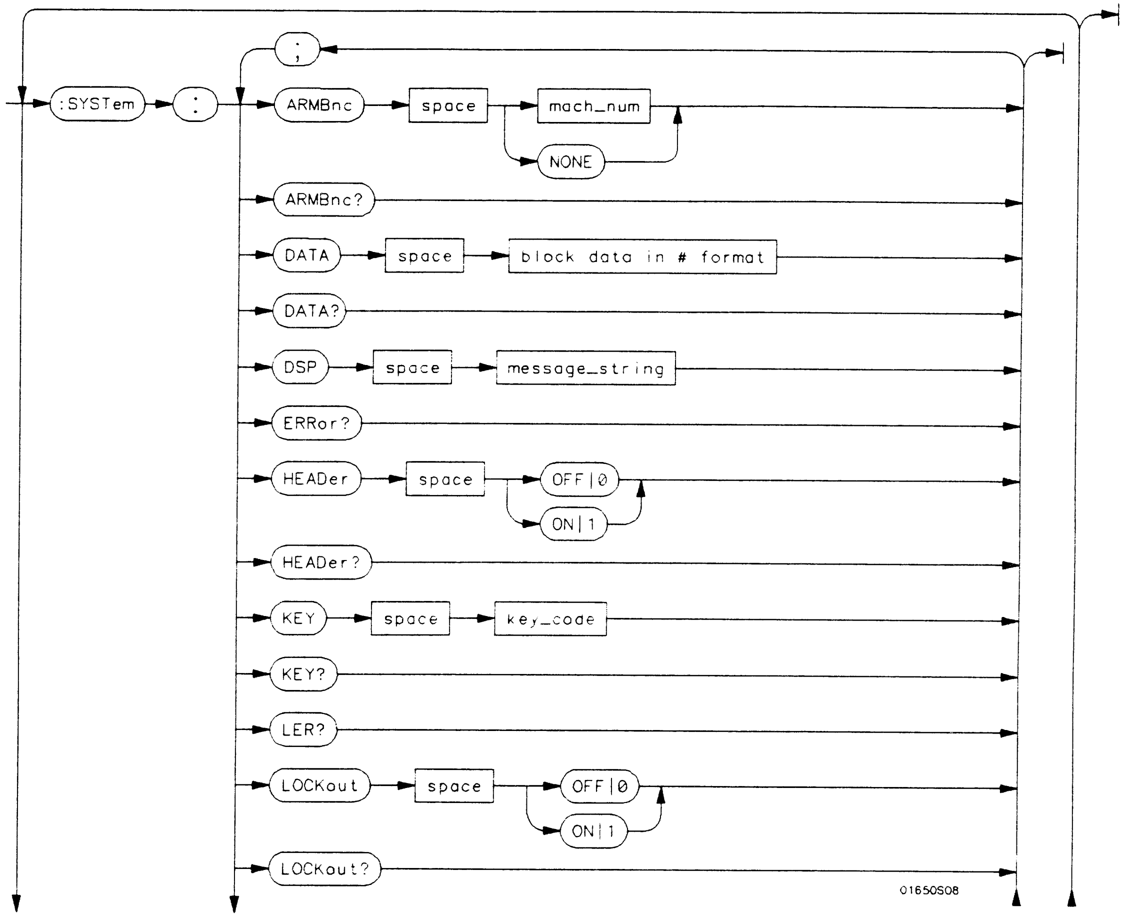
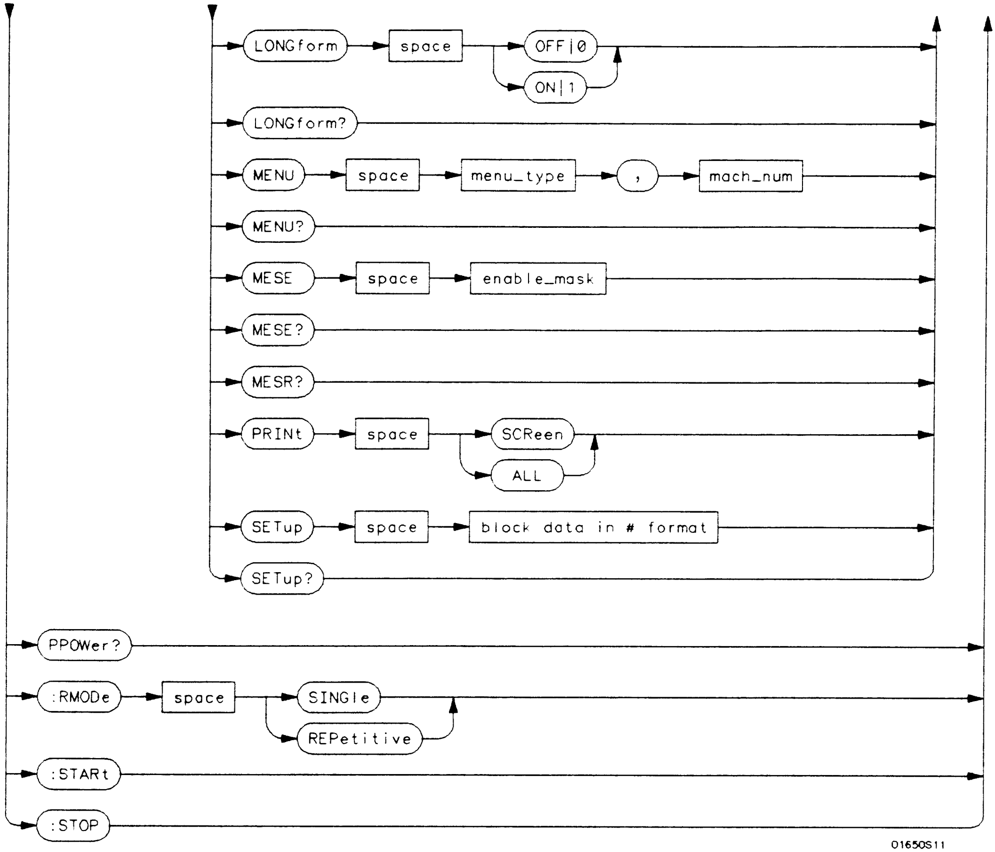


Figure 6-1. System Commands Syntax Diagram



value = *integer from 0 to 255.*
menu = *integer. Refer to the individual programming manuals for each module and the system for specific menu number definitions.*
enable_value = *integer from 0 to 255.*
index = *integer from 0 to 5.*
block_data = *data in IEEE 488.2 format.*
string = *string of up to 60 alphanumeric characters.*

Figure 6-1. System Commands Syntax Diagram (continued)

ARMBnc

command/query

The ARMBnc command selects the source that will generate the arm out signal that will appear on the rear panel BNC labelled External Trigger Out.

The ARMBnc query returns the source currently selected.

Command Syntax: :SYSTem:ARMBnc {MACHine{1|2}|SCOPe | NONE}

Example:

```
OUTPUT XXX;":SYSTEM:ARMBNC MACHINE1"
```

Query Syntax: :SYSTem:ARMBnc?

Returned Format: [:SYSTem:ARMBnc] {MACHine{1|2}|SCOPe |NONE}<NL>

Example:

```
10 DIM Mode$[100]
20 OUTPUT XXX;":ARMBNC?"
30 ENTER XXX;Mode$
40 PRINT Mode$
50 END
```

DATA

command/query

The DATA command allows you to send and receive acquired data to and from a controller in block form. This helps saving block data for:

- Re-loading to the logic analyzer
- Processing data later
- Processing data in the controller.

The format and length of block data depends on the instruction being used and the configuration of the instrument. This section describes each part of the block data as it will appear when used by the DATA instruction. The beginning byte number, the length in bytes, and a short description is given for each part of the block data. This is intended to be used primarily for processing of data in the controller.

Note Do not change the block data in the controller if you intend to send the block data back into the logic analyzer for later processing. Changes made to the block data in the controller could have unpredictable results when sent back to the logic analyer.

The SYSTem:DATA query returns the block data.

Note The data sent by the SYSTem:DATA query reflects the configuration of the machines when the last run was performed. Any changes made since then through either front-panel operations or programming commands do not affect the stored configuration.

For the DATA instruction, block data consists of either 14506 bytes containing logic analyzer only information or 26794 bytes containing both logic analyzer and oscilloscope information. This information is captured by the acquisition systems. The information for the logic analyzer will be in one of four formats depending on the type of data captured. The logic analyzer format is described in the "Acquisition Data Description" section in "Logic Analyzer Block Data." The oscilloscope format is described in the "Acquisition Data Description" section in "Oscilloscope Block Data." Since no parameter checking is performed, out-of-range values could cause instrument lockup; therefore, care should be taken when transferring the data string into the HP 1652B/53B.

The <block data> parameter can be broken down into a <block length specifier> and a variable number of <section>s.

The <block length specifier> always takes the form #8DDDDDDDD. Each D represents a digit (ASCII characters "0" through "9"). The value of the eight digits represents the total length of the block (all sections). For example, if the total length of the block is 14522 bytes, the block length specifier would be "#800014522".

Each <section> consists of a <section header> and <section data>. The <section data> format varies for each section and may be any length. For this instruction, the <section data> section is composed of a data preamble section and an acquisition data section.

Command Syntax: :SYSTem:DATA <block data>

Example: `OUTPUT XXX;":SYSTEM:DATA"` <block data>

where:

<block data>	::= <block length specifier> <section>...
<block length specifier>	::= #8<length>
<length>	::= the total length of all sections in byte format (must be represented with 8 digits)
<section>	::= <section header> <section data>
<section header>	::= 16 bytes, described in the following "Section Header" sections
<section data>	::= format depends on the type of data

Note

The total length of a section is 16 (for the section header) plus the length of the section data. So when calculating the value for <length>, don't forget to include the length of the section headers.

Query Syntax: :SYSTem:DATA?

Returned Format: [:SYSTem:DATA] <block data> <NL>

HP-IB Example:

```
10 DIM Num$[2], Block$[32000]     !  allocate enough memory for block data
20 OUTPUT XXX;":SYSTEM:HEAD OFF"
30 OUTPUT XXX;":SYSTEM:DATA?"     ! send data query
40 ENTER XXX USING "#,2A";Num$    !read in #8
50 ENTER XXX USING "#,8D";Blocklength! read in block length
60 ENTER XXX USING "-K";Block$    ! read in data
70 END
```

Logic Analyzer Block Data

The logic analyzer block data is described in the following sections. The oscilloscope block data is appended at the end of the logic analyzer block data when the oscilloscope is on and has acquired and stored waveform data. The oscilloscope block data is described in "Oscilloscope Block Data" later in this section.

Section Header Description

The section header uses bytes 1 through 16 (this manual begins counting at 1; there is no byte 0). The **16 bytes** of the section header are as follows:

1 **10 bytes** - section name, such as "DATA   " (six trailing spaces)

11 **1 byte** - reserved

12 **1 bytes** - module ID (31 for HP 1652B/53B)

13 **4 bytes** - length (14506 for the logic analyzer only and 26794 for both the logic analyzer and oscilloscope).

Section Data

For the SYSTem:DATA command, the <section data> parameter consists of two parts: the data preamble and the acquisition data. These are described in the following two sections.

Data Preamble Description

The block data is organized as 160 bytes of preamble information, followed by 1024 14-byte groups of information, followed by 10 reserved bytes. The preamble gives information for each analyzer describing the amount and type of data captured, where the trace point occurred in the data, which pods are assigned to which analyzer, and other information.

Each 14-byte group is made up of two bytes (16 bits) of status for Analyzer 1, two bytes of status for Analyzer 2, then five sets of two bytes of information for each of the five 16-bit pods of the HP 1652B. In the HP 1653B, the status and format for the sets of bytes are the same, but the data in not valid on pods 3, 4, and 5.

Note

One analyzer's information is independent of the other analyzer's information. In other words, on any given line, one analyzer may contain data information for a timing machine, while the other analyzer may contain count information for a state machine with time tags enabled. The status bytes for each analyzer describe what the information for that line contains. Therefore, when describing the different formats that data may contain below, keep in mind that this format pertains only to those pods that are assigned to the analyzer of the specified type. The other analyzer's data is TOTALLY independent and conforms to its own format.

The preamble (bytes 17 through 176) consists of the following **160 bytes**:

17 **2 bytes** - Instrument ID (always 1652 for HP 1652B and HP 1653B)

19 **2 bytes** - Revision Code

Note

The values stored in the preamble represent the captured data currently stored in this structure and not what the current configuration of the analyzer is. For example, the mode of the data (bytes 21 and 99) may be STATE with tagging, while the current setup of the analyzer is TIMING.

The next **78 bytes** are for Analyzer 1 Data Information.

21 **1 byte** - Machine data mode, one of the following values:
0 = off
1 = state data (with either time or state tags)
2 = state data (without tags)
3 = glitch timing data
4 = transitional timing data

22 **1 byte** - List of pods in this analyzer, where a 1 indicates that the corresponding pod is assigned to this analyzer.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1
unused	unused	Pod 1	Pod 2	Pod 3	Pod 4	Pod 5	unused

23 **1 byte** - Master chip in this analyzer - When several chips are grouped together in a single analyzer, one chip is designated as a master chip. This byte identifies the master chip. A value of 4 represents POD 1, 3 for POD 2, 2 for POD 3, 1 for POD 4, and 0 for POD 5.

24 **1 byte** - Reserved

25 **10 bytes** - Number of rows of valid data for this analyzer - Indicates the number of rows of valid data for each of the five pods. Two bytes are used to store each pod value, with the first 2 bytes used to hold POD 5 value, the next 2 for POD 4 value, and so on.

35 **1 byte** - Trace point seen in this analyzer - Was a trace point seen (value = 1) or forced (value = 0)

36 **1 byte** - Reserved

37 **10 bytes** - Trace point location for this analyzer - Indicates the row number in which the trace point was found for each of the five pods. Two bytes are used to store each pod value, with the first 2 bytes used to hold POD 5 value, the next 2 for POD 4 value, and so on.

47 **4 bytes** - Time from arm to trigger for this analyzer - The number of 40 ns ticks that have taken place from the arm of this machine to the trigger of this machine. A value of -1 (all 32 bits set to 1) indicates counter overflow.

51 **1 byte** - Armer of this analyzer - Indicates what armed this analyzer (1 = RUN, 2 = BNC, 3 = other analyzer, 4 = SCOPE)

52 **1 byte** - Devices armed by this analyzer - Bitmap of devices armed by this machine

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1
unused	unused	unused	unused	SCOPE	BNC out	Mach. 2	Mach. 1

A 1 in a given bit position implies that this analyzer arms that device, while a 0 means the device is not armed by this analyzer.

53 **4 bytes** - Sample period for this analyzer (timing only) - Sample period at which data was acquired. Value represents the number of nanoseconds between samples.

57 **4 bytes** - Delay for this analyzer (timing only) - Delay at which data was acquired. Value represents the amount of delay in nanoseconds.

61 **1 byte** - Time tags on (state with tagging only) - In state tagging mode, was the data captured with time tags (value = 1) or state tags (value = 0).

62 **1 byte** - Reserved

63 **5 bytes** - Demultiplexing (state only) - For each of the five pods (first byte is POD 5, fifth byte is POD 1) in a state machine, describes multiplexing of each of the five pods. (0 = NO DEMUX, 1 = TRUE DEMUX, 2 = MIXED CLOCKS).

68 **1 byte** - Reserved

69 **20 bytes** - Trace point adjustment for pods - Each pod uses 4 bytes to show the number of nanoseconds that are to be subtracted from the trace point described above to get the actual trace point value. The first 4 bytes are for Pod 5, the next four are for Pod 4, and so on.

89 **10 bytes** - Reserved

The next **78 bytes** are for Analyzer 2 Data Information. They are organized in the same manner as Analyzer 1 above, but they occupy bytes 99 through 176

Acquisition Data Description

The acquisition data section consists of 14336 bytes (1024 14-byte groups), appearing in bytes 177 through 14512. The last ten bytes (14513 through 14522) are reserved. The data contained in the data section will appear in one of four forms depending on the mode in which it was acquired (as indicated in byte 21 for machine 1 and byte 99 for machine 2). The four modes are:

- State Data (without tags)
- State Data (with either time or state tags)
- Glitch Timing Data
- Transitional Timing Data

The following four sections describe the four data modes that may be encountered. Each section describes the Status bytes (shown under the Machine 1 and Machine 2 headings), and the Information bytes (shown under the Pod 5 through Pod 1 headings).

State Data (without tags)

Status Bytes. In normal state mode, only the least significant bit (bit 1) is used. When bit 1 is set, this means that there has been a sequence level transition.

Information Bytes. In state acquisition with no tags, data is obtained from the target system with each clock and checked with the trace specification. If the state matches this specification, the data is stored, and is placed into the memory.

	Machine 1	Machine 2	Pod 5	Pod 4	Pod 3	Pod 2	Pod 1*
177	Status	Status	Data	Data	Data	Data	Data
191	Status	Status	Data	Data	Data	Data	Data
205	Status	Status	Data	Data	Data	Data	Data
⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮
14499	Status	Status	Data	Data	Data	Data	Data

*The headings are not a part of the returned data.

State Data (with either time or state tags)

Status Bytes. In state tagging mode, the tags indicate whether a given row of the data is a data line, a count (tag) line, or a prestore line.

Bit 2 is the Data vs. Count bit. Bit 3 is the Prestore vs. Tag bit. The two bits together show what the corresponding Information bytes represent.

Bit 3	Bit 2	Information byte represents:
0	0	Acquisition Data
0	1	Count
1	0	Prestore Data
1	1	Invalid

If Bit 2 is clear, the information contains either actual acquisition data as obtained from the target system (if Bit 3 is clear), or prestore data (if Bit 3 is set). If Bit 2 is set and Bit 3 is clear, this row's bytes for the pods assigned to this machine contain tags. If Bit 2 and Bit 3 are set, the corresponding Information bytes are invalid and should be ignored. Bit 1 is used only when Bit 2 is clear. Whenever there has been a sequence level transition Bit 1 will be set, and otherwise will be clear.

Information Bytes. In the State acquisition mode with tags, data is obtained from the target system with each clock and checked with the trace specification. If the state does not match the trace specification, it is checked against the prestore qualifier. If it matches the prestore qualifier, then it is placed in the prestore buffer. If the state does not match either the sequencer qualifier or the prestore qualifier, it is discarded.

The type of information in the bytes labeled Data depends on the Prestore vs. Tags bit. When the Data bytes are used for prestore information, the following Count bytes (in the same column) should be ignored. When the Data bytes are used for tags, the Count bytes are formatted as floating-point numbers in the following fashion:

bits 16 through 12	bits 11 through 1
EEEEE	MMMMMMMMMMM

The five most-significant bits (EEEEE) store the exponent, and the eleven least-significant bits (MMMMMMMMMMM) store the mantissa. The actual value for Count is given by the equation:

$$\text{Count} = (2048 + \text{mantissa}) \times 2^{\text{exponent}} - 2048$$

Since the counts are relative counts from one state to the one previous, the count for the first state in the data structure is invalid.

If time tagging is on, the count value represents the number of 40 nanosecond ticks that have elapsed between the two stored states. In the case of state tagging, the count represents the number of qualified states that were encountered between the stored states.

If a state matches the sequencer qualifiers, the prestore buffer is checked. If there are any states in the prestore buffer at this time, these prestore states are first placed in memory, along with a dummy count row. After this check, the qualified state is placed in memory, followed by the count row which specified how many states (or 40 ns ticks) have elapsed since the last stored state. If this is the first stored state in memory, then the count information that is stored should be discarded.

	Machine 1	Machine 2	Pod 5	Pod 4	Pod 3	Pod 2	Pod 1*
177	Status	Status	Data	Data	Data	Data	Data
191	Status	Status	⊗	⊗	⊗	⊗	⊗
205	Status	Status	Data	Data	Data	Data	Data
219	Status	Status	Count	Count	Count	Count	Count
⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮
14485	Status	Status	Data	Data	Data	Data	Data
14499	Status	Status	Count	Count	Count	Count	Count

*The headings are not a part of the returned data.

⊗ = Invalid data

Glitch Timing Data

Status Bytes. In glitch timing mode, the status bytes indicate whether a given row in the data contains actual acquisition data information or glitch information.

Bit 1 is the Data vs. Glitch bit. If Bit 1 is set, this row of information contains glitch information. If Bit 1 is clear, then this row contains actual acquisition data as obtained from the target system.

Information Bytes. In the Glitch timing mode, the target system is sampled at every sample period. The data is then stored in memory and the glitch detectors are checked. If a glitch has been detected between the previous sample and the current sample, the corresponding glitch bits are set. The glitch information is then stored. If this is the first stored sample in memory, then the glitch information stored should be discarded.

	Machine 1	Machine 2	Pod 5	Pod 4	Pod 3	Pod 2	Pod 1*
177	Status	Status	Data	Data	Data	Data	Data
191	Status	Status	⊗	⊗	⊗	⊗	⊗
205	Status	Status	Data	Data	Data	Data	Data
219	Status	Status	Glitch	Glitch	Glitch	Glitch	Glitch
:	:	:	:	:	:	:	:
14485	Status	Status	Data	Data	Data	Data	Data
14499	Status	Status	Glitch	Glitch	Glitch	Glitch	Glitch

*The headings are not a part of the returned data.

⊗ = Invalid data

Transitional Timing Data

Status Bytes. In transitional timing mode, the status bytes indicate whether a given row in the data contains acquisition information or transition count information.

bits 10-9	bits 8-7	bits 6-5	bits 4-3	bits 2-1
Pod 5	Pod 4	Pod 3	Pod 2	Pod 1

Each pod uses two bits to show what is being represented in the corresponding Information bytes. Bits 10, 8, 6, 4 and 2 are set when the appropiate pod's Information bytes represent acquisition data. When that bit is clear, the next bit shows if the Information bytes represent the first word of a count. Together there are three possible combinations:

10 - This pod's Information bytes contain acquisition data as obtained from the target system.
01 - This pod's Information bytes contain the first word of a count.
00 - This pod's Information bytes contain part of a count other than the first word.

Information Bytes. In the Transitional timing mode the logic analyzer performs the following steps to obtain the information bytes:

1. Four samples of data are taken at 10 nanosecond intervals. The data is stored and the value of the last sample is retained.

2. Four more samples of data are taken. If any of these four samples differ from the last sample of the step 1, then these four samples are stored and the last value is once again retained.

3. If all four samples of step 2 are the same as the last sample taken in step 1, then no data is stored. Instead, a counter is incremented. This process will continue until a group of four samples is found which differs from the retained sample. At this time, the count will be stored in the memory, the counters reset, the current data stored, and the last sample of the four once again retained for comparison.

Note The stored count indicates the number of 40 ns intervals that have elapsed between the old data and the new data.

The rows of the acquisition data may, therefore, be either four rows of data followed by four more rows of data, or four rows of data followed by four rows of count. Rows of count will always be followed by four rows of data except for the last row, which may be either data or count.

Note This process is performed on a pod-by-pod basis. The individual status bits will indicate what each pod is doing.

The following table is just an example. The meaning of the Information bytes (Data or Count) depends upon the corresponding Status bytes.

Example:	Machine 1	Machine 2	Pod 5	Pod 4	Pod 3	Pod 2	Pod 1*
177	Status	Status	Data	Data	Data	Data	Data
191	Status	Status	Data	Data	Data	Data	Data
205	Status	Status	Data	Data	Data	Data	Data
219	Status	Status	Data	Data	Data	Data	Data
233	Status	Status	Data	Count	Count	Data	Data
247	Status	Status	Data	Count	Count	Data	Data
261	Status	Status	Data	Count	Count	Data	Data
275	Status	Status	Data	Count	Count	Data	Data
289	Status	Status	Count	Data	Data	Count	Data
303	Status	Status	Count	Data	Data	Count	Data
317	Status	Status	Count	Data	Data	Count	Data
331	Status	Status	Count	Data	Data	Count	Data
345	Status	Status	Data	Data	Count	Data	Data
359	Status	Status	Data	Data	Count	Data	Data
373	Status	Status	Data	Data	Count	Data	Data
387	Status	Status	Data	Data	Count	Data	Data
⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮
14457	Status	Status	Data	Data	Data	Data	Data
14471	Status	Status	Data	Data	Data	Data	Data
14485	Status	Status	Data	Data	Data	Data	Data
14499	Status	Status	Data	Data	Data	Data	Data

*The headings are not a part of the returned data.

Oscilloscope Block Data

The oscilloscope block data is described in the following sections. This data is appended to the logic analyzer block data and is present only when the oscilloscope is on and waveform data has been acquired and stored.

The oscilloscope data contains both a section header and section data similar to the logic analyzer for both of its sections. The oscilloscope block data sections are Oscilloscope Data and Oscilloscope Display Data.

- Oscilloscope Data - the raw data captured on the last acquisition.
- Oscilloscope Display Data - the segment of data displayed after each acquisition.

The oscilloscope data and oscilloscope display data sections are sent only when the oscilloscope is on and there is waveform data stored in the oscilloscope memory .

Oscilloscope Data Section

The Oscilloscope Data section contains the raw data the oscilloscope acquired on the last acquisition.

Section Header Description

The oscilloscope data <section header> used bytes 14523 through 14539. The **16 bytes** of the section header are as follows:

14523 **10 bytes** - Section name, "SCOPEDAT " (two trailing spaces)

14533 **1 byte** - Reserved (always 0)

14534 **1 byte** - Unused

14535 **4 bytes** - Length of oscilloscope data

Section Data

The oscilloscope raw data <section data> contains the initially acquired data. Each data unit is contained in a byte. The lower six bits contain the data, while the upper two bits are not used and as a result, each data unit can represent a value from 0 to 63. The total number of bytes is this section is 4096 with the first 2048 bytes for channel 1 and the remaining 2048 bytes for channel 2.

14539 **2048 bytes** - raw oscilloscope data for channel 1.

16587 **2048 bytes** - raw oscilloscope data for channel 1.

Oscilloscope Display Data Section

The display data section <section data> contains the initial data displayed after an acquisition. Each data unit is represented by a 16 bit value which is generated by taking the raw oscilloscope data and shifting it the the left by 8 bits.

Note

Changing the seconds-per-division after the oscilloscope has stopped will change the data displayed on the screen but it will not change the display data in this section.

18635 **4096 bytes** - Displayed oscilloscope data for channel 1

22731 **4096 bytes** - Displayed oscilloscope data for channel 2

DSP (Display) command

The DSP command writes the specified quoted string to a device dependent portion of the instrument display.

Command Syntax: :SYSTem:DSP <string>

where:

<string> ::= string of up to 60 alphanumeric characters

Examples:

```
OUTPUT XXX;":SYSTEM:DSP 'The message goes here'"
```

ERRor

query

The ERRor query returns the oldest error number from the error queue. A complete list of error numbers for the HP 1652B/53B is shown in appendix C, "Error Messages." If no errors are present in the error queue, a zero is returned.

Query Syntax: :SYSTem:ERRor?

Returned Format: [:SYSTem:ERRor] <error number> <NL>

Example:

```
10 OUTPUT XXX;":SYSTEM:ERROR?"
20 ENTER XXX;Err_num
30 PRINT Err_num
40 END
```

HEADer

command/query

The HEADER command tells the instrument whether or not to output a header for query responses. When HEADer is set to ON, query responses will include the command header.

The HEADer query returns the current state of the HEADer command.

Command Syntax: :SYSTem:HEADer {{ON|1}|{OFF|0}}

Example: OUTPUT XXX;":SYSTEM:HEADER ON"

Query Command: :SYSTem:HEADer?

Returned Format: [:SYSTem:HEADer] {1|0}<NL>

Example:

```
10 DIM Mode$[100]
20 OUTPUT XXX;":SYSTEM:HEADER?"
30 ENTER XXX;Mode$
40 PRINT Mode$
50 END
```

Note Headers should be turned off when returning values to numeric variables.

KEY

command/query

The KEY command allows you to simulate pressing a specified front-panel key. Key commands may be sent over the bus in any order that is legal from the front panel. Be sure the instrument is in a desired setup before executing the KEY command. Key codes range from 0 to 36 with 99 representing no key (returned at power-up). See table 6-1 for key codes.

Note The external KEY buffer is only two keys deep; therefore, attempting to send KEY commands too rapidly will cause a KEY buffer overflow error to be displayed on the HP 1652B/53B screen.

The KEY query returns the key code for the last front- panel key pressed or the last simulated key press over the bus.

Command Syntax: :SYSTem:KEY <key_code>

where:

<key_code> ::= integer from 0 to 36

Example:

```
OUTPUT XXX;":SYSTEM:KEY 24"
```

Query Syntax: :SYSTem:KEY?

Returned Format: [:SYSTem:KEY] <key_code> <NL>

Example:

```
10 DIM Key$[100]
20 OUTPUT XXX;":SYSTEM:KEY?"
30 ENTER XXX; KEY$
40 PRINT KEY$
50 END
```

Table 6-1. Key codes

Key Value	HP 1652B/53B Key	Key Value	HP1652B/53B Key
0	RUN	19	D
1	STOP	20	E
2	unused	21	F
3	SELECT	22	unused
4	CHS	23	unused
5	Don't Care	24	Knob left
6	0	25	Knob right
7	1	26	L/R Roll
8	2	27	U/D Roll
9	3	28	unused
10	4	29	unused
11	5	30	unused
12	6	31	"."
13	7	32	Clear Entry
14	8	33	FORMAT/CHAN
15	9	34	TRACE/TRIG
16	A	35	DISPLAY
17	B	36	I/O
18	C	99	Power Up

LER (LCL Event Register) query

The LER query allows the LCL (local) Event Register to be read. After the LCL Event Register is read, it is cleared. A one indicates a remote-to-local transition has taken place. A zero indicates a remote-to-local transition has not taken place.

Query Syntax: :SYSTem:LER?

Returned Format: [:SYSTem:LER] {0|1}<NL>

Example:

```
10 DIM Event$[100]
20 OUTPUT XXX;":SYSTEM:LER?"
30 ENTER XXX;Event$
40 PRINT Event$
50 END
```

LOCKout

command/query

The LOCKout command locks out or restores front-panel operation. When this function is on, all controls (except the power switch) are entirely locked out.

The LOCKout query returns the current status of the LOCKout command.

Command Syntax: :SYSTem:LOCKout {{ON|1}|{OFF|0}}

Example:
```
OUTPUT XXX;":SYSTEM:LOCKOUT ON"
```

Query Syntax: :SYSTem:LOCKout?

Returned Format: [:SYSTem:LOCKout] {0|1}<NL>

Example:
```
10 DIM Status$[100]
20 OUTPUT XXX;":SYSTEM:LOCKOUT?"
30 ENTER XXX;Status$
40 PRINT Status$
50 END
```

LONGform

command/query

The LONGform command sets the longform variable which tells the instrument how to format query responses. If the LONGform command is set to OFF, command headers and alpha arguments are sent from the instrument in the abbreviated form. If the LONGform command is set to ON, the whole word will be sent to the controller.

This command has no affect on the input data messages to the instrument. Headers and arguments may be input in either the longform or shortform regardless of how the LONGform command is set.

The query returns the status of the LONGform command.

Command Syntax: :SYSTem:LONGform {{ON|1}|{OFF|0}}

Example:
```
OUTPUT XXX;":SYSTEM:LONGFORM ON"
```

Query Syntax: :SYSTem:LONGform?

Returned Format: [:SYSTem:LONGform] {1|0}<NL>

Example:
```
10 DIM Mode$[100]
20 OUTPUT XXX;":SYSTEM:LONGFORM?"
30 ENTER XXX;Mode$
40 PRINT Mode$
50 END
```

MENU

command/query

The MENU command puts a menu on the display.

The MENU query returns the current menu selection.

Command Syntax: :SYSTem:MENU <menu_type>,<mach_num>

where:

<menu_type> ::= {SCONfig | FORMat | CHANnel | TRACe | TRIGger | DISPlay | WAVeform | SWAVeform | COMPare | SCHart | SLISt}

<mach_num> ::= {0 | 1 | 2 | 3}

0 ::= mixed mode

1 ::= analyzer 1

2 ::= analyzer 2

3 ::= oscilloscope

Example:

```
OUTPUT XXX;"SYSTEM:MENU FORMAT,1"
```

Query Syntax: :SYSTem:MENU?

Returned Format: [:SYSTem:MENU] <menu_type>,<mach_num>

Example:

```
10 DIM Response$[100]
20 OUTPUT XXX;":SYSTEM:MENU?"
30 ENTER XXX;Response$
40 PRINT Response$
50 END
```

MESE

command/query

The MESE command sets the Module Event Status Enable Register bits. The MESE register contains a mask value for the bits enabled in the MESR register. A one in the MESE will enable the corresponding bit in the MESR, a zero will disable the bit.

The MESE query returns the current setting.

Refer to table 6-2 for information about the Module Event Status Enable register bits, bit weights, and what each bit masks for the logic analyzer.

Command Syntax: :SYSTem:MESE <enable_mask>

where:

<enable mask> ::= integer from 0 to 255

Example:

```
OUTPUT XXX;":SYSTEM:MESE 1"
```

MESE

Query Syntax: :SYSTem:MESE?

Returned Format: [:SYSTem:MESE] <enable_mask> <NL>

Example:

```
10 OUTPUT XXX;":SYSTEM:MESE?"
20 ENTER XXX; Mes
30 PRINT Mes
40 END
```

Table 6-2. Module Event Status Enable Register

Module Event Status Enable Register (A "1" enables the MESR bit)		
Bit	Weight	Enables
7	128	Not used
6	64	Not used
5	32	Not used
4	16	Not used
3	8	Not used
2	4	Not used
1	2	RNT - Run until satisified
0	1	MC - Measurement complete

MESR

query

The MESR query returns the contents of the Module Event Status register.

Note Reading the register clears the Module Event Status Register.

Table 6-3 shows each bit in Module Event Status Register and their bit weights for the logic analyzer. When you read the MESR, the value returned is the total bit weights of all bits that are set at the time the register is read.

Query Syntax: :SYSTem:MESR?

Returned Format: [:SYSTem:MESR] <status> <NL>

where:

<status> ::= integer from 0 to 255

Example:

```
10 OUTPUT XXX;":SYSTem:MESR?"
20 ENTER XXX; Mer
30 PRINT Mer
40 END
```

Table 6-3. Module Event Status Register

Module Event Status Register		
Bit	Weight	Condition
7	128	Not used
6	64	Not used
5	32	Not used
4	16	Not used
3	8	Not used
2	4	Not used
1	2	1 = Run until satisified 0 = Run until not satisified
0	1	1 = Measurement complete 0 = Measurement not complete

PPOWer

query

The PPOWer (preprocessor power) query returns the current status of the HP 1652B/53B's high-current limit circuit. If it is functioning properly, 1 is returned. If the current draw is too high, 0 is returned until the problem is corrected and the circuit automatically resets.

Query Syntax: :PPOWer?

Returned Format: [:PPOWer] {0 | 1}

Example:

```
10 DIM Response$[10]
20 OUTPUT XXX;":PPOWER?"
30 ENTER XXX; Response$
40 PRINT Response$
50 END
```

PRINt

command

The PRINt command initiates a print of the screen or print all over the RS-232C bus. The PRINt parameters SCReen or ALL specify how the screen data is sent to the controller. PRINt SCReen transfers the data to the controller in a printer specific graphics format. PRINt ALL transfers the data in a raster format for the following menus:

- State and Timing Format menus
- Disk menu
- State and Timing Symbol menus
- State Listing menu
- State Trace
- State Compare

Command Syntax: :SYSTem:PRINt {SCReen|ALL}

Example: `OUTPUT XXX;":SYSTEM:PRINT SCREEN"`

RMODe

command/query

The RMODe command is a run control command that specifies the run mode for logic analyzer and oscilloscope. It is at the same level in the command tree as SYSTem; therefore, it is not preceded by :SYSTem.

The query returns the current setting.

Note After specifying the run mode, use the STARt command to start the acquisition.

Command Syntax: :RMODe {SINGle|REPetitive}

Example: `OUTPUT XXX;":RMODE SINGLE"`

Query Syntax: :RMODe?

Returned Format: [:RMODe] {SINGle|REPetitive}<NL>

Example:

```
10 DIM Mode$[100]
20 OUTPUT XXX;":RMODE?"
30 ENTER XXX;Mode$
40 PRINT Mode$
50 END
```

SETup

command/query

The SYStem:SETup command configures the logic analyzer module as defined by the block data sent by the controller.

The SYStem:SETup query returns a block of data that contains the current configuration to the controller.

There are three data sections which are always returned and a fourth header when the oscilloscope is on and has acquired and stored waveform data. These are the strings which would be included in the section header:

- `"CONFIG    "`
- `"1650 RS232"`
- `"1650 DISP "`
- `"1650 DISP2"`
- `"SCOPECNF  "`

Additionally, the following sections may also be included, depending on what's loaded:

- `"SYMBOLS A "`
- `"SYMBOLS B "`
- `"SPA DATA A"`
- `"SPA DATA B"`
- `"INVASM A  "`
- `"INVASM B  "`
- `"COMPARE   "`

Command syntax: :SYStem:SETup <block data>

where:

<block data>	::= <block length specifier> <section>...
<block length specifier>	::= #8<length>
<length>	::= the total length of all sections in byte format (must be represented with 8 digits)
<section>	::= <section header> <section data>
<section header>	::= 16 bytes in the following format: 10 bytes for the section name 1 byte reserved 1 byte for the module ID code (31 for the logic analyzer) 4 bytes for the length of the section data in bytes
<section data>	::= format depends on the type of data

Note

The total length of a section is 16 (for the section header) plus the length of the section data. So when calculating the value for <length>, don't forget to include the length of the section headers.

Example:

```
OUTPUT XXX USING "#,K";":SYSTEM:SETUP  "    <block data>
```

Query Syntax: :SYStem:SETup?

Returned Format: [:SYStem:SETup] <block data><NL>

HP-IB Example:

```
10 DIM Block$[32000]              !allocate enough mcmory for block data
20 DIM Specifier$[2]
30 OUTPUT XXX;":SYSTEM:HEAD OFF"
40 OUTPUT XXX;":SYSTEM:SETUP?"    ! send setup query
50 ENTER XXX USING "#,2A";Specifier$! read in #8
60 ENTER XXX USING "#,8D";Blocklength! read in block length
70 ENTER XXX USING "-K";Block$    ! read in data
80 END
```

STARt command

The STARt command is a run control command that starts the logic analyzer running in the specified run mode (see RMODe). The STARt command is on the same level in the command tree as SYSTem; therefore, it is not preceded by :SYSTem.

Note The STARt command is an Overlapped Command. An Overlapped Command is a command that allows execution of subsequent commands while the device operations initiated by the Overlapped Command are still in progress.

Command Syntax: :STARt

Example: `OUTPUT XXX;":START"`

STOP

command

The STOP command is a run control command that stops the logic analyzer. The STOP command is on the same level in the command tree as SYSTem; therefore, it is not preceded by :SYSTem.

Note The STOP command is an Overlapped Command. An Overlapped Command is a command that allows execution of subsequent commands while the device operations initiated by the Overlapped Command are still in progress.

Command Syntax: :STOP

Example: OUTPUT XXX;":STOP"

7 - MMEMory Subsystem

MMEMory Subsystem 7

Introduction

MMEMory subsystem commands provide access to the disk drive. The MMEMory subsystem commands are:

- AUToload
- CATalog
- COPY
- DOWNload
- INITialize
- LOAD
- PACK
- PURGe
- REName
- STORe
- UPLoad

Note

If you are not going to store information to the configuration disk, or if the disk you are using contains information you need, it is advisable to write protect your disk. This will protect the contents of the disk from accidental damage due to incorrect commands, etc.

