

Logic Analyzer Short Guide

Accessing the Menus

When you power up the logic analyzer, the first screen after the system tests is the Analyzer Configuration menu. Menus are identified by two fields in the upper left corner. The leftmost field shows Analyzer. This field is sometimes referred to as the "mode field" because it controls which other set of menus you can access. The second field, just to the right of the mode field, accesses menus within the mode and so is called the "menu field." For example, if you are in Analyzer mode, the menus for the analyzer are accessed from the menu field. Menus are referred to by the titles that appear in the mode and menu fields, for example, the Analyzer Configuration menu.

To access the System menus

The System menus allow you to perform operations that affect the entire logic analyzer, such as load configurations, change shades, and perform system diagnostics.

1 Select the mode field.

Use the arrow keys to highlight the mode field, then press the Select key. Or, if you are using the mouse, click on the field. This operation is referred to as "select." A pop-up menu appears with the choices System and Analyzer. (If you have installed any optional software, there may be other choices as well.)

2 Select System.

3 Select the menu field.

The pop-up lists five menus: Hard Disk, Flexible Disk, External I/O, Utilities, and Test.

- Hard Disk allows you to perform file operations on the hard disk.
- Flexible Disk allows you to perform file operations on the flexible disk.
- External I/O allows you to configure your HP-IB, RS-232-C, and LAN interfaces and connect to a printer and controller.
- Utilities allows you to set the clock, update the operating system software, and adjust the display.
- Test displays the installed software version number and loads the self tests.

To access the Analyzer menus

The Analyzer menus allow you to control the analyzer to make your measurement, perform operations on the data, and view the results on the display.

1 Select the mode field.

A pop-up menu appears with the choices System, Analyzer, and Patt Gen. (If you have installed any optional software, there may be other choices as well.)

2 Select Analyzer.

3 Select the menu field.

The Compare menu is only available when you set an analyzer to state mode, and the SPA menu requires an analyzer set to SPA.

- Configuration is always available in Analyzer mode. Use Configuration to assign pods and set the analyzer type.
- Format is available whenever an analyzer is set to a type other than "Off." Use Format to create data labels and symbols, adjust the pod threshold level, and set modes and clocks.
- Trigger is available when an analyzer is set to State or Timing. Use Trigger to specify a trigger sequence which will filter the raw information into the measurement you want to see.
- Listing is available when an analyzer is set to State or Timing. Use Listing to view your measurement as a list of states. Using an inverse assembler, a state analyzer can display the measurement as though it were assembly code.
- Compare is available only when an analyzer is set to State. Use Compare to compare two listings and quickly scroll to the sections where they differ.
- Mixed Display always appears in the menu list when an analyzer is set to State or Timing, but it requires a State analyzer with time tags enabled.

- Waveform is available when an analyzer is set to State or Timing. Use Waveform to view the data as logic levels on discrete lines.
- Chart is available only when an analyzer is set to State. Use Chart to view your measurement as a graph of states versus time.
- SPA is available only when an analyzer is set to SPA. Use SPA to gather and view overall statistics about your system performance.

Type field

The Type field lets you configure the logic analyzer with either an internal clock (Timing mode) or an external clock (State and SPA). When the Type field is selected, the following choices are available.

Timing

When Timing is selected, the analyzer uses its own internal clock to clock measurement data into the acquisition memory. This clock is asynchronous to the signals in the target system. Fields relating to external clocks such as the Analyzer Format menu's Master Clock do not appear and certain menus such as Compare are not available. The analyzer can only be configured with one timing analyzer. If two are selected, the first will be turned off.

State

When State is selected, the analyzer uses a clock from the system under test to clock measurement data into acquisition memory. This clock is synchronous to the signals in the target system.

SPA

SPA stands for System Performance Analysis. It uses an external clock like a state analyzer but measures overall system performance rather than recording discrete activity.

To access the Pattern Generator menus

The Pattern Generator menus let you configure the pattern generator, build test vector files, and use and create user macros.

1 Select the mode field.

A pop-up menu appears with the choices System, Analyzer, and Patt Gen. (If you have installed any optional software, there may be other choices as well.)

