TEK 1230

070-6878-02
PRODUCT GROUP 43

1230
LOGIC ANALYZER

OPERATOR'S MANUAL

This manual supports 1230 Logic Analyzers that have software versions of 4.XX and below.

Please check for change information at the back of this manual.

FIRST PRINTING: JANUARY 1990

Tektronix®
COMMITTED TO EXCELLENCE

Copyright © 1990, Tektronix, Inc. All rights reserved. Contents of this publication may not be reproduced in any form without the written permission of Tektronix Inc.

Products of Tektronix, Inc. and its subsidiaries are covered by U. S. and foreign patents and/or pending patents.

TEKTRONIX, TEK, SCOPE-MOBILE, are registered trademarks of Tektronix, Inc. TELEQUIPMENT is a registered trademark of Tektronix U.K. Limited.

Printed in the U.S.A. Specification and price change privileges are reserved.

Tektronix, Inc.
Walker Road Industrial Park
P.O. Box 4600
Beaverton, OR 97076

Table of Contents

List of Figures

List of Tables

PREFACE

This manual describes the TEKTRONIX 1230 Logic Analyzer. The 1230 is a modular, general-purpose test and measurement tool for designing, manufacturing, and servicing digital devices. The analyzer mainframe is lightweight, rugged, portable, and quiet. Because it is a modular instrument, you can configure the analyzer to meet your specific application. The 1230 Logic Analyzer supports hardware debugging, software analysis, microprocessor disassembly, and hardware/software integration.

ABOUT THE 1230 LOGIC ANALYZER

In its basic configuration, the 1230 Logic Analyzer can acquire data on 16 channels at 25 MHz, 8 channels at 50 MHz (asynchronous), or 4 channels at 100 MHz (asynchronous). You can also install up to three 16-channel 1230E1 Expander Cards in the analyzer, increasing the number of acquisition channels to 32, 48, or 64.

With each additional 16-channel expander card installed, the analyzer also lets you use an additional timebase. This lets you collect data simultaneously from system under test (SUT) at different sampling rates and threshold voltages. For example, a 64-channel analyzer can disassemble an 8-bit microprocessor and acquire 16 channels of data at a sampling rate of 40 ns at the same time.

Optional RS-232C and GPIB interfaces let you use the 1230 Logic Analyzer as part of an integrated system. These interfaces let you transfer acquired data and analyzer setup information between the 1230 and a remote host system. With these interfaces, you can also control the analyzer from your host keyboard.

The 1230 is easy to use. Screen and pop-up menus let you make changes quickly and easily, while a menu bar at the bottom of each menu lists the keys to press for menu features. The analyzer also provides online help if you need more information about the screens.

ABOUT THIS MANUAL

You don't have to read this manual from front to back. In general, the manual is organized so that user information is provided in the first sections, and reference information is listed in later sections.

The manual is divided into these sections:

Section 1 **Getting Started.** This section describes the analyzer front and back panels, tells you how to power up, and shows you how to acquire data in the default setup. This section also describes basic analyzer menus and features.

Section 2 **Procedures.** This section gives you simple procedures and examples for changing setup parameters and acquiring data using the demonstration test card supplied with your logic analyzer.

Section 3 **Setup Menus.** This section describes the setup menus and shows you how to change setup parameters for your applications.

Section 4 **Data Menus.** This section describes the analyzer's acquisition screen and data menus. This section also shows you how to manipulate acquired data in state and timing displays. For in-depth information about disassembly, refer to your microprocessor disassembly probe operator's manual.

Section 5 **Utility Menus.** This section describes the analyzer's standard and optional utilities, except for the optional GPIB and RS-232C interfaces.

For in-depth information about these interfaces, refer to their operator's manuals.

Appendix A **Logic Analyzer Concepts.** This appendix introduces you to logic analysis and explains basic logic analyzer concepts such as clocking, triggering, and glitches.

Appendix B **Default Menus.** This appendix lists the default analyzer menus.

Appendix C **Installation and Setup.** This appendix shows you how to install optional analyzer accessories. This appendix also shows you how to connect acquisition and microprocessor probes, and how to power up the analyzer.

Appendix D **Accessories and Specifications.** This appendix lists all the standard accessories, optional accessories, and specifications for the analyzer.

Glossary

Index

GENERAL SAFETY SUMMARY

The general safety information in this summary is for operating and servicing personnel. Specific warnings and cautions can be found throughout the manual where they apply, and may not appear in this summary.

TERMS IN THIS MANUAL

CAUTION — CAUTION statements identify conditions or practices that could result in damage to the equipment or other property.

WARNING — WARNING statements identify conditions or practices that could result in personal injury or loss of life.

TERMS AS MARKED ON EQUIPMENT

CAUTION indicates a hazard to property, including the equipment itself, and could cause minor personal injury.

WARNING indicates solely a personal injury hazard not immediately accessible as you read the marking.

DANGER indicates a personal injury hazard immediately accessible as you read the marking.

SYMBOLS AS MARKED ON EQUIPMENT

DANGER—High voltage.

Protective ground (earth) terminal.

ATTENTION—REFER TO MANUAL.

GROUNDING THE PRODUCT

This product is intended to operate from a power source that does not apply more than 250 volts rms between the supply conductors or between either supply conductor and ground.

WARNING: This product is grounded through the grounding conductor of the power cord. To avoid electrical shock, plug the power cord into a properly wired receptacle. A protective-ground connection by way of the grounding conductor in the power cord is essential for safe operation. (I.E.C. Safety Class I)

DANGER ARISING FROM LOSS OF GROUND

Upon loss of the protective-ground connection, all accessible conductive parts (including knobs and controls that may appear to be insulated) can render an electric shock.

POWER DISCONNECT

The main power disconnect is by means of the power cord or, if provided, an ac power switch.

USE THE PROPER POWER CORD

Use only the power cord and connector specified for your product. Use only a power cord that is in good condition. CSA Certification includes the equipment plus those power cords appropriate for use on the North America power network. All other power cords supplied are approved for the country of use.

USE THE PROPER FUSE

To avoid fire hazard use only a fuse of the correct type, voltage rating, and current rating.

USE THE PROPER VOLTAGE SETTING

Make sure the line selector is in the proper position for the power source being used.

REMOVE LOOSE OBJECTS

During disassembly or installation procedures, screws or other small objects may fall to the bottom of the mainframe. To avoid shorting out the power supply, do not power-up the instrument until such objects have been removed.

DO NOT OPERATE WITHOUT COVERS

To avoid personal injury or damage to the product, do not operate this product with covers or panels removed.

USE CARE WITH COVERS REMOVED

To avoid personal injury, remove jewelry such as rings, watches, and other metallic objects before removing the cover. Do not touch exposed connections and components within the product while the power cord is connected.

REMOVE FROM OPERATION

If you have reason to believe that the instrument has suffered a component failure, do not operate the instrument until the cause of the failure has been determined and corrected.

DO NOT OPERATE IN EXPLOSIVE ATMOSPHERES

To avoid explosion, do not operate this product in an explosive atmosphere unless it has been specifically certified for such operation.

Section 1

GETTING STARTED

This section introduces you to the 1230 Logic Analyzer and takes you through the setup menus so you can get started acquiring data. Section 2, *Procedures*, gives you several simple examples that show you how to acquire data, trigger on specific values, and detect glitches. Refer to Appendix A, *Logic Analyzer Concepts*, for information about logic analyzers in general.

This section describes:

- 1230 front and back panels
- Powering up and acquiring data
- Basic features of the 1230

The first discussion explains each key and connection on the front and back panel of the analyzer. The second discussion, *Power Up and Acquire Data*, shows you how easy it is to acquire data with the 1230. The Initialization and Main menus are discussed in-depth after that procedure. Features common to most menus are described last.

1230 FRONT AND BACK PANEL

Figures 1-1 and 1-2 show the keys and connections on the 1230's front and back panel. The numbered lists after each figure describe the keys and connections.

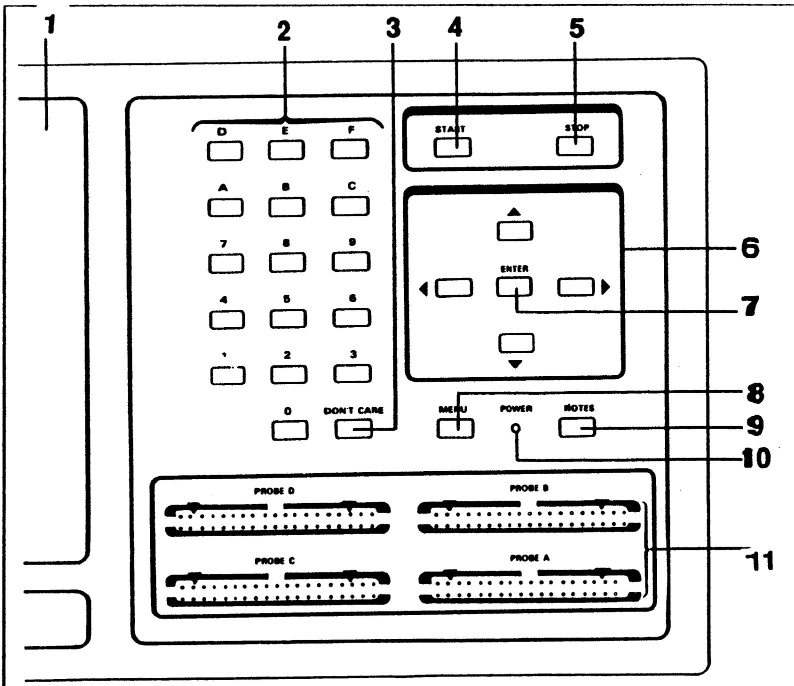


Figure 1-1. 1230 front panel.

1 **Display screen.**

2 **Data entry keypad.** You can enter hexadecimal characters directly from the keypad. You can also use the hexadecimal characters to directly enter the desired menu after you press the MENU key.

3 **DON'T CARE key.** The 1230 represents don't care values as Xs in the displays. Use this key to enter a don't care value in the Conditions menu and to enter a blank space in certain menu fields when changing (editing) the field names.

4 **START key.** Press this key to start a data acquisition.

5 **STOP key.** Press this key to stop a data acquisition manually.

[6] **Cursor keys.** Use these keys to scroll through menus or data. You can also use the cursor keys to enter symbols and alphanumeric characters when changing (editing) certain menu fields.

[7] **ENTER key.** Use this key to confirm certain changes when prompted for a response. You also use this key to enter and exit an editing mode when changing certain menu fields.

[8] **MENU key.** Use this key to call up the Main menu or return to the immediately previous menu you displayed. You can also use this key to exit the on-line notes which give you information about each 1230 menu.

[9] **NOTES key.** Use this key to display on-line help about each 1230 menu. You can also initialize the analyzer without turning the 1230 off by pressing NOTES and ENTER simultaneously.

[10] **Power indicator.** This light is on when the analyzer is turned on.

[11] **Probe slots.** To acquire data, you must always have a probe plugged into probe slot A. If you're using more than one probe, other probes can be plugged into slots B, C, and D, in that order.

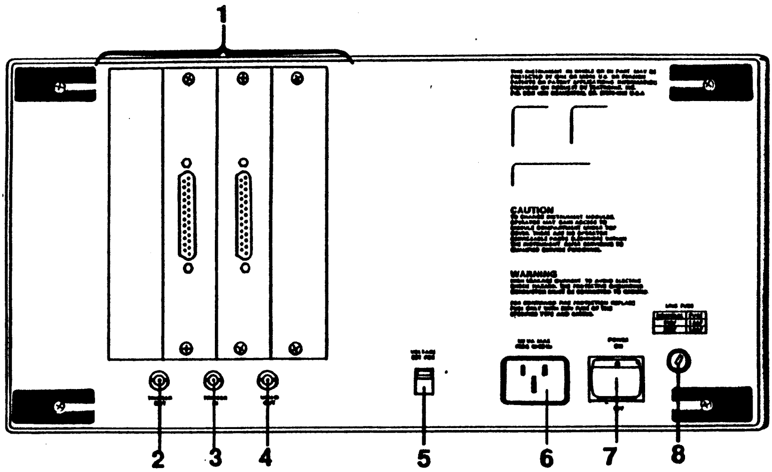


Figure 1-2. 1230 back panel.

[1] **Option card connection ports.** When you install option cards, the option card cables connect to the inside of these ports.

[2] TRIGGER OUT. Use this connection to send a signal to another device.

[3] TRIGGER IN. Use this connection to receive a signal from another device.

[4] VIDEO OUT. A composite of the display screen.

[5] Line voltage selector. The selector switch lets you choose between 115 V and 230 V for the input power voltage.

[6] Power cord connector.

[7] Power switch.

[8] Fuse.

POWER UP AND ACQUIRE DATA

This procedure shows you how easy it is to start acquiring data with the 1230 Logic Analyzer.

1. Make sure power to the 1230 and your system under test (SUT) is off.

CAUTION

Make sure power to the 1230 and SUT is off. If you connect the acquisition probe when the 1230 is off and the SUT is on, too much power may flow through the probe inputs and damage the probe.

2. Plug your acquisition probe into slot A on the 1230.
3. Connect your acquisition probe to your SUT.
4. Turn on the 1230, which supplies power to the probe. Instruments with version 4.0 software will first display diagnostics if this parameter is set to "on" in menu C. The Initialization menu is now displayed on your 1230 screen. Figure 1-3 shows the Initialization menu.
5. Turn on your SUT.
6. Press MENU to initialize the 1230 and call up the Main menu. Figure 1-4 shows the Main menu.
7. Press START to acquire data with the default 1230 setup. In the default setup, the 1230 triggers on all don't cares at an asynchronous sampling rate of 1 μsec.

During the acquisition, the Acquisition Process screen is displayed, telling you the status of the acquisition; for example, that the 1230 is waiting for the trigger to occur. Figure 1-5 shows a sample Acquisition Process screen.

As soon as the acquisition is complete, the analyzer automatically displays the acquired data in the default display format, a timing diagram. Figure 1-6 shows a sample timing diagram.

```
WED, NOV 29, 1989                        11:02   -DEFAULT

Tektronix 1230/32 Channel Logic Analyzer. V4.01
(C) Tektronix, Inc. 1988, 1989 All rights reserved.


     Use the NOTES key whenever information is needed,
              or consult the Operator's Manual.

              X represents DON'T CARE condition.

                    Press MENU to continue
```

Figure 1-3. Initialization menu. When you first turn on the analyzer, this menu is displayed. This menu won't be displayed again unless you turn on the analyzer again or reset it by firmly pressing NOTES and ENTER at the same time. The menu bar at the bottom of the screen tells you that pressing MENU calls up the Main menu.

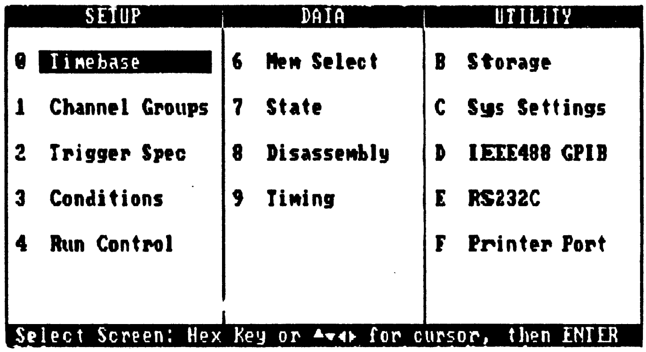


Figure 1-4. Main menu. Press MENU to call up the Main menu. This menu gives you access to the analyzer's setup, data, and utility menus. The menu bar at the bottom of the screen tells you which keys to press to select other analyzer menus.

```
TUE, JAN 03, 1989  Acquisition Process  14:14   -DEFAULT
                 Timebase         Status

                    T1          Memory Full




                  < Acquisition Complete >

                  Trigger Spec Level   FILL

                  Stop Acquisition: STOP
```

Figure 1-5. Acquisition Process screen. As soon as the 1230 is connected to your SUT and initialized, you can acquire data from your SUT. This screen tells you the status of the acquisition. For example, the screen can tell you which trigger statement is being executed, whether or not the trigger has been found, and whether or not memory is being filled.

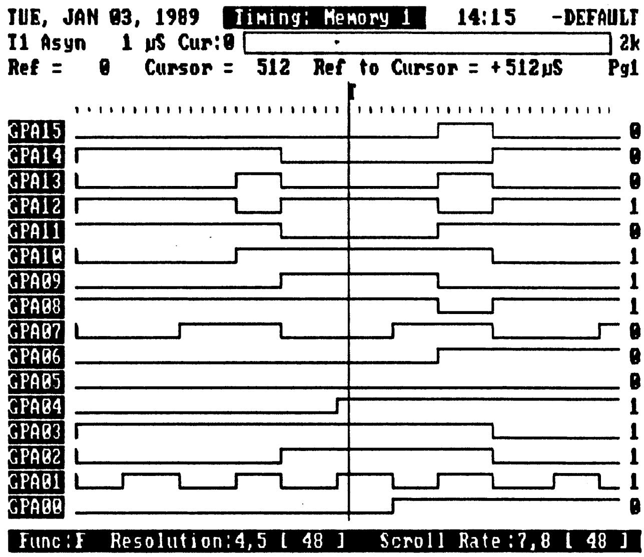


Figure 1-6. Sample Timing Diagram. As soon as the acquisition is complete, the 1230 automatically displays the acquired data for you. The default data display is a timing diagram. Press MENU, then 7 to call up the corresponding state table for your data acquisition.

The rest of this section describes basic features of the 1230. Section 2 shows you some simple examples of setting up the 1230 to acquire data, trigger on a word condition, and capture glitches. Sections 3 through 5 explain setup, data and utility menus in detail.

BASIC FEATURES

The 1230 has many features that make it easy to use. Among them are the 1230's modular configuration, single-keystroke functions, pop-up menus, and on-line note screens. This discussion explains the menus and on-line note screens.

Initialization Menu

When you first turn on the 1230, an Initialization menu is displayed. This tells you the name and firmware version number of the 1230 Logic Analyzer. The menu also gives the time and date, tells you how to get on-line notes, and tells you that in the displays, an X represents a don't care. If diagnostics are set to "on", the 1230 runs diagnostics tests on power up. Figure 1-3, earlier in this section, shows the Initialization menu.

Main Menu

Press MENU to call up the Main menu. This menu lets you choose setup, data, and utility menus for using and configuring the 1230. Figure 1-4, earlier in this section, shows the Main Menu. When you first call up the Main menu, the cursor highlights the upper left field, Timebase. When you return to the menu, the cursor highlights the last selection you made.

To choose a menu from the Main Menu, press the hexadecimal key associated with the menu. For example, press 0 for the Timebase menu, 7 for a State Table, or B for the Storage menu. You can also use the cursor keys to scroll to the menu selection and then press ENTER. The selected menu is then shown on the screen.

There are (fields) reserved for menus associated with options. The analyzer displays a menu only if the option is installed. If you don't have any options, these fields are blank. If you press those keys or try to select those fields, the analyzer ignores your request. You can install up to three of these options in the 1230:

- Parallel printer
- RS-232C
- GPIB
- International notes

Other Menus

The 1230 has three types of menus: setup, data, and utility menus. These menus are shown as screen or pop-up menus.

Screen and Pop-up Menus

Some 1230 menus are screen menus, and some are pop-up menus. Screen menus, such as the Channel Grouping menu, fill the entire display. Pop-up menus, such as the Conditions menu, don't fill the entire display. Pop-up menus are displayed on top of screen menus. This is especially handy for the Conditions menu, because you can look at your defined conditions and the information on a screen menu at the same time.

Setup, Data, and Utility Menus

Setup menus let you set up the 1230 for different kinds of acquisitions. For example, set-up menus let you link probes for parallel data acquisition, specify channel groups, and set trigger events. Data menus display acquired data in different formats, such as timing diagrams and state tables. Data menus also let you manipulate data; for example, compare two different acquisitions. Utility menus control support features or menus you add by installing options, such as parallel printer or RS-232C ports.

The next three tables briefly describe the standard and optional menu selections.

Table 1-1
SETUP MENUS

Setup	Description
Timebase	Sets the sampling rate and clocking format (asynchronous or synchronous), glitch detection, and probe threshold voltage. This menu also lets you link probes in the same timebase or in separate timebases.
Channel Grouping	Assigns probe channels to groups so that you can acquire and display data in groups meaningful to your application.
Trigger Spec	Defines the trigger statements that control data acquisition and storage. Trigger statements include fields that control trigger conditions, number of occurrences of the conditions, trigger actions, and destinations.
Conditions	Defines the word values for word recognizers used in the Trigger Spec menu. Conditions may also be used as comparison masks and in the Search For function of the State and Timing displays.
Run Control	Specifies which memory data is stored in, at what memory location the trigger is stored, and when the 1230 begins looking for the trigger condition. This menu also controls the memory comparisons executed from the state table display.

Table 1-2
DATA MENUS

Data Menu	Description
Memory Select	Shows the main setup parameters for each of the four memories. This menu also lets you choose the memory you want to display as reference memory.
State	Displays acquired data in state table format. This screen also shows the differences from memory comparisons.
Disassembly	If you're using one of the optional microprocessor disassembly probes, the screen displays disassembly information in hardware or software format.
Timing	Displays data using timing traces. This screen also lets you reorder, duplicate, and delete traces from the display to show only the information you want.

Table 1-3
UTILITY MENUS

Utility Menu	Description
Storage	Stores user-specified menu settings (setups). This menu lets you reload stored setups ad data previously stored.
System Settings	Sets the date, time, screen intensity, screen saver, and power-up diagnostics features.
RS-232C	Sets the baud rate, word length, parity, and stop bits for the optional RS-232C communication protocol. Refer to the RS-232C Operator's Manual for information.
GPIB	Sets up the General Purpose Interface Bus interface option. Refer to the 1230 GPIB Operator's Manual for GPIB information.
Printer Port	Specifies how to use the optional printer port, including the range of memory to print, the print density, and other details.
International Notes	Selects the language of the notes information (optional).

Using the Cursor

You can use the 1230 cursor keys to move around in menus and screen displays.

In most cases, when you reach the end of a line or screen, the cursor wraps to the other end of the line or screen.

The first time you call up a menu, the cursor appears in the upper-left field for that menu. In general, if you move the cursor, leave the menu and then return to it before making changes in another menu, the cursor appears in the same field it last occupied.

For some menus, the cursor tracks position between the displays. For example, in data menus, the cursor tracks the same memory location (such as 0815) in each display so that you see the same information in different display formats (state, disassembly, and timing).

Changing Menu Fields

Each menu has fields that you can change to define the way the analyzer triggers, displays, and stores data. For example, in the Timebase menu, you can change fields that describe probe links, data sampling formats and rates, glitch capture, and probe threshold voltage.

Some fields have a list of parameters you can choose from. For example, in the Timebase menu, you can choose two sampling formats: Sync (for synchronous sampling) or Async (for asynchronous sampling). Other menu fields are fields you can edit by entering values from the keypad. For example, you can edit a channel group name to be any 3-character string you choose, such as DAT or INT.

There are different ways to change menu fields. For example, if there's a list of parameters to choose from, you can usually press 0 or 2 to cycle through the list of selections. The menu bar at the bottom of each screen lists the keys to press for each menu function. For example, in the Trigger Spec menu, the trigger timebase field is listed in the menu bar as Triggger Timebase:D[T1]. This tells you to press D to change the trigger timebase from T1 to another timebase. The value in brackets tells you what the current selection is for that field.

Sometimes, just putting the cursor on the field lets you change the field. For example, you can change a condition word definition, such as XXFF, as soon as you move the cursor to the definition field. Other times you may have to press ENTER to tell the 1230 that you want to change a field. For example, to change a channel group name, move the cursor to the name field and press ENTER, then enter the group name you want by using the keypad and cursor keys to enter characters.

Scrolling Through Data

Some menus have more information than fits on the screen at one time. For example, the state table contains information for the entire 2K memory, but only 20 lines fit on the screen at a time. For these menus, use the cursor keys to scroll more data onto the screen.

When you reach the end of memory or the display, the data wraps around to the beginning again so you can display the beginning and end of memory at the same time. A blank line separates the beginning and end of memory.

Using Menu Bars

At the bottom of each screen, a one-line help message (a menu bar) tells you which keys to press for each menu function. Figures 1-3, 1-4, and 1-6 show menu bars at the bottom of the menus.

Some data and utility menus have more than one menu bar. In these cases, press F to cycle through available menu bars. If there's more than one menu bar for the menu, you don't have to display a specific bar to use the functions listed on it. Menu bars just remind you about menu functions. For example, one of the State Menu bars tells you to press C to compare the display and reference memories. You don't have to cycle through the menu bars until you display the bar that lists the compare instruction. As long as you're not in an editor, you can press C at any time in the state table to compare memories and display differences.

An editor is a tool that lets you enter the characters you want instead of selecting a predetermined value. For example, in the Channel Grouping menu, you can list only valid channels in the groups (such as A15-A00). However, you can edit the channel group name to be any 3-character name you want (such as DAT for data bus, or I/O for input-output lines).

The menu bars that list editor functions are displayed only when you're in an editor. These bars tell you which keys to press to edit or change the field.

For example, if you press ENTER while on the channel group name, you can edit the name, and the menu bar automatically changes to list the keys you can use to change the name.

Using Online Help

If you need information about a menu you are viewing, press NOTES. The 1230 then displays a screen of information about the current menu. You can press NOTES at any time to get information. Press MENU to exit the on-line notes and return to the previous display. Figure 1-7 shows an on-line help screen for the Timebase menu.

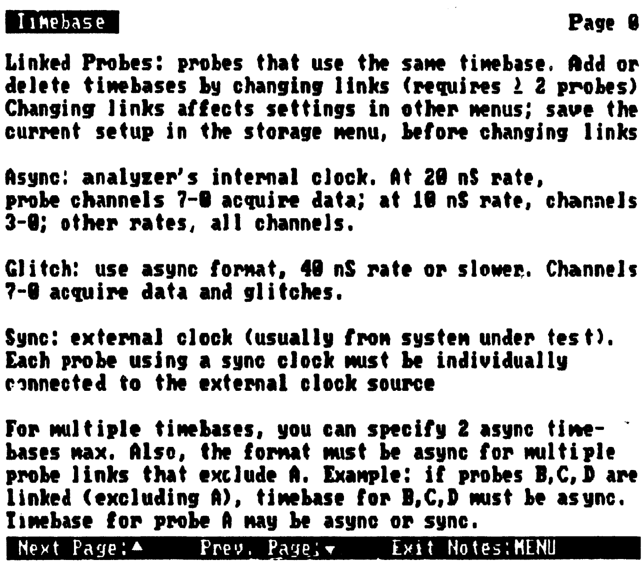


Figure 1-7. Notes for Timebase menu. Press NOTES to display information about the current menu. You can use the cursor keys to scroll through other pages of notes about other menus. After the notes information is displayed, you can press MENU to return to the previous menu.

Section 2

PROCEDURES

This section shows you how to set up the analyzer, restore the default setup, and acquire data by triggering on events, glitches, and clock cycles. These examples also show you how to mask channels from the data display, compare two memories, and store acquisition setups. This section is divided into these discussions:

- Connecting to the SUT
- Setting up the 1230
- Acquiring data

The examples of acquiring data use the demonstration test card included with the 1230 Logic Analyzer. For more in-depth examples, refer to the 1230 Logic Analyzer Workbook. The workbook uses the test card to show more advanced examples for acquiring data.

CONNECTING TO THE SUT

Before you use the analyzer, make sure you connect and power up the probes to your system as shown in the following procedure. Figures 2-1 and 2-2 show an example of how the analyzer, probes, and SUT (system under test) can be connected. Appendix C, Installation and Power Up, explains the power-up procedure in detail.

1. Make sure power to the analyzer is off.

CAUTION

Do not connect any probe to the analyzer or SUT unless power to the analyzer and SUT is off. If you connect a probe when the analyzer is off and the SUT power is on, too much current may flow through the probe's inputs and damage the probe.

2. Connect the probes to the analyzer. The first probe should always be plugged into probe slot A. Figure 2-1 shows the analyzer and probe configuration.
3. Connect the probes to the SUT.
4. For synchronous acquisition, make sure that each probe's clock lines are connected to the same points in the SUT. For asynchronous acquisition, clock lines don't have to be connected; the 1230 supplies an internally generated clock.
5. Power up the analyzer and probe.
6. Power up the SUT.

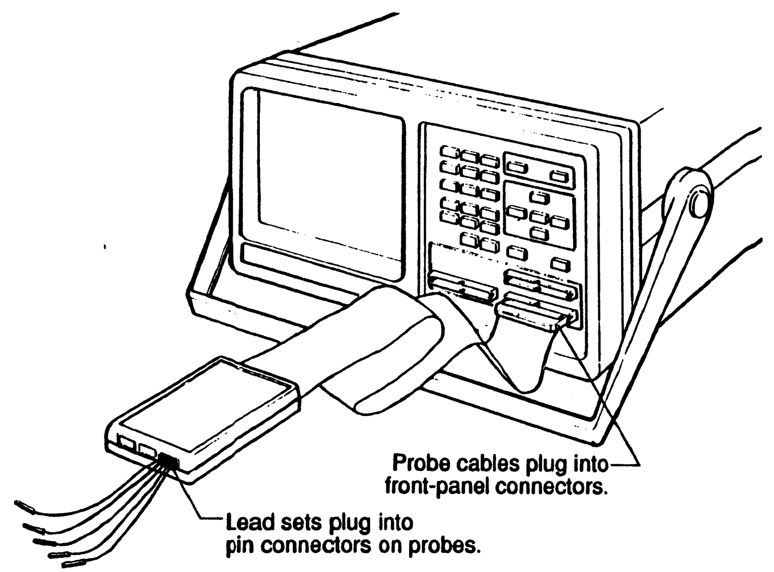


Figure 2-1. Analyzer and probe connections. Make sure you turn the analyzer and SUT power off before plugging in any probes.

Setting the Clock Rate

You can acquire data at the rate of the clock in your SUT or the analyzer's internal clock. Using the SUT clock is synchronous sampling. Using the logic analyzer clock is asynchronous sampling.

If you acquire data synchronously, make sure that each probe's clock lines are connected to the same SUT points. If you acquire data asynchronously, you can set the clock rate in the Timebase menu to incremental values between 40 ms and 10 ns. The examples in this section show you how to set the analyzer's clock rate for sampling data.

SETTING UP THE 1230

A setup is a set of parameters that tells the analyzer how to acquire data from your SUT and display it in a format appropriate for your application. For example, a setup includes information about probe links, sampling rate, input signal groups, and trigger conditions.

The Default Setup

These procedures and examples use a 16-channel analyzer in the default setup. The default setup puts all 16 input channels in one channel group (named by default, GPA) and triggers on all don't cares. The asynchronous sampling rate is 1 µs.

If the default setup is loaded and the analyzer connected to your SUT, you can press START at any time to acquire data. The analyzer will trigger on the 512th data sample and fill memory, stopping when the acquisition is complete. You can also press STOP to stop the acquisition at any time. Appendix A explains acquisition concepts in detail. Appendix B lists the default analyzer menu settings.

Changing the Default Setup

To change the default setup, call up the appropriate menu and use the keys listed in the menu bars to select values you want for the menu fields. For example, you may want to acquire data at a faster sampling rate. You may want to group the input signals so that the address bus is in a different channel group than the data bus. You may want to define a trigger condition so that the analyzer acquires data only after a certain event occurs.

These procedures show simple examples of how and why you would want to change the default setup. The three examples that follow show how to set up the analyzer to do specific tasks, such as trigger on a glitch. This discussion shows you how to:

- Set the asynchronous sampling rate
- Select a variable threshold voltage for your P6444 probe
- Group input signals
- Rename channel groups
- Define a condition
- Rename a condition
- Define simple and complex trigger statements
- Save (store) a setup

Making Changes Efficiently

When you make changes to the setup, it's important that you make them in the right order. Some changes, such as changes to the timebase in which you acquire data, affect many other analyzer menus. Other changes affect only one or two fields in a submenu. You can save time by setting up the menus in the order of the greatest impact to the least:

1. Timebase
2. Channel Grouping
3. Conditions

For example, the analyzer must know how the acquisition probes are linked before it can group the input signals into channel groups in the right timebase. If you change a channel group and then change probe links, the analyzer may have to overwrite your channel group changes to make sure the groups are appropriate for your probe links. However, if you change the probe links first, you know that any changes you make to channel groups will automatically be appropriate for your links.

Restoring the Default Setup.

When you turn on or reset the 1230, the analyzer keeps the same setup it had before being reset. Even if the setup name field in the menu header says DEFAULT, if there is an asterisk before the name (* -DEFAULT), there have been changes to the default setup since it was loaded.

To load the original default setup listed in Appendix B, follow these steps:

1. Press NOTES and ENTER at the same time. The Initialization menu should now be displayed on your screen. If they have been set to "on" in menu C, the diagnostics will also run.
2. Press D to load the default setup.
3. Check the header line at the top of the menu. In the upper right corner, the setup name field in the header line should now say -DEFAULT. The asterisk, if it was displayed before, should now be gone from this field. The asterisk meant that the setup had been changed since it was loaded.

You can now press MENU to call up the Main menu. The other menus should reflect the default setup listed in Appendix B.

Set a Sampling Rate

The default sampling rate is 1 µsec. To change the rate, follow these steps.

1. Press MENU, then 0 to call up the Timebase menu.
2. Use the cursor keys to move to the rate field.
3. Press 0 or 2 to cycle through the rate selections from slower to faster or faster to slower. Figure 2-2 shows a Timebase menu with a sampling rate of 80 ns.

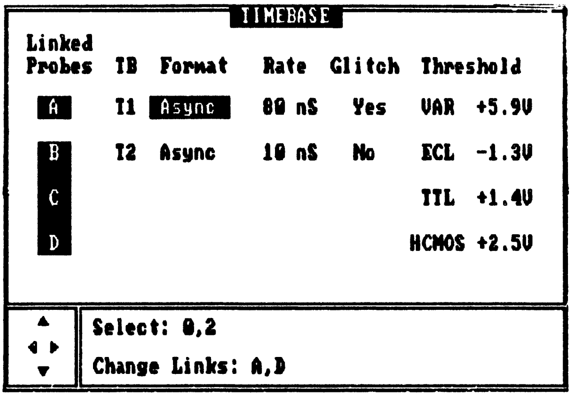


Figure 2-2. Sampling rate and threshold voltage. **Using an asynchronous sampling format lets you choose a sampling rate appropriate for your SUT. Using a P6444 probe (instead of a P6443 probe) lets you select a variable threshold voltage for your acquisition.**

Select a Variable Threshold Voltage

If you're using a P6444 acquisition probe, you can choose the threshold voltage at which the 1230 recognizes a data high or low. The P6443 acquisition probe uses a threshold voltage of TTL +1.4 V (you can't change the threshold voltage for a P6443 probe). To change the threshold voltage for a P6444 probe, follow these steps:

1. In the Timebase menu, use the cursor keys to move to the threshold voltage field.
2. Press 0 or 2 to cycle through the voltage selections (TTL, HCMOS, ECL, and VAR) until you select VAR.
3. Move the cursor into the threshold polarity field and press 0 or 2 to toggle from positive to negative polarity. For example, select a negative polarity.
4. Move the cursor into the voltage field, and use the 0 or 2 keys to cycle through the digits 0-9 to select the digits you want for the threshold. Move the cursor to the next digit. You can choose a threshold from ±0.1 to ±9.0 V (you can't choose a threshold of 0.0 V).

Figure 2-2 shows different threshold voltages.

Group Input Signals

The default setup puts each probe's 16 input channels in a different group. You can change the groups so that up to 32 channels are grouped together. You can also delete input channels from the groups so that data is displayed only on specific channels. Grouping channels lets you display acquired data in the groups and radixes most appropriate for your application. In this example, the 16-channel acquisition probe is connected to an 8-bit data bus (D7-D0) and an 8-bit address bus (A7-A0). SUT address lines A7-A0 are probe inputs A07-A00 and SUT data lines D7-D0 are probe inputs A15-A08. Follow these steps to group the address and data lines separately.

1. Press MENU, then 1 to call up the Channel Grouping menu. All 16 input channels are in channel group GPA by default.
2. Move the cursor to the channel labeled A15.
3. Delete channels A15-A08 from the group (you must delete a channel from the group it's in before you can add it to another channel group). Press 0 to delete A15, press 0 to delete the next channel, and so on. Channels A15-A08 are now listed in the unused channels list at the bottom of the screen.
4. Add channels A15-A08 to channel group GPB. Move the cursor to channel group GPB and enter the first channel, A15 (press A, 1, 5, on the keypad). Then press E seven times to add the rest of the channels. Pressing E adds the next available consecutive channel to the group.

Figure 2-3, displayed after the next procedure, shows the Channel Grouping menu with the channel groups changed. (Figure 2-3 also shows that the channel group names are changed.) The next procedure shows how to change channel group names.

Rename Channel Groups

The 1230 lets you rename channel groups to any 3-character name you want. Being able to name channel groups lets you see quickly and easily what information is acquired for each group in the data displays. Follow these steps to rename channel group GPA to ADD (for address bus) and group GPB to DAT (for data bus).

1. In the Channel Grouping menu, move the cursor to the name of the first group, GPA.
2. Press ENTER to tell the analyzer you want to change the name of the channel group. The cursor now highlights only the first character of the name.
3. Enter the new name for the channel group; for example, since this group represents the 8-bit address bus, enter ADD directly from the keypad. As soon as you enter the third character, the cursor saves your changes and moves to the next field. (You can also press ENTER while still on the group name to save your changes.)
4. Move the cursor to the group GPB and press ENTER to change this name too.
5. Enter the new name for this channel group; for example, DAT for the 8-bit data bus. Enter the hexadecimal characters D and A directly from the keypad, and use ▲ (or ▼) to cycle through the available characters until you select T for the third character. The cursor automatically moves to the next field when you enter the third character.

The Channel Grouping menu should now look like Figure 2-3.

```
TUE, JAN 10, 1989  Channel Grouping   14:27 ▣ -DEFAULT
Group  Radix  Pol TB  Channel Definitions
 ADD    HEX    +  T1  AAAAAAAA
                      00000000
                      76543210

 DAT    HEX    +  T1  AAAAAAAA
                      11111100
                      54321098

 GPC    HEX


 GPD    HEX

Probe               UNUSED CHANNELS
  A |

Cursor:▲▼ ◄►    Edit name:ENTER        Default Groups:F
```

Figure 2-3. Channel Grouping menu. The channels are split into two 8-bit groups, ADD and DAT. In this example, group ADD contains A07-A00, the input address signals A7-A0. Group DAT contains A15-A00, the input data signals D7-D0. When you delete channels, spaces may appear in the display. When you call up the menu again, the spaces will automatically be deleted.

Define a Trigger Condition

The 1230 lets you define conditions on which the analyzer will trigger. Defining conditions lets you acquire only the data you need from your SUT. For example, you may want to store only the sequence of a subroutine, so you would define the beginning and end of a subroutine as conditions of the trigger program.

The example shown here uses one channel group called ADD (for address bus) with 16 channels. Follow these steps to define the beginning and end of a subroutine as conditions of the trigger program.

1. Press MENU, then 3 to call up the Conditions menu. The default condition word definitions are all don't cares (except for MASK in software version 4.0).
2. Move the cursor to the condition word definition for condition A.
3. Enter the value for the beginning of the subroutine; for example, 2053. As soon as you enter the value for the last digit, the cursor automatically wraps around to the condition name again.
4. Move the cursor to the condition word definition for condition B.
5. Enter the value for the end of the subroutine; for example, 2080. Figure 2-4 shows these condition word definitions.

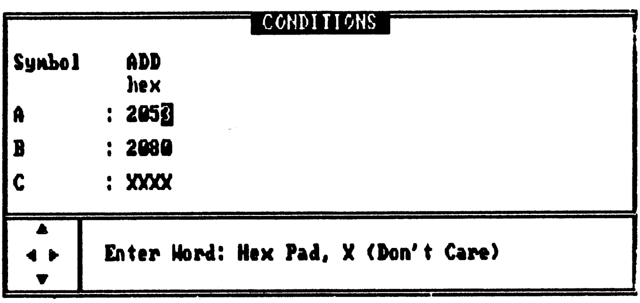


Figure 2-4. Defining conditions. Condition A, the beginning of the subroutine, is 2053; condition B, the end of the subroutine, is 2080. The cursor highlights each digit of the definition separately. You can re-size the menu to show more or fewer conditions by pressing F and C.

Rename a Condition

You can rename any of the 24 conditions you can define. In this example, since you already defined two conditions to be the beginning and end of a subroutine, the following steps show you how to rename those conditions to names meaningful for the example.

1. Move the cursor to the condition name A.
2. Press ENTER to tell the analyzer you want to edit the condition name.
3. Enter the name SUB BEG to change A to a name more representative of the beginning of the subroutine. Enter hexadecimal characters B and E directly from the keypad. Use the cursor keys ▲ and ▼ to enter the other characters (S, U, and G). Press DON'T CARE to enter a space.
4. Press ENTER when you're done to save your change. The cursor then highlights the entire condition name.
5. Follow steps 1 through 4 for condition B to rename that condition SUB END (for the end of the subroutine). The Conditions menu should now look like Figure 2-5.

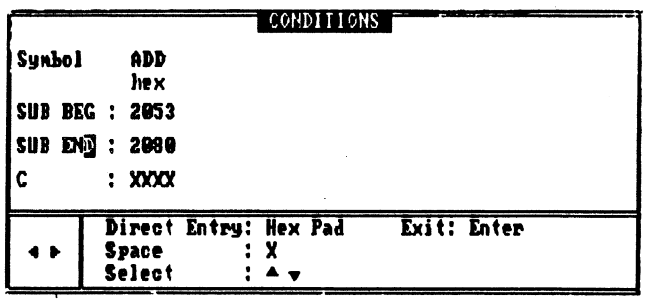


Figure 2-5. Renaming conditions. Condition A, the beginning of the subroutine, is renamed SUB BEG. Condition B, the end of the subroutine, is renamed SUB END. The cursor highlights each digit separately as you change it. Conditions are always listed in the default order: A-X. Make sure you don't give two conditions the same name (for example, define two conditions as SUB END), because the analyzer looks for conditions in the default order A-X. If the first SUB END occurs, the analyzer will never look for the second SUB END.

At this point and for this example, you would define the trigger statements so that when SUB BEG occurs, the 1230 turns storage on and when SUB END occurs, the analyzer turns storage off. The next procedure shows how to define simple and more advanced trigger statements.

Choose a Trigger Statement

A trigger statement tells the 1230 what to trigger on, how to store data, and when to fill memory and stop. You can choose if-then or if-then-else trigger statements to define your trigger specification. The default configuration is an if-then statement for simplicity. This discussion shows you how to:

- Trigger on a clock cycle
- Trigger and store only a subroutine
- Trigger on an intermittent condition

Trigger on a Clock Cycle. Follow these steps to define a simple if-then trigger statement that tells the 1230 to trigger on the 1000th clock cycle.

1. Press MENU, then 2 to call up the Trigger Spec menu. The cursor highlights the condition in the trigger statement, which by default is A.
2. Press 0 to cycle through the conditions and select Clock as the trigger condition. Since the default condition is A, you should only have to press 0 once to select the clock. (Pressing 2 cycles through the trigger conditions in the opposite direction.)
3. Move the cursor to the count field and press ENTER to tell the 1230 you want to change the clock count.
4. Enter the value 1000 directly from the keypad. As soon as you enter the last digit, the cursor automatically saves your change and moves to the next field.

The trigger statement should now look like Figure 2-6.

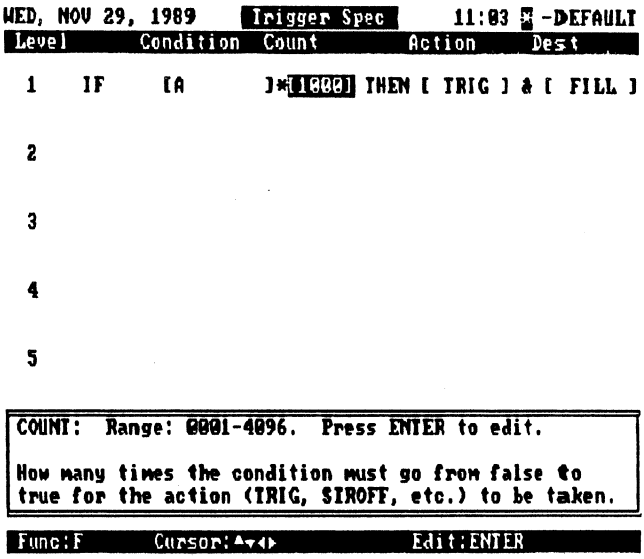


Figure 2-6. Trigger on a clock cycle. This trigger statement tells the 1230 to trigger and fill memory on the 1000th clock cycle.