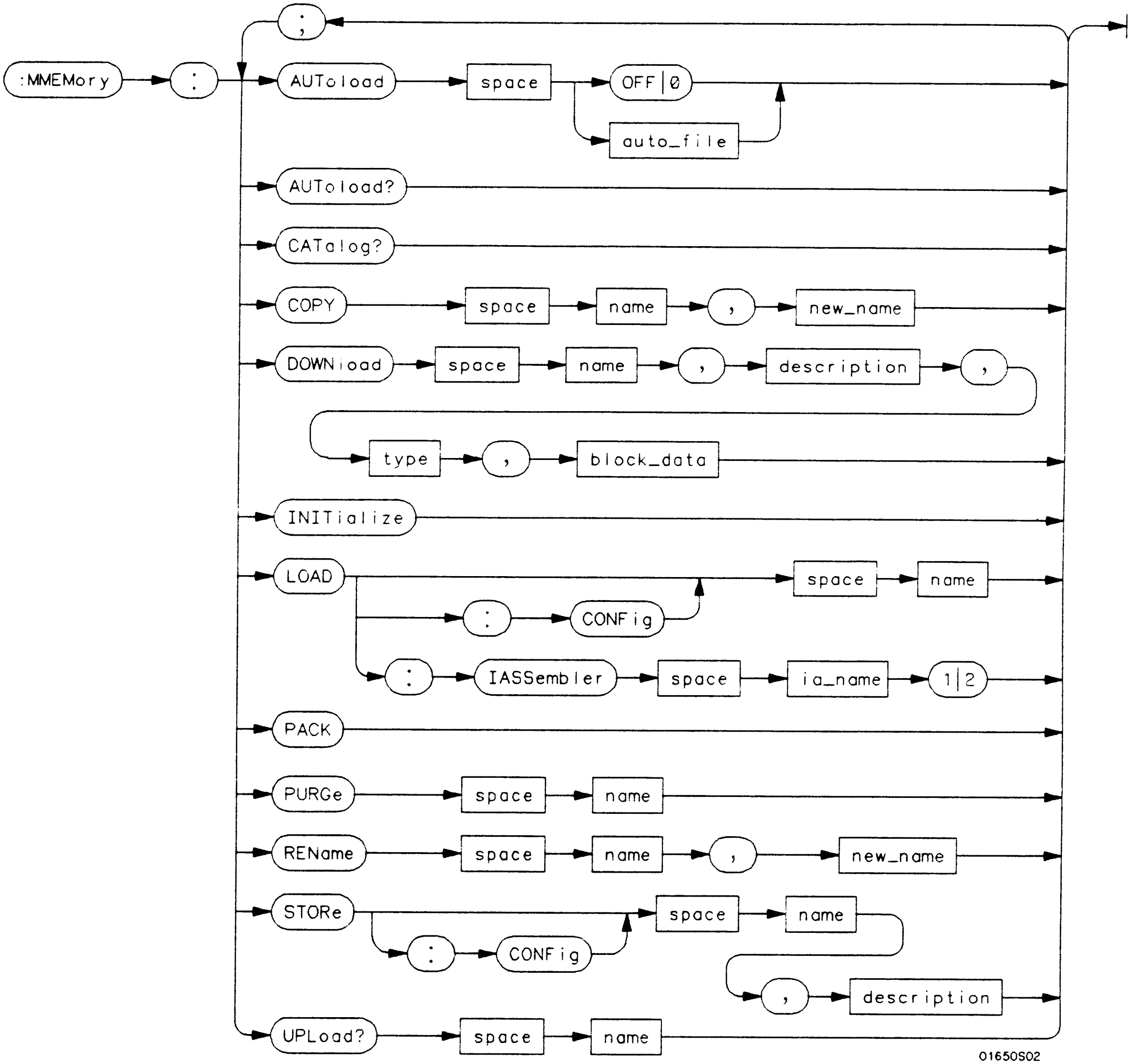


Figure 7-1. MMEMory Subsystem Commands Syntax Diagram

auto_file = *string of up to 10 alphanumeric characters representing a valid file name.*
name = *string of up to 10 alphanumeric characters representing a valid file name.*
description = *string of up to 32 alphanumeric characters.*
type = *integer, refer to table 7-1.*
block_data = *data in IEEE 488.2 # format.*
ia_name = *string of up to 10 alphanumeric characters representing a valid file name.*
new_name = *string of up to 10 alphanumeric characters representing a valid file name*

Figure 7-1. MMEMory Subsystem Commands Syntax Diagram (continued)

Note Refer to "Disk Operations" in chapter 5 of the *HP 1652B/53B Logic Analyzers Reference* manual for a description of a valid file name.

AUToload

command/query

The AUToload command controls the autoload feature which designates a configuration file to be loaded automatically the next time the instrument is turned on. The OFF parameter (or 0) disables the autoload feature. When a string parameter is specified it represents the desired autoload file.

The AUToload query returns 0 if the autoload feature is disabled. If the autoload feature is enabled, the query returns a string parameter that specifies the current autoload file.

Command Syntax: :MMEMory:AUToload {{OFF|0}|<auto_file>}

where:

<auto_file> ::= string of up to 10 alphanumeric characters

Examples:

```
OUTPUT XXX;":MMEMORY:AUTOLOAD OFF"
OUTPUT XXX;":MMEMORY:AUTOLOAD 'FILE1'"
OUTPUT XXX;":MMEMORY:AUTOLOAD 'FILE2'"
```

Query Command: :MMEMory:AUToload?

Returned Format: [:MMEMory:AUToload] {0|<auto_file>}<NL>

Example:

```
10 DIM Auto_status$[100]
20 OUTPUT XXX;":MMEMORY:AUTOLOAD?"
30 ENTER XXX;Auto_status$
40 PRINT Auto_status$
50 END
```

CATalog

query

The CATalog query returns the directory of the disk in block data format. The directory consists of a 51-character string for each file on the disk. Each file entry is formatted as follows:

"NNNNNNNNNN TTTTTTT DDDDDDDDDDDDDDDDDDDDDDDDDDDDDDDD"

where N is the filename, T is the file type (a number), and D is the file description.

Query Syntax: :MMEMory:CATalog?

Returned Format: [:MMEMory:CATalog] <block size> <block data>

where:

<block size> ::= #8dddddddd (#8 followed by an eight-digit number)

<block data> ::= [<filename> <file type> <file description>]...

Example:

```
10  DIM File$[51]
20  DIM Specifier$[2]
30  OUTPUT XXX;":SYSTEM:HEAD OFF"
40  OUTPUT XXX;":MMEMORY:CATALOG?"      !send catalog query
50  ENTER XXX USING "#,2A";Specifier$   !read in #8
60  ENTER XXX USING "#,8D";Length       !read in length
70  FOR I=1 TO Length STEP 51           !read and print each file
80    ENTER XXX USING "#,51A";File$
90    PRINT File$
100   NEXT I
110 ENTER XXX USING "A";Specifier$      !read in final line feed
120 END
```

COPY

command

The COPY command copies the contents of a file to a new file. The two <name> parameters are the filenames. The first parameter specifies the source file. The second specifies the destination file. An error is generated if the source file doesn't exist, if the destination file already exists, or any other disc error is detected.

Command Syntax: :MMEMory:COPY <name>,<name>

where:

<name> ::= string of up to 10 alphanumeric characters representing a valid file name

Example: To copy the contents of "FILE1" to "FILE2":

```
OUTPUT XXX;":MMEMORY:COPY 'FILE1','FILE2'"
```

DOWNload command

The DOWNload command downloads a file to the disk. The <name> parameter specifies the filename, the <description> parameter specifies the file description, and the <block_data> contains the contents of the file to be downloaded.

Table 7-1 lists the file types for the <type> parameter.

Command Syntax: :MMEMory:DOWNload <name>,<description>,<type>,<block_data>

where:

<name> ::= string of up to 10 alphanumeric characters representing a valid file name
<description> ::= string of up to 32 alphanumeric characters
<type> ::= integer (see Table 7-1)
<block_data> ::= contents of file in block data format

Example:

```
OUTPUT XXX;":MMEMORY:DOWNLOAD 'SETUP__';'FILE CREATED FROM SETUP
QUERY',-16127,#800000643..."
```

Table 7-1. File Types

File	File Type
HP 1652/3 SYSTEM	-16383
1652/3 CONFIG	-16096
AUTOLOAD TYPE	-15615
INVERSE ASSEMBLER	-15614
TEXT TYPE	-15610

INITialize

command

The INITialize command formats the disk.

Note Once executed, the initialize command formats the specified disk, permanently erasing all existing information from the disk. After that, there is no way to retrieve the original information.

Command Syntax: :MMEMory:INITialize

Example:
```
OUTPUT XXX;":MMEMORY:INITIALIZE"
```

LOAD [:CONFig] command

The LOAD command loads a file from the disk into the analyzer. The [:CONfig] specifier is optional and has no effect on the command. The <name> parameter specifies the filename that will be loaded into the logic analyzer.

Note Any previous setups and data in the instrument are replaced by the contents of the configuration file.

Command Syntax: :MMEMory:LOAD[:CONfig] <name>

where:

<name> ::= string of up to 10 alphanumeric characters representing a valid file name

Examples:

```
OUTPUT XXX;":MMEMORY:LOAD:CONFIG 'FILE__'"
OUTPUT XXX;":MMEMORY:LOAD 'FILE__'"
OUTPUT XXX;":MMEM:LOAD:CONFIG 'FILE_A'"
```

LOAD [:IASSembler] command

This variation of the LOAD command allows inverse assembler files to be loaded into analyzer 1 or analyzer 2 of the HP 1652B/1653B. The <IA_name> parameter specifies the inverse assembler filename. The parameter after the <IA_name> parameter specifies into which machine the inverse assembler is loaded.

Note Inverse assembler files should only be loaded into the state analyzer. If an inverse assembler file is loaded into the timing analyzer no error will be generated; however, it will not be accessible.

Command Syntax: :MMEMory:LOAD:IASSembler <IA_name>,{1|2}

where:

<IA_name> ::= string of up to 10 alphanumeric characters representing a valid file name

Examples:

```
OUTPUT XXX;":MMEMORY:LOAD:IASSEMBLER 'I68020_IP',1"
OUTPUT XXX;":MMEM:LOAD:IASS 'I68020_IP'1"
```

PACK

command

The PACK command packs the files on a disk in the disk drive.

Command Syntax: :MMEMory:PACK

Example:
```
OUTPUT XXX;":MMEMORY:PACK"
```

PURGe

command

The PURGe command deletes a file from the disk. The <name> parameter specifies the filename to be deleted.

Note Once executed, the purge command permanently erases all the existing information from the specified file. After that, there is no way to retrieve the original information.

Command Syntax: :MMEMory:PURGe <name>

where:

<name> ::= string of up to 10 alphanumeric characters representing a valid file name

Examples:

```
OUTPUT XXX;":MMEMORY:PURGE 'FILE1'"
```

REName

command

The REName command renames a file on the disk. The <name> parameter specifies the filename to be changed and the <new_name> parameter specifies the new filename.

Note

You cannot rename a file to an already existing filename.

Command Syntax: :MMEMory:REName <name>,<new_name>

where:

<name> ::= string of up to 10 alphanumeric characters representing a valid file name

<new_name> ::= string of up to 10 alphanumeric characters representing a valid file name

Examples:

```
OUTPUT XXX;":MMEMORY:RENAME 'OLDFILE','NEWFILE'"
```

STORe [:CONFig] command

The STORe command stores a configuration onto a disk. The [:CONFig] specifier is optional and has no effect on the command. The <name> parameter specifies the file to be stored to the disk. The <description> parameter specifies the file description.

Command Syntax: :MMEMory:STORe [:CONfig] <name>,<description>

where:

<name>	::= string of up to 10 alphanumeric characters representing a valid file name
<description>	::= string of up to 32 alphanumeric characters

Example:

```
OUTPUT XXX;":MMEM:STORE 'DEFAULTS','DEFAULT SETUPS'"
```

UPLoad

query

The UPLoad query uploads a file. The <name> parameter specifies the file to be uploaded from the disk. The contents of the file are sent out of the instrument in block data form.

Query Syntax: :MMEMory:UPLoad? <name>

where:

<name> ::= string of up to 10 alphanumeric characters representing a valid file name

Returned Format: [:MMEMory:UPLoad] <block_data><NL>

Example:

```
10 DIM Block$[32000]              !allocate enough memory for block data
20 DIM Specifier$[2]
30 OUTPUT XXX;":SYSTEM HEAD OFF"
40 OUTPUT XXX;":MMEMORY:UPLOAD? 'FILE1'"  !send upload query
50 ENTER XXX USING "#,2A";Specifier$      !read in #8
60 ENTER XXX USING "#,8D";Length          !read in block length
70 ENTER XXX USING "-K";Block$            !read in file
80 END
```

8 - DLISt Subsystem

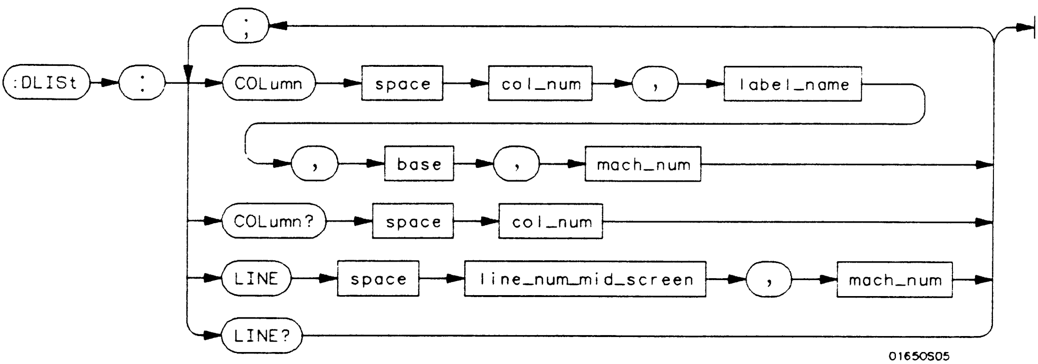
DLISt Subsystem

8

Introduction

The DLISt (dual list) subsystem contains the commands in the dual state listing menu. These commands are:

- COLumn
- LINE



col_num = *integer from 1 to 8*
label_name = *a string of up to 6 alphanumeric characters*
base = *{BINary|HEXacecimal|OCTal|DECimal|ASCii|SYMBol}*
mach_num = *{1|2}*
line_num_mid_screen = *integer from -1023 to +1023*

Figure 8-1. DLISt Subsystem Syntax Diagram

DLISt

selector

The DLISt selector (dual list) is used as part of a compound header to access those settings normally found in the Dual State Listing menu. The dual list displays data when two state analyzers are run simultaneously.

Command Syntax: :DLISt

Example: `OUTPUT XXX;":DLIST:LINE 0,1"`

COLumn

command/query

The COLumn command allows you to configure the state analyzer list display by assigning a label name and base to one of eight vertical columns in the menu. The machine number parameter is required since the same label name can occur in both state machines at once. A column number of 1 refers to the left-most column. When a label is assigned to a column it replaces the original label in that column. The label originally in the specified column is placed in the column the specified label is moved from.

When "TAGS" is the label name, the TAGS column is assumed and the next parameter must specify RELative or ABSolute. The machine number should be 1.

The COLumn query returns the column number, label name, and base for the specified column.

Command Syntax: :DLISt:COLumn <col_num>,{"TAGS",{RELative|ABSolute}|<label_name>,<base>},<mach_num>

where:

<col_num>	::= {1|2|3|4|5|6|7|8}
<label_name>	::= a string of up to 6 alphanumeric characters
<base>	::= {BINary|HEXadecimal|OCTal|DECimal|ASCii|SYMBol}
<mach_num>	::= {1|2}

Example:

```
OUTPUT XXX;":DLIST:COLUMN 4,'DATA',HEXADECIMAL,1"
```

COLumn

Query Syntax: :DLISt:COLumn? <col_num>

Returned Format: [:DLISt:COLumn] <col_num>,<label_name>,<base>,<mach_num><NL>

Example:

```
10 DIM C1$[100]
20 OUTPUT XXX;":DLIST:COLUMN? 4"
30 ENTER XXX;C1$
40 PRINT C1$
50 END
```

LINE

command/query

The LINE command allows you to scroll the state analyzer listing vertically. The command specifies the state line number relative to the trigger that the specified analyzer will highlight at center screen.

The LINE query returns the line number for the state currently in the box at center screen and the machine number to which it belongs.

Command Syntax: :DLISt:LINE <line_num_mid_screen>,<mach_num>

where:

<line_num_mid_screen> ::= integer from -1023 to +1023
<mach_num> ::= {1|2}

Example:

```
OUTPUT XXX;":DLIST:LINE 511,1"
```

Query Syntax: :DLISt:LINE?

Returned Format: [DLISt:LINE] <line_num_mid_screen>,<mach_num><NL>

Example:

```
10 DIM Ln$[100]
20 OUTPUT XXX;":DLIST:LINE?"
30 ENTER XXX;Ln$
40 PRINT Ln$
50 END
```

9 - WLISt Subsystem

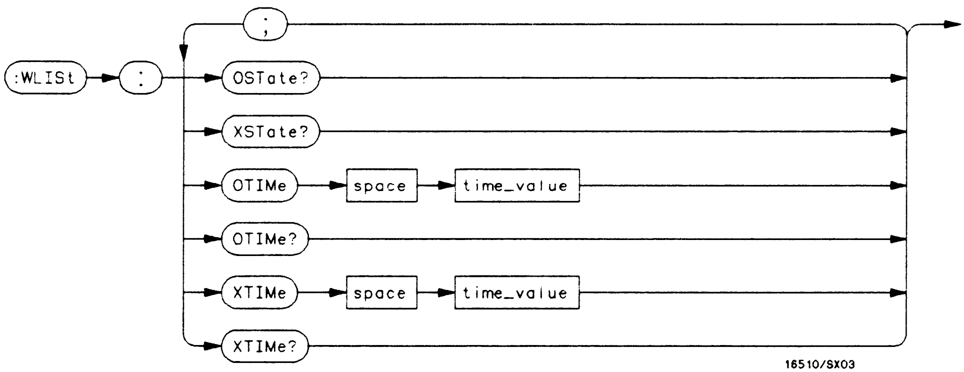
WLISt Subsystem 9

Introduction

Two commands in the WLISt subsystem control the X and O marker placement on the waveforms portion of the Timing/State mixed mode display. These commands are XTIMe and OTIMe. The XSTate and OSTate queries return what states the X and O markers are on. Since the markers can only be placed on the timing waveforms, the queries return what state (state acquisition memory location) the marked pattern is stored in.

Note

In order to have mixed mode, one machine must be a timing analyzer and the other must be a state analyzer with time tagging on (use MACHine<N>:STRace:TAG TIME).



time_value = *real number*

Figure 9-1. WLISt Subsystem Syntax Diagram

WLISt

selector

The WLISt (Waveforms/listing) selector is used as a part of a compound header to access the settings normally found in the Mixed Mode menu. Since the WLISt command is a root level command, it will always appear as the first element of a compound header.

Note The WLISt Subsystem is only available when one state analyzer (with time tagging on) and one timing analyzer are specified.

Command Syntax: :WLISt

Example: `OUTPUT XXX;":WLIST:XTIME 40.0E-6"`

OSTate

query

The OSTate query returns the state where the O Marker is positioned. If data is not valid, the query returns 32767.

Query Syntax: :WLISt:OSTate?

Returned Format: [:WLISt:OSTate] <state_num><NL>

where:

<state_num> ::= integer

Example:

```
10 DIM So$[100]
20 OUTPUT XXX;":WLIST:OSTATE?"
30 ENTER XXX;So$
40 PRINT So$
50 END
```

XSTate

query

The XSTate query returns the state where the X Marker is positioned. If data is not valid, the query returns 32767.

Query Syntax: :WLISt:XSTate?

Example:

```
OUTPUT XXX,":WLIST:XSTATE?
```

Returned Format: [:WLISt:XSTate] <state_num> <NL>

where:

<state_num> ::= integer

Example:

```
10 DIM Sx$[100]
20 OUTPUT XXX;":WLIST:XSTATE?"
30 ENTER XXX;Sx$
40 PRINT Sx$
50 END
```

OTIMe

command/query

The OTIMe command positions the O Marker on the timing waveforms in the mixed mode display. If the data is not valid, the command performs no action.

The OTIMe query returns the O Marker position in time. If data is not valid, the query returns 9.9E37.

Command Syntax: :WLISt:OTIMe <time_value>

where:

<time_value> ::= real number

Example:

```
OUTPUT XXX,":WLIST:OTIME 40.0e-6"
```

Query Syntax: :WLISt:OTIMe?

Returned Format: [:WLISt:OTIMe] <time_value><NL>

Example:

```
10 DIM To$[100]
20 OUTPUT XXX;":WLIST:OTIME?"
30 ENTER XXX;To$
40 PRINT To$
50 END
```

XTIMe

command/query

The XTIMe command positions the X Marker on the timing waveforms in the mixed mode display. If the data is not valid, the command performs no action.

The XTIMe query returns the X Marker position in time. If data is not valid, the query returns 9.9E37.

Command Syntax: :WLISt:XTIMe <time_value>

where:

<time_value> ::= real number

Example:

```
OUTPUT XXX,":WLIST:XTIME 40.0E-6"
```

Query Syntax: :WLISt:XTIMe?

Returned Format: [:WLISt:XTIMe] <time_value><NL>

Example:

```
10 DIM Tx$[100]
20 OUTPUT XXX;":WLIST:XTIME?"
30 ENTER XXX;Tx$
40 PRINT Tx$
50 END
```

10 - MACHine Subsystem

MACHine Subsystem 10

Introduction

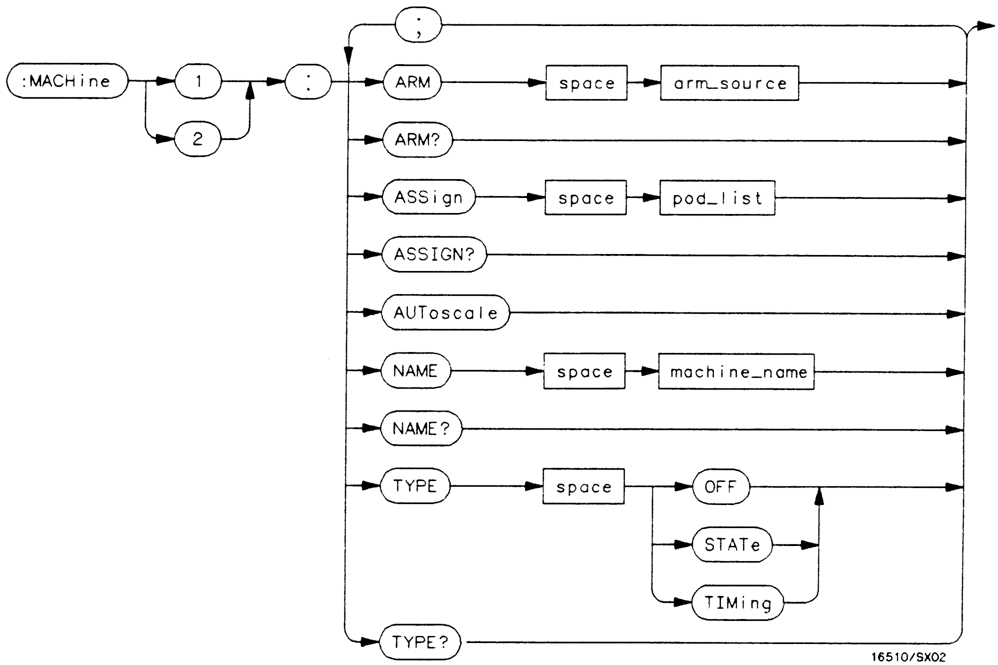
The MACHine subsystem contains the commands available for the State/Timing Configuration menu. These commands are:

- ARM
- ASSign
- AUToscale (Timing Analyzer only)
- NAME
- TYPE

There are actually two MACHine subsystems: MACHine1 and MACHine2. Unless noted, they are identical. In the syntax definitions you will see MACHine{1|2} anytime the subject is applicable to both subsystems.

Additionally, the following subsystems are a part of the MACHine subsystem. Each is explained in a separate chapter.

- SFORmat subsystem (chapter 11)
- STRace subsystem (chapter 12)
- SLISt subsystem (chapter 13)
- SWAVeform subsystem (chapter 14)
- SCHart subsystem (chapter 15)
- COMPare subsystem (chapter 16)
- TFORmat subsystem (chapter 17)
- TTRace subsystem (chapter 18)
- TWAVeform subsystem (chapter 19)
- SYMBol subsystem (chapter 20)



arm_source = *{RUN | MAChine {1 | 2}}*
pod_list = *{NONE | <pod_num>[, <pod_num>]...}*
pod_num = *{1 | 2 | 3 | 4 | 5}*
machine_name = *string of up to 10 alphanumeric characters*

Figure 10-1. Machine Subsystem Syntax Diagram

MACHine

selector

The MACHine<N> selector specifies which of the two analyzers (machines) available in the HP 1652B/53B the commands or queries following will refer to. Since the MACHine<N> command is a root level command, it will normally appear as the first element of a compound header.

Command Syntax: :MACHine<N>

where:

<N> ::= {1|2} (the number of the machine)

Example:

```
OUTPUT XXX; ":MACHINE1:NAME 'DRAMTEST'"
```

ARM

command/query

The ARM command specifies the arming source of the specified analyzer (machine).

The ARM query returns the source that the current analyzer (machine) will be armed by.

Command Syntax: :MACHine{1|2}:ARM <arm_source>

where:

<arm_source> ::= {RUN|MACHine{1|2}|BNC|SCOPe}

Example:

```
OUTPUT XXX;":MACHINE1:ARM MACHINE2"
```

Query Syntax: :MACHine {1|2}:ARM?

Returned Format: [:MACHine {1|2}:ARM] <arm_source> <NL>

Example:

```
10 DIM String$ [100]
20 OUTPUT XXX; ":MACHINE1:ARM?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

ASSign

command/query

The ASSign command assigns pods to a particular analyzer (machine).

The ASSign query returns which pods are assigned to the current analyzer (machine).

Command Syntax: :MACHine{1|2}:ASSign <pod_list>

where:

<pod_list> ::= {NONE|<pod #>[, <pod #>]...}
<pod #> ::= {1|2|3|4|5}

Example:

```
OUTPUT XXX;":MACHINE1:ASSIGN 5, 2, 1"
```

Query Syntax: :MACHine {1|2}:ASSign?

Returned Format: [:MACHINE {1|2}:ASSign] <pod_list> <NL>

Example:

```
10 DIM String$ [100]
20 OUTPUT XXX;":MACHINE1:ASSIGN?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

AUToscale

command

The AUToscale command causes the current analyzer (machine) to autoscale if the current machine is a timing analyzer. If the current machine is not a timing analyzer, the AUToscale command is ignored.

AUToscale is an Overlapped Command. Overlapped Commands allow execution of subsequent commands while the logic analyzer operations initiated by the Overlapped Command are still in progress. Command overlapping can be avoided by using the *OPC and *WAI commands in conjunction with AUToscale (see chapter 5, "Common Commands.")

Note When the AUToscale command is issued, existing timing analyzer configurations are erased and the other analyzer is turned off.

Command Syntax: :MACHine{1|2}:AUToscale

Example:
```
OUTPUT XXX;":MACHINE1:AUTOSCALE"
```

NAME

command/query

The NAME command allows you to assign a name of up to 10 characters to a particular analyzer (machine) for easier identification.

The NAME query returns the current analyzer name as an ASCII string.

Command Syntax: :MACHine{1|2}:NAME <machine_name>

where:

<machine_name> ::= string of up to 10 alphanumeric characters

Example:

```
OUTPUT XXX;":MACHINE1:NAME 'DRAMTEST'"
```

Query Syntax: :MACHine{1|2}:NAME?

Returned Format: [MACHine{1|2}:NAME] <machine name><NL>

Example:

```
10 DIM String$ [100]
20 OUTPUT XXX;":MACHINE1:NAME?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

TYPE

command/query

The TYPE command specifies what type a specified analyzer (machine) will be. The analyzer types are state or timing. The TYPE command also allows you to turn off a particular machine.

Note Only one of the two analyzers can be specified as a timing analyzer at one time.

The TYPE query returns the current analyzer type for the specified analyzer.

Command Syntax: :MACHine{1|2}:TYPE <analyzer type>

where:

<analyzer type> ::= {OFF|STATe|TIMing}

Example:

```
OUTPUT XXX;":MACHINE1:TYPE STATE"
```

Query Syntax: :MACHine{1|2}:TYPE?

Returned Format: [:MACHine{1|2}:TYPE] <analyzer type><NL>

Example:

```
10 DIM String$ [100]
20 OUTPUT XXX;":MACHINE1:TYPE?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

11 - SFORmat Subsystem

SFORmat Subsystem 11

Introduction

The SFORmat subsystem contains the commands available for the State Format menu in the HP 1652B/53B logic analyzer. These commands are:

- CLOCk
- CPERiod
- LABel
- MASTer
- REMove
- SLAVe
- THReshold

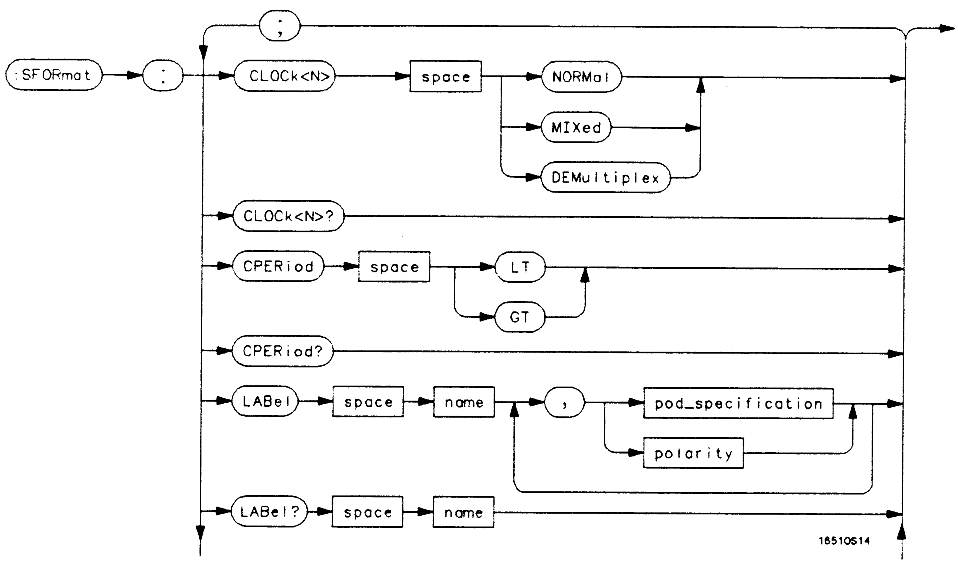
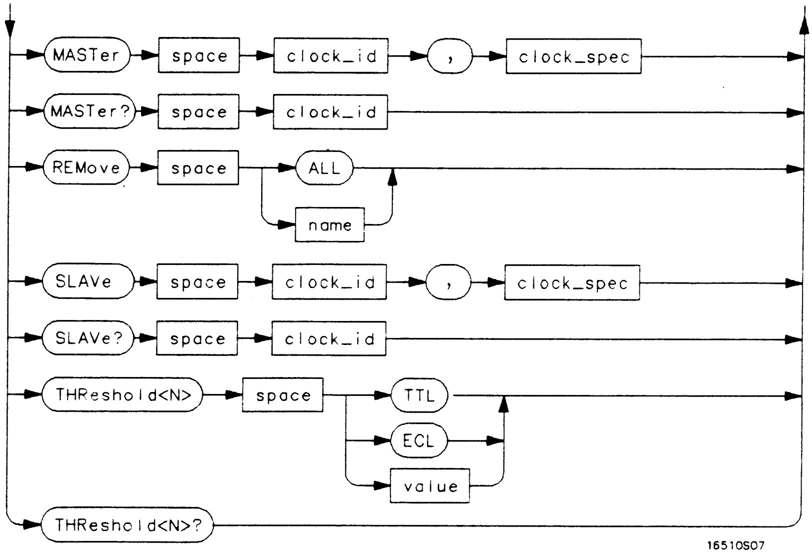


Figure 11-1. SFORmat Subsystem Syntax Diagram



<N> = *{1 | 2 | 3 | 4 | 5}*
GT = *Greater Than 60 ns*
LT = *Less Than 60 ns*
name = *string of up to 6 alphanumeric characters*
polarity = *{POSitive | NEGative}*
pod_specification = *format (integer from 0 to 65535) for a pod (pods are assigned in decreasing order)*
clock_id = *{J | K | L | M | N}*
clock_spec = *{OFF | RISing | FALLing | BOTH | LOW | HIGH}*
value = *voltage (real number) -9.9 to +9.9*

Figure 11-1. SFORmat Subsystem Syntax Diagram (continued)

SFORmat

selector

The SFORmat (State Format) selector is used as a part of a compound header to access the settings in the State Format menu. It always follows the MACHine selector because it selects a branch directly below the MACHine level in the command tree.

Command Syntax: :MACHine{1|2}:SFORmat

Example:

```
OUTPUT XXX;":MACHINE2:SFORMAT:MASTER J, RISING"
```

CLOCk

command/query

The CLOCk command selects the clocking mode for a given pod when the pod is assigned to the state analyzer. When the NORMal option is specified, the pod will sample all 16 channels on the master clock. When the MIXed option is specified, the upper 8 bits will be sampled by the master clock and the lower 8 bits will be sampled by the slave clock. When the DEMultiplex option is specified, the lower 8 bits will be sampled on the slave clock and then sampled again on the master clock. The master clock always follows the slave clock when both are used.

The CLOCk query returns the current clocking mode for a given pod.

Command Syntax: :MACHine{1|2}:SFORmat:CLOCk<N> <clock_mode>

where:

<N> ::= Pod {1|2|3|4|5}
<clock_mode> ::= {NORMal | MIXed | DEMultiplex}

Example:

```
OUTPUT XXX;":MACHINE1:SFORMAT:CLOCK2 NORMAL"
```

Query Syntax: :MACHine{1|2}:SFORmat:CLOCk<N>?

Returned Format: [:MACHine{1|2}:SFORmat:CLOCK<N>] <clock_mode><NL>

Example:

```
10 DIM String$ [100]
20 OUTPUT XXX; ":MACHINE1:SFORMAT:CLOCK2?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

CPERiod

command/query

The CPERiod command allows you to set the state analyzer for input clock periods of greater than or less than 60 ns. Either LT or GT can be specified. LT signifies a state input clock period of less than 60 ns, and GT signifies a period of greater than 60 ns.

Because count tagging requires a minimum clock period of 60 ns, the CPERiod and TAG commands are interrelated (the TAG command is in the STRace subsystem). When the clock period is set to Less Than, count tagging is turned off. When count tagging is set to either state or time, the clock period is automatically set to Greater Than.

The CPERiod query returns the current setting of clock period.

Command Syntax: :MACHine{1|2}:SFORmat:CPERiod {LT|GT}

where:

GT ::= greater than 60 ns
LT ::= less than 60 ns

Example:

```
OUTPUT XXX;":MACHINE2:SFORMAT:CPERIOD GT"
```

Query Syntax: :MACHine{1|2}:SFORmat:CPERiod?

Returned Format: [:MACHine{1|2}:SFORmat:CPERiod] {GT|LT}<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:SFORMAT:CPERIOD?
30 ENTER XXX; String$
40 PRINT String$
50 END
```

LABel

command/query

The LABel command allows you to specify polarity and assign channels to new or existing labels. If the specified label name does not match an existing label name, a new label will be created.

The order of the pod-specification parameters is significant. The first one listed will match the highest-numbered pod assigned to the machine you're using. Each pod specification after that is assigned to the next-highest-numbered pod. This way they match the left-to-right descending order of the pods you see on the Format display. Not including enough pod specifications results in the lowest-numbered pod(s) being assigned a value of zero (all channels excluded). If you include more pod specifications than there are pods for that machine, the extra ones will be ignored. However, an error is reported anytime more than five pod specifications are listed.

The polarity can be specified at any point after the label name.

Since pods contain 16 channels, the format value for a pod must be between 0 and 65535 (2^{16}-1). When giving the pod assignment in binary (base 2), each bit will correspond to a single channel. A "1" in a bit position means the associated channel in that pod is assigned to that pod and bit. A "0" in a bit position means the associated channel in that pod is excluded from the label. For example, assigning #B1111001100 is equivalent to entering "......****..**.." through the front-panel user interface.

A label can not have a total of more than 32 channels assigned to it.

The LABel query returns the current specification for the selected (by name) label. If the label does not exist, nothing is returned. The polarity is always returned as the first parameter. Numbers are always returned in decimal format.

Command Syntax: :MACHine{1|2}:SFORmat:LABel <name>[, {<polarity> | <assignment>}]...

where:

<name> ::= string of up to 6 alphanumeric characters
<polarity> ::= {POSitive | NEGative}
<assignment> ::= format (integer from 0 to 65535) for a pod (pods are assigned in decreasing order)

Examples:

```
OUTPUT XXX;":MACHINE2:SFORMAT:LABEL 'STAT', POSITIVE, 65535,127,40312"
OUTPUT XXX;":MACHINE2:SFORMAT:LABEL 'SIG 1', 64, 12, 0, 20, NEGATIVE"
OUTPUT XXX;":MACHINE1:SFORMAT:LABEL 'ADDR', NEG, #B0011110010101010"
```

Query Syntax: :MACHine{1|2}:SFORmat:LABel?<name>

Returned Format: [:MACHine{1|2}:SFORmat:LABel] <name>,<polarity>[, <assignment>]...<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:SFORMAT:LABEL? 'DATA'"
30 ENTER XXX String$
40 PRINT String$
50 END
```

MASTer

command/query

The MASTer clock command allows you to specify a master clock for a given machine. The master clock is used in all clocking modes (Normal, Mixed, and Demultiplexed). Each command deals with only one clock (J,K,L,M,N); therefore, a complete clock specification requires five commands, one for each clock. Edge specifications (RISing, FALLing, or BOTH) are ORed. Level specifications (LOW or HIGH) are ANDed.

Note At least one clock edge must be specified.

The MASTer query returns the clock specification for the specified clock.

Command Syntax: :MACHine{1|2}:SFORmat:MASTer <clock_id>,<clock_spec>

where:

<clock_id> ::= {J|K|L|M|N}
<clock_spec> ::= {OFF|RISing|FALLing|BOTH|LOW|HIGH}

Example:

```
OUTPUT XXX;":MACHINE2:SFORMAT:MASTER J, RISING"
```

Query Syntax: :MACHine{1|2}:SFORmat:MASTer? <clock_id>

Returned Format: [:MACHine{1|2}:SFORmat:MASTer] <clock_id>,<clock_spec><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:SFORMAT:MASTER?<clock_id>"
30 ENTER XXX String$
40 PRINT String$
50 END
```

REMove

command

The REMove command allows you to delete all labels or any one label for a given machine.

Command Syntax: :MACHine{1|2}:SFORmat:REMove {<name>|ALL}

where:

<name> ::= string of up to 6 alphanumeric characters

Examples:

```
OUTPUT XXX;":MACHINE2:SFORMAT:REMOVE 'A'"
OUTPUT XXX;":MACHINE2:SFORMAT:REMOVE ALL"
```

SLAVe

command/query

The SLAVe clock command allows you to specify a slave clock for a given machine. The slave clock is only used in the Mixed and Demultiplexed clocking modes. Each command deals with only one clock (J,K,L,M,N); therefore, a complete clock specification requires five commands, one for each clock. Edge specifications (RISing, FALLing, or BOTH) are ORed. Level specifications (LOW or HIGH) are ANDed.

Note The slave clock must have at least one edge specified.

The SLAVe query returns the clock specification for the specified clock.