2 Select Patt Gen.

3 Select the menu field.

The menus for the pattern generator:

- Patt Gen Sequence menu is used to build test vector files. There are two sequences: an initialization sequence and a main sequence.
- Patt Gen Format menu is used to configure the pattern generator with a clock source and parameters, to generate a symbol table, to select its output mode, to assign which vector output channels are used, and to group and label the vector output channels.
- Pat Gen User Macros menu is used to create new macros and to edit existing macros.

To label channel groups

Hewlett-Packard logic analyzers give you the ability to separate or group data channels and label the groups with a name that is meaningful to your measurement. Labels also assist you in triggering only on states of interest. Labels can only be assigned in the Analyzer Format menu. Once assigned, the labels are available in all display menus, where they can be added to or deleted from the display. Use labels when you want to group data channels by function with a name that has meaning to that function.

The default label names are Lab1 through Lab126. However, you can modify a name to any six-character string.

To create or modify a label and assign channel groups, use the following procedure.

1 Press the Format key to go to the Format menu.

2 Select a label under the Labels heading. In the pop-up menu, select Modify Label.

3 Use the front panel to enter a name for the label and press Done.

4 Select the pod containing the channels for the label. Use the knob or the arrow keys to position the selector over a channel you want to change.

An asterisk indicates the channel is selected; a dot indicates the channel is not part of the current group.

5 Toggle the channel's group status by pressing Select.

To create a symbol

Symbols are alphanumeric mnemonics that represent specific data patterns or ranges. When you define a symbol and set the the base type to Symbol in the Listing menu, the symbol is displayed in the data listing where the bit pattern would normally be displayed. The symbols also appear in the Waveform menu when you view a label in bus form. Symbols allow you to quickly identify data of interest.

To create a symbol, use the following procedure.

1 In the Analyzer Format menu, select Symbols.

The symbol table menu appears. The symbol table is where all user symbols are created and maintained. If you get a message, "No labels specified," check that you have at least one label turned on with channels assigned to it.

2 In the Symbol menu, select the Label field. In the pop-up menu, select the label that contains the channel groups you want.

When you open the symbol table menu, the Label field displays the name of the first active label. If the label you want does not appear in the pop-up menu, the label is probably off. Return to the Format menu, select the label you want, and select Turn Label On. Another possibility is that the label is on the other analyzer. The two analyzers manage resources separately.

3 Select the Base field. In the pop-up menu, select the base for the pattern.

In this example, binary is used because CYCLE only contains three channels.

4 Select the field below Symbol. Select Add a Symbol, type in the symbol name, then press Done.

5 If additional Symbols are needed, repeat step 4 until you have added all symbols.

In this example, three symbols are added: MEM RD, MEM WR, and DATA RD.

6 Toggle the Type field to "range" or "pattern".

When Type is range, a third field appears under the Stop column. To fully specify a range, you need to enter a value for it, too.

7 Select the Pattern/Start field and use the keypad to enter an appropriate value in the selected base. Use X for "don't care."

8 When the pattern is specified, press Done. If you created additional Symbols, repeat steps 6 and 7 until all symbols are specified.

9 To close the symbol table menu, select Done.

To examine an analyzer waveform

The Analyzer Waveform menu lets you view state or timing data in a format similar to an oscilloscope display. The horizontal axis represents states (in state mode) or time (in timing mode) and the vertical axis represents logic highs and lows.

1 In Analyzer mode, press the Run key to acquire data.

In any mode other than Analyzer, pressing the Run key has no effect. The menus which ignore Run lack the Run field onscreen. In Analyzer mode with Run available, the menu changes to a display menu.

2 Go to the Analyzer Waveform menu.

3 To adjust the horizontal axis (sec/Div or states/Div), use the knob.

If nothing happens when you turn the knob, make sure the Div field has a roll indicator above it, as in the figures on the next page. When you first enter the Waveform menu, the knob adjusts the horizontal axis but if you select another rollable field, the knob will control that field instead.

4 To adjust the display relative to the trigger, select the Delay field and enter a value or use the knob.

The portion of memory being displayed is indicated by a white bar along the bottom of the display area.