Store Only a Subroutine. To define a more complex trigger statement, you can use up to 14 levels of if-then statements or if-then-else statements. For example, you can trace a subroutine with two simple if-then statements:

1. Turn storage on at the beginning of the routine.
2. Turn storage off at the end of the routine.

Follow these steps to define two trigger statements. For this example, condition A is the beginning of the routine and condition B is the end of the routine.

1. In the Trigger Spec menu, move the cursor to the condition field and press 0 or 2 to cycle through the available conditions until you select A.

2. Move the cursor to the count field and press ENTER to change the clock-cycle count, then enter the value 0001 directly from the keypad.
3. Move the cursor to the action field and press 0 or 2 to cycle through available actions until you display STR ON. This tells the 1230 to turn storage on when A occurs.
4. Move the cursor to the destination field and press 0 or 2 to cycle through the available destinations until you display CONTIN. This tells the analyzer to execute the instruction on the next trigger level.
5. Move the cursor down to level 2 and press ENTER to display another if-then statement. When you do this, the cursor highlights the condition field, which by default, is always A (or the condition name representing A, the first definable condition).
6. Press 2 (or 0) to cycle through the available conditions until you display B. Since the default condition is A, you should have to press 2 only twice to display B. The count field should specify 0001 as the default cycle count.
7. Move the cursor to the action field and press 0 or 2 to cycle through the actions until you display STROFF. This tells the 1230 to turn storage off when B (the end of the subroutine) occurs.
8. Move to the destination field and press 2 (or 0) to cycle through the destinations until you display GOTO 1. This tells the analyzer to go back to trigger statement 1 and look for the beginning of the routine again, repeating the storage sequence.

The Trigger Spec menu should now look like Figure 2-7.

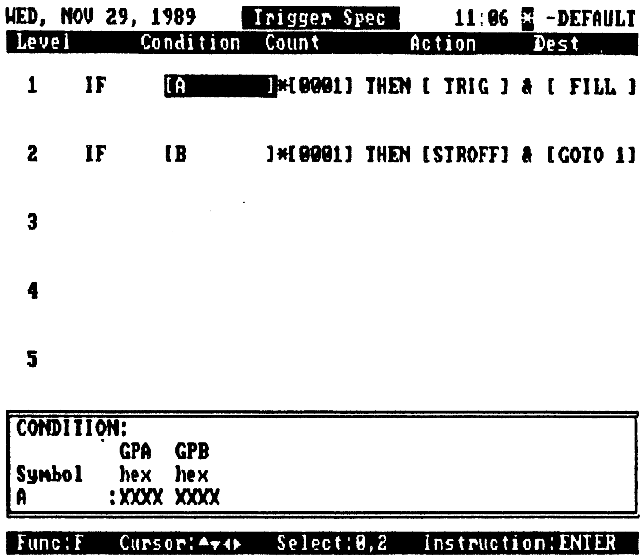


Figure 2-7. Store a subroutine. This trigger statement tells the 1230 to turn storage on when A (the beginning of the subroutine) occurs and turn storage off when B (the end of the subroutine) occurs. Because the destination field is a GOTO statement, the 1230 returns to trigger statement 1 and repeats the sequence until you press STOP.

Since you didn't specify a trigger action for this series of statements, you have to press STOP to stop an acquisition after you press START. If you define your trigger statements to enable the analyzer to trigger, the analyzer stops the acquisition at the trigger point and fills memory with the data you specified for the acquisition.

Trigger on an Intermittent Condition. You may want to trigger and store data if a condition doesn't occur. For example, if your SUT calls a subroutine, and then an interrupt sometimes occurs within 100 clock cycles after the subroutine is called, you want to know why the interrupt is occurring. To do this, because the interrupt is intermittent, you want to store information when the interrupt occurs, but not when the interrupt doesn't.

You want two trigger statements:

1. Look for the beginning of the routine.
2. If the interrupt doesn't occur within 100 clock cycles, look for the beginning of the subroutine again. Otherwise, if the interrupt occurs, trigger and fill memory.

Because you're counting clock cycles, you must consider the number of triggering levels you are using. The analyzer uses 2 clock cycles (or 80 ns, whichever is greater) to go to the next trigger level. To count 100 clock cycles in statement level 2 from the beginning of the subroutine, which is in statement 1, use 98 as the count number instead of 100. The analyzer uses 2 cycles to go from level 1 to 2, and then counts 98 more clock cycles before looking through its acquired memory for the interrupt condition.

The first trigger statement is a simple if-then statement. The second statement is an if-then-else statement. The 1230 features an advanced mode of triggering that lets you use if-then-else statements. Follow these steps to set up this more complex trigger specification.

1. In the Trigger Spec menu, press F to display the Mode menu bar, then press 1 to toggle the triggering mode from basic to advanced. The menu bar at the bottom of the screen should now say Mode 1: [Advanced], the current triggering mode. Pressing 1 again will toggle you back to Basic mode.
2. If trigger statement 1 is not already a simple if-then statement, move the cursor to statement 1 and press ENTER to cycle through the statements until you display the if-then statement. (Pressing ENTER on the count field puts you in an edit mode for changing the count cycles. Pressing ENTER on any other field cycles through the available trigger statements.)

3. Use the cursor keys and the 0 and 2 select keys to change the trigger statement for level 1 to look like this:

```
IF [A    ]*[0001] THEN [NOP ] & [CONTIN]
```

4 Move the cursor to the trigger statement for level 2 and press ENTER until you display an if-then-else statement as shown in step 5. (You should cycle through the statements in this order: if-then, if-then-else-if-then, if-then else, and if-then-else-store-only.)

5. Use the cursor keys and the 0 and 2 select keys to change the statement to look like this:

```
IF  [CLOCK  ]*[0098] THEN [NOP ] & [GOTO1]
ELSE IF [INT   ]      THEN [TRIG] & [FILL ]
```

Figure 2-8 shows the Trigger Spec menu now.

```
TUE, JAN 10, 1989      Trigger Spec        14 33 -DEFAULT
 Level          Condition  Count          Action      Dest

  1     IF      [SUB BEG ]*[0001] THEN [ NOP  ] & [CONTIN]


  2     IF      [ Clock  ]*[0098] THEN [ NOP  ] & [GOTO 1]
  ELSE  IF      [INT     ]        THEN [ TRIG ] & [ FILL ]

  3
+------------------------------ CONDITIONS ----------------------------+
| Symbol       ADD                                                     |
|              hex                                                     |
| SUB BEG :    2023                                                    |
| SUB END :    2080                                                    |
| INT     :    0066                                                    |
+----------------------------------------------------------------------+
|   ▲   | Edit Symbol: ENTER                                           |
|  ◄ ►  | Window Up  : F                                               |
|   ▼   | Window Down: C                                               |
+----------------------------------------------------------------------+
 Menu:MENU   Return:MENU twice   New:MENU, then Hex Key
```

Figure 2-8. Trigger if INT occurs. This trigger statement tells the analyzer to trigger if the interrupt condition occurs within 100 clock cycles of the beginning of the subroutine. The clock count field is 98 instead of 100 because the analyzer uses 2 clock cycles to jmove between trigger statement levels.

Set the Trigger Position

It's important that you specify the trigger position so that memory is filled with the data you need to see. For example, if you want to see what led up to a trigger condition, tell the analyzer to fill memory with pre-trigger data and position the trigger near the end of the analyzer's memory. This lets you see almost 2K of acquired data that occurred before the trigger condition. If you want to see what happens after a condition occurs, position the trigger near the beginning of memory so that most of the stored data is post-trigger data.

The Run Control menu in Figure 2-9 shows how to set up the analyzer so that you look at the events leading up to the trigger position.

```
TUE, JAN 10, 1989        Run Control        14:35   -DEFAULT

 Update Memory    : [1]          Display: [Timing]
 Trigger Position: [1536]        0 [            ] 2K
 Look for Trigger: [After Pre-Trigger Memory Full]
═══════════════════════════════════════════════════════════
 Compare           : [Manual]
 Compare Memory 1 to Memory: [2]


 Compare Mem Locations: [0000] to [2047]
 Use Channel Mask     : [SUB BEG ]
 Display Data at least: [5] seconds
═══════════════════════════════════════════════════════════
            ADD DAT
 Symbol     hex hex
 SUB BEG :  23  XX
═══════════════════════════════════════════════════════════
 Cursor: ▲▼◄►        Select: 0,2
```

Figure 2-9. Setting the trigger position. The trigger position is 1536, about three-quarters of the way through the analyzer's memory. The analyzer will look for the trigger after the pretrigger memory is full so that you can see those 1536 locations filled with acquired data.

Save a Setup

The analyzer features a storage menu to let you save your setup parameters. This saves you time because you can reload the saved setup with a few keystrokes instead of manually changing menus to match the previous setup. Section 5, Utility Menus, gives in-depth information about storing setups.

ACQUIRING DATA

These examples show you how to set up and acquire data for three cases:

- Trigger on anything
- Trigger on a specific value
- Capture a glitch

The procedures take you through the complete setup and acquisition for each example, showing the timing and state table displays for the acquisitions. Use the demonstration test card supplied with this manual to make the acquisitions. For more in-depth examples using the test card, refer to the 1230 Logic Analyzer Workbook.

Required Equipment

For these procedures, you need a 1230 Logic Analyzer with:

- Test circuit (part number: 671-0049-00)
- 1 P6444 or P6443 data acquisition probe
- 2 lead sets

The analyzer used in these examples is a 16-channel analyzer. You can follow the examples if you have more channels installed in your analyzer, but the test card will use only 16. If your analyzer has more than 16 channels, the menus will display the extra channel groups that the analyzer sets up by default.

You can use a P6444 or P6443 acquisition probe to do the examples in this section. These procedures refer to the P6444 probe, which lets you select a variable threshold voltage for sampling data. There are no differences in these procedures for using a P6443 probe rather than a P6444.

The test circuit (shown in Figure 2-10) generates sample signals. Test cards vary, so your acquisitions may not look exactly like those shown here. A 3 V battery powers the circuit, which includes 24 data pins, 3 clock pins, 3 grounds pins, and a glitch pin. All the pins are clearly labeled on the test card. The circuit oscillator runs at about 220 KHz, and several of the data lines have glitches mixed with data.

NOTE

To prolong the life of the battery, keep the test circuit turned off when not in use. Do not leave the test circuit turned on with the 1230 turned off; do not leave the test circuit turned off when the analyzer is turned on. In either case, the battery will drain rapidly. The battery' can be replaced with Tektronix part number 146-0069-00. Lift the clip over the battery to remove or insert the battery.

WARNING

Handle and dispose of lithium batteries with special considerations. Improper handling may cause fire, explosion, or severe burns. Do not recharge, crush, disassemble, heat the battery above 100° Celsius, incinerate, or expose contents of the battery to water. To avoid personal injury, observe the proper procedures for handling and disposing of lithium batteries given next.

If you remove the test circuit's lithium battery, dispose of the battery in accordance with local, state, and national regulations. Typically, small quantities of batteries (less than 20) can be safely disposed of with ordinary garbage in a sanitary landfill. Larger quantities must be sent by surface transport to a Hazardous Waste Disposal Facility. The batteries should be individually packaged to prevent shorting, and packed in a sturdy container that is clearly labeled:

Lithium Batteries -- DO NOT OPEN.

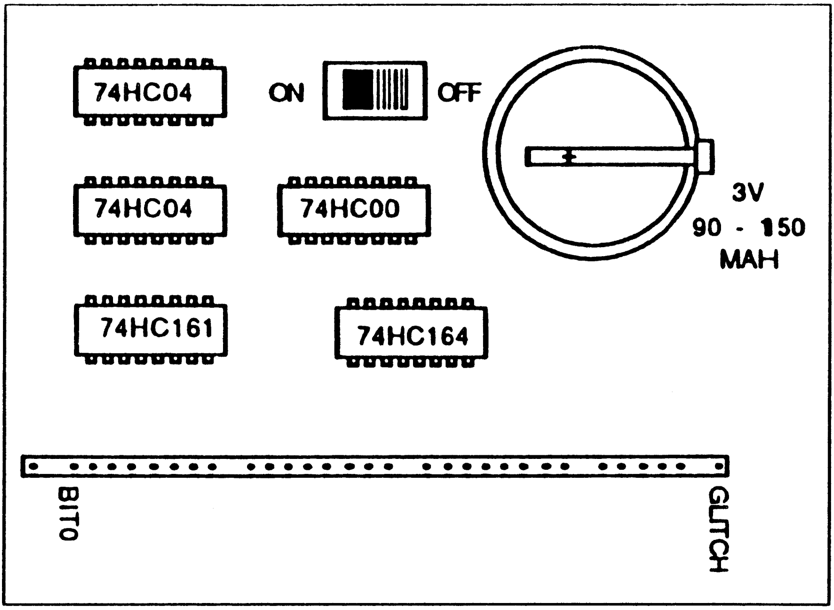


Figure 2-10. Test circuit.

PROBE CONNECTIONS

Follow these steps to connect the 16-channel acquisition probe to the analyzer.

1. Make sure power to the 1230 and test card is off.

CAUTION

Make sure power to the analyzer and test card is off. If you connect the acquisition probe to the 1230 and test card when power to the 1230 is off and power to the test card is on, too much power may flow through the probe and damage the probe.

2. Plug the acquisition probe into slot A on the analyzer's front panel.

3. Connect two of the lead sets to the test circuit as follows. Do not connect lead sets to probes yet. Figure 2-11 shows how to connect probe leads to the test circuit.
   a. Connect one lead set to test card bits 0-7. Start by connecting the black wire to bit 0; connect sequential wires (black, brown, red, orange, yellow, green, blue, violet), ending with the violet wire to bit 7.
   b. Connect the other lead set to bits 8-15 in the same manner, starting with the black wire to bit 8.
   c. Connect the white wire on each lead set to the ground (GND). There are three GNDs on the test circuit (one by bit 0 and two near the CLKs); you can connect to any of them. Do not connect the gray wire on either lead set.

NOTE

Always connect the white lead to ground. This improves performance by limiting noise and cross talk between channels.

4. Plug the lead set connected to bits 0-7 into the probe's low-order channels (D0-D7). Plug the other lead set into the probe's high-order channels (D8-D15). Grooves in the lead-set plugs show how to align the plugs in the acquisition probe sockets.
5. Turn on the analyzer, then turn on the test circuit.
6. Press D on the analyzer's front panel to reset the 1230 to the default setup. The header line at the top of the screen should now say -DEFAULT.
7. Press MENU to call up the Main menu. You're now ready to go through the examples.

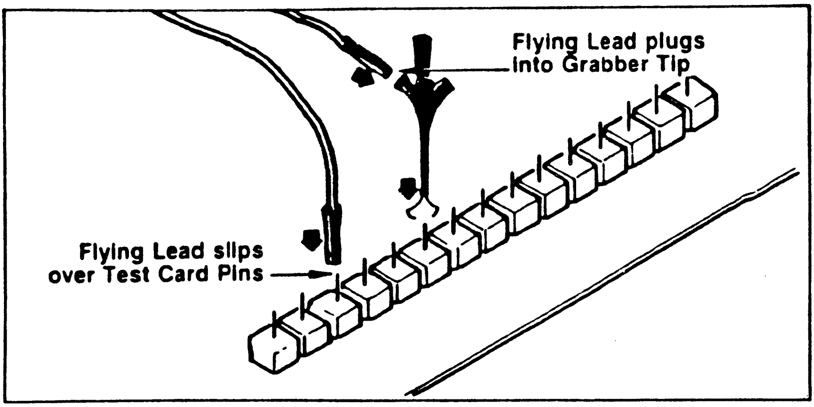


Figure 2-11. Connecting to the test card. Make sure that power to the SUT is off before connecting probe leads. When the probe leads are connected, make sure you turn the analyzer on before turning on the SUT.

EXAMPLE 1: TRIGGER ON ANYTHING

Because you're using the default setup to trigger on anything (Condition A = all don"t cares), you shouldn't have to change any of the setup menus to acquire data. If you do not have the default setup loaded, follow these steps:

1. Press NOTES and ENTER at the same time until the analyzer resets.
2. Press D to upload the default setup. The header line at the top of the menu should now say -DEFAULT. The setup name field should not include a highlighted asterisk. If your setup name field says * -DEFAULT, you've made changes to the default menus, and should reload the default setup by initializing the analyzer and pressing D again.
3. Display each of the setup menus to make sure they match the menus shown in Figures 2-12 through 2-16. For example, press MENU, then 0 to call up the Timebase menu.
4. Press START to acquire data on the first clock pulse after you press START.

As soon as the pretrigger memory is full, the analyzer looks for the trigger, which is all don't cares. Since the trigger condition is all don't cares, the analyzer triggers on the first data sample after pre-trigger memory is filled. When the analyzer acquires enough data to fill memory, it stops the acquisition and displays the data in timing format, as shown in Figure 2-17.

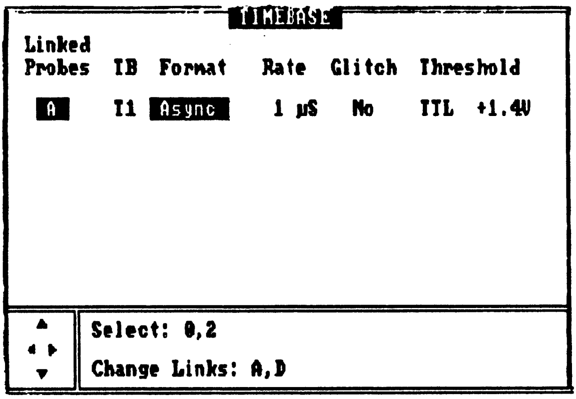


Figure 2-12. Default Timebase menu.

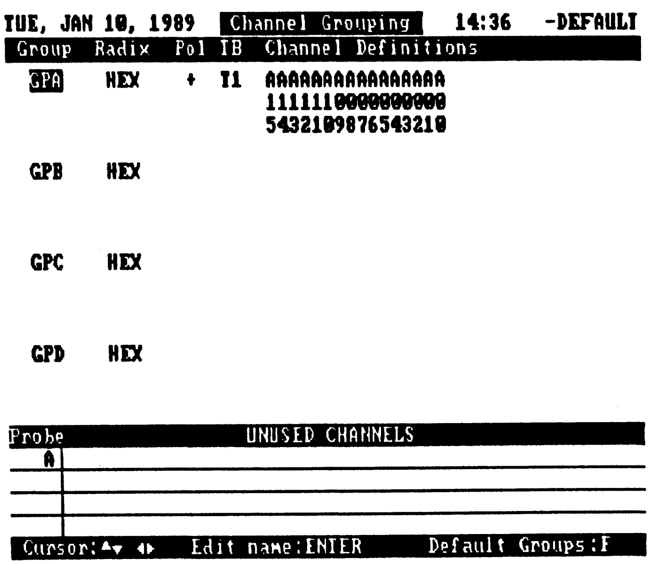


Figure 2-13. Default Channel Grouping menu.

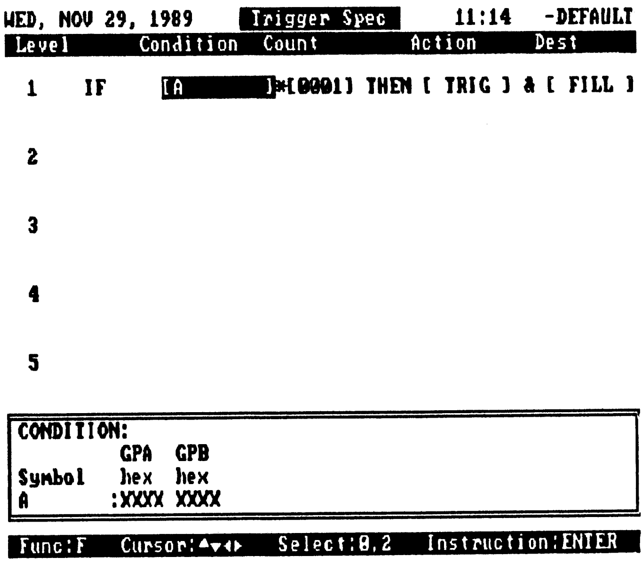


Figure 2-14. Default Trigger Spec menu. If condition A occurs once, the 1230 triggers, fills acquisition memory, then displays data. Since no specific data value is being looked for, the 1230 triggers on the first sample after the trigger is enabled.

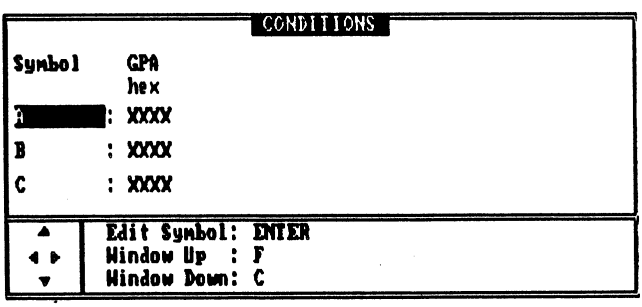


Figure 2-15. Default Conditions menu. The Conditions menu is a pop-up menu; in this case, it is displayed on top of the Trigger Spec menu.

```
TUE, JAN 10, 1989        Run Control      14 37    -DEFAULT

 Update Memory    : [1]      Display: [Timing]

 Trigger Position: [0512]     0 [   T            ] 2K

 Look for Trigger: [After Pre-Trigger Memory Full]
==============================================================
 Compare          : [Manual]

 Compare Memory 1 to Memory: [2]


 Compare Mem Locations: [0000] to [2047]
 Use Channel Mask     : [A      ]

 Display Data at least: [5] seconds

 Symbol     GPA
            hex
 A        : XXXX

 Cursor: ▲▼◀▶       Select: 0,2
```

Figure 2-16. Default Run Control menu. **New data will be stored in memory 1 and displayed in timing diagram format. The trigger will be positioned at memory location 512, and the 1230 will begin the trigger search after pretrigger memory (in this case, 511 clock cycles) is full.**

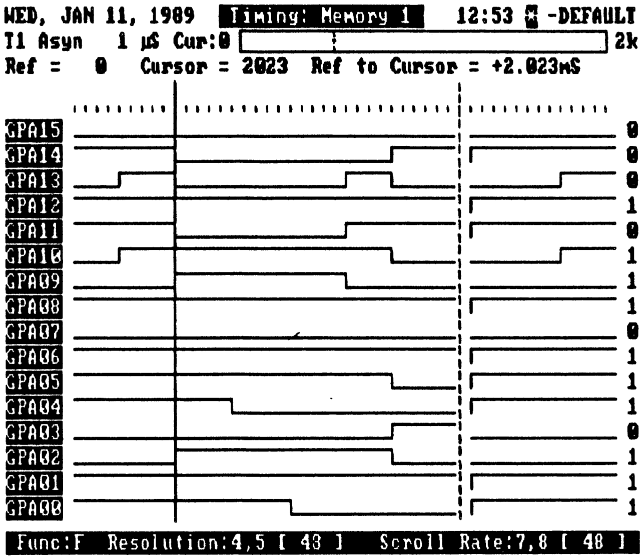


Figure 2-17. Timing diagram for Example 1. For this acquisition, the cursor appears near the end of memory. Use the cursor keys to move the cursor through the acquired data. To move the cursor quickly, set a higher scroll rate. You can set a scroll rate from 6 to 96 as long as the scroll rate is not greater than the resolution. The menu bar shown here tells you which keys to press to change the resolution and scroll rates.

You can change the display resolution for the Timing menu so that you show more or less clock cycles of acquired data on the screen at the same time. When you do this, an indicator in the memory bar changes size to show you how much of the memory you are actually viewing. Press 4 to increase the resolution. Increasing the resolution lets you show fewer clock cycles in more detail. If data transitions are close together, increasing the resolution lets you show these transitions in more detail.

NOTE

Use resolutions less than 2048 for making timing mensurements with the reference cursor.

Press 5 to decrease the display resolution. If you want an overview of the entire acquired memory, decrease the resolution to 2048. You can also use the resolution of 2048 to quickly locate a trigger or other event. Figure 2-18 shows the entire memory for the test card acquisition.

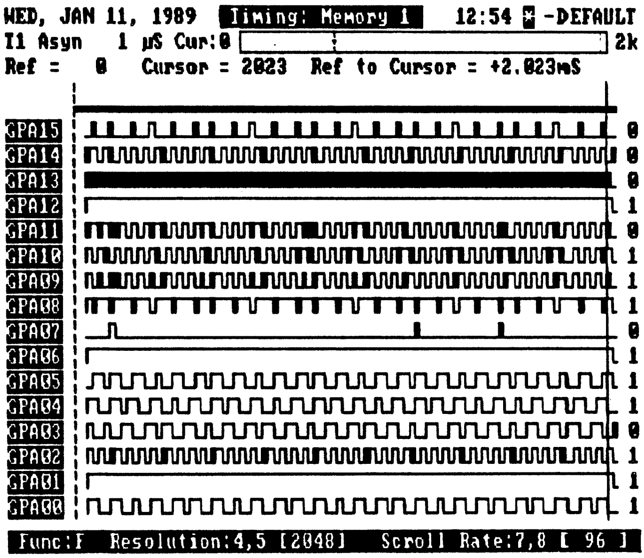


Figure 2-18. Decreased resolution. You can increase the display resolution to see more detail of fewer clock cycles, and decrease the display resolution to see more clock cycles at less detail. Press 4 to increase the resolution (show more detail of less acquisition cycles) and 5 to decrease the resolution. An indicator in the memory bar changes size as the display resolution changes, showing you how much of the memory you are actually viewing. This figure shows a resolution of 2048 so you can see the entire contents of memory in one screen. The solid bars mean that there is too much information for the detail to be shown at this resolution.

You can display acquired data in timing, state table, and disassembly formats (disassembly display requires that an optional disassembly probe be connected to the 1230). To show the acquired data in a state table, press MENU, then 7. The display should look like Figure 2-19.

```
WED, JAN 11, 1989    State: Memory 1     12 55 ■ -DEFAULT
 Loc    GPA
        hex
 1994   175F
 1995   175F
 1996   175F
 1997   175F
 1998   175F
 1999   BC5F
 2000   BC5F
 2001   BC5F
 2002   BC5F
-2003---BC5F-------------------------------------------
 2004   5973
 2005   5973
 2006   5973
 2007   5973
 2008   5973
 2009   5973
 2010   5973
 2011   5973
 2012   5973
 2013   5973
 Func:F   Scroll:▲▼   Cursor:◄►   Jump:ENTER   Radix:E
```

Figure 2-19. State table for Example 1.

The state table and timing diagrams show the data at the same locations. For example, if you display locations 1994 to 2013 in the state table, the timing diagram also displays those locations in the center of the screen, and vice-versa. The state table displays whatever 20 addresses are located around the cursor in the timing diagram screen, no matter what the timing display resolution is.

EXAMPLE 2: TRIGGER ON A SPECIFIC DATA VALUE

To help you locate specific events in your system under test, logic analyzers have the ability to recognize and trigger on a specific data pattern. This lets you store only the data you need.

In this example, you trigger on the hexadecimal value A6E5. To do this, the only change you need to make from Example 1 is to set up a condition defined as A6E5. Follow these steps to define a specific trigger condition:

1. Press MENU, then 3 to call up the Conditions menu.
2. Move the cursor into condition A's condition word and enter the value A6E5. Figure 2-20 shows the new Conditions menu.
3. Press MENU, then 2 to call up the Trigger Spec menu.
4. Move the cursor into the condition field, which should still specify A as the trigger condition. The viewport (window) at the bottom of the screen should list the new definition for condition A. Figure 2-21 shows the new Trigger Spec menu.

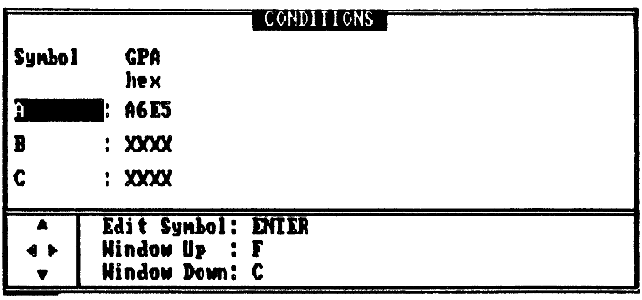


Figure 2-20. Define a specific condition.

```
WED, NOV 29, 1989    Trigger Spec       11:15  -DEFAULT
Level        Condition  Count        Action      Dest

 1     IF        [A        ]*[0001] THEN [ TRIG ] & [ FILL ]

 2

 3

 4

 5

CONDITION:
          GPA  GPB
Symbol    hex  hex
A        :A6E5 XXXX

Func:F    Cursor:▲▼◀▶    Select:0,2    Instruction:ENTER
```

Figure 2-21. Make sure the Trigger Spec matches.

5. Press START. The 1230 begins acquiring data on the first clock pulse after START is pressed. Once the pretrigger memory is full, the 1230 starts looking for the trigger. The next time A6E5 occurs, the 1230 will trigger, fill memory, and display data in a timing diagram.
6. Press MENU, then 7 to call up the corresponding state table. Figure 2-22 shows the state table after you find the trigger event in step 7.
7. Press 1 to begin a search for the trigger condition which is the default search event in the state table. If the trigger is not the current search event, press 0 or 2 to cycle through available conditions until you select the trigger. The menu bar at the bottom of the screen automatically changes to display the search event when you press 1 to do the search or 0 or 2 to select another event. Figure 2-22 shows the trigger event in the state table.

8. Call up the Timing menu again to see the trigger event in the timing traces. Figure 2-23 shows the trigger event in the timing diagram.

```
WED, JAN 11, 1989   State: Memory 1    13 00   -DEFAULT
Loc    GPA
       hex
0502   7CF3
0503   7CF3
0504   7CF3
0505   7CF3
0506   7CF3
0507   A6F4
0508   A6F4
0509   A6F4
0510   A6F4
0511   A6F4
TRIG   A6E5
0513   A6E5
0514   A6E5
0515   A6E5
0516   A6E5
0517   A6C6
0518   A6C6
0519   A6C6
0520   A6C6
0521   A6C6
Func:F    Search for: 0,2 [Trigger ]   Do Search: 1
```

Figure 2-22. State table with trigger event. The trigger event in the state table shows that the trigger occurred on the hexadecimal value A6E5.

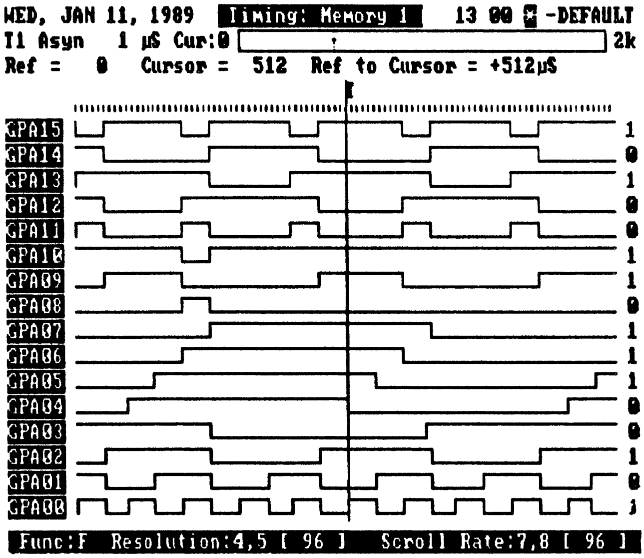


Figure 2-23. Timing diagram with trigger event. When the cursor is on the trigger point (location 512), the binary readout at the far right of the screen shows that the analyzer triggered on the hexadecimal value A6E5.

Example 3: Capturing Glitches

For simple acquisitions, you can acquire glitches generated by your SUT by toggling glitch capture from no to yes in the Timebase menu. Glitches are explained in detail in Appendix A, Logic Analyzer Concepts.

Probe Connections

The test circuit generates glitches intermixed with data on bits 9 and 12 as well as on the pin labeled Glitch. To change the connections to the test card so that you can monitor the glitches, follow these steps:

1. Turn off the test circuit, then turn off the analyzer.
2. Disconnect both lead sets from the acquisition probe.
3. Plug the lead set connected to bits 8-15 into the probe's low-order channels (D0-D07).

4. Disconnect the violet lead from bit 15 (probe channel D7) and attach it to the test card pin labeled GLITCH.
5. Turn on the analyzer, then turn on the test card.

Menu Setups

Follow these steps to set up to acquire glitches.

1. Call up the Timebase menu.
2. Move the cursor to the glitch field and press 0 or 2 to toggle glitch capture to yes. Turning glitches on may change your channel groups and conditions because the analyzer acquires glitches only on the lower 8 bits of the probe. Section 3, Setup Menus, describes the features and restrictions of glitch capture in detail. Figure 2-24 shows the Timebase menu.
3. Press MENU to save your change and leave the menu.
4. Press ENTER to confirm your change when the analyzer prompts you with this message:

```
              OK to change Glitch?
     May change Channel Groups and Conditions.
      Press ENTER to Proceed or MENU to Abort
```

   When you press ENTER, the analyzer is set up to acquire data and glitches on the lower 8 channels (0-7) of the acquisition probe and use the memory from channels 8-15 to store glitch information. When you confirm glitch capture, the analyzer automatically drops channels 8-15 from the channel groups and defined conditions. Figures 2-25 and 2-26 show the new Channel Grouping and Conditions menus.

5. Call up the Conditions Menu.
6. Move the cursor to condition A's condition word and enter all don't cares for the definition. Figure 2-26 shows the new Conditions menu.

7. Press START. The analyzer acquires data and glitches. When the acquisition is complete, the analyzer displays the data in timing format as shown in Figure 2-27. Glitches are shown as condensed, highlighted marks.

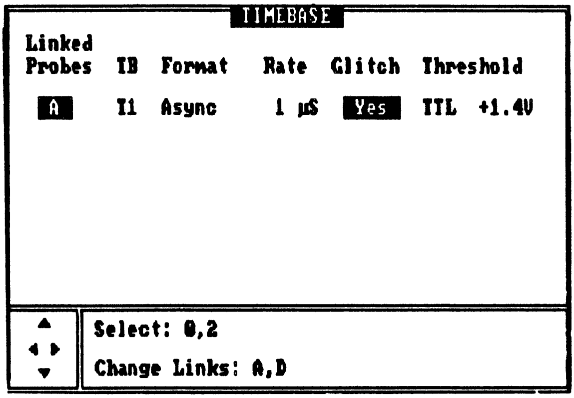


Figure 2-24. Turn glitch capture on.

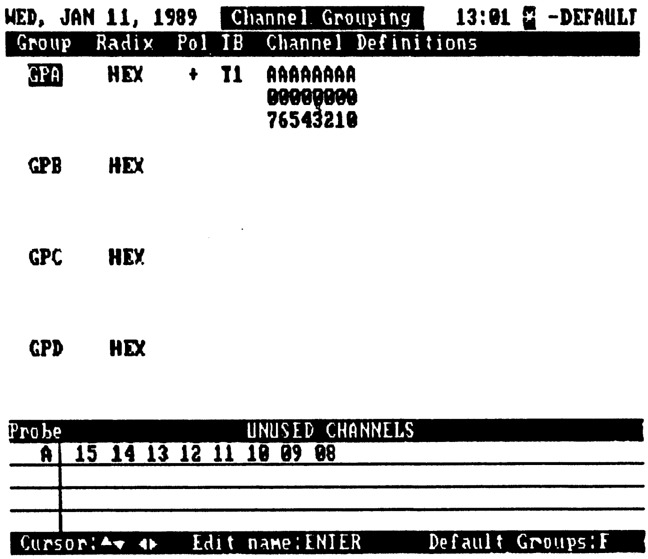


Figure 2-25. Channel groups for glitches. Glitches are acquired only on probe channels 0-7. When you turn glitch capture on, channels 8-15 are automatically removed from the channel groups and shown in the unused channels list at the bottom of the screen. You can't use these channels again until you turn glitch capture off in the Timebase menu.

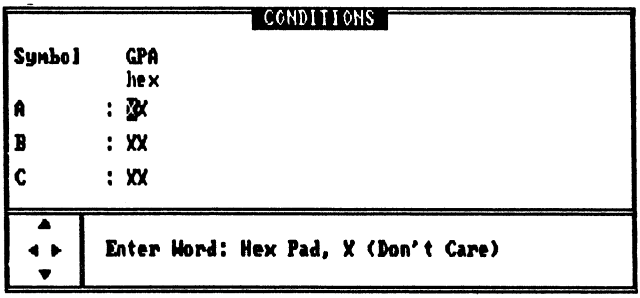


Figure 2-26. Conditions for glitches. Set condition A to all don't cares (X). Because the analyzer acquires glitches only on the lower 8 bits of the probes, the first two hex digits of condition A from the last example were automatically removed when you turned glitch capture on.

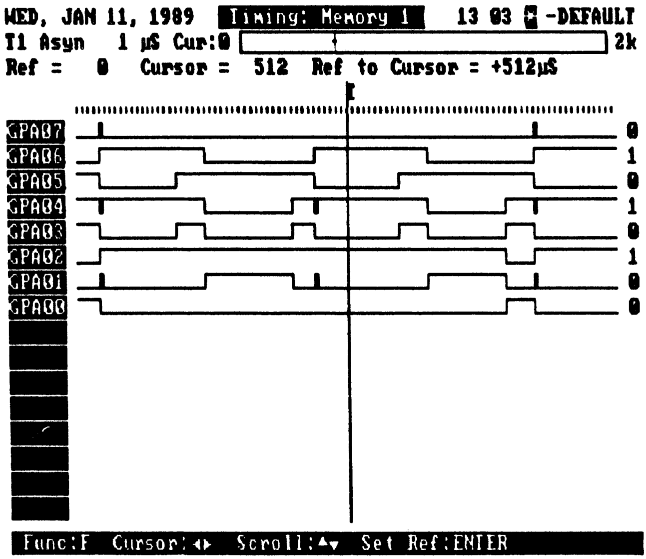


Figure 2-27. Timing diagram with glitches. Since you are set up for 16 channels and the analyzer acquires on only the lower 8 bits of the probe (channels A07-A00), the other 8 timing traces in the screen are blank. Because test cards vary, your acquisition may look different.

For more examples using the test card and 1230 Logic Analyzer, refer to the *1230 Logic Analyzer Workbook.*

Section 3
SETUP MENUS

This section describes the five menus that let you set up the analyzer for your application. For each menu, this section also gives you step-by-step procedures for changing setup parameters. Most changes can be made with one or two keystrokes. This section has the following discussions:

- Using the setup menus
- Making changes efficiently
- Timebase menu
- Channel Grouping menu
- Trigger Spec menu
- Conditions menu
- Run Control menu

USING THE SETUP MENUS

Before you start acquiring data, use the setup menus to specify which probes you're using, whether you want them to acquire in parallel or not, how input channels are grouped together, how trigger conditions are defined, and other configuration information. Table 3-1 briefly describes the setup menus.

Table 3-1
SETUP MENUS

Menu	Description
Timebase	Lets you specify how you want to link probes (for example, to acquire data in the same or a different timebase). This menu also lets you set the sampling rate and clocking format (asynchronous or synchronous), whether or not you want to capture glitches, and what the variable probe threshold voltage is for sampling data (P6444 probe only).
Channel Grouping	Lets you group input channels together so you can look at specific signals in different display radixes. This menu also lets you mask channels from the data display so you can show data meaningful to your application.
Trigger Spec	Lets you define how the analyzer triggers and how data is acquired and stored. You can use up to 14 specifications.
Conditions	Lets you define the conditions on which the analyzer triggers and acquires data. This menu also lets you set up comparison masks. In the State Table data menu (and the version 4.0 Timing menu) you can search for any condition (event) defined in the Conditions menu.
Run Control	Specifies which of the four analyzer memories acquired data is stored in, which memory location the trigger is stored at, and when to start looking for the trigger condition. The default data display format can be state, timing, or disassembly (optional). You can also control memory comparison features in this menu.

MAKING CHANGES EFFICIENTLY

In general, when setting up the analyzer for your application, use the menus in the order they are listed in the Main menu. The reason for this is that they are listed in the order of their impact on other menus. These three are particularly important:

1. Timebase menu
2. Channel Grouping menu
3. Conditions

For example, the Timebase menu is listed first. Changing your probe configuration in this menu often changes channel groups and conditions. If you have 32 input channels (probes A and B) grouped together and then change probe links so that probe A acquires data separately from probe B, the analyzer must separate probe B's input channels from the group containing probe A's input channels. Changing the probe links before setting up your channel groups lets the analyzer show you what channel groupings are valid before you spend time grouping input channels the wrong way.

It is particularly important that you set up the Timebase, Channel Grouping, and Conditions menus in that order. However, changing the Trigger Spec and Run Control menus won't affect many fields in the other menus. For example, changing the data display in the Run Control menu (the last setup menu) from state to timing doesn't affect parameters in any other setup menu.

TIMEBASE MENU

The Timebase menu, shown in Figure 3-1, lists the available probes for the analyzer: one for the analyzer's basic 16 channels, and one additional probe for each installed 16-channel 1230E1 Expander Card (refer to Appendix C for information about expander cards).

The Timebase menu also shows how the installed probes are linked, which timebase they acquire in, the clocking format, data sampling rate, whether or not glitch detection is enabled, and what logic threshold voltage is being used for each probe.

The Timebase menu lets you:

- Link probes for parallel or separate data acquisition
- Specify asynchronous or synchronous clocking
- Set the asynchronous sampling rate
- Tell the analyzer to capture glitches or not
- Change the threshold voltage (P6444 probe only)

NOTE

A P6443 probe acquires data at TTL voltage levels. With a P6444 probe, you can acquire data at ECL or HCMOS levels as well as at TTL. You can also select a variable threshold voltage from -0.1 V to -9.0 V and from +0.1 V to +9.0 V.

Once you've set up your probe links, you can define the other parameters in the Timebase menu. Changes are stored when you save the menu and return to the analyzer's Main menu. If you turn the analyzer off before storing the Timebase menu, the analyzer doesn't save your changes. Make sure you save the Timebase menu after you make changes to the menu.

The rest of this discussion explains how to set up the Timebase menu for different data acquisitions. Figure 3-1 shows a Timebase menu. The numbered list that follows Figure 3-1 explains each field in the menu.

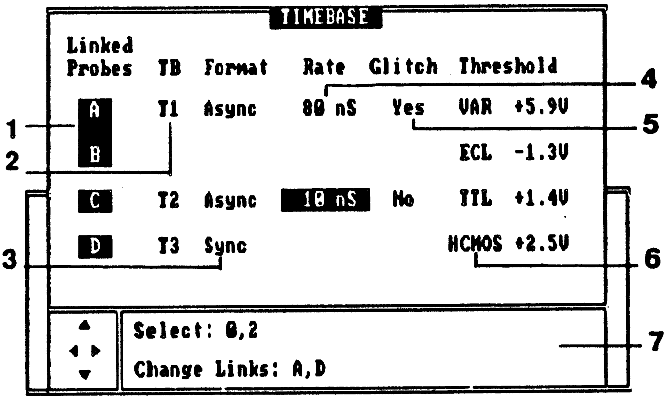


Figure 3-1. Timebase menu. This screen shows a 64-channel configuration. Probes A and B are linked; probes C and D acquire on separate timebases. Probes A, B, and C use the clock in the analyzer at rates you define (asynchronous acquisition). The analyzer doesn't acquire glitches for synchronous acquisitions.

[1] **Probe Links.** Highlighting shows how probes are linked together. Linking probes puts them in the same timebase. Press A or D (don't use the cursor) to cycle through probe links. Press MENU, then ENTER (to confirm the change in links) to save the new links and leave this menu.

[2] **Timebase.** This field lists each probe group's timebase. For example, probe A might be in timebase T1, probes B and C linked in timebase T2. This field automatically changes when you change the probe links. You cannot select this field.