Command Syntax: :MACHine{1|2}:SFORmat:SLAVe <clock_id>,<clock_spec>

where:

<clock_id> ::= {J|K|L|M|N}
<clock_spec> ::= {OFF|RISing|FALLing|BOTH|LOW|HIGH}

Example:

```
OUTPUT XXX;":MACHINE2:SFORMAT:SLAVE J, RISING"
```

Query Syntax: :MACHine{1|2}:SFORmat:SLAVe?<clock_id>

Returned Format: [:MACHine{1|2}:SFORmat:SLAVe] <clock_id>,<clock_spec><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:SFORMAT:SLAVE? <clock_id>"
30 ENTER XXX String$
40 PRINT String$
50 END
```

THReshold

command/query

The THReshold command allows you to set the voltage threshold for a given pod to ECL, TTL, or a specific voltage from -9.9V to +9.9V in 0.1 volt increments.

Note

On the HP 1652B, the pod thresholds of pods 1, 2 and 3 can be set independently. The pod thresholds of pods 4 and 5 are slaved together; therefore, when you set the threshold on either pod 4 or 5, both thresholds will be changed to the specified value. On the HP 1653B, pods 1 and 2 can be set independently.

The THReshold query returns the current threshold for a given pod.

Command Syntax: :MACHine{1|2}:SFORmat:THReshold<N> {TTL|ECL|<value>}

where:

<N> ::= pod number {1|2|3|4|5}
<value> ::= voltage (real number) -9.9 to +9.9
TTL ::= default value of +1.6V
ECL ::= default value of -1.3V

Example:

```
OUTPUT XXX;":MACHINE1:SFORMAT:THRESHOLD1 4.0"
```

Query Syntax: :MACHine{1|2}:SFORmat:THReshold<N>?

Returned Format: [:MACHine{1|2}:SFORmat:THReshold<N>] <value><NL>

Example:

```
10 DIM Value$ [100]
20 OUTPUT XXX;":MACHINE1:SFORMAT:THRESHOLD4?"
30 ENTER XXX;Value$
40 PRINT Value$
50 END
```

12 - STRace Subsystem

STRace Subsystem 12

Introduction

The STRace subsystem contains the commands available for the State Trace menu in the HP 1652B/53B logic analyzer. The STRace subsystem commands are:

- BRANch
- FIND
- PREStore
- RANGe
- RESTart
- SEQuence
- STORe
- TAG
- TERM

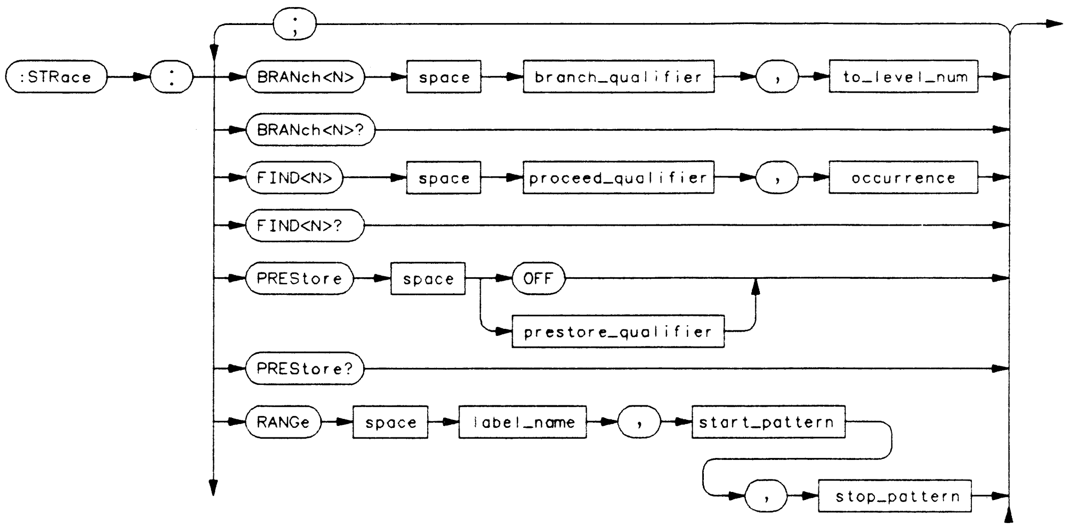


Figure 12-1. STRace Subsystem Syntax Diagram

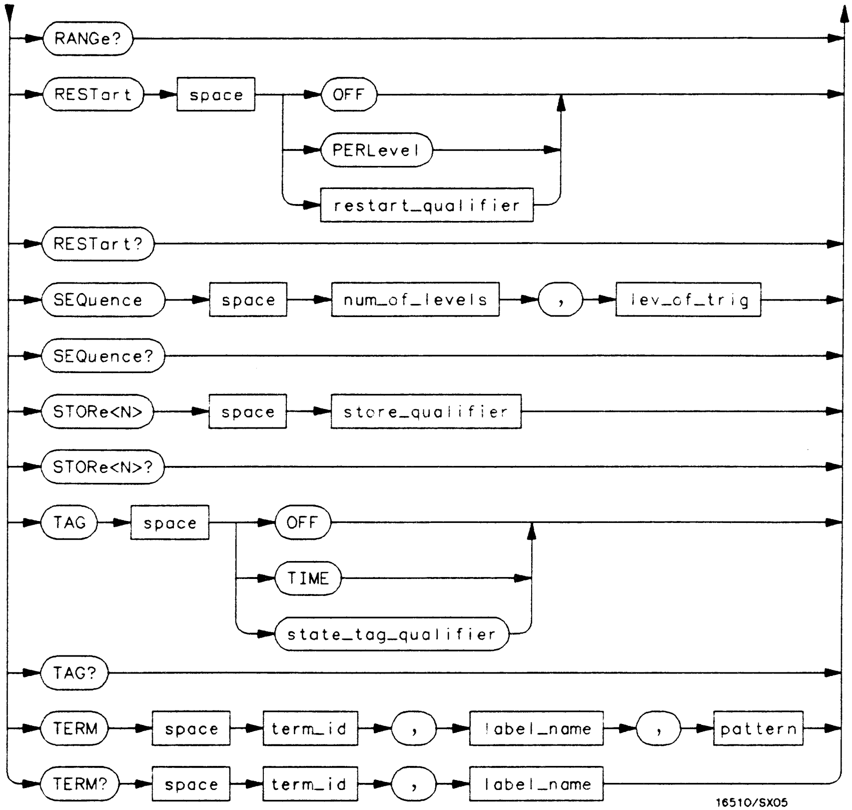


Figure 12-1. STRace Subsystem Syntax Diagram (continued)

branch_qualifier = *<qualifier>*

to_lev_num = *integer from 1 to trigger level when <N> is less than or equal to the trigger level, or from (trigger level + 1) to <num_of_levels> when <N> is greater than the trigger level*

proceed_qualifier = *<qualifier>*

occurrence = *number from 1 to 65535*

prestore_qual = *<qualifier>*

label_name = *string of up to 6 alphanumeric characters*

start_pattern = "{*#B{0|1}*. . . |
#Q{0|1|2|3|4|5|6|7}. . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F}. . . |
{0|1|2|3|4|5|6|7|8|9}. . . }"

stop_pattern = "{*#B{0|1}*. . . |
#Q{0|1|2|3|4|5|6|7}. . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F}. . . |
{0|1|2|3|4|5|6|7|8|9}. . . }"

restart_qualifier = *<qualifier>*

num_of_levels = *integer from 2 to 8 when ARM is RUN or from 2 to 7 otherwise*

lev_of_trig = *integer from 1 to (number of existing sequence levels - 1)*

store_qualifier = *<qualifier>*

state_tag_qualifier = *<qualifier>*

term_id = *{A|B|C|D|E|F|G|H}*

pattern = "{*#B{0|1|X}*. . . |
#Q{0|1|2|3|4|5|6|7|X}. . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X}. . . |
{0|1|2|3|4|5|6|7|8|9}. . . }"

qualifier = { *ANYState* | *NOSTate* | *<any_term>* | *(expression1[{AND|OR} <expression2>])* | *(expression2[{AND|OR} <expression1>])* }

any_term = {*<or_term1>* | *<and_term1>* | *<or_term2>* | *and_term2*}

expression1 = {*<or_term1>[OR <or_term1>]...* | *<and_term1>[AND <and_term1>]...*}

expression2 = {*<or_term2>[OR <or_term2>]...* | *<and_term2>[AND <and_term2>]...*}

or_term1 = *{A|B|C|D|INRange|OUTRange}*

and_term1 = *{NOTA|NOTB|NOTC|NOTD|INRange|OUTRange}*

or_term2 = *{E|F|G|H}*

and_term2 = *{NOTE|NOTF|NOTG|NOTH}*

Figure 12-1. STRace Subsystem Syntax Diagram (continued)

STRace

selector

The STRace (State Trace) selector is used as a part of a compound header to access the settings found in the State Trace menu. It always follows the MACHine selector because it selects a branch directly below the MACHine level in the command tree.

Command Syntax: :MACHine{1|2}:STRace

Example: `OUTPUT XXX;":MACHINE1:STRACE:TAG TIME"`

BRANch

command/query

The BRANch command defines the branch qualifier for a given sequence level. When this branch qualifier is matched, it will cause the sequencer to jump to the specified sequence level.

Note "RESTART PERLEVEL" must have been invoked for this command to have an effect (see RESTart command).

The terms used by the branch qualifier (A through H) are defined by the TERM command. The meaning of INRange and OUTRange is determined by the RANGe command.

Within the limitations shown by the syntax definitions, complex expressions may be formed using the AND and OR operators. Expressions are limited to what you could manually enter through the front panel. Regarding parentheses, the syntax definitions on the next page show only the required ones. Additional parentheses are allowed as long as the meaning of the expression is not changed. For example, the following two statements are both correct and have the same meaning. Notice that the conventional rules for precedence are not followed.

```
OUTPUT XXX;":MACHINE1:STRACE:BRANCH1 (C OR D AND F OR G), 1"
OUTPUT XXX;":MACHINE1:STRACE:BRANCH1 ((C OR D) AND (F OR G)), 1"
```

Figure 12-2 shows a complex expression as seen on the Format display.

Note Branching across the trigger level is not allowed. Therefore, the values for <N> and <to_level_num> must both be either on or before the trigger level, or they must both be after the trigger level . The trigger level is determined through the SEQuence command.

The BRANch query returns the current branch qualifier specification for a given sequence level.

BRANch

Command Syntax: :MACHine{1|2}:STRace:BRANch<N> <branch_qualifier>,<to_level_number>

where:

<N>	::= an integer from 1 to <number_of_levels>
<to_level_number>	::= integer from 1 to trigger level, when <N> is less than or equal to the trigger level or from (trigger level + 1) to <number_of_levels>, when <N> is greater than the trigger level
<number_of_levels>	::= integer from 2 to the number of existing sequence levels (maximum 8)
<branch_qualifier>	::= { ANYState | NOSTate | <any_term> | (<expression1>[{AND|OR} <expression2>]) | (<expression2>[{AND|OR} <expression1>]) }
<any_term>	::= {<or_term1> | <and_term1> | <or_term2> | <and_term2>}
<expression1>	::= {<or_term1>[OR <or_term1>]... | <and_term1>[AND <and_term1>]...}
<expression2>	::= {<or_term2>[OR <or_term2>]... | <and_term2>[AND <and_term2>]...}
<or_term1>	::= {A|B|C|D|INRange|OUTRange}
<and_term1>	::= {NOTA|NOTB|NOTC|NOTD|INRange|OUTRange}
<or_term2>	::= {E|F|G|H}
<and_term2>	::= {NOTE|NOTF|NOTG|NOTH}

Examples:

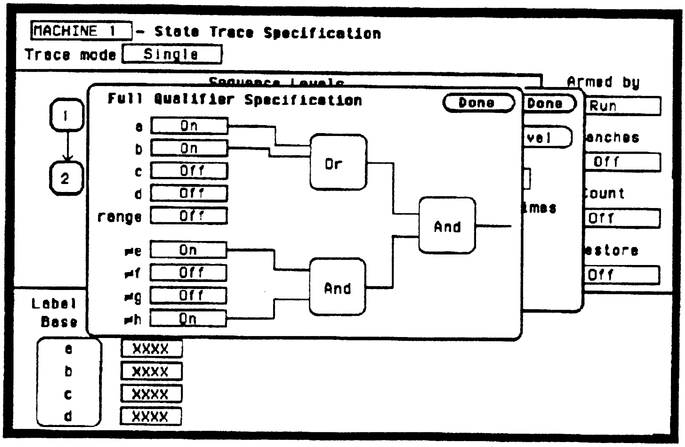
```
OUTPUT XXX;":MACHINE1:STRACE:BRANCH1 ANYSTATE, 3"
OUTPUT XXX;":MACHINE2:STRACE:BRANCH2 A, 7"
OUTPUT XXX;":MACHINE1:STRACE:BRANCH3 ((A OR B) OR NOTG), 1"
```

Query Syntax :MACHine{1|2}:STRace:BRANch<N>?

Returned Format: [:MACHine{1|2}:STRace:BRANch<N>]<branch_qualifier>,<to_level_num><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:BRANCH3?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```



.Figure 12-2. Complex qualifier

Figure 12-2 is a front panel representation of the complex qualifier **(a Or b) And (≠e And ≠h)**. The following example would be used to specify this complex qualifier.

```
OUTPUT XXX;":MACHINE1:STRACE:BRANCH1 ((A OR B) AND (NOTE AND NOTH)), 2"
```

Note

Terms **A** through **D** and **RANGE** must be grouped together and terms **E** through **H** must be grouped together. In the first level, terms from one group may not be mixed with terms from the other. For example, the expression ((A OR INRANGE) AND (C OR H)) is not allowed because the term C cannot be specified in the **E** through **H** group.

Keep in mind that, at the first level, the operator you use determines which terms are available. When AND is chosen, only the **NOT** terms may be used. Either AND or OR may be used at the second level to join the two groups together. It is acceptable for a group to consist of a single term. Thus, an expression like (B AND G) is legal, since the two operands are both simple terms from separate groups.

FIND

command/query

The FIND command defines the proceed qualifier for a given sequence level. The qualifier tells the state analyzer when to proceed to the next sequence level. When this proceed qualifier is matched the specified number of times, the sequencer will proceed to the next sequence level. The state that causes the sequencer to switch levels is automatically stored in memory whether it matches the associated store qualifier or not. In the sequence level where the trigger is specified, the FIND command specifies the trigger qualifier (see SEQuence command).

The terms A through H are defined by the TERM command. The meaning of INRange and OUTRange is determined by the RANGe command. Expressions are limited to what you could manually enter through the Format menu. Regarding parentheses, the syntax definitions below show only the required ones. Additional parentheses are allowed as long as the meaning of the expression is not changed. See figure 6-2 for a detailed example.

The FIND query returns the current proceed qualifier specification for a given sequence level.

Command Syntax: :MACHine{1|2}:STRace:FIND<N> <proceed_qualifier>,<occurrence>

where:

<N> ::= integer from 1 to the number of existing sequence levels (maximum 8)
<occurrence> ::= integer from 1 to 65535
<proceed_qualifier> ::= { ANYState | NOSTate | <any_term> |
(<expression1>[{AND|OR} <expression2>]) |
(<expression2>[{AND|OR} <expression1>]) }
<any_term> ::= {<or_term1> | <and_term1> | <or_term2> | <and_term2>}
<expression1> ::= {<or_term1>[OR <or_term1>]... | <and_term1>[AND <and_term1>]...}
<expression2> ::= {<or_term2>[OR <or_term2>]... | <and_term2>[AND <and_term2>]...}
<or_term1> ::= {A|B|C|D|INRange|OUTRange}
<and_term1> ::= {NOTA|NOTB|NOTC|NOTD|INRange|OUTRange}
<or_term2> ::= {E|F|G|H}
<and_term2> ::= {NOTE|NOTF|NOTG|NOTH}

Examples:

```
OUTPUT XXX;":MACHINE1:STRACE:FIND1 ANYSTATE, 1"
OUTPUT XXX;":MACHINE1:STRACE:FIND2 A, 512"
OUTPUT XXX;":MACHINE1:STRACE:FIND3 ((NOTA AND NOTB) OR G), 1"
```

Query Syntax: :MACHine{1|2}:STRace:FIND4?

Returned Format: [:MACHine{1|2}:STRace:FIND<N>] <proceed_qualifier>,<occurrence><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:FIND<N>?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

PREStore

command/query

The PREStore command turns the prestore feature on and off. It also defines the qualifier required to prestore only selected states. The terms A through H are defined by the TERM command. The meaning of INRange and OUTRange is determined by the RANGe command.

Expressions are limited to what you could manually enter through the Format menu. Regarding parentheses, the syntax definitions below show only the required ones. Additional parentheses are allowed as long as the meaning of the expression is not changed.

A detailed example is provided in figure 12-2.

The PREStore query returns the current prestore specification.

Command Syntax: :MACHine{1|2}:STRace:PREStore {OFF | <prestore_qualifier>}

where:

<prestore_qualifier>	::= { ANYState | NOSTate | <any_term> | (<expression1>[{AND|OR} <expression2>]) | (<expression2>[{AND|OR} <expression1>]) }
<any_term>	::= {<or_term1> | <and_term1> | <or_term2> | <and_term2>}
<expression1>	::= {<or_term1>[OR <or_term1>]... | <and_term1>[AND <and_term1>]...}
<expression2>	::= {<or_term2>[OR <or_term2>]... | <and_term2>[AND <and_term2>]...}
<or_term1>	::= {A|B|C|D|INRange|OUTRange}
<and_term1>	::= {NOTA|NOTB|NOTC|NOTD|INRange|OUTRange}
<or_term2>	::= {E|F|G|H}
<and_term2>	::= {NOTE|NOTF|NOTG|NOTH}

Examples:

```
OUTPUT XXX;":MACHINE1:STRACE:PRESTORE OFF"
OUTPUT XXX;":MACHINE1:STRACE:PRESTORE ANYSTATE"
OUTPUT XXX;":MACHINE1:STRACE:PRESTORE (E)"
OUTPUT XXX;":MACHINE1:STRACE:PRESTORE (A OR B OR D OR F OR H)"
```

Query Syntax: :MACHine{1|2}:STRace:PREStore?

Returned Format: [:MACHine{1|2}:STRace:PREStore] {OFF|<prestore_qualifier>}<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:PRESTORE?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

RANGe

command/query

The RANGe command allows you to specify a range recognizer term in the specified machine. Since a range can only be defined across one label and, since a label must contain 32 or less bits, the value of the start pattern or stop pattern will be between $(2^{32})-1$ and 0.

Note Since a label can only be defined across a maximum of two pods, a range term is only available across a single label; therefore, the end points of the range cannot be split between labels.

When these values are expressed in binary, they represent the bit values for the label at one of the range recognizers' end points. Don't cares are not allowed in the end point pattern specifications. Since only one range recognizer exists, it is always used by the first state machine defined.

The RANGe query returns the range recognizer end point specifications for the range.

Note When two state analyzers are on, the RANGe term is not available in the second state analyzer assigned and there are only 4 pattern recognizers per analyzer.

Command Syntax: :MACHine{1|2}:STRace:RANGE <label_name>,<start_pattern>,<stop_pattern>

where:

<label_name> ::= string of up to 6 alphanumeric characters

<start_pattern> ::= "{#B{0|1} . . . |
#Q{0|1|2|3|4|5|6|7} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

<stop_pattern> ::= "{#B{0|1} . . . |
#Q{0|1|2|3|4|5|6|7} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Examples:

```
OUTPUT XXX;":MACHINE1:STRACE:RANGE 'DATA', '127', '255' "
OUTPUT XXX;":MACHINE1:STRACE:RANGE 'ABC', '#B00001111', '#HCF' "
```

Query Syntax: :MACHine{1|2}:STRace:RANGe?

Returned Format: [:MACHine{1|2}:STRAce:RANGe]
<label_name>,<start_pattern>,<stop_pattern><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:RANGE?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

RESTart

command/query

The RESTart command selects the type of restart to be enabled during the trace sequence. It also defines the global restart qualifier that restarts the sequence in global restart mode. The qualifier may be a single term or a complex expression. The terms A through H are defined by the TERM command. The meaning of INRange and OUTRange is determined by the RANGe command.

Expressions are limited to what you could manually enter through the Format menu. Regarding parentheses, the syntax definitions below show only the required ones. Additional parentheses are allowed as long as the meaning of the expression is not changed.

A detailed example is provided in figure 12-2.

The RESTart query returns the current restart specification.

Command Syntax: :MACHine{1|2}:STRace:RESTart {OFF | PERLevel | <restart_qualifier>}

where:

```
<restart_qualifier>  ::= { ANYState | NOSTate | <any_term> |
                           (<expression1>[{AND|OR} <expression2>]) |
                           (<expression2>[{AND|OR} <expression1>]) }
<any_term>           ::= {<or_term1> | <and_term1> | <or_term2> | <and_term2>}
<expression1>        ::= {<or_term1>[OR <or_term1>]... | <and_term1>[AND <and_term1>]...}
<expression2>        ::= {<or_term2>[OR <or_term2>]... | <and_term2>[AND <and_term2>]...}
<or_term1>           ::= {A|B|C|D|INRange|OUTRange}
<and_term1>          ::= {NOTA|NOTB|NOTC|NOTD|INRange|OUTRange}
<or_term2>           ::= {E|F|G|H}
<and_term2>          ::= {NOTE|NOTF|NOTG|NOTH}
```

Examples:

```
OUTPUT XXX;":MACHINE1:STRACE:RESTART OFF"
OUTPUT XXX;":MACHINE1:STRACE:RESTART PERLEVEL"
OUTPUT XXX;":MACHINE1:STRACE:RESTART (NOTA AND NOTB AND INRANGE)"
OUTPUT XXX;":MACHINE1:STRACE:RESTART (B OR (NOTE AND NOTF))"
```

Query Syntax: :MACHine{1|2}:STRace:RESTart?

Returned Format: [:MACHine{1|2}:STRace:RESTart] {OFF | PERLevel | <restart_qualifier>}<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:RESTART?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

SEQuence

command/query

The SEQuence command redefines the state analyzer trace sequence. First, it deletes the current trace sequence. Then it inserts the number of levels specified, with default settings, and assigns the trigger to be at a specified sequence level. The number of levels can be between 2 and 8 when the analyzer is armed by the RUN key. When armed by the BNC or the other machine, a level is used by the arm in; therefore, only seven levels are available in the sequence.

The SEQuence query returns the current sequence specification.

Command Syntax: :MACHine{1|2}:STRace:SEQuence <number_of_levels>,<level_of_trigger>

where:

<number_of_levels> ::= integer from 2 to 8 when ARM is RUN or from 2 to 7 otherwise

<level_of_trigger> ::= integer from 1 to (number of existing sequence levels - 1)

Example:

```
OUTPUT XXX;":MACHINE1:STRACE:SEQUENCE 4,3"
```

Query Syntax: :MACHine{1|2}:STRace:SEQuence?

Returned Format: [:MACHine{1|2}:STRace:SEQuence] <number_of_levels>,<level_of_trigger><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:SEQUENCE?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

STORe

command/query

The STORe command defines the store qualifier for a given sequence level. Any data matching the STORe qualifier will actually be stored in memory as part of the current trace data. The qualifier may be a single term or a complex expression. The terms A through H are defined by the TERM command. The meaning of INRange and OUTRange is determined by the RANGe command.

Expressions are limited to what you could manually enter through the Format menu. Regarding parentheses, the syntax definitions below show only the required ones. Additional parentheses are allowed as long as the meaning of the expression is not changed.

A detailed example is provided in figure 12-2.

The STORe query returns the current store qualifier specification for a given sequence level <N>.

Command Syntax: `:MACHine{1|2}:STRace:STORe<N> <store_qualifier>`

where:

```
<N>                ::= an integer from 1 to the number of existing sequence levels (maximum 8)
<store_qualifier>  ::= { ANYState | NOSTate | <any_term> |
                         (<expression1>[{AND|OR} <expression2>]) |
                         (<expression2>[{AND|OR} <expression1>]) }
<any_term>         ::= {<or_term1> | <and_term1> | <or_term2> | <and_term2>}
<expression1>      ::= {<or_term1>[OR <or_term1>]... | <and_term1>[AND <and_term1>]...}
<expression2>      ::= {<or_term2>[OR <or_term2>]... | <and_term2>[AND <and_term2>]...}
<or_term1>         ::= {A|B|C|D|INRange|OUTRange}
<and_term1>        ::= {NOTA|NOTB|NOTC|NOTD|INRange|OUTRange}
<or_term2>         ::= {E|F|G|H}
<and_term2>        ::= {NOTE|NOTF|NOTG|NOTH}
```

STORe

Examples:

```
OUTPUT XXX;":MACHINE1:STRACE:STORE1 ANYSTATE"
OUTPUT XXX;":MACHINE1:STRACE:STORE2 OUTRANGE"
OUTPUT XXX;":MACHINE1:STRACE:STORE3 (NOTC AND NOTD AND NOTH)"
```

Query Syntax: :MACHine{1|2}:STRace:STORe<N>?

Returned Format: [:MACHine{1|2}:STRace:STORe<N>] <store_qualifier><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:STORE4?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

TAG

command/query

The TAG command selects the type of count tagging (state or time) to be performed during data acquisition. State tagging is indicated when the parameter is the state tag qualifier, which will be counted in the qualified state mode. The qualifier may be a single term or a complex expression. The terms A through H are defined by the TERM command. The terms INRange and OUTRange are defined by the RANGe command.

Expressions are limited to what you could manually enter through the Format menu. Regarding parentheses, the syntax definitions below show only the required ones. Additional parentheses are allowed as long as the meaning of the expression is not changed. A detailed example is provided in figure 12-2.

Because count tagging requires a minimum clock period of 60 ns, the CPERiod and TAG commands are interrelated (the CPERiod command is in the SFORmat subsystem). When the clock period is set to Less Than, count tagging is turned off. When count tagging is set to either state or time, the clock period is automatically set to Greater Than.

The TAG query returns the current count tag specification.

Command Syntax: :MACHine{1|2}:STRace:TAG {OFF | TIME | <state_tag_qualifier>}

where:

<state_tag_qualifier>	::= { ANYState | NOSTate | <any_term> | (<expression1> [{AND|OR} <expression2>]) | (<expression2> [{AND|OR} <expression1>]) }
<any_term>	::= {<or_term1> | <and_term1> | <or_term2> | <and_term2>}
<expression1>	::= {<or_term1>[OR <or_term1>]... | <and_term1>[AND <and_term1>]...}
<expression2>	::= {<or_term2>[OR <or_term2>]... | <and_term2>[AND <and_term2>]...}
<or_term1>	::= {A|B|C|D|INRange|OUTRange}
<and_term1>	::= {NOTA|NOTB|NOTC|NOTD|INRange|OUTRange}
<or_term2>	::= {E|F|G|H}
<and_term2>	::= {NOTE|NOTF|NOTG|NOTH}

TAG

Examples:

```
OUTPUT XXX;":MACHINE1:STRACE:TAG OFF"
OUTPUT XXX;":MACHINE1:STRACE:TAG TIME"
OUTPUT XXX;":MACHINE1:STRACE:TAG (INRANGE OR NOTF)"
OUTPUT XXX;":MACHINE1:STRACE:TAG ((INRANGE OR A) AND E)"
```

Query Syntax: :MACHine{1|2} :STRace:TAG?

Returned Format: [:MACHine{1|2}:STRace:TAG] {OFF|TIME|<state_tag_qualifier>}<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:TAG?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

TERM

command/query

The TERM command allows you to a specify a pattern recognizer term in the specified machine. Each command deals with only one label in the given term; therefore, a complete specification could require several commands. Since a label can contain 32 or less bits, the range of the pattern value will be between $2^{32} - 1$ and 0. When the value of a pattern is expressed in binary, it represents the bit values for the label inside the pattern recognizer term. Since the pattern parameter may contain don't cares and be represented in several bases, it is handled as a string of characters rather than a number.

When a single state machine is on, all eight terms (A through H) are available in that machine. When two state machines are on, terms A through D are used by the first state machine defined, and terms E through H are used by the second state machine defined.

The TERM query returns the specification of the term specified by term identification and label name.

Command Syntax: :MACHine{1|2}:STRace:TERM <term_id>,<label_name>,<pattern>

where:

<term_id> ::= {A|B|C|D|E|F|G|H}

<label_name> ::= string of up to 6 alphanumeric characters

<pattern> ::= "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Example:

```
OUTPUT XXX;":MACHINE1:STRACE:TERM A,'DATA','255' "
OUTPUT XXX;":MACHINE1:STRACE:TERM B,'ABC','#BXXXX1101' "
```

TERM

Query Syntax: :MACHine{1|2}:STRace:TERM? <term_id>,<label_name>

Returned Format: [:MACHine{1|2}:STRAce:TERM] <term_id>,<label_name>,<pattern><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:STRACE:TERM? B,'DATA' "
30 ENTER XXX;String$
40 PRINT String$
50 END
```

13 - SLISt Subsystem

SLISt Subsystem 13

Introduction

The SLISt subsystem contains the commands available for the State Listing menu in the HP 1652B/53B logic analyzer. These commands are:

- COLumn
- DATA
- LINE
- MMODe
- OPATtern
- OSEarch
- OSTate
- OTAG
- RUNTil
- TAVerage
- TMAXimum
- TMINimum
- VRUNs
- XOTag
- XPATtern
- XSEarch
- XSTate
- XTAG

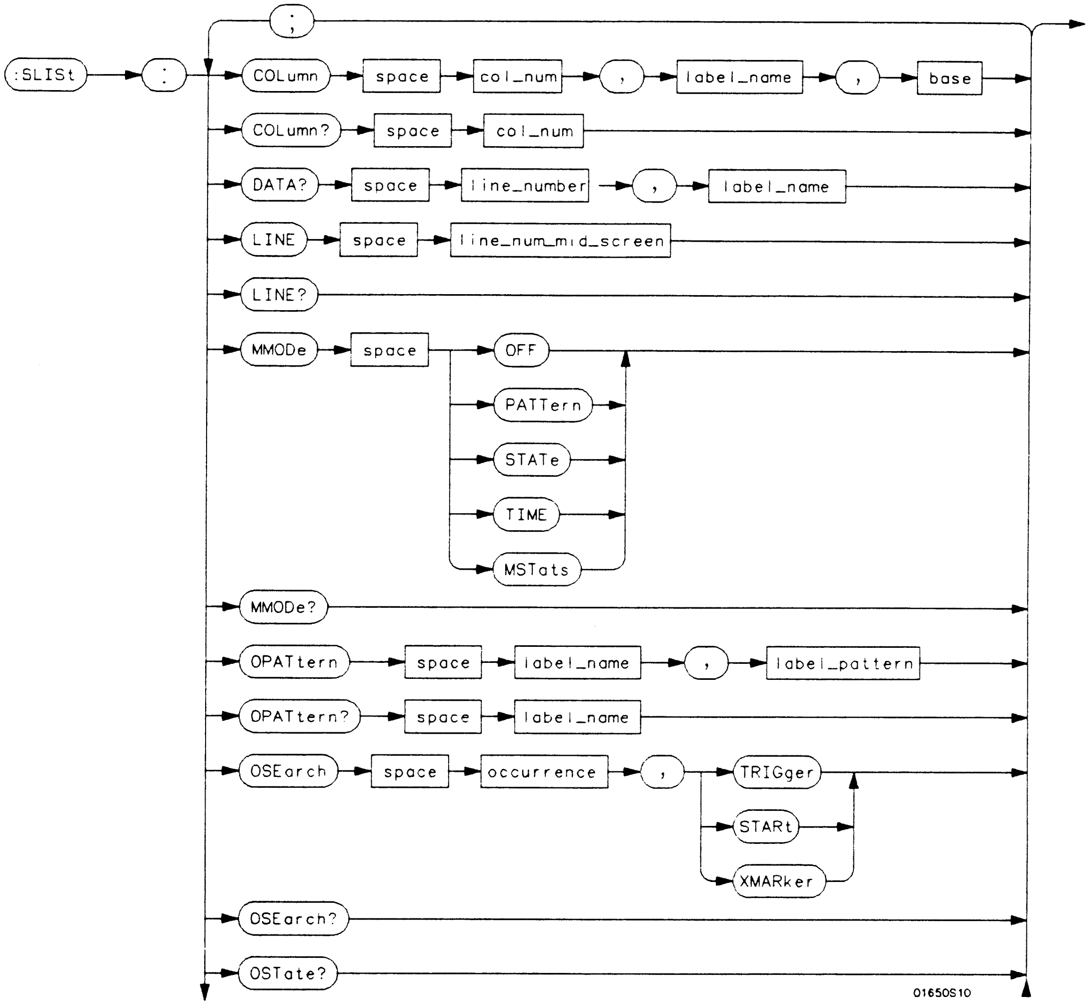


Figure 13-1. SLISt Subsystem Syntax Diagram

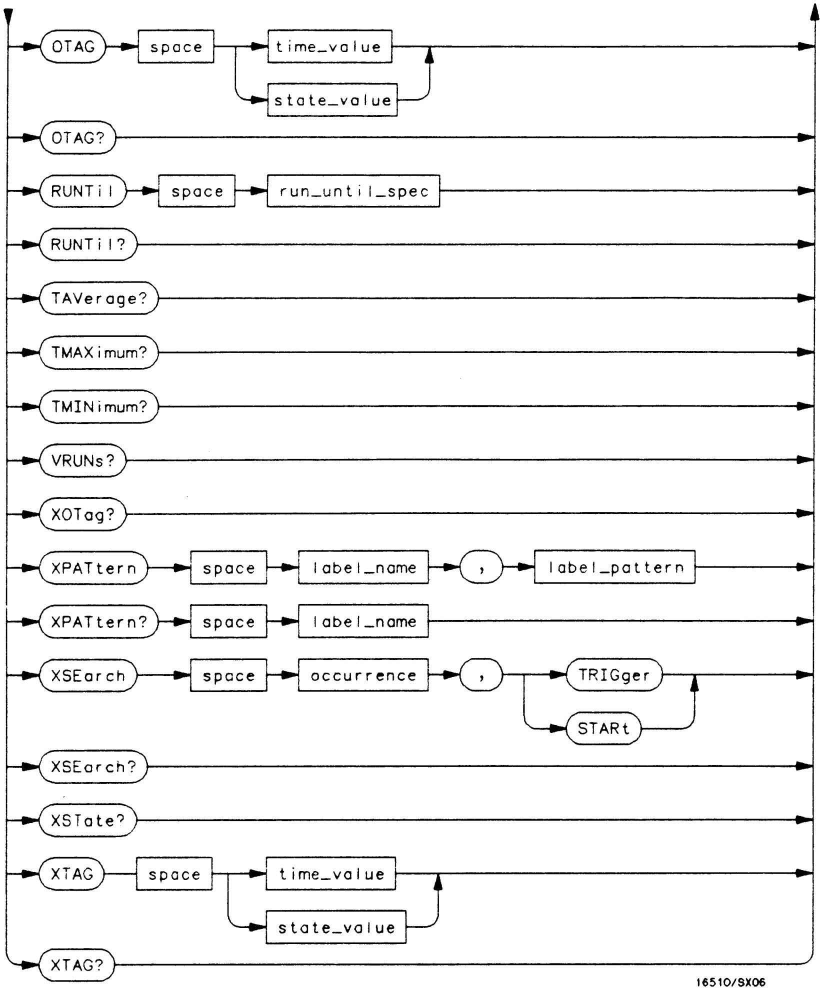


Figure 13-1. SLISt Subsystem Syntax Diagram (continued)

module_num = {*1*|*2*|*3*|*4*|*5*}
mach_num = {*1*|*2*}
col_num = {*1*|*2*|*3*|*4*|*5*|*6*|*7*|*8*}
line_number = *integer from -1023 to +1023*
label_name = *a string of up to 6 alphanumeric characters*
base = {*BINary*|*HEXadecimal*|*OCTal*|*DECimal*|*ASCii*|*SYMBol*|*IASSembler*} *for labels or*
{*ABSolute*|*RELative*} *for tags*
line_num_mid_screen = *integer from -1023 to +1023*
label_pattern = "{*#B*{*0*|*1*|*X*} . . . |
#Q{*0*|*1*|*2*|*3*|*4*|*5*|*6*|*7*|*X*} . . . |
#H{*0*|*1*|*2*|*3*|*4*|*5*|*6*|*7*|*8*|*9*|*A*|*B*|*C*|*D*|*E*|*F*|*X*} . . . |
{*0*|*1*|*2*|*3*|*4*|*5*|*6*|*7*|*8*|*9*} . . . }"
occurrence = *integer from -1023 to +1023*
time_value = *real number*
state_value = *real number*
run_until_spec = {*OFF*|*LT*, *<value>* |*GT*, *<value>* |*INRange*, *<value>*, *<value>* |
OUTRange, *<value>*, *<value>* }
value = *real number*

Figure 13-1. SLISt Subsystem Syntax Diagram (continued)

SLISt

selector

The SLISt selector is used as part of a compound header to access those settings normally found in the State Listing menu. It always follows the MACHine selector because it selects a branch directly below the MACHine level in the command tree.

Command Syntax: :MACHine{1|2}:SLISt

Example:

```
OUTPUT XXX;":MACHINE1:SLIST:LINE 256"
```

COLumn

command/query

The COLumn command allows you to configure the state analyzer list display by assigning a label name and base to one of the eight vertical columns in the menu. A column number of 1 refers to the left most column. When a label is assigned to a column it replaces the original label in that column. The label originally in the specified column is placed in the column the specified label is moved from.

When the label name is "TAGS," the TAGS column is assumed and the next parameter must specify RELative or ABSolute.

The COLumn query returns the column number, label name, and base for the specified column.

Command Syntax: :MACHine{1|2}:SLISt:COLumn <col_num>,<label_name>,<base>

where:

<col_num> ::= {1|2|3|4|5|6|7|8}

<label_name> ::= a string of up to 6 alphanumeric characters

<base> ::= {BINary|HEXadecimal|OCTal|DECimal|ASCii|SYMBol|IASSembler} for labels

or

::= {ABSolute|RELative} for tags

Note

A label for tags must be assigned in order to use ABSolute or RELative state tagging.

Examples:

```
OUTPUT XXX;":MACHINE1:SLIST:COLUMN 4,2,MACHINE1,'A',HEX"
OUTPUT XXX;":MACHINE1:SLIST:COLUMN 1,2,MACHINE1,'TAGS', ABSOLUTE"
```

Query Syntax: :MACHine{1|2}:SLISt:COLumn? <col_num>

Returned Format: [:MACHine{1|2}:SLISt:COLumn] <col_num>,<label_name>,<base><NL>

Example:

```
10 DIM C1$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:COLUMN? 4"
30 ENTER XXX;C1$
40 PRINT C1$
50 END
```

DATA

query

The DATA query returns the value at a specified line number for a given label. The format will be the same as the one shown in the Listing display except for ASCII, Symbols, or Inverse Assembly which will be returned in HEX.

Query Syntax: :MACHine{1|2}:SLISt:DATA? <line_number>,<label_name>

Returned Format: [:MACHine{1|2}:SLISt:DATA]
<line_number>,<label_name>,<pattern_string><NL>

where:

<line_number> ::= integer from -1023 to +1023

<label_name> ::= string of up to 6 alphanumeric characters

<pattern_string> ::= "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Example:

```
10 DIM Sd$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:DATA? 512, 'RAS'"
30 ENTER XXX;Sd$
40 PRINT Sd$
50 END
```

LINE

command/query

The LINE command allows you to scroll the state analyzer listing vertically. The command specifies the state line number relative to the trigger that the analyzer will be highlighted at center screen.

The LINE query returns the line number for the state currently in the box at center screen.

Command Syntax: :MACHine{1|2}:SLISt:LINE <line_num_mid_screen>

where:

<line_num_mid_screen> ::= integer from -1023 to +1023

Example:

```
OUTPUT XXX;":MACHINE1:SLIST:LINE 0"
```

Query Syntax: :MACHine{1|2}:SLISt:LINE?

Returned Format: [:MACHine{1|2}:SLISt:LINE] <line_num_mid_screen><NL>

Example:

```
10 DIM Ln$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:LINE?"
30 ENTER XXX;Ln$
40 PRINT Ln$
50 END
```

MMODe

command/query

The MMODe command (Marker Mode) selects the mode controlling the marker movement and the display of marker readouts. When PATTern is selected, the markers will be placed on patterns. When STATe is selected and state tagging is on, the markers move on qualified states counted between normally stored states. When TIME is selected and time tagging is enabled, the markers move on time between stored states. When MSTats is selected and time tagging is on, the markers are placed on patterns, but the readouts will be time statistics.

The MMODe query returns the current marker mode selected.

Command Syntax: :MACHine{1|2}:SLISt:MMODe <marker_mode>

where:

<marker_mode> ::= {OFF|PATTern|STATe|TIME|MSTats}

Example:

```
OUTPUT XXX;":MACHINE1:SLIST:MMODE TIME"
```

Query Syntax: :MACHine{1|2}:SLISt:MMODe?