The position of the trigger in memory is indicated by a white dot on the same line. When the bar includes the dot, then the trigger is visible on the display as indicated by a vertical line with a "t" underneath.

5 To scroll through waveforms, select the large rectangle below the Div field and use the knob.

The roll indicator appears at the top of the rectangle and the name of the first waveform is highlighted. The highlight moves as you turn the knob.

6 To insert waveforms, select the large rectangle under the Div field. In the pop-up, select Insert, and then select the labels and channels.

The Sequential field inserts all the channels of the label as individual waveforms; the Bus field groups the waveforms; the Bit N field inserts just the Nth bit. Waveforms are inserted after the currently highlighted one.

7 To take measurements, select the Markers field and choose the appropriate marker type.

The markers available depend on the type of analyzer and whether or not tagging is enabled. Use markers to locate patterns quickly.

To examine an analyzer listing

The Analyzer Listing menu displays state or timing data as patterns (states).

The Listing menu uses any of several formats to display the data such as binary, ASCII, or symbols. If you are using an inverse assembler and select Invasm, the data is displayed in mnemonics that closely resemble the microprocessor source code.

1 In Analyzer mode, press the Run key to acquire data.

In any mode other than Analyzer, pressing the Run key has no effect. The menus which ignore Run lack the Run field onscreen. In Analyzer mode with Run available, the menu changes to a display menu.

2 Go to the Analyzer Listing menu.

All labels defined in the Analyzer Format menu appear in the listing. If there are more labels than will fit on the screen, the Label/Base field is shaded like a normal field.

3 To scroll the labels, select the Label/Base field and use the knob or press the blue shift key and a page key.

If the Label/Base field is selectable, the roll indicator appears over the field. To move the labels one full screen at a time, press Shift and a Page key.

4 To scroll the data, use the Page keys or select the data roll field and use the knob.

If you select the data roll field, the roll indicator moves to it. No matter which field is currently controlled by the knob, however, the Page keys page the data up or down. The numbers in the data roll column indicate how many samples the data is from the trigger. Negative numbers occurred before the trigger and positive numbers occurred after.

5 If the labels have symbols associated with them, set the base to Symbol.

The symbols you defined appear in the listing.

6 To insert a label, select one of the label fields, then select Insert from the pop-up and the label you want to insert.

The last label cannot be deleted, so there is always at least one label. You can insert the same label multiple times and display it in different bases.

7 To take measurements, select the Markers field and choose the appropriate marker type.

The markers available depend on the type of analyzer and whether or not tagging is enabled. Use markers to locate states quickly.

To compare two listings

The Compare menu allows you to take two state analyzer acquisitions and compare them to find the differences. You can use this function to quickly find all the effects after changing the target system or to quickly compare the results of quality tests with results from a working system.

1 In Analyzer mode, press the Run key to acquire data.

In any mode other than Analyzer, pressing the Run key has no effect. The menus which ignore Run lack the Run field onscreen. In Analyzer mode with Run available, the menu changes to a display menu.

2 Go to the Analyzer Compare menu, select Copy Listing to Reference, and then select Execute.

The Compare menu initially is empty, but when you select Execute the data appears.

3 Set up the other test that you want to compare to the first.

This can be a change to the hardware, or a different system. Do not change the trigger, however, or all the states will be different.

4 Run the test again, then select the Reference listing field to toggle to Difference listing.

5 Select the Find Error field and use the knob to scroll through the errors.

Transferring Files Using the Flexible Disk Drive

Because the flexible disk drive on the HP 1660CP-series logic analyzer will read and write double-sided, double-density, or high-density disks in MS-DOS format, it is a useful tool for transferring data to and from IBM PC-compatible computers as well as transferring data to and from other systems that can read and write MS-DOS format. You can save configuration files, measurement results, and even menu and measurement images from the screen.

To save a configuration

You can save configurations on a 3.5-inch disk or on the internal hard disk for later use. This is especially useful for automating repetitive measurements for production testing.

1 Go to the System Hard Disk or System Flexible Disk menu.

- 2 Set the field under System to Store.
- 3 Select the type of configuration you want to save in the field to the right of Store.
You can save the analyzer configuration, the system configuration, or both.
- 4 Specify a file name into which to save the configuration using the to file field.
- 5 Enter a description for the file using the file description field.
- 6 Select Execute.