[3] **Sampling clock format.** You can choose Async or Sync. Synchronous sampling uses the SUT clock to determine the data sampling rate. Asynchronous sampling uses the analyzer's internal clock, which has a sampling rate you can set. To change the clock format, move the cursor to this field and press 0 or 2.

[4] **Sampling rate.** Asynchronous acquisitions can be taken at rates between 10 ns and 40 ms. Place the cursor on this field and press 0 or 2 to choose faster or slower rates. Synchronous sampling uses the clock rate of the SUT.

[5] **Glitch capture.** The analyzer lets you capture glitches when you sample data asynchronously at a rate at or under 25 MHz (40 ns). When glitches are turned on, the analyzer acquires data on the probe's lower 8 bits (0-7). To turn glitches on and off, place the cursor on the field and press 0 or 2.

[6] **Probe threshold.** If you're using a P6444 probe, you can set the voltage threshold at which the analyzer recognizes a data high or low. You can set HCMOS +2.5 V, ECL -1.3 V, TTL +/-1.4 V, or a variable threshold from -9.0 to +9.0 V (excluding 0 V). The P6443 probe sets the threshold to TTL +1.4 V.

[7] **Menu bar.** This bar lists the keys to press for menu features.

Probe Configuration and Links

The 1230 can acquire data on 16, 32, 48, or 64 channels, depending on how many 16-channel 1230E1 Expander Cards you install. You can use one 16-channel acquisition probe for each set of 16 input channels the analyzer is configured for.

Linking probes lets you set up the probes to acquire in parallel so that they use the same timebase (sampling rate and format). For example, you can set up two 16-channel probes to acquire data as if they were one 32-channel probe. You can do this for up to 4 acquisition probes so that up to 64 channels acquire data in parallel.

You can also link the probes separately so that they acquire data in up to 4 timebases. For example, if you're using two probes, you can acquire 16 channels of data in one timebase at one sampling rate, and 16 channels of data in another timebase at a different sampling rate.

If you're using one probe (16 channels), the Timebase menu shows only the one probe on the screen. There are no probe links for a 16-channel (one-probe) configuration.

Probe links are the highest level of control in a setup. When you change probe links, the analyzer acquires data with a new setup. You can no longer acquire or display data with the previous setup until you change the probe links back to match the previous setup. These things happen when you change probe links:

1. Channel groups are changed (if necessary) so that only input channels acquired in the same timebase will be in the same channel groups.
2. Condition words are changed (if necessary) to reflect the new channel groups.

Changing Probe Links

To change probe links, follow these steps:

1. In the Timebase menu, press A or D to cycle through the probe links to the configuration you want. Figures 3-2 through 3-5 show the probe links possible for analyzer configurations.
2. Press MENU to save your change and leave the Timebase menu.
3. Press ENTER to confirm the new probe links when the analyzer prompts you with this message:

```
OK to change links?
May change Channel Groups and Conditions
Press ENTER to Proceed or MENU to Abort
```

Once you confirm the probe links, the analyzer creates the new probe configuration. If you press `MENU`, the analyzer ignores your new links (and all other changes you made since the Timebase menu was last stored) and returns the Timebase menu to the previous setup.

If the channel groups change because of the new links, the analyzer makes sure that only channels that acquire data in the same timebase are in the same channel groups. The analyzer also updates the condition words to reflect channel group changes. (These changes are explained in more detail in the discussions about channel groups and conditions.)

The following figures show the probe links you can specify for an analyzer with 16, 32, 48, and 64 channels. If your analyzer has only 16 channels, you have only one acquisition probe and there are no probe links. An analyzer with 16 channels always uses timebase T1.

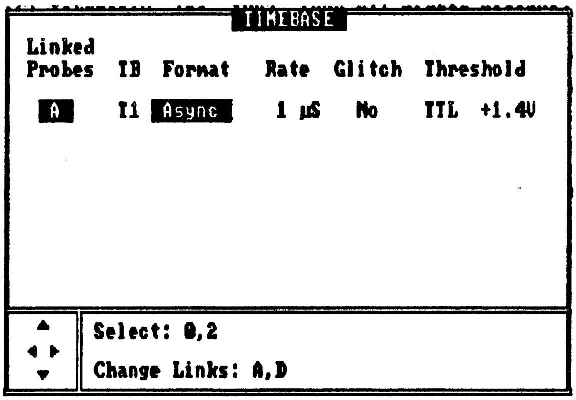
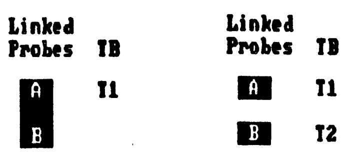


Figure 3-2. **Probe link for 16 channels. If you have only 16 channels, you have the basic analyzer configuration with no expander cards installed. With only one 16-channel acquisition probe, there are no probe links, and the only timebase available is T1.**



6878-37

Figure 3-3. **Probe links for 32 channels. For an analyzer with 32 channels, you can use up to two timebases with up to two links.**

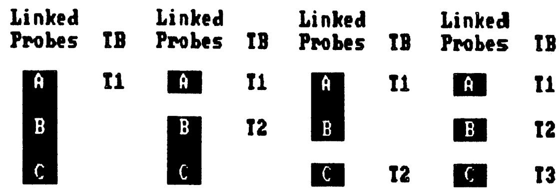


Figure 3-4. Probe links for 48 channels. For an analyzer with 48 channels, you can specify up to three timebases with up to four links.

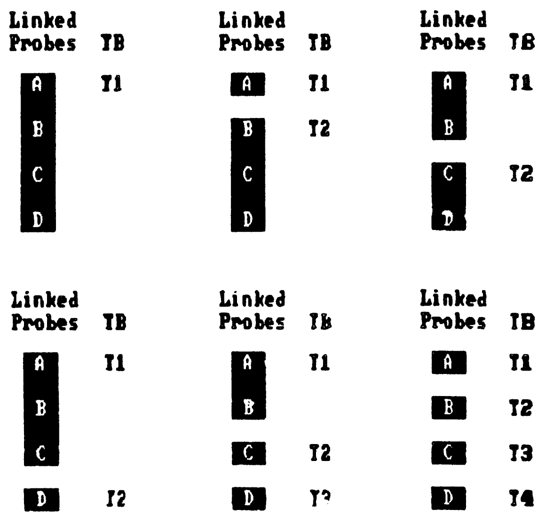


Figure 3-5. Probe links for 64 channels. For an analyzer with 64 channels, you can use up to four timebases and six links.

Acquiring and Displaying Data with Different Links

You can change the probe links to acquire data with a different setup. There are two ways to change links so that a setup doesn't match a previous acquisition's setup. You can re-link the available probes, or you can remove one or more of the 16-channel 1230E1 Expander Cards (which deletes probes from the setup). In either case, you cannot display data acquired with a previous setup until the current probe links match those of the previous setup. This means you must relink the probes or reinstall the expander cards and relink the probes.

For example, for a 32-channel analyzer, you can change the default setup of probes A and B linked in T1 to a setup in which A and B acquire data in the two timebases T1 and T2. To display the data acquired when both probes were linked in T1, you have to change the probe links back to the setup in which the data was originally acquired. This means linking probes A and B in T1 at the same sampling rate and format they had for the previous acquisition.

To acquire data with different links, and then display data acquired with previous links, follow these steps:

1. Store the current setup in the Storage menu (section 5 describes the Storage menu).
2. Change the probe links and acquire data on the new links.
3. Store the new setup in the Storage menu.
4. Change the probe links back to the first setup.
5. Reload the first setup (from the Storage menu) and display the first acquisition.

Probe Link Examples

The next three examples show how changing probe links can affect channel groups and conditions.

Example 1: Fewer to More Links. In this example, the analyzer has 32 channels, all linked in timebase T1. The Channel Grouping menu shows that you've grouped all 32 channels together in the group called ADD. The Conditions menu shows that a two-byte hexadecimal word represents the input channels for the group ADD. Figure 3-6 shows these menus.

When you change probe links so that each probe acquires data separately, the analyzer automatically changes the channel groups and conditions to match the links. Figure 3-7 shows the menus after you change the probe links so that probe B acquires data in a different timebase than probe A.

The analyzer updates the channel groups so that only input channels in the same timebase are in the same channel group. The analyzer uses the first channel in the list (the leftmost channel) to determine which timebase the input channels are using. In this example, since all 32 channels were in GPA and probe A's channels were listed first, the analyzer deletes the 16 channels from probe B and puts them in the unused list at the bottom of the screen.

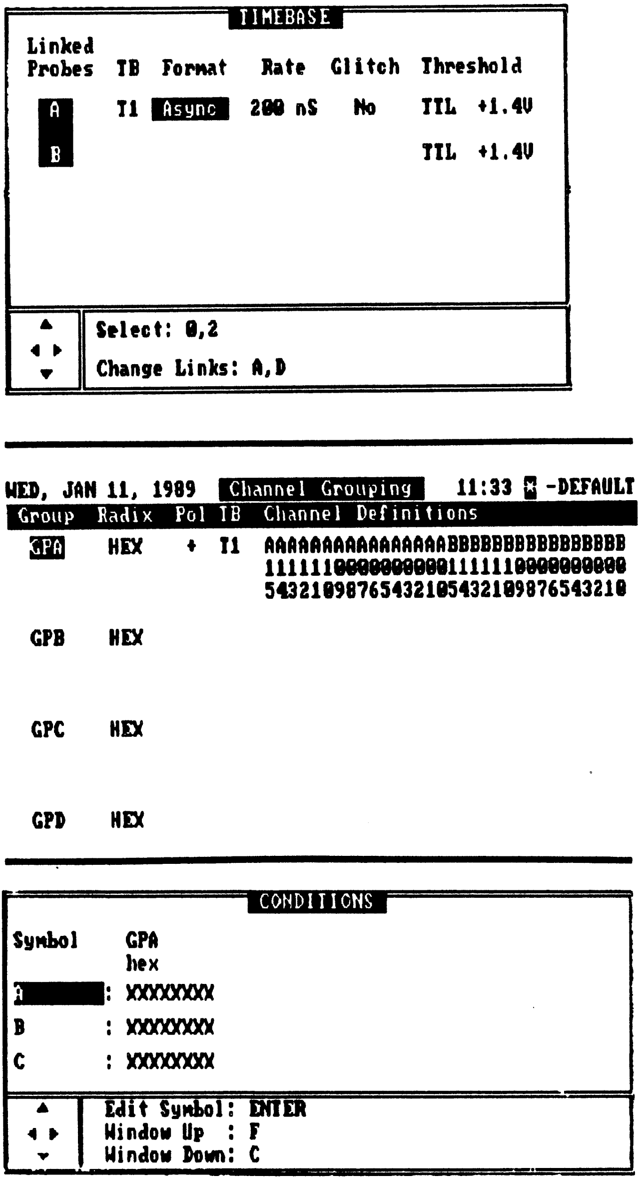


Figure 3-6. **One timebase for 32 channels.** All 32 channels are linked in timebase T1. In the Channel Grouping menu, all 32 channels are in group GPA. The conditions menu reflects the channel group setup, showing a two-byte condition word for group GPA.

```
                         TIMEBASE
Linked
Probes   TB   Format    Rate   Glitch   Threshold

  A      T1   Async    200 nS    No     TTL  +1.4V

  B      T2   Async      1 µS    No     TTL  +1.4V
```

```
WED, JAN 11, 1989   Channel Grouping   11:34   -DEFAULT
Group  Radix  Pol  TB  Channel Definitions
GPA    HEX     +   T1  AAAAAAAAAAAAAAAA
                       1111110000000000
                       5432109876543210

GPB    HEX


GPC    HEX


GPD    HEX


Probe              UNUSED CHANNELS
  A
  B  15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00


Cursor:▲▼ ◀▶     Edit name:ENTER        Default Groups:F
```

```
                    CONDITIONS
Symbol      GPA
            hex
A         : XXXX
B         : XXXX
C         : XXXX

  ▲    Edit Symbol: ENTER
 ◀ ▶   Window Up   : F
  ▼    Window Down: C
```

Figure 3-7. **Probe link change and affected menus.** After the links are changed, 16 channels are in timebase T1 and 16 in timebase T2. Because probes A and B are no longer in the same timebase, probe B's channels can no longer be in the same channel group as probe A's channels. The 1230 deletes probe B's channels and puts them in the unused list at the bottom of the screen. The Conditions menu shows that the condition word is only 1 byte long now.

Example 2: More to Fewer Links. In this example, the analyzer has 32 channels, 16 (probe A) in timebase T1 and 16 (probe B) in T2. The Channel Grouping menu shows how the channels are grouped, with each group containing only the input channels from the probe in each timebase. The Conditions menu shows four condition words since you're using four channel groups. Figure 3-8 shows these menus.

When you link probes A and B together, all channels are in the same timebase. The timebase rate is adjusted so that probe A's rate and sampling format is used for the timebase. Since all channels are now in the same timebase, you can group them together any way you like (Channel Grouping menu). Because of this, all previous channel groups are still valid; the analyzer makes no changes to the groups. The only change the analyzer makes is to the timebase field TB to show that all groups are now in timebase T1. Figure 3-9 shows the Timebase and Channel Grouping menus after you change the probe links.

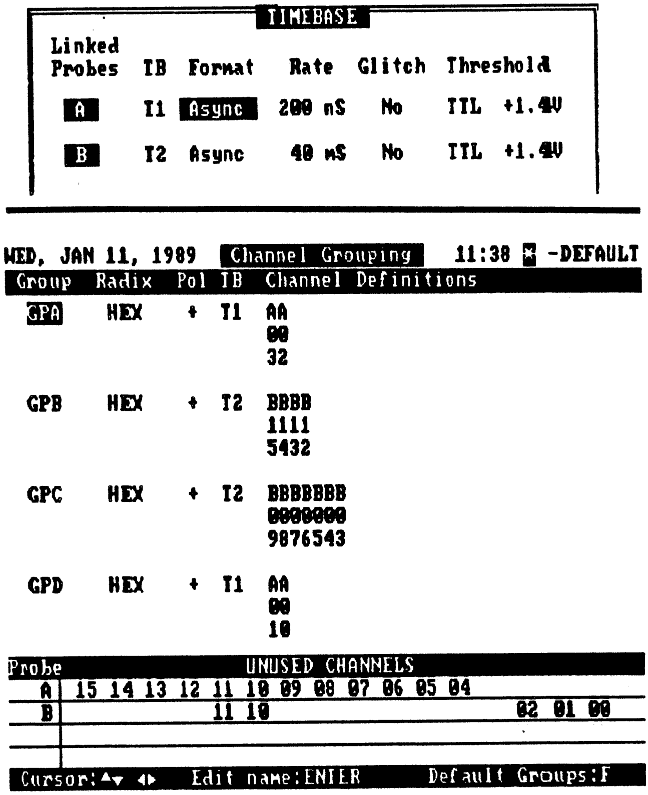


Figure 3-8. **Two timebases for 32 channels. Timebase T1 has 16 channels and timebase T2 has 16 channels. In the Channel Grouping menu, probe A and B's channels can't be mixed since the two probes aren't in the same timebase. The conditions reflect the channel groups.**

TIMEBASE

Linked Probes	TB	Format	Rate	Glitch	Threshold	
A	T1	Async	200 nS	No	TTL	+1.4V
B					TTL	+1.4V

WED, JAN 11, 1989 Channel Grouping 11 39 -DEFAULT

Group	Radix	Pol	TB	Channel Definitions
GPA	HEX	+	T1	AA 00 32
GPB	HEX	+	T1	BBBB 1111 5432
GPC	HEX	+	T1	BBBBBBB 0000000 9876543
GPD	HEX	+	T1	AA 00 10

Probe	UNUSED CHANNELS
A	15 14 13 12 11 10 09 08 07 06 05 04
B	11 10 02 01 00

Cursor:▲▼ ◀▶ -Edit name:ENTER Default Groups:F

Figure 3-9. Changing to one timebase. After the links are changed, all 32 channels are in the same timebase. Since the current channel groups are still valid, the analyzer makes no changes to the channels definitions, and changes only the timebase fields to show that the probes are in the same timebase, T1. The conditions menu also shows no changes.

Example 3: Re-linking with Mixed Groups. In this example, the analyzer has 32 channels, all linked in timebase T1. The Channel Grouping menu shows that groups ADD and STB have mixed probe A and probe B's input channels. Group INT has only channels from group B. Figure 3-10 shows the menus for this example.

When you change probe links, the analyzer looks at the first channel in the group to determine the timebase for the group. Channels that no longer match the group's timebase are deleted from the group and placed in the unused list at the bottom of the screen.

In this example, since probe B's channels were listed first, group ADD is timebase T2. Probe A's channels are not in T2, so those channels are deleted from the group and placed in the unused list. Because group INT has only probe B's channels, that group is also in timebase T2. Group STB has probe A's channels listed first, so STB is now in timebase T1, and probe B's channels are deleted and placed in the unused list.

Figure 3-11 shows these menus.

```
                         TIMEBASE
Linked
Probes  TB  Format   Rate   Glitch  Threshold

  A     T1  Async    80 nS   No     TTL  +1.4V

  B                                 TTL  +1.4V
```

```
WED, JAN 11, 1989  Channel Grouping   11 42 ▒ -DEFAULT
Group  Radix  Pol TB  Channel Definitions
ADD    HEX    +   T1  BBBBBBBB AAAAAAAA
                      00000000 00000000
                      76543210 76543210

DAT    HEX    +   T1  BB
                      11
                      54

STB    HEX    +   T1  AAAA BB
                      1100 11
                      1098 10

GPD    HEX


Probe               UNUSED CHANNELS
  A | 15 14 13 12
  B |       13 12        09 08
    |
    |
Cursor:▲▼ ◄►    Edit name:ENTER        Default Groups:F
```

```
                    CONDITIONS
Symbol      ADD  DAT STB
            hex  hex hex
A         : XXXX X   XX
B         : XXXX X   XX
C         : XXXX X   XX

  ▲     Edit Symbol: ENTER
 ◄ ►    Window Up  : F
  ▼     Window Down: C
```

Figure 3-10. One timebase with mixed channels. All 32 channels are linked in timebase T1, so the input channels from the probes can be mixed in the groups.

```
                         TIMEBASE
 Linked
 Probes   TB  Format    Rate   Glitch  Threshold

   A      T1  Async     80 nS   No     TTL  +1.4V

   B      T2  Async      1 mS   No     TTL  +1.4V



   ▲    | Select: 0,2
  ◄ ►   |
   ▼    | Change Links: A,D
```

```
WED, JAN 11, 1989  Channel Grouping   11:43  -DEFAULT
 Group  Radix  Pol  TB  Channel Definitions
 ADD    HEX     +   T2  BBBBBBBB
                        00000000
                        76543210

 DAT    HEX     +   T2  BB
                        11
                        54

 STB    HEX     +   T1  AAAA
                        1100
                        1098

 GPD    HEX
```

```
                       CONDITIONS
Symbol      ADD DAT STB
            hex hex hex
A         : XX  X   X
B         : XX  X   X
C         : XX  X   X

   ▲    | Edit Symbol: ENTER
  ◄ ►   | Window Up  : F
   ▼    | Window Down: C
```

Figure 3-11. **Invalid channels deleted.** After the links are changed, Probe A and B's channels are in different timebases. The analyzer looks at the first (leftmost) channel to determine the timebase for the group. For ADD, probe A's channels are deleted; for STB, probe B's channels are deleted. Since INT has input channels only from one probe, the analyzer doesn't change that group. The timebase field lists each group's current timebase.

Choosing the Sampling Format

You can acquire data asynchronously or synchronously. Asynchronous sampling uses the internal analyzer clock. Synchronous sampling uses the SUT clock. Press 0 or 2 while the cursor is in the format field to toggle between Sync and Async sampling formats.

Each timebase can have an asynchronous or synchronous sampling format:

- You can have as many synchronous timebases as you have acquisition probes acquiring data.
- You can have only two asynchronous timebases. (For example, if you have three timebases, two can acquire data asynchronously but the third must acquire data synchronously.)
- If you have two or more probes linked together and the link doesn't include probe A, the link can only acquire data asynchronously.

Figure 3-12 shows different probe links and formats.

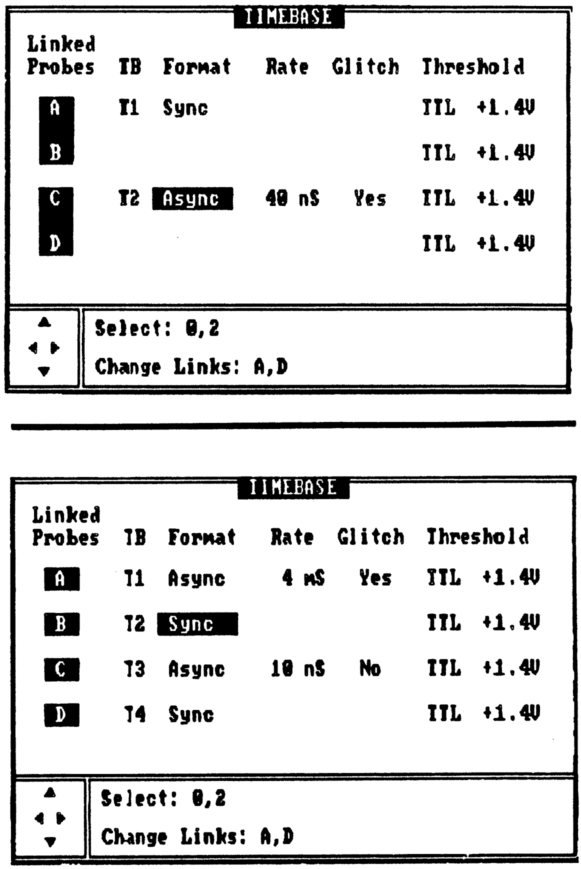


Figure 3-12. **Probe links and formats.** In the first example, T1 can be synchronous or asynchronous. However, the link for T2 has more than one probe and does not contain probe A, so it can only use an asynchronous format. In the second example, T1 and T3 are asynchronous. Since there are already two timebases acquiring asynchronously, T2 and T4 must be synchronous.

Setting the Async Sampling Rate

You can set data acquisition rates for asynchronous acquisition. (Synchronous acquisition uses the SUT clock rate.) You can have up to two asynchronous timebases in a setup, so you can have two asynchronous rates. Figure 3-12 shows different sampling rates.

Choosing the Sampling Rate

You can choose asynchronous rates in the range from 10 ns (100 MHz) to 40 ms (25 Hz). To select the rate, place the cursor on the field, then press 0 or 2 to cycle through the sampling rates faster or slower.

Choosing a Rate of 10 or 20 ns

The analyzer asks for confirmation if you change the rate to 10 or 20 ns. This is because at 20 ns, the analyzer acquires data only on the lower eight bits (input channels 7-0) of the probe. At 10 ns, the analyzer acquires on the lower four bits (input channels 3-0) of the probe. By changing to these rates, you may change channel groups and conditions.

For example, all 16 channels from probe A are in channel group GPA. If you change the probe's sampling rate to 20 ns, the analyzer acquires on only the lower 8 bits (channels A07-A00). To make the channel groups match the sampling rate, the analyzer deletes the upper 8 bits (A15-A08) from the group GPA. These channels are shown dimmed in the unused list at the bottom of the screen so that you know you can't use them in the current setup. The Conditions menu reflects the channel groups, so the condition word for GPA is also shortened 8 bits.

If you change the sampling rate to 10 or 20 ns and then press MENU to leave the Timebase menu, the analyzer prompts you for confirmation. Press ENTER to confirm and MENU to abort a rate change. If you press ENTER, the analyzer makes any channel groups are valid for the selected sampling rate. If you press MENU to abort the rate change, any changes you made to the menu since it was last called up are cancelled.

Figures 3-13 and 3-14 show how rate changes can affect channel groups and conditions.

```
                          TIMEBASE
 Linked
 Probes   TB   Format    Rate    Glitch   Threshold

   A      T1   Async     40 nS    No      TTL  +1.4V

   B      T2   Async     80 nS    No      TTL  +1.4V
```

```
WED, JAN 11, 1989   Channel Grouping     11:51  -DEFAUL1
Group  Radix  Pol TB  Channel Definitions
 ADD    HEX    +  T1  AAAAAAAAAAAA
                      110000000000
                      109876543210

 SIB    HEX    +  T1            AAAA
                                1111
                                5432

 DAT    HEX    +  T2      BBBBBBBBBBBBBBBB
                          1111110000000000
                          5432109876543210

 GPD    HEX
```

```
                        CONDITIONS
Symbol      ADD SIB DAT
            hex hex hex
A         : XXX X   XXXX
B         : XXX X   XXXX
C         : XXX X   XXXX
```

Figure 3-13. **Timebase rates. When the sampling rate is 40 ns or greater, the acquisition probes acquire on all 16 channels. The channel groups show that all 16 channels are still available for use. The condition words reflect the channel groups.**

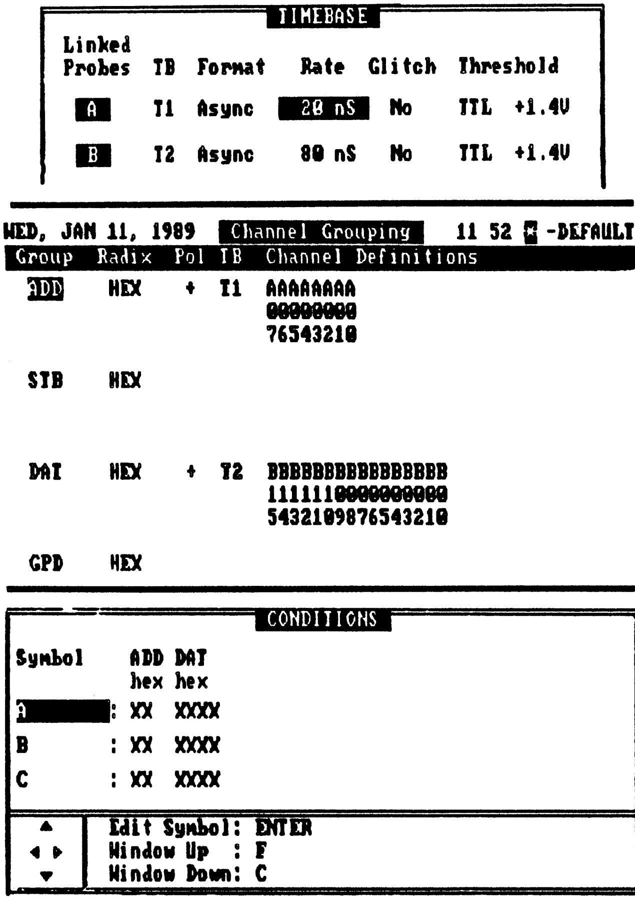


Figure 3-14. **Rate change and affected menus.** If you change probe A's sampling rate from 40 ns to 20 ns, the probe acquires on only the lower 8 bits (input channels 7-0). The analyzer deletes the upper 8 bits (A15-A08) from the channel groups and puts them in the unused list in dimmed video (to show that they are now unavailable for use). Because group STB had channels only from the upper 8 bits (A15-A12), that group is no longer active. Group ADD lost 4 channels also (A11-A08). Group DAT didn't lose any channels because you didn't conditions reflect the change in channel groups.

Glitch Detection

You can acquire and display glitches at 40 ns and slower rates. The analyzer ignores attempts to turn glitch detection on for rates of 10 or 20 ns.

Turning Glitches On

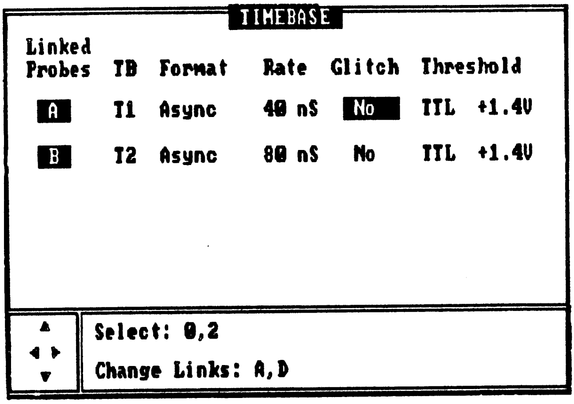
When you turn glitches on, the acquisition probes acquire data on only the lower 8 bits (for example, input channels A07-A00 for probe A). Follow these steps to turn glitches on (or off) in the Timebase menu.

1. In the Timebase menu, move the cursor to the glitch field and press 0 or 2 to toggle glitches on or off (yes for on; no for off).
2. Press MENU to save your change and leave the menu.
3. Press ENTER to confirm your change when the analyzer prompts you with this message:

```
OK to change Glitch?
May change Channel Groups and Conditions
Press ENTER to proceed or MENU to abort
```

When you confirm that you want to capture glitches, the analyzer checks the channel groups to see if you're using the upper 8 channels of the acquisition probes (15-08) for which glitch detection is turned on. If you're using those channels in groups, the analyzer deletes those channels and puts them in the unused list at the bottom of the Channel Grouping menu. They are dim to show that they are now unavailable. If you aren't using them, the analyzer leaves them in the unused list and dims them to show that you can no longer use them in the current setup.

Figures 3-15 and 3-16 show how turning glitch detection on can affect channel groups. The Conditions menu, not shown in the figures, always reflects the channel groups.



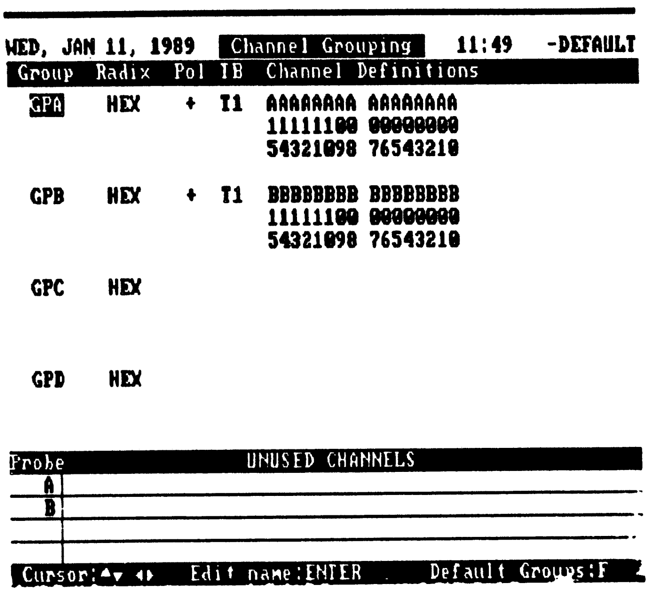


Figure 3-15. **Channel groups without glitches on.** The Timebase menu shows that glitches are not turned on for either acquisition probe. Since probe B is in a different timebase than probe A, changing probe A's setup parameters won't affect probe B's setup parameters. The channel groups show the default setup: 16 channels from probe A in group GPA; 16 channels from probe B in group GPB. The spaces are shown to make the screen easy to read for this example.

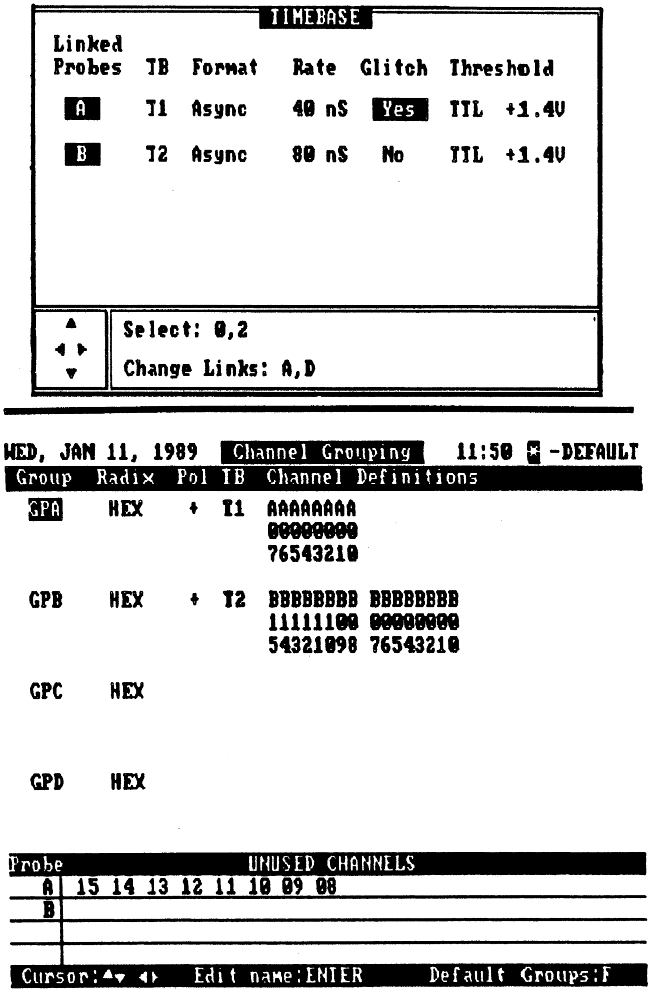


Figure 3-16. **Channel groups with glitches on. The Timebase menu shows that you turned glitches on for probe A. This means that probe A now acquires on only the lower 8 bits (input channels A07-00). The channel groups show that the upper 8 bits (input channels A15-A08) have been deleted and put in the unused list at the bottom of the screen. You cannot use these channels until you turn glitch detection off again in the Timebase menu. The spaces in the screen are shown to make it easy to read this example.**

Defining Glitches

If you turn glitch detection on, you can incorporate glitches in the condition words you define in the Conditions menu. You can define glitches when the condition word is in a binary radix (this is set in the Channel Grouping menu). Follow these steps to define glitches:

1. In the Timebase menu, turn glitch detection on.
2. In the Channel Grouping menu, make sure the channel group for which you want to define glitches has a binary input radix.
3. In the Conditions menu, enter glitch definitions in the condition words as you would enter binary values for input channels. The menu bar lists the types of glitches (generic, rising-edge, and falling-edge) you can define.

Refer to the discussion on conditions, later in this section, for more information about defining glitches in condition words.

Probe Threshold

The 1230 has two acquisition probe options: P6444 and P6443. The P6444 probe lets you choose the logic threshold voltage for each probe. The P6443 probe only acquires data at TTL logic levels.

The P6444 probe has DIP switches on the top of the probe so you can set the external, qualifier, and clock lines quickly and easily. The P6443 probe does not use DIP switches. Instead, you set up the external, qualifier, and clock lines for the P6443 probe by the way you connect those lines to your SUT. Appendix C, *Installation and Setup*, explains how to set up the external, qualifier, and clock lines for the acquisition probes.

Using Different Thresholds for Probes

With the P6444, even if probes are linked in the same timebase, you can set each probe's threshold differently. For example, if you have probes A and B linked in timebase T1, probe A could trigger on a threshold of ECL -1.3 V and probe B could trigger on a variable threshold of +7.6 V. For the P6444 probe, you can choose these threshold voltages:

TTL +1.4 V
HCMOS +2.5 V
ECL -1.3 V
VAR. -9.0 to +9.0 V (excluding 0 V)

Selecting a Threshold Voltage

To select the threshold voltage, move the cursor to the threshold field and press 0 or 2 to cycle through the voltage options. To set a variable threshold voltage, follow these steps:

1. In the threshold field, press 0 or 2 to cycle through the voltage options until you choose VAR for variable.
2. Move the cursor to the polarity field and use 0 and 2 to toggle the voltage polarity from positive to negative.
3. Move the cursor to the first digit of the voltage level and use 0 and 2 to cycle through the digits 0-9. When you're done, move to the second digit of the voltage level and use 0 and 2 to select that value.
4. Press ENTER to save your changes.

Figure 3-17 shows different threshold voltages for installed acquisition probes.

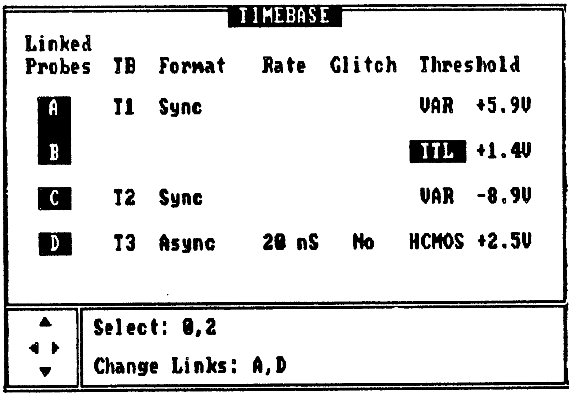
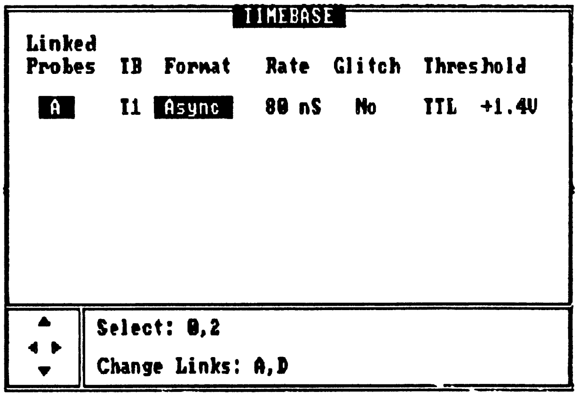


Figure 3-17. **Threshold voltages for acquisition probes.** Probes A and B are linked in timebase T1, but probe B can still use a threshold voltage different from probe A. Probes C and D also use different thresholds for triggering.

Sample Timebase Menus

The next five figures show sample Timebase menus and some corresponding channel groups. The Timebase menus show various probe links, timebases, and clocking formats.



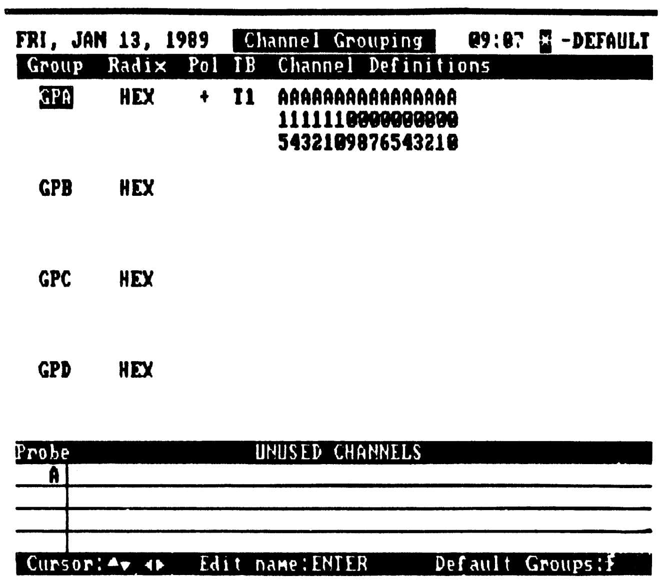


Figure 3-18. One probe, one timebase.

```
                        TIMEBASE
Linked
Probes   TB   Format    Rate    Glitch   Threshold

  A      T1   Sync                       TTL  +1.4V

  B      T2   Async     20 nS   No       TTL  +1.4V
```

```
WED, JAN 11, 1989   Channel Grouping    11:54   -DEFAULT
 Group  Radix  Pol  TB  Channel Definitions
 GPA    HEX    +    T1  AAAAAAAAAAAAAAAA
                        1111110000000000
                        5432109876543210

 GPB    HEX    +    T2  BBBBBBBB
                        00000000
                        76543210

 GPC    HEX


 GPD    HEX


Probe                 UNUSED CHANNELS
  A |
  B | 15 14 13 12 11 10 09 08
    |
    |
 Cursor:▲▼ ◄►     Edit name:ENTER      Default Groups:F
```

Figure 3-19. Two probes, two timebases. This figure shows two timebases, one for each probe. Because T2 acquires at a rate of 20 ns, the corresponding channel group has only eights bits. Channels B15 through B08 are dimmed in the unused channels list to show that they are unavailable at the current acquisition rate.

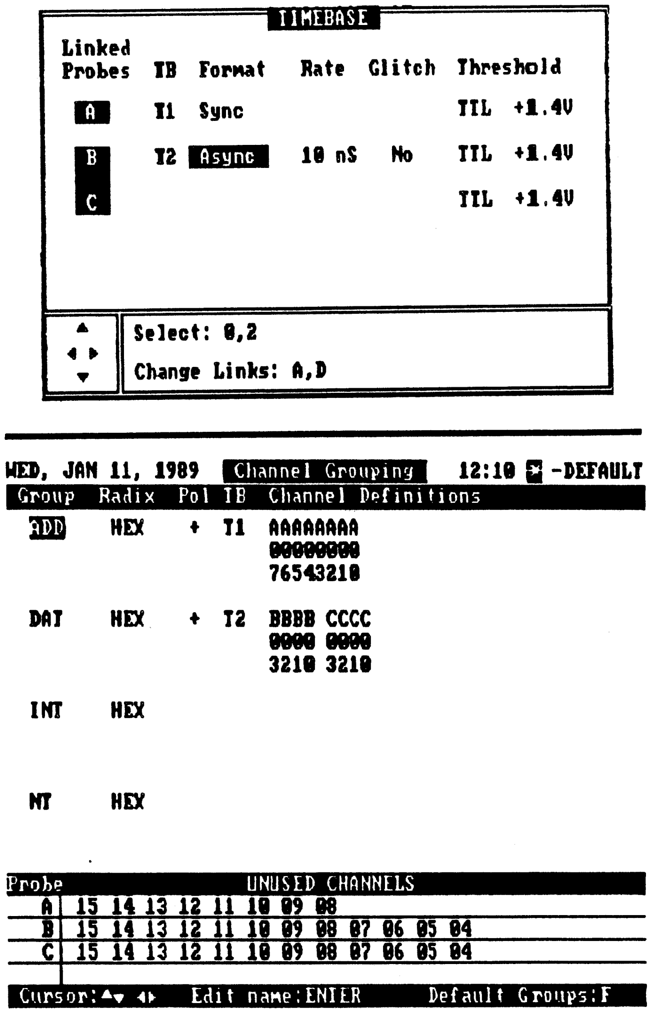
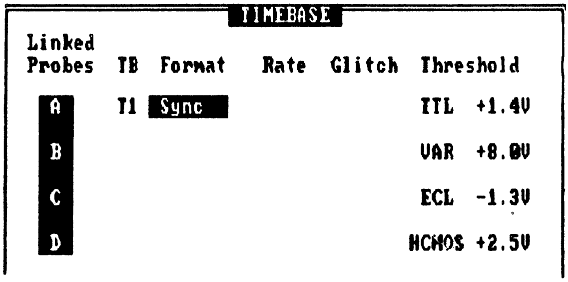


Figure 3-20. **Three probes, two timebases. Probe A acquires synchronously in T1, and probes B and C acquire data asynchronously in T2. Because probe B and C are linked without probe A, T2 must be asychronous.**



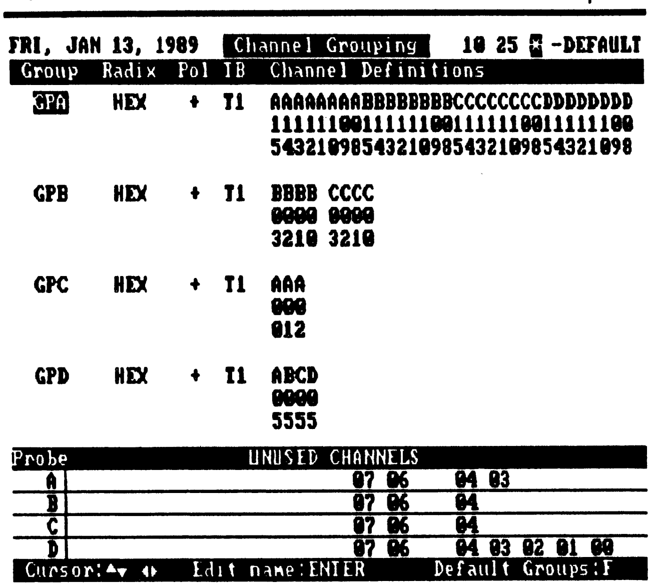
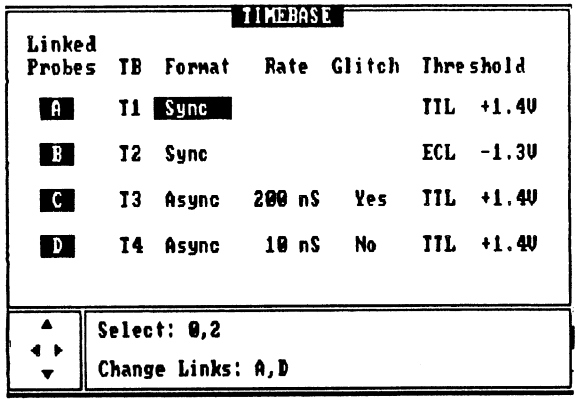


Figure 3-21. **Four probes, one timebase. All four probes are linked so that all 64 channels acquire data synchronously. Even though all four probes are linked, each probe can have a different voltage threshold. In this example, the Channel Grouping menu shows the highest eight bits of each probe in group A. (You can have a maximum of 32 channels in one group).**



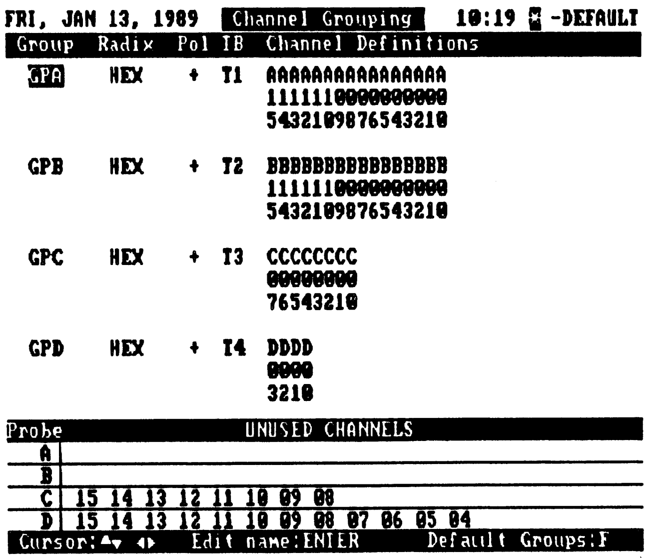


Figure 3-22. **Four probes, four timebases. In this example, each probe uses a separate timebase so that each probe acquires data separately from the other probes. Since you don't specify any linkages, you can set any of the timebases to be synchronous or asynchronous. However, you can specify only two different asynchronous rates. In this example, T1 and T2 are synchronous.**

CHANNEL GROUPING MENU

The Channel Grouping menu lets you organize input channels into groups. This lets you acquire and display data in a format that makes sense for your application. While the analyzer acquires data on all channels, you can choose to ignore channels that you don't need. For example, if you need only 8 channels but have 16 connected to your SUT, you could make a channel group consisting of only those 8 channels on which you want to acquire data.