Returned Format: [:MACHine{1|2}:SLISt:MMODe] <marker_mode><NL>

Example:

```
10 DIM Mn$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:MMODE?"
30 ENTER XXX;Mn$
40 PRINT Mn$
50 END
```

OPATtern

command/query

The OPATtern command allows you to construct a pattern recognizer term for the O Marker which is then used with the OSEarch criteria when moving the marker on patterns. Since this command deals with only one label at a time, a complete specification could require several invocations.

When the value of a pattern is expressed in binary, it represents the bit values for the label inside the pattern recognizer term. In whatever base is used, the value must be between 0 and 2^{32} - 1, since a label may not have more than 32 bits. Because the <label_pattern> parameter may contain don't cares, it is handled as a string of characters rather than a number.

The OPATtern query returns the pattern specification for a given label name.

Command Syntax: :MACHine{1|2}:SLISt:OPATtern <label_name>,<label_pattern>

where:

<label_name> ::= string of up to 6 alphanumeric characters

<label_pattern> ::= "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Examples:

```
OUTPUT XXX;":MACHINE1:SLIST:OPATTERN 'DATA','255' "
OUTPUT XXX;":MACHINE1:SLIST:OPATTERN 'ABC','#BXXXX1101' "
```

OPATtern

Query Syntax: :MACHine{1|2}:SLISt:OPATtern? <label_name>

Returned Format: [:MACHine{1|2}:SLISt:OPATtern] <label_name>,<label_pattern><NL>

Example:

```
10 DIM Op$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:OPATTERN? 'A'"
30 ENTER XXX;Op$
40 PRINT Op$
50 END
```

OSEarch

command/query

The OSEarch command defines the search criteria for the O marker, which is then used with associated OPATtern recognizer specification when moving the markers on patterns. The origin parameter tells the marker to begin a search with the trigger, the start of data, or with the X marker. The actual occurrence the marker searches for is determined by the occurrence parameter of the OPATtern recognizer specification, relative to the origin. An occurrence of 0 places the marker on the selected origin. With a negative occurrence, the marker searches before the origin. With a positive occurrence, the marker searches after the origin.

The OSEarch query returns the search criteria for the O marker.

Command Syntax: :MACHine{1|2}:SLISt:OSEarch <occurrence>,<origin>

where:

<occurrence> ::= integer from -1023 to +1023

<origin> ::= {TRIGger|STARt|XMARker}

Example:

```
OUTPUT XXX;":MACHINE1:SLIST:OSEARCH +10,TRIGGER"
```

Query Syntax: :MACHine{1|2}:SLISt:OSEarch?

Returned Format: [:MACHine{1|2}:SLISt:OSEarch] <occurrence>,<origin><NL>

Example:

```
10 DIM Os$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:OSEARCH?"
30 ENTER XXX;Os$
40 PRINT Os$
50 END
```

OSTate

query

The OSTate query returns the line number in the listing where the O marker resides (-1023 to +1023). If data is not valid, the query returns 32767.

Query Syntax: :MACHine{1|2}:SLISt:OSTate?

Returned Format: [:MACHine{1|2}:SLISt:OSTate] <state_num><NL>

where:

<state_num> ::= an integer from -1023 to +1023, or 32767

Example:

```
10 DIM Os$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:OSTATE?"
30 ENTER XXX;Os$
40 PRINT Os$
50 END
```

OTAG

command/query

The OTAG command specifies the tag value on which the O Marker should be placed. The tag value is time when time tagging is on or states when state tagging is on. If the data is not valid tagged data, no action is performed.

The OTAG query returns the O Marker position in time when time tagging is on or in states when state tagging is on, regardless of whether the marker was positioned in time or through a pattern search. If data is not valid, the query returns 9.9E37 for time tagging, 32767 for state tagging.

Command Syntax: :MACHine{1|2}:SLISt:OTAG {<time_value>|<state_value>}

where:

<time_value> ::= real number

<state_value> ::= integer

Example:

```
:OUTPUT XXX;":MACHINE1:SLIST:OTAG 40.0E-6"
```

Query Syntax: :MACHine{1|2}:SLISt:OTAG?

Returned Format: [:MACHine{1|2}:SLISt:OTAG] {<time_value>|<state_value>}<NL>

Example:

```
10 DIM Ot$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:OTAG?"
30 ENTER XXX;Ot$
40 PRINT Ot$
50 END
```

RUNTil

command/query

The RUNTil (run until) command allows you to define a stop condition when the trace mode is repetitive. Specifying OFF causes the analyzer to make runs until either the display's STOP field is touched or the STOP command is issued.

There are four conditions based on the time between the X and O markers. Using this difference in the condition is effective only when time tags have been turned on (see the TAG command in the STRace subsystem). These four conditions are as follows:

- The difference is less than (LT) some value.
- The difference is greater than (GT) some value.
- The difference is inside some range (INRange).
- The difference is outside some range (OUTRange).

End points for the INRange and OUTRange should be at least 40 ns apart since this is the minimum time resolution of the time tag counter.

There are two conditions which are based on a comparison of the acquired state data and the compare data image. You can run until one of the following conditions is true:

- Compare Equal (EQUal) - Every channel of every label has the same value.
- Compare not equal (NEQual) - Any channel of any label has a different value.

The RUNTil query returns the current stop criteria.

Note The RUNTil instruction (for state analysis) is available in both the SLISt and COMPare subsystems.

Command Syntax: :MACHine{1|2}:SLISt:RUNTil <run_until_spec>

where:

<run_until_spec> ::= {OFF|LT,<value>|GT,<value>|INRange,<value>,<value>
|OUTRange,<value>,<value>|EQUal|NEQual}

<value> ::= real number from 10E-9 to +9E9

Example:

```
OUTPUT XXX;":MACHINE1:SLIST:RUNTIL GT,800.0E-6"
```

Query Syntax: :MACHine{1|2}:SLISt:RUNTil?

Returned Format: [:MACHine{1|2}:SLISt:RUNTil] <run_until_spec><NL>

Example:

```
10 DIM Ru$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:RUNTIL?"
30 ENTER XXX;Ru$
40 PRINT Ru$
50 END
```

TAVerage

query

The TAVerage query returns the value of the average time between the X and O Markers. If the number of valid runs is zero, the query returns 9.9E37. Valid runs are those where the pattern search for both the X and O markers was successful, resulting in valid delta-time measurements.

Query Syntax: :MACHine{1|2}:SLISt:TAVerage?

Returned Format: [:MACHine{1|2}:SLISt:TAVerage] <time_value><NL>

where:

<time_value> ::= real number

Example:

```
10 DIM Tv$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:TAVERAGE?"
30 ENTER XXX;Tv$
40 PRINT Tv$
50 END
```

TMAXimum

query

The TMAXimum query returns the value of the maximum time between the X and O Markers. If data is not valid, the query returns 9.9E37.

Query Syntax: :MACHine{1|2}:SLISt:TMAXimum?

Returned Format: [:MACHine{1|2}:SLISt:TMAXimum] <time_value><NL>

where:

<time_value> ::= real number

Example:

```
10 DIM Tx$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:TMAXIMUM?"
30 ENTER XXX;Tx$
40 PRINT Tx$
50 END
```

TMINimum

query

The TMINimum query returns the value of the minimum time between the X and O Markers. If data is not valid, the query returns 9.9E37.

Query Syntax: :MACHine{1|2}:SLISt:TMINimum?

Returned Format: [:MACHine{1|2}:SLISt:TMINimum] <time_value> <NL>

where:

<time_value> ::= real number

Example:

```
10 DIM Tm$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:TMINIMUM?"
30 ENTER XXX;Tm$
40 PRINT Tm$
50 END
```

VRUNs

query

The VRUNs query returns the number of valid runs and total number of runs made. Valid runs are those where the pattern search for both the X and O markers was successful resulting in valid delta time measurements.

Query Syntax: :MACHine{1|2}:SLISt:VRUNs?

Returned Format: [:MACHine{1|2}:SLISt:VRUNs] <valid_runs>,<total_runs><NL>

where:

<valid_runs> ::= zero or positive integer

<total_runs> ::= zero or positive integer

Example:

```
10 DIM Vr$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:VRUNS?"
30 ENTER XXX;Vr$
40 PRINT Vr$
50 END
```

XOTag

query

The XOTag query returns the time from the X to O markers when the marker mode is time or number of states from the X to O markers when the marker mode is state. If there is no data in the time mode the query returns 9.9E37. If there is no data in the state mode, the query returns 32767.

Query Syntax: :MACHine{1|2}:SLISt:XOTag?

Returned Format: [:MACHine{1|2}:SLISt:XOTag] {<XO_time>|<XO_states>}<NL>

where:

<XO_time> ::= real number

<XO_states> ::= integer

Example:

```
10 DIM Xot$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:XOTAG?"
30 ENTER XXX;Xot$
40 PRINT Xot$
50 END
```

XPATtern

command/query

The XPATtern command allows you to construct a pattern recognizer term for the X Marker which is then used with the XSEarch criteria when moving the marker on patterns. Since this command deals with only one label at a time, a complete specification could require several invocations.

When the value of a pattern is expressed in binary, it represents the bit values for the label inside the pattern recognizer term. In whatever base is used, the value must be between 0 and 2^{32} - 1, since a label may not have more than 32 bits. Because the <label_pattern> parameter may contain don't cares, it is handled as a string of characters rather than a number.

The XPATtern query returns the pattern specification for a given label name.

Command Syntax: :MACHine{1|2}:SLISt:XPATtern <label_name>,<label_pattern>

where:

<label_name> ::= string of up to 6 alphanumeric characters

<label_pattern> ::= "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Examples:

```
OUTPUT XXX;":MACHINE1:SLIST:XPATTERN 'DATA','255' "
OUTPUT XXX;":MACHINE1:SLIST:XPATTERN 'ABC','#BXXXX1101' "
```

XPATtern

Query Syntax: :MACHine{1|2}:SLISt:XPATtern? <label_name>

Returned Format: [:MACHine{1|2}:SLISt:XPATtern] <label_name>,<label_pattern><NL>

Example:

```
10 DIM Xp$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:XPATTERN? 'A'"
30 ENTER XXX;Xp$
40 PRINT Xp$
50 END
```

XSEarch

command/query

The XSEarch command defines the search criteria for the X Marker, which is then with associated XPATtern recognizer specification when moving the markers on patterns. The origin parameter tells the Marker to begin a search with the trigger or with the start of data. The occurrence parameter determines which occurrence of the XPATtern recognizer specification, relative to the origin, the marker actually searches for. An occurrence of 0 places a marker on the selected origin.

The XSEarch query returns the search criteria for the X marker.

Command Syntax: :MACHine{1|2}:SLISt:XSEarch <occurrence>,<origin>

where:

<occurrence> ::= integer from -1023 to +1023

<origin> ::= {TRIGger|STARt}

Example:

```
OUTPUT XXX;":MACHINE1:SLIST:XSEARCH +10,TRIGGER"
```

Query Syntax: :MACHine{1|2}:SLISt:XSEarch?

Returned Format: [:MACHine{1|2}:SLISt:XSEarch] <occurrence>,<origin><NL>

Example:

```
10 DIM Xs$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:XSEARCH?"
30 ENTER XXX;Xs$
40 PRINT Xs$
50 END
```

XSTate

query

The XSTate query returns the line number in the listing where the X marker resides (-1023 to +1023). If data is not valid, the query returns 32767.

Query Syntax: :MACHine{1|2}:SLISt:XSTate?

Returned Format: [:MACHine{1|2}:SLISt:XSTate] <state_num><NL>

where:

<state_num> ::= an integer from -1023 to +1023, or 32767

Example:

```
10 DIM Xs$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:XSTATE?"
30 ENTER XXX;Xs$
40 PRINT Xs$
50 END
```

XTAG

command/query

The XTAG command specifies the tag value on which the X Marker should be placed. The tag value is time when time tagging is on or states when state tagging is on. If the data is not valid tagged data, no action is performed.

The XTAG query returns the X Marker position in time when time tagging is on or in states when state tagging is on, regardless of whether the marker was positioned in time or through a pattern search. If data is not valid tagged data, the query returns 9.9E37 for time tagging, 32767 for state tagging.

Command Syntax: :MACHine{1|2}:SLISt:XTAG {<time_value> | <state_value>}

where:

<time_value> ::= real number

<state_value> ::= integer

Example:

```
:OUTPUT XXX;":MACHINE1:SLIST:XTAG 40.0E-6"
```

Query Syntax: :MACHine{1|2}:SLISt:XTAG?

Returned Format: [:MACHine{1|2}:SLISt:XTAG] {<time_value> | <state_value>}<NL>

Example:

```
10 DIM Xt$[100]
20 OUTPUT XXX;":MACHINE1:SLIST:XTAG?"
30 ENTER XXX;Xt$
40 PRINT Xt$
50 END
```

14 - SWAVeform Subsystem

SWAVeform Subsystem 14

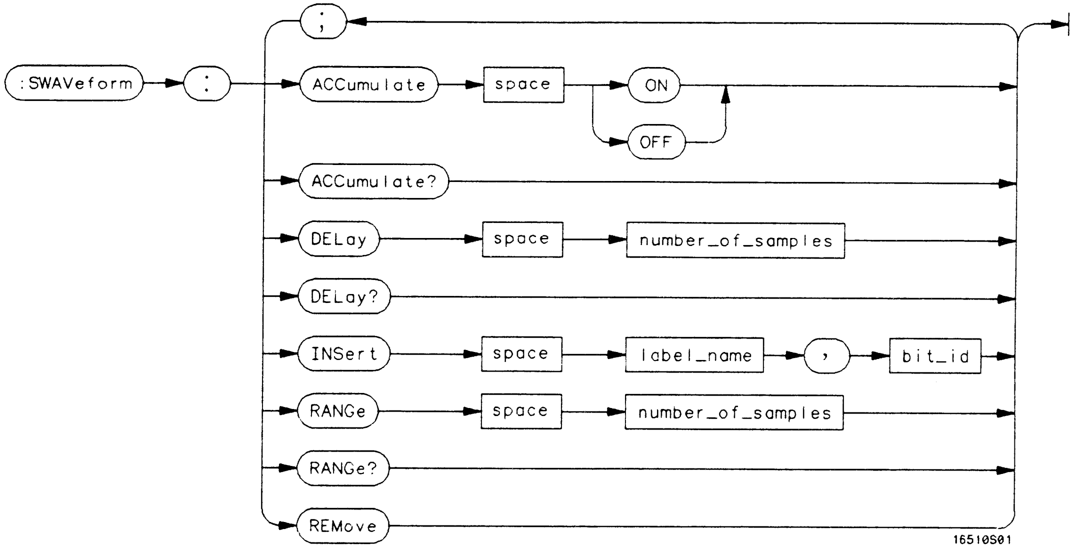
Introduction

The commands in the State Waveform subsystem allow you to configure the display so that you can view state data as waveforms on up to 24 channels identified by label name and bit number. The five commands are analogous to their counterparts in the Timing Waveform subsystem. However, in this subsystem the x-axis is restricted to representing only samples (states), regardless of whether time tagging is on or off. As a result, the only commands which can be used for scaling are DELay and RANge.

The way to manipulate the X and O markers on the Waveform display is through the State Listing (SLISt) subsystem. Using the marker commands from the SLISt subsystem will affect the markers on the Waveform display.

The commands in the SWAVeform subsystem are:

- ACCumulate
- DELay
- INSert
- RANGe
- REMove



number_of_samples = *integer from -1023 to + 1024*
label_name = *string of up to 6 alphanumeric characters*
bit_id = *{OVERlay | <bit_num>}*
bit_num = *integer representing a label bit from 0 to 31*

Figure 14-1. SWAVeform Subsystem Syntax Diagram

SWAVeform

selector

The SWAVeform (State Waveform) selector is used as part of a compound header to access the settings in the State Waveform menu. It always follows the MACHine selector because it selects a branch directly below the MACHine level in the command tree.

Command Syntax: :MACHine{1|2}:SWAVeform

Example:

```
OUTPUT XXX;":MACHINE2:SWAVEFORM:RANGE 40"
```

ACCumulate

command/query

The ACCumulate command allows you to control whether the waveform display gets erased between individual runs or whether subsequent waveforms are allowed to be displayed over the previous waveforms.

The ACCumulate query returns the current setting. The query always shows the setting as the character "0" (off) or "1" (on).

Command Syntax: :MACHine{1|2}:SWAVeform:ACCumulate {{ON | 1} | {OFF | 0}}

Example:

```
OUTPUT XXX;":MACHINE1:SWAVEFORM:ACCUMULATE ON"
```

Query Syntax: MACHine{1|2}:SWAVeform:ACCumulate?

Returned Format: [MACHine{1|2}:SWAVeform:ACCumulate] {0 | 1}<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:SWAVEFORM:ACCUMULATE?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

DELay

command/query

The DELay command allows you to specify the number of samples between the timing trigger and the horizontal center of the screen for the waveform display. The allowed number of samples is from -1023 to +1024.

The DELay query returns the current sample offset value.

Command Syntax: :MACHine{1|2}:SWAVeform:DELay <number_of_samples>

where:

<number_of_samples> ::= integer from −1023 to +1024

Example:

```
OUTPUT XXX;":MACHINE2:SWAVEFORM:DELAY 127"
```

Query Syntax: MACHine{1|2}:SWAVeform:DELay?

Returned Format: [MACHine{1|2}:SWAVeform:DELay] <number_of_samples><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:SWAVEFORM:DELAY?"
30 ENTER XXX;String$
40 PRINT String$
50 END
```

INSert

command

The INSert command allows you to add waveforms to the state waveform display. Waveforms are added from top to bottom on the screen. When 24 waveforms are present, inserting additional waveforms replaces the last waveform. Bit numbers are zero based, so a label with 8 bits is referenced as bits 0-7. Specifying OVERlay causes a composite waveform display of all bits or channels for the specified label.

Command Syntax: MACHine{1|2}:SWAVeform:INSert <label_name>,<bit_id>

where:

<label_name>	::=	string of up to 6 alphanumeric characters
<bit_id>	::=	{OVERlay|<bit_num>}
<bit_num>	::=	integer representing a label bit from 0 to 31

Examples:

```
OUTPUT XXX;":MACHINE1:SWAVEFORM:INSERT 'WAVE', 19"
OUTPUT XXX;":MACHINE1:SWAVEFORM:INSERT 'ABC', OVERLAY"
OUTPUT XXX;":MACH1:SWAV:INSERT 'POD1', #B1001"
```

RANGe

command/query

The RANGe command allows you to specify the number of samples across the screen on the State Waveform display. It is equivalent to ten times the states per division setting (st/Div) on the front panel. A number between 10 and 10240 may be entered.

The RANGe query returns the current range value.

Command Syntax: MACHine{1|2}:SWAVeform:RANGe <number_of_samples>

where:

<number_of_samples> ::= integer from 10 to 10240

Example:

```
OUTPUT XXX;":MACHINE2:SWAVEFORM:RANGE 80"
```

Query Syntax: MACHine{1|2}:SWAVeform:RANGe?

Returned Format: [MACHine{1|2}:SWAVeform:RANGe] <number_of_samples><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:SWAVEFORM:RANGE?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

REMove

command

The REMove command allows you to clear the waveform display before building a new display.

Command Syntax: :MACHine{1|2}:SWAVeform:REMove

Example:

```
OUTPUT XXX;":MACHINE1:SWAVEFORM:REMOVE"
```

15 - SCHart Subsystem

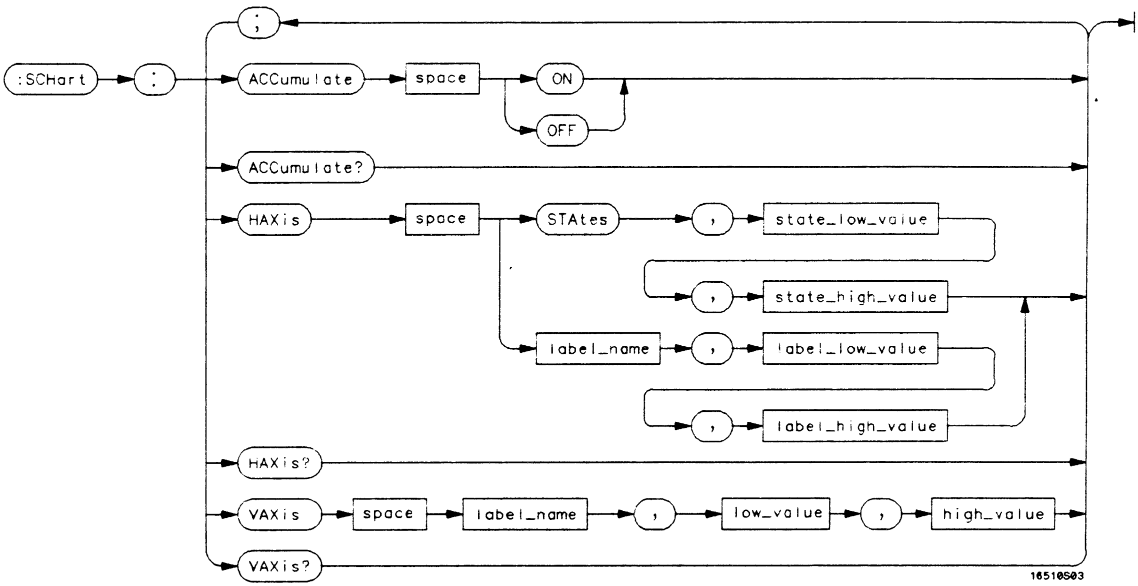
SCHart Subsystem 15

Introduction

The State Chart subsystem provides the commands necessary for programming the HP 1652B/53B's Chart display. The commands allow you to build charts of label activity, using data normally found in the Listing display. The chart's y-axis is used to show data values for the label of your choice. The x-axis can be used in two different ways. In one, the x-axis represents states (shown as rows in the State Listing display). In the other, the x-axis represents the data values for another label. When states are plotted along the x-axis, X and O markers are available. Since the State Chart display is simply an alternative way of looking at the data in the State Listing, the X and O markers can be manipulated through the SLISt subsystem. In fact, because the programming commands do not force the menus to switch, you can position the markers in the SLISt subsystem and see the effects in the State Chart display.

The commands in the SCHart subsystem are:

- ACCumulate
- HAXis
- VAXis



state_low_value = *integer from −1023 to +1024*
state_high_value = *integer from <state_low_value> to +1024*
label_name = *a string of up to 6 alphanumeric characters*
label_low_value = *string from 0 to* 2^{32} *- 1 (#HFFFFFFFF)*
label_high_value = *string from <label_low_value> to* 2^{32} *- 1 (#HFFFFFFFF)*
low_value = *string from 0 to* 2^{32} *- 1 (#HFFFFFFFF)*
high_value = *string from < low_value> to* 2^{32} *- 1 (#HFFFFFFFF)*

Figure 15-1. SCHart Subsystem Syntax Diagram

SCHart

selector

The SCHart selector is used as part of a compound header to access the settings found in the State Chart menu. It always follows the MACHine selector because it selects a branch below the MACHine level in the command tree.

Command Syntax: :MACHine{1|2}:SCHart

Example: `OUTPUT XXX;":MACHINE1:SCHART:VAXIS 'A', '0', '9'"`

ACCumulate

command/query

The ACCumulate command allows you to control whether the chart display gets erased between each individual run or whether subsequent waveforms are allowed to be displayed over the previous waveforms.

The ACCumulate query returns the current setting. The query always shows the setting as the character "0" (off) or "1" (on).

Command Syntax: MACHine{1|2}:SCHart:ACCumulate {{ON | 1} | {OFF | 0}}

Example:

```
OUTPUT XXX;":MACHINE1:SCHART:ACCUMULATE OFF"
```

Query Syntax: MACHine{1|2}:SCHart:ACCumulate?

Returned Format: [MACHine{1|2}:SCHart:ACCumulate] {0 | 1}<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:SCHART:ACCUMULATE?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

HAXis

command/query

The HAXis command allows you to select whether states or a label's values will be plotted on the horizontal axis of the chart. The axis is scaled by specifying the high and low values.

Note

The shortform for STATES is STA. This is an intentional deviation from the normal truncation rules.

The HAXis query returns the current horizontal axis label assignment and scaling.

Command Syntax: MACHine{1|2}:SCHart:HAXis {STATES,<state_low_value>,<state_high_value> | <label_name>,<label_low_value>,<label_high_value>}

where:

<state_low_value>	::= integer from -1023 to 1024
<state_high_value>	::= integer from <state_low_value> to +1024
<label_name>	::= a string of up to 6 alphanumeric characters
<label_low_value>	::= string from 0 to 2^{32}-1 (#HFFFFFFFF)
<label_high_value>	::= string from <label_low_value> to 2^{32}-1 (#HFFFFFFFF)

Examples:

```
OUTPUT XXX;":MACHINE1:SCHART:HAXIS STATES, -100, 100"
OUTPUT XXX;":MACHINE1:SCHART:HAXIS 'DATA', '100', '511'"
```

HAXis

Query Syntax: MACHine{1|2}:SCHart:HAXis?

Returned Format: [MACHine{1|2}:SCHart:HAXis] {STATES,<state_low_value>,<state_high_value> | <label_name>,<label_low_value>,<label_high_value>}

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:SCHART:HAXIS?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

VAXis

command/query

The VAXis command allows you to choose which label will be plotted on the vertical axis of the chart and scale the vertical axis by specifying the high value and low value.

The VAXis query returns the current vertical axis label assignment and scaling.

Command Syntax: MACHine{1|2}:SCHart:VAXis <label_name>,<low_value>,<high_value>

where:

<label_name> ::= a string of up to 6 alphanumeric characters
<low_value> ::= string from 0 to 2^{32}-1 (#HFFFFFFFF)
<high_value> ::= string from <low_value> to 2^{32}-1 (#HFFFFFFFF)

Examples:

```
OUTPUT XXX;":MACHINE2:SCHART:VAXIS 'SUM1', '0', '99'"
OUTPUT XXX;":MACHINE1:SCHART:VAXIS 'BUS', '#H00FF', '#H0500'"
```

Query Syntax: MACHine{1|2}:SCHart:VAXis?

Returned Format: [MACHine{1|2}:SCHart:VAXis] <label_name>,<low_value>,<high_value><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE1:SCHART:VAXIS?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

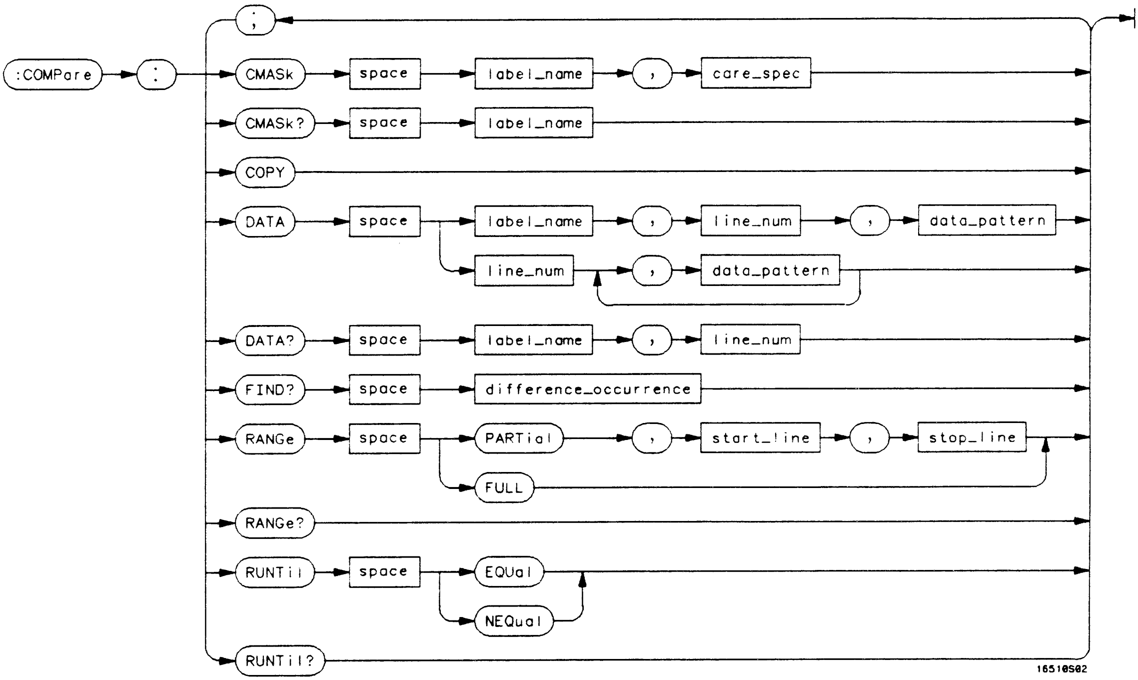
16 - COMPare Subsystem

COMPare Subsystem 16

Introduction

Commands in the state COMPare subsystem provide the ability to do a bit-by-bit comparison between the acquired state data listing and a compare data image. The commands are:

- COPY
- DATA
- CMASk
- RANGe
- RUNTil
- FIND



label_name = *string of up to 6 characters*
care_spec = *string of characters* "{*|.}..."
***** = *care*
. = *don't care*
line_num = *integer from –1023 to +1023*
data_pattern = "{#B{0|1|X}... |
#Q{0|1|2|3|4|5|6|7|X}... |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X}... |
{0|1|2|3|4|5|6|7|8|9}... }"
difference_occurence = *integer from 1 to 1024*
start_line = *integer from –1023 to +1023*
stop_line = *integer from <start_line> to +1023*

Figure 16-1. COMPare Subsystem Syntax Diagram

COMPare

selector

The COMPare selector is used as part of a compound header to access the settings found in the Compare menu. It always follows the MACHine selector because it selects a branch directly below the MACHine level in the command tree.

Command Syntax: :MACHine{1|2}:COMPare

Example:

```
OUTPUT XXX;":MACHINE1:COMPARE:FIND? 819"
```

CMASk

command/query

The CMASk (Compare Mask) command allows you to set the bits in the channel mask for a given label in the compare listing image to "compares" or "don't compares."

The CMASk query returns the state of the bits in the channel mask for a given label in the compare listing image.

Command Syntax: MACHine{1|2}:COMPare:CMASk <label_name>,<care__spec>

where:

<label_name> ::= a string of up to 6 alphanumeric characters
<care_spec> ::= string of characters "{*|.}..." (32 characters maximum)
* ::= care
. ::= don't care

Example:

```
OUTPUT XXX;":MACHINE2:COMPARE:CMASK 'STAT', '*.**..**'"
```

Query Syntax: MACHine{1|2}:COMPare:CMASk? <label_name>

Returned Format: [MACHine{1|2}:COMPare:CMASk] <label_name>,<care_spec><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:COMPARE:CMASK? 'POD5'"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

COPY

command

The COPY command copies the current acquired State Listing for the specified machine into the Compare Listing template. It does not affect the compare range or channel mask settings.

Command Syntax: MACHine{1|2}:COMPare:COPY

Example:
```
OUTPUT XXX;":MACHINE2:COMPARE:COPY"
```

DATA

command/query

The DATA command allows you to edit the compare listing image for a given label and state row. When DATA is sent to an instrument where no compare image is defined (such as at power-up) all other data in the image is set to don't cares.

Not specifying the <label_name> parameter allows you to write data patterns to more than one label for the given line number. The first pattern is placed in the left-most label, with the following patterns being placed in a left-to-right fashion (as seen on the Compare display). Specifying more patterns than there are labels simply results in the extra patterns being ignored.

Because don't cares (Xs) are allowed in the data pattern, it must always be expressed as a string. You may still use different bases, though don't cares cannot be used in a decimal number.

The DATA query returns the value of the compare listing image for a given label and state row.

Command Syntax:

```
MACHine{1|2}:COMPare:DATA {<label_name>,<line_num>,<data_pattern> |
                           <line_num>,<data_pattern>[, <data_pattern>]... }
```

where:

```
<label_name>    ::= a string of up 6 alphanumeric characters
<line_num>      ::= integer from -1023 to +1023
<data_pattern>  ::= "{#B{0|1|X} . . . |
                     #Q{0|1|2|3|4|5|6|7|X} . . . |
                     #H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
                     {0|1|2|3|4|5|6|7|8|9} . . . }"
```

Examples:

```
OUTPUT XXX;":MACHINE2:COMPARE:DATA 'CLOCK', 42, '#B011X101X'"
OUTPUT XXX;":MACHINE2:COMPARE:DATA 'OUT3', 0, '#HFF40'"
OUTPUT XXX;":MACHINE1:COMPARE:DATA 129, '#BXX00', '#B1101', '#B10XX'"
OUTPUT XXX;":MACH2:COMPARE:DATA -511, '4', '64', '16', 256', '8', '16'"
```

Query Syntax: MACHine{1|2}:COMPare:DATA? <label_name>,<line_num>

Returned Format: [MACHine{1|2}:COMPare:DATA] <label_name>,<line_num>,<data_pattern> <NL>

Example:

```
10 DIM Label$[6], Response$[80]
15 PRINT "This program shows the values for a signal's Compare listing"
20 INPUT "Enter signal label: ", Label$
25 OUTPUT XXX;":SYSTEM:HEADER OFF"    !Turn headers off (from responses)
30 OUTPUT XXX;":MACHINE2:COMPARE:RANGE?"
35 ENTER XXX; First, Last                 !Read in the range's end-points
40 PRINT "LINE #", "VALUE of "; Label$
45 FOR State = First TO Last          !Print compare value for each state
50   OUTPUT XXX;":MACH2:COMPARE:DATA? '" & Label$ & "'," & VAL$(State)
55   ENTER XXX; Response$
60   PRINT State, Response$
65   NEXT State
70 END
```

FIND query

The FIND query is used to get the line number of a specified difference occurence (first, second, third, etc) within the current compare range, as dictated by the RANGe command (see RANGe). A difference is counted for each line where at least one of the current labels has a discrepancy between its acquired state data listing and its compare data image.

Invoking the FIND query updates both the Listing and Compare displays so that the line number returned is in the center of the screen.

Query Syntax: MACHine{1|2}:COMPare:FIND? <difference_occurrence>

Returned Format: [MACHine{1|2}:COMPare:FIND] <difference_occurrence>, <line_number><NL>

where:

<difference_occurrence> ::= integer from 0 to 1024

<line_number> ::= integer from −1023 to +1023

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:COMPARE:FIND? 26"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

RANGe

command/query

The RANGe command allows you to define the boundaries for the comparison. The range entered must be a subset of the lines in the aquisition memory.

The RANGe query returns the current boundaries for the comparison.

Command Syntax: MACHine{1|2}:COMPare:RANGe {FULL | PARTial,<start_line>,<stop_line>}

where:

<start_line> ::= integer from −1023 to +1023

<stop_line> ::= integer from <start_line> to +1023

Examples:

```
OUTPUT XXX;":MACHINE2:COMPARE:RANGE PARTIAL,  -511, 512"
OUTPUT XXX;":MACHINE2:COMPARE:RANGE FULL"
```

Query Syntax: MACHine{1|2}:COMPare:RANGe?

Returned Format: [MACHine{1|2}:COMPare:RANGe] {FULL | PARTial,<start_line>, <stop_line>}<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE4:COMPARE:RANGE?"
30 ENTER XXX; String$
40 REM   See if substring "FULL" occurs in response string:
50 PRINT "Range is ";
60 IF POS(String$,"FULL") > 0 THEN PRINT "Full" ELSE PRINT "Partial"
70 END
```

RUNTil

command/query

The RUNTil (run until) command allows you to define a stop condition when the trace mode is repetitive. Specifying OFF causes the analyzer to make runs until either the display's STOP field is touched or the STOP command is issued.

There are four conditions based on the time between the X and O markers. Using this difference in the condition is effective only when time tags have been turned on (see the TAG command in the STRace subsystem). These four conditions are as follows:

- The difference is less than (LT) some value.
- The difference is greater than (GT) some value.
- The difference is inside some range (INRange).
- The difference is outside some range (OUTRange).

End points for the INRange and OUTRange should be at least 40 ns apart.

There are two conditions which are based on a comparison of the acquired state data and the compare data image. You can run until one of the following conditions is true:

- Compare equal (EQUal) - Every channel of every label has the same value.
- Compare not equal (NEQual) - Any channel of any label has a different value .

The RUNTil query returns the current stop criteria for the comparison when running in repetitive trace mode.

Note The RUNTil instruction (for state analysis) is available in both the SLISt and COMPare subsystems.

Command Syntax: MACHine{1|2}:COMPare:RUNTil {OFF|LT,<value>|GT,<value>|
INRange,<value>,<value>|OUTRange,<value>,<value>|EQUal|NEQual}

Example:

```
OUTPUT XXX;":MACHINE2:COMPARE:RUNTIL EQUAL"
```

Query Syntax: MACHine{1|2}:COMPare:RUNTil?

Returned Format: [MACHine{1|2}:COMPare:RUNTil] {OFF|LT,<value>|GT,<value>|
INRange,<value>,<value>|OUTRange,<value>,<value>|EQUal|NEQual}<NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:COMPARE:RUNTIL?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

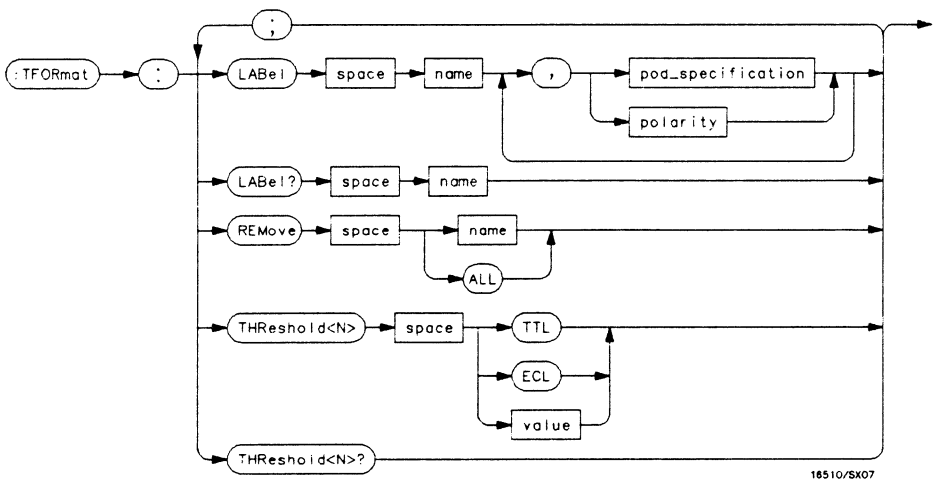
17 - TFORmat Subsystem

TFORmat Subsystem 17

Introduction

The TFORmat subsystem contains the commands available for the Timing Format menu in the HP 1652B/53B logic analyzer. These commands are:

- LABel
- REMove
- THReshold



<N> = *{1 | 2 | 3 | 4 | 5}*
name = *string of up to 6 alphanumeric characters*
polarity = *{POSitive | NEGative}*
pod_specification = *format (integer from 0 to 65535) for a pod (pods are assigned in decreasing order)*
value = *voltage (real number) -9.9 to +9.9*

Figure 17-1. TFORmat Subsystem Syntax Diagram

TFORmat

selector

The TFORmat selector is used as part of a compound header to access those settings normally found in the Timing Format menu. It always follows the MACHine selector because it selects a branch directly below the MACHine level in the language tree.

Command Syntax: :MACHine{1|2}:TFORmat

Example:

```
OUTPUT XXX;":MACHINE1:TFORMAT:LABEL?"
```

LABel

command/query

The LABel command allows you to specify polarity and assign channels to new or existing labels. If the specified label name does not match an existing label name, a new label will be created.

The order of the pod-specification parameters is significant. The first one listed will match the highest-numbered pod assigned to the machine you're using. Each pod specification after that is assigned to the next-highest-numbered pod. This way they match the left-to-right descending order of the pods you see on the Format display. Not including enough pod specifications results in the lowest-numbered pod(s) being assigned a value of zero (all channels excluded). If you include more pod specifications than there are pods for that machine, the extra ones will be ignored. However, an error is reported anytime more than five pod specifications are listed.

The polarity can be specified at any point after the label name.