If you want to save your file in a directory other than the root, you can select Change Directory from the disk operations field, then type the name of the desired directory in the directory name field or select it from the list of visible directories using the knob.

To load a configuration

You can quickly load a previously saved configuration, so that you will not have to manually set up the measurement parameters.

- 1 Go to the System Hard Disk or System Flexible Disk menu.
Your choice here depends on where you saved the configuration.
- 2 Select the field below System and select Load from the pop-up menu.
- 3 Select the destination from the module list.

"System" loads only settings available under the System menus. "Analyzer" loads data and settings for the analyzer. "All" loads both the system and analyzer configurations. You can only load configuration files to the area from which the configuration was taken. For instance, you cannot load an analyzer configuration file to the system. Thus, if you select System, then select a file that contains only an analyzer configuration, the configuration will fail. The file type field tells you what type of information is in a file. Analyzer configuration files show "166xan_config", and system configurations show "16[6/7]x_cnfg" in the file type field.

- 4 Specify a file name from which to load the configuration using the from file field or scrolling with the knob.
- 5 Select Execute.

To save a trace list in ASCII format

Some screens, such as file lists and trace lists, contain columns of ASCII data that you may want to move to a computer for further manipulation or analysis. You can save these displays as ASCII files. While a screen capture saves only the data shown onscreen, saving the display as an ASCII file captures all data in the list, even if it is offscreen.

- 1 Insert a DOS-formatted 3.5-inch disk in the flexible disk drive.
- 2 Set up the menu you want to capture, or run a measurement from which you want to save data.
Remember that only displays that present lists of textual data can be captured as ASCII files.
- 3 Select Print and choose Print Disk from the pop-up menu.
- 4 Select the Filename field and specify a file name to which the data will be saved.
- 5 Select ASCII (ALL) from the Output Format field.

If the current display contents can not be saved as an ASCII file, this option will not be present in the Output Format field.

- 6 Select Flexible Disk from the Output Disk menu, then select Execute.

To save a screen's image

You can save menus and measurements to disk in one of four different graphical formats.

- 1 Insert a formatted flexible disk in the flexible disk drive.
- 2 Set up the menu whose image you want to capture, or run a measurement from which you want to save data.
- 3 Select Print and choose Print Disk from the pop-up menu.

If the screen contains a pop-up menu, the Print field is not available. Pop-up menus cannot be saved to file unless you are using a controller.

- 4 Select the Filename field and specify a file name to save to.
- 5 Select the Output Format field and choose the output format for the graphics file from the pop-up menu.

Choose one of the following formats:

- B/W TIF is a black-and-white TIFF (Tagged Image File Format), v 5.0.
- GRAY TIF is a grayscale TIFF file in TIFF version 5.0 format.

- PCX is a grayscale PCX file (PCX is the PC Paintbrush and Publisher's Paintbrush format from ZSoft).
 - EPS is a black-and-white Encapsulated PostScript™ file.
- 6 Select Flexible Disk from the Output Disk menu, then select Execute.

To store and time the execution of a subroutine

Most system software of any kind is composed of a hierarchy of functions and procedures. During integration, testing, and performance evaluation, you want to look at specific procedures to verify that they are executing correctly and that the implementation is efficient. The analyzer lets you do this by triggering on entry to the address range of the subroutine and counting the elapsed time since the trigger state.

1 Go to the state analyzer's Trigger menu.

2 Set Count to Time.

3 Define a range term, such as Range1, to represent the address range of the particular subroutine.

You may need to examine the structure of your code to help determine this. If your subroutine calls are really procedure calls, then there is likely to be some code at the beginning of the routine that adjusts the stack for local variable allocation. This will precede the address of the first statement in the procedure. If your subroutine has no local storage and is called by a jump or branch, then the first statement will also be the entry address.