This discussion is divided into these topics:

- Changing group names
- Selecting radixes for condition definitions
- Choosing the input polarity
- Grouping channels
- The unused channels list

You can specify up to six channel groups. The default names for the groups are GPA, GPB, ..., GPE. Each group can include up to 32 active channels (up to 64 channels for the 1230). Only channels in the same timebase can be in the grouped together.

Four channel groups are displayed on the screen at once. Use ▲ and ▼ to scroll through the visible groups and scroll the other groups onto the screen.

The analyzer always displays the unused channels list at the bottom of this menu. Channels that are dimmed are unavailable because of your sampling rate or glitch selection in the Timebase menu. Channels in normal video are available but not currently assigned to a group.

Figure 3-23 shows the default Channel Grouping menu.

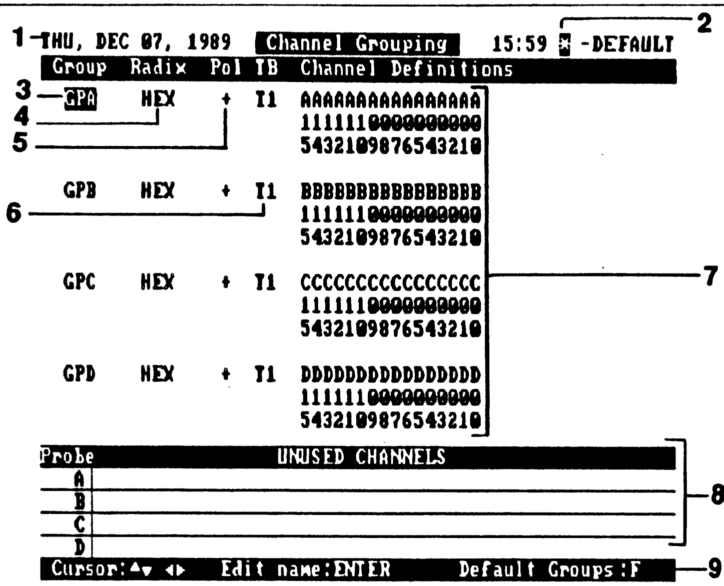


Figure 3-23. **Channel Grouping menu. All available channels are assigned to timebase T1. Each probe's 16 channels are assigned to a different channel group: GPA for probe A, GPB for probe B, and so on. The default display radix for each channel group is hexadecimal.**

[1] **Menu header.** This line gives the menu name, the current date, the current time, and the current setup name.

[2] **Asterisk.** If you made changes to the setup since you last stored it in the analyzer, the analyzer puts an asterisk before the setup name.

[3] **Group name.** The group name can be up to three characters long. Change a channel group name by moving the cursor to the name and pressing ENTER. You can then enter hexadecimal characters directly from the keypad and use ▲ and ▼ for other characters and symbols. Press ENTER again when you have finished making changes to save the new group name.

[4] **Radix field.** This field sets the input radix. The radix listed for the channel group defines the radix used in the Conditions menu to display (and define) the condition words for triggering. You can specify hexadecimal, octal, and binary radixes. Use the binary radixes if you want to define glitches for conditions. Press 0 or 2 to cycle through the radixes.

[5] **Polarity field.** You can choose positive true or negative true polarity. Press 0 or 2 to toggle polarity positive or negative.

[6] **Timebase field.** This field shows which timebase the probe's channels are in. Probe channels must be in the same timebase to be in the same channel group. When you put the first channel in a blank group, the analyzer automatically updates this field to reflect the channel's timebase. You can then add only channels also in that timebase. You cannot change this field.

[7] **Channel groups.** This field lists the channels from each probe that are in this channel group. You can add, delete, and overwrite channels. To insert a channel, move the cursor to the space in which you want to insert a channel, press 1, and then enter the channel number (for example, A07). Press E to enter the next (subsequent) channel if available. Press 0 to delete a channel. Enter a complete channel number directly to replace an existing channel. Press DON'T CARE to abort a change you're currently making.

[8] **Unused channel list.** Channels you delete from channel groups are displayed here. If a channel is dimmed, it is unavailable because of the sampling rate or glitch selection.

[9] **Menu bars.** Menus bars tell you what keys to press to use menu features. Menu bars for editing group names or changing channel groups appear when the cursor is on those fields.

Changing Group Names

Channel group names can be up to three characters long. Default names are GPA, GPB, ..., GPE, but you can change a group name to any three-character string you want. Follow these steps to edit a group name:

1. Move the cursor to the group name and press ENTER. The cursor now highlights the first character of the name. (You can use ◀ and ▶ to move from one character to the next.)
2. Enter hexadecimal characters directly from the keypad, use ▲ and ▼ to cycle through other characters and symbols, and press DON'T CARE to enter a blank space. As soon as you enter a character, the cursor automatically moves to the next digit. If you enter the third character, the cursor leaves the edit mode and moves on to the next menu field (radix).
3. If you didn't change the third character so that the cursor automatically leaves the edit mode and moves on to the next field, press ENTER to leave edit mode and save your new name. You can also leave edit mode by using the cursor keys to scroll to another field.

Changing channel group names affects the Conditions, State, and Timing menus. Condition-word group names and state-table display group names reflect any changes you make in the Channel Grouping menu. In the timing diagram, the channel group name affects the trace names and any reordering of traces you've done since you last changed the group name.

In the timing diagram, the group name is reflected in trace names that haven't been manually defined. For example, Figure 3-24 shows a group of 8 traces with 4 traces named by default after the channel group DAT, and 4 traces manually renamed to reflect SUT lines. If you change the channel group name from DAT to ADD, the 4 traces named by default after the channel group now reflect the new group name. The names of the 4 traces manually renamed do not change (see Figure 3-25).

A change in the channel group name also affects the order of the traces. If you reordering the traces in the timing diagram so that they no longer reflect the order of channels in the Channel Grouping menu, changing the channel group name automatically changes the order of the traces back to that shown in the Channel Grouping menu. Figure 3-24 shows the change from the group name DAT to ADD. Figure 3-25 shows that the traces you reordered are now changed back to reflect the Channel Grouping menu.

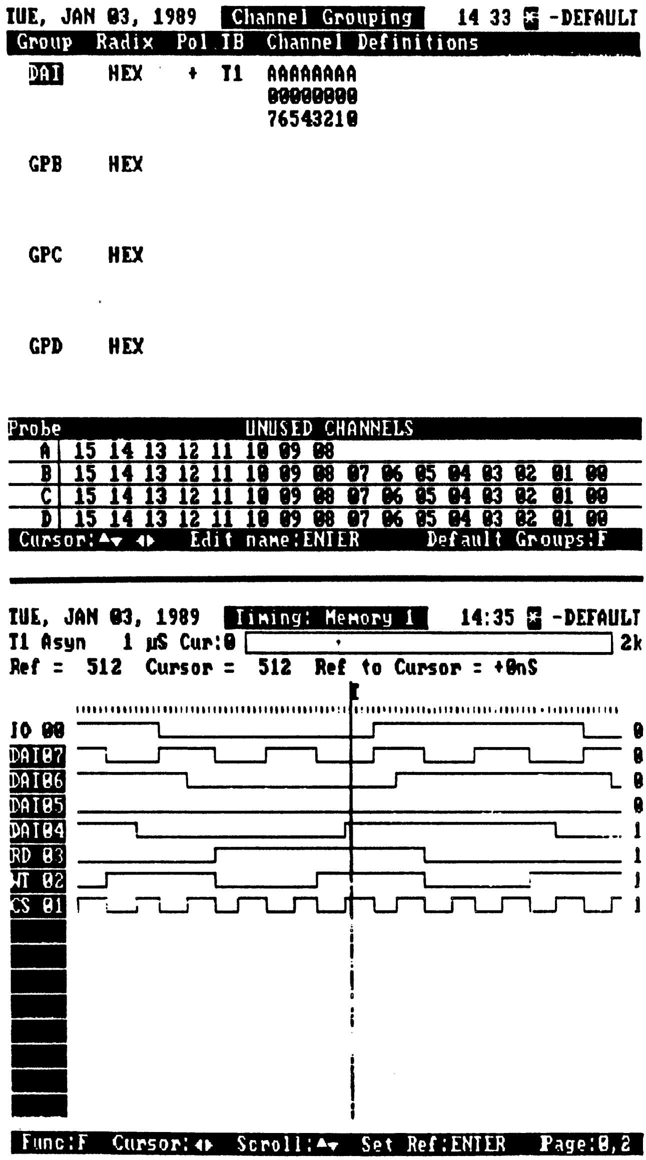


Figure 3-24. Change the channel group name. When you change the channel group name, the change affects other menus. In the timing diagram, this change affects not only the trace names you haven't manually defined, but also the order of traces you've moved around.

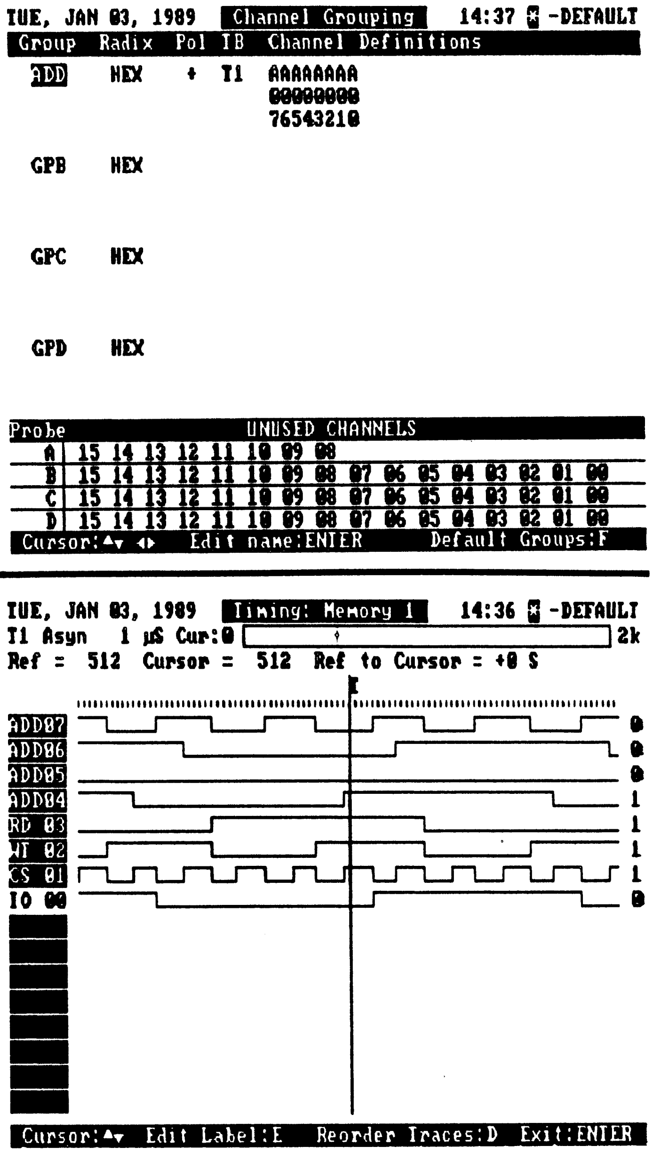


Figure 3-25. **Traces are renamed and reordered.** The change to the channel group you've manually renamed are not changed. However, the order of the traces in the timing diagram is also changed back to reflect the order of channels in the Channel Grouping menu. The highlighting on the trace names shows how the order was changed.

Selecting Radixes

The radix field sets the input radix for word definitions in the Conditions menu. You can set this field to hexadecimal (default), binary, or octal. If you set a group's radix to binary, you can then specify glitches as trigger events for that channel group in the Conditions menu.

To change the radix, move the cursor to this field and press 0 or 2 to cycle through the three selections. Figure 3-26 shows how radixes in channel groups affect conditions.

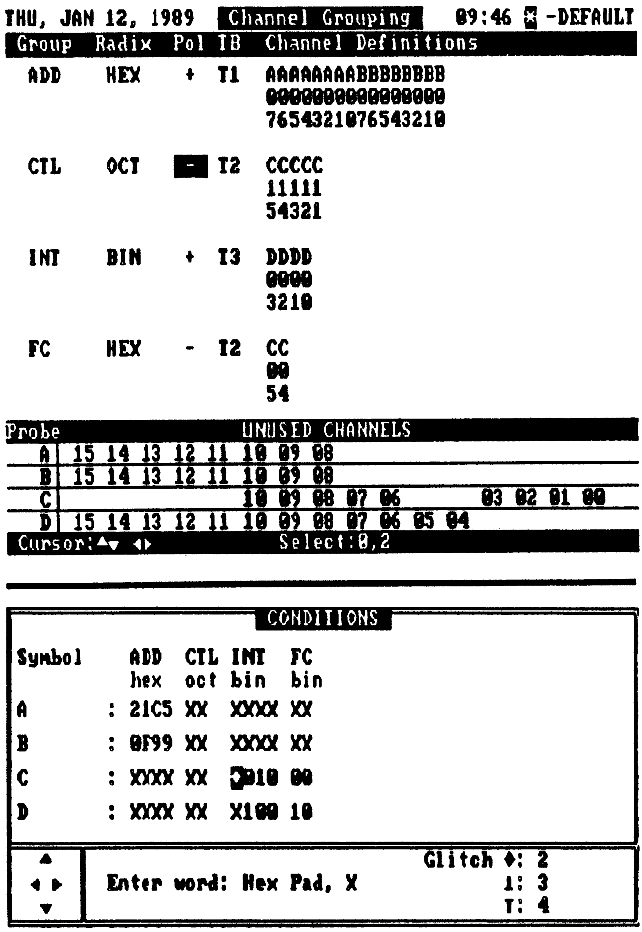


Figure 3-26. **Channel group radixes. Channel groups in different timebases can have different polarities. Regardless of timebases, each channel group can have different input radixes. The Conditions menu reflects channel group radix, you can include glitches in the condition definition (if glitches are turned on in the Timebase menu).**

Choosing the Input Polarity

This field lets you choose positive true or negative true polarity for each channel group. This lets you define whether you want the channels to recognize a rising or falling signal as True.

In the state table, when polarity is positive, a logic low is False and a logic high is True. Conversely, when polarity is negative, a logic low is True and a logic high is False. The input polarity does not affect the timing diagram, condition words, or the value of the search condition. However, the binary readout along the right side of the timing traces always matches the state table.

Figure 3-26 shows channel groups with different polarities.

Grouping Channels

The analyzer lets you add, delete, and change the channels assigned to channel groups. When you move to the channel group field, the cursor highlights the channel you can change. You can then use the cursor keys to move around in the fields and groups.

Grouping channels lets you do three things:

- Specify the input channels on which you want to see acquired data.
- Display acquired data in groups of specific channels in state table format.
- Define trigger conditions in different radixes.

You can group channels together with three restrictions (listed after Figure 3-27). If you try to use a channel that is invalid for that group or setup, the analyzer displays one of the messages shown in Figure 3-27. If this happens, press DON'T CARE to abort the change you tried to make. You can then enter the name of a valid channel.

```
                 Selected Channel is Not Available

ADD     HEX     +  T1   AAAAAAAABBBBBBBB
                        0000000000000000
                        7654321076543210

DAT     OCT     +  T1   CCCCCCCC
                        00000000
                        76543210

CTL     BIN     +  T1   DDDDDDDD
                        00000000
                        76543210

GPD     HEX

Probe                    UNUSED CHANNELS
    A | 15 14 13 12 11 10 09 08
    B | 15 14 13 12 11 10 09 08
    C | 15 14 13 12 11 10 09 08
    D | 15 14 13 12 11 10 09 08
              Select:A-D            Abort:X
```

Figure 3-27. **Invalid channel grouping. If you try to use a channel that is unavailable, the analyzer displays an error message. Press DON'T CARE to abort your change and then enter a valid channel for the group.**

The error messages mean:

1. The channel is already being used. In this case, you must delete the channel from the group it's being used in before you can add it to another group.
2. The channel is not in the same timebase as other channels in the group. Only channels in the same timebase (sampling rate and format) can be grouped together. Figure 3-29 shows this.

3. The channel is not available because of the sampling rate or glitch selection. For example, if you're acquiring data at 20 ns, the acquisition probe acquires on only the lower 8 bits (input channels 07-00). In this case you can't add channel 15 to any group, because the acquisition probe won't acquire on that channel. At a sampling rate of 10 ns, an acquisition probe acquires on only the lower 4 bits (input channels 03-00). When glitch capture is turned on, the probe acquires on only the lower 8 bits (input channels 07-00).

As soon as the cursor is in the channel definition field, you can enter a channel name directly from the keypad. You can also use these keys to insert, add, delete, and change channels in the groups.

1 Insert a channel between two others

0 Delete a channel

E Add the next consecutive channel if available

X Abort a change

Figures 3-28 and 3-29 show different channel groups.

```
WED, JAN 11, 1989   Channel Grouping     12 02 ▮ -DEFAULT
 Group  Radix   Pol  TB  Channel Definitions
  ADD    HEX     +   T1   A AAA A
                          0 110 0
                          0 549 2

  DAT    HEX     +   T1   AAAAAA
                          000000
                          345678

  STB    HEX     +   T1   A
                          1
                          3

  CTL    HEX     +   T1   AAA A
                          111 0
                          210 1
Probe                 UNUSED CHANNELS
    A|
    B| 15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00
    C| 15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00
    D| 15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00
 Cursor:▲▼ ◄► Select:A-D  Next Ch:E  Insert:1  Delete:0
```

Figure 3-28. One probe with channel groups. In this example, probe A's channels are split into different groups. Because all channels belong to probe A, all channel groups are in T1, the timebase for probe A.

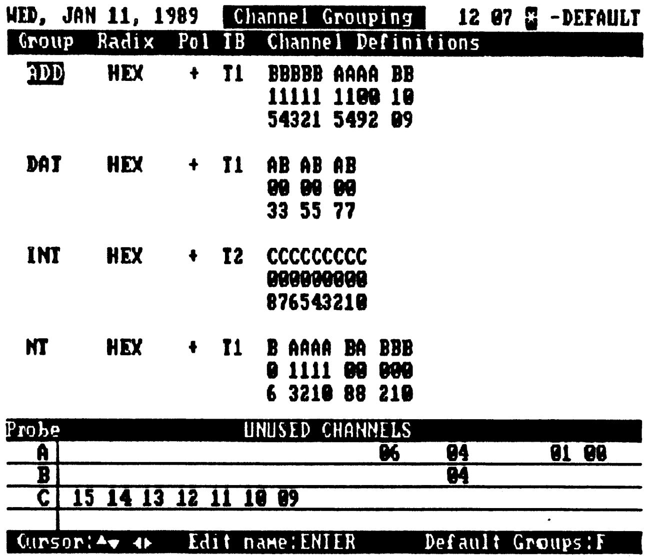


Figure 3-29. **Three probes with channel groups. In this example, probes A and B are in T1 and probe C is in T2. Because of this, the channels from probes A and B can be mixed, but probe C's channels cannot be mixed in a group with the other probes' channels.**

The next three discussions show you in detail how to add and delete channels, and how to reset the channel groups to the default settings.

Adding Channels

You can add input channels to groups if the new channels are in the same timebase as the other channels in the group. You can add channels in any order you want: one at a time, in a series of descending numbers, and between other channels. These steps tell you how to add channels to groups:

1. Move the cursor to the channel group to which you want to assign channel

2. Enter the full name of the channel you want to add, for example, A03 (not A3). The cursor automatically moves from one digit to the next. If you make a mistake, press DON'T CARE to abort the change, then try again.
3. To add a channel between two others, press 1 to insert, and then enter the full name of the channel you want to add. If you press 1 and move the cursor, the 1230 leaves a space at that location. The space will disappear the next time you call up the Channel Grouping menu.
4. To add the next consecutive channel (if available) automatically, press E. For example, if you added A15, then pressed E, the analyzer adds A14 (if available) to the channel group.

If you add spaces to a channel group, they will remain until you exit and call up the Channel Grouping menu again. When the channel group reappears, the analyzer automatically takes out extra spaces. However, the analyzer leaves the channels in the order you specify. For example, if you specify B07-B00, then A03, and then B15-B08, the analyzer leaves the channels in that order in the group.

Deleting Channels

You can delete channels to keep their data from being displayed. The analyzer still acquires data on those channels, however, the data is not shown in the state table or timing diagram after the acquisition.

Follow these steps to delete channels from channel groups:

1. Move the cursor to the channel you want to delete, for example, A15.
2. Press 0 to delete the unwanted channel from the group. The 1230 puts this channel in the unused list at the bottom of the screen. For example, A15 should now be listed in the unused channel list. Figures earlier in this section show unused channels lists.

When you delete a channel from a group, the condition words defined for that group also drop the channel. For example, if a 16-channel group (A15-A00) had a condition word FFFF and you dropped A15 from the group, the new condition word would be 7FFF. If you add the channel back to the group, the channel is added to the condition word as an X (don't care) value. For example, if you add A15 back to the group, the new condition word is X111 1111 1111 1111 in binary. This is ?FFF in hexadecimal, because the analyzer doesn't know what value you want for the new channel.

Resetting Groups to the Defaults

The analyzer lets you return to the default channel groups at any time by pressing F in the Channel Grouping menu. The default setting is all the channels from each available probe assigned in descending order to a single group. For example, if the analyzer has 32 channels, 16 channels (A15-A00) are in group GPA and 16 channels (B15-B00) are in group GPB.

When you press F to reset the groups to the defaults, the analyzer asks for confirmation with this message:

```
OK to set channel grouping to default?
Press ENTER to proceed or MENU to abort
```

If you press ENTER, the analyzer resets the channel groups to the defaults. Pressing MENU aborts the change and leaves you with the previous channel assignments.

Changes Affecting Other Menus

Changing channel groups affects the Conditions, State, and Timing menus. In the Conditions and State menus, the condition words and state-table displays always show the input channels in the order they are listed in the Channel Grouping menu.

For the Timing menu, when you add, delete, or reorder channels in channel groups, the timing diagram automatically changes to show the timing traces in the same order you list them in the

Channel Grouping menu. If you reordering channels in the Channel Grouping menu, changing the channel definitions automatically changes the order of the traces back to that shown in the Channel Grouping menu.

For example, Figures 3-30, 3-31, and 3-32 show how a change in channel definitions affects the order of the timing traces in the timing diagram.

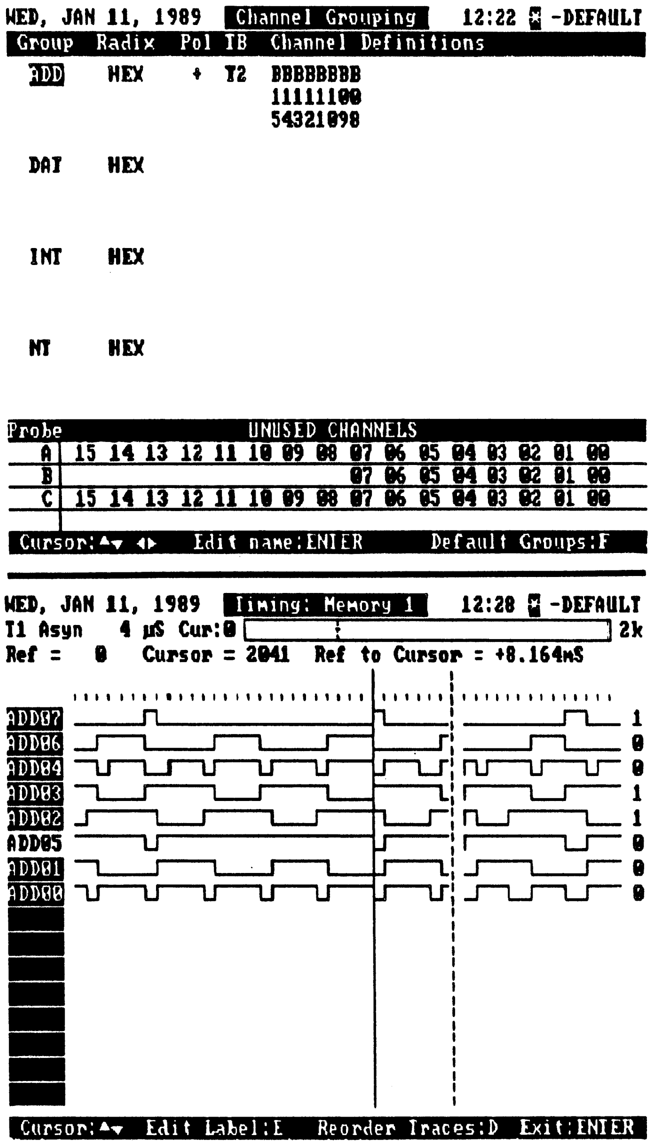


Figure 3-30. Change the order of traces. The channel group shows the default order of the timing traces. In the timing diagram, the traces are manually changed around to show data in a specific order.

```
WED, JAN 11, 1989   Channel Grouping      12 29 ▣ -DEFAULT
 Group   Radix   Pol  TB   Channel Definitions
  ADD     HEX     +   T1   BBBBBBBB
                           00111111
                           89012345

  DAT     HEX


  INT     HEX


  NT      HEX

Probe                    UNUSED CHANNELS
    A | 15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00
    B |                         07 06 05 04 03 02 01 00
    C | 15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00
      |
 Cursor:▲▼ ◄►     Edit name:ENTER        Default Groups:F
```

Figure 3-31. Change the channel definitions. The channels are now displayed in this order: B08-B15. The timing traces reflect this new order of input channels.

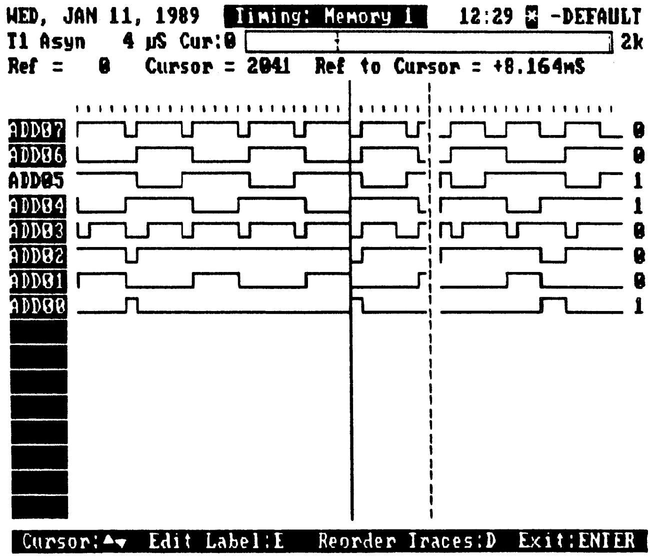


Figure 3-32. Traces are automatically reordered. The change to the channel definitions is reflected in the order of the traces in the timing diagram. The 8 traces are now displayed 00-07. You can still reorder traces in the timing diagram. However, each time you change the channel definitions, the analyzer updates the timing diagram to match the Channel Grouping menu.

Changes Caused by the Timebase Menu

Channel groups reflect changes in probe links, sampling rates, and glitch capture. When you change these Timebase menu features, the analyzer looks at each channel group to determine which channels are still valid for the grouping. For sampling rates and glitch capture, the analyzer drops invalid channels and places them in the unused list at the bottom of the screen.

When probe links change, the analyzer looks at the first channel in each channel group to determine the new timebase for that group. Channels that are in the new timebase remain in the group in the order you listed them. Channels that are no longer in that timebase are deleted and placed in the unused list at the bottom of the screen. You can then add the available, unused input channels to groups that have the same timebase. The discussion on the Timebase menu, earlier in this section, explains this in detail.

The Unused Channel List

The unused channel list displays any channels not currently in the menu's channel groups. Channels may be in the unused list because you deleted them, in which case they are available for use in other groups, or because the timebase rate or glitch selection makes the input channel unavailable.

If a channel is unavailable because of the rate or glitch selection, the analyzer dims that channel in the list. You cannot assign unavailable channels to channel groups. No matter what changes you make in other menus, the analyzer doesn't automatically add any channel back to a group once it's been deleted and placed in the unused list. You must add them back manually. Figures earlier in this section show unused channels lists at the bottom of Channel Grouping menus.

TRIGGER SPECIFICATION MENU

The Trigger Spec menu lets you define trigger statements and choose the trigger timebase used for acquiring data. This discussion covers these topics:

- Choosing the trigger timebase
- Using trigger statements

Figure 3-33 shows a Trigger Spec menu.

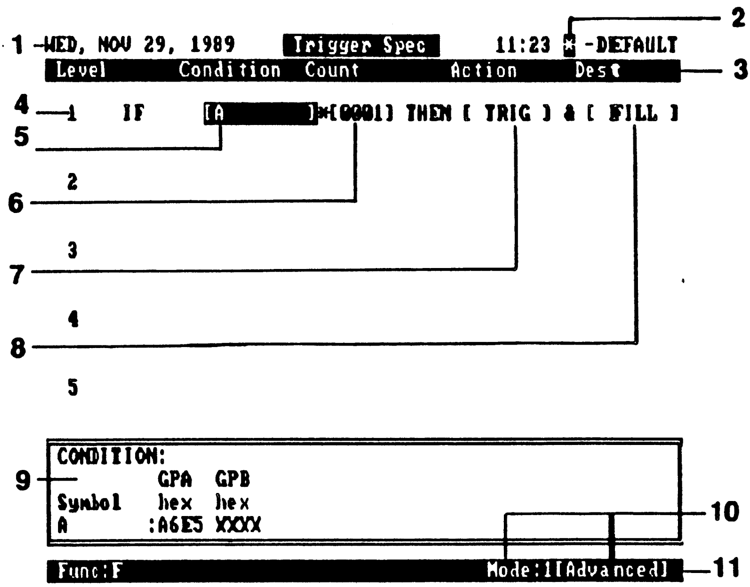


Figure 3-33. **Trigger Spec menu. The viewport at the bottom of the screen briefly describes the field highlighted by the cursor. In this figure, the cursor highlights the conditions field, so the viewport shows the condition's word value.**

[1] **Menu header.** This field lists the menu name, current date and time, and the current setup name.

[2] **Asterisk.** If you made changes to the setup since you stored it, the analyzer displays an asterisk before the setup name.

[3] **Statement header line.** This line lists a one-word header for each field: level, condition, count, action, and destination.

[4] **Level number.** The Trigger Spec menu provides 14 trigger levels. Add a statement to a blank level by moving the cursor to a blank level and pressing ENTER. If you're in advanced trigger mode (press 1 to toggle between modes), press ENTER to cycle through the available statements.

[5] **Condition.** Use this field to specify the condition you want the analyzer to recognize. Press 0 or 2 to cycle through available conditions. When you're on this field, the viewport displays the current condition's word value.

[6] **Count.** This field lists the number of times the condition must go through the transition from False to True in order for the action to be taken. You can select a count number in the range from 1 to 4096.

[7] **Action.** Use this field to tell the analyzer to trigger, turn data storage on or off, or do nothing. You can select TRIG, TRIG O, TRGOUT, STR ON, STROFF, or NOP. The viewport explains each action. When you move the cursor to the action field, the viewport describes the currently selected action. Press 0 or 2 to cycle through the actions.

[8] **Destination.** Use this field to tell the analyzer to fill memory, continue reading through the statement levels, or go to a specific level and continue from there. You can choose FILL, CONTIN, or GOTO# (where # is a number from 1-14). The viewport discussion explains each destination. When you move the cursor to the destination field, the viewport describes the currently selected destination. Press 0 or 2 to cycle through available destinations.

[9] **Viewport.** The viewport gives you a description of the field the cursor is currently on. You can turn the viewport off and on by pressing `DON'T CARE`.

[10] **Trigger timebase.** If you're using more than one timebase, the main menu bar lists the trigger timebase field (TrigTB). You can press D to change the trigger timebase from one timebase to another. The brackets surround the current timebase. Regardless of what probe is in the trigger timebase, the analyzer automatically cross-triggers all other installed probes to acquire when the trigger occurs.

[11] **Menu bars.** These bars give you brief instructions about command keys and menu fields.

Choosing the Trigger Timebase

The analyzer looks for the trigger condition on the timebase you specify, then automatically cross-triggers the other probes (in the other timebases) when the trigger condition is found. You specify the trigger timebase with a field in the menu bar at the bottom of the Trigger Spec menu: TrigTB: D[T1]. To select a different timebase on which to trigger, press D. The brackets enclose the new timebase.

If you're using only 16 channels, you only have one acquisition probe, so you only have one timebase to acquire data in. In this case, the menu bar doesn't display the trigger timebase field, and the D key is disabled.

When you select a timebase in which to look for the trigger event, the viewport at the bottom of the screen shows the valid condition word for that timebase. For example, Figure 3-34 shows a viewport with two channel groups for the defined condition word, one in each timebase. The condition word for the channel group in the trigger timebase is displayed in normal video. The condition word for the channel group in the other timebase is dimmed in the display. This shows you the specific value on which the analyzer triggers.

```
WED, JAN 11, 1989   Channel Grouping    12 34   80386
Group  Radix  Pol TB  Channel Definitions

 ADD    HEX    +  T1  AAAAAAAAAAAAAAAA
                      1111110000000000
                      5432109876543210

 DAT    HEX    +  T2  BBBBBBBBBBBBBBBB
                      1111110000000000
                      5432109876543210

 INT    BIN

 NT     BIN

Probe                 UNUSED CHANNELS
   A|
   B|
   C| 15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00
   D| 15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00
Cursor:▲▼ ◄►    Edit name:ENTER       Default Groups:F
```

```
WED, NOV 29, 1989     Trigger Spec        11:25   -DEFAULT
Level        Condition  Count          Action        Dest

 1   IF     [A        ]*[0001] THEN [ TRIG ] & [ FILL ]

 2

 3

 4

 5

CONDITION:
         GPA  GPB
Symbol   hex  hex
A       :21C5 00FF

Func:F   Cursor:▲▼◄►   Select:0,2   Instruction:ENTER
```

Figure 3-34. Conditions reflecting the trigger timebase. The Channel Grouping menu shows that there are two timebases and two channel groups. The Trigger Spec menu shows that timebase T1 is the trigger timebase. When the cursor is on the trigger condition field, the viewport shows the trigger condition word values: 21C5 and 00FF. Since group ADD is in timebase T1, the analyzer triggers when it acquires the value 21C5. Timebase T2 (acquisition probe B) is automatically cross-triggered when the value 21C5 occurs in T1.

Using Trigger Statements

You can use up to 14 statements to specify when and how to trigger and store data. Each trigger statement has four parts:

1. A data condition or event for the analyzer to recognize
2. The number of times the event must occur
3. An action to be performed when the event is recognized action
4. A destination that tells the analyzer what to do after it finishes the action.

Read each statement like this: If the [condition] occurs [count] times, then perform the [action] and go to the [destination].

In the basic triggering mode, if-then statements define your trigger specifications. In advanced triggering mode, you can use more complex, if-then-else statements to define the way the analyzer triggers:

if-then

if-then-else-if-then

if-then-else

if-then-else-store-only

To toggle between triggering modes, press 1. The menu bar at the bottom of the screen tells you which mode (Basic or Advanced) is currently selected. Figures 3-35 through 3-38 show the four types of triggering statements.

```
WED, JAN 11, 1989     Trigger Spec      12:37   EXT_TEST
Level        Condition  Count          Action       Dest

 1    IF      [SUBA     ]*[0001] THEN [ TRIG ] & [ FILL ]


 2


 3
```

Figure 3-35. **If-then statement.** In this example, the condition SUB A is the subroutine A occurs, trigger and fill memory.

```
WED, JAN 11, 1989     Trigger Spec      12:37   EXT_TEST
Level        Condition  Count          Action       Dest

 1    IF      [SUBA     ]*[0001] THEN [TRGOUT] & [GOTO 1]
 ELSE IF      [SUBB     ]        THEN [ TRIG ] & [ FILL ]

 2


 3
```

Figure 3-36. **If-then-else-if-then statement.** One condition is the beginning of subroutine SUB A and the other condition is the beginning of subroutine SUB B. You want to make sure SUB B occurs sometime within the first 10 occurrences of SUB A. Read this statement like this: If subroutine SUB A occurs 10 times, send a trigger-out signal to the back panel connector and take the acquisition again. Otherwise, if SUB B occurs before SUB A occurs 10 times, trigger and fill memory.

```
WED, JAN 11, 1989     Trigger Spec      12:41   EXT_TEST
Level        Condition  Count          Action       Dest

 1    IF      [SUBA     ]*[0001] THEN [ TRIG ] & [ FILL ]
 ELSE                                 [ NOP  ] & [GOTO 2]

 2


 3
```

Figure 3-37. **If-then-else statement.** In this example, if the subroutine SUB A doesn't occur, you want the analyzer to go to level 2 and execute that statement. Read the statement like this: If SUB A occurs, trigger and fill memory. Otherwise, go to trigger statement 2.

```
WED, JAN 11, 1989     Trigger Spec       12:42   EXT_TEST
Level        Condition  Count           Action     Dest

1    IF      [SUBEND2 ]*[0068] THEN [ TRIG ] & [ FILL ]
ELSE STORE ONLY      [NMI      ]

2


3
```

Figure 3-38. **If-then-else-store-only statement. In this example, the condition SUBEND2 is the end of a subroutine. The second condition, NMI, is a non-maskable interrupt. Read the statement like this: If SUBEND2 occurs 68 times, trigger and fill memory. Otherwise, story only data values that match condition NMI.**

Entering Trigger Statements

The Trigger Spec menu has up to 14 levels you can use to control the trigger. To enter a trigger statement at a blank level, move the cursor to a blank level and press ENTER.

In basic trigger mode, pressing ENTER toggles between an if-then statement and a blank level. In advanced trigger mode, pressing ENTER cycles through the four available statements for advanced triggering along with a blank level.

Make sure the cursor is not on the count field when you try to cycle through trigger statements. Pressing ENTER while on the count field puts you in an edit mode for the count field. If this happens, press ENTER to leave the count field's edit mode, move the cursor to some other field, and press ENTER again to cycle through statements.

Deleting Trigger Statements

You can delete levels between two existing levels. To delete a trigger statement, move the cursor to the statement level and press ENTER until you display a blank line.

If you want to jump a blank level to the next one, you should use a goto instruction in the destination field of the first level's statement. If you don't use a goto, the analyzer continues to the new blank level. Since the level is blank, the analyzer assumes you meant to trigger and fill memory, which it will do for you, until either memory is full or you press STOP.

Choosing the Trigger Condition

Conditions are defined in the Conditions menu and specified for triggering in the Trigger Spec menu. The Trigger Spec menu lets you specify which of the conditions you want the analyzer to look for on each trigger level. Press 0 (or 2) when on the conditions field to cycle through available conditions in this order:

1. 24 conditions (named by default A-W and MASK) from the Conditions menu and the NO of each condition. You can define and rename these conditions. For example, A might be SUB1 END, condition B might be RESET, and so on.

   NOTE

   If a condition is all don't cares and you use the negated condition, the analyzer will never find the condition because it will never occur. For example, if A is defined as all don't cares and you tell the analyzer to trigger if NOT A occurs, the analyzer will never trigger and you will have to manually halt the acquisition by pressing STOP.

2. External. The signal received from the EXT inputs on the acquisition probe. These inputs are edge sensitive, rather than level sensitive as are the other inputs. Use these inputs to look for a specific transition on a single line. You cannot negate this condition.
3. Trigin. The signal received from the trigger input on the analyzer's rear panel. You cannot negate this condition.

4. Clock. Clock pulses. Use this selection to count clock cycles when you are looking for a condition to occur a specific number of cycles after another condition. If you selected asynchronous as the format in the Timebase menu, the clock pulses are those from the analyzer's internal acquisition clock. If you selected synchronous, the clock pulses are signals acquired from the SUT through the CLK inputs on the acquisition probe. You cannot negate this condition.

When you move the cursor to the conditions field, the analyzer displays a viewport at the bottom of the screen. The viewport shows the value of the condition from the conditions menu. For External, Trigin, or Clock, the viewport contains descriptive messages about those conditions.

Entering a Count Value

This field lists the number of times the condition must go through a transition from False to True in order for the action to be taken. Follow these steps to edit the count field:

1. Move the cursor to the count field and press ENTER to edit the field.
2. Enter a count in the range 0001-4096 directly from the keypad. If you try to enter a count larger than 4096, the analyzer defaults to 4096. The cursor automatically moves to the next digit, but you can also use v and w to move it to different digits.
3. Press ENTER to quit edit mode. You can also move the cursor on to the next field (action) to leave edit mode.

When the count field uses a condition with a specific data value, the analyzer counts False-to-True transitions before taking an action. When the condition is all don't cares, the analyzer counts clock cycles. For example, if A=XXXX, the following two trigger statements are the same.

If [A] * [0001] Then [Trig] & [Fill]

If [Clock] * [0001] Then [Trig] & [Fill]

Figure 3-39 shows an example of counted transitions.

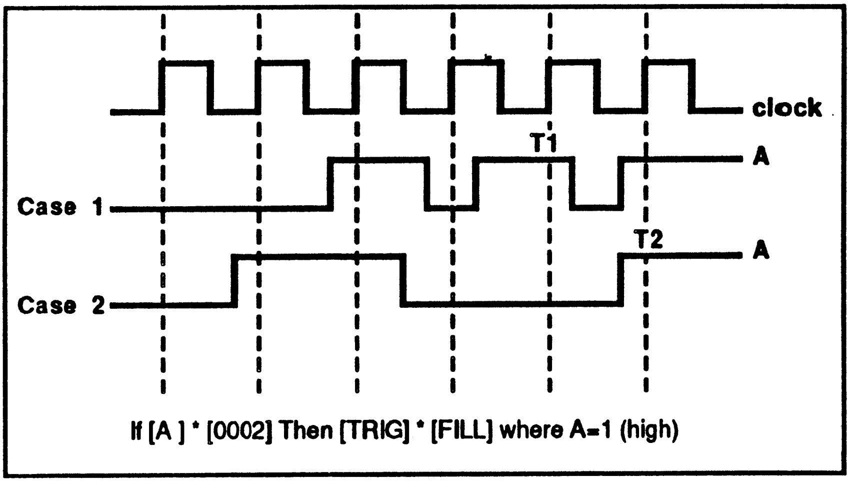


Figure 3-39. Counting Transitions. In case 1, the analyzer will trigger at the mark (T1) because A has gone through two transitions from False to True. In case 2, the analyzer will trigger at the later mark (T2) after A has gone through two transitions of False to True. In case 2, the analyzer does not trigger on a two-cycle-wide pulse because A has not gone through two transitions in that period.