Since pods contain 16 channels, the format value for a pod must be between 0 and 65535 (2^{16}-1). When giving the pod assignment in binary (base 2), each bit will correspond to a single channel. A "1" in a bit position means the associated channel in that pod is assigned to that pod and bit. A "0" in a bit position means the associated channel in that pod is excluded from the label. For example, assigning #B1111001100 is equivalent to entering "......****..**.." through the front-panel user interface.

A label can not have a total of more than 32 channels assigned to it.

The LABel query returns the current specification for the selected (by name) label. If the label does not exist, nothing is returned. Numbers are always returned in decimal format.

LABel

Command Syntax: :MACHine{1|2}:TFORmat:LABel <name>[, {<polarity> | <assignment>}]...

where:

<name> ::= string of up to 6 alphanumeric characters
<polarity> ::= {POSitive | NEGative}
<assignment> ::= format (integer from 0 to 65535) for a pod (pods are assigned in decreasing order)

Examples:

```
OUTPUT XXX;":MACHINE2:TFORMAT:LABEL 'DATA', POS, 65535, 127, 40312"
OUTPUT XXX;":MACHINE2:TFORMAT:LABEL 'STAT', 1, 8096, POSITIVE"
OUTPUT XXX;":MACHINE1:TFORMAT:LABEL 'ADDR', NEGATIVE, #B11110010101010"
```

Query Syntax: :MACHine{1|2}:TFORmat:LABel? <name>

Returned Format: [:MACHine{1|2}:TFORmat:LABel] <name>[,<assignment>]...,<polarity><NL>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":MACHINE2:TFORMAT:LABEL? 'DATA'"
30 ENTER XXX String$
40 PRINT String$
50 END
```

REMove

command

The REMove command allows you to delete all labels or any one label specified by name for a given machine.

Command Syntax: :MACHine{1|2}:TFORmat:REMove {<name>|ALL}

where:

<name> ::= string of up to 6 alphanumeric characters

Examples:

```
OUTPUT XXX;":MACHINE1:TFORMAT:REMOVE 'A'"
OUTPUT XXX;":MACHINE1:TFORMAT:REMOVE ALL"
```

THReshold

command/query

The THReshold command allows you to set the voltage threshold for a given pod to ECL, TTL or a specific voltage from -9.9V to +9.9V in 0.1 volt increments.

Note On the HP 1652B, the pod thresholds of pods 1, 2, and 3 can be set independently. The pod thresholds of pods 4 and 5 are slaved together; therefore, when you set the threshold on pod 4 or 5, both thresholds will be changed to the specified value. On the HP 1653B, both pods 1 and 2 can be set independently.

The THReshold query returns the current threshold for a given pod.

Command Syntax: :MACHine{1|2}:TFORmat:THReshold<N> {TTL|ECL|<value>}

where:

<N>	::= pod number {1|2|3|4|5}
<value>	::= voltage (real number) -9.9 to +9.9
TTL	::= default value of +1.6V
ECL	::= default value of -1.3V

Example: OUTPUT XXX;":MACHINE1:TFORMAT:THRESHOLD1 4.0"

Query Syntax: :MACHine{1|2}:TFORmat:THReshold<N>?

Returned Format: [:MACHine{1|2}:TFORmat:THReshold<N>] <value><NL>

Example:

```
10 DIM Value$ [100]
20 OUTPUT XXX;":MACHINE1:TFORMAT:THRESHOLD2?"
30 ENTER XXX;Value$
40 PRINT Value$
50 END
```

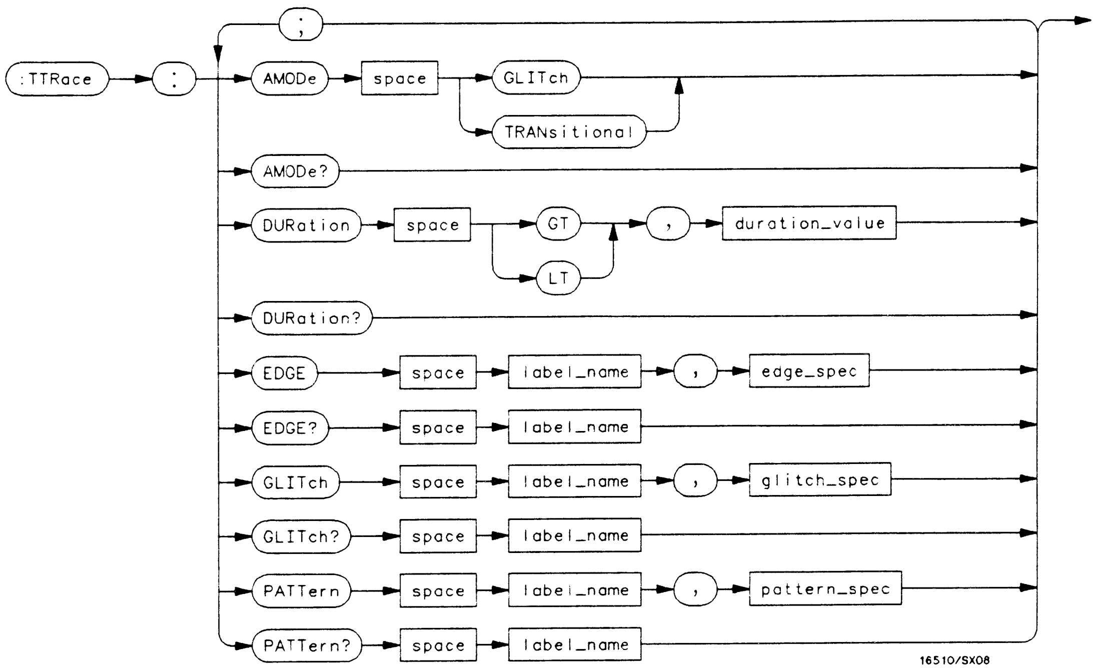
18 - TTRace Subsystem

TTRace Subsystem 18

Introduction

The TTRace subsystem contains the commands available for the Timing Trace menu in the HP 1652B/53B logic analyzer. These commands are:

- AMODe
- DURation
- EDGE
- GLITch
- PATTern



GT = *greater than*
LT = *less than*
duration_value = *real number*
label_name = *string of up to 6 alphanumeric characters*
edge_spec = *string of characters* "{R|F|T|X}..."
R = *rising edge*
F = *falling edge*
T = *toggling or either edge*
X = *don't care or ignore this channel*
glitch_spec = *string of characters* "{*|.}..."
***** = *search for a glitch on this channel*
. = *ignore this channel*
pattern_spec = "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Figure 18-1. TTRace Subsystem Syntax Diagram

TTRace

selector

The TTRace selector is used as part of a compound header to access the settings found in the Timing Trace menu. It always follows the MACHine selector because it selects a branch directly below the MACHine level in the language tree.

Command Syntax: :MACHine{1|2}:TTRace

Example:
```
OUTPUT XXX;":MACHINE1:TTRACE:GLITCH 'ABC', '....****'"
```

AMODe

command/query

The AMODe command allows you to select the acquisition mode used for a particular timing trace. The acquisition modes available are TRANsitional and GLITch.

The AMODe query returns the current acquisition mode.

Command Syntax: :MACHine{1|2}:TTRace:AMODe <acquisition_mode>

where:

<acquisition_mode> ::= {GLITch|TRANsitional}

Example:

```
OUTPUT XXX; ":MACHINE1:TTRACE:AMODE GLITCH"
```

Query Syntax: :MACHine1:TTRace:AMODe?

Returned Format: [:MACHine1:TTRace:AMODe] {GLITCH|TRANSITIONAL}

Example:

```
10 DIM M$[100]
20 OUTPUT XXX; ":MACHINE1:TTRACE:AMODE?"
30 ENTER XXX;M$
40 PRINT M$
50 END
```

DURation

command/query

The DURation command allows you to specify the duration qualifier to be used with the pattern recognizer term in generating the timing trigger. The duration value can be specified in 10 ns increments within the following ranges:

- Greater than (GT) qualification - 30 ns to 10 ms
- Less than (LT) qualification - 40 ns to 10 ms.

The DURation query returns the current pattern duration qualifier specification.

Command Syntax: :MACHine{1|2}:TTRace:DURation {GT|LT},<duration_value>

where:

GT	::=	greater than
LT	::=	less than
<duration_value>	::=	real number

Example:

```
OUTPUT XXX; ":MACHINE1:TTRACE:DURATION GT, 40.0E-9"
```

Query Syntax: :MACHine{1|2}:TTRace:DURation?

Returned Format: [:MACHine{1|2}:TTRace:DURation] {GT|LT},<duration_value><NL>

Example:

```
10 DIM D$[100]
20 OUTPUT XXX; ":MACHINE1:TTRACE:DURATION?"
30 ENTER XXX;D$
40 PRINT D$
50 END
```

EDGE

command/query

The EDGE command allows you to specify the edge recognizer term for the timing analyzer trigger on a per label basis. Each command deals with only one label in the given edge specification; therefore, a complete specification could require several commands. The edge specification uses the characters R, F, T, X to indicate the edges or don't cares as follows:

R = rising edge
F = falling edge
T = toggling or either edge
X = don't care or ignore the channel

The position of these characters in the string corresponds with the position of the channels within the label. All channels without "X" are ORed together to form the edge trigger specification.

The EDGE query returns the edge specification for the specified label.

Command Syntax: :MACHine{1|2}:TTRace:EDGE <label_name>,<edge_spec>

where:

<label_name> ::= string or up to 6 alphanumeric characters
<edge_spec> ::= string of characters "{R|F|T|X}..."

Example:

```
OUTPUT XXX; ":MACHINE1:TTRACE:EDGE 'POD1','XXXXXXXR'"
```

Query Syntax: :MACHine{1|2}:TTRace:EDGE? <label_name>

Returned Format: [:MACHine{1|2}:TTRace:] <label_name>,<edge_spec><NL>

Example:

```
10 DIM E$[100]
20 OUTPUT XXX; ":MACHINE1:TTRACE:EDGE? 'POD1'"
30 ENTER XXX;E$
40 PRINT E$
50 END
```

GLITch

command/query

The GLITch command allows you to specify the glitch recognizer term for the timing analyzer trigger on a per label basis. Each command deals with only one label in a given glitch specification, and, therefore a complete specification could require several commands. The glitch specification uses the characters "*" and "." as follows:

"*" (asterisk) = search for a glitch on this channel

"." (period) = ignore this channel

The position of these characters in the string corresponds with the position of the channels within the label. All channels with the "*" are ORed together to form the glitch trigger specification.

The GLITch query returns the glitch specification for the specified label.

Command Syntax: :MACHine{1|2}:TTRace:GLITch <label_name>,<glitch_spec>

where:

<label_name> ::= string of up to 6 alphanumeric characters

<glitch_spec> ::= string of characters "{*|.}..."

Example:

```
OUTPUT XXX; ":MACHINE1:TTRACE:GLITCH 'POD1','**.......*'"
```

Query Syntax: :MACHine1:TTRace:GLITch? <label_name>

Returned Format: [:MACHine1:TTRace:GLITch] <label_name>,<glitch_spec><NL>

Example:

```
10 DIM G$[100]
20 OUTPUT XXX; ":MACHINE1:TTRACE:GLITCH? 'POD1'"
30 ENTER XXX;G$
40 PRINT G$
50 END
```

PATTern

command/query

The PATTern command allows you to construct a pattern recognizer term for the timing analyzer trigger on a per label basis. Each command deals with only one label in the given pattern; therefore, a complete timing trace specification could require several commands. Since a label can contain up to 32 bits, the range of the pattern value will be between 0 and (2^{32})-1. The value may be expressed in binary (#B), octal (#Q), hexadecimal (#H) or decimal (default). When the value of a pattern is expressed in binary, it represents the bit values for the label inside the pattern recognizer term. Since a pattern value can contain don't cares, the pattern specification parameter is handled as a string of characters instead of a number.

The PATTern query returns the pattern specification for the specified label in the base previously defined for the label.

Command Syntax: :MACHine{1|2}:TTRace:PATTern <label_name>,<pattern_spec>

where:

<label_name> ::= string of up to 6 alphanumeric characters

<pattern_spec> ::= "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Example:

```
OUTPUT XXX; ":MACHINE1:TTRACE:PATTERN 'DATA', '255'"
```

PATTern

Query Syntax: :MACHine{1|2}:TTRace:PATTern? <label_name>

Returned Format: [:MACHine{1|2}:TTRace:PATTern] <label_name>,<pattern_spec><NL>

Example:

```
10 DIM P$[100]
20 OUTPUT XXX; ":MACHINE2:TTRACE:PATTERN? 'DATA'"
30 ENTER XXX;P$
40 PRINT P$
50 END
```

19 - TWAVeform Subsystem

TWAVeform Subsystem 19

Introduction

The TWAVeform subsystem contains the commands available for the Timing Waveforms menu in the HP 1652B/53B. These commands are:

- ACCumulate
- DELay
- INSert
- MMODe
- OCONdition
- OPATtern
- OSEarch
- OTIMe
- RANGe
- REMove
- RUNTil
- SPERiod
- TAVerage
- TMAXimum
- TMINimum
- VRUNs
- XCONdition
- XOTime
- XPATtern
- XSEarch
- XTIMe

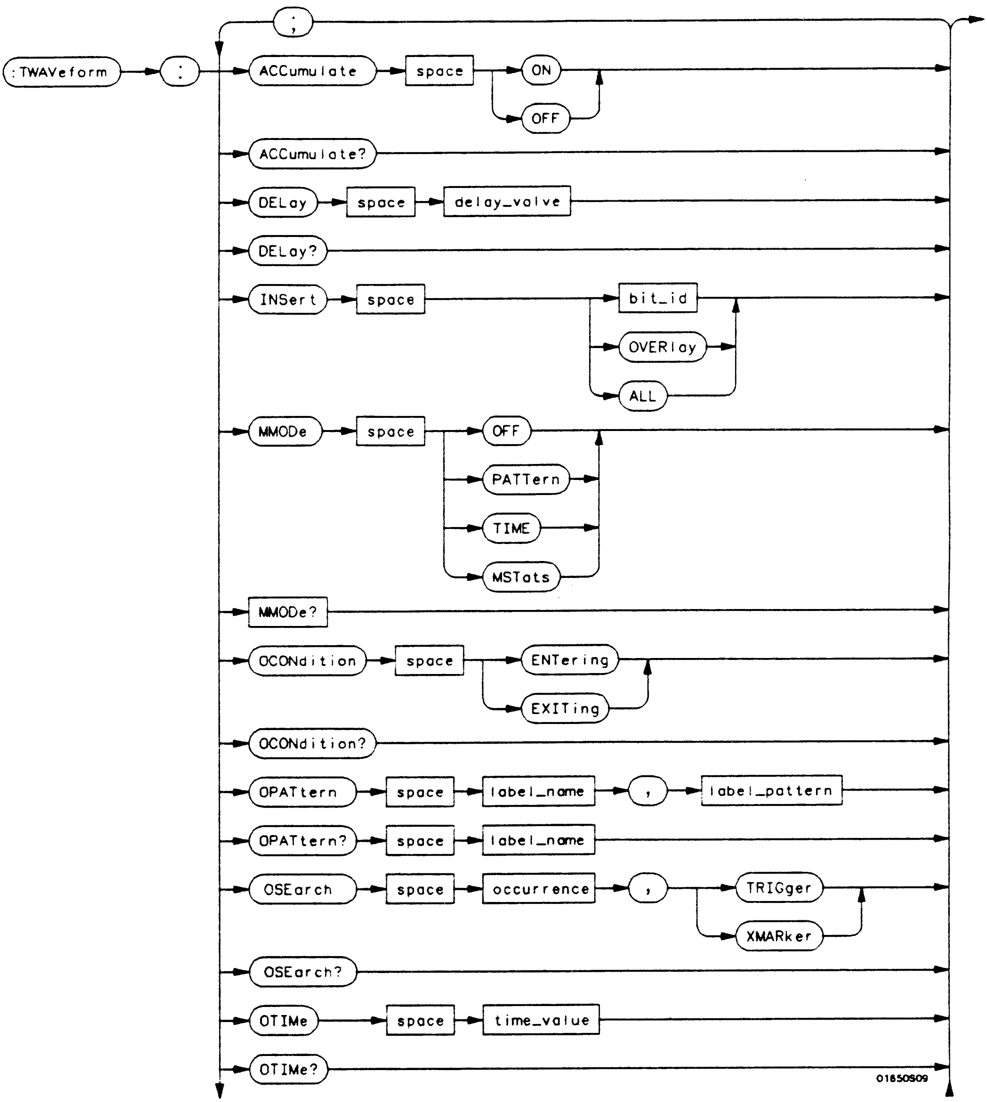


Figure 19-1. TWAVeform Subsystem Syntax Diagram

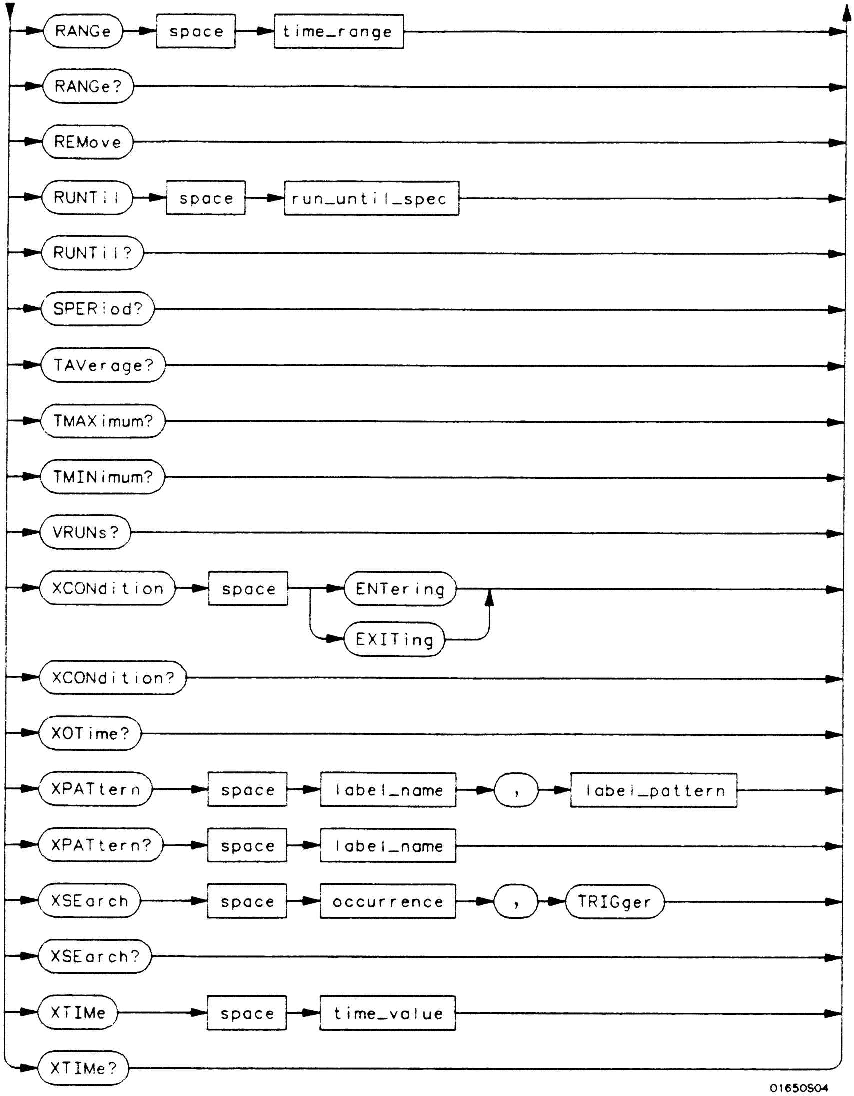


Figure 19-1. TWAVeform Subsystem Syntax Diagram (continued)

delay_value = *real number between -2500 s and +2500 s*
module_spec = {1|2|3|4|5}
bit_id = *integer from 0 to 31*
waveform = *string containing <acquisition_spec>* {1|2}
acquisition_spec = {A|B|C|D|E} *(slot where acquisition card is located)*
label_name = *string of up to 6 alphanumeric characters*
label_pattern = "{#B{0|1|X} . . . |
    #Q{0|1|2|3|4|5|6|7|X} . . . |
    #H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
    {0|1|2|3|4|5|6|7|8|9} . . . }"
occurrence = *integer*
time_value = *real number*
label_id = *string of one alpha and one numeric character*
module_num = *slot number in which the timebase card is installed*
time_range = *real number between 100 ns and 10 ks*
run_until_spec = {OFF|LT, <value>|GT, <value>|INRange<value>, <value>|
    OUTRange<value>, <value>}
GT = *greater than*
LT = *less than*
value = *real number*

Figure 19-1. TWAVeform Subsystem Syntax Diagram (continued)

TWAVeform

Selector

The TWAVeform selector is used as part of a compound header to access the settings found in the Timing Waveforms menu. It always follows the MACHine selector because it selects a branch below the MACHine level in the command tree.

Command Syntax: :MACHine{1|2}:TWAVeform

Example: `OUTPUT XXX;":MACHINE1:TWAVEFORM:DELAY 100E-9"`

ACCumulate

command/query

The ACCumulate command allows you to control whether the chart display gets erased between each individual run or whether subsequent waveforms are allowed to be displayed over the previous ones.

The ACCumulate query returns the current setting. The query always shows the setting as the character "0" (off) or "1" (on).

Command Syntax: :MACHine{1|2}:TWAVeform:ACCumulate <setting>

where:

<setting> ::= {0|OFF} or {1|ON}

Example:

```
OUTPUT XXX;":MACHINE1:TWAVEFORM:ACCUMULATE ON"
```

Query Syntax: :MACHine{1|2}:TWAVeform:ACCumulate?

Returned Format: [:MACHine{1|2}:TWAVeform:ACCumulate] {0|1}<NL>

Example:

```
10 DIM P$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:ACCUMULATE?"
30 ENTER XXX; P$
40 PRINT P$
50 END
```

DELay

command/query

The DELay command specifies the amount of time between the timing trigger and the horizontal center of the the timing waveform display. The allowable values for delay are −2500 s to +2500 s. In glitch acquisition mode, as delay becomes large in an absolute sense, the sample rate is adjusted so that data will be acquired in the time window of interest. In transitional acquisition mode, data may not fall in the time window since the sample period is fixed at 10 ns and the amount of time covered in memory is dependent on how frequent the input signal transitions occur.

The DELay query returns the current time offset (delay) value from the trigger.

Command Syntax: :MACHine{1|2}:TWAVeform:DELay <delay_value>

where:

<delay_value> ::= real number between -2500 s and +2500 s

Example:

```
OUTPUT XXX;":MACHINE1:TWAVEFORM:DELAY 100E-6"
```

Query Syntax: :MACHine{1|2}:TWAVeform:DELay?

Returned Format: [:MACHine{1|2}:TWAVeform:DELay] <time_value><NL>

Example:

```
10 DIM D1$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:DELAY?"
30 ENTER XXX; D1$
40 PRINT D1$
50 END
```

INSert

command

The INSert command inserts waveforms in the timing waveform display. The waveforms are added from top to bottom. When 24 waveforms are present, inserting additional waveforms replaces the last waveform .

The first parameter specifies the label name that will be inserted. The second parameter specifies the label bit number or overlay.

If OVERLAY is specified, all the bits of the label are displayed as a composite overlaid waveform.

Command Syntax: :MACHine{1|2}:TWAVeform:INSert <label_name> {<bit_id>|OVERlay}

where:

<label_name> ::= string of up to 6 alphanumeric characters
<bit_id> ::= integer from 0 to 31

Example:

```
OUTPUT XXX;":MACHINE1:TWAVEFORM:INSERT 'WAVE',10"
```

MMODe

command/query

The MMODe (Marker Mode) command selects the mode controlling marker movement and the display of the marker readouts. When PATTern is selected, the markers will be placed on patterns. When TIME is selected, the markers move on time. In MSTats, the markers are placed on patterns, but the readouts will be time statistics.

The MMODe query returns the current marker mode.

Command Syntax: :MACHine{1|2}:TWAVeform:MMODe {OFF|PATTern|TIME|MSTats}

Example:

```
OUTPUT XXX; ":MACHINE1:TWAVEFORM:MMODE TIME"
```

Query Syntax: :MACHine{1|2}:TWAVeform:MMODe?

Returned Format: [:MACHine{1|2}:TWAVeform:MMODe] <marker_mode> <NL>

where:

<marker_mode> ::= {OFF|PATTern|TIME|MSTats}

Example:

```
10 DIM M$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:MMODE?"
30 ENTER XXX; M$
40 PRINT M$
50 END
```

OCONdition

command/query

The OCONdition command specifies where the O marker is placed. The O marker can be placed on the entry or exit point of the OPATtern when in the PATTern marker mode.

The OCONdition query returns the current setting.

Command Syntax: :MACHine{1|2}:TWAVeform:OCONdition {ENTering|EXITing}

Example:
```
OUTPUT XXX; ":MACHINE1:TWAVEFORM:OCONDITION ENTERING"
```

Query Syntax: :MACHine{1|2}:TWAVeform:OCONdition?

Returned Format: [:MACHine{1|2}:TWAVeform:OCONdition] {ENTering|EXITing}<NL>

Example:
```
10 DIM Oc$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:OCONDITION?"
30 ENTER XXX; Oc$
40 PRINT Oc$
50 END
```

OPATtern

command/query

The OPATtern command allows you to construct a pattern recognizer term for the O marker which is then used with the OSEarch criteria and OCONdition when moving the marker on patterns. Since this command deals with only one label at a time, a complete specification could require several invocations.

When the value of a pattern is expressed in binary, it represents the bit values for the label inside the pattern recognizer term. In whatever base is used, the value must be between 0 and 2^{32} - 1, since a label may not have more than 32 bits. Because the <label_pattern> parameter may contain don't cares, it is handled as a string of characters rather than a number.

The OPATtern query, in pattern marker mode, returns the pattern specification for a given label name. In the time marker mode, the query returns the pattern under the O marker for a given label. If the O marker is not placed on valid data, don't cares (XX...X) are returned.

Command Syntax: :MACHine{1|2}:TWAVeform:OPATtern <label_name>,<label_pattern>

where:

<label_name> ::= string of up to 6 alphanumeric characters

<label_pattern> ::= "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Example:

```
OUTPUT XXX; ":MACHINE1:TWAVEFORM:OPATTERN 'A','511'"
```

OPATtern

Query Syntax: :MACHine{1|2}:TWAVeform:OPATtern? <label_name>

Returned Format: [:MACHine{1|2}:TWAVeform:OPATtern] <label_name>,<label_pattern><NL>

Example:

```
10 DIM Op$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:OPATTERN? 'A'"
30 ENTER XXX; Op$
40 PRINT Op$
50 END
```

OSEarch

command/query

The OSEarch command defines the search criteria for the O marker which is then used with the associated OPATtern recognizer specification and the OCONdition when moving markers on patterns. The origin parameter tells the marker to begin a search with the trigger or with the X marker. The actual occurrence the marker searches for is determined by the occurrence parameter of the OPATtern recognizer specification, relative to the origin. An occurrence of 0 places a marker on the selected origin. With a negative occurrence, the marker searches before the origin. With a positive occurrence, the marker searches after the origin.

The OSEarch query returns the search criteria for the O marker.

Command Syntax: :MACHine{1|2}:TWAVeform:OSEarch <occurrence>,<origin>

where:

<origin> ::= {TRIGger|XMARker}

<occurrence> ::= integer from -9999 to +9999

Example:

```
OUTPUT XXX; ":MACHINE1:TWAVEFORM:OSEARCH +10,TRIGGER"
```

Query Syntax: :MACHine{1|2}:TWAVeform:OSEarch?

Returned Format: [:MACHine{1|2}:TWAVeform:OSEarch] <occurrence>,<origin><NL>

Example:

```
10 DIM Os$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:OSEARCH?"
30 ENTER XXX; Os$
40 PRINT Os$
50 END
```

OTIMe

command/query

The OTIMe command positions the O marker in time when the marker mode is TIME. If data is not valid, the command performs no action.

The OTIMe query returns the O marker position in time. If data is not valid, the query returns 9.9E37.

Command Syntax: :MACHine{1|2}:TWAVeform:OTIMe <time_value>

where:

<time_value> ::= real number -2.5Ks to +2.5Ks

Example:

```
OUTPUT XXX; ":MACHINE1:TWAVEFORM:OTIME 30.0E-6"
```

Query Syntax: :MACHine{1|2}:TWAVeform:OTIMe?

Returned Format: [:MACHine{1|2}:TWAVeform:OTIMe] <time_value><NL>

Example:

```
10 DIM Ot$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:OTIME?"
30 ENTER XXX; Ot$
40 PRINT Ot$
50 END
```

RANGe

command/query

The RANGe command specifies the full-screen time in the timing waveform menu. It is equivalent to ten times the seconds-per-division setting on the display. The allowable values for RANGe are from 100 ns to 10 ks.

The RANGe query returns the current full-screen time.

Command Syntax: :MACHine{1|2}:TWAVeform:RANGe <time_value>

where:

<time_range> ::= real number between 100 ns and 10 ks

Example:

```
OUTPUT XXX;":MACHINE1:TWAVEFORM:RANGE 100E-9"
```

Query Syntax: :MACHine{1|2}:TWAVeform:RANGe?

Returned Format: [:MACHine{1|2}:TWAVeform:RANGe] <time_value><NL>

Example:

```
10 DIM Rg$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:RANGE?"
30 ENTER XXX; Rg$
40 PRINT Rg$
50 END
```

REMove

command

The REMove command deletes all waveforms from the display.

Command Syntax: :MACHine{1|2}:TWAVeform:REMove

Example:
```
OUTPUT XXX;":MACHINE1:TWAVEFORM:REMOVE"
```

RUNTil

command/query

The RUNTil (run until) command defines stop criteria based on the time between the X and O markers when the trace mode is in repetitive. When OFF is selected, the analyzer will run until either the "STOP" touch screen field is touched or the STOP command is sent. Run until the time between X and O marker options are:

- Less Than (LT) a specified time value
- Greater Than (GT) a specified time value
- In the range (INRange) between two time values
- Out of the range (OUTRange) between two time values

End points for the INRange and OUTRange should be at least 10 ns apart since this is the minimum time at which data is sampled.

This command affects the timing analyzer only, and has no relation to the RUNTil commands in the SLISt and COMPare subsystems.

The RUNTil query returns the current stop criteria.

Command Syntax: :MACHine{1|2}:TWAVeform:RUNTil <run_until_spec>

where:

<run_until_spec> ::= {OFF | LT,<value> | GT,<value> | INRange<value>,<value> | OUTRange<value>,<value>}

<value> ::= real number

Examples:

```
OUTPUT XXX;":MACHINE1:TWAVEFORM:RUNTIL GT, 800.0E-6"
OUTPUT XXX;":MACHINE1:TWAVEFORM:RUNTIL INRANGE, 4.5, 5.5"
```

RUNTil

Query Syntax: :MACHine{1|2}:TWAVeform:RUNTil?

Returned Format: [:MACHine{1|2}:TWAVeform:RUNTil] <run_until_spec> <NL>

Example:

```
10 DIM Ru$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:RUNTIL?"
30 ENTER XXX; Ru$
40 PRINT Ru$
50 END
```

SPERiod

query

The SPERiod query returns the sample period of the last run.

Query Syntax: :MACHine{1|2}:TWAVeform:SPERiod?

Returned Format: [:MACHine{1|2}:TWAVeform:SPERiod] <time_value><NL>

where:

<time_value> ::= real number

Example:

```
10 DIM Sp$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:SPERIOD?"
30 ENTER XXX; Sp$
40 PRINT Sp$
50 END
```

TAVerage query

The TAVerage query returns the value of the average time between the X and O markers. If there is no valid data, the query returns 9.9E37.

Query Syntax: :MACHine{1|2}:TWAVeform:TAVerage?

Returned Format: [:MACHine{1|2}:TWAVeform:TAVerage] <time_value><NL>

where:

<time_value> ::= real number

Example:

```
10 DIM Tv$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:TAVERAGE?"
30 ENTER XXX; Tv$
40 PRINT Tv$
50 END
```

TMAXimum

query

The TMAXimum query returns the value of the maximum time between the X and O markers. If there is no valid data, the query returns 9.9E37.

Query Syntax: :MACHine{1|2}:TWAVeform:TMAXimum?

Returned Format: [:MACHine{1|2}:TWAVeform:TMAXimum] <time_value><NL>

where

<time_value> ::= real number

Example:

```
10 DIM Tx$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:TMAXIMUM?"
30 ENTER XXX; Tx$
40 PRINT Tx$
50 END
```

TMINimum query

The TMINimum query returns the value of the minimum time between the X and O markers. If there is no valid data, the query returns 9.9E37.

Query Syntax: :MACHine{1|2}:TWAVeform:TMINimum?

Returned Format: [:MACHine{1|2}:TWAVeform:TMINimum] <time_value><NL>

where:

<time_value> ::= real number

Example:

```
10 DIM Tm$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:TMINIMUM?"
30 ENTER XXX; Tm$
40 PRINT Tm$
50 END
```

VRUNs

query

The VRUNs query returns the number of valid runs and total number of runs made. Valid runs are those where the pattern search for both the X and O markers was successful resulting in valid delta time measurements.

Query Syntax: :MACHine{1|2}:TWAVeform:VRUNs?

Returned Format: [:MACHine{1|2}:TWAVeform:VRUNs] <valid_runs>,<total_runs><NL>

where:

<valid_runs> ::= zero or positive integer
<total_runs> ::= zero or positive integer

Example:

```
10 DIM Vr$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:VRUNS?"
30 ENTER XXX; Vr$
40 PRINT Vr$
50 END
```

XCONdition

command/query

The XCONdition command specifies where the X marker is placed. The X marker can be placed on the entry or exit point of the XPATtern when in the PATTern marker mode.

The XCONdition query returns the current setting.

Command Syntax: :MACHine{1|2}:TWAVeform:XCONdition {ENTering|EXITing}

Example: `OUTPUT XXX; ":MACHINE1:TWAVEFORM:XCONDITION ENTERING"`

Query Syntax: :MACHine{1|2}:TWAVeform:XCONdition?

Returned Format: [:MACHine{1|2}:TWAVeform:XCONdition] {ENTering|EXITing}<NL>

Example:

```
10 DIM Xc$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:XCONDITION?"
30 ENTER XXX; Xc$
40 PRINT Xc$
50 END
```

XOTime

query

The XOTime query returns the time from the X marker to the O marker. If data is not valid, the query returns 9.9E37.

Query Syntax: :MACHine{1|2}:TWAVeform:XOTime?

Returned Format: [:MACHine{1|2}:TWAVeform:XOTime] <time_value> <NL>

where:

<time_value> ::= real number

Example:

```
10 DIM Xot$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:XOTIME?"
30 ENTER XXX; Xot$
40 PRINT Xot$
50 END
```

XPATtern

command/query

The XPATtern command allows you to construct a pattern recognizer term for the X marker which is then used with the XSEarch criteria and XCONdition when moving the marker on patterns. Since this command deals with only one label at a time, a complete specification could require several invocations.

When the value of a pattern is expressed in binary, it represents the bit values for the label inside the pattern recognizer term. In whatever base is used, the value must be between 0 and 2^{32} - 1, since a label may not have more than 32 bits. Because the <label_pattern> parameter may contain don't cares, it is handled as a string of characters rather than a number.

The XPATtern query, in pattern marker mode, returns the pattern specification for a given label name. In the time marker mode, the query returns the pattern under the X marker for a given label. If the X marker is not placed on valid data, don't cares (XX...X) are returned.

Command Syntax: :MACHine{1|2}:TWAVeform:XPATtern <label_name>,<label_pattern>

where:

<label_name> ::= string of up to 6 alphanumeric characters

<label_pattern> ::= "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Example:

```
OUTPUT XXX; ":MACHINE1:TWAVEFORM:XPATTERN 'A','511'"
```

Query Syntax: :MACHine{1|2}:TWAVeform:XPATtern? <label_name>

Returned Format: [:MACHine{1|2}:TWAVeform:XPATtern] <label_name>,<label_pattern><NL>

Example:

```
10 DIM Xp$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:XPATTERN? 'A'"
30 ENTER XXX; Xp$
40 PRINT Xp$
50 END
```

XSEarch

command/query

The XSEarch command defines the search criteria for the X marker which is then used with the associated XPATtern recognizer specification and the XCONdition when moving markers on patterns. The origin parameter tells the marker to begin a search with the trigger. The occurrence parameter determines which occurrence of the XPATtern recognizer specification, relative to the origin, the marker actually searches for. An occurrence of 0 (zero) places a marker on the origin.

The XSEarch query returns the search criteria for the X marker.

Command Syntax: :MACHine{1|2}:TWAVeform:XSEarch <occurrence>,<origin>

where:

<origin> ::= TRIGger
<occurrence> ::= integer from -9999 to +9999

Example:

```
OUTPUT XXX; ":MACHINE1:TWAVEFORM:XSEARCH,+10,TRIGGER"
```

Query Syntax: :MACHine{1|2}:TWAVeform:XSEarch? <occurrence>,<origin>

Returned Format: [:MACHine{1|2}:TWAVeform:XSEarch] <occurrence>,<origin><NL>

Example:

```
10 DIM Xs$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:XSEARCH?"
30 ENTER XXX; Xs$
40 PRINT Xs$
50 END
```

XTIMe

command/query

The XTIMe command positions the X marker in time when the marker mode is TIME. If data is not valid, the command performs no action.

The XTIMe query returns the X marker position in time. If data is not valid, the query returns 9.9E37.

Command Syntax: :MACHine{1|2}:TWAVeform:XTIMe <time_value>

where:

<time_value> ::= real number from −2.5Ks to +2.5Ks

Example: OUTPUT XXX; ":MACHINE1:TWAVEFORM:XTIME 40.0E-6"

Query Syntax: :MACHine{1|2}:TWAVeform:XTIMe?

Returned Format: [:MACHine{1|2}:TWAVeform:XTIMe] <time_value><NL>

Example:

```
10 DIM Xt$ [100]
20 OUTPUT XXX;":MACHINE1:TWAVEFORM:XTIME?"
30 ENTER XXX; Xt$
40 PRINT Xt$
50 END
```

20 - SYMBol Subsystem

SYMBol Subsystem 20

Introduction

The SYMBol subsystem contains the commands that allow you to define symbols on the controller and download them to the HP 1652B/53B logic analyzer. The commands in this subsystem are:

- BASE
- PATTern
- RANGe
- REMove
- WIDTh

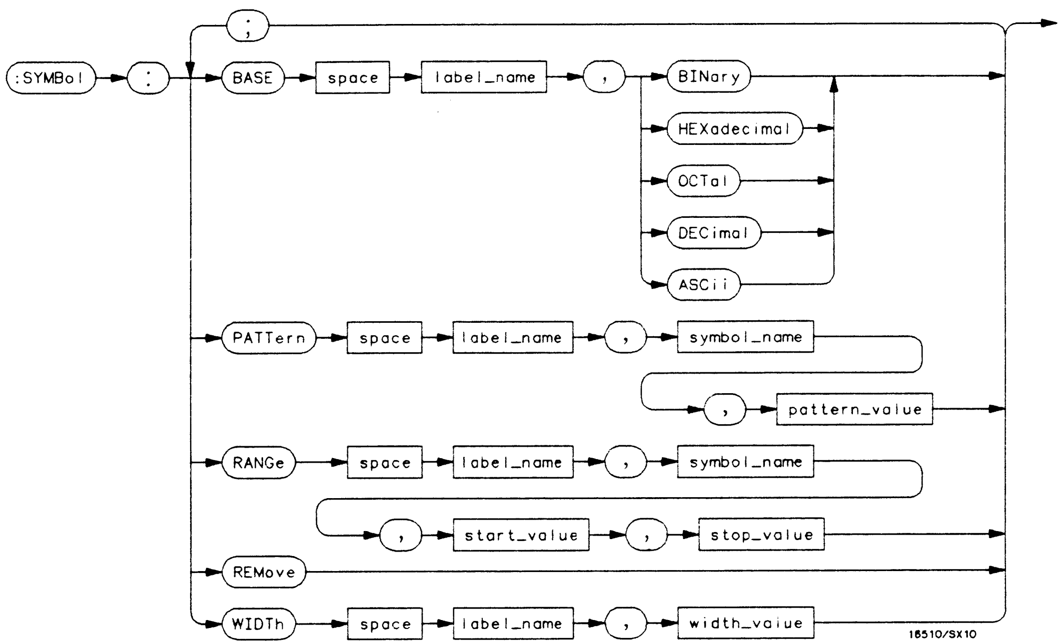


Figure 20-1. SYMBol Subsystem Diagram

label_name = *string of up to 6 alphanumeric characters*
symbol_name = *string of up to 16 alphanumeric characters*
pattern_value = "{*#B{0|1|X}*... |
#Q{0|1|2|3|4|5|6|7|X}... |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X}... |
{0|1|2|3|4|5|6|7|8|9}... }"
start_value = "{*#B{0|1}*... |
#Q{0|1|2|3|4|5|6|7}... |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F}... |
{0|1|2|3|4|5|6|7|8|9}... }"
stop_value = "{*#B{0|1}*... |
#Q{0|1|2|3|4|5|6|7}... |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F}... |
{0|1|2|3|4|5|6|7|8|9}... }"
width_value = *integer from 1 to 16*

Figure 20-1. SYMBol Subsystem Syntax Diagram (continued)

SYMBol

selector

The SYMBol selector is used as a part of a compound header to access the commands used to create symbols. It always follows the MACHine selector because it selects a branch directly below the MACHine level in the command tree.

Command Syntax: :MACHine{1|2}:SYMBol

Example:

```
OUTPUT XXX;":MACHINE1:SYMBOL:BASE 'DATA', BINARY"
```

BASE

command

The BASE command sets the base in which symbols for the specified label will be displayed in the symbol menu. It also specifies the base in which the symbol offsets are displayed when symbols are used.

Note

BINary is not available for labels with more than 20 bits assigned. In this case the base will default to HEXadecimal.

Command Syntax: :MACHine{1|2}:SYMBol:BASE <label_name>,<base_value>

where:

<label_name> ::= string of up to 6 alphanumeric characters
<base_value> ::= {BINary | HEXadecimal | OCTal | DECimal | ASCii}

Example:

```
OUTPUT XXX;":MACHINE1:SYMBOL:BASE 'DATA',HEXADECIMAL"
```

PATTern

command

The PATTern command allows you to create a pattern symbol for the specified label.

Because don't cares (X) are allowed in the pattern value, it must always be expressed as a string. You may still use different bases, though don't cares cannot be used in a decimal number.

Command Syntax: :MACHine{1|2}:SYMBol:PATTern<label_name>,<symbol_name>,<pattern_value>

where:

<label_name> ::= string of up to 6 alphanumeric characters
<symbol_name> ::= string of up to 16 alphanumeric characters
<pattern_value> ::= "{#B{0|1|X} . . . |
#Q{0|1|2|3|4|5|6|7|X} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F|X} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Example:

```
OUTPUT XXX;":MACHINE1:SYMBOL:PATTERN 'STAT', 'MEM_RD','#H01XX'"
```

RANGe

command

The RANGe command allows you to create a range symbol containing a start value and a stop value for the specified label. The values may be in binary (#B), octal (#Q), hexadecimal (#H) or decimal (default). You may not use "don't cares" in any base.

Command Syntax: :MACHine{1|2}:SYMBol:RANGe <label_name>,<symbol_name>,<start_value>, <stop_value>

where:

<label_name> ::= string of up to 6 alphanumeric characters
<symbol_name> ::= string of up to 16 alphanumeric characters
<start_value> ::= "{#B{0|1} . . . |
#Q{0|1|2|3|4|5|6|7} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"
<stop_value> ::= "{#B{0|1} . . . |
#Q{0|1|2|3|4|5|6|7} . . . |
#H{0|1|2|3|4|5|6|7|8|9|A|B|C|D|E|F} . . . |
{0|1|2|3|4|5|6|7|8|9} . . . }"

Example:

```
OUTPUT XXX;":MACHINE1:SYMBOL:RANGE 'STAT', 'IO_ACC','0','#H000F'"
```

REMove

command

The REMove command deletes all symbols from a specified machine.

Command Syntax: :MACHine{1|2}:SYMBol:REMove

Example:

```
OUTPUT XXX;":MACHINE1:SYMBOL:REMOVE"
```

WIDTh

command

The WIDTh command specifies the width (number of characters) in which the symbol names will be displayed when symbols are used.

Note The WIDTh command does not affect the displayed length of the symbol offset value.

Command Syntax: :MACHine{1|2}:SYMBol:WIDTh <label_name>,<width_value>

where:

<label_name> ::= string of up to 6 alphanumeric characters

<width_value> ::= integer from 1 to 16

Example:

```
OUTPUT XXX;":MACHINE1:SYMBOL:WIDTH 'DATA',9 "
```

21 - SCOPe Subsystem

SCOPe Subsystem 21

Introduction

The SCOPe subsystem provides access to the commands and the oscilloscope subsystem commands that control the basic operation of the oscilloscope. At the SCOPe subsystem level is a command that turns the oscilloscope on or off (SMODe), specifies how the oscilloscope is Armed (ARM), and the AUToscale command.

Additionally, the following subsystems are a part of the SCOPe subsystem. Each is explained in a separate chapter.

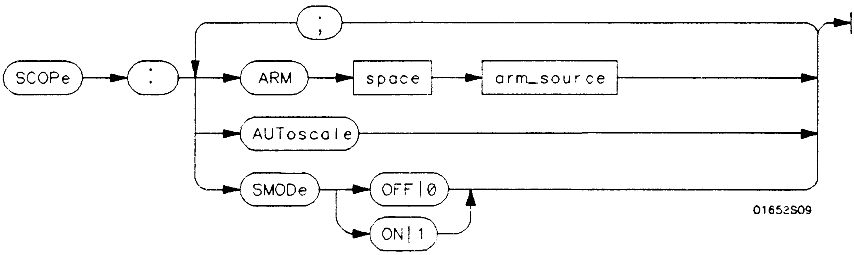
- CHANnel subsystem (chapter 22)
- TRIGger subsystem (chapter 23)
- ACQuire subsystem (chapter 24)
- TIMebase subsystem (chapter 25)
- WAVeform subsystem (chapter 26)
- MEASure subsystem (chapter 27)

Not all scope-related functions can be duplicated with programming instructions. If you are unable to get a desired configuration strictly through programming instruction, try the following steps:

1. Manually configure the HP 1652B/53B through the front panel.

2. Save configuration to a disk (through the front panel or through the `:MMEM:STORE "CONFIG","Setups"` instruction).

Now you can use the command `MMEM:LOAD "CONFIG"` to load in the desired configuration.



arm_source = {*RUN* | *MACHine*{*1* | *2*} | *BNC*}

Figure 21-1. SCOPe Subsystem Syntax Diagram

SCOPe

selector

The SCOPe selector is used to indicate the beginning of a compound command (or query) for a function within the SCOPe subsystem. Since SCOPe is a root-level command, it will normally appear as the first element of a compound header.

Command Syntax: :SCOPe

Example: `OUTPUT XXX; ":SCOPE:TRIGGER:SLOPE NEGATIVE"`

Arm

command/query

The ARM command specifies the arming source of the oscilloscope.

The ARM query returns the source that the oscilloscope is armed by.

Command Syntax: :SCOPe:ARM <arm_source>

where:

<arm_source> ::= {RUN | MACHine{1|2} | BNC}

Example:

```
OUTPUT XXX;":SCOPE:ARM:MACHINE2"
```

Query Syntax: :SCOPe:ARM?

Returned Format: [:SCOPe:ARM] <arm_source>

Example:

```
10 DIM String$[100]
20 OUTPUT XXX;":SCOPE:ARM?"
30 ENTER XXX; String$
40 PRINT String$
50 END
```

AUToscale

command

The AUToscale command causes the oscilloscope to automatically select the vertical sensitivity, vertical offset, trigger level and timebase settings for a stable display on one or both channels. The input signal required for Autoscale must have an amplitude above 10 mV peak, and a frequency between 50 Hz and 100 MHz..

Command Syntax: :SCOPe:AUToscale

Example:
```
OUTPUT XXX;":SCOPE:AUTOSCALE"
```

SMODe

command/query

The SMODe command allows the oscilloscope to be turned on or off over the bus.

The SMODe query returns the current status of the oscillosocpe.

Command Syntax: :SCOPe:SMODe {ON|OFF}

Example:

```
OUTPUT XXX;":SCOPe:SMODe ON"
```

Query Syntax: :SCOPe:SMODe?

Returned Format: [:SCOPe:SMODe] {ON|OFF}<NL>

Example:

```
10 DIM Sm$[100]
20 OUTPUT XXX;":SCOPE:SMODE?"
30 ENTER XXX;Sm$
40 PRINT Sm$
50 END
```

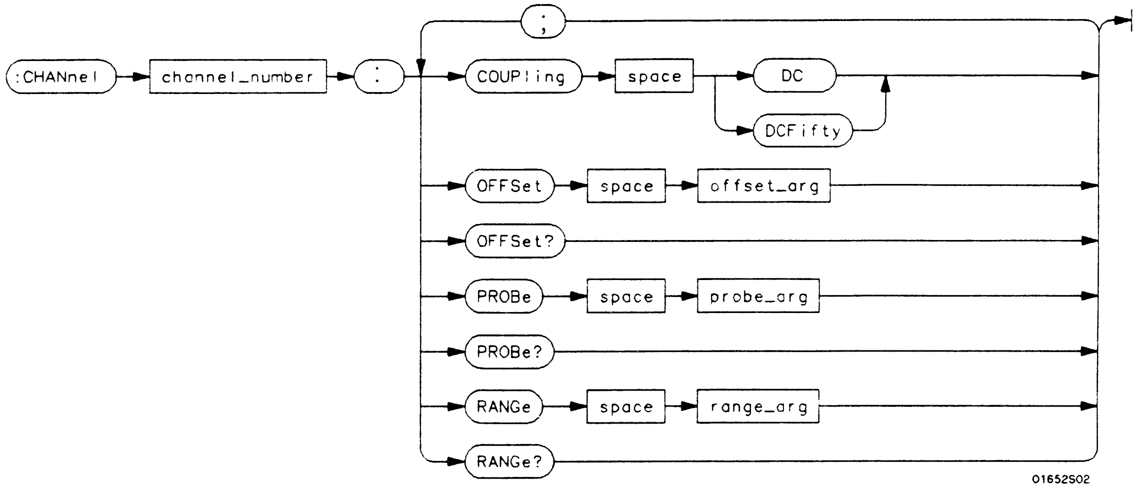
22 - CHANNel Subsystem

CHANnel Subsystem 22

Introduction

The CHANnel subsystem commands control the channel display and the vertical axis of the oscilloscope. Each channel must be programmed independently for all offset, range and probe functions. The commands are:

- CHANnel
- COUPling
- OFFSet
- PROBe
- RANGe



channel_number = *{1 | 2}*
offset_arg = *real number defining the voltage at the center of the display. The offset range depends on the input impedance setting. The offset range for 1 MΩ input is −125 V to +125 V. The offset range for 50 Ω input is −5 V to + 5 V.*
probe_arg = *integer from 1 through 1000, specifying the probe attenuation with respect to 1.*
range_arg = *real number specifying vertical sensitivity. The allowable range is 15 mV to 10 V for a probe attenuation of 1. The specified range is equal to 4 times Volts/Div.*

Figure 22-1. CHANnel Subsystem Syntax Diagram

CHANnel

selector

The CHANnel selector is used as part of a compound command header to access the settings found in oscilloscope's CHANnel menu. It always follows the SCOPe selector because it selects a branch below the SCOPe level in the command tree.

Command Syntax: :SCOPe:CHANnel<N>

where:

<N> ::= {1 | 2}

Example:

```
OUTPUT XXX; ":SCOPE:CHANNEL2:OFFSET 2.5"
```

COUPling

command/query

The COUPling command sets the input impedance for the selected channel. The choices are either 1M Ohm (DC) or 50 Ohms (DCFifty).

The query returns the current input impedance for the specified channel.

Command Syntax: :SCOPe:CHANnel{1|2}:COUPling {DC|DCFifty}

Example:
```
OUTPUT XXX;":SCOPE:CHANNEL1:COUPLING DC"
```

Query Syntax: :SCOPe:CHANnel{1|2}:COUPling?

Returned Format: [:SCOPe:CHANnel{1|2}:COUPling] {DC|DCFifty}<NL>

Example:
```
10 DIM Cc$[100]
20 OUTPUT XXX;":SCOPE:CHANNEL1:COUPLING?"
30 ENTER XXX;Cc$
40 PRINT Cc$
50 END
```

OFFSet

command/query

The OFFSet command sets the voltage that is represented at center screen for the selected channel. The allowable offsets for 1:1 probes are:

± 2 V < 74 mV/div
± 10 V between 74 mV/div and 370 mV/div
± 50V between 370 mV/div and 1.85 V/div
± 125 V > 1.85 V/div

When the input impedance is set to 50 Ω the maximum offset is æ 2 V for V/Div settings less than 74 mV and is ±5 V for V/Div settings greater than 74 mV.

The offset value is recompensated whenever the probe attenuation factor is changed.

The query returns the current value for the selected channel.

Command Syntax: :SCOPe:CHANnel{1|2}:OFFSet <value>

where:

<value> ::= {− 250V to + 250 V max. at 1 MΩ | − 5 V to + 5 V at 50 Ω}

Example:

```
OUTPUT XXX;":SCOP:CHAN1:OFFS 1.5"
```

Query Syntax: :SCOPe:CHANnel{1|2}:OFFSet?

Returned Format: [:SCOPe:CHANnel{1|2}:OFFSet] <value><NL>

Example:

```
10 DIM Co$[100]
20 OUTPUT XXX;":SCOPE:CHANNEL1:OFFSET?"
30 ENTER XXX;Co$
40 PRINT Co$
50 END
```

PROBe

command/query

The PROBe command specifies the attenuation factor for an external probe connected to a channel. The command changes the channel voltage references such as range, offset, trigger levels and automatic measurements. The actual sensitivity is not changed at the channel input. The allowable probe attenuation factor is an integer from 1 to 1000.

The query returns the probe attenuation factor for the selected channel.

Command Syntax: :SCOPe:CHANnel{1|2}:PROBe <atten>

where:

<atten> ::= integer from 1 to 1000

Example:

```
OUTPUT XXX;":SCOPe:CHAN1:PROB 10"
```

Query Syntax: :SCOPe:CHANnel{1|2}:PROBe?

Returned Format: [:SCOPe:CHANnel{1|2}:PROBe] <atten><NL>

Example:

```
10 DIM Att$[100]
20 OUTPUT XXX;":SCOPE:CHANNEL1:PROBE?"
30 ENTER XXX;Att$
40 PRINT Att$
50 END
```

RANGe

command/query

The RANGe command defines the full-scale (4 × Volts/Div) vertical axis of the selected channel. The values for the RANGe command are dependent on the current probe attenuation factor for the selected channel. The allowable range for a probe attenuation factor of 1:1 is 60 mV to 40 V. For a larger probe attenuation factor, multiply the range limit by the probe attenuation factor.

The RANGe query returns the current range setting.

Command Syntax: :SCOPe:CHANnel{1|2}:RANGe <range>

where:

<range> ::= 60 mV to 40 V for a probe attenuation factor of 1:1

Example:

```
OUTPUT XXX;":SCOPE:CHANNEL1:RANGE 4.8"
```

Query Syntax: :SCOPe:CHANnel{1|2}:RANGe?

Returned Format: [:SCOPe:CHANnel{1|2}:RANGe] <range><NL>

Example:

```
10 DIM Pr$[100]
20 OUTPUT XXX;":SCOPE:CHANNEL1:RANGE?"
30 ENTER XXX;Pr$
40 PRINT Pr$
50 END
```

23 - TRIGger Subsystem

TRIGger Subsystem 23

Introduction

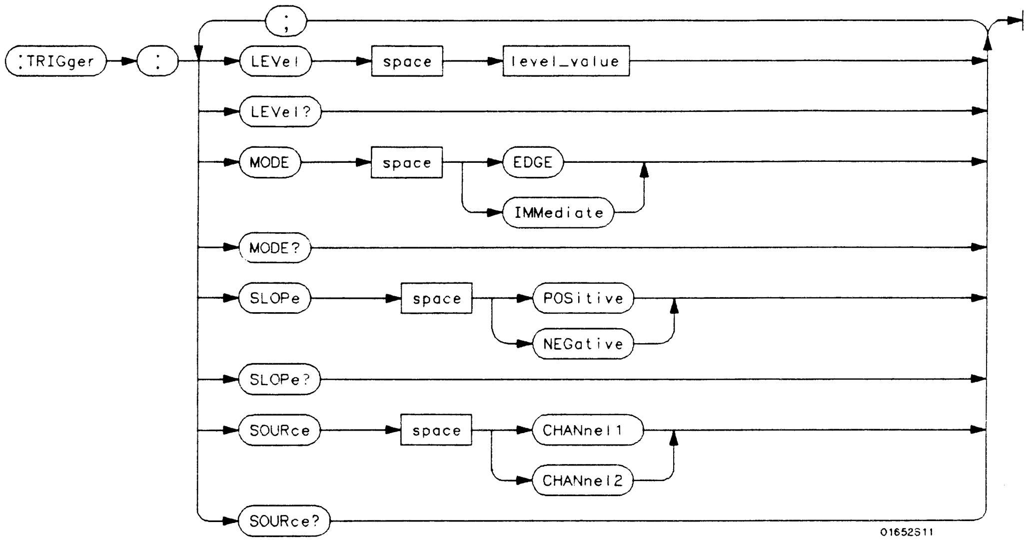
The commands of the TRIGger subsystem allow you to set all the trigger conditions necessary for generating a trigger. There are two trigger modes: Edge and Immediate. If a command is valid for the chosen trigger mode, then that setting will be accepted by the oscilloscope. However, if the command is not valid for the trigger mode, an error will be generated. None of the commands of this subsystem are used in conjunction with Immediate trigger mode. See Figure 23-1 for the TRIGger subsystem syntax diagram.

The Edge Trigger Mode

In the Edge trigger mode, the oscilloscope triggers on an edge of a waveform, specified by the SOURce, LEVel, and SLOPe commands. If a source is not specified, then the current source is assumed.

The Immediate Trigger Mode

In the Immediate trigger mode, the oscilloscope will trigger by itself when the arming requirements are met.



level _value = *trigger level in volts*

Figure 23-1. TRIGger Subsystem Syntax Diagram

TRIGger

selector

The TRIGger selector is used as part of a compound command header to access the settings found in oscilloscope's Trigger menu. It always follows the SCOPe selector because it selects a branch below the SCOPe level in the command tree.

Command Syntax: :SCOPe:TRIGger

Example:

```
OUTPUT XXX; ":SCOPE:TRIGGER:CHANNEL1;LEVEL 2.0"
```

LEVEL

command/query

The LEVEL command sets the trigger level voltage for the selected source or path. This command cannot be used in the IMMEDIATE trigger mode.

The query returns the trigger level for the current path or source.

Note There is no shortform for LEVEL. This is an intentional deviation from the normal truncation rule.

Command Syntax: :SCOPe:TRIGger:LEVEL <value>

where:

<value> ::= Trigger level in volts

Example:

```
OUTPUT XXX;":SCOPE:TRIG:LEVEL 1.0"
```

Query Syntax: :SCOPe:TRIGger:LEVEL?

Returned Format: [:SCOPe:TRIGger:LEVEL] <value><NL>

Example:

```
10 DIM E1$[100]
20 OUTPUT XXX;":SCOPE:TRIGGER:SOURCE CHANNEL1;LEVEL?"
30 ENTER XXX;E1$
40 PRINT E1$
50 END
```

MODE

command/query

The MODE command allows you to select the trigger mode for the oscilloscope. The EDGE mode will trigger the oscilloscope on an edge whose slope is determined by the SLOPe command at a voltage set by the LEVEL command. In the IMMediate trigger mode, the oscilloscope goes to a freerun mode and does not wait for a trigger. The IMMediate mode is used in armed-by other machine applications.

The query returns the current mode.

Command Syntax: :SCOPe:TRIGger:MODE {EDGE|IMMediate}

Example:

```
OUTPUT XXX;":SCOPE:TRIGGER:MODE EDGE"
```

Query Syntax: :SCOPe:TRIGger:MODE?

Returned Format: [:SCOPe:TRIGger:MODE] {EDGE|IMMediate}<NL>

Example:

```
10 DIM Md$[100]
20 OUTPUT XXX;":SCOPE:TRIGGER:MODE?"
30 ENTER XXX;Md$
40 PRINT Md$
50 END
```

SLOPe

command/query

The SLOPe command selects the trigger slope for the previously specified trigger source. This command can only be used in the EDGE trigger mode.

The query returns the slope of the current trigger source.

Command Syntax: :SCOPe:TRIGger:SLOPe {POSitive|NEGative}

Example:

```
OUTPUT XXX;":SCOP:TRIG:SOURCE CHAN1;SLOPE POS"
```

Query Syntax: :SCOPe:TRIGger:SLOPe?

Returned Format: [:SCOPe:TRIGger:SLOPe] {POSitive|NEGative}<NL>

Example:

```
10 DIM Ts$[100]
20 OUTPUT XXX;":SCOP:TRIG:SOUR CHAN1;SLOP?"
30 ENTER XXX;Ts$
40 PRINT Ts$
50 END
```

SOURce

command/query

The SOURce command is used to select the trigger source and is used for any subsequent SLOPe and LEVEL commands. This command can only be used in the EDGE trigger mode.

The query returns the current trigger source.

Command Syntax: :SCOPe:TRIGger:SOURce {CHANnel{1|2}}

Example:
```
OUTPUT XXX;":SCOP:TRIG:SOUR CHAN1"
```

Query Syntax: :SCOPe:TRIGger:SOURce?

Returned Format: [:SCOPe:TRIGger:SOURce] {CHANnel{1|2}}<NL>

Example:
```
10 DIM Tso$[100}
20 OUTPUT XXX;":SCOPE:TRIGGER:SOURCE?"
30 ENTER XXX;Tso$
40 PRINT Tso$
50 END
```

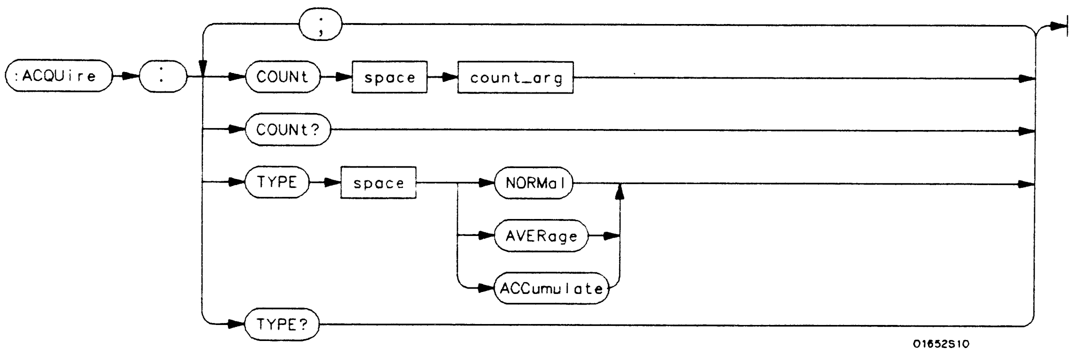
24 - ACQuire Subsystem

ACQuire Subsystem 24

Introduction

The ACQuire subsystem commands are used to select the type of acquisition and the number of averages to be taken if the average type is chosen. The commands are:

- COUNt
- TYPE



count_arg = {2|4|8|16|32|64|128|256} *An integer that specifies the number of averages to be taken of each time point.*

Figure 24-1. ACQuire Subsystem Syntax Diagram

Acquisition Type Normal

In the Normal mode, with the ACCumulate command OFF, the oscilloscope acquires waveform data and then displays the waveform. When the oscilloscope makes a new acquisition, the previously acquired waveform is erased from the display and replaced by the newly acquired waveform.

When the ACCumulate command is ON, the oscilloscope displays all the waveform acquisitions without erasing the previously acquired waveform.

Acquisition Type Average

In the Average mode, the oscilloscope averages the data points on the waveform with previously acquired data. Averaging helps eliminate random noise from the displayed waveform. In this mode the ACCumulate command is OFF. When Average mode is selected, the number of averages must also be specified using the COUNt command. Previously averaged waveform data is erased from the display and the newly averaged waveform is displayed.

ACQuire

selector

The ACQuire selector is used as part of a compound command header to access the settings found in oscilloscope's Acquire menu. It always follows the SCOPe selector because it selects a branch below the SCOPe level in the command tree.

Command Syntax: :SCOPe:ACQuire

Example: `OUTPUT XXX; ":SCOPE:ACQUIRE:TYPE NORMAL"`

COUNt

command/query

The COUNt command specifies the number of acquisitions for the running weighted average. This command generates an error if Normal acquisition mode is specified.

The query returns the last specified count.

Command Syntax: :SCOPe:ACQuire:COUNt <count>

where

<count> ::= {2|4|8|16|32|64|128|256}

Example

```
OUTPUT XXX;":SCOPE:ACQUIRE:COUNT 16"
```

Query Syntax: :SCOPe:ACQuire:COUNt?

Returned Format [:SCOPe:ACQuire:COUNt] <count><NL>

Example:

```
10 DIM Ac$[100]
20 OUTPUT XXX;":SCOPE:ACQ:COUN?"
30 ENTER XXX;Ac$
40 PRINT Ac$
50 END
```

TYPE

command/query

The TYPE command selects the type of acquisition that is to take place when the STARt command is executed. One of three acquisition types may be selected: the NORMal, AVERage, or ACCumulate mode.

The query returns the last specified type.

Command Syntax :SCOPe:ACQuire:TYPE {NORMal|AVERage|ACCumulate}

Example:
```
OUTPUT XXX;":SCOPE:ACQUIRE:TYPE NORMAL"
```

Query Syntax: :SCOPe:ACQuire:TYPE?

Returned Format: [:SCOPe:ACQuire:TYPE] {NORMal|AVERage}<NL>

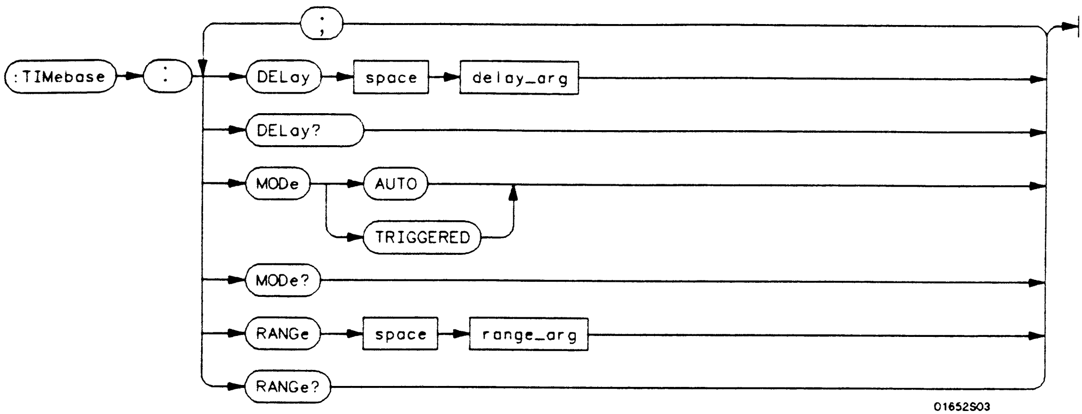
Example:
```
10 DIM At$[100]
20 OUTPUT XXX;":SCOPE:ACQUIRE:TYPE?"
30 ENTER XXX;At$
40 PRINT At$
50 END
```

25 - TIMebase Subsystem

TIMebase Subsystem 25

Introduction

The commands of the TIMebase subsystem control the Timebase, Trigger Delay Time, and the Timebase Mode. If TRIGGERED mode is to be used, ensure that the trigger specifications of the TRIGger subsystem have been set. Refer to Figure 25-1 for the TIMebase subsystem syntax diagram.



delay_arg = *delay time in seconds, from -2500 seconds through +2500 seconds*
range_arg = *a real number from 5 ns through 10s*

Figure 25-1. TIMebase Subsystem Syntax Diagram

TIMebase

selector

The TIMebase selector is used as part of a compound command header to access the settings found in oscilloscope's Timebase menu. It always follows the SCOPe selector because it selects a branch below the SCOPe level in the command tree.

Command Syntax: :SCOPe:TIMebase

Example: `OUTPUT XXX; ":SCOPE:TIMEBASE:MODE AUTO"`

DELAY

command/query

The DELAY command sets the time between the trigger and the center of the screen if the trigger events count is zero. If the trigger events count is non-zero, the center of the screen is the trigger events count plus the delay time.

The query returns the current delay setting.

Note The DELAY command in the TIMebase subsystem has no shortform. This is an intentional deviation from the normal truncation rules.

Command Syntax: :SCOPe:TIMebase:DELAY <delay time>

where:

<delay time> ::= delay time in seconds

Example:

```
OUTPUT XXX;":SCOPe:TIMebase:DELAY 2US"
```

Query Syntax: :SCOPe:TIMebase:DELAY?

Returned Format: [:SCOPe:TIMebase:DELAY] <value> <NL>

Example:

```
10 DIM Dt$[100]
20 OUTPUT XXX;":SCOPe:TIMebase:DELAY?"
30 ENTER XXX;Dt$
40 PRINT Dt$
50 END
```

MODE

command/query

The MODE command sets the oscilloscope timebase to either Auto or Triggered mode. When the AUTO mode is chosen, the oscilloscope waits approximately one second for a trigger to occur. If a trigger is not generated within that time, then auto trigger is executed. If a signal is not applied to the input, a baseline is displayed. If there is a signal at the input and the specified trigger conditions have not been met within one second, the waveform display will not be synchronized to a trigger.

When the TRIGGERED mode is chosen, the oscilloscope waits until a trigger is received before data is acquired. The TRIGGERED mode should be used when the trigger source signal is less than at a 40 Hz repetition rate.

The Auto-Trig On field in the trigger menu is the same as the AUTO mode over HP-IB or RS-232C. Setting the mode to TRIGGERED is the same as the Auto-Trig Off on the front panel.

The query returns the current TIMebase mode.

Note The TRIGGERED argument for MODE has no shortform. This is an intentional deviation from the normal truncation rule.

Command Syntax: :SCOPe:TIMebase:MODE {TRIGGERED|AUTO}

Example: OUTPUT XXX;":SCOPE:TIME:MODE AUTO"

Query Syntax: :SCOPe:TIMebase:MODE?

Returned Format: [:SCOPe:TIMebase:MODE] {AUTO|TRIGGERED}<NL>

Example:

```
10 DIM Tm$[100]
20 OUTPUT XXX;":SCOPe:TIMEBASE:MODE?"
30 ENTER XXX;Tm$
40 PRINT Tm$
50 END
```

RANGe

command/query

The RANGE command sets the full-scale horizontal time in seconds. The RANGE value is ten times the front panel field of s/div.

The query returns the current range.

Command syntax: :SCOPe:TIMebase:RANGe <range>

where:

<range> ::= time in seconds

Example:

```
OUTPUT XXX;":SCOPE:TIMEBASE:RANGE 2US"
```

Query Syntax: :SCOPe:TIMebase:RANGe?

Returned Format: [:SCOPe:TIMebase:RANGe] <range><NL>

Example:

```
10 DIM Tr$[100]
20 OUTPUT XXX;":SCOPE:TIMEBASE:RANGE?"
30 ENTER XXX;Tr$
40 PRINT Tr$
50 END
```

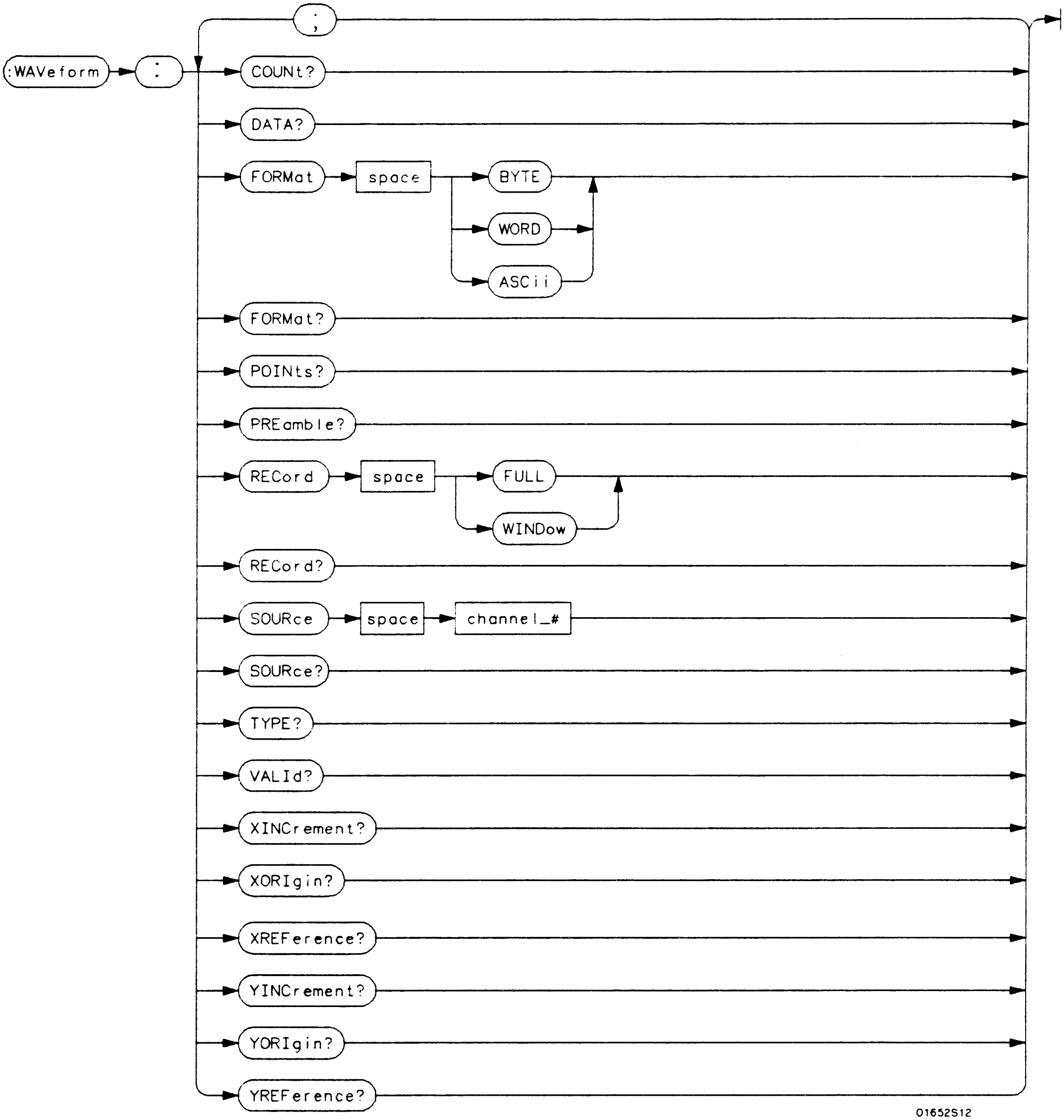
26 - WAVeform Subsystem

WAVeform Subsystem 26

Introduction

The commands of the WAVeform subsystem are used to transfer waveform data from the oscilloscope to a controller. The commands are:

- COUNt
- DATA
- FORMat
- POINts
- PREamble
- RECord
- SOURce
- TYPe
- VALid
- XINCrement
- XORigin
- XREFerence
- YINCrement
- YORigin
- YREFerence



channel_# = {1|2}

Figure 26-1. WAVeform Subsystem Syntax Diagram

Waveform Record

The waveform record is actually contained in two portions; the waveform data and preamble. The waveform data is the actual data acquired for each point. The preamble contains the information for interpreting waveform data. Data in the preamble includes number of points acquired, format of acquired data, average count and the type of acquired data. The preamble also contains the X and Y increments, origins, and references for the acquired data for translation to time and voltage values.

The values set in the preamble are based on the settings of the variables in the ACQuire, WAVeform, CHANnel, and TIMebase subsystems. The ACQuire subsystem determines the acquisition type and the average count, the WAVeform subsystem sets the number of points and the format mode for sending waveform data over the remote interface and the CHANnel and TIMebase subsystems set all the X - Y parameters.

Data Acquisition Types

The two acquisition types that may be chosen are Normal and Average.

Normal Mode

In the Normal mode, with ACCumulate command OFF, the oscilloscope acquires waveform data and then displays the waveform. When the oscilloscope takes a new acquisition, the previously acquired waveform is erased from the display and replaced by the newly acquired waveform.

When ACCumulate is set ON, the oscilloscope displays all the waveform acquisitions without erasing the previously acquired waveform.

Average Mode

In the Average mode, the oscilloscope averages the data points on the waveform with previously acquired data. Averaging helps eliminate random noise from the displayed waveform. In this mode ACCumulate is set to OFF. When Average mode is selected the number of averages must also be specified using the COUNt command. Previously displayed waveform data is erased from the display and the newly averaged waveform is displayed.

Format for Data Transfer

There are three formats for transferring waveform data over the remote interface. The formats are WORD, BYTE, and ASCII.

WORD and BYTE formatted waveform records are transmitted using the arbitrary block program data format specified in IEEE-488.2. When you use this format, the ASCII character string "#8 < DDDDDDDD > " is sent before the actual data. Each D represents an ASCII digit. The eight-digit number represents the number of bytes to follow.

For example, if 2048 points of data are to be transmitted, the ASCII string #800002048 would be sent.

BYTE Format

In BYTE format, the six least significant bits represent the waveform data. This means that the display is divided into 64 vertical increments. The most significant bit is not used. The second most significant bit is the overflow bit. If this bit is set to "1" and all data bits are set to "0" then the waveform is clipped at the top of the screen. If all "0"s are returned, then the waveform is clipped on the bottom of the display (see figure 26-2).

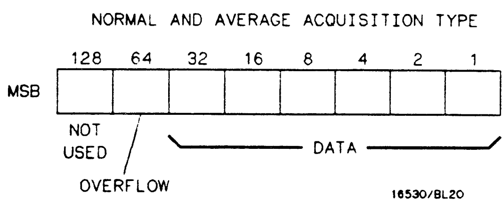


Figure 26-2. Byte Data Structure

The data returned in BYTE format are the same for either Normal or Average acquisition types. The data transfer rate in this format is faster than the other two formats.

WORD Format Word data is two bytes wide with the most significant byte of each word being transmitted first. Each 16-bit value effectively places a data point on screen. The screen therefore is divided into 16384 vertical increments. The WORD data structure for normal and average acquisition types are shown in figure 26-3.

The relationship between BYTE and WORD formats are similar. Byte data values equal word data values divided by 256. This is the reason that the least significant byte in the normal acquisition mode always contains "0"s. In the average acquisition mode, the extra bits of resolution gained by averaging occupy the least significant byte of the word. However, this is only true when RECord type is set to WINDow.

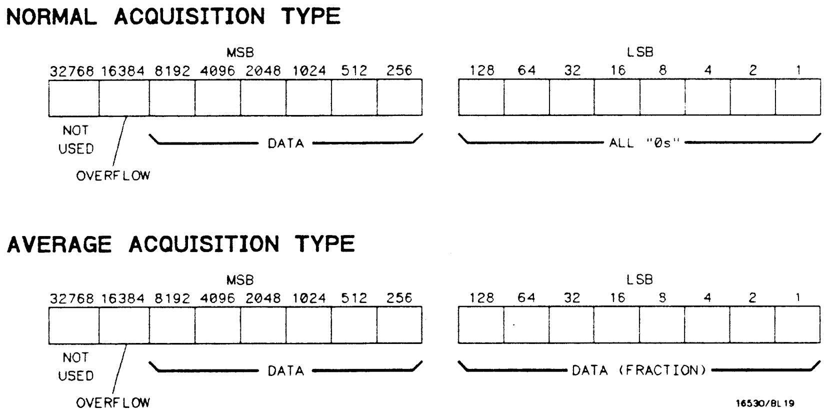


Figure 26-3. Word Data Structure

ASCII Format ASCII formatted waveform records are transmitted one value at a time, separated by a comma. The data values transmitted are the same as would be sent in the WORD format except that they are converted to an integer ASCII format (six or less characters) before being transmitted. The header before the data is not included in this format.

Data Conversion

Data sent from the HP 1652B/53B is raw data and must be scaled for useful interpretation. The values used to interpret the data are the X and Y references, X and Y origins, and X and Y increments. These values are read from the waveform preamble or by the queries of these values.

Conversion from Data Value to Voltage

The formula to convert a data value returned by the instrument to a voltage is:

voltage = [(data value − yreference) × yincrement] + yorigin

Conversion from Data Value to Time

The time value of a data point can be determined by the position of the data point. As an example, the third data point sent with XORIGIN = 16ns, XREFERENCE = 0 and XINCREMENT = 2ns. Using the formula:

time = [(data point number − xreference) × xincrement] + xorigin

would result in the following calculation:

time = [(3 - 0) × 2ns] + 16ns = 22ns.

Conversion from Data Value to Trigger Point

The trigger data point can be determined by calculating the closest data point to time 0.

WAVeform

selector

The WAVeform selector is used as part of a compound command header to access the settings found in oscilloscope's Waveform menu. It always follows the SCOPe selector because it selects a branch below the SCOPe level in the command tree.

Command Syntax: :SCOPe:WAVeform

Example:

```
OUTPUT XXX; ":SCOPE:WAVEFORM:"
```

COUNt

query

The COUNt query returns the AVERage count that was last specified in the Acquire subsystem. If the display mode is either NORMal or ACCumulate, a 1 is returned. If the display mode is AVERage, the average number is returned.

Query Syntax: :SCOPe:WAVeform:COUNt?

Returned Format: [:SCOPe:WAVeform:COUNt] <count> <NL>

where:

<count> ::= {2|4|8|16|32|64|128|256}

Example:

```
10 DIM Ac$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM COUNT?"
30 ENTER XXX;Ac$
40 PRINT Ac$
50 END
```

DATA

query

The DATA query returns the waveform record stored in a specified channel buffer. The SOURce command of this subsystem has to be used to select the specified channel. The data is transferred based on the FORMAT (BYTE, WORD or ASCII) chosen and the RECORD specified (FULL or WINDOW). Since WAVeform:DATA is a query only, it can not be used to send a waveform record back to the oscilloscope from the controller. If a waveform record is to be saved for later reloading into the oscilloscope, the SYSTem:DATA command should be used. See the DATA instruction in the SYSTem subsystem for information concerning the <block data> parameter.

Query Syntax: :SCOPe:WAVeform:[SOURce CHANnel{1|2};]DATA?

Returned Format: [:SCOPe:WAVeform:DATA]#800004096 <block data><NL>

The following example program moves data from the HP 1652B/53B to a controller.

Example:

```
100 CLEAR XXX
110 OUTPUT XXX;":SYSTEM:HEADER OFF"
120 OUTPUT XXX;":SCOPE:ACQUIRE:TYPE NORMAL"
130 OUTPUT XXX;":SCOPE:WAVEFORM:SOURCE CHANNEL1"
140 OUTPUT XXX;":SCOPE:WAVEFORM:FORMAT BYTE"
150 OUTPUT XXX;":SCOPE:WAVEFORM:RECORD FULL"
160 OUTPUT XXX;":SCOPE:AUTOSCALE"
170 DIM Header$[20]
180 Length=2048
190 ALLOCATE INTEGER WAVEFORM(1:Length)
200 OUTPUT XXX;":SCOPE:WAVEFORM:DATA?"
210 ENTER XXX USING "#,10A";Header$
220 ENTER XXX USING "#,B";Waveform(*)
230 ENTER XXX USING "#,B";Lastchar
240 END
```

FORMat

command/query

The FORMat command specifies the data transmission mode of waveform data over the remote interface.

The query returns the currently specified format.

Command Syntax: :SCOPe:WAVeform:FORMat {BYTE|WORD|ASCii}

Example:

```
OUTPUT XXX;":SCOPE:WAV:FORMAT"
```

Query Syntax: :SCOPe:WAVeform:FORMat?"

Returned Format: [:SCOPe:WAVeform:FORMat] {BYTE|WORD|ASCii}<NL>

Example:

```
10 DIM Fo$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:FORMAT?"
30 ENTER XXX;Fo$
40 PRINT Fo$
50 END
```

POINts

query

When WAVeform RECord is set to FULL, the POINts query always returns a value of 2048 points. When WAVeform RECord is set to WINDow, then the query returns the number of points displayed on screen.

Query Syntax: :SCOPe:WAVeform:POINts?

Returned Format: [:SCOPe:WAVeform:POINts] <points> <NL>

where:

<points> ::= number of points depending on setting of WAVeform RECord command

Example:

```
10 DIM Po$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:POINTS?"
30 ENTER XXX;Po$
40 PRINT Po$
50 END
```

PREAmble

query

The PREAmble query returns the preamble of the specified channel. The channel is specified using the SOURce command.

Note The short form for PREAMBLE is PREAmble. This is an intentional deviation from the normal truncation rule.

Query Syntax: :SCOPe:WAVeform:[SOURce CHANnel{1|2};]PREAmble?

Returned Format: [:SCOPe:WAVeform:PREAmble]

<format>,
<type>,
<points>,
<count>,
<Xincrement>,
<Xorigin>,
<Xreference>,
<Yincrement>,
<Yorigin>,
<Yreference><NL>

Example:

```
10 DIM Pr$[300]
20 OUTPUT XXX;":SCOPE:WAVEFORM:PREAMBLE?"
30 ENTER XXX;Pr$
40 PRINT Pr$
50 END
```

RECord

command/query

The RECord command specifies the data you want to receive over the bus. The choices are FULL or WINDOW. When FULL is chosen the entire 2048 point record of the specified channel is transmitted over the bus. In WINDOW mode, only the data displayed on screen will be returned. Use the SOURce command to select the channel of interest. The query returns the present mode chosen.

Command Syntax: :SCOPe:WAVeform:RECord {FULL|WINDow}

Example:

```
OUTPUT XXX;":SCOPE:WAV:SOUR CHAN1:REC FULL"
```

Query Syntax: :SCOPe:WAVeform:RECord?

Returned Format: [:SCOPe:WAVeform:RECord] {FULL|WINDow}<NL>

Example:

```
10 DIM Wr$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:SOURCE CHANNEL1:RECORD?"
30 ENTER XXX;Wr$
40 PRINT Wr$
50 END
```

SOURce

command/query

The SOURce command specifies the channel that is to be used for all subsequent waveform commands.

The query returns the presently selected channel.

Command Syntax: :SCOPe:WAVeform:SOURce CHANnel{1|2}

Example:

```
OUTPUT XXX;":SCOPE:WAVEFORM:SOURCE CHANNEL1"
```

Query Syntax: :SCOPe:WAVeform:SOURce?

Returned Format: [:SCOPe:WAVeform:SOURce] CHANnel<N> <NL>

Example:

```
10 DIM Ws$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:SOURCE?"
30 ENTER XXX;Ws$
40 PRINT Ws$
50 END
```

TYPE

query

The TYPE query returns the present acquisition type which was specified in the ACQuire subsystem.

Query Syntax: :SCOPe:WAVeform:TYPE?

Returned Format: [:SCOPe:WAVeform:TYPE]{NORmal|AVERage|ACCumulate}<NL>

Example:

```
10 DIM Wt$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:TYPE?"
30 ENTER XXX;Wt$
40 PRINT Wt$
50 END
```

VALid query

The VALid query checks the oscilloscope for acquired data. If a measurement is completed, and data has been acquired by all channels, then the query reports a 1. A 0 is reported if no data has been acquired for the last acquisition.

Query Syntax: :SCOPe:WAVeform:VALid?

Returned Format: [:SCOPe:WAVeform:VALid] {0|1}<NL>

where:

0 ::= No data acquired
1 ::= Data has been acquired

Example:

```
10 DIM Da$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:VALID?"
30 ENTER XXX;Da$
40 PRINT Da$
50 END
```

XINCrement query

The XINCrement query returns the X-increment currently in the preamble. This value is the time between the consecutive data points.

Query Syntax: :SCOPe:WAVeform:XINCrement?

Returned Format: [:SCOPe:WAVeform:XINCrement] <value> <NL>

where:

<value> ::= X-increment value currently in preamble

Example:

```
10 DIM Xi$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:XINCREMENT?"
30 ENTER XXX;Xi$
40 PRINT Xi$
50 END
```

XORigin

query

The XORigin query returns the X-origin value currently in the preamble. The value represents the time of the first data point in memory with respect to the trigger point.

Query Syntax: :SCOPe:WAVeform:XORigin?

Returned Format: [:SCOPe:WAVeform:XORigin] <value> <NL>

where:

<value> ::= X-origin value currently in preamble

Example:

```
10 DIM Xo$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:XORigin?"
30 ENTER XXX;Xo$
40 PRINT Xo$
50 END
```

XREFerence

query

The XREFerence query returns the X-reference value in the preamble. The value specifies the first data point in memory and is always 0.

Query Syntax: :SCOPe:WAVeform:XREFerence?

Returned Format: [:SCOPe:WAVeform:XREFerence] <value> <NL>

where:

<value> ::= X-reference value in preamble

Example:

```
10 DIM Xo$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:XREFerence?"
30 ENTER XXX;Xo$
40 PRINT Xo$
50 END
```

YINCrement

query

The YINCrement query returns the Y-increment currently in the preamble. This value is the voltage difference between consecutive data values.

Query Syntax: :SCOPe:WAVeform:YINCrement?

Returned Format: [:SCOPe:WAVeform:YINCrement]<value> <NL>

where:

<value> ::= Y-increment value currently in preamble

Example:

```
10 DIM Yi$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:YINCREMENT?"
30 ENTER XXX;Yi$
40 PRINT Yi$
50 END
```

YORigin

query

The YORigin query returns the Y-origin value currently in the preamble. This value is the voltage at the center of the screen.

Query Syntax: :SCOPe:WAVeform:YORigin?

Returned Format: [:SCOPe:WAVeform:YORigin]<value><NL>

where:

<value> ::= Y-origin value currently in preamble

Example:

```
10 DIM Yo$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:YORigin?"
30 ENTER XXX;Yo$
40 PRINT Yo$
50 END
```

YREFerence

query

The YREFerence query returns the Y-reference value in the preamble. The value specifies the data value at center screen where Y-origin occurs.

Query Syntax: :SCOPe:WAVeform:YREFerence?

Returned Format: [:SCOPe:WAVeform:YREFerence]<value><NL>

where:

<value> ::= Y-reference value in preamble

Example:

```
10 DIM Yo$[100]
20 OUTPUT XXX;":SCOPE:WAVEFORM:YREFerence?"
30 ENTER XXX;Yo$
40 PRINT Yo$
50 END
```

27 - MEASure Subsystem

MEASure Subsystem 27

Introduction

The instructions in the MEASure subsystem are used to make automatic parametric measurements on displayed waveforms. The instructions are:

- ALL
- FALLTime
- FREQuency
- NWIDth
- OVERShoot
- PERiod
- PRESHoot
- PWIDth
- RISETime
- SOURce
- VAMPlitude
- VBASe
- VMAX
- VMIN
- VPP
- VTOP

Before using any of the MEASure subsystem queries, be sure that you have used to SOURce command to specify which channel is to be used. All subsequent measurements will be made from that channel's waveform.

If a waveform characteristic cannot be measured, the instrument responds with 9.9E+37.

The following characteristics can be measured:

Frequency The frequency of the first complete cycle displayed is measured using the 50% level.

Period The period of the first displayed waveform is measured at the 50% level.

Peak-to-Peak The absolute minimum and the maximum voltages for the selected source are measured.

Positive Pulse Width Pulse width is measured at the 50% level of the first displayed pulse.

Negative Pulse Width Pulse width is measured at the 50% level of the first displayed pulse.

Risetime The risetime of the first displayed rising edge is measured. To obtain the best possible measurement accuracy, select the fastest sweep speed while keeping the rising edge on the display. The risetime is determined by measuring time at the 10% and the 90% voltage points of the rising edge.

Falltime Falltime is measured between the 10% and the 90% points of the first displayed falling edge. To obtain the best possible measurement accuracy, select the fastest sweep speed possible while keeping the falling edge on the display.

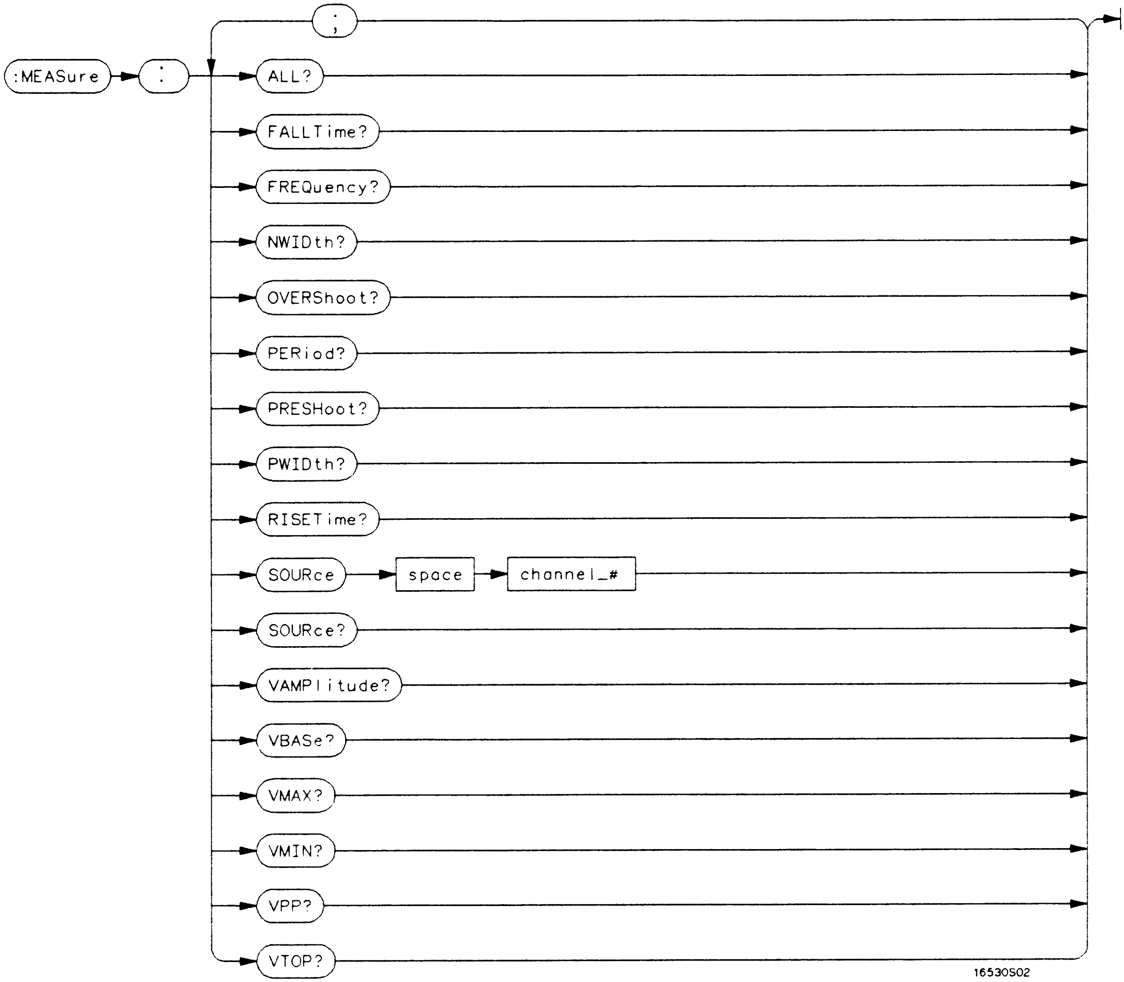
Preshoot and Overshoot Preshoot and overshoot measure the perturbation on a waveform above or below the top and base voltages.

Preshoot is a perturbation before a rising or a falling edge and measured as a percentage of the top-base voltage.

Overshoot is a perturbation after a rising or falling edge and is measured as a percentage of the top-base voltage.

For complete details of the measurement algorithms, refer to the *Front-panel Operating Reference Manual.*

Refer to figure 27-1 for the MEASure subsystem syntax diagram.



channel_# = *an integer* {*1* | *2*}.

Figure 27-1. MEASure Subsystem Syntax Diagram

MEASure

selector

The MEASure selector is used as part of a compound command header to access the settings found in oscilloscope's Measure menu. It always follows the SCOPe selector because it selects a branch below the SCOPe level in the command tree.

Command Syntax: :SCOPe:MEASure

Example:

```
OUTPUT XXX; ":SCOPE:MEASURE:SOURCE CHAN2"
```

Note

All queries in this subsystem return the measurement results of the last channel specified by the SOURce command. If you want measurement results from the other channel, you must use the SOURce command before using any of the queries.

ALL

query

The ALL query makes a set of measurements on the displayed waveform using the selected source.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]ALL?

Returned Format:

[:SCOPe:MEASure:ALL PERiod] <real number>;
[RISETime] <real number>;
[FALLTime] <real number>;
[FREQuency] <real number>;
[PWIDtH] <real number>;
[NWIDtH] <real number>;
[VPP] <real number>;
[VAMPlitude] <real number>;
[PRESHoot] <real number>;
[OVERShoot] <real number> <NL>

Example:

```
10 DIM Query$[300]
20 !PRINTER IS 701 !THIS LINE SENDS RESULTS TO PRINTER
30 OUTPUT XXX;":SCOPE:MEASURE:SOUR CHAN1"
40 OUTPUT XXX;":SCOPE:MEASURE:ALL?"
50 ENTER XXX;Query$
60 Query$=Query$[POS(Query$,";")+1]
70 LOOP
80 I=POS(Query$,";")
90 EXIT IF NOT I
100 PRINT Query$[1,I-1]
110 Query$=Query$[I+1]
120 END LOOP
130 PRINT Query$
140 PRINTER IS 1
150 END
```

FALLTime

query

The FALLTime query makes a fall time measurement on the selected channel. The measurement is made between the 90% to the 10% voltage point of the first falling edge displayed on screen.

Note The short form of FALLTIME is FALLTime. This is an intentional deviation of the normal truncation rule.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]FALLTime?

Returned Format: [:SCOPe:MEASure:FALLTime] <value> <NL>

where:

<value> ::= time in seconds between 10% and 90% voltage points

Example:

```
10 DIM Ft$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHANNEL2;FALLTIME?"
30 ENTER XXX;Ft$
40 PRINT Ft$
50 END
```

FREQuency

query

The FREQency query makes a frequency measurement on the selected channel. The measurement is made using the first complete displayed cycle at the 50% voltage level.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]FREQuency?

Returned Format: [:MEAsure:FREQuency] <value> <NL>

where:

<value> ::= frequency in Hertz

Example:

```
10 DIM Frcy$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOUR CHAN1;FREQ?"
30 ENTER XXX;Frcy$
40 PRINT Frcy$
50 END
```

NWIDth

query

The NWIDth query makes a negative width time measurement on the selected channel. The measurement is made between the 50% points of the first falling and the next rising edge displayed on screen.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]NWIDth?

Returned Format: [:SCOPe:MEASure:NWIDth] <value><NL>

where:

<value> ::= negative pulse width in seconds

Example:

```
10 DIM Nw$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHAN2;NWID?"
30 ENTER XXX;Nw$
40 PRINT Nw$
50 END
```

OVERShoot

query

The OVERShoot query makes an overshoot measurement on the selected channel. The measurement is made by finding a distortion following the first major transition. The result is the ratio of VMAX or VMIN vs. VAMPlitude.

Note

The short form of OVERSHOOT is OVERShoot. This is an intentional deviation from the normal truncation rule.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]OVERShoot?

Returned Format: [:SCOPe:MEASure:OVERShoot] <value> <NL>

where:

<value> ::= ratio of overshoot to Vamplitude

Example:

```
10 DIM Ovs$[100]
20 OUTPUT XXX;":SCOPE:MEASURE SOURCE CHAN1;OVER?"
30 ENTER XXX;Ovs$
40 PRINT Ovs$
50 END
```

PERiod

query

The PERiod query makes a period measurement on the selected channel. The measurement equivalent to the inverse of frequency.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]PERiod?

Returned Format: [:SCOPe:MEASure:PERiod] <value><NL>

where:

<value> ::= waveform period in seconds

Example:

```
10 DIM Pd$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHANNEL1;PERIOD?"
30 ENTER XXX;Pd$
40 PRINT Pd$
50 END
```

PRESHoot

query

The PRESHoot query makes the preshoot measurement on the selected channel. The measurement is made by finding a distortion which precedes the first major transition on screen. The result is the ratio of VMAX or VMIN vs. VAMPlitude.

Note The short form of PRESHOOT is PRESHoot. This is an intentional deviation of the normal truncation rule.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]PRESHoot?

Returned Format: [:SCOPe:MEASure:PRESHoot] <value><NL>

where:

<value> ::= ratio of preshoot to Vamplitude

Example:

```
10 DIM Prs$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:CHANNEL2;PRESH?"
30 ENTER XXX;Prs$
40 PRINT Prs$
50 END
```

PWIDth query

The PWIDth query makes a positive pulse width measurement on the selected channel. The measurement is made by finding the time difference between the 50% points of the first rising and the next falling edge displayed on screen.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]PWIDth?

Returned Format: [:SCOPe:MEASure:PWIDth] <value> <NL>

where:

<value> ::= positive pulse width in seconds

Example:

```
10 DIM Pw$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHANNEL2;PWIDTH?"
30 ENTER XXX;Pw$
40 PRINT Pw$
50 END
```

RISETime

query

The RISETime query makes a risetime measurement on the selected channel by finding the 10% and 90% voltage levels of the first rising edge displayed on screen.

Note

The short form of RISETIME is RISETime. This is an intentional deviation from the normal truncation rule.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]RISETime?

Returned Format: [:SCOPe:MEASure:RISETime] <value> <NL>

where:

<value> ::= risetime in seconds

Example:

```
10 DIM Tr$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHANNEL1;RISETIME?"
30 ENTER XXX;Tr$
40 PRINT Tr$
50 END
```

SOURce

command/query

The SOURce command specifies the source to be used for subsequent measurements. If the source is not specified, the last waveform source is assumed.

The query returns the presently specified channel.

Command Syntax: :SCOPe:MEASure:SOURce <source>

where:

<source> ::= {1 | 2}

Example:

```
OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHAN1"
```

Query Syntax: :SCOPe:MEASure:SOURce?

Returned Format: [:SCOPe:MEASure:SOURce] CHANnel<N><NL>

Example:

```
10 DIM So$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE?"
30 ENTER XXX;So$
40 PRINT So$
50 END
```

VAMPlitude

query

The VAMPlitude query makes a voltage measurement on the selected channel. The measurement is made by finding the relative maximum and minimum points on screen.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]VAMPlitude?

Returned Format: [:SCOPe:MEASure:VAMPlitude] <value><NL>

where:

<value> ::= difference between top and base voltage

Example:

```
10 DIM Va$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHANNEL2;VAMP?"
30 ENTER XXX;Va$
40 PRINT Va$
50 END
```

VBASe query

The VBASe query returns the base voltage (relative minimum) of a displayed waveform. The measurement is made on the selected source.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]VBASe?

Returned Format: [:SCOPe:MEASure:VBASe] <value><NL>

where:

<value> ::= voltage at base level of selected waveform

Example:

```
10 DIM Vb$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHAN1;VBAS?"
30 ENTER XXX;Vb$
40 PRINT Vb$
50 END
```

VMAX

query

The VMAX query returns the absolute maximum voltage of the selected source.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]VMAX?

Returned Format: [:SCOPe:MEASure:VMAX] <value> <NL>

where:

<value> ::= maximum voltage of selected waveform

Example:

```
10 DIM Vma$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHAN2;VMAX?"
30 ENTER XXX;Vma$
40 PRINT Vma$
50 END
```

VMIN

query

The VMIN query returns the absolute minimum voltage present on the selected source.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]VMIN?

Returned Format: [:SCOPe:MEASure VMIN] <value> <NL>

where:

<value> ::= minimum voltage of selected waveform

Example:

```
10 DIM Vmi$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHAN1;VMIN?"
30 ENTER XXX;Vmi$
40 PRINT Vmi$
50 END
```

VPP

query

The VPP query makes a peak-to-peak voltage measurement on the selected source. The measurement is made by finding the absolute maximum and minimum points on the displayed waveform.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]VPP?

Returned Format: [:SCOPe:MEASure:VPP] <value> <NL>

where:

<value> ::= peak to peak voltage of selected waveform

Example:

```
10 DIM Vpp$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHAN1;VPP?"
30 ENTER XXX;Vpp$
40 PRINT Vpp$
50 END
```

VTOP

query

The VTOP query returns the voltage at the top (relative maximum) of waveform on the selected source.

Query Syntax: :SCOPe:MEASure:[SOURce CHANnel{1|2};]VTOP?

Returned Format: [:SCOPe:MEASure:VTOP] <value> <NL>

where:

<value> ::= voltage at the top of the selected waveform

Example:

```
10 DIM Vt$[100]
20 OUTPUT XXX;":SCOPE:MEASURE:SOURCE CHAN2;VTOP?"
30 ENTER XXX;Vt$
40 PRINT Vt$
50 END
```

A - Message Communication
and System Functions

Message Communication and System Functions

A

Introduction

This appendix describes the operation of instruments that operate in compliance with the IEEE 488.2 (syntax) standard. Although the HP 1652B and HP 1653B logic analyzers are RS-232C instruments, they were designed to be compatible with other Hewlett-Packard IEEE 488.2 compatible instruments.

The IEEE 488.2 standard is a new standard. Instruments that are compatible with IEEE 488.2 must also be compatible with IEEE 488.1 (HP-IB bus standard); however, IEEE 488.1 compatible instruments may or may not conform to the IEEE 488.2 standard. The IEEE 488.2 standard defines the message exchange protocols by which the instrument and the controller will communicate. It also defines some common capabilities, which are found in all IEEE 488.2 instruments. This appendix also contains a few items which are not specifically defined by IEEE 488.2, but deal with message communication or system functions.

Note

The syntax and protocol for RS-232C program messages and response messages for the HP 1652B/1653B are structured very similar to those described by 488.2. In most cases, the same structure shown in this appendix for 488.2 will also work for RS-232C. Because of this, no additional information has been included for RS-232C.

Protocols

The protocols of IEEE 488.2 define the overall scheme used by the controller and the instrument to communicate. This includes defining when it is appropriate for devices to talk or listen, and what happens when the protocol is not followed.

Functional Elements

Before proceeding with the description of the protocol, a few system components should be understood.

Input Buffer. The input buffer of the instrument is the memory area where commands and queries are stored prior to being parsed and executed. It allows a controller to send a string of commands to the instrument which could take some time to execute, and then proceed to talk to another instrument while the first instrument is parsing and executing commands.

Output Queue. The output queue of the instrument is the memory area where all output data (<response messages>) are stored until read by the controller.

Parser. The instrument's parser is the component that interprets the commands sent to the instrument and decides what actions should be taken. "Parsing" refers to the action taken by the parser to achieve this goal. Parsing and executing of commands begins when either the instrument recognizes a <program message terminator> (defined later in this appendix) or the input buffer becomes full. If you wish to send a long sequence of commands to be executed and then talk to another instrument while they are executing, you should send all the commands before sending the <program message terminator>.

Protocol Overview

The instrument and controller communicate using <program message>s and <response message>s. These messages serve as the containers into which sets of program commands or instrument responses are placed. <program message>s are sent by the controller to the instrument, and <response message>s are sent from the instrument to the controller in response to a query message. A <query message> is defined as being a <program message> which contains one or more queries. The instrument will only talk when it has received a valid query message, and therefore has something to say. The controller should only attempt to read a response after sending a complete query message, but before sending another <program message>. The basic rule to remember is that the instrument will only talk when prompted to, and it then expects to talk before being told to do something else.

Protocol Operation

When the instrument is turned on, the input buffer and output queue are cleared, and the parser is reset to the root level of the command tree.

The instrument and the controller communicate by exchanging complete <program message>s and <response message>s. This means that the controller should always terminate a <program message> before attempting to read a response. The instrument will terminate <response message>s except during a hardcopy output.

If a query message is sent, the next message passing over the bus should be the <response message>. The controller should always read the complete <response message> associated with a query message before sending another <program message> to the same instrument.

The instrument allows the controller to send multiple queries in one query message. This is referred to as sending a "compound query." As will be noted later in this appendix, multiple queries in a query message are separated by semicolons. The responses to each of the queries in a compound query will also be separated by semicolons.

Commands are executed in the order they are received.

Protocol Exceptions

If an error occurs during the information exchange, the exchange may not be completed in a normal manner. Some of the protocol exceptions are shown below.

Command Error. A command error will be reported if the instrument detects a syntax error or an unrecognized command header.

Execution Error. An execution error will be reported if a parameter is found to be out of range, or if the current settings do not allow execution of a requested command or query.

Device-specific Error. A device-specific error will be reported if the instrument is unable to execute a command for a strictly device dependent reason.

Query Error. A query error will be reported if the proper protocol for reading a query is not followed. This includes the interrupted and unterminated conditions described in the following paragraphs.

Syntax Diagrams

The syntax diagrams in this appendix are similar to the syntax diagrams in the IEEE 488.2 specification. Commands and queries are sent to the instrument as a sequence of data bytes. The allowable byte sequence for each functional element is defined by the syntax diagram that is shown with the element description.

The allowable byte sequence can be determined by following a path in the syntax diagram. The proper path through the syntax diagram is any path that follows the direction of the arrows. If there is a path around an element, that element is optional. If there is a path from right to left around one or more elements, that element or those elements may be repeated as many times as desired.

Syntax Overview

This overview is intended to give a quick glance at the syntax defined by IEEE 488.2. It should allow you to understand many of the things about the syntax you need to know. This appendix also contains the details of the IEEE 488.2 defined syntax.

IEEE 488.2 defines the blocks used to build messages which are sent to the instrument. A whole string óf commands can therefore be broken up into individual components.

Figure A-1 shows a breakdown of an example <program message>. There are a few key items to notice:

1. A semicolon separates commands from one another. Each <program message unit> serves as a container for one command. The <program message unit>s are separated by a semicolon.

2. A <program message> is terminated by a <NL> (new line). The recognition of the <program message terminator>, or <PMT>, by the parser serves as a signal for the parser to begin execution of commands. The <PMT> also affects command tree traversal (see the Programming and Documentation Conventions chapter).

3. Multiple data parameters are separated by a comma..

4. The first data parameter is separated from the header with one or more spaces.

5. The header MACHINE1:ASSIGN 2,3 is an example of a compound header. It places the parser in the machine subsystem until the <NL> is encountered.

6. A colon preceding the command header returns you to the top of the command tree.

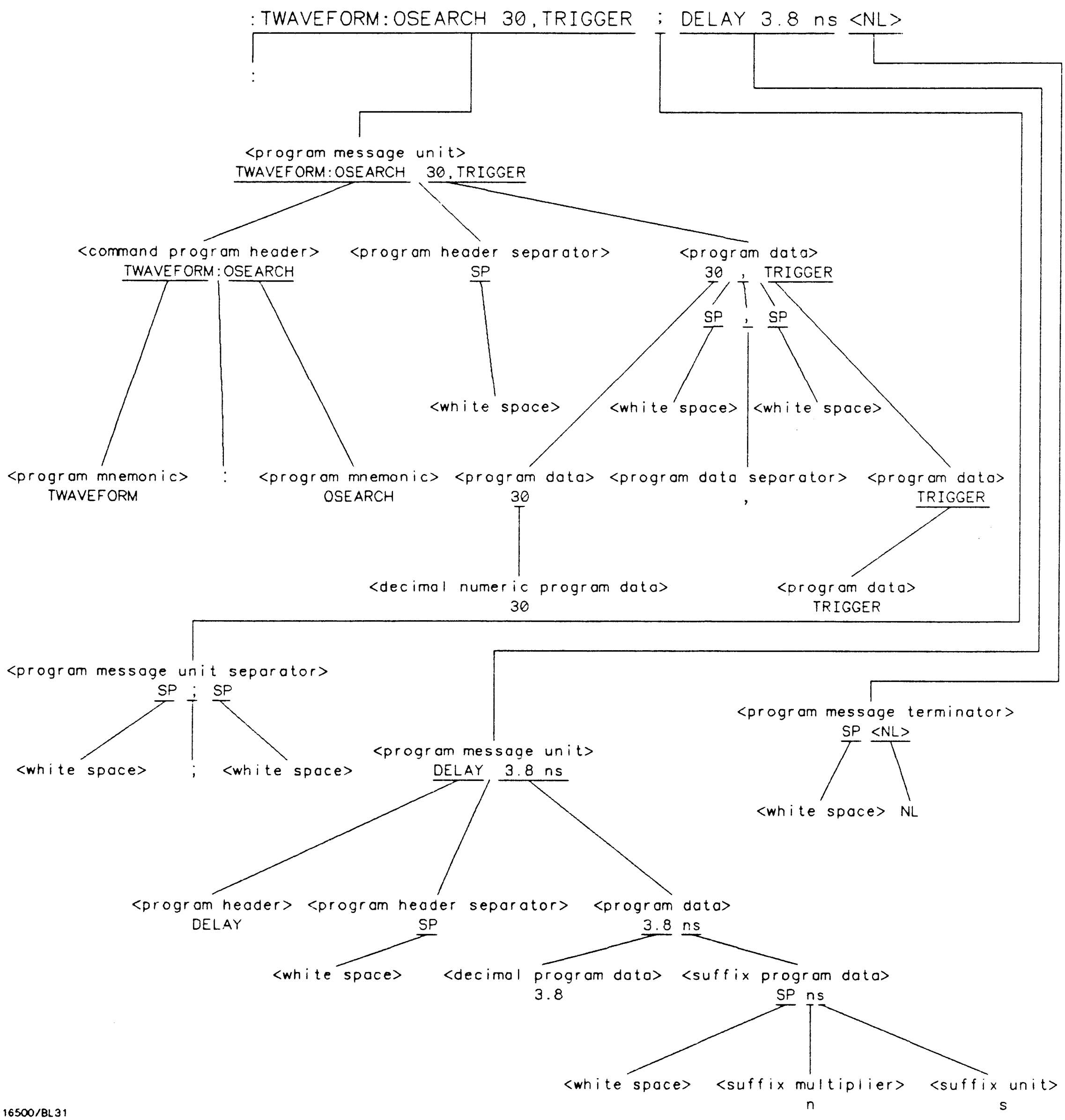


Figure A-1. <program message> Parse Tree

Device Listening Syntax

The listening syntax of IEEE 488.2 is designed to be more forgiving than the talking syntax. This allows greater flexibility in writing programs, as well as allowing them to be easier to read.

Upper/Lower Case Equivalence. Upper and lower case letters are equivalent. The mnemonic SINGLE has the same semantic meaning as the mnemonic single.

<white space>. <white space> is defined to be one or more characters from the ASCII set of 0 - 32 decimal, excluding 10 decimal (NL). <white space> is used by several instrument listening components of the syntax. It is usually optional, and can be used to increase the readability of a program.

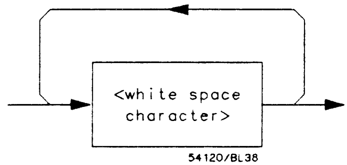


Figure A-2. <white space>

<program message>. The <program message> is a complete message to be sent to the instrument. The instrument will begin executing commands once it has a complete <program message>, or when the input buffer becomes full. The parser is also repositioned to the root of the command tree after executing a complete <program message>. Refer to "Tree Traversal Rules" in the "Programming and Documentation Conventions," chapter 4 for more details.

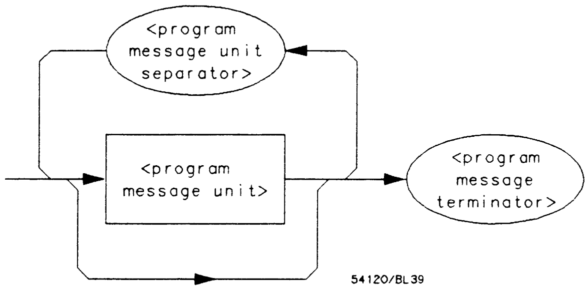


Figure A-3. <program message>

<program message unit>. The <program message unit> is the container for individual commands within a <program message>.

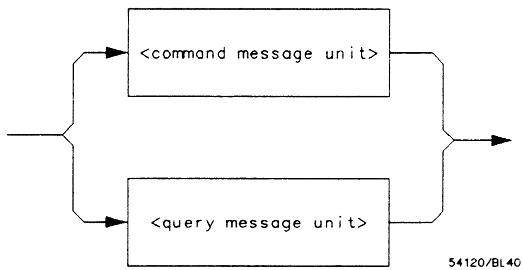


Figure A-4. <program message unit>

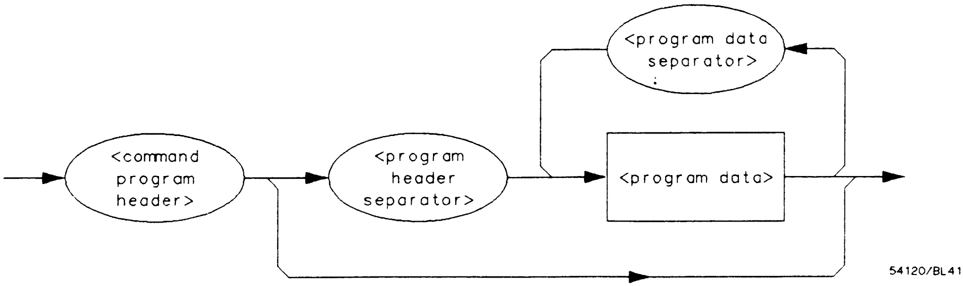


Figure A-5. <command message unit>

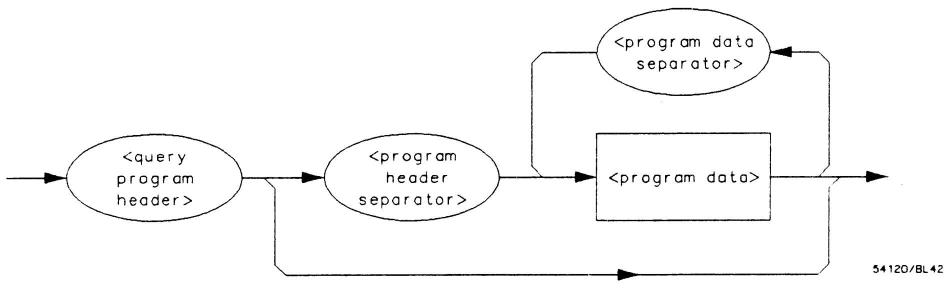


Figure A-6. <query message unit>

<program message unit separator>. A semicolon separates <program message unit>s, or individual commands.

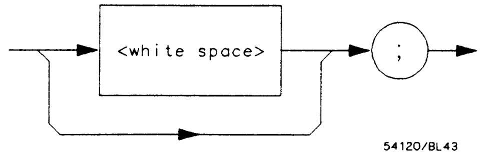


Figure A-7. <program message unit separator>

<command program header>/<query program header>. These elements serve as the headers of commands or queries. They represent the action to be taken.

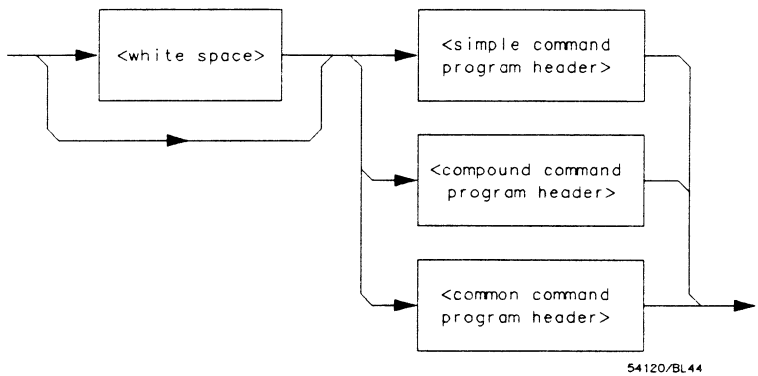
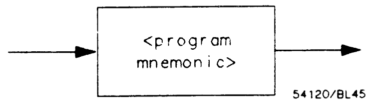
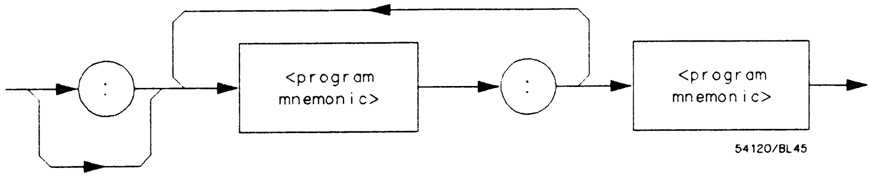


Figure A-8. <command program header>

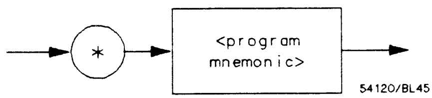
Where <simple command program header> is defined as



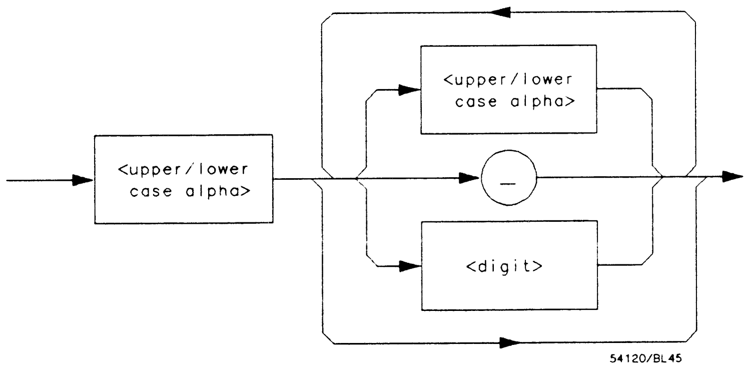
Where <compound command program header> is defined as



Where <common command program header> is defined as



Where <program mnemonic> is defined as

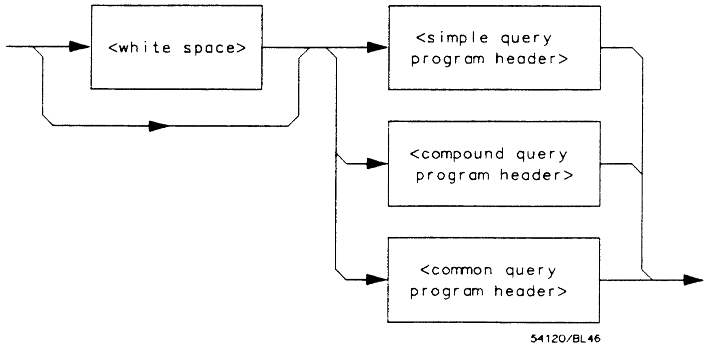


Where <upper/lower case alpha> is defined as a single ASCII encoded byte in the range 41 - 5A, 61 - 7A (65 - 90, 97 - 122 decimal).

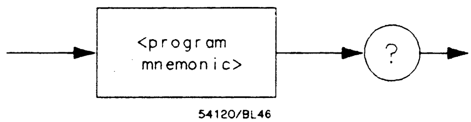
Where <digit> is defined as a single ASCII encoded byte in the range 30 - 39 (48 - 57 decimal).

Where (_) represents an "underscore", a single ASCII-encoded byte with the value 5F (95 decimal).

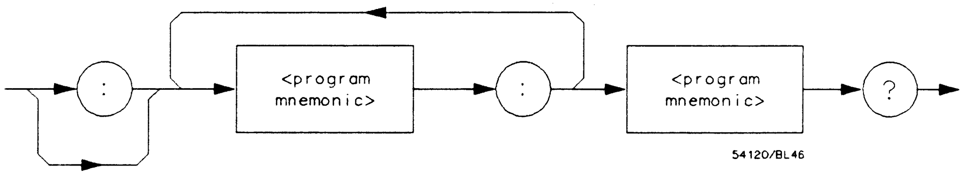
Figure A-8. <command program header> (continued)



Where <simple query program header> is defined as



Where <compound query program header> is defined as



Where <common query program header> is defined as

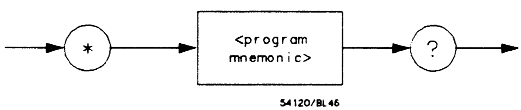


Figure A-9. <query program header>

<program data>. The <program data> element represents the possible types of data which may be sent to the instrument. The HP 1652B/1653B will accept the following data types: <character program data>, <decimal numeric program data>, <suffix program data>, <string program data>, and <arbitrary block program data>.

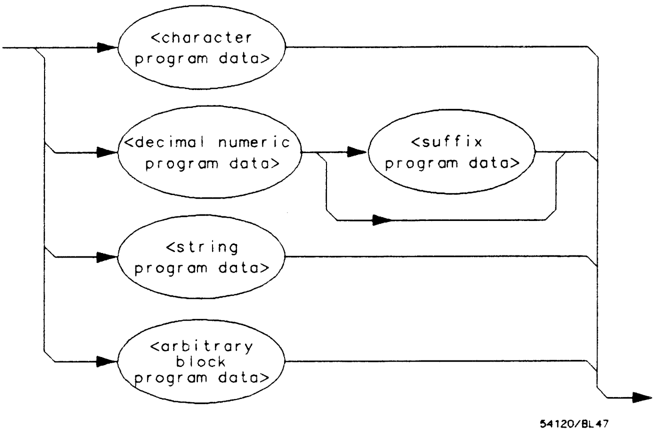


Figure A-10. <program data>

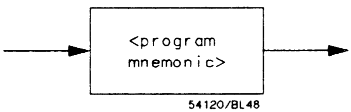
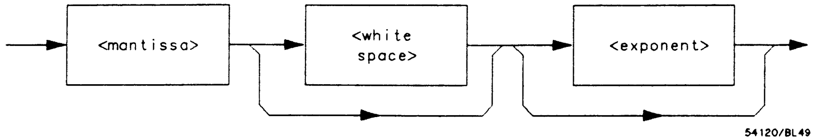
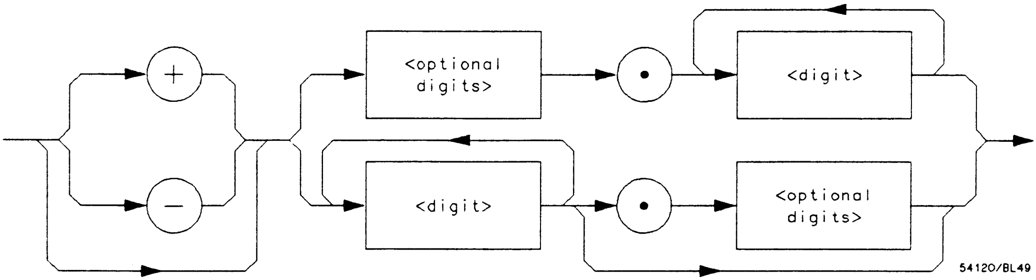


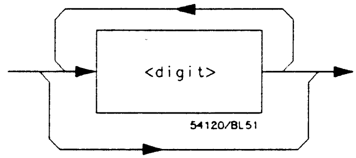
Figure A-11. <character program data>



Where <mantissa> is defined as



Where <optional digits> is defined as



Where <exponent> is defined as

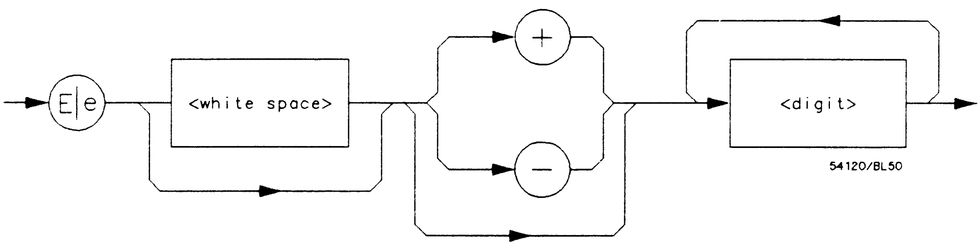


Figure A-12. <decimal numeric program data>

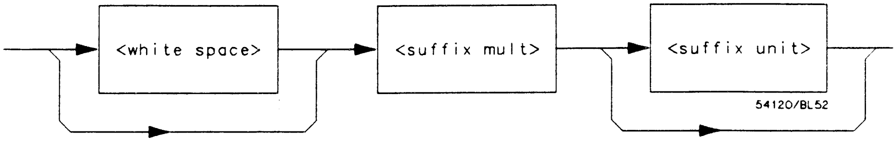


Figure A-13. <suffix program data>

Suffix Multiplier. The suffix multipliers that the instrument will accept are shown in table A-1.

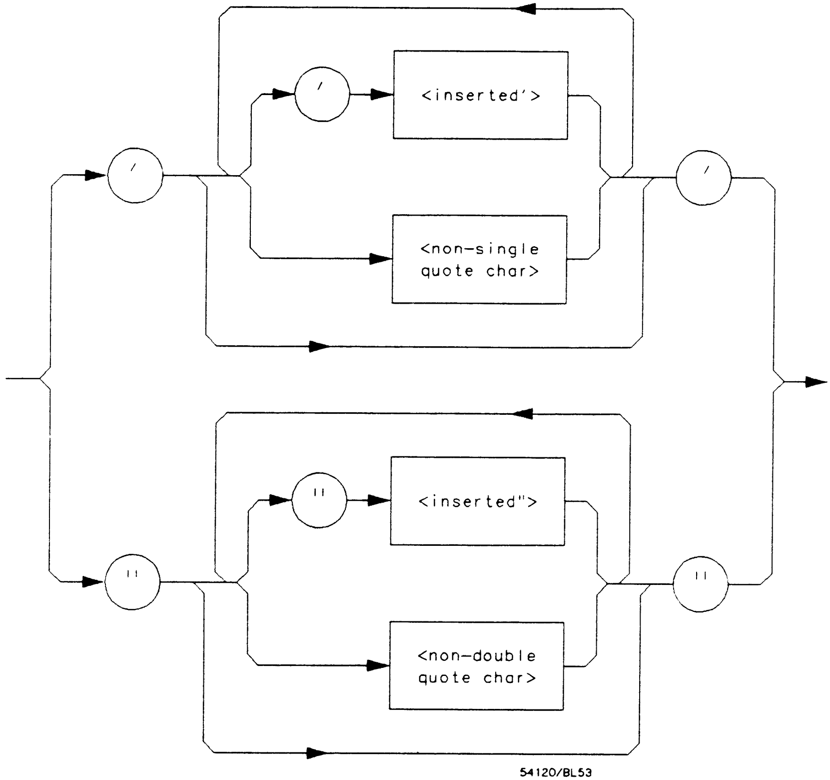
Table A-1. <suffix mult>

Value	Mnemonic
1E18	EX
1E15	PE
1E12	T
1E9	G
1E6	MA
1E3	K
1E-3	M
1E-6	U
1E-9	N
1E-12	P
1E-15	F
1E-18	A

Suffix Unit. The suffix units that the instrument will accept are shown in table A-2.

Table A-2. <suffix unit>

Suffix	Referenced Unit
V	Volt
S	Second



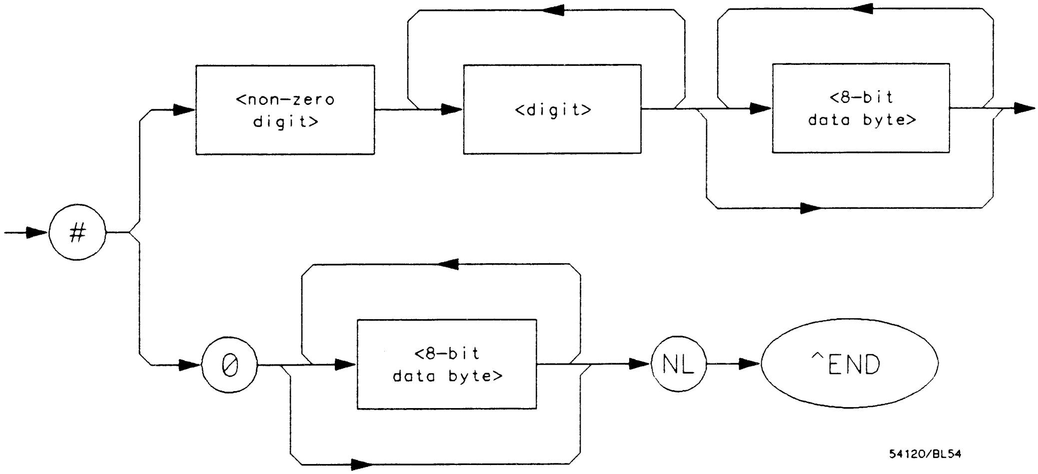
Where <inserted '> is defined as a single ASCII character with the value 27 (39 decimal).

Where <non-single quote char> is defined as a single ASCII character of any value except 27 (39 decimal).

Where <inserted "> is defined as a single ASCII character with the value 22 (34 decimal).

Where <non-double quote char> is defined as a single ASCII character of any value except 22 (34 decimal)

Figure A-14. <string program data>



Where <non-zero digit> is defined as a single ASCII encoded byte in the range 31 - 39 (49 - 57 decimal).

Where <8-bit byte> is defined as an 8-bit byte in the range 00 - FF (0 - 255 decimal).

Figure A-15. <arbitrary block program data>

<program data separator>. A comma separates multiple data parameters of a command from one another.

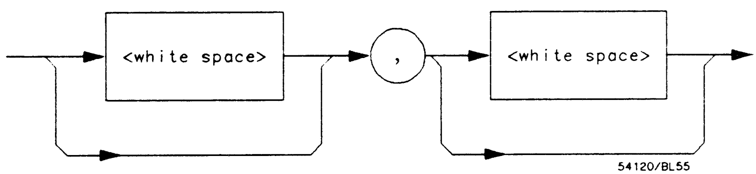


Figure A-16. <program data separator>

<program header separator>. A space separates the header from the first or only parameter of the command.

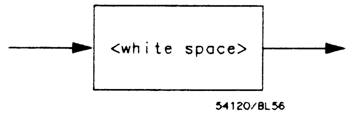
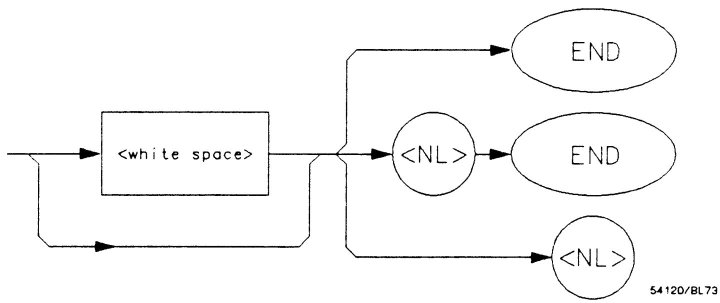


Figure A-17. <program header separator>

<program message terminator>. The <program message terminator> or <PMT> serves as the terminator to a complete <program message>. When the parser sees a complete <program message> it will begin execution of the commands within that message. The <PMT> also resets the parser to the root of the command tree.



Where <NL> is defined as a single ASCII-encoded byte 0A (10 decimal).

Figure A-18. <program message terminator>

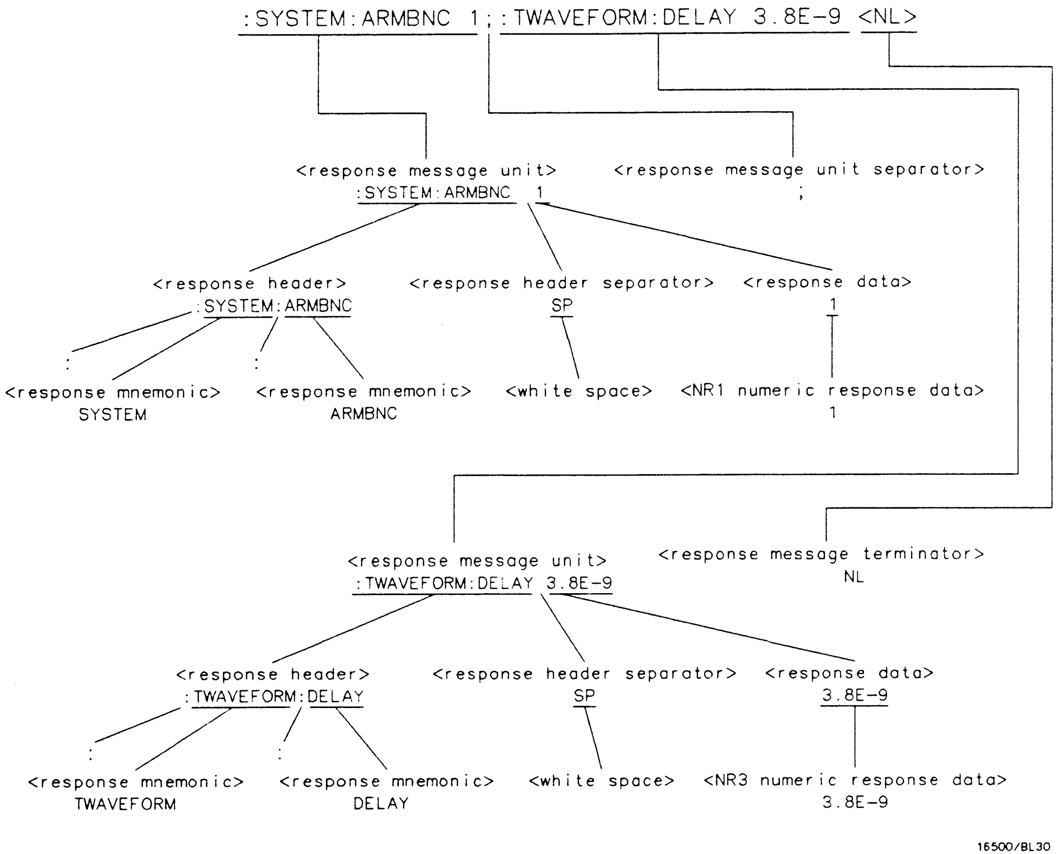


Figure A-19. <response message> Tree

Device Talking Syntax

The talking syntax of IEEE 488.2 is designed to be more precise than the listening syntax. This allows the programmer to write routines which can easily interpret and use the data the instrument is sending. One of the implications of this is the absence of <white space> in the talking formats. The instrument will not pad messages which are being sent to the controller with spaces.

<response message>. This element serves as a complete response from the instrument. It is the result of the instrument executing and buffering the results from a complete <program message>. The complete <response message> should be read before sending another <program message> to the instrument.

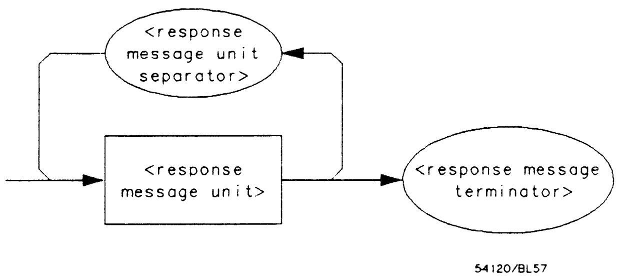
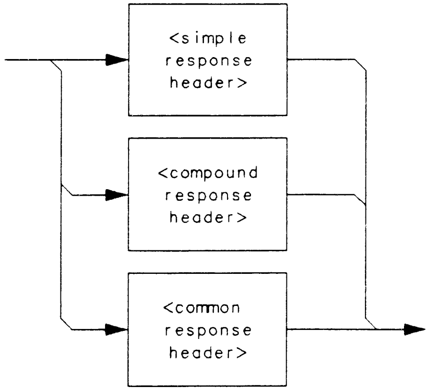


Figure A-20. <response message>

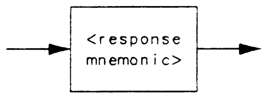
<response message unit>. This element serves as the container of individual pieces of a response. Typically a <query message unit> will generate one <response message unit>, although a <query message unit> may generate multiple <response message unit>s.

<response header>. The <response header>, when returned, indicates what the response data represents.



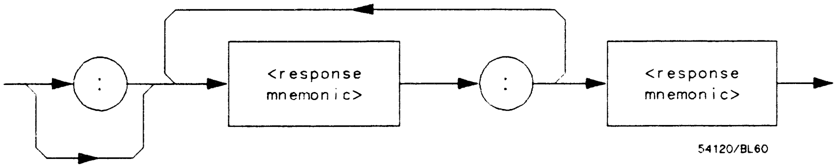
54120/BL58

Where <simple response mnemonic> is defined as

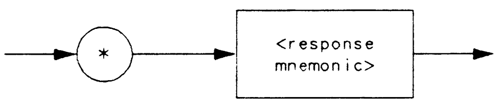


54120/BL59

Where <compound response header> is defined as



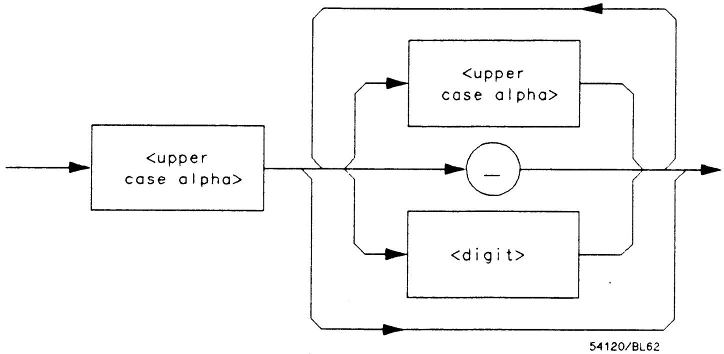
Where <common response header>' is defined as



54120/BL61

Figure A-21. <response message unit>

Where <response mnemonic> is defined as



Where <uppercase alpha> is defined as a single ASCII encoded byte in the range 41 - 5A (65 - 90 decimal).

Where (_) represents an "underscore", a single ASCII-encoded byte with the value 5F (95 decimal).

Figure A-21. <response message unit> (Continued)

<response data>. The <response data> element represents the various types of data which the instrument may return. These types include: <character response data>, <nr1 numeric response data>, <nr3 numeric response data>, <string response data>, <definite length arbitrary block response data>, and <arbitrary ASCII response data>.

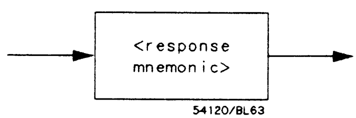


Figure A-22. <character response data>

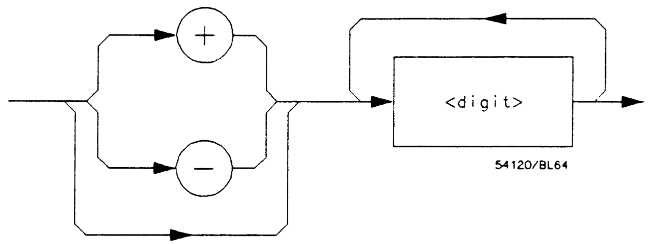


Figure A-23. <nr1 numeric response data>

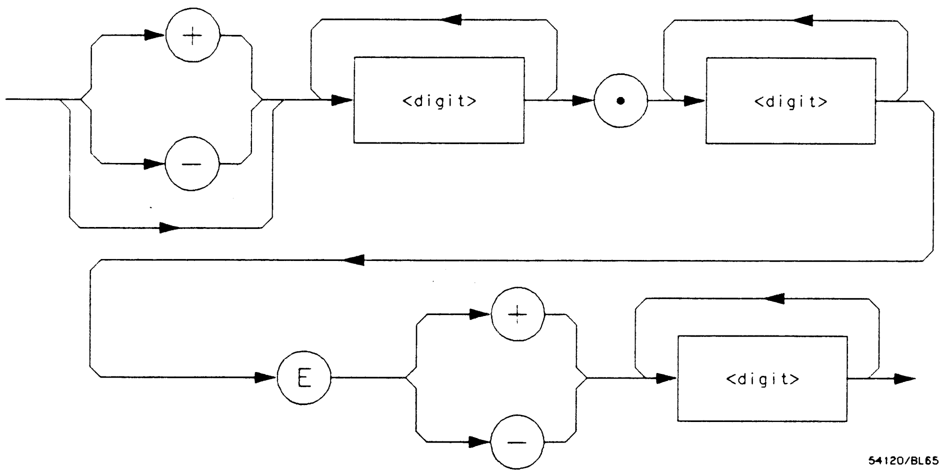


Figure A-24. <nr3 numeric response data>

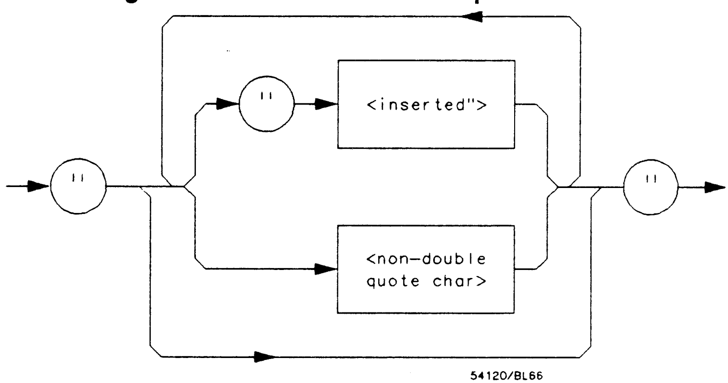


Figure A-25. <string response data>

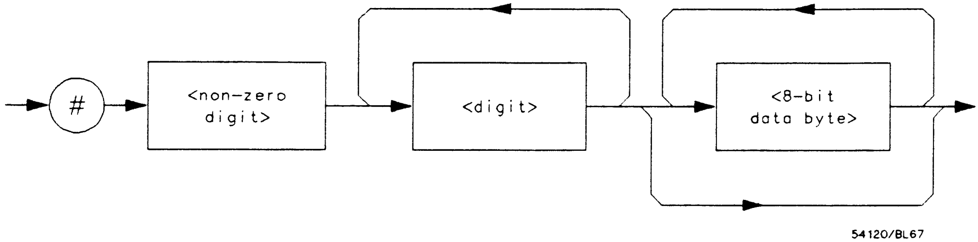
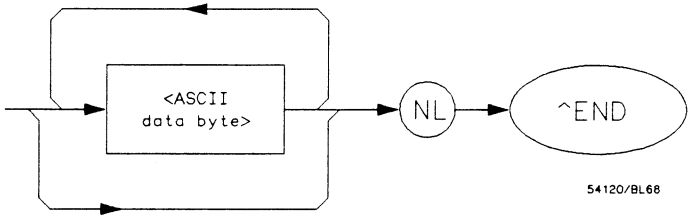


Figure A-26. <definite length arbitrary block response data>



Where <ASCII data byte> represents any ASCII-encoded data byte except <NL> (0A, 10 decimal).

Notes

1. The END message provides an unambiguous termination to an element that contains arbitrary ASCII characters.

2. The IEEE 488.1 END message serves the dual function of terminating this element as well as terminating the <RESPONSE MESSAGE>. It is only sent once with the last byte of the indefinite block data. The NL is present for consistency with the <RESPONSE MESSAGE TERMINATOR>. Indefinite block data format is not supported in the HP 1652B/1653B.

Figure A-27. <arbitrary ASCII response data>

<response data separator>. A comma separates multiple pieces of response data within a single <response message unit>.

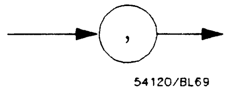


Figure A-28. <response data separator>

<response header separator>. A space (ASCII decimal 32) delimits the response header, if returned, from the first or only piece of data.

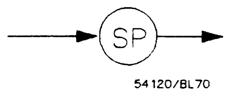


Figure A-29. <response header separator>

<response message unit separator>. A semicolon delimits the <response message unit>s if multiple responses are returned.

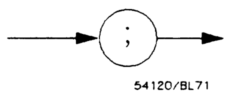


Figure A-30. <response message unit separator>

<response message terminator>. A <response message terminator> (NL) terminates a complete <response message>. It should be read from the instrument along with the response itself.

Common Commands

IEEE 488.2 defines a set of common commands. These commands perform functions which are common to any type of instrument. They can therefore be implemented in a standard way across a wide variety of instrumentation. All the common commands of IEEE 488.2 begin with an asterisk. There is one key difference between the IEEE 488.2 common commands and the rest of the commands found in this instrument. The IEEE 488.2 common commands do not affect the parser's position within the command tree. More information about the command tree and tree traversal can be found in the Programming and Documentation Conventions chapter.

Table A-3. HP 1652B/53B's Common Commands

Command	Command Name
*CLS	Clear Status Command
*ESE	Event Status Enable Command
*ESE?	Event Status Enable Query
*ESR?	Event Status Register Query
*IDN?	Identification Query
*OPC	Operation Complete Command
*OPC?	Operation Complete Query
*RST	Reset (not implemented on HP 1652B/1653B)
*SRE	Service Request Enable Command
*SRE?	Service Request Enable Query
*STB?	Read Status Byte Query
*WAI	Wait-to-Continue Command

B - Status Reporting

Status Reporting B

Introduction

The status reporting feature available over the bus is the serial poll. IEEE 488.2 defines data structures, commands, and common bit definitions. There are also instrument defined structures and bits.

The bits in the status byte act as summary bits for the data structures residing behind them. In the case of queues, the summary bit is set if the queue is not empty. For registers, the summary bit is set if any enabled bit in the event register is set. The events are enabled via the corresponding event enable register. Events captured by an event register remain set until the register is read or cleared. Registers are read with their associated commands. The "*CLS" command clears all event registers and all queues except the output queue. If "*CLS" is sent immediately following a <program message terminator> , the output queue will also be cleared.

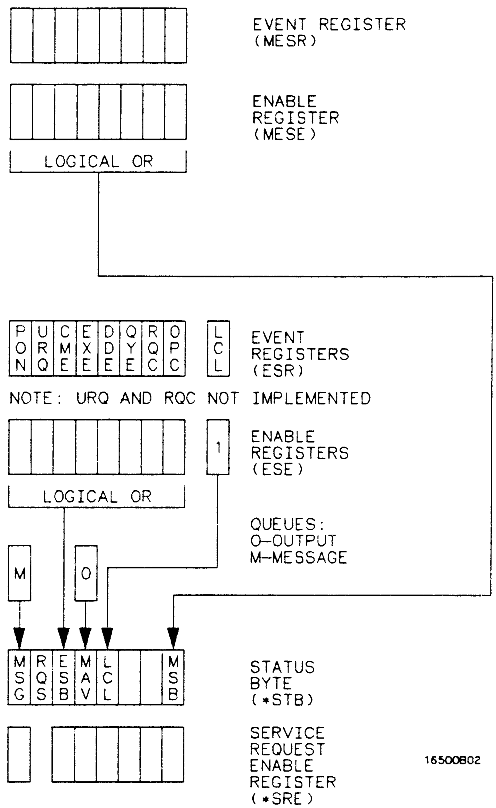


Figure B-1. Status Byte Structures and Concepts

Event Status Register The Event Status Register is a 488.2 defined register. The bits in this register are "latched." That is, once an event happens which sets a bit, that bit will only be cleared if the register is read.

Service Request Enable Register The Service Request Enable Register is an 8-bit register. Each bit enables the corresponding bit in the status byte to cause a service request. The sixth bit does not logically exist and is always returned as a zero. To read and write to this register use the *SRE? and *SRE commands.

Bit Definitions The following mnemonics are used in figure B-1 and in the "Common Commands" chapter:

MAV - message available. Indicates whether there is a response in the output queue.

ESB - event status bit. Indicates if any of the conditions in the Standard Event Status Register are set and enabled.

MSS - master summary status. Indicates whether the device has a reason for requesting service. This bit is returned for the *STB? query.

RQS - request service. Indicates if the device is requesting service. This bit is returned during a serial poll. RQS will be set to 0 after being read via a serial poll (MSS is not reset by *STB?).

MSG - message. Indicates whether there is a message in the message queue (Not implemented in the HP 1652B/1653B).

PON - power on. Indicates power has been turned on.

URQ - user request. Always 0 on the HP 1652B/1653B.

CME - command error. Indicates whether the parser detected an error.

Note The error numbers and/or strings for CME, EXE, DDE, and QYE can be read from a device defined queue (which is not part of 488.2) with the query :SYSTEM:ERROR?.

EXE - execution error. Indicates whether a parameter was out of range, or inconsistent with current settings.

DDE - device specific error. Indicates whether the device was unable to complete an operation for device dependent reasons.

QYE - query error. Indicates whether the protocol for queries has been violated.

RQC - request control. Always 0 on the HP 1652B/1653B.

OPC - operation complete. Indicates whether the device has completed all pending operations. OPC is controlled by the *OPC common command. Because this command can appear after any other command, it serves as a general purpose operation complete message generator.

LCL - remote to local. Indicates whether a remote to local transition has occurred.

MSB - module summary bit. Indicates that an enable event in one of the modules Status registers has occurred.

Key Features

A few of the most important features of Status Reporting are listed in the following paragraphs.

Operation Complete. The IEEE 488.2 structure provides one technique which can be used to find out if any operation is finished. The *OPC command, when sent to the instrument after the operation of interest, will set the OPC bit in the Standard Event Status Register. If the OPC bit and the RQS bit have been enabled a service request will be generated. The commands which affect the OPC bit are the overlapped commands.

```
OUTPUT XXX;"*SRE 32 ; *ESE 1" !enables an OPC service request
```

Status Byte. The Status Byte contains the basic status information which is sent over the bus in a serial poll. If the device is requesting service (RQS set), and the controller serial polls the device, the RQS bit is cleared. The MSS (Master Summary Status) bit (read with *STB?) and other bits of the Status Byte are not be cleared by reading them. Only the RQS bit is cleared when read.

The Status Byte is cleared with the *CLS common command.

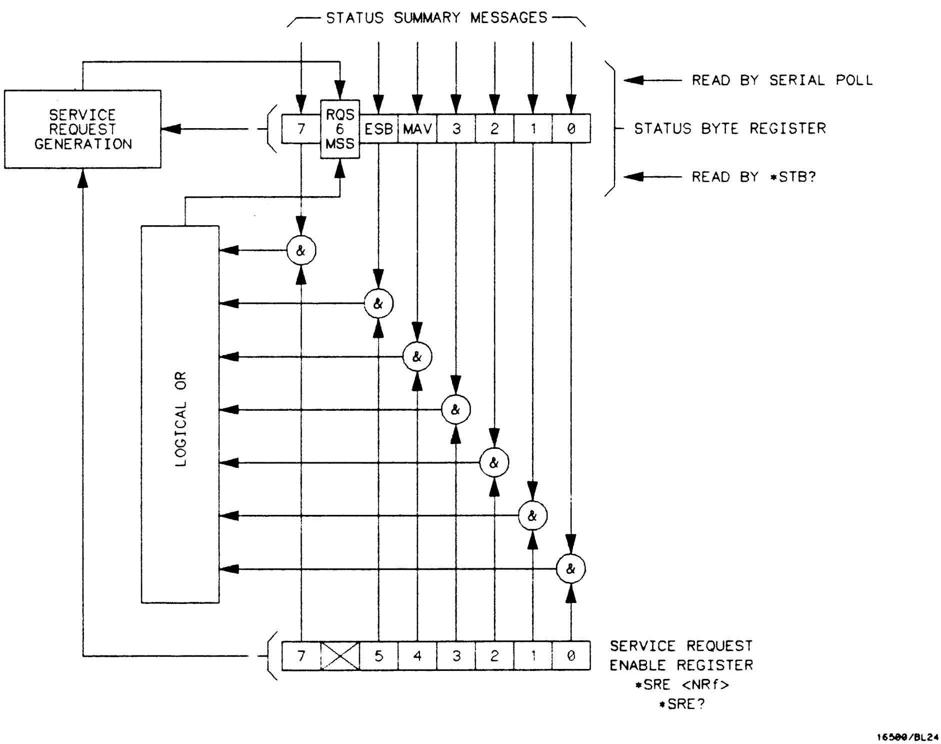


Figure B-2. Service Request Enabling

Serial Poll

The HP 1652B/1653B supports the IEEE 488.1 serial poll feature. When a serial poll of the instrument is requested, the RQS bit is returned on bit 6 of the status byte.

Using Serial Poll (HP-IB)

This example will show how to use the service request by conducting a serial poll of all instruments on the HP-IB bus. In this example, assume that there are two instruments on the bus; a Logic Analyzer at address 7 and a printer at address 1.

The program command for serial poll using HP BASIC 4.0 is Stat = SPOLL(707). The address 707 is the address of the oscilloscope in the this example. The command for checking the printer is Stat = SPOLL(701) because the address of that instrument is 01 on bus address 7. This command reads the contents of the HP-IB Status Register into the variable called Stat. At that time bit 6 of the variable Stat can be tested to see if it is set (bit 6 = 1).

The serial poll operation can be conducted in the following manner:

1. Enable interrupts on the bus. This allows the controller to "see" the SRQ line.

2. Disable interrupts on the bus.

3. If the SRQ line is high (some instrument is requesting service) then check the instrument at address 1 to see if bit 6 of its status register is high.

4. To check whether bit 6 of an instruments status register is high, use the following Basic statement:

```
IF BIT (Stat, 6) THEN
```

5. If bit 6 of the instrument at address 1 is not high, then check the instrument at address 7 to see if bit 6 of its status register is high.

6. As soon as the instrument with status bit 6 high is found check the rest of the status bits to determine what is required.

The SPOLL(707) command causes much more to happen on the bus than simply reading the register. This command clears the bus automatically, addresses the talker and listener, sends SPE (serial poll enable) and SPD (serial poll disable) bus commands, and reads the data. For more information about serial poll, refer to your controller manual, and programming language reference manuals.

After the serial poll is completed, the RQS bit in the HP 1652B/1653B Status Byte Register will be reset if it was set. Once a bit in the Status Byte Register is set, it will remain set until the status is cleared with a *CLS command, or the instrument is reset.

C - Error Messages

Error Messages C

This section covers the error messages that relate to the HP 1652B/53B Logic Analyzers.

Device Dependent Errors

200 Label not found

201 Pattern string invalid

202 Qualifier invalid

203 Data not available

300 RS-232C error

Command Errors

-100 Command error (unknown command)(generic error)

-101 Invalid character received

-110 Command header error

-111 Header delimiter error

-120 Numeric argument error

-121 Wrong data type (numeric expected)

-123 Numeric overflow

-129 Missing numeric argument

-130 Non numeric argument error (character,string, or block)

-131 Wrong data type (character expected)

-132 Wrong data type (string expected)

-133 Wrong data type (block type #D required)

-134 Data overflow (string or block too long)

-139 Missing non numeric argument

-142 Too many arguments

-143 Argument delimiter error

-144 Invalid message unit delimiter

Execution Errors

-200 No Can Do (generic execution error)

-201 Not executable in Local Mode

-202 Settings lost due to return-to-local or power on

-203 Trigger ignored

-211 Legal command, but settings conflict

-212 Argument out of range

-221 Busy doing something else

-222 Insufficient capability or configuration

-232 Output buffer full or overflow

-240 Mass Memory error (generic)

-241 Mass storage device not present

-242 No media

-243 Bad media

-244 Media full

-245 Directory full

-246 File name not found

-247 Duplicate file name

-248 Media protected

Internal Errors

-300 Device Failure (generic hardware error)

-301 Interrupt fault

-302 System Error

-303 Time out

-310 RAM error

-311 RAM failure (hardware error)

-312 RAM data loss (software error)

-313 Calibration data loss

-320 ROM error

-321 ROM checksum

-322 Hardware and Firmware incompatible

-330 Power on test failed

-340 Self Test failed

-350 Too Many Errors (Error queue overflow)

Query Errors

-400 Query Error (generic)

-410 Query INTERRUPTED

-420 Query UNTERMINATED

-421 Query received. Indefinite block response in progress

-422 Addressed to Talk, Nothing to Say

-430 Query DEADLOCKED

Index

Index

D

E

F

G

H

I

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Y

HEWLETT
PACKARD

Printed in U.S.A.

HP 1652B/1653B Logic Analyzers Programming Reference

HEWLETT PACKARD