4 Under State Sequence Levels, enter the following sequence specification:

- While storing "no state" TRIGGER on "In_Range1" Occurs 1 Else on "no state" go to level 1
- While storing "In_Range1" Then find "Out_Range1" Occurs 1 Else on "no state" go to level 2
- Store "no state" on "no state" go to level 1

For processors that prefetch instructions or have pipelined architectures, you may want to add part or all of the depth of the pipeline to the start address for In_Range1 to ensure that the analyzer does not trigger on a prefetched but unexecuted state.

Triggering Examples

To trigger on the nth iteration of a loop

Traditional debugging requires print statements around the area of interest. This is not possible in most embedded systems designs, but the analyzer lets you view the system's behavior when a particular event occurs. Suppose that your system behaves incorrectly on the last iteration of a loop, which, in this instance, happens to be the 10th iteration. You can use the analyzer's triggering capabilities to capture that iteration and subsequent processor activity.

1 Go to the state analyzer's Trigger menu.

2 Define the terms LP_START and LP_END to represent the start and end addresses of statements in the loop, and LP_EXIT to represent the first statement executed after the loop terminates.

3 Change State Sequence Level 1's macro to "Find event2 n times after event1 before event3 occurs."

4 In the pop-up, enter the following sequence specification:

- While storing anystate Find "LP_START" "9" times after "LP_END" before "LP_EXIT" occurs.

You should use your value for n-1 instead of "9" in the sequence specification above.

To trigger on entry to a function

This sequence triggers on entry to a function only when it is called by one particular function.

1 Go to the state analyzer's Trigger menu.

2 Define the terms F1_START and F1_END to represent the start and end addresses of the calling function. Define F2_START to represent the start address of the called function.

3 Change State Sequence Level 1's macro to "Find event2 n times after event1 before event3 occurs."

4 In the pop-up menu, enter the following sequence specification:

- While storing anystate Find "F2_START" "1" times after "F1_START" before "F1_END" occurs.

This sequence specification assumes there is some conditional logic in function F1 that chooses whether or not to call function F2. Thus, if F1 ends without the analyzer having seen F2, the sequence restarts.

The specification also stores all execution inside function F1, whether or not F2 was called. If you are interested only in the execution of F1, without the code that led to its invocation, you can change the storage specification from "anystate" to "nostate" for the second sequence term.

To capture a write of known bad data to a particular variable

The trigger specification ANDs the bad data on the data bus, the write transaction on the status bus, and the address of the variable on the address bus.

1 Go to the state analyzer's Trigger menu.

2 Define the terms BAD_DATA, WRITE, and VAR_ADDR to represent the bad data value, write status, and the address of the variable.

3 Under State Sequence Level 1, enter the following sequence specification (use the Combination trigger term):

- While storing "anystate" TRIGGER on "BAD_DATA • WRITE • VAR_ADDR" Occurs "1" Else on "no state" go to level "1"

To trigger on a loop that occasionally runs too long

This example assumes the loop normally executes in 14 ms.

1 Go to the state analyzer's Trigger menu.

2 Define terms LP_START and LP_END to represent the start and end addresses of the loop, and set Timer1 to the normal duration of the loop.

3 Change State Sequence Level 1's macro to "Find event2 occurring too late after event1."

4 In the pop-up menu, enter the following sequence specification:

- While storing anystate Find "LP_END" occurring too late after "LP_START" Use Timer: "Timer1" Time="14 ms" Of course, you use your normal loop duration in place of "14 ms." The macro will automatically start Timer1 for you.

To verify correct return from a function call

The exit code for a function will often contain instructions for deallocating stack storage for local variables and restoring registers that were saved during the function call. Some language implementations vary on these points, with the calling function doing some of this work, so you may need to adapt the procedure to suit your system.

1 Go to the state analyzer's Trigger menu.

2 Define terms SR_START and SR_END to represent the start and end addresses of the subroutine.

3 Under State Sequence Levels, insert 2 more sequence levels and enter the following sequence specification:

- While storing "anystate" Find "SR_START" Occurs "1" Else on "no state" go to level "1"
- While storing "anystate" Then find "SR_END" Occurs "1" Else on "no state" go to level "2"
- While storing "anystate" TRIGGER on $\forall \neq$ SR_START" Occurs "1" Else on "SR_START" go to level "2"