Selecting an Action

The action field allows you to select an action for the analyzer to perform when the trigger condition is recognized. You can select these actions:

Table 3-2
TRIGGER SPEC ACTIONS

Action	Description
TRIG	Trigger and start storing post-trigger data. TRIG triggers all channels on all timebases.
TRIG O	Trigger and send a trigger-out signal to the back-panel connector.
TRIGOUT	Don't trigger, but send a signal to the trigger-out connector on the back panel when the trigger condition is recognized.
STR ON	Start storing data in memory.
STROFF	Stop storing data. Use STR ON, TRIG, or TRIG O to start storing data again.
NOP	No action; do nothing.

Choosing a Destination

The destination tells the analyzer what to do after it executes the trigger statement it's currently working on. You can use these destinations:

Table 3-3
TRIGGER SPEC DESTINATIONS

Destination	Description
FILL	Fill the rest of acquisition memory. Use FILL with TRIG or TRIG 0. Once acquisition memory is filled, the analyzer halts.
CONTIN	Continue to the next trigger level. If you tell the analyzer to continue to a level that has no instruction, the analyzer will fill acquisition memory until either you press STOP or memory gets full.
GOTO#	Go to the indicated trigger level. If you tell the analyzer to go to a level that has no instruction, the analyzer will fill memory and halt.

Using the Viewport

The viewport at the bottom of the screen lists a description of the field the cursor currently highlights. For example, if the cursor highlights the conditions field, the viewport lists the word definitions for that condition. Figures 3-33 and 3-34, earlier in this section, show viewports.

CONDITIONS MENU

The Conditions menu lets you define up to 24 conditions (A-X) (or A-W and MASK, in version 4.0 only) so that you can:

- Trigger on specific values
- Search for specific events in the state table
- Use comparison masks when you compare memories (see Run Control menu later in this section).

Figure 3-40 shows a Conditions menu and explains the menu's fields. The discussion that follows Figure 3-40 covers these topics:

- Displaying condition words
- Editing condition names
- Specifying glitches
- Changing radixes for glitches

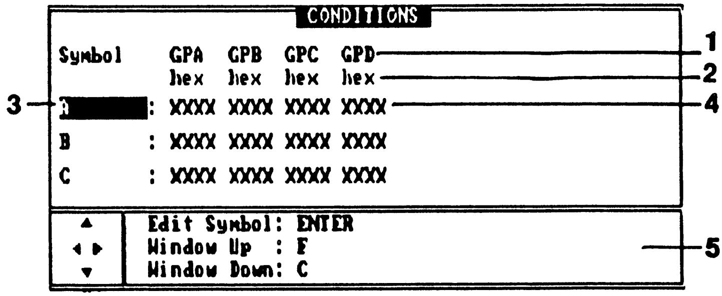


Figure 3-40. Conditions menu. The default Conditions menu shows three conditions. Default conditions are A-X (A-W and MASK in version 4.0). Default word definitions are all don't cares. Use F and C to expand or shrink the conditions window. The radix for condition words is set in the Channel Grouping menu.

[1] **Channel group names.** The header column displays the names of the groups listed in the Channel Grouping menu. Names are displayed from left to right as they were defined in the Channel Grouping menu from top to bottom.

[2] **Radix.** This field shows the radixes selected in the Channel Grouping menu. You can't select this field, but when you change channel group radixes, the Conditions menu radixes are automatically updated. Changes to display radixes in the State menu don't affect the Conditions menu because those changes are for state table display only.

[3] **Condition names.** This field lists the 24 conditions you can define. The default names are A-X (A-W and MASK in version 4.0 only). Press ENTER while the cursor is on the symbol name to edit the condition name. You can rename a condition to any eight-character name you want, but you should avoid naming two conditions the same thing. You cannot rename a condition to all blanks spaces.

[4] **Condition words.** This field shows the condition word value. The default word definitions are all don't cares. When the cursor is in a condition word definition, you can edit the definition directly from the keypad with values corresponding to the current group radix. For example, for a binary radix, you can enter 0, 1, X (don't care), and glitch symbols in the condition definition. Alternately, you can load the condition values from the cursor location in the state table by placing the cursor anywhere on the condition word and pressing ENTER.

[5] **Menu bar.** This field tells you which keys to press to use menu functions. When you scroll into the condition word, the menu bar automatically changes to tell you how to edit condition words.

Displaying Condition Words

The default conditions window shows three conditions: A, B, and C. You can re-size the window with the C and F keys to show one to eight conditions. You can also use the ▲ and ▼ keys to scroll other conditions through the conditions window.

You can display all six channel-group condition words on the screen if each channel group has at least one channel in it. However, displaying condition words in binary radix takes up a lot of room. If the displayed groups can't all fit because of the selected radixes, you cannot scroll more information onto the screen. Instead, call up the Channel Grouping menu and change group radixes until you see the information you want.

Figures 3-41 and 3-42 show condition words displayed in different radixes.

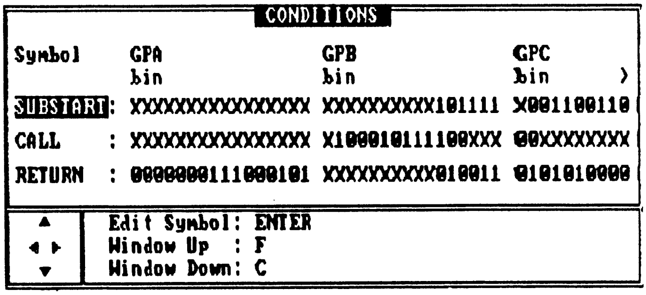


Figure 3-41. Conditions menu with binary radixes. All condition words are in binary radix and can't fit on the screen. There is a > at the right side of the screen to tell you that there is more information to be shown.

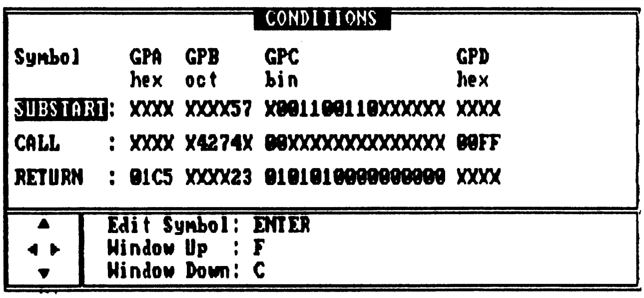


Figure 3-42. Conditions menu with different radixes. Only one group is now in binary radix. The other word definitions are now in their corresponding hexadecimal and octal representations. All four groups of condition word definitions can now fit on the screen.

The Conditions menu also tracks the condition selected as the memory comparison mask in the Run Control menu and the last condition selected in the Trigger Spec menu. For example, if the trigger condition is K and you call up the Conditions menu, condition K is displayed at the top of the conditions window. (Scrolling through the Conditions menu has no effect on the Trigger Spec menu.)

Editing Condition Names

Condition names (or symbols) can be up to eight characters long. The default condition names are A-X (A-W and MASK in version 4.0 only). To change a condition name, follow these steps:

1. Move the cursor to the condition name and press ENTER.
2. Enter 0-9 and A-F directly from the keypad. Use the cursor keys ▲ and ▼ to enter other characters. Press DON'T CARE to enter a space. When you enter a character, the cursor automatically moves to the next digit, but you can also use ◀ and ▶ to move the cursor from one character to the next.
3. Press ENTER when you finish entering the new name.

NOTE

Make sure you give each condition a unique name. If you accidentally give two conditions the same name, you won't be able to tell which of them you have actually selected in the Trigger Spec and Run Control menus unless you look at the condition definition in the viewport. Only the first definition in the list is used when two with like names exist.

Editing Conditions

To enter a value for a condition word, scroll into the word field. As soon as the cursor is on the word, you can enter alphanumeric characters directly from the keypad. Alternately, you can load the condition values from the cursor location in the state table by placing the cursor anywhere on the condition word and pressing ENTER. If glitch capture is set up and the word is in the binary display radix, you can also define glitches in the condition word.

Specifying Glitches

A glitch is two or more transitions through the logic threshold voltage (from low to high or high to low) within one clock cycle. Glitches are frequently symptoms of a circuit problem. Follow these steps to set up the analyzer to capture glitches:

1. Turn glitch capture on in the Timebase menu.

2. Set the appropriate channel group radix to binary in the Channel Grouping menu.
3. Specify glitches as part of a condition in the Conditions menu. Press 2 to define a generic glitch, 3 to define a rising-edge glitch, and 4 to define a falling-edge glitch. Table 3-4, later in this section, describes these glitches.

Figure 3-43 shows glitch symbols in condition words.

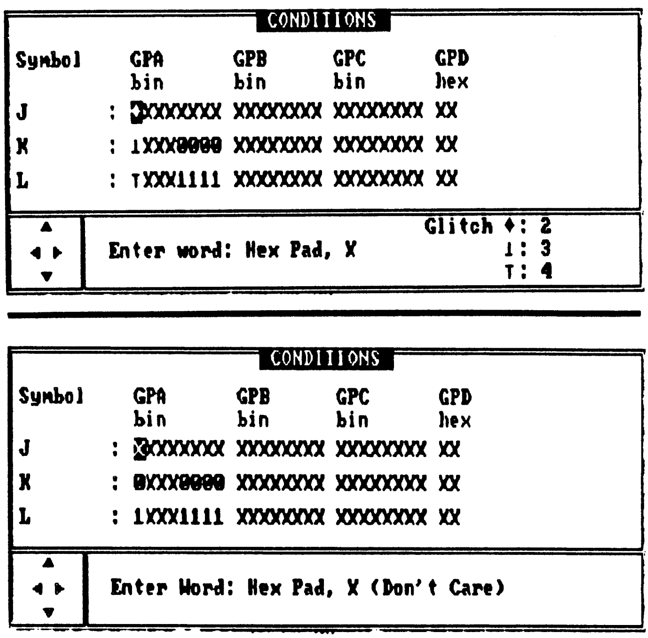


Figure 3-43. **Glitches in word definitions. For channel group GPA, the top Conditions menu shows a generic glitch on the leftmost channel for condition J. Condition K shows a rising-edge glitch, and condition L shows a falling-edge glitch. The lower Conditions menu shows how these glitch symbols revert to Xs, 0s, and 1s when glitch capture is turned off in the Timebase menu.**

Table 3-4
TYPES OF GLITCHES

Press	Type of Glitch	Reverts To	Description
2	◆	X	Generic glitch, which is any occurrence of one or more transitions in any direction within one clock cycle.
3	_|‾	0	Rising-edge glitch. Indicates one or more transitions within one clock cycle starting from a low state and rising to a high state.
4	‾|_	1	Falling-edge glitch. Indicates one or more transitions within one clock cycle starting from a high state and falling to a low state.

Changing Radixes for Glitches.

You can define glitches for condition words that are in binary radixes. If you have glitches defined for condition words and then change the channel group radixes to octal or hexadecimal, the glitch definitions revert to Xs, 0s, and 1s, as described in Table 3-4. If you change radixes to octal or hexadecimal and then change back to binary, the previous glitch definition is restored. Figure 3-44 shows how glitch definitions can revert to values when you change channel group radixes.

Remember that generic glitches (represented by a ◆) are really don't-care values to the analyzer. If you include a generic glitch in a condition word and then change the group's radix to octal or hexadecimal, the analyzer might display a question mark (?) in the condition word. A question mark means that some of the bits in the nibble are Xs (don't care values) and some are numerical digits. Because some values are Xs and some are numbers, the analyzer doesn't know what value you want defined for the octal or hexadecimal condition word. The example under *How Channel Groups Affect Conditions* shows how the analyzer can display a question mark.

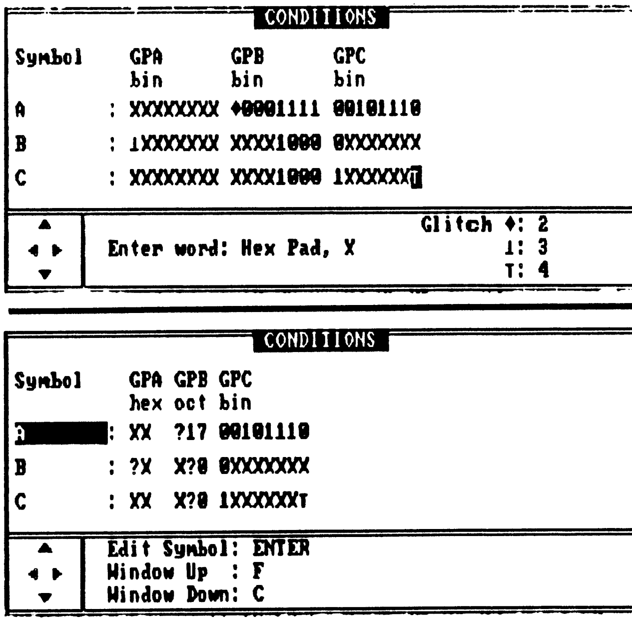


Figure 3-44. **Radix changes affecting glitch definitions. The top figure shows three conditions, with glitches in the word definitions. The bottom menu shows the corresponding hexadecimal and octal words for the conditions when the radix is changed. The generic glitch reverted to a don't-care value. Since other bits in the nibble were numerical values instead of other don't-care values, the analyzer displays a question mark for the nibble.**

Turning Glitches On and Off in Timebase

If you turn off glitch detection in the Timebase menu after defining glitches in condition words, the glitch symbols revert to X, 0, or 1, as described in Table 3-4. If you then turn glitches on again in the Timebase menu, the previous glitch definition is restored.

How Channel Groups Affect Conditions

Changing the order of channels in the Channel Grouping menu affects condition word definitions. When you add channels, the analyzer adds don't-care bits (Xs) to the condition words. When you delete channels, the analyzer deletes bits from the condition words. If you add deleted channels back to a channel group, the word definition for those channels is don't-care values. The analyzer displays a question mark in the condition word if don't-care values and numerical values are in the same nibble (if the nibble isn't all don't-care values or all numerical values).

Figures 3-45 and 3-46 show how conditions reflect changes in channel groups.

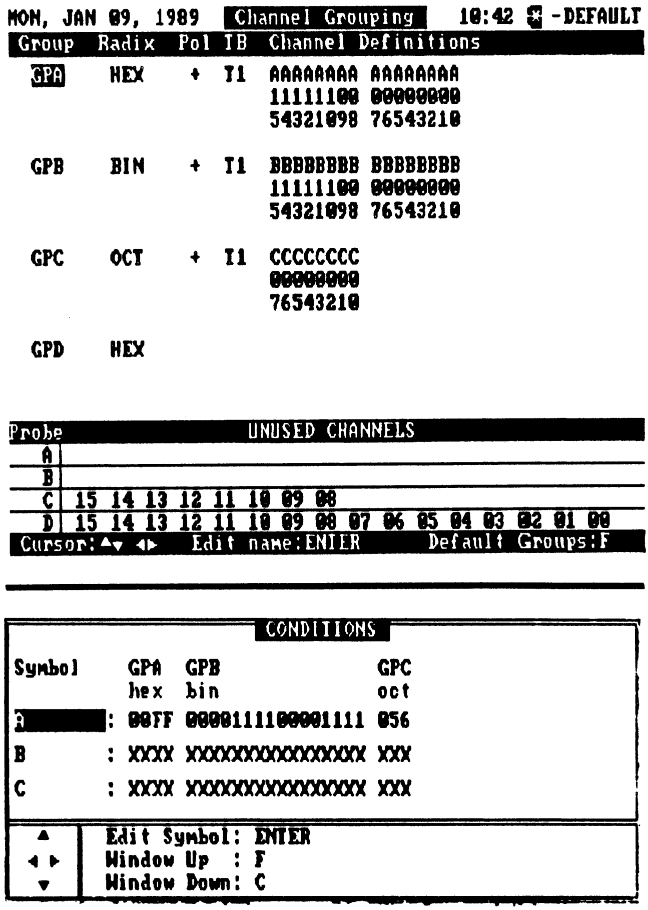


Figure 3-45. **Channel groups and conditions. The Conditions menu shows three condition words (one for each channel group) in the radixes selected in the Channel Grouping menu.**

```
MON, JAN 09, 1989  Channel Grouping    10:45 ▯ -DEFAULT
Group  Radix  Pol  TB  Channel Definitions
GPA    HEX    +    T1  AAAAAAAA
                       11111100
                       54321098

GPB    BIN    +    T1  BBBBBBBB BBBBBBBB
                       00111111 00000000
                       89012345 76543210

GPC    OCT    +    T1  CCCCCCCC CCCCCCCC
                       11111100 00000000
                       54321098 76543210

GPD    HEX

Probe                UNUSED CHANNELS
  A |                         07 06 05 04 03 02 01 00
  B |
  C |
  D | 15 14 13 12 11 10 09 08 07 06 05 04 03 02 01 00
Cursor:▲▼ ◄►    Edit name:ENTER      Default Groups:F
```

```
                  CONDITIONS
Symbol      GPA GPB                  GPC
            hex bin                  oct
A         : 00  XXXXXXXX00001111     XXX?56
B         : XX  XXXXXXXXXXXXXXXX     XXXXXX
C         : XX  XXXXXXXXXXXXXXXX     XXXXXX

  ▲    | Edit Symbol: ENTER
 ◄ ►   | Window Up  : F
  ▼    | Window Down: C
```

Figure 3-46. **Changed groups and corresponding conditions. Eight channels are deleted from GPA, eight channels are added to GPC, and eight channels are reordered in GPB. The corresponding condition words show that the analyzer deleted the digits in the condition word corresponding to GPA's channels. The analyzer then replaced the eight digits in group GPB's binary condition word with Xs for the eight channels in GPB that were deleted and later added back. The analyzer also added Xs for the new digits in group GPC's condition word. The question mark means that the nibble represented by the ? has both don't-care and numerical values in it.**

If you delete channels, the value of a condition word might change. For example, group GPA has eight channels: A07-A00. The condition word has the value 0B in hexadecimal, which is

0000 1011 in binary. If you drop channel A00 from the group (the rightmost binary digit), the binary value becomes 0000 101, which is 05 in hexadecimal. Now if you add A00 back to the group, the binary value is 0000 010X. Since don't-care and numerical values are mixed in the condition word's second nibble, the analyzer displays the new hexadecimal word value as 0?.

Always change channel groups before you define conditions. If you change the groups after you define the conditions, you may have to redefine your conditions to match the new order of the input channels.

RUN CONTROL MENU

The Run Control menu is divided into two parts by a double horizontal line (see Figure 3-47).

The upper part of the menu lets you specify which of the four memories the acquisition will be stored in, how you want data displayed after the acquisition, where in memory the trigger should be positioned, and when during the acquisition the actual trigger search should begin. The update memory you specify here controls the parameters in the Memory Select menu discussed in Section 4.

The lower portion of the menu lets you control memory comparison features. This part also lets you set the display screen update rate for continuous or auto-run acquisitions. The memory comparison feature is described at the end of this discussion. Figure 3-47 shows the Run Control menu.

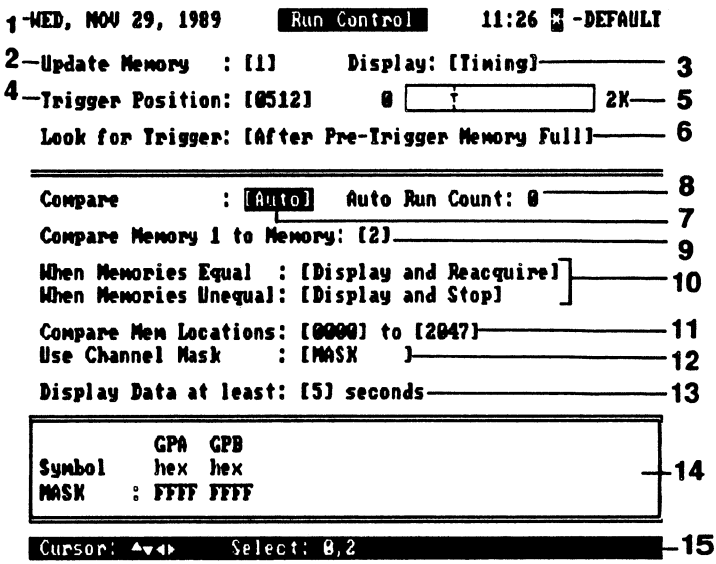


Figure 3-47. Run Control menu.

[1] **Menu header.** This line lists the menu name, current date and time, and current setup name. If there is an asterisk before the setup name, you made changes to this menu since you last stored it in the Storage Menu.

[2] **Update memory.** This field lets you specify the memory you want updated (and overwritten) and displayed when you complete the next acquisition. You can specify memory 1, 2, 3, or 4. Press 0 or 2 to cycle through the selections.

[3] **Display format.** This field lets you specify the default display format for the update memory. For example, if you specify timing, when the acquisition stops, the data is displayed as a timing diagram. Press 0 or 2 to change the format from state, to disassembly, to timing. You must have an optional disassembly probe connected to get a disassembly display.

[4] **Trigger Position.** You can specify the trigger position in increments of 128: 128, 256, 384, and so on, through 1920. The trigger position indicator (the box) to the right of this field shows graphically where the specified trigger is in memory. Press 0 or 2 to cycle through the selections.

[5] **Trigger position indicator.** This box shows where the trigger specified in the trigger position field is in memory. For example, if you set the trigger position to be 0512, the trigger position indicator shows that 0512 is approximately one-quarter of the way through memory.

[6] **Look for Trigger.** This field tells the analyzer when to begin searching the trigger immediately, continuously (any time), or after the pre-trigger memory is full. Press 0 or 2 to cycle through the selections.

[7] **Memory Compare.** Use this field to specify a manual or automatic comparison of the update memory to the specified reference memory. If you do an automatic comparison, you can then tell the analyzer to display and stop, discard and stop, or discard and re-acquire (memory compare actions). You can also specify that the analyzer compares memories for equalities or inequalities. Press 0 or 2 to toggle from manual to automatic memory comparisons. You can't do an automatic memory comparison if you tell the analyzer to look for the trigger continuously.

[8] **Auto Run Count.** This field shows how many comparisons of the update memory to the specified reference memory have been performed.

[9] **Memory compare specification.** This field specifies the reference memory you want to compare the update memory to for automatic memory comparisons. (Manual memory comparisons are done by specifying the compare memory in the Mem Select menu.) You can specify memory 1, 2, 3, or 4. The update memory is selected in the upper portion of this memory. Press 0 or 2 to cycle through the memories.

10 **Memory compare actions.** If you do an automatic data comparison, you can use these fields to tell the analyzer to do certain actions depending on whether memories are equal or unequal. You can specify these actions: display and reacquire, display and stop, or discard and reacquire. Press 0 or 2 to cycle through the selections for each field.

11 **Memory compare locations.** You can specify a range of memory (instead of the entire 2K analyzer memory) to be compared. You can specify ranges from 0000 to 2047. Press ENTER while the cursor is on this field, then enter address values directly from the keypad. Press ENTER again when you finish entering the new memory locations.

12 **Channel Mask.** You can set a channel mask for memory comparisons by specifying a condition's name in this field. If the specified condition word includes don't-care values, the analyzer won't compare data acquired on those channels (the channels will be masked). This is done for you with the version 4.0 software—just press C. If you're doing a memory comparison, memory differences on those channels will be ignored. Data from channels containing any value other than don't care will be compared.

13 **Data display update.** This field applies when you tell the analyzer to look for the trigger continuously or when you specify an automatic memory comparison. This field lets you specify the amount of time the analyzer displays data before going back to the Trigger Spec menu and acquiring data again.

14 **Conditions viewport.** This viewport shows the comparison mask condition you're currently using. For example, if you're using a channel mask called EXTERN4, the viewport shows the channel definition word of EXTERN4. You cannot select this field.

15 **Menu bar.** The menu bar at the bottom of the screen lists the keys to press for menu functions.

The top part of the screen controls the display and storage of data features:

- Look-for-trigger instruction
- Trigger position in increments of 128
- Default display format (state, timing, or disassembly)
- Update memory specification

Features in the lower half of the menu include:

- Automatic or manual memory comparison
- Memory comparison specification
- Memory range for comparison
- Channel mask qualification
- Data display time

Trigger Specification

You can specify a trigger after the pre-trigger memory is full, immediately, or continuously. It's important that you specify the trigger position so that memory is filled with the data you need to see.

For example, if you want to see what occurred before a trigger condition, tell the analyzer to fill memory with pre-trigger data and position the trigger near the end of the analyzer's memory. This lets you see almost 2K of acquired data that led up to the trigger condition. If you want to see what happens after a condition occurs, position the trigger near the beginning of memory; for example, at location 0128.

How Triggering Works. When a logic analyzer acquires data, it fills the pre-trigger memory you specify in the Run Control menu and then looks for the trigger. At this point, the analyzer looks at each sample as it occurs. As each sample is acquired, the analyzer stores the sample in the trigger position, shifting the previously acquired data back in memory and overwriting the pre-trigger memory as if it was a first-in-first-out queue. When the analyzer finds the trigger event, it stores that data in the trigger position, then acquires enough data to fill the rest of memory with post-trigger data before it stops.

If the trigger occurs before the trigger position is filled the first time, the analyzer won't see the event. For example, if the trigger position is 512 but the event occurs on the 400th cycle of the acquisition, the analyzer isn't enabled to see the trigger. Instead, the analyzer continues to run until you press STOP, or until the trigger event occurs again. In this case, you should tell the analyzer to look for the trigger immediately. As soon as the trigger occurs, the analyzer stores that sample, fills the post-trigger memory, and you specified.

Memory Compare

You can use the memory compare feature to compare a memory to another stored acquisition, or to compare a memory to itself. If you compare one memory to another, the state table displays the differences between the two. If you compare a memory to itself, you clear any marked differences (from a previous comparison) from the state display. 1230 Logic Analyzers with version 3.5 software highlight data differences by group. 1230 Logic Analyzers with version 4.0 software highlight data differences by bit.

You can compare memories manually or automatically. If you do an automatic comparison, you can then tell the analyzer to display or discard, and reacquire or stop, whether memory is equal or unequal. However, if you tell the analyzer to look for the trigger continuously, you can't make an automatic memory comparison at the same time. The two operations are not compatible.

Section 4

DATA MENUS

Data menus let you specify how you want to display and manipulate data. Table 4-1 briefly describes the data menus and screens. This section explains the menus and screens in the order they are listed in the table.

Table 4-1
DATA MENUS

Data Menu	Descriptions
Memory Select	Shows the main setup parameters for each of the four memories and also specifies the display memory.
Acquisition Process	Displays timebase and acquisition status during data sampling.
State	Displays acquired data in state table format and also shows the differences from memory comparisons.
Disassembly	Shows disassembled data in hardware or software format only if you use an optional microprocessor disassembly probe. Refer to your disassembly probe operator's manual for detailed information about disassembly.
Timing	Displays data as timing traces. This screen also lets you reorder, duplicate, and delete traces from the display to show only the information you want.

MEMORY SELECT MENU

The Memory Select menu, shown in Figure 4-1, lets you choose a different memory for display than is specified in the Run Control menu. It also lists the main setup parameters for the data stored in each of the analyzer's four memories:

- The name of the setup used for the acquisition

- The date and time the data was acquired
- The probe links used for the acquisition
- The timebases for each group of linked probes
- The sampling rate and format
- Whether or not glitch detection was turned on

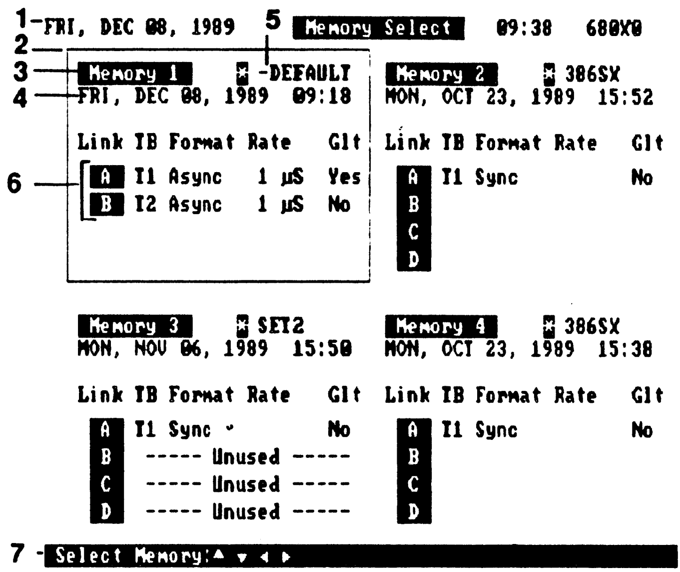


Figure 4-1. Memory Select menu. This menu displays the main setup parameters for the acquired data stored in the analyzer memories.

[1] **Menu header.** This line lists the menu name, current date and time, and the current setup name. If an asterisk is shown before the setup name, you changed the setup since you last stored it in the Storage menu.

[2] **Box.** A box encloses the memory currently selected for display. Use the cursor keys to select a different memory.

[3] **Memory number.** Identifies each of the four memories.

[4] **Date and time.** This field lists the date and time at which you acquired the data stored in this memory.

5 **Setup name.** This field lists the name of the setup you used to acquire the data. If there is an asterisk before the setup name in a memory configuration, you made changes to the setup after you acquired data with it and before you stored it last in the Storage menu.

6 **Setup fields.** These columns show the timebase, probe links, sampling rate and format, and glitch capture status used to acquire the data in this memory.

7 **Menu bar.** The menu bar lists the keys to press to move the selection box from memory to memory.

Immediately after an acquisition, the analyzer displays the data sampled from your SUT. There are many times though, when you want to see other data-data from other acquisitions, data acquired with different sampling rates or formats, data acquired from other SUTs, and so on. By moving the box in the Memory Select menu to the memory from which you want to display data, you can choose to display the stored data in any of the four 2K-deep analyzer memories.

The analyzer can display stored data if the current sampling rate is as fast or faster than the rate at which the stored data was sampled. For example, you acquire data at 40 ns, store it, and then acquire more data at 20 ns. In the Mem Select menu, you can still choose to display the data acquired at 40 ns. The analyzer will show the lower 8 bits of the stored data (only the lower 8 bits are enabled at the current rate selection, which is 20 ns).

If you sample data at 10 ns, store it, and then sample data at 100 ns, you cannot display the data acquired at 10 ns until you change the rate back to 10 ns. This is because the analyzer is set up to show data acquired at the slower rates.

If the analyzer can't display the data, it gives you an error message telling you whether the probe links, rate, or glitch selection doesn't match. Pressing MENU again saves the Memory Select menu and returns you to the Main menu. From there, you can call up the setup menus and change the required parameters needed to match the stored data.

To display stored data, follow these steps:

1. Save your current setup.
2. Select the memory you want to display in the Memory Select menu, noting the required timebase parameters.
3. Display the data by calling up the State or Timing menus.

ACQUISITION PROCESS SCREEN

The Acquisition Process screen tells you what the analyzer is doing after you press START to acquire data. The screen also lists the status of each timebase in which the trigger specification is running, and displays the statement level currently being processed. This screen also tells you when memory is full and acquisition is complete, and shows the autorun count.

Figure 4-2 shows a sample Acquisition Process screen for an analyzer set up for 48 channels (16 in each timebase).

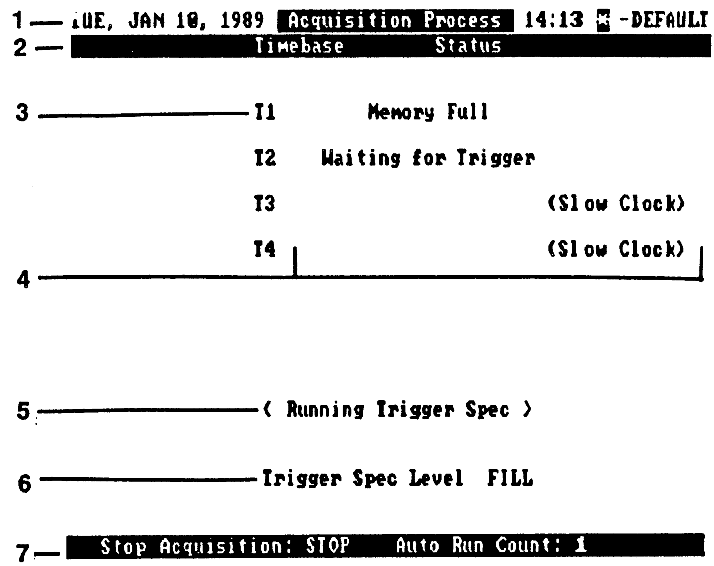


Figure 4-2. Acquisition Process screen. This screen appears while the analyzer samples data from your SUT. The messages tell you the timebase, trigger statement level, and acquisition status for the current process.

[1] **Menu header.** This line lists the menu name, current date and time, and current setup name. If there is an asterisk before the setup name, you made changes to the setup since you last stored it in the Storage menu.

[2] **Column headings.**

[3] **Defined timebases.** Lists the timebases specified in the Timebase menu.

[4] **Timebase status.** This field tells you what the analyzer is doing in each timebase: waiting for the trigger, triggered, and memory full. If the clocks are occurring slower than 40 ms, the field tells you (to the right) there is a slow clock.

[5] **Acquisition status.** Displays the status of the acquisition. The brackets flash on and off to tell you if the acquisition is still going on. Status messages are: initializing, running the Trigger Spec, acquisition complete, acquisition aborted, and comparing memories.

6	**Trigger level.** Identifies the statement level in the Trigger Spec that is currently being executed. If memory is being filled with data, this field says FILL.
7	**Menu bar.** The menu bar at the bottom of the screen tells you that during acquisition, the analyzer ignores input keystrokes except for STOP, which stops the current acquisition.

When you press START, the analyzer displays the Acquisition Process screen until memory is full and the acquisition is complete, or until you press STOP. When the acquisition is complete, the analyzer immediately displays the acquired data in the format specified in the Run Control menu. If you stop the acquisition before memory is completely filled, the analyzer displays as much acquired data as possible. If the analyzer suspects that invalid data was acquired, it dims that data in the state table.

The Acquisition Process screen will not be displayed if the analyzer is looking for the trigger continuously (specified in the Run Control menu). The next two lists explain the timebase status and acquisition status messages.

Table 4-2
TIMEBASE STATUS MESSAGES

Message	Description
Waiting for trigger	The analyzer is sampling data in the SUT and looking for the trigger.
Triggered	The analyzer found the trigger and is filling or trying to fill memory.
Memory full	The analyzer found the trigger and filled memory as you specified.
Slow clock	The analyzer is trying to acquire data You may have specified an external clock that does not exist or one that is extremely slow (for example, 10 ms or less).

Table 4-3
ACQUISITION STATUS MESSAGES

Message	Description
Initializing	The trigger specification is not enabled. Pre-trigger data is being acquired, but the analyzer is not yet looking for the trigger condition.
Running trigger spec	The analyzer is executing the trigger specification statements and looking for the trigger.
Acquisition complete	The acquisition is finished, and the analyzer is processing the acquired data to display it in the specified format (state, timing, or disassembly).
Acquisition aborted	You pressed STOP before the acquisition completed.
Comparing memories	The analyzer is doing an automatic memory comparison and is discarding and re-acquiring data as specified in the Run Control menu. This message can also mean that you told the analyzer to display and stop, and the comparison takes more than one or two seconds.

STATE MENU

The State menu lists the data acquired from your digital circuit in state table format. You can display this data in hexadecimal, channel groups that are in the same timebase. You can display other timebases simply by pressing 9. You can also choose to ignore channel groups and display only data from specific groups. Figure 4-3 shows a sample state table.

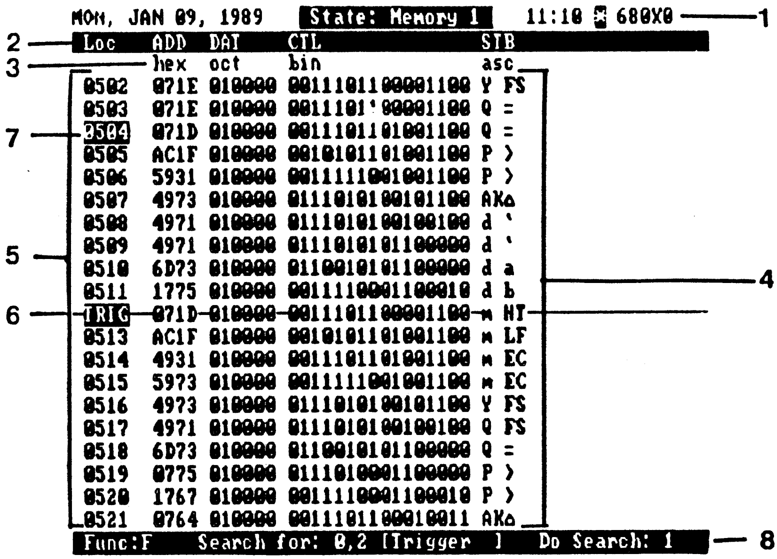


Figure 4-3. State menu. The cursor is on the trigger location and the channel group GPC has been turned off. The location marked in reverse video shows that the trigger condition occurred more than once.

[1] **Menu header.** This line lists the menu name, current date and time, and current setup name. If an asterisk appears before the setup name, you made changes to the setup since you last stored it in the Storage menu.

[2] **Group names.** This field displays the group names assigned in the Channel Grouping menu. If there isn't enough room on the screen to show all groups, the analyzer puts a > in the last column of the radix line to show that there is more channel group information. To see these groups, change the first groups' radixes to radixes that use fewer characters, such as hexadecimal.

[3] **Radix.** This field lists the display radixes of the active channel groups. This field affects the state table display only. If you change the radix here, it won't affect the Channel Grouping or Conditions menus. Press E to change radixes. Press 0 to cycle through the groups and 2 to cycle through the radix options: hexadecimal, binary, octal, ASCII, or off.

[4] **Data.** The analyzer can display up to 20 lines of state table data on the screen at one time. Use ▲ and ▼ to scroll through data in increments determined by the scroll rate. Use ◀ and ▶ to scroll up or down data one line at a time. Dim video identifies data that is invalid. Reverse video indicates data that was compared to another memory and is different from the other memory.

[5] **Memory location number.** This column lists the location of the data in memory. If locations are in dim video, the memory locations are invalid (data at those locations is probably invalid also).

[6] **Trigger location.** The trigger location is marked with the word TRIG. When the analyzer triggers and stops acquiring data, it labels the trigger location.

[7] **Subsequent trigger mark.** If the trigger condition occurs more than once, each additional occurrence is marked with inverse video in the location column.

[8] **Menu bars.** The State menu has four menu bars that tell you which keys to press for menu functions. Press F to cycle through the menu bars.

The State menu has several features that help you display and compare acquired data quickly and easily. You can:

- Scroll data by lines or by screens
- Jump to a specific memory location
- Specify the display radix for acquired data
- Search for specific events
- Display data acquired with different timebases

- Remove a channel group from the display
- Toggle between update and reference memories
- Compare memories
- Count the differences between memories

The next several discussions explain these features in detail.

Table 4-4 briefly describe the keys to use for state table features.

Table 4-4
STATE TABLE KEYS AND FUNCTIONS

KEY	FUNCTION
s t	Scroll back or forward by the scroll rate.
◀ ▶	Move the cursor up or down one line at a time.
7, 8	Decrease or increase the scroll rate.
ENTER	Enter the location to which you want to jump. You don't have to enter all four digits to jump to a new location. If you don't enter all four digits, scroll out of the field or press ENTER again to make the jump
E	Edit channel group radixes. 0 — Select channel group. 2 — Choose the radix for the channel group. ENTER — Save your changes and exit.
0, 2	Select the condition to search for (press 2 to cycle through alphabetically; press 0 to cycle through in reverse order). You can search for any of the 24 conditions you can define, the beginning or end of acquisition, the trigger event, the next occurrence of the trigger event, and the next difference between memories (if you compared memories).
1	Search for the specified condition.
9	Select the timebase for displaying data (you can display one timebase on the screen at a time).
X	Toggle between display and reference memories.
C	Compare the display memory to the reference memory. The menu bar lists the number of differences between memories. Press X to toggle between display and reference memories.
F	Display the next menu bar.

The next several discussions explain these features in detail.

Scroll by Lines or Screens

Use the cursor keys ◀ and ▶ to move up or down the screen one line at a time. Since the analyzer memory wraps back onto the screen, you can scroll up or down to show the beginning (0000) and end (2047) of memory on the screen at the same time. A blank line separates the beginning from the end of memory.

Use ▲ and ▼ to scroll data onto the screen by the scroll factor, which can be from 2 to 20 lines of data. Set the scroll factor by pressing 7 to lower and 8 to raise it. When you press 7 or 8, the menu bar is updated to show the current scroll factor. You can also press F to cycle through the menu bars.

Jump to a Specific Location

The analyzer lets you jump to specific memory locations. You can specify a location in the range 0000 to 2047. If you choose a location outside this range, the jump location defaults to 2047. Follow these steps to jump to a specific location:

1. Press ENTER to specify a location. The cursor highlights the first character in the location at the top of the screen.
2. Enter the memory location (such as 0821) directly from the keypad. The cursor automatically moves from one character to the next, but you can also use ◀ and ▶ to move within the range.

As soon as you enter a location, the analyzer jumps to that location, displaying the location at the top of the screen.

Anytime you leave the location field, whether you entered an entire location or not (for example, 0156 rather than 01--), the analyzer jumps to the specified location. If you don't enter an entire location, the analyzer uses the value of the old location in the jump field to fill in the digits you didn't specify. Because of this, there are several ways to leave the field and make the jump to the new location:

- Finish entering a location.
- Scroll left or right out of the location field.
- Scroll up or down off the line you are editing.
- Press ENTER.

Change the Radix or Turn Off a Group

The analyzer lets you choose the radix for displaying each channel group. You can choose to show state table information in hexadecimal, octal, binary, and ASCII radixes. You can also turn a channel group's radix off. Turning a group's radix off removes the entire group from the display. Follow these steps to change a group's radix.

1. Press E to edit the radixes. The menu bar shows the currently selected group and that group's current radix.
2. Press 0 to cycle through available groups till the menu bar shows the one you want to change.
3. Press 2 to cycle through available radixes, including OFF.
4. Press ENTER when done, to reset the state table in the new radixes.

If there are less than seven channels in a group's byte and you change that group's radix to ASCII, the byte won't be displayed. The byte also won't be displayed if you change the radix to ASCII, then go to the Channel Grouping menu and delete channels from the specific group so that there are less than seven channels in the byte.

Figures 4-3 and 4-4 show state tables with different radixes. Figure 4-4 also shows that channel group GPC has been turned off and removed from the display.

Search for Specific Events

You can search for a specific event in the state table (and in the Timing diagram with version 4.0 software). One of the menu bars lists the current search event (press F to cycle through the menu bars). You can search for:

- The 24 conditions defined in the Conditions menu
- The end of the acquisition
- The beginning of the acquisition
- The trigger

- The next difference (if you compared memories)

Follow these steps to search for an event:

1. Press 0 or 2 to cycle through the available conditions and events till you display the event for which you want to search. The menu bar automatically changes to show the current search event.
2. Press 1 to search for the event.

After you press 1, the analyzer searches forward through memory for the next occurrence of the event. If the search takes longer than 1 or 2 seconds, the analyzer displays a message telling you it is searching for the event. If the analyzer doesn't find the event before reaching the end of memory, it wraps around to the beginning of memory and continues to search until it reaches the point where you began. If you choose an event that does not occur, the analyzer displays a message telling you the event was not found.

When the analyzer finds the event, it places the cursor on it. If the analyzer redraws the screen, it displays the event in the center of the screen. Figure 4-4 shows an example of a search for the trigger.

```
MON, JAN 09, 1989   State: Memory 1     11:15   680X0
Loc    CTL               STB
       bin               asc
0502   0110010100111011  Y FS
0503   0110010101111111  Y FS
0504   1001110001111101  Q =
0505   0011101100001100  P >
0506   0011101100001100  P >
0507   0010101100001100  AKo
0508   0011101101001100  d `
0509   0011111001001100  d `
0510   0111010100101100  d a
0511   0111010100100100  d a
TRIG   0110010100100100  d b
0513   0111010101100000  d s
0514   0011110001100010  d s
0515   0011101100010011  P t
0516   0011101100010011  X U
0517   0010101100010011  X U
0518   0011101101010011  Y V
0519   0011111001010011  M EB
0520   0111010100110011  M EB
0521   0111010100111011  m BS
Func:F Scroll Rate:7,8[20] Timebase:9[T2 Async   4 µS]
```

Figure 4-4. Search for the trigger. The event being searched for, the trigger, is placed in the center of the screen in the state table.

Display Data in Different Timebases

Only data acquired with the same timebase can be displayed on the screen at the same time. One of the menu bars shows the timebase in which the current data display was acquired. Press 9 to display data acquired with a different timebase. In the menu bar, the brackets containing the timebase also contain the format and rate of the acquired data.

For example, you display data acquired with timebase T2. To show data acquired with timebase T3, press 9. The brackets show that T3 is now the current display timebase, and the state table screen now shows data acquired with T3.

If a timebase has no channels assigned to it, the state display for that timebase is blank (the analyzer didn't acquire any data for that timebase). If a timebase doesn't exist because of the probe links in the Timebase menu, you cannot display a State menu for that timebase.

Compare Memories

The state table includes a memory compare feature. This lets you compare the same data locations in different memories, count the differences, and display the results. Follow these steps to compare memories:

1. Call up the Run Control menu and specify the reference memory you want to compare the currently displayed data to. (The Mem Select menu shows which memory is selected as the currently displayed or update memory.) The reference memory may be any of the four analyzer memories.
2. Specify a range of locations to compare, along with a comparison mask. Any condition defined in the Conditions menu may be selected as the comparison mask. Channels defined in condition words as an X are not compared. Any other value in the condition definition (0 or 1) causes the channel to be included in the comparison.
3. Call up the state table and press C to compare the memories. When the comparison is complete, the analyzer shows the update memory on the screen, regardless of what was there before. The menu bar lists the number of differences between the memories. The data differences are displayed in reverse video; invalid locations or data are displayed in dim video.
4. Press DON'T CARE to toggle the display between the update memory and the reference memory. Note: If the reference memory contains reverse-video data locations, they are the result of a previous comparison. To clear the reverse video of the memory comparison from the screen, compare the memory to itself. Call up the Run Control menu and set the reference memory field to the memory you want to clear. In the Mem Select menu, select the same memory. For example, compare memory 3 to memory 3. Then, return to the State menu and press C. The differences are cleared from the display.

Figure 4-5 shows an example of a compared memory with differences. If the compare range or the differences you want aren't currently displayed, you can use the search function to search for the next difference. To do this, Press 0 or 2 to cycle through the search events till you specify Next Diff for the next difference in memories. Press 1 to search for the event.

```
TUE, DEC 12, 1989    State: Memory 1     01:56 ■ -DEFAULT
 Loc    GPA   GPB        GPC      GPD
        hex   bin        oct      asc
 0500   C43D  00001111   177616   △ SP
 0501   C43D  00001111   177616   △ SP
 0502   C43D  00001111   177616   △ SP
 0503   C47F  10001111   177616   △ 0
 0504   C47F  10001111   177616   △ 0
 0505   0A3D  10001111   177616   △ b
 0506   0A3D  10001111   177616   △ b
 0507   0A7F  10001111   177672   △ r
 0508   0A7F  10001111   177672   △ r
 0509   0A7D  10031111   177372   △ r
 0510   FA1C  10031111   177372   △ NL
 0511   FA1C  10031111   177372   △ NL
-TRIG---FA5E--00011111--177744---△ DE-----------------------
 0513   FA5E  00011111   177744   △ DE
 0514   FA1C  00011111   177744   △ D
 0515   FA1C  00011111   177744   △ D
 0516   FA5E  00010111   177744   △ I
 0517   FA5E  00010111   177744   △ I
 0518   FA5E  30010111   177744   △ I
 0519   FB1C  30010111   177744   △ EI
 Func:F     Search for: 0,2 [Trigger  ]   Do Search: 1
```

Figure 4-5. State table with memory comparison. In the data columns, reverse video identifies data locations where the update memory differed from the reference memory.

You can compare any two memories. The links and channel groups of the reference and update memories don't have to match, but if they don't, you may not receive information useful to you from the comparison.

Reverse video marks differences between the compared memories. To clear the reverse video of the memory comparison from the screen, compare the memory to itself. Call up the Run Control menu and set the reference memory field to the memory you want to clear. In the Mem Select menu, select the same memory. For example, compare memory 3 to memory 3. Then, return to the State menu and press C. The differences are cleared from the display.

In the locations column, reverse video marks other occurrences of the trigger condition. Dim video indicates invalid data or locations (not memory differences). For example, you set up the analyzer to acquire on 64 channels but use only three probes to take the acquisition. In this case, the analyzer still displays the groups on which it expects data from the fourth probe. However, since no data was acquired on the fourth probe, the displayed information for that probe's channel groups is in dimmed video so that you know the data is invalid.

Track Data Locations for the Timing Diagram

The cursor tracks the same memory location in the state table and timing display. If you call up the State menu and then call up the Timing menu, the cursor is positioned on the same memory location in both menus. This lets you get timing information for the same data you're displaying in the state table.

For example, if the cursor is on location 0185 in the state table, when you call up the Timing menu, the cursor is on location 0185 in that screen too. If you change the cursor position in the timing diagram, the cursor in the state table changes also.

DISASSEMBLY DISPLAY

Microprocessor disassembly is an option to the 1230 Logic Analyzer. When you power up with an optional microprocessor probe connected, the analyzer asks if you want to use the setup for the microprocessor probe. If you want to use the microprocessor probe's setup, press ENTER to load it. Otherwise, press MENU to preserve the current analyzer setup and access the normal analyzer menus.

Disassembly information is discussed in the operator's manual for each microprocessor disassembly probe.

TIMING MENU

The Timing menu displays data from up to 64 channels (16 on a page at a time) as digital waveforms or traces. The horizontal axis is time, and the vertical axis is the logic level (high or low). Figure 4-6 shows a Timing menu.

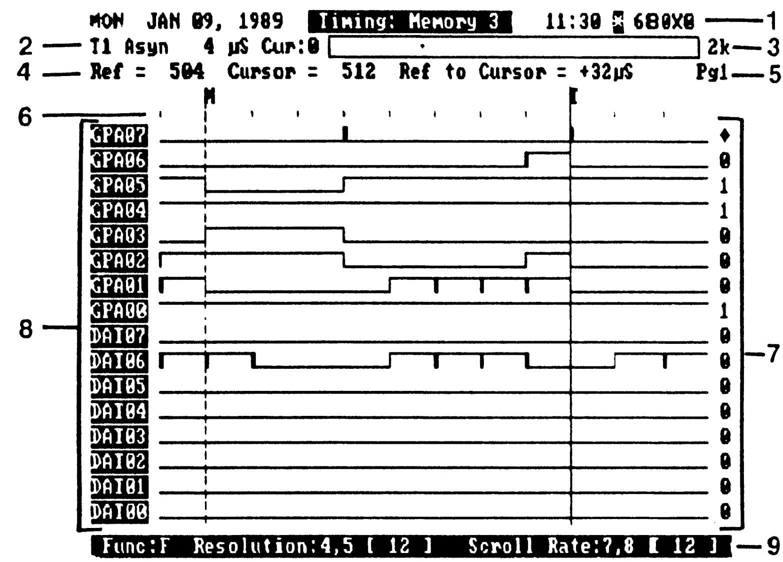


Figure 4-6. **Timing diagram.** The dotted line is the cursor reference location; the solid line is the cursor. The T marks the trigger location; the M marks other occurrences of the trigger condition. Press 0 or 2 to display the next available page of traces. If you acquired glitches, they are displayed as small, bright vertical lines rising or falling from the trace.

[1] **Menu header.** This line lists the menu name, the memory currently being displayed, the current date and time, and the current setup name. If there is an asterisk before the setup name, you made changes to the setup since you last stored it in the Storage menu.

[2] **Timebase.** Identifies the timebase and clock format with which you acquired the currently displayed data. You can display traces together on the screen only if they were acquired with the same setup and timebase.

[3] **Memory indicator.** This box represents the analyzer's 2 K of memory. The T shows the relative location of the trigger word. A vertical tic mark shows the relative position of the timing diagram's cursor.

[4] **Ref, cursor, and ref to cursor.** These three fields show the cursor reference location, the cursor position, and the distance between the two. The reference location is the dotted line; the cursor is the solid line. If you use an asynchronous sampling format, the distance between the cursor and reference location is in units of time; if you use a synchronous sampling format, the distance is in your SUT's clock cycles.

[5] **Channels.** This field tells you which page of channels is currently displayed if you're using more than 16 channels. Depending on the number of probes you're using, you can display one of four pages of channels.

[6] **Graticule.** The line of tic marks represents the number of samples displayed on the screen. The analyzer displays a tic mark for each sample. As you display more and more samples on the screen (resolution decreases) the tic marks move closer together. In the graticule, the trigger location is marked by a T. Subsequent triggers and locations where storage was turned on are marked with an M. If you set resolution to 2048, the graticule is a solid bar, and you can see the entire range of memory on the screen at once.

[7] **Binary trace value.** This field gives you the logic level of each displayed trace at the cursor location.

[8] **Trace names.** The analyzer lists the name of each trace (for each channel) in the timing diagram. By default, the traces are named from the 3-character channel group name and the 2-digit channel position in the Channel Grouping menu. You can edit the traces by pressing E, then using the keys listed in the menu bar.

[9] **Menu bar.** The menu bar tells you which keys to press for the menu functions. Press F to cycle through the menu bars.

The timing display has several features that let you manipulate data. You can:

- Scroll through data
- Change the data resolution
- Display different pages of timing traces
- Set a reference point for the cursor
- Search for events
- Track the cursor location in the state table
- Edit the timing trace labels
- Restore the trace names to the defaults
- Reorder, duplicate, and delete timing traces
- Restore the traces to the default order

Table 4-5
TIMING DIAGRAM KEYS AND FUNCTIONS

<table>
<tr><th>Key</th><th colspan="2">Function</th></tr>
<tr><td>0</td><td colspan="2">Displays the previous page of traces.</td></tr>
<tr><td>2</td><td colspan="2">Displays the next page of traces.</td></tr>
<tr><td>4</td><td colspan="2">Decreases the display resolution.</td></tr>
<tr><td>5</td><td colspan="2">Increases the display resolution.</td></tr>
<tr><td>◀ ▶</td><td colspan="2">Scrolls left or right one sample at a time.</td></tr>
<tr><td>▲ ▼</td><td colspan="2">Scrolls right or left by the scroll rate.</td></tr>
<tr><td>7</td><td colspan="2">Decreases the scroll rate.</td></tr>
<tr><td>8</td><td colspan="2">Increases the scroll rate.</td></tr>
<tr><td>B</td><td colspan="2">Resets the timing traces to the default order. The default order is determined by the current order of channels in the Channel Grouping menu.</td></tr>
<tr><td>C</td><td colspan="2">Returns the timing trace labels to the default names. The default trace names are the 3-character channel group names and the 2-digit channel number (00, 01, etc).</td></tr>
<tr><td rowspan="4">E</td><td colspan="2">Lets you edit timing traces:</td></tr>
<tr><td>E</td><td>Lets you edit trace names with these keys:
0-F Hexadecimal values 0-F
X Space
▲ ▼ Select other characters
ENTER Save the new name and exit</td></tr>
<tr><td>D</td><td>Lets you reorder, duplicate, or mask timing traces with these keys:
0 Display previous trace here
2 Display next trace here
1 Display next trace in the next line
3 Turn this trace off or on
ENTER Save the new order and exit</td></tr>
<tr><td>ENTER</td><td>Saves your changes and exits.</td></tr>
<tr><td>F</td><td colspan="2">Displays the next menu bar.</td></tr>
<tr><td>X</td><td colspan="2">Toggles between display and reference memories.</td></tr>
<tr><td>ENTER</td><td colspan="2">Sets a reference location for the cursor.</td></tr>
</table>

Move Through the Display

For each displayed page of timing traces, you can move through the traces in two ways:

- Scroll one data sample at a time with ◀ and ▶.
- Scroll the number of samples specified in the menu bar's scroll rate field. Use ▲ and ▼ to scroll the specified number of locations. The cursor stays in its current position and the data scrolls by the cursor.

The scroll rate lets you specify how fast you move through the timing display when you use ▲ and ▼. Press 7 to decrease the scroll rate or 8 to increase it. One of the menu bars lists the current scroll rate.

You can set a scroll rate of 6, 12, 24, 48, 96 (default), 128, 256, 512, 1024, or 2048 data samples. The only restriction is that the scroll rate cannot be larger than the data resolution. When you change the data resolution, the scroll rate is automatically updated to the largest value possible. This makes it easy to scroll through data quickly.

The analyzer also lets you show the beginning and end of memory at the same time. A blank vertical line separates the two in the display. Figure 4-7 shows the beginning and end of memory on the same screen.

Set the Data Resolution

The resolution feature lets you choose the number of samples displayed on the screen. Press 4 to decrease the resolution or 5 to increase it. You can set a resolution of 6, 12, 24, 48, 96 (the default), 128, 256, 512, 1024, or 2048. One of the menu bars lists the current resolution.

If you set the data resolution to 2048, the entire 2K memory is displayed on the screen. The graticule becomes a solid bar at the top of the timing diagram. Solid bars in the traces represent timing information too dense to show distinct rising and falling edges. You can use the resolution of 2048 to see or move to a specific event quickly. You can then increase the resolution so that the specific timing information you want is more clear. Figure 4-7 shows different data resolutions.

At resolutions of 96 or lower, each time you press the ◀ and ▶ keys the cursor moves one time unit. However, at resolutions above 96, each key press moves the cursor more than one time unit. Generally, if you want to accurately measure the duration of a signal, make sure the resolution is set to 96 or lower.

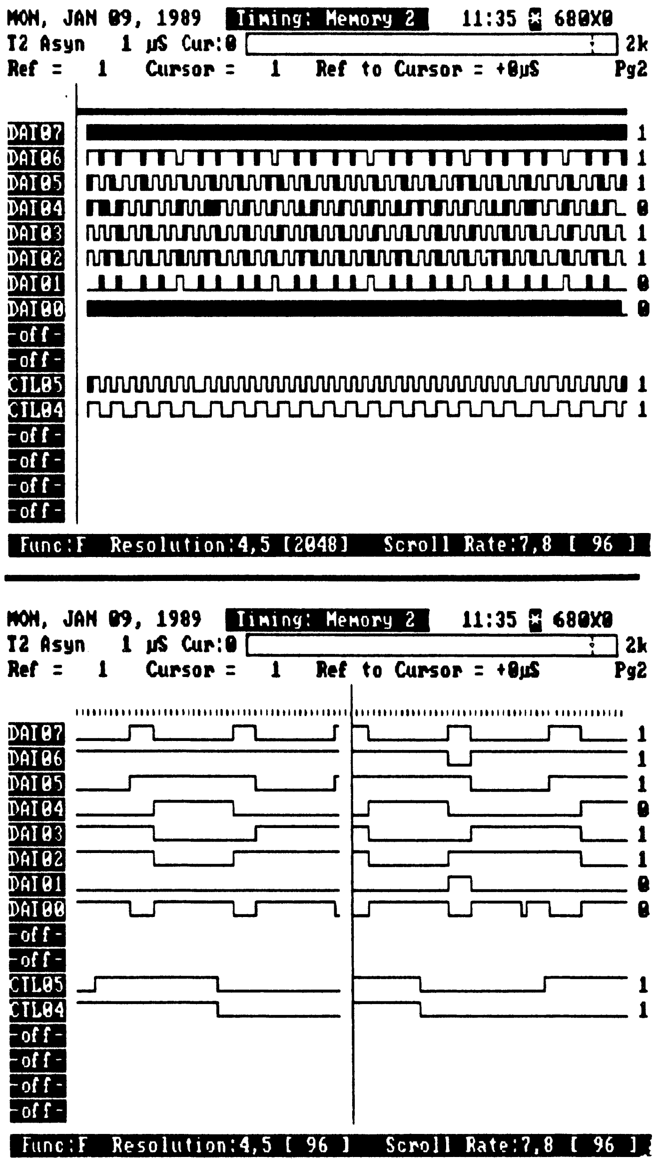


Figure 4-7. Data resolution at 2048 and 96. The top half-screen shows the entire memory displayed at once. Solid lines indicate that too much data is being displayed to show separate rising and falling edges. The bottom screen shows fewer samples, but shows the data more clearly.

Sometimes glitches appear in displays that have a lower resolution (greater number of cycles displayed on the screen). If the glitch is a result of the resolution, you can increase the resolution to see the cycles actually occurring at that point in your circuit.

Display Pages of Traces

Each page of timing traces displays up to 16 traces for each acquisition probe on which data was acquired. For example, if you use one probe and 16 channels, there is one page of 16 traces available for display. If you use four probes and 64 channels, there are four pages of 16 traces each available for display. Press 0 or 2 to cycle through the available pages.

Only timing traces acquired in the same timebase can be displayed on the same page. For example, if you use two probes, one in T1 and one in T2, one page shows traces in T1 and the other page shows traces in T2. Figure 4-8 shows how data acquired in different timebases is displayed on separate timing diagram pages.

The analyzer tries to show as many channels from each timebase as can fit on the screen. If both probes are in the same timebase and you use 8 channels from each probe, both probes' channels (all 16) are shown on the same page.

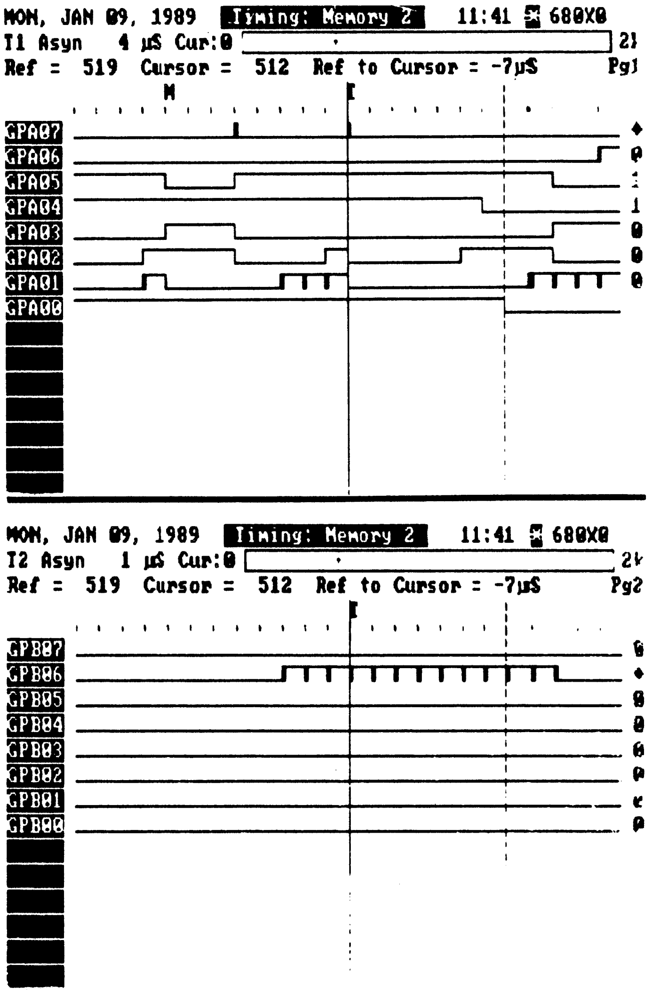


Figure 4-8. **Displaying channels in different timebases.** Channel group GPA has 8 channels: A07-A00. Channel group GPB has 8 channels: B07-B00. The two channel groups GPA and GPB are in different timebases, so traces GPA07-GPA00 are shown on page 1 and GPB07-GPB00 on page 2. Press 0 or 2 to cycle through the pages. Glitches are displayed as intensified marks. The binary read-out for the timing traces shows that the acquired glitches at the cursor are generic glitches.

Set a Cursor Reference Location

You can set a reference location for the data cursor. This makes it easy for you to see when events occurred in relation to each other. The Ref-to-Cursor field shows the time that elapsed between when the data at the reference location occurred and when the data at the cursor position occurred. Figure 4-8, earlier in this section, shows a cursor reference set near the trigger point.

To set the cursor reference, move the cursor to the location and press ENTER. When you move the cursor away, the dotted reference line remains at that spot. At the top of the screen, the reference location, (for example, 0802), the cursor location (for example, 0825), and the distance between the two (for example, +2.3 µsec) are listed. The analyzer keeps this line updated as you move the cursor through the timing diagram.

In asynchronous sampling format, the reference-to-cursor interval is shown in units of time. In synchronous sampling, the interval is shown in clock cycles.

Track the Display in the State Table

The timing diagram cursor marks the location of the data called up in the state table. For example, if the cursor marks location 1052 in the timing diagram, the next time you call up the state table, the cursor will be on the same location.

Search for Events

The easiest way to search for a particular event is to call up the state table, do the search, then call up the timing diagram again. If your 1230 is using software version 4.0 you can do the search from the Timing screen. The cursor marks the location of the event in both menus. For example, Figures 4-9 and 4-10 show how you can quickly and easily search for an event in the state table, then display it in the timing diagram.

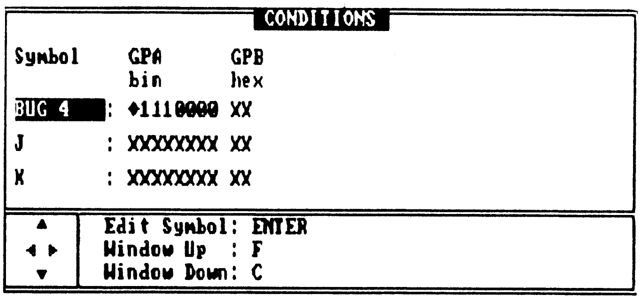
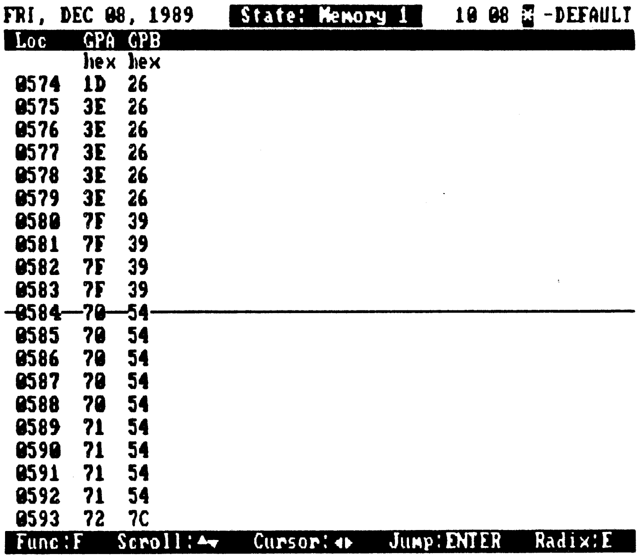


Figure 4-9. Conditions for a search. The trigger condition is named BUG 4, and is defined with generic glitches on three channels.


FRI, DEC 08, 1989 State: Memory 1 10 08 -DEFAULT

Loc	GPA hex	GPB hex
0574	1D	26
0575	3E	26
0576	3E	26
0577	3E	26
0578	3E	26
0579	3E	26
0580	7F	39
0581	7F	39
0582	7F	39
0583	7F	39
0584	70	54
0585	70	54
0586	70	54
0587	70	54
0588	70	54
0589	71	54
0590	71	54
0591	71	54
0592	71	54
0593	72	7C

Func:F Scroll:▲▼ Cursor:◄► Jump:ENTER Radix:E

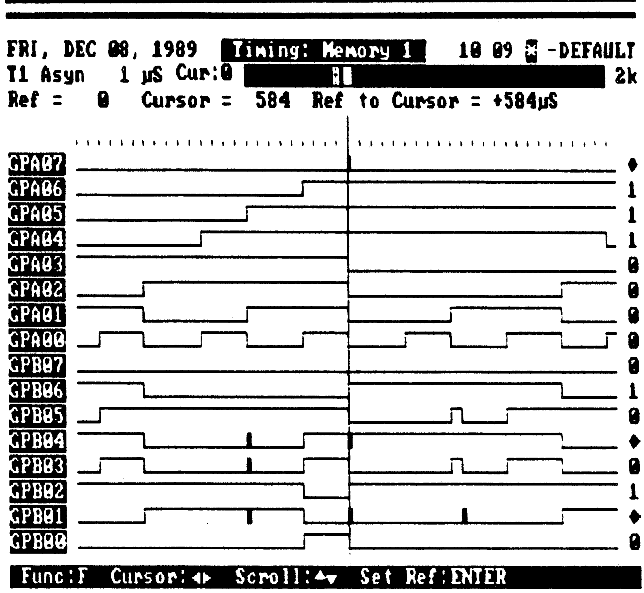


Figure 4-10. Searching in state and timing diagrams. First call up the state table, search for the event (BUG 4), and then call up the timing diagram. The Timing menu shows the search event in the center of the screen. The binary read-out for the timing traces includes the glitch definitions.

Edit Trace Names

Each trace in the timing diagram has a name. The default name is the 3-character group name from the channel group menu plus the 2-digit number of the channel in the channel group. For example, if group GPA has 32 channels (A15-A00 and B15-B00), the default trace names are GPA31-GPA00. If group DAT has 8 channels (C15-C08), the default trace names are DAT07-DAT00.

You can rename traces with any 5-character names you want. Once you start editing or reordering traces, you can't use other timing display menu functions until you leave edit mode by pressing ENTER. Figures 4-10 and 4-11 show edited trace names. Follow these steps to name a trace.

1. Press E to edit traces. The cursor highlights the first trace name in the list.
2. Move the cursor to the trace you want to change and press E again to edit the trace name.
3. Enter the new trace name. Enter 0-F directly from the keypad. Use ▲ and ▼ to enter other characters. Press DON'T CARE to enter a space. The cursor automatically moves to the next digit when you enter a character, but you can also use ◀ and ▶ to move to other characters.
4. Press ENTER to leave edit mode for that trace name. When you enter the last (fifth) digit of the name, the cursor automatically scrolls out of the field and leaves the editor, saves the change, and highlights the whole name.
5. Press ENTER to leave edit mode and return to the timing display's menu functions.

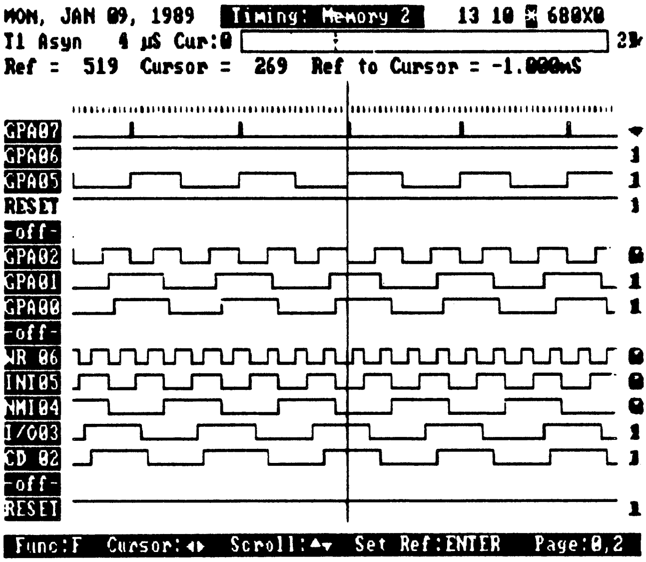


Figure 4-11. Edited and reordered traces. You can reorder, duplicate, and mask traces. In this example, the RESET line is displayed twice in this menu. Three other traces are turned off to create blank spaces and make the display easy to read. Several traces are renamed to reflect SUT signal lines.

When you edit a trace name, you are not changing the trace itself. For example, page 1 displays 16 traces (A15-A00). If you change name GPA00 to GPA15, trace GPA15 is not displayed in that position. Instead, the new name GPA15 is a user-defined label, and you will have two different traces named the same thing (GPA15). To change the timing trace instead of the trace name, reorder, duplicate, or mask the traces as explained next.

Edit Traces

The analyzer displays up to 16 channels on the screen at a time and up to four pages of traces depending on how many probes you have connected. By default, traces are displayed as they are listed in the Channel Grouping menu. For example, if group ADD lists channels A15-A00, the timing diagram lists the traces for those channels in the same order: ADD15-ADD00.

You can change the order of the traces, duplicate the traces, or turn them off so they don't show in the display. Figure 4-11, earlier in this section, shows reordered, duplicated, and masked traces. The only restriction to changing traces around is that traces can be displayed on the same page only if they were acquired with the same timebase. Follow these steps to reorder, duplicate, and remove traces from the display.

1. Press E to edit the traces.
2. Press D to change the order of traces.
3. Use these keys to reorder, duplicate, and delete traces:
   - 0,2 Cycle through traces at the current cursor position. This is how you display a specific trace in a specific position. For example, if the cursor is on GPA04 and you press 0, the analyzer displays GPA03 in that position. For 16 channels, if the cursor is on GPA15 and you press 2, the analyzer displays GPA00 in that position.
   - 1 Display the next (subsequent) trace in the next position. For example, if the cursor is on trace GPA09, pressing 1 will display trace GPA08 in the position immediately below the cursor.
   - 3 Toggle the current trace on or off.
4. Press ENTER to save your changes and return to Timing menu functions.

How Channel Groups Affect Traces

Changes to the order or names of timing traces don't affect channels in the Channel Grouping menu. However, because timing traces reflect the channel groups, changes to the Channel Grouping menu can affect timing trace names and the order traces are displayed.

The traces are named by default according to the 3-character channel group name and the 2-digit channel position in the group. If you change a channel group's name in the Channel Grouping menu, the Timing menu is automatically updated to reflect the new name. If you change a channel's position in the channel group, the Timing menu is restored to the default (Channel Grouping) order to reflect this change also. This is explained in detail in Section 3 under *Channel Grouping Menu.*

Section 5
UTILITY MENUS

Utility menus let you access various standard and optional analyzer features. Table 5-1 lists the utility menus.

Table 5-1
UTILITY MENUS

Utility Menu	Description
Storage	Stores menu settings (setups) you define. This menu also lets you name and load stored setups so you can display data acquired using previous setups.
System Settings	Sets the date, time, screen-intensity, screen-saver, and power-on diagnostics features.
Printer Port	Optional. Specifies how to use the parallel printer port, including identifying the range of memory to print, the print density, and other details.
RS-232C	Optional. Sets the baud rate, word length, parity, and stop bits for the optional RS-232C communication protocol. See the *RS-232C Operator's Manual* for details.
GPIB	Optional. Sets up the General Purpose Interface Bus interface. Refer to the 1230 GPIB Operator's Manual for GPIB information.
International Notes	Optional. Selects the language of the notes information (optional).

STORAGE MENU

The Storage menu lets you name, save, protect, and restore (load) up to eight setup configurations. A setup configuration is a set of parameters the analyzer uses to acquire data. This discussion covers these topics:

- Setups and probe links
- Saving setups

- Loading setups
- Protecting setups
- Setups after powering up
- Restoring the default setup

For each setup listed in the Storage menu, the menu shows you the setup name, the date and time the setup was stored, the setup's protection status, and the probe links used in the setup. Figure 5-1 shows an example Storage menu.

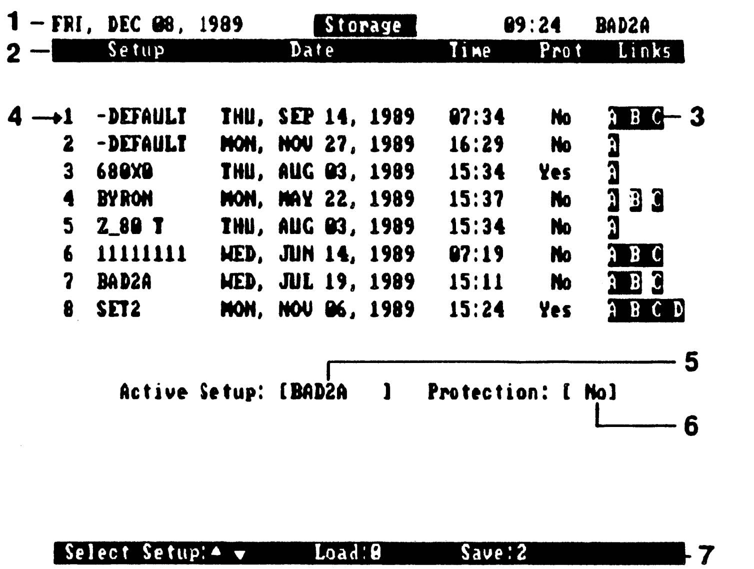


Figure 5-1. Storage menu. This menu displays the name of each setup that can be restored, the date and time that setup was created, the setup's file protection status, and the probe link configuration necessary to restore this setup.

[1] **Menu header.** This field lists the menu name, current date and time, and current setup name. An asterisk before the setup name means that you changed the current analyzer setup since it was last uploaded.

[2] **Column headings.**

[3] **Setup description.** This line gives you the name, date and time of storage, file protection status, and probe links for the stored parameters.

[4] **Selection indicator.** The > indicates the setup you want to load or save. Use the cursor keys to move the indicator through the menu.

[5] **Active setup name.** Use this field to change the name of the current (active) setup.

[6] **Setup protection status.** You can protect a setup from being accidentally overwritten. Press 0 or 2 to toggle the protection for the active setup from yes to no.

[7] **Menu bar.** The menu bar lists the keys to press to load, save, edit, and protect setups.

Setups and Probe Links

The next two discussions explain the setup parameters stored by the analyzer and describe how probe links affect the uploading of setups and displaying of stored data.

Setup Parameters

The setup parameters stored by the analyzer include information about probe links, sampling rates, channel groups, condition word definitions, and trigger statements.

For example, one setup may have 48 channels in one timebase for asynchronous data acquisition at 80 ns. The trigger statement might be an if-then-else statement that uses a counter to define the trigger condition. With the Storage menu, you can save these parameters so that you can reload them with one keystroke and acquire data again on that setup. This saves you time because you don't have to manually reset the analyzer to those parameters.

Naming Setups

Naming setups lets you keep track of the purpose of each group of setup parameters. When you first receive your analyzer from the factory, all eight setups are the same default configuration and are named DEFAULT. Follow these steps to rename the current setup.

1. Place the cursor on the Active Setup field and press ENTER.
2. Name the setup. Enter 0-F directly from the keypad, use ▲ and ▼ to enter other characters, and press DON'T CARE to enter a space.
3. Press ENTER to save the new setup name.

Saving Setups

You can save the current group of setup parameters with the Storage menu. This lets you reload those parameters quickly and easily so you don't have to manually change the analyzer back to a previously created (and stored) setup. To save the current configuration, follow these steps:

1. Move the > indicator to the line on which you want to save the current setup.
2. Press 2 to save the current setup parameters.
3. Press ENTER to confirm overwriting the setup currently stored on that line (for example, Z80-20NS) when you see this message:

```
            Ok to replace Z80-20NS?
    Press ENTER to confirm, MENU to abort.
```

   If the setup you want to overwrite is currently protected, the analyzer first warns you of the protection:

```
   * * * WARNING * * *  Z80-20NS   is protected!
Press setup number to overwrite, or MENU to abort.
```

   In this case, press the number of the setup space (for example, 6) in which you want to store the current configuration, then press ENTER to confirm the change when you see the prompt.

When you confirm saving the setup, the analyzer stores the current setup parameters. The Storage menu is automatically updated so that the new setup is listed on the designated line.

Loading Setups

You can load a stored setup into the analyzer's setup menus, overwriting the current menu setups. You can do this as long as the probe links of the current setup are the same as the one you want to upload. Follow these steps to load a stored setup into the analyzer's current memory:

1. Move the cursor to the setup you want to load.
2. Press 0 to load the specified setup. Because this overwrites the current setup, the analyzer asks for confirmation. In the following message, SETUP3 is the name of the current analyzer menu setup.

```
OK to replace SETUP3?
Press ENTER to confirm, MENU to abort
```

3. Press ENTER to load the specified setup and overwrite the current one. If you press MENU, the analyzer ignores the load command.

Protecting Setups

You can protect a setup so that you can't accidentally overwrite it. You can also clear a setup's protection so that you can store a different setup in that space instead. To protect a setup or clear a protection, follow these steps:

1. Load the setup you want to protect/overwrite.
2. Move the cursor to the Protection field.
3. Press 0 (or 2) to turn protection on or off. The menu then lists the new protection status in the protection column for the setup.

Restoring the Default Setup

When you power up the analyzer, the setup is the same as when the analyzer was last powered down. However, while the Initialization menu is still displayed, you can restore the default analyzer setup by pressing D. The analyzer then changes the setup menus to the factory default for your configuration.

The default analyzer setup links all available probes in one timebase at an asynchronous sampling rate of 1 μsec. Appendix B, *Default Menus*, shows the default setups for each setup menu. Follow these steps to upload (from ROM) the default analyzer setup:

1. Press NOTES and ENTER at the same time until the analyzer resets.
2. Press D to restore the default setup and overwrite the existing setup (for example, *-68K@40NS).
3. Check the header line of the menu. The asterisk (if there was one) should disappear from the setup name, and the name should now say -DEFAULT. If the asterisk is still displayed, the default setup is not loaded, and you should repeat the steps listed here.

SYSTEM SETTINGS MENU

The System Settings menu lets you set these system features:

1. Date and time for the nonvolatile clock and calendar.
2. Whether or not you want to run the power-on diagnostics.
3. Intensity (brightness) for screen displays.
4. Whether or not you want to enable the screen-saver, a feature that prevents screen burn-in by blanking the screen after about 18 minutes without a keystroke.

Figure 5-2 shows a System Settings menu.

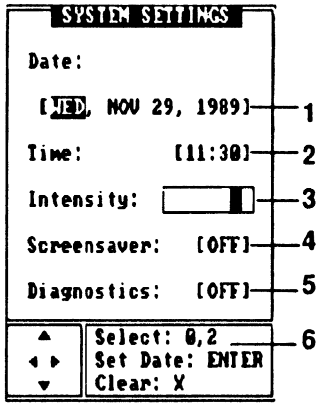


Figure 5-2. System Settings menu.

[1] **Date.** Press 0 to 2 to cycle through the selections for the month, day, and year fields. Pressing DON'T CARE sets the highlighted field back to the default. Pressing ENTER sets the new date so that other analyzer menus reflect your changes.

[2] **Time.** Press 0 or 2 to cycle through the selections for the hours and minutes fields. Pressing DON'T CARE sets the highlighted field back to the default (00). Pressing ENTER sets the new time so that other analyzer menus reflect your changes.

[3] **Screen intensity.** Press 0 or 2 to move the intensity indicator left or right so that the screen dims or brightens.

[4] **Screen-saver.** Press 0 or 2 to toggle the screen-saver feature on or off. This feature helps prevent screen burn-in by blanking the screen after about 18 minutes of inactivity.

[5] **Diagnostics.** Press 0 or 2 to toggle the power-on diagnostics on or off. If this feature is on, the 1230 will run power-on diagnostics before it displays the initialization screen.

[6] **Menu bar.** This bar tells you the keys to press to use this menu's functions.

Setting the Date and Time

In the date fields, you can set the day of the week, month, date, and year. In the time fields, you can set the minutes and hours. To change any field, follow these steps:

1. Move the cursor to the date or time field you want to change and press 0 or 2 to cycle through the field's selections. You can also press DON'T CARE to set the field back to its default value (sometimes it's faster to set the date from the defaults).
2. Before moving the cursor from the field, press ENTER to store that field's change. If you don't press ENTER, the analyzer won't store the new values, and the date/time line at the top of other analyzer screens won't be updated to reflect your changes.

Setting Screen Intensity and Screen Saver

The screen intensity lets you change the brightness of the screen. When the cursor is on this field, press 0 or 2 to lessen or brighten the screen intensity. The box scale shows how bright the screen is compared to the analyzer's capabilities.

The screen-saver feature blanks the screen after about 18 minutes of inactivity. This helps save the CRT phosphors and prevent screen burn-in. Press 0 or 2 to toggle the screen saver on or off. If the screen is blanked because of the screen saver, press any key to bring back the current display.

PRINTER MENU

The parallel printer option lets you customize your interface to an output device. If this option is installed, the Main menu lists the printer as a menu option. You can then print data acquisitions and screens on a dot matrix printer such as an Epson or other parallel, Centronix-interface, dot-matrix printer that is completely compatible with the Epson graphics character set and interface specifications. Figure 5-3 shows the default Printer Port menu.

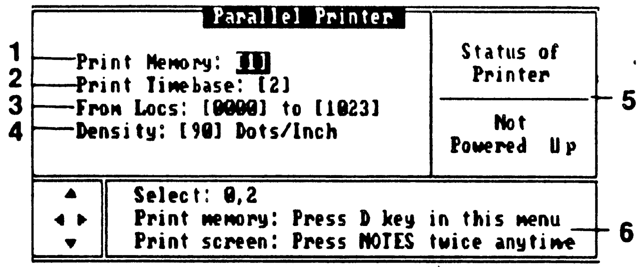


Figure 5-3. **Printer Port menu.** Use this menu to specify which analyzer memory you want to print, the range of memory to print, and the print density.

[1] **Memory specification.** Identifies one of the four analyzer memories. Press 0 or 2 to select the memory from which you want to print the data.

[2] **Timebase specification.** Identifies the timebase from which you want to print acquired data. For example, if the analyzer is set up to acquire in two timebases, you can choose to print the data acquired in T1 or T2.

[3] **Memory range.** Lets you specify a range of memory to print out. Press ENTER, then enter the memory range directly with numeric keys. The lower (left) range is 0000 to 2046; the upper (right) range is 0001 to 2047.

[4] **Printout density.** Lets you choose 60, 80, or 90 dots per inch. A density of 90 gives the printout with the best resolution; a density of 60 gives the printout with the lowest resolution. Press 0 or 2 to cycle through the selections.

[5] **Printer status.** Tells you if the printer is connected and ready to print, busy or off-line, or not powered up.

[6] **Menu bar.** These lines tell you which keys to press to change the menu fields, print the specified memory, or print the current analyzer screen. If the cursor is on the memory range field, this bar tells you how to change the specified memory range.

You can print the screen at any time by pressing NOTES twice, quickly. If you wait too long before pressing NOTES the second time, the online help screen for the current menu will appear. In this case, press MENU to get out of the online help and then press NOTES again twice.

Pressing D while in the Printer menu prints the currently specified memory (or range of memory). The printout is in state-table format, with trigger event occurrences and memory differences in bold. You can print timing traces if you call up the Timing menu and press NOTES twice to print the screen.

INTERNATIONAL NOTES MENU

The analyzer features optional online help in English, Danish, Dutch, German, Finnish, French, French Canadian, Italian, Norwegian, Spanish, and Swedish. If you install international notes, the Main menu lists notes as an option.

To change the current language selection, call up the International Notes menu and press 0 or 2 till the language you want is displayed. (If you press NOTES, the analyzer displays a screen telling you which languages you can choose.) After that, online help is displayed in the language you specify. Figure 5-4 shows an International Notes menu. Figure 5-5 shows the online help for the International notes option. Figure 5-6 shows a page of notes in Dutch.

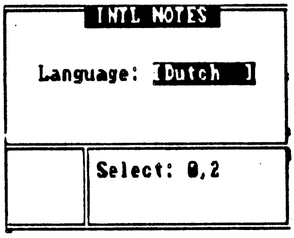


Figure 5-4. **International Notes menu. Press 0 or 2 to cycle through the available languages. When you leave the menu, the analyzer saves your change. After that, the notes information is displayed in the language you specified.**

International Notes Version 1.01

Select "English" for English system notes

Wähle "German" für deutsche System Mitteilungen

Seleccione "Spanish" para obtener las notas de
referencia del sistema en español

Kies "Dutch" voor Nederlandse help mededelingen

Selectionnez "French" pour le système de notes Français

Selezionare "Italian" per le Note di Sistema in
Italiano

Choisissez "Fre Can" pour le système de notes
Canadiennes françaises

Exit Notes: Hit any key

Figure 5-5. International Notes online help page. If you press NOTES while in the International Notes menu, the analyzer displays a help screen telling you which languages you can choose (Italian is not supported for this product release).

Timebase / Tijdbasis　　　　　　　　　　　　　Pag. 0

Gebruik het "Tijdbasis Menu" om aan te geven
hoe de data dient te worden gemeten door het instrument
(synchroon of asynchroon).
"Linked (gecombineerde) Probes" zijn probes welke
dezelfde tijdbasis gebruiken. Een tijdbasis kan worden
toegevoegd of verwijderd door "links" te veranderen.
Door het combineren van tijdbasissen worden bepaalde
instellingen in andere menu's veranderd. Sla de instru-
ment instelling altijd op in het geheugen alvorens
de tijdbases te veranderen.

Async: maakt gebruik van de interne analyzer klok.
10 ns instelling, data wordt gemeten op kanaal 0 tot 3
kanaal 0 tot 7 voor 20 nS, alle andere instelling
meten op alle kanalen. Max. 2 async klokken beschikbaar
Sync: maakt gebruik van een externe klok. Elke probe
die voor een synchrone meting wordt gebruikt dient te
worden aangesloten op het externe klok circuit.
Glitch detectie: gebruikt een tijdbasis instelling van
minstens 40 nS asynchroon. Kanalen 0 tot 7 kunnen
worden gebruikt om data en glitches te analyseren.

volgende pagina:▲　vorige pagina:▼　beëindig:MENU

Figure 5-6. **Notes in Dutch.** When the International Notes option is installed, you can choose the language in which the notes are displayed. For example, this figure shows the notes screen for the Timebase menu in Dutch.

Appendix A
LOGIC ANALYZER CONCEPTS

Logic analyzers are designed to check out digital hardware and software. Before logic analyzers were developed, the only instrument that allowed designers and technicians to monitor digital operation was the oscilloscope.

LOGIC ANALYZERS VS. OSCILLOSCOPES

Most oscilloscopes display only two to four channels. However, in today's digital applications, you often need to look at up to 48 or more channels at the same time. The strengths of logic analyzers are that they can:

- Sample many signals at once
- Sample non-repetitive signals
- Display information in different formats
- Trigger on a particular combination of signals
- Show timing relationships of many signals at once
- Capture data that occurs before a predefined trigger
- Collect and store data selectively

When would you still need to use an oscilloscope? Use an oscilloscope when you want to display signal amplitude vs. time in fine detail. Some scopes can measure the time interval between events with sub-nanosecond accuracy. Therefore, scopes are most useful when you need a small (1-4 channels), detailed picture for parametric measurements, such as signal rise or fall times, amplitude, and pulse width.

SAMPLING DATA

A logic analyzer samples the input waveforms to determine whether they are high or low in relation to a threshold voltage (see Figure A-1). If the signal is above the threshold when it is sampled, the logic analyzer stores it as a 1 or high. Similarly, if the signal is below the threshold, it is stored as a 0 or low.

Exactly when each high or low is sampled and stored into memory is determined by the sample clock (see Figure A-2). This clock can be either supplied internally by the logic analyzer or externally by the system under test (SUT). Another word for sample clock is timebase.

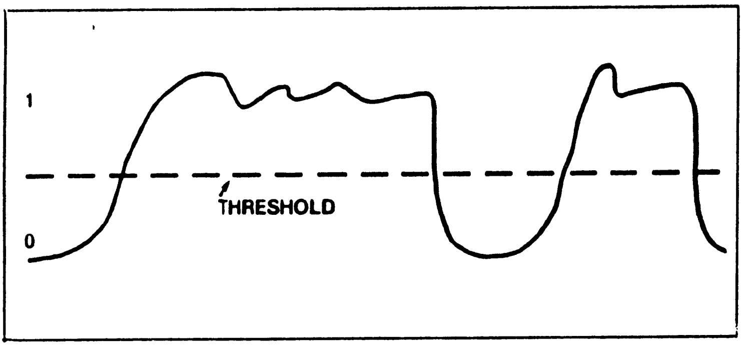


Figure A-1. Threshold voltage. Logic analyzers digitize incoming signals by comparing the signal voltage to a reference threshold voltage and storing the result (1 or 0) in memory.

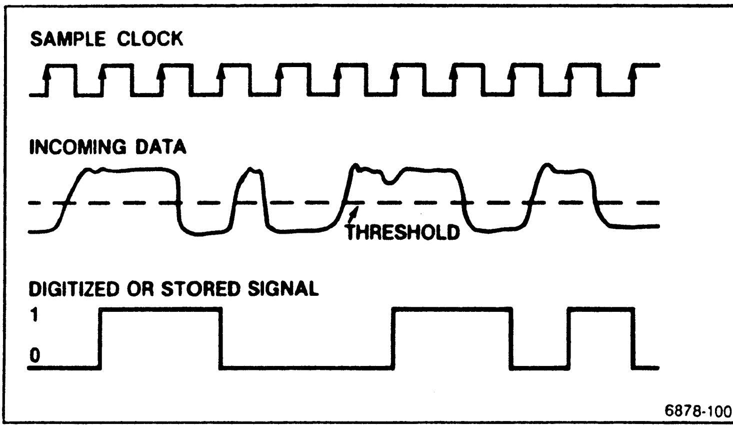


Figure A-2. Sample clock. The sample clock determines when the analyzer looks at the incoming signals. On the rising edge of the sample clock, the analyzer compares the amplitude of the incoming data to the threshold. If the signal is above the threshold when the sample clock occurs, the analyzer stores a 1; if the signal is below the threshold, the analyzer stores a 0.

Asynchronous Clock

If the sample clock is generated by the logic analyzer, it is called an internal or asynchronous clock. You can choose the asynchronous clock rate by choosing one of the rate selections on your logic analyzer.

In general, the asynchronous sampling rate (clock) should be at least twice as fast as the fastest frequency of data being sampled. This makes sure that every high and low transition is captured. For example, in the top half of Figure A-3, the sampling rate is not fast enough to detect the narrow pulse. The faster clock rate in the bottom half of the figure captures the narrow pulse; it also shows a more accurate representation of the incoming data waveform.

You get the best signal resolution when the sample clock is ten or more times faster than the data rate.

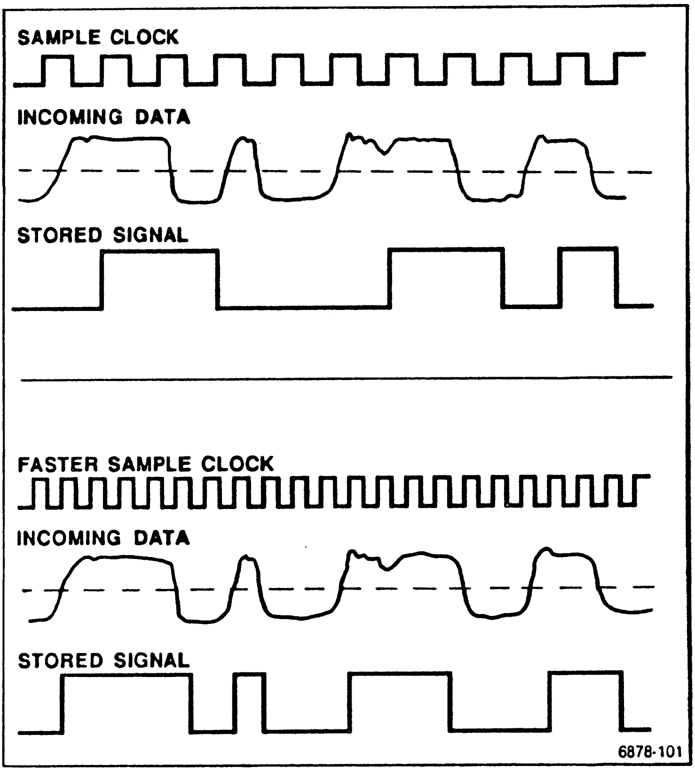


Figure A-3. Sample clock rates. The faster sample clock captures the narrow pulse missed by the slower sample clock (top). To make sure all data transitions are captured, use an asynchronous clock rate at least twice as fast as the fastest data being sampled.

Synchronous Clock

In many cases, data is meaningful to the system under test (SUT) only at specific points in time. For example, data might be meaningful only when:

- A processor bus has a valid address or instruction

- A counter outputs its next value
- A character is received from a control panel

In these cases, it's easier to analyze meaningful data if you use a signal from your SUT to control the data sampling rate. When your SUT supplies the sample clock, the clock is called an external or synchronous clock. Synchronous analysis tells you what state your SUT was in for each clock pulse at which data was sampled. Figure A-4 shows a synchronous clock cycle.

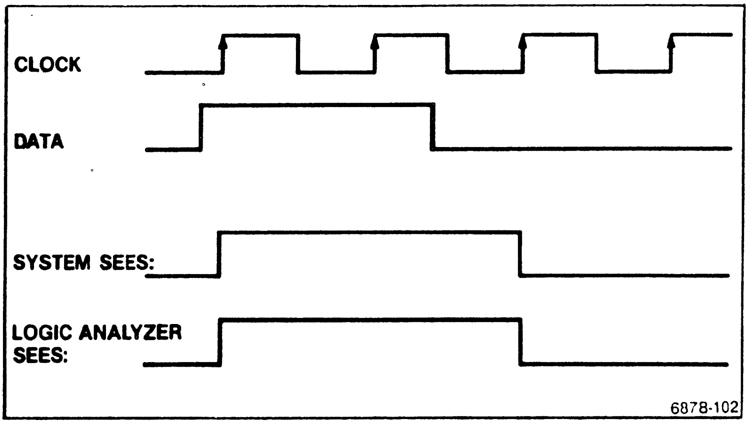


Figure A-4. Synchronous sampling. When the sample clock is generated by the SUT, the clock is called a synchronous clock.

DISPLAY FORMATS

A major benefit of logic analyzers is their ability to display acquired data in a format appropriate for your application. This makes it easier for you to read and analyze the information acquired from your SUT. The three common display formats are state tables, disassembly mnemonics, and timing diagrams. Section 4 explains the features of these 1230 Logic Analyzer displays in detail.

State Tables

State tables display data in a list format (Figure A-5). Each horizontal row represents one sample; each vertical column represents one channel. State tables are useful when you want to see the word flow, or data sequencing. While timing diagrams are most often associated with hardware troubleshooting, state tables lend themselves more readily to software debugging.

Most analyzers can display state data in different radixes. For the 1230 Logic Analyzer, a radix is the alphanumeric format in which a value is displayed; for example, binary or ASCII. Being able to choose the radix for data displays makes it easier for you to relate acquired data to other documentation (data books, listings, and so on) about your SUT. Figure A-5 shows a state table using different display radixes.

Most analyzers can store more data than a state table can display at one time. For example, some analyzers, like the 1230, can store up to 2K bytes of data and display up to 20 lines of data for each channel on the screen. You can move through the acquired data backwards, forwards, or in jumps by scrolling.

```
FRI, JAN 06, 1989    State: Memory 3     13:03 ▣ -DEFAULT
 Loc    ADD  DAT                 CTL  FFM
        hex  bin                 asc  oct
 0118   0073 1111110011111110 ( SB 006000
 0119   0073 1111110011111110 ( SB 026400
 0120   0073 1111110011111110 ( SB 026400
 0121   0073 1111110011111110 ( Z  026400
 0122   0073 1111110011111110 ( Z  026400
 0123   0073 0110101100001111 ) Z  026400
 0124   0073 0110101100001111 ) Z  037000
 0125   0073 0110101100001111 ) Z  037000
 0126   0073 0010101100001111 = Z  037000
 0127   0073 0010101100001111 = Z  037000
-TRIG---00F6-0010101110001111-= Z -077400-----------------
 0129   00F6 0010101110001111 = Z  077400
 0130   00F6 0010101110001111 = Z  077400
 0131   00F6 0100101110001111 e :  077400
 0132   00F6 0100101110001111 e :  077400
 0133   005F 0100101101001111 e :  170000
 0134   005F 0100101101001111 e :  170000
 0135   005F 0100101101001111 e :  170000
 0136   005F 0100101101001111 e :  170000
 0137   005F 0100101101001111 e :  170400
 Func:F     Search for: 0,2 [Trigger  ]   Do Search: 1
```

Figure A-5. State table. In a state table, each horizontal row represents one data sample - what the channels were doing at that point in time. For displays in binary radix, each vertical column represents the data from one channel recorded at successive points in time. If you're displaying data in radixes other than binary, each row represents several channels. For example, in hexadecimal radix, each displayed digit represents four channels.

Disassembly Mnemonics

In many logic analyzer applications, you need to monitor microprocessor address, data, and control buses. Logic analyzers do this by analyzing the microprocessor bus cycles and then decoding those logic states into microprocessor mnemonics. For example, a data value of 6600 may be decoded to mean BNE (branch if not equal).

By analyzing and decoding bus cycles, a logic analyzer can correlate addresses and data associated with each logic state. The analyzer can then display that information in a disassembly format, as shown in Figure A-6. These mnemonic-coded displays let you analyze state machine problems without having to manually translate displayed binary or hexadecimal data. Being able to disassemble microprocessor instructions can shorten the time you spend debugging your system.

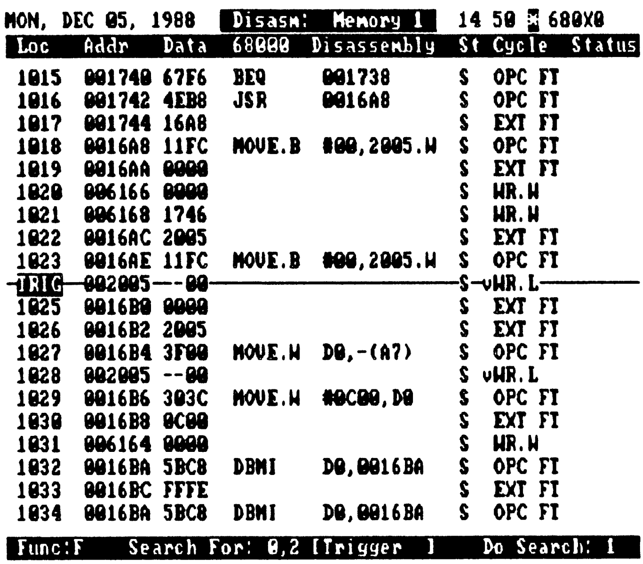


Figure A-6. Disassembly mnemonics. Mnemonic-decoded displays help reduce the time you spend debugging your SUT. To do disassembly, you usually need a microprocessor-specific data probe and microprocessor disassembler (a reverse assembler) software.

Timing Diagram

Timing diagrams (as shown in Figure A-7) are pseudo-waveform displays that represent the highs and lows of data states as they change, rather than actual voltages across the line. Figure A-2, earlier in this section, shows a timing diagram's digitized signal (a trace) and the corresponding actual incoming signal.

For each timing diagram, the vertical axis lists the channels on which you acquired (or tried to acquire) data. The horizontal axis lists the samples taken across a period of time. You can decrease or increase the display resolution to show more or less samples on the screen. This lets you show both very detailed information about the logic states, and also see an overview of all the acquired data in memory. Figure A-7 shows a high resolution (fine detail) for the acquired data.

Because you can see the logic states across each signal line, or channel, timing diagrams are useful for hardware troubleshooting. They let you look at the timing relationships between signals and check propagation delay times.

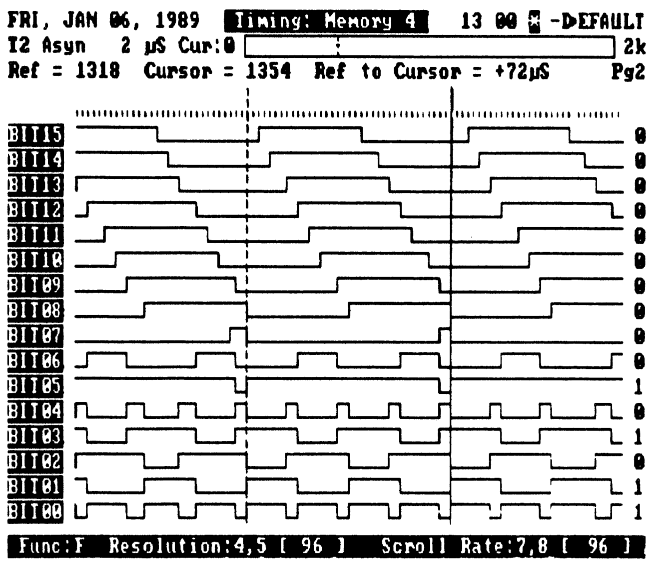


Figure A-7. Timing diagram. A timing diagram is a pseudo-waveform display. You can use timing diagrams to look at timing relationships between signals. In these diagrams, the horizontal axis shows samples and the vertical axis shows data channels.

GLITCH DETECTION

When using an analyzer's internal clock, the rate you select for sampling data should be faster than the rate of the data being sampled. Sometimes, activity occurs in such fast pulses that even the fastest sampling rate is not fast enough.

When activity occurs so much faster than the sampling rate, the pulses are generally many times narrower than any other typical pulse in the data. These extremely fast pulses are called glitches. A glitch is defined as two or more transitions on a signal occurring between two adjacent sampling points.

Glitches can be caused by sampling rates that are too slow, capacitive coupling between circuit board traces, by power supply ripples, by high instantaneous current demands of several devices at the same time, inadvertent interrupts, and many other events.

You can often ignore glitches. However, when you're working with sequential logic, such as flip-flops and counters, glitches can signify a problem. Logic analyzers that can capture and display glitches can reduce debugging time for digital troubleshooters.

Figure A-8 shows how glitches are displayed by the 1230 Logic Analyzer. In the timing diagram, glitches are shown as intensified marks.

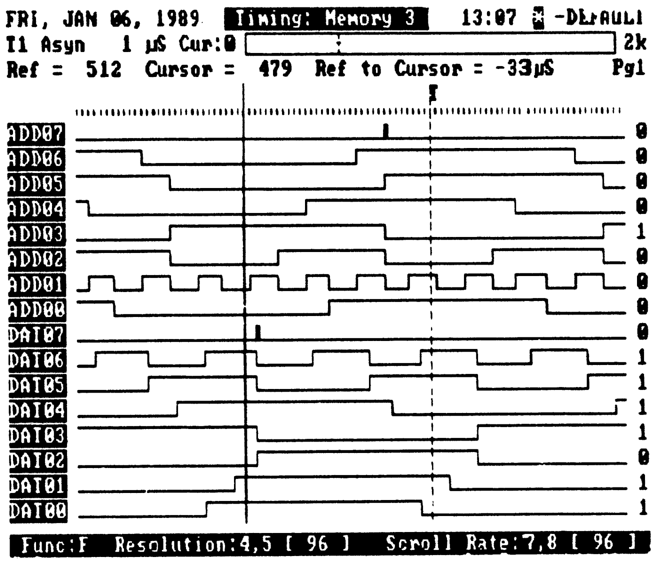


Figure A-8. Glitches. Glitches are displayed as intensified marks in the timing diagram. A glitch is two or more transitions of a signal occurring between two adjacent sample clocks.

TRIGGERING

In a storage oscilloscope, the trigger event starts the sweep and, therefore, the acquisition and storage of data. In a logic analyzer, the trigger event stops data acquisition, either immediately or after a specific number of clock cycles. This means that a logic analyzer can display data before and after the trigger event. Because of this, the trigger event gives you a reference point in the acquired data.

Trigger Events and Word Recognizers

The trigger event is detected by the logic analyzer's word recognizer circuits. Word recognizers let you specify the type of trigger event on which you want data acquisition to stop. For example, you could specify the rising or falling edge of a signal line (a single input). You could also specify a combination of highs and lows across parallel channels. Most logic analyzers provide trigger word recognizers with widths equal to the number of available data channels.

Figure A-9 shows a logic diagram of a word recognizer. A word recognizer includes a set of programmable AND gates. Each gate has one input for a data channel and one input for a programmable word recognizer value.

Each acquired data sample is compared by the word recognizer to the trigger word. When a match is found, the logic analyzer stops data acquisition and displays the results.

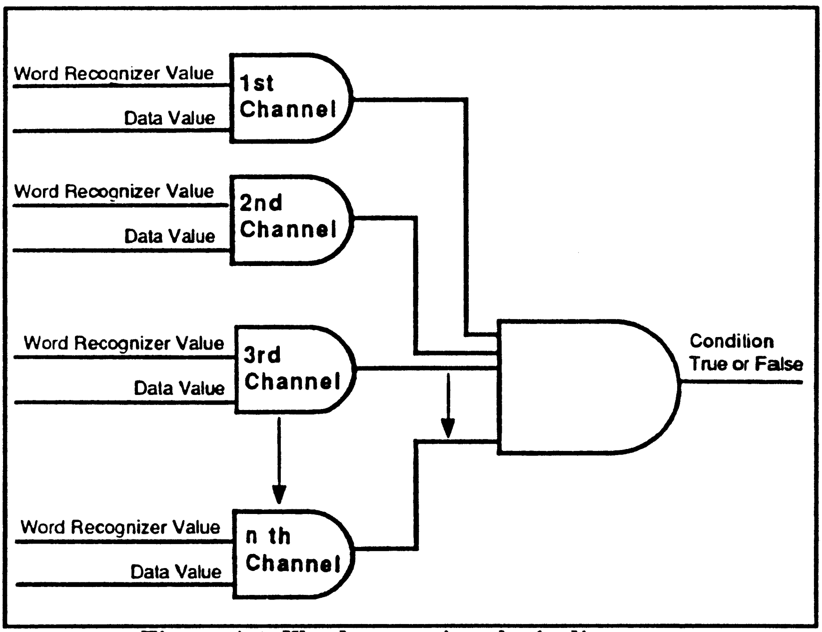


Figure A-9. Word recognizer logic diagram.

Trigger Position

Most logic analyzers let you control how much data is stored before and after the trigger event. In this way, you can specify the position of the actual trigger event in the logic analyzer's memory, which contains the acquired data. If you position the trigger toward the end of the sample memory, you can look ability to display negative time is one of the main advantages of using a logic analyzer over an oscilloscope.

The amount of data before the trigger is called pre-trigger data and the amount after the trigger is called post-trigger data. The total of the two is equal to the sample memory depth, usually about 2K words.

ACQUISITION AND REFERENCE MEMORIES

Most logic analyzers have two memories: acquisition memory and reference memory. Acquisition memory contains the most recently acquired data. As you acquire data from your SUT, that information is immediately placed in the acquisition memory. Reference memory contains data acquired some time in the past. You can store the data from different acquisitions in reference memories.

Comparing acquisition memory to reference memory gives you another evaluation tool for your SUT. For example, if you compare a reference memory from a known-good circuit to other circuits, you can see if the other circuits are also good by seeing if the data stored in the memories matches. This approach is especially useful in manufacturing test applications.

Figure A-10 shows a state table display of the results of an acquisition-reference memory comparison. Patterns that do not match are displayed in reverse video.

```
FRI, JAN 06, 1989    State: Memory 4     13:18 ▪ -DEFAULT
Loc    ADD  DAT               CTL  FFM
       hex  bin               asc  oct
0508   6923 0111101100111110  u ;  037160
0509   6927 0111101100111110  u ;  037160
0510   6927 0001001111010000  u ∆  077160
0511   6923 0001011111010000  u ∆  077646
-0512--3B23-0001011111010000--FS}  -077646--------------
0513   3B23 0001011111010000  FS}  070140
0514   3FA7 0001011111010000  FF}  070144
0515   3DA7 0001011111010000  + FF 070144
0516   30EB 0001011111010000  ; FF 070544
0517   30EB 0101011111010001  ; ET 070544
0518   30EE 0101011111010001  ; ET 070544
0519   30EE 0101011111010001  ; D  071144
0520   30EE 0101011110011101  ; H  071144
0521   30E8 0101011110011101  ; H  071144
0522   32E8 0101011110011101  > N  071744
0523   67AC 0011011111010101  . N  071744
0524   65AC 0011011111010101  e -  071744
0525   4930 0011011111010101  u -  072160
0526   4930 0010011111010101  u -  072160
0527   4930 0010011111010101  u $  072160
Func:F  Compare Mem 4-3:C  #Diff 00046  Mem:X[Display]
```

Figure A-10. Memory comparison. State table display of the results of an acquisition-reference memory comparison. Samples where the two memories are different are marked by inverse video.

Appendix B
DEFAULT MENUS

This appendix describes the default conditions for most of the analyzer menus. To upload the default setup instead of a previously stored setup, press NOTES and ENTER firmly at the same to reset the analyzer then press D for default. The header line for the menu should list the setup as -DEFAULT. If the header lists the setup as * -DEFAULT (with an asterisk), you've made changes to the default setup since it was last stored.

The menus shown in this appendix are for a 64-channel analyzer. If you're using an analyzer with less than 64 channels, the difference for your default setup is that the analyzer will have a default channel group for each group of 16 channels. This difference in the number of installed channels will be reflected in almost all analyzer menus.

The default menus shown here do not include the Disassembly or GPIB menus. The default Disassembly menus are different depending on the microprocessor probe you plug into the analyzer. Refer to your disassembly probe operator's manual for information about the Disassembly menu. Refer to the *1230 GPIB Operator's Manual* for information about the GPIB option.

WED, NOV 29, 1989 11 32 -DEFAULT

Tektronix 1230/32 Channel Logic Analyzer. V4.01
(C) Tektronix, Inc. 1988, 1989 All rights reserved.

Use the NOTES key whenever information is needed,
or consult the Operator's Manual.

X represents DON'T CARE condition.

Press MENU to continue

Figure B-1. Initialization menu.

SETUP	DATA	UTILITY
0 Timebase	6 Mem Select	B Storage
1 Channel Groups	7 State	C Sys Settings
2 Trigger Spec	8 Disassembly	
3 Conditions	9 Timing	
4 Run Control		

Select Screen: Hex Key or ▲▼◄► for cursor, then ENTER

Figure B-2. Main menu. This menu shows no options installed. If you install options, spaces in the Utility column will list the installed options.

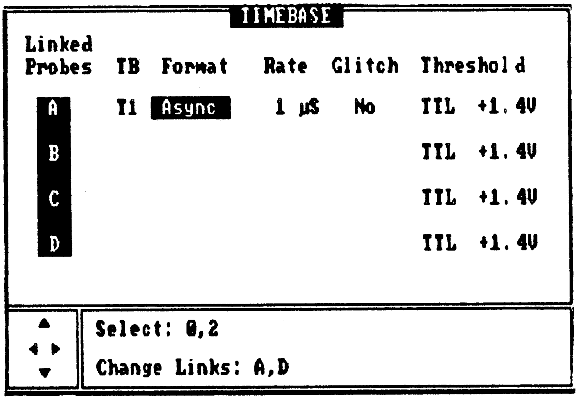


Figure B-3. Timebase menu. The default Timebase menu is set up with all probes linked. By default, the 1230 uses its own clock to acquire data (asynchronously) at a default sampling rate of 1 µs. Glitch detection is off.

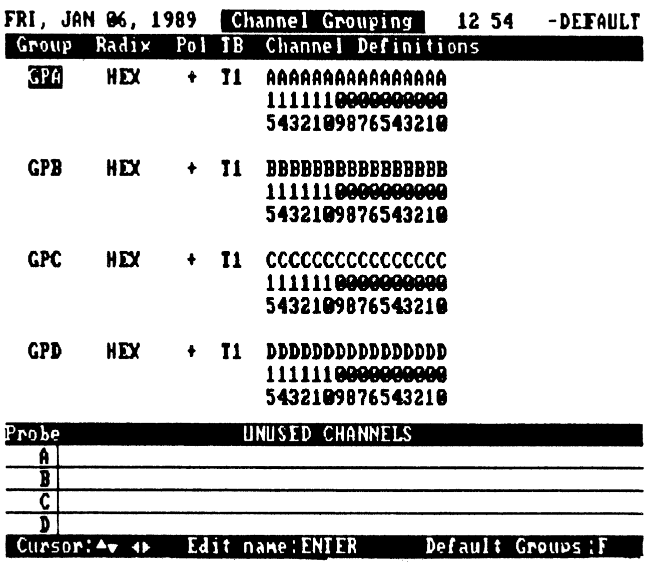


Figure B-4. Channel Grouping menu. All probes are assigned to timebase T1, with channels listed in descending order.

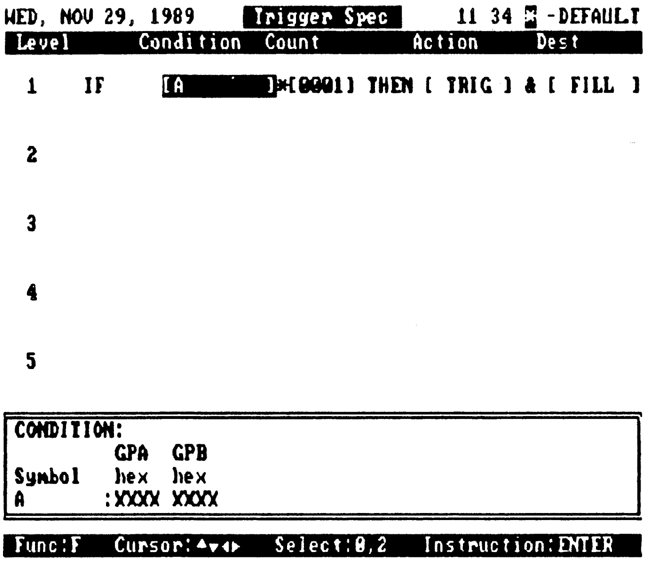


Figure B-5. **Trigger Spec menu. The default trigger statement is an if-then statement and the default trigger condition is A, defined as all don't cares.**

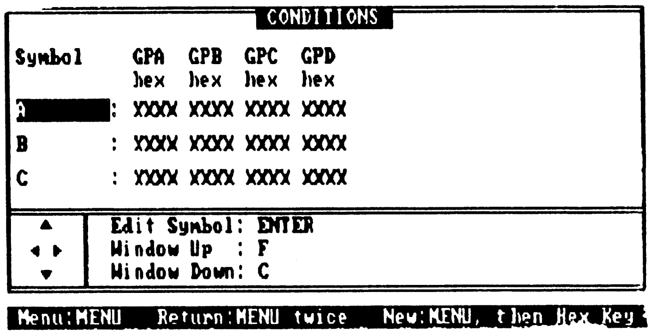


Figure B-6. Conditions menu. The default Conditions menu has all 24 conditions set to don't care values. The default names for the conditions are A, B, C, ..., W (and MASK for version 4.0 software). By default, the conditions window shows only three conditions. You can re-size the window by pressing F or C.

```
WED, NOV 29, 1989        Run Control        11:36   -DEFAULT

 Update Memory   : [1]         Display: [Timing]
 Trigger Position: [0512]       0 [     T          ] 2K
 Look for Trigger: [After Pre-Trigger Memory Full]
═══════════════════════════════════════════════════════
 Compare          : [Manual]
 Compare Memory 1 to Memory: [2]


 Compare Mem Locations: [0000] to [2047]
 Use Channel Mask     : [A      ]

 Display Data at least: [5] seconds
┌─────────────────────────────────────────────────────┐
│           GPA  GPB                                  │
│ Symbol    hex  hex                                  │
│ A       : FFFF FFFF                                 │
└─────────────────────────────────────────────────────┘
 Cursor: ▲▼◄►        Select: 0,2
```

Figure B-7. Run Control menu. In the default setup, the 1230 displays timing information when you acquire data. The default trigger position is 512. The default memory compare setting is manual.

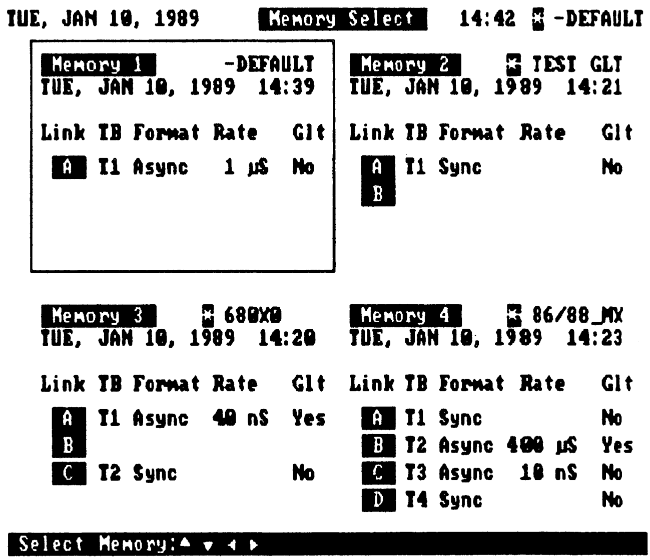


Figure B-8. **Mem Select menu.** This menu displays the main setup parameters for each of the four analyzer memories. Although this menu shows setups for several analyzer configurations, the box shows that memory 1 is selected as the display memory.

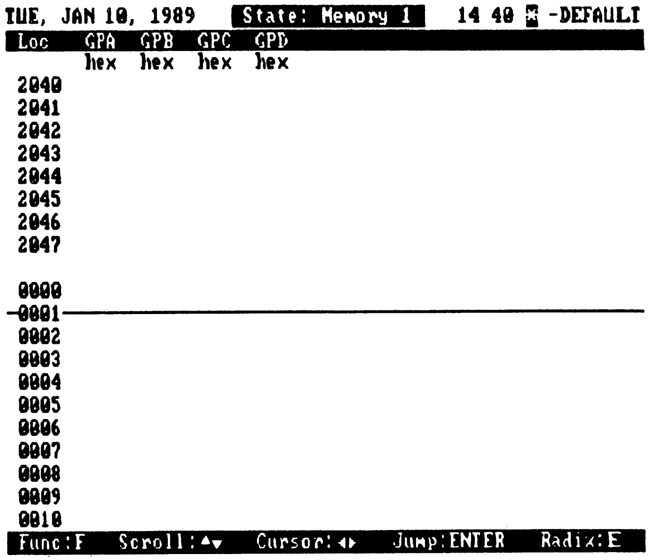


Figure B-9. **State menu. The default State menu shows no acquired data. All channel groups are displayed (none are masked from the display), and all groups are in the hexadecimal display radix.**

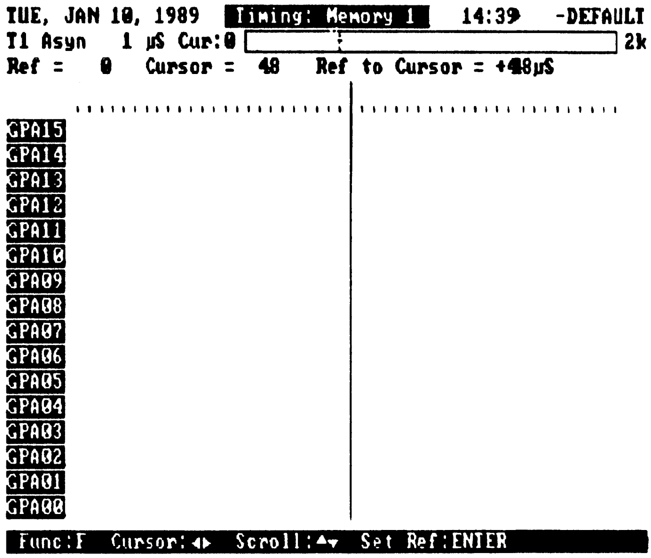


Figure B-10. Timing menu. The default Timing menu shows no acquired data. The default names for the timing traces reflect the default channel groups: GPA, GPB, GPC, and GPD, each with input channels 15-0. The default Timing menu shows the first page of timing traces. To see other pages, press 0 or 2.

TUE, JAN 10, 1989 **Storage** 14:41 -DEFAULT

	Setup	Date	Time	Prot	Links
▸1	-DEFAULT	TUE, SEP 20, 1988	15:22	No	A B
2	S_MR_1	THU, DEC 01, 1988	10:12	Yes	A B C D
3	EXT_TEST	MON, DEC 12, 1988	08:24	Yes	A B C D
4	TST GLT	FRI, DEC 02, 1988	09:52	Yes	A B C D
5	86/88_MX	THU, SEP 01, 1988	15:00	Yes	A B C D
6	TRGIO_1	MON, AUG 27, 1988	14:12	Yes	A B C D
7	680X0	TUE, DEC 06, 1988	13:11	Yes	A B C D
8	TESTGPIB	WED, DEC 07, 1988	09:12	Yes	A B C D

Active Setup: [-DEFAULT] Protection: [No]

Select Setup:▲ ▼ Load:0 Save:2

Figure B-11. Storage menu. This menu stores the user-specified menu settings (setups). This menu also lets you load stored setups so you can display data acquired with previous setups, and also acquire new data with a setup you've previously defined. When you first receive your analyzer from the factory, the default setup is stored in every space. This menu shows eight possible configurations for different acquisitions.

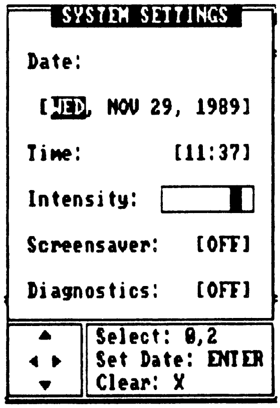


Figure B-12. System Settings menu. This pop-up menu sets the date, time, screen intensity, screen-saver features, and power-up disgnostics.

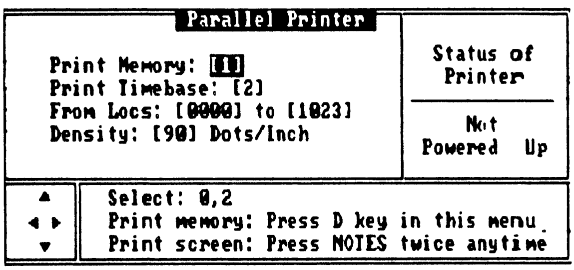


Figure B-13. Printer Port menu. This menu lets you specify the range of memory you want to print, the print density at which you want to print, and other details.

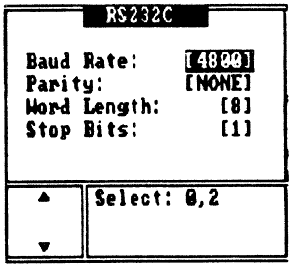


Figure B-14. RS-232C menu. This pop-up menu sets the baud rate, word length, parity, and stop bits for the optional RS-232C communication protocol

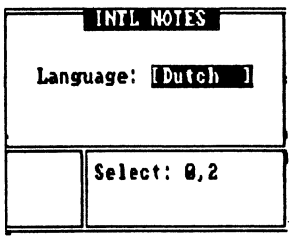


Figure B-15. International notes menu. This pop-up menu specifies the language the on-line help will appear in. The default setting is English; this menu shows that Dutch has been selected for the on-line help.

Appendix C
INSTALLATION AND SETUP

This appendix shows you how to install and set up the analyzer for your application. The installation procedure includes instructions for installing expansion cards and probes. Refer to Section 1, *Getting Started* for figures and descriptions of the analyzer front and rear panels.

These installation and setup procedures assume that you're facing the analyzer and that the analyzer is in a normal, upright position. Refer to the *1230 Logic Analyzer Service Manual* for detailed assembly and disassembly instructions.

WARNING

Turn the power off and disconnect the power source before removing the top cover. Do not try to install or remove anything from the analyzer with the instrument on or the power cord connected. Doing so may result in personal injury.

INSTALLING EXPANSION CARDS

The analyzer can have up to three 16-channel 1230E1 expansion cards installed at one time. Expansion cards are used with acquisition probes. The connection procedure for acquisition probes is later in this section.

You need a magnetic-tip #1 POZIDRIV screwdriver to install or remove boards or probes. The following steps tell you how to install expansion cards for the analyzer:

1. Make sure power to the analyzer is off and the power cord is disconnected.
2. Make sure that the analyzer is protected from static.

CAUTION

Static discharge can damage any semiconductor in this instrument. Damage to electrical components may not be immediately apparent. Take standard antistatic precautions.

3. Remove the top cover of the analyzer by removing the four Phillips-head screws on the cover, two on each side. Figure C-1 shows the analyzer's top cover and the circuit board cover.
4. Lift the cover and set it aside.
5. Refer to Figure C-1. Remove the three screws on the right side of the metal plate that covers the circuit boards, then loosen the three captive screws on the left side of the metal plate.
6. Lift off the metal plate and set it aside.
7. Firmly but gently plug the expansion card (or cards) into the appropriate card slot. Plug the first card into the slot on the right (near the outside of the analyzer), the second card in the middle slot, and the third card in the slot on the left (near the Controller board). Make sure each expansion card is well-seated.
8. Connect the cards to the probe-interconnect cables as described in the following list. Figure C-2 shows the probe-interconnect cables and cards.
   - Connect the right-expansion card to the cable connector for probe B.
   - Connect the middle-expansion card to the cable connector for probe C.
   - Connect the left-expansion card, the one closest to the Controller board, to probe D's cable connector.
   - Connect the Master Analyzer board to the cable connector for probe A (done at the factory).
9. Replace the circuit-board cover and screws.

10. Replace the analyzer's top cover and screws.

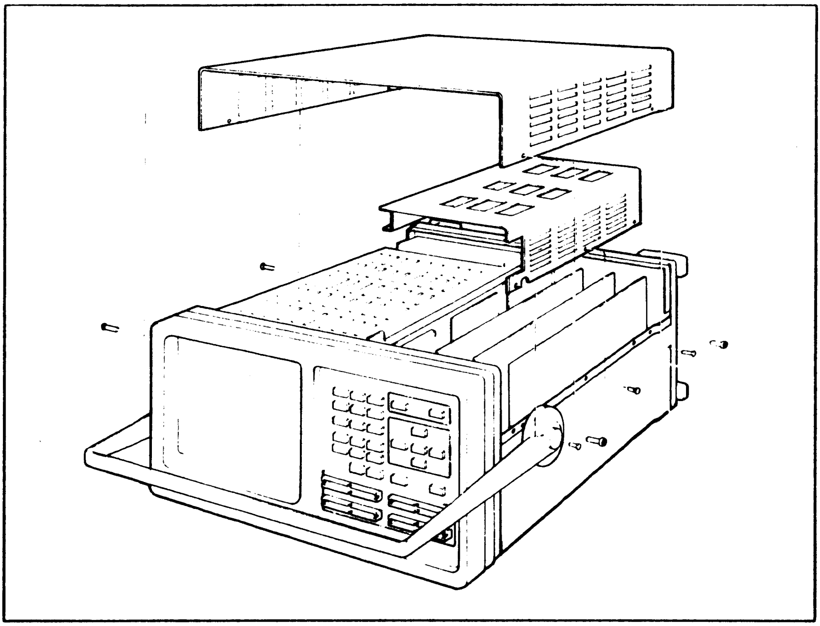


Figure C-1. Top cover and circuit board cover.

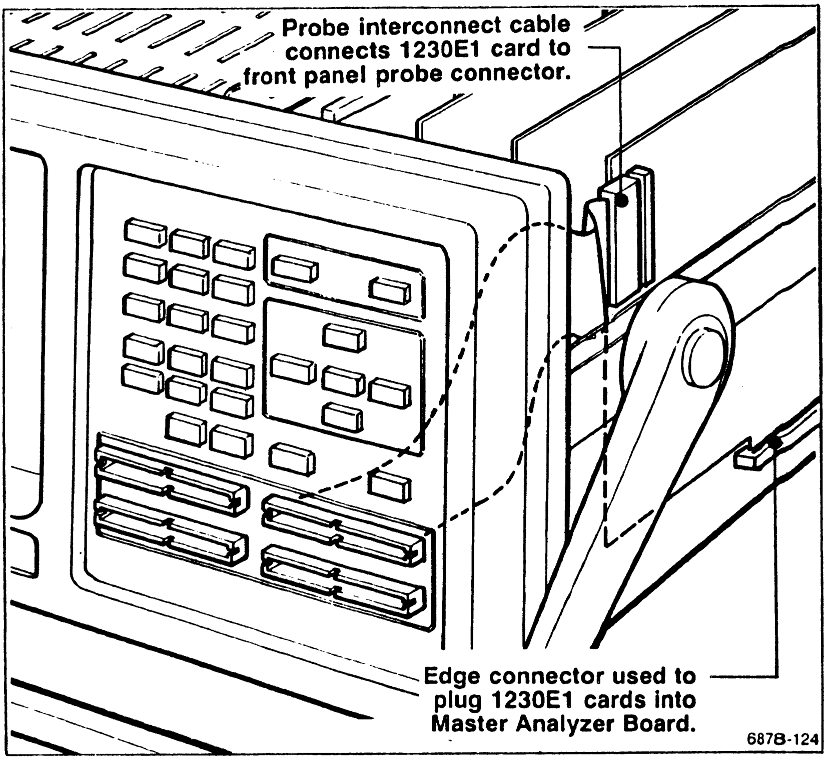


Figure C-2. Probe interconnect cables and expansion cards. Connect the probe cables to the expansion cards as shown here. Interconnect cable A should already be attached to the Master Analyzer board.

INSTALLING OPTION CARDS

Option cards, such as RS-232C and GPIB interfaces, are mounted on the Controller board. The analyzer currently supports these options:

- RS-232C protocol, which consists of a circuit board, cable, and back-panel connector

- GPIB interface, which consists of a circuit board, cable, and back-panel connector
- Parallel-printer interface, which consists of a circuit board, cable, and back panel connector
- International notes, which consists of a circuit board only

If you're installing the RS-232C or GPIB interface, install the GPIB card in the front slot, the printer card in the middle, and the RS-232C card in the rear slot. The Notes option doesn't use interrupts, so you may install this option in another slot, closer to the front panel.

Follow these steps to install an option card. (Remove a card by reversing the appropriate steps.) Figure C-3 shows the option cards.

1. Make sure the power is off and the power cord is disconnected from the analyzer.
2. Make sure the analyzer is protected from static.

CAUTION

Static discharge can damage any semiconductor in this instrument. Damage to electrical components may not be immediately apparent. Take standard anti-static precautions.

3. Take off the analyzer's covers as described in the procedure listed earlier in this appendix.
4. If an expansion card is installed in position D, close to the Controller board, disconnect the card's probe interconnect cable and remove the card. This gives you enough room in the mainframe to install or remove option cards from the Controller board.

5. Gently but firmly install the option card in one of the option slots. (Install the RS-232C or GPIB option in the slot closest to the back panel. Install other options toward the front panel.) Make sure the card is well-seated.
6. Unscrew the two Phillips-head screws from one of the blank option plates on the analyzer's back panel. Remove the screws and plate and set them aside.
7. Replace the blank plate with the new option plate that has a probe connector on it. Screw the plate down with the two Phillips-head screws you set aside in step 6.
8. Attach the option's cable to the appropriate option card.
9. Plug the option cables into the back-panel port connectors in this order:
   a. The back option card connects to the left option cable.
   b. The middle card connects to the middle option cable.
   c. The front card connects to the right option cable.
10. Replace the 16-channel expansion card in position D and the probe interconnect cable.
11. Replace the circuit board cover.
12. Replace the analyzer's top cover.

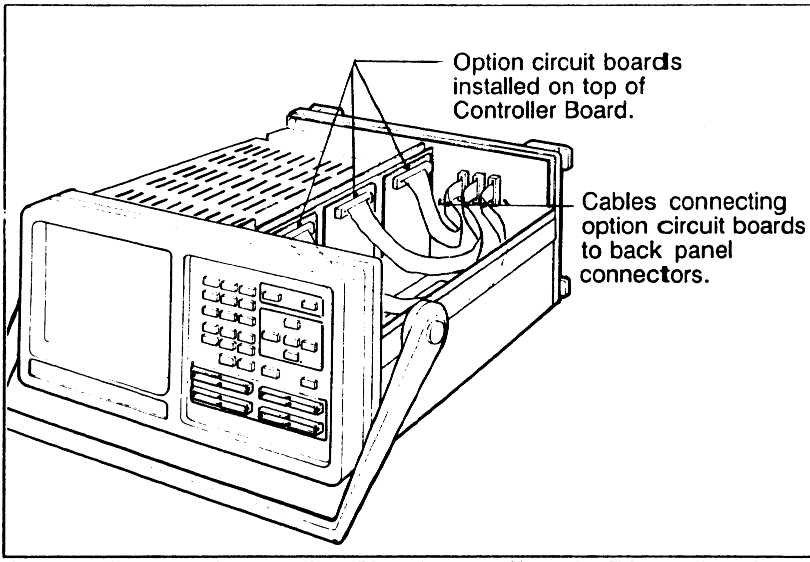


Figure C-3. Option cards. If you're installing the RS-232C or GPIB interface, install the GPIB card in the front slot, the printer card in the middle, and the RS-232C card in the rear slot. The Notes option doesn't use interrupts, so you may install this option in another slot, closer to the front panel.

PROBES

You can order three kinds of probes for the 1230 Logic Analyzer.

- P6444 acquisition probes
- P6443 acquisition probes
- Disassembly probes

The P6444 probes are standard 16-channel acquisition probes that let you specify a variable threshold voltage for triggering. The P6443 probes are optional 16-channel acquisition probes that acquire data at the standard TTL threshold voltage level (+1.4 V). The microprocessor disassembly probes are also optional. Refer to Appendix D, *Options and Accessories,* for a list of microprocessors the analyzer supports.

Connecting Probes

This procedure tells you how to connect probes to the analyzer's front panel.

1. Make sure power to the analyzer is off and the power cord disconnected.

CAUTION

Do not connect any probe to the analyzer unless power to the analyzer is off and the probe is not connected to a powered system. Always power up the analyzer before powering up the SUT.

2. Refer to Figure C-4 and plug the first probe into probe slot A on the front of the analyzer. To acquire data, probe slot A must always have a probe plugged in.
3. As required, plug in the rest of the probes to probe slots B, C, and D on the front panel.
4. Power up the analyzer, which powers up the probes.
5. Connect the probe leads to your SUT.
6. Power up the SUT.

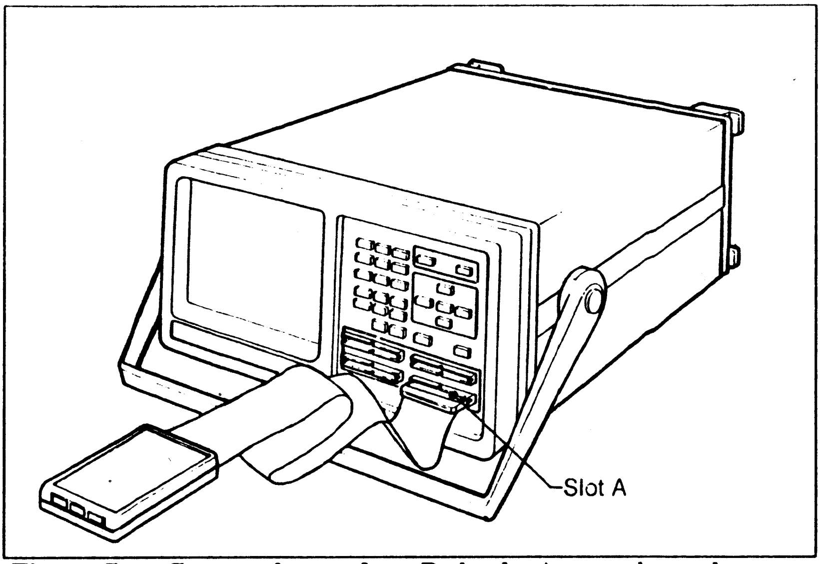


Figure C-4. Connecting probes. Probe slot A must always have a probe plugged in to acquire data. If you have more than one probe, you don't have to connect them in any particular order, but they must be connected. For simplicity, it's best to plug the second probe into slot B, the third in C, and so on.

Setting Up Acquisition Probes

This discussion tells you how to set up acquisition probes, which are used with the expansion cards installed earlier in this section.

The analyzer can have up to four acquisition probes. You may connect as many probes to the analyzer as you have expansion cards installed. If you don't plug in the probes but have expansion cards installed for those probes, the analyzer assumes you are still doing acquisition on those missing channels. The state display will show an invalid acquisition (dimmed, to show that it is invalid) for the channels on the missing probes. The timing diagram won't show traces for the invalid data.

The P6444 Probe

The P6444 acquisition probe lets you specify different threshold voltages at which the analyzer recognizes a data high or low. This probe also uses DIP switches to let you quickly and easily configure clock, qualifier, and external lines for your SUT. Figure C-5, later in this section, shows the DIP switches.

Variable Threshold Voltages. The P6444 probe lets you use these voltages:

TTL +1.4 V
HCMOS +2.5 V
ECL -1.3 V
VAR. -9.0 to +9.0 V (excluding 0 V)

To change the threshold voltage, move the cursor to the threshold field in the Timebase menu and press 0 or 2 to cycle through the voltage options. When you set the threshold field to VAR, you can then change each voltage digit by pressing 0 or 2 to increase or decrease the field's value. You can also use those keys to change the polarity for logic True.

If you set a threshold voltage other than TTL, then plug in a P6443 probe without changing the threshold back to TTL, the analyzer gives you a warning message when you power up again. The message tells you that the current threshold voltage settings are incompatible with the installed probes. When you call up the Timebase menu, the analyzer asks if it can change the settings to compatible values. Follow these steps to restore valid threshold voltage settings.

1. Call up the Timebase menu and press ENTER when you see this message:

```
OK to change incompatible threshold values?
   Press ENTER to confirm, MENU to abort
```

2. Press MENU to save the Timebase menu and return to the analyzer's Main menu.

Figure C-5 shows the P6444 probe label, listing the probe connections and DIP switches for the clock, qualifier, and external lines. Use a pointed tool, such as a pin or paper clip, to toggle the DIP switches.

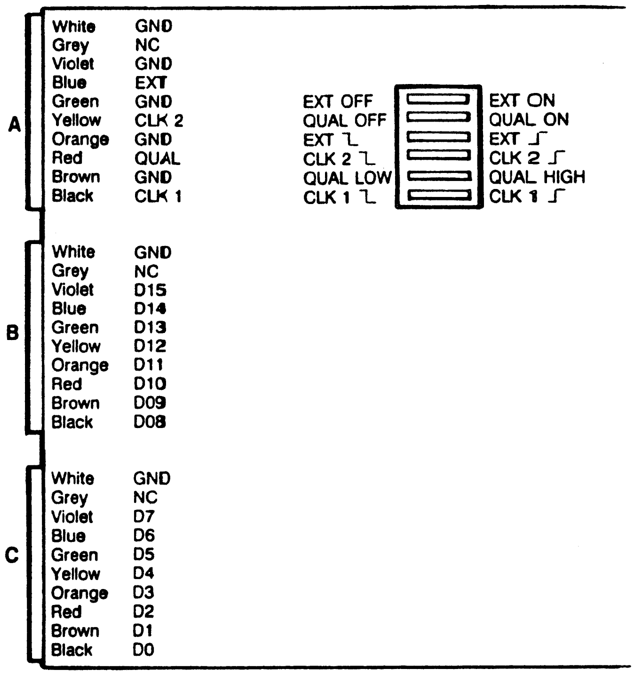


Figure C-5. P6444 label and switches. The probe's DIP switches are set into the probe casing to keep them from being accidentally toggled during setups and acquisitions. Use a pointed tool, such as a pin or paper clip, to toggle the DIP switches.

P6444 Clock Lines. When acquiring data with a synchronous clock, you can choose to acquire data samples on either rising edges or falling edges of the clock signal. Use the CLK 1 and CLK 2 switches on the top of the probe to select either a rising or falling edge for the corresponding clock inputs (see Figure C-5).

The following examples explain some synchronous clock settings.

NOTE

When you connect the clock or qualifier lines you must also connect the ground lines that are associated with that lead set. The grounds for the external, clock, and qualifier lines are not white. Check the probe label to make sure you use the proper line for ground.

- If you want to acquire data on the rising edge of the clock pulse, connect CLK 1 to the SUT clock source and set the CLK 1 switch on the probe to _|¯.
- If you want to acquire data on the falling edge of the clock pulse, connect CLK 1 to the SUT clock source and set the CLK 1 switch on the probe to ¯|_.
- If you want to acquire data on the rising or falling edge of the clock pulse, connect CLK 1 and CLK 2 to the SUT clock source and set the CLK 1 switch to _|¯ and the CLK 2 switch to ¯|_.
- If you want two probes to acquire data in parallel on the rising edge of the clock pulse, connect the CLK 1 line from both probes to the same clock source point in the SUT. Then, set the CLK 1 switches on both probes to _|¯.
- If you want to make a twisted pair for the clock line to make sure no noise appears on the line, twist the nearest ground to the CLK line before connecting it to the SUT clock source. It is especially advisable to twist a ground around the clock lead if you are acquiring at rates faster than 15 MHz.

You don't have to connect CLK lines that aren't being used. However, you should pull the unused leads away from the others to reduce cross talk between the leads.

P6444 Qualifier Line. You can use the P6444 qualifier input to qualify acquisition clock signals. This means that the analyzer doesn't acquire data on a clock pulse unless a True signal is present on the qualifier line at the same time the clock pulse occurs. Use the QUAL ON/OFF switch to turn on the qualifier input line. Use the QUAL LOW/HIGH switch to tell the analyzer to recognize a logic low or a logic high as a True signal.

NOTE

When the QUAL line is not being used, the QUAL ON / OFF switch must be in the off position.

P6444 External Line. The EXT input is edge-sensitive. It recognizes signal transitions, not just whether a signal is high or low. To use the EXT input line, turn on the EXT ON/OFF switch and use the EXT switch to select whether a rising or falling edge should be recognized.

If you use the EXT line, you can select the EXT line's signal as a trigger condition in the Trigger Spec menu. Section 3, *Setup Menus*, explains how to specify trigger conditions in the Trigger Spec menu.

The P6443 Probe

The P6443 acquisition probe has a threshold voltage of TTL +1.4 V. This discussion tells you how to connect the clock, qualifier, and external lines from the probe to your SUT.

P6443 Clock Lines. Each P6443 probe has four clock lines and two clock qualification lines (see P6443 Qualifier Lines later in this section). The CLK HIGH inputs cause data to be acquired on rising edges of the clock signal, and CLK LOW inputs cause data to be acquired on falling edges. The way you connect the probe CLK lines determines each probe's synchronous clocking. The following examples explain some synchronous connections.

NOTE

When you connect the clock or qualifier lines you must also connect the ground lines that are associated with that lead set. The grounds for the clocks and qualifiers lines are not white. Check the probe label to make sure you use the proper line for ground.

- If you want to acquire data on the rising edge of the clock pulse, connect a CLK HIGH to the SUT clock source.
- If you want to acquire data on the falling edge of the clock pulse, connect a CLK LOW to the SUT clock source.
- If you want to acquire data on the rising or falling edge of the clock pulse, connect a CLK HIGH and a CLK LOW line to the SUT clock source.
- If you want two probes to acquire data in parallel on the rising edge of the clock pulse, connect a CLK HIGH line from both probes to the same clock source point in the SUT.
- If you want to make a twisted pair for the clock line to make sure no noise appears on the line, twist the nearest ground to the CLK line before connecting it to the SUT clock source. Clock and ground lines should be twisted around each other if your sample rate is greater than 15 MHz.

You don't have to connect CLK and QUAL lines that aren't being used; unused CLK and QUAL lines are automatically pulled to the True state.

When more than one clock signal is connected, the signals are ORed together:

(CLK High OR CLK High OR CLK Low OR CLK Low)

P6443 Qualifier Lines. Each acquisition probe has two clock-qualifier lines. If qualifier lines are connected, a True signal (logic high for QUAL HIGH, and low for QUAL LOW) must be present on them at the same time the acquisition clock pulse occurs for a data sample to be acquired. If you're not using the qualifier lines, you don't have to connect them. The analyzer automatically pulls them to the True state. However, to reduce noise and cross talk in the leads, you should pull unused leads away from the others.

If more than one qualifier line is being used, they are ANDed together with the clock signals. The clock lines themselves are ORed:

(CLK High OR CLK High OR CLK Low OR CLK Low)
AND
(QUAL High AND QUAL Low)

P6443 External Lines. Each acquisition probe has two external lines. These lines can acquire an external signal that you can use in the Trigger Spec menu as a condition on which to trigger. The external lines recognize a transition from high to low, or low to high. To use the high external line (EXT HIGH), connect it to your SUT and connect the low external line (EXT LOW) to ground. To use the low external line, connect it to your SUT and connect the high to a logical high point in the circuit.

If you're acquiring data in parallel on two or more probes, connect both (or all) EXT HIGH lines to the same point in your SUT and connect both EXT LOW lines to the same point in your SUT. This lets you use the external lines for triggering parallel acquisition across the probes.

Setting Up Microprocessor Probes

Follow these steps to connect a microprocessor disassembly probe to the analyzer. Refer to the operator's manual for your disassembly probe for detailed instructions about using the microprocessor probe.

1. Make sure power to the analyzer and SUT is off.

CAUTION

Do not connect any probe to the analyzer unless power to the analyzer and SUT is off. If you connect a probe when the analyzer power is off and the SUT is on, the probe may be damaged.

2. Plug the bottom cable from the probe into slot A on the front of the analyzer.
3. Plug the top cable from the probe into slot B on the front of the analyzer.
4. If your disassembly probe has three probe cables, connect the third probe cable to probe slot C on the front panel of the analyzer.
5. Connect the microprocessor-probe clip to the system you're testing.
6. Power up the analyzer, which powers up the probes.
7. Turn on power to the system.
8. Press ENTER to upload the microprocessor disassembly setup. If you press MENU instead, the previous 1230 setup is saved and the microprocessor setup is not loaded into the analyzer. Whether you press ENTER or MENU, the Main menu is displayed as soon as you upload the disassembly setup or save the existing setup.

You can change the analyzer setup for the probe later, but if you make a mistake changing the setup, the probe may not acquire or disassemble properly. Refer to your disassembly probe's operator's manual for more information about your microprocessor-disassembly probe.

As long as the data in memory is valid for the microprocessor and you have the disassembly probe plugged in, you can display acquired data in disassembly format on the analyzer.

Appendix D
ACCESSORIES AND SPECIFICATIONS

This appendix lists the standard and optional accessories for the 1230 Logic Analyzer. This appendix also lists electrical, performance, environmental, and physical specifications.

OPTIONS

This list identifies the analyzer's options:

1230E1	16-channel expand card, includes one P6444 probe
1230 Option 01	RS232 port & PC software (S43R101)
1230 Option 02	Parallel printer port
1230 Option 03	GPIB port
1230 Option 04	International notes
1230 Option 05	Rack-mount hardware
1230 Option 06	Accessory bag
1230 Option 07	GPIB acquisition leadset
1230 Option 1S	Substitute P6443 Probe for P6444
1230 Option 21	1230 Logic Analyzer Service Manual
1230 Option 22	Service Maintenance Kit
1230 Option 23	VHS-PAL Videotape
1230 Option A1	230V/6A 50 Hz Universal Euro plug
1230 Option A2	230V/6A 50 Hz U.K. plug
1230 Option A3	230V/6A 50 Hz Australian plug
1230 Option A4	230V/6A 60 Hz North American plug[1]
1230 Option A5	230V/6A 50 Hz Swiss plug

[1] Canadian Standards Association certification

STANDARD ACCESSORIES

The following list identifies the 1230 Standard Accessories.

1 1230 Operator's	070-6878-02
1 Quick Reference Guide	070-6880-02
1 Training videotape	068-0249-04 (VHS-NTSC format)
1 Workbook	062-9731-02
1 Power Cord	161-0104-00
1 Test Circuit	671-0049-00
1 P6444 Probe	

P6444 Standard Accessories

1 black lead set	174-1264-00
1 red lead set	174-1265-00
1 white lead set	174-1266-00
22 IC grabber tips, double sided	206-0364-00

OPTIONAL ACCESSORIES

The following list identifies the 1230 Optional Accessories.

A6740G Serial Probe	A67040G
GPIB Acquisition Lead Set	012-1276-00
1230 Rack-mount Kit	016-0947-00
1230 Service Maintenance Kit	020-1674-00
Protective Cover	200-3639-00
Accessory Bag	016-0707-00
1230 Training Videotape, U-Matic format	068-0249-00
1230 Training Videotape, Beta I format	068-0249-01
1230 Training Videotape, Beta II format	068-0249-02

1230 Training Videotape, Beta III format	068-0249-03
1230 Training Videotape, VHS-NTSC format	068-0249-04
1230 Training Videotape, VHS-PAL format	068-0249-06
1230 Training Videotape, VHS-SECAM format	068-0249-00
Package of 12 IC grabber tips, double sided	206-0364-00
RS232 cable for use with the Option 01	012-0911-00
Parallel printer cable for use with Option 02	012-0997-00
GPIB cable for use with Option 03	012-0630-03
Training/test circuit	671-0049-00
8-Inch test lead for test circuit	196-3222-00
Package of 5 training/test circuits	016-0948-00
Package of 5 quick reference guides	016-0949-00
Package of 5 workbooks	016-0950-00

SPECIFICATIONS

The following tables list the specifications for the 1230 Logic Analyzer:

- D-1 Electrical Specifications: Power Requirements
- D-2 1230 Performance Specifications
- D-3 1230 Environmental Specifications
- D-4 1230 Physical Specifications
- D-5 P6444/P6443 Performance Specifications

The performance specifications in Table D-2 are valid under the following conditions:

1. The 1230 must be in an operating environment whose limits are listed in Table D-3, Environmental Specifications.
2. The 1230 must have been calibrated at an ambient temperature between +20 C and +30 C, after a 30-minute warm-up.

3. Any applicable conditions, not listed above but unique to a particular specification, are stated as part of that specification.

The performance requirements column in Table D-2 contains values you can verify. Refer to your *1230 Logic Analyzer Service Manual* for information about verifying the requirements. The supplemental information column in Table D-2 contains explanatory notes.

Table D-1
ELECTRICAL SPECIFICATIONS: POWER REQUIREMENTS

Characteristic	Description
AC input	115/230 VAC, 47 - 63 Hz Nominal
Range	90 - 132/180-264 VAC
Maximum power input	120 VA

Table D-2
1230 PERFORMANCE SPECIFICATIONS

Characteristic	Performance Requirement	Supplement Information
Batteries		2 Lithium, 3 V 250 Ma each, replaceable
RAM battery life		2 years typical
Clock/calendar battery life		12 years typical
Asynchronous clocking maximum number of rates	2	Requires at least one 1230E1 installed
Active channels per probe	16 channels, glitch off 8 channels, glitch on 8 channels, 20 ns rate 4 channels, 10 ns rate	40 ns or slower 40 ns or slower

Table D-2 (cont.)
1230 PERFORMANCE SPECIFICATIONS

Characteristic	Performance Requirement	Supplement Information
Range for each of the two async timebases	100 MHz 10 ns 50 MHz 20 ns 25 MHZ 40 ns 12.5 MHz 80 ns 5 MHz 200 ns 2.5 MHz 400 ns 1 MHz 1 µs 500 KHz 2 µs 250 KHz 4 µs 100 KHz 10 µs 50 KHz 20 µs 25 KHz 40 µs 10 KHz 100 µs 5 KHz 200 µs 2.5 KHz 400 µs 1 KHz 1 ms 500 Hz 2 ms 250 Hz 4 ms 100 Hz 10 ms 50 Hz 20 ms 25 Hz 40 ms	Two async timebases can be used at once if one 1230E1 is installed

Table D-2 (cont.)
1230 PERFORMANCE SPECIFICATIONS

Characteristics	Performance Requirements	Supplemental Information
Timebase accuracy @ 25 MHz	Within .01%	
Synchronous clocking		
Maximum frequency	25 MHz (40 ns)	
Setup time	10 ns min	
Hold time	0 ns max	
Clock pulse	10 ns min	
Maximum number of rates	4	Requires three 1230E1s
Active channels per probe	16	
External trigger-out pulse width	40 ns or 1 cycle whichever is greater	TTL-level output back panel connector
External trigger-out Delay		4 cycles or 160 ns whichever is greater back panel connector
External trigger-in pulse width	20 ns minimum	TTL-level input, positive edge triggered, back panel connector
External trigger-in delay		20 ns + 2 cycles back panel connector
External trigger input resistance		100 ΩK nominal back panel connector

Table D-2 (cont.)
1230 PERFORMANCE SPECIFICATIONS

Characteristic	Performance Requirement	Supplemental Information
External trigger input capacitance		10 pF nominal, back panel connector
External trigger input maximum voltage range		-5 V to +10 V back panel connector
Trigger position		128 to 1920 user selectable in increments of 128
Trigger levels		14
Word recognizers per level		1 Basic Mode 2 Advanced Mode
Minimum word recognition periods with one probe:		
25 MHz async		43 ns or 1 clock period whichever is greater
50 MHz async		23 ns
100 MHz async		13 ns, basic-mode triggering 23 ns, advanced-mode triggering

Table D-2 (cont.)
1230 PERFORMANCE SPECIFICATIONS

Characteristic	Performance Requirement	Supplemental Information
Minimum word recognition periods with multiple probes:		
25 MHz async		52 ns or 1 clock period whichever is greater
50 MHz async		32 ns
100 MHz async		22 ns, basic-mode triggering
		32 ns, advanced-mode triggering
Word recognizer reset time		40 ns minimum
Level-level transition time, Trigger Spec menu	80 ns or 2 cycles	whichever is greater
Acquisition memory depth	2047 bits/channel	25 MHz and below
	2048 bits/channel	50 MHz, 100 MHz
Minimum detectable glitch	5 ns	3 ns typical, 40 ns async rate or slower
Condition counter range	1-4096	
Video output level		RS170 compatible
EXTERNAL minimum pulse width		10 ns EXT signals on acquisition probe(s)

Table D-3
1230 ENVIRONMENTAL SPECIFICATIONS

Characteristic	Description
Temperature	
operating	0 C min.; +55 C max.
storage	-40 C min.; +70 C max.
Humidity	
operating & non-operating	95%-97% max.
Altitude	
operating	4500 m (15,000 ft.)
non-operating	15000 m (50,000 ft.)
Electromagnetic interference	meets FCC Part 15, Sub-part J, Class A, without probes
Electrostatic immunity	The instrument will discharge through 200 Ωs resistance of a 500 pF capacitor charged to 4 KV without component failure.
Safety	This instrument complies with the requirements of UL1244, IEC348, and CSA 556B.
Cooling clearances	
top	None required
bottom	2 mm (0.5 in.)
rear	50 mm (2 in.)
sides	25 mm (1 in.)

Table D-4
1230 PHYSICAL SPECIFICATIONS

Characteristic	Description
Weight (no accessories)	8.6 kg (19 lbs), 1230 mainframe only 9.7 kg (21.5 lbs), 1230 with three 1230E1 expander cards
Overall dimensions	See also Figure 2-1
height	19 cm (7.48 in), handle folded back
width	37.6 cm (14.8 in), including handle
depth	
(handle folded back)	42. cm (16.5 in)
(including handle)	48.9 cm (19.3 in)
Probe cable length	29 inches

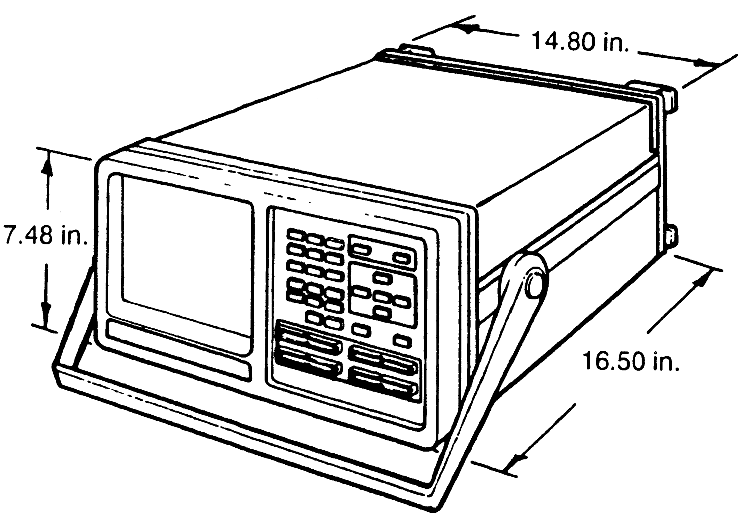


Figure D-1. Physical dimensions of the 1230

Table D-5
P6444/P6443 PERFORMANCE SPECIFICATIONS

Characteristic	Performance Requirement	Supplemental Information
Input resistance (all inputs)		1 MΩ ±5%
Input capacitance (all inputs		8 pF nominal, without test leads
Nondestructive input voltage range		
P6444		±40 V
P6442		-5 to +10 V
Threshold range		
P6444		Selectable from +9 V to -9 V (excluding 0 V) in 100 mV increments; also includes preset values for TTL (+1.4 V), ECL (-1.3 V), and HCMOS (+2.5 V)
Input voltage	V in high 2.0 V min.	TTL-level, 1.4 V
P6443	V in low 0.8 V max.	nominal
Minimum input voltage swing P6444	600 mV	Centered around voltage swing detected

Appendix E
DIAGNOSTICS

The 1230 Logic Analyzer diagnostics functionally check the instrument at power up if this feature is turned on in the System Settings menu.

The 1230 performs these diagnostics:

1. A test of the four system RAM banks.
2. A status check of the backup RAM battery.
3. A test of the four video RAM buffers.
4. Test access to all RAM banks.
5. Check that the setups stored in the instrument when it was powered down remain valid at power up.
6. A test of the checksums for the four banks of ROM and comparison of these checksums to calculated values.
7. Configuration check of any installed expansion cards.
8. Acquisition card tests. This test is performed only if test 7 passes.
9. A query of the "smart" expansion cards, such as the RS-232C card, for pass/fail of on-card diagnostics routines.

The 1230 Logic Analyzer remains operational as long as any error message displayed does not contain the word "FATAL".

If you suspect that a board or component failure is compromising the functions or capabilities of the instrument, contact your Tektronix field office for more information.

GLOSSARY

acquisition - The sampling of data from a system under test by a logic analyzer. Data is conditionally stored in acquisition memory so that you can display it in different formats or compare it to other acquired data.

acquisition clock - The clock that determines the rate at which the logic analyzer will sample data. The clock can be an internal analyzer clock (asynchronous) with sampling rates you can specify, or an external clock (synchronous) with the sampling rate determined by your SUT. See *external clock* and *internal clock*.

acquisition cycle - A complete data acquisition sequence including start, search, sample, and stop phases.

acquisition memory - The RAM in which data acquired by the analyzer is stored.

acquisition memory sequence - A single sample of acquired data to which a unique identification number is assigned. As data is acquired, each sample is assigned a consecutive sequence number.

action - An instruction in the trigger specification menu that tells the analyzer to trigger, turn data storage on or off, or do nothing (NOP).

active setup - The set of parameters that the analyzer is currently using to specify probe links, timebase formats, sampling rates, channel groups, conditions, reference and update memories, and so on.

address - A number or expression that designates a specific location in a storage or memory device.

aliasing - The condition that occurs when data from an SUT is sampled at a rate slower than the rate at which data changes.

When this happens, meaningless data is displayed because the analyzer misses the changes in data that occurred between sample points. Data pulses that fall between samples meet the technical definition of a glitch and will be stored and displayed as glitches. See also *asynchronous acquisition* and *glitch.*

ASCII - American Standard Code for Information Interchange. ASCII is an eight-bit code representing characters and control functions.

assert - To cause a signal or line to change from its logic false state to its logic true state.

asynchronous acquisition - An acquisition that is made using a clock rate and signal generated internally by the analyzer. Such a signal is asynchronous to your circuit. You should use an asynchronous rate that is considerably faster than your data rate to avoid aliasing. See also *aliasing.*

channel - A signal line. For example, if you have 16 channels in a channel group and you connect probe lead 16 to signal line A15 on your SUT, then channel A15 corresponds to SUT signal line A15.

channel group - A user-defined group of input channels that use the same timebase, polarity, and radix. Channel groups can include up to 32 input channels.

channel mask - When you mask a channel, you're omitting from the display the data acquired on that signal line. This can be very useful if you only want to see the signals on specific channels. You can mask channels three ways. You can use the channel mask field in the run control menu to specify channels you don't want to see data on. You can delete unwanted channels from the channel group in the channel group menu and leave them in the unused channels list. You can also (in the state table) turn a channel group's radix off to mask it from the display.

clock - The regular signal pulse that determines the sampling rate for the logic analyzer. See also *acquisition clock, external clock, internal clock,* and *sample clock.*

clock cycle - A clock sequence that includes both high- and low-going transitions.

clock equation - The Boolean combination of events needed to generate a store command. You can define a variety of clock inputs and link them using Boolean operators. Data will only be sampled and stored in memory when this clock equation is true.

clock qualification - The process of filtering out irrelevant data by combining an acquisition clock signal with one or more bus signals.

clock qualifier - An external signal that acts as a gate for the acquisition clock. When the external signal is false, the acquisition clock is not allowed to load acquired data into the acquisition memory. See also *acquisition memory.*

comparing memory - The process through which you compare data stored in one acquisition to data stored in another acquisition. The Run Control menu lets you specify the memories you want to compare.

condition - A value or event on which the analyzer will trigger. The analyzer has 51 conditions: trigin, external, clock, A, NOT A, B, NOT B, and so on through X and NOT X (and MASK, with version 4.0 software). Use the conditions menu to define conditions. You can define the 24 conditions A, B, C, and so on through X.

correlation - The tracking of independent events captured by different acquisition modules and indicating how they relate to each other in time. Specifically, the chronological interleaving of data from two or more different acquisition modules into a single display; this shows real-time interactions between independently clocked circuits.

count - A field in the trigger specification menu that lets you specify the number of times you want the condition or event to occur before the analyzer triggers.

cursor - A marker representing a specific location on the terminal screen. When the cursor highlights a field, you can select functions for that field (for example, edit a channel group name or reorder traces). In some menus, such as the state and timing displays, the cursor is a line showing the current location.

cycle - See *acquisition cycle* or *clock cycle*.

data correlation - See *correlation*.

destination - A field in the trigger specification menu that lets you specify whether data is stored, memory is filled, or the analyzer jumps to another level of conditional statements before finishing the acquisition cycle.

display memory - the memory which is automatically displayed after the analyzer completes an acquisition or after you press STOP. You specify the display memory in the Run Control menu.

don't care - A symbol (X) used in place of a numeric character to indicate that the value of a channel or character is not to be considered.

download - The process of writing data from the host to the analyzer. For example, you can download a data acquisition into the 1230 from a host system in order to do a memory comparison. You can download a stored setup into the analyzer from the host so that you can acquire data on a setup you stored on the host system. See also *load* and *upload*.

edge - A signal transition from low to high, or high to low.

edge triggering - Triggering on a low-to-high or high-to-low transition instead of triggering on the low or high threshold.

event - A defined set of values the analyzer uses to determine when to trigger. For example, an event could be the beginning of a subroutine that has the value 0F15.

expansion probe - A 16-channel probe with which you can acquire data from your SUT. The analyzer has two expansion probes: the P6444 and the P6443. The P6444 expansion probe lets you set the threshold voltage at which the analyzer triggers. See also *microprocessor disassembly probe.*

external clock - A clock external to the logic analyzer and usually synchronous with your SUT. See also *acquisition clock* and *internal clock.*

external clocking - A clock mode in which the sampling of input logic signals is synchronized with the activity of your SUT. The representation of the signals is stored in memory by the SUT clock which provides a signal external to the analyzer.

glitch - A signal that makes a transition through the threshold voltage two or more times between successive sample clock cycles. Signals that are faster than the sampling rate, such as noise spikes or pulse ringing, can be captured by a logic analyzer as glitches.

graticule - A line of tic marks near the top of the timing diagram that shows how many samples are being displayed on the screen. Each tic mark represents one sample. If the graticule is a solid bar, the display resolution is at 2048, and the entire contents of memory are displayed at once.

hold time - The length of time a signal must remain stable after the acquisition clock has occurred in order to be considered valid.

internal clock - The internal analyzer clock, which is used to determine the rate at which you will acquire data asynchronously. See also *acquisition clock* and *external clock.*

internal clocking - A clock mode in which the sampling of input logic signals occurs asynchronously to the activity of your SUT. The representation of the signals is stored in memory by the internal (asynchronous) clock.

level - A line in the trigger specification menu that can contain a trigger statement. You can have up to 14 levels of trigger statements to qualify triggers, data acquisition, and data storage.

linking probes - A way to expand acquisition so that you can acquire data with the same word recognizers on more than 16 channels. For example, if you have two probes and link them together, the 1230 acquires data on all 32 channels using the same timebase and sampling rate.

load - The process of writing data from one memory to another. For example, you can load the stored default setup into the analyzer from the storage menu so that you can display previously acquired data. See also *download* and *upload*.

logic analyzer - An instrument, useful in both hardware and software analysis, used to acquire logic signals in real time with respect to a trigger event. The 1230 Logic Analyzer acquires up to 64 signals and displays them in state, timing, and disassembly formats.

mainframe - A mechanical cabinet providing power, cooling, and backplane resources to analyzer cards. The mainframe also houses I/O connectors.

mask - The process of omitting certain data from the acquisition or display. For example, if you mask channels in the conditions menu, the analyzer still acquires data on those channels, but doesn't display it when you look at the acquisition display.

memory select - A menu that lets you choose the memory you want to display.

menu - A screen display that lists fields you can choose, edit, and turn on or off. A menu can also show setups, data, and status information. For example, the trigger specification menu lets you specify the conditions on which the analyzer will trigger and acquire data. The state table menu displays acquired data in state table format, and lets you choose the radix for the data displays.

menu bar - A one-line message at the bottom of the menu or display screen that tells you which keys to press for screen functions. If the menu or display has more than one menu bar, you can usually press **F** to cycle through the bars.

microprocessor disassembly probe - A probe that disassembles microprocessor instructions for data acquired across the probe's lines. For example, a Z80 probe disassembles Z80 microprocessor instructions. The analyzer's microprocessor probes disassemble data and display it in hardware and software formats.

pipeline - The several stages of latches that acquired data must be clocked through to reach acquisition memory.

pop-up menu - A menu that covers a portion of the complete screen when it appears. For example, the timebase menu is a pop-up menu.

post-fill - When the analyzer continues to acquire data after the trigger until the specified number of samples has been taken. When the trigger position is programmed to occur in the center of memory, the analyzer takes enough samples after the trigger to fill one-half of the acquisition memory. See also *trigger position.*

post-processing - Any type of acquisition or reference memory analysis that occurs after (rather than during) the data acquisition.

pre-fill - When the analyzer fills the specified number of acquisition memory sequences with new data before accepting a trigger. If the trigger is programmed to occur in the center of memory, one-half of the memory must be filled before the trigger. See also *trigger position.*

pre-trigger data - Data acquired before the trigger condition occurred. This data is continually overwritten with newly acquired data as the analyzer continues to look for the trigger. When the trigger occurs, the analyzer positions the trigger event at the location specified in the Run Control menu. The analyzer then fills the rest of memory as you specified in the Trigger Spec menu. See also *trigger position.*

probe - An input device, constructed as a separate unit, that transmits the input signal from the SUT to the analyzer.

probe links - The sampling setup links between acquisition probes. For example, if you link two 16-channel probes in the same timebase, the two probes acquire in parallel-as if they were one 32-channel probe. The two probes then use the same sampling rate and format to acquire data.

protection - A condition that prevents accidental overwriting of stored setups in the storage menu.

qualification - See *clock* or *storage qualification.*

qualifier - See *clock qualifier.*

radix - The alphanumeric base in which data is displayed. For example, the analyzer lets you display data in the state table in hexadecimal, binary, octal, and ASCII radixes. You can display and define condition words and channel masks in hexadecimal, octal, and binary radixes.

reference memory - The memory containing previously acquired data to which you want to compare the current acquisition memory.

resolution - The minimum detectable interval between data transitions on a single acquisition channel or between data transitions on any two acquisition channels.

run control - The menu parameters that control the way data is acquired, displayed, and compared. For example, you can specify a pre-trigger acquisition instruction so that the analyzer fills memory before triggering on the specified condition.

sample clock - A timing signal that establishes the rate of data sampling.

sampling format - The format, asynchronous or synchronous, that the analyzer uses to determine whether to use an internal or external clock for sampling data.

sampling rate - The clock rate at which the analyzer will sample data. If you're using an asynchronous format, you can set the rate to incremental values between 40 ms and 10 ns. If you're using synchronous format, you cannot set the timebase rate because the analyzer will use the clock from the circuit under test.

screensaver - A feature that automatically turns off the analyzer screen after 18 minutes of inactivity. This helps prevent screen burn-in. You can turn the screensaver feature on or off in the system settings menu.

scroll - A method of moving through a data display that's too big to fit on one display screen. For example, the state table displays 20 lines of data at a time; you can use the cursor keys to scroll through the rest of the 2047 lines of memory.

sequence - See *acquisition memory sequence.*

setup - A set of parameters the analyzer uses to trigger, acquire data, and store data acquisition. Setup parameters include probe links, timebase formats, sampling rates, channel groups, condition word definitions, trigger statements, and data display formats. You can store setups in nonvolatile memory with the Storage menu.

skew - The relative time difference between input channels, specified in terms of one edge relative to another. If parallel input channels have different propagation delay times, the data can be skewed and misrepresented by the analyzer.

slot - A backplane connector position.

state table - Tabular representation of logic states of input data channels.

storage qualification - The process of filtering out data that has been acquired but which you do not want to store in acquisition memory. This lets you avoid filling up your module's acquisition memory with irrelevant data samples.

storing setups - The process of storing a set of acquisition parameters in nonvolatile memory. Use the storage menu to save setup parameters.

synchronous acquisition - An acquisition mode using a clock external to the logic analyzer; usually the clock in your SUT. The external clock is usually synchronous with the SUT and may or may not be periodic.

system settings - A menu that lists system fields you can adjust. For example, you can adjust the brightness of the screen and you can also turn the screensaver feature on so that the display turns itself off after 18 minutes of inactivity. You can also turn the power-up diagnostics on and off.

threshold - A voltage to which input signals are compared. If you're using a P6444 expansion probe instead of the P6443, you can set the threshold to be HCMOS, TTL, ECL, or VAR (variable) from -9.0 to +9.0 V (excluding 0 V).

tic marks - At the top of the timing display, a line of tic marks (a graticule) shows the resolution at which you sampled data.

For example, if the tic marks are far apart, you are displaying data at a high resolution - looking at few samples closely. If the tic marks are close together, you are displaying data at a low resolution - looking at many samples with less detail.

timebase - Source of the acquisition clock. The timebase can be an internal source, a single external source, or a Boolean combination of several external signals. See also *acquisition clock, external clock,* and *internal clock.*

timer - A device that allows you to make time measurements during a data acquisition. The 1230 Logic Analyzer uses delayed clocking for this purpose.

timing display - Graphic representation of data states and timing relationships as digital (two-state) waveforms.

trigger - An event or condition that leads to the end of the acquisition cycle. When started, the analyzer continuously acquires data from your SUT until the trigger occurs. After triggering, the analyzer continues to load data until the post-fill requirement is met. See also *pre-fill* and *post-fill.*

trigger position - The location of the trigger in memory. For example, you can position the trigger at location 0512, about one-quarter of the way through memory. When the analyzer acquires data, it fills the pre-trigger memory, continually over-writing that data with newly sampled information. When the analyzer finds the trigger event, it positions that event at the location you specified, then fills the rest of memory with post-trigger data. If you want to see data that occurred before a trigger event, position the trigger near the end of memory. If you want to see what occurred after the trigger event, position the trigger near the beginning of memory.

trigger specification - The trigger specification menu allows you to define the acquisition triggers, conditions, and storage. The trigger specification is a statement or set of statements the analyzer uses to determine when to trigger, when to acquire data, and how to store data once it has been acquired.

update memory - A memory reserved for newly acquired data. When you acquire data, the analyzer stores new information in the update memory. If you then compare data, the analyzer compares the newly acquired data in the update memory to the previously stored data in the reference memory. Use the Run Control menu to specify the memories to use for the reference and update memories.

upload - The process of writing data from the analyzer up to a host system. For example, you can upload a data acquisition from the analyzer to a host system to save it in the host memory. You can also save more than eight setups by uploading some to the host system. See also *load* and *download*.

viewport - A window at the bottom of the screen or menu which gives additional information about that screen or menu. For example, in the Trigger Spec menu, the viewport lists definitions for the cursor-highlighted condition, count, action, or destination field.

word recognition - The matching of a specific data word with the presence or absence of that word in signals acquired by the analyzer.

INDEX