Help Volume

Instrument: HP 16522A 200 MHz Pattern Generator

The HP 16522A Pattern Generator is used by digital design teams to emulate digital signals in circuits under development. The pattern generator can take the place of missing devices, or can act as a stimulus to functionally test prototypes.

Getting Started

- "Overview of the HP 16522A Pattern Generator" on page 12
- "A Beginner's Exercise" on page 19

Creating the Program

- "Building an Initialization Sequence" on page 22
- "Building a Main Sequence" on page 24
- "Building a User Macro" on page 26
- "Importing HP 16522A ASCII Files" on page 29
- "Importing System Data Files" on page 40
- "Loading and Saving Pattern Generator Configurations" on page 43

See Also

"Selecting the Correct Probe Pod" on page 44

"Connecting the Probe Pods" on page 48

"Editing Sequences" on page 50

"Working with Instruction Types" on page 56

"Working with Labels and Pods" on page 66

"Working with Macro Parameters" on page 79

"Working with Automatic Pattern Fills" on page 82

"Loading and Saving Pattern Generator Configurations" on page 43

"Printing the Pattern Generator Window" on page 89

"Printing Vector Sequences to a File" on page 90

Contents

Contents

Contents

Contents

1

Using the HP 16522A Pattern Generator

Overview of the HP 16522A Pattern Generator

Description of the HP 16522A Pattern Generator

The HP 16522A Pattern Generator is a tool that generates digital signals. It is used in applications that require an external source to simulate digital circuitry or generate digital signals for functionally testing prototype hardware.

Combined with the analog and digital measurement capabilities of the logic analysis system, you have a tightly integrated solution to your digital stimulus and response measurement needs.

A Conceptual Measurement Example

The exact output pattern, clock type and speed, and number of required signals depends on your specific application. How you configure the pattern generator and what kind of signal generation sequence you create will vary. However, from a procedural standpoint, the steps are the same each time to set up, create a sequence, and run the pattern generator.

1. Select the probing (see page 44) that is compatible with your target circuit.
2. Set the Vector Output mode (see page 14) and the Clock Source (see page 15) parameters.
3. Connect the probes (see page 48) to your circuit and map the probe channels (see page 13) into the interface of the pattern generator.
4. Build a sequence of test vectors (see page 17) to generate the desired output signals.
5. Run (see page 18) the pattern generator and measure the active target circuit or prototype for the desired results.

Re-using Pattern Generator Programs

After you set up a pattern generator configuration, you may want to store it away so you can use it again. Perhaps you want to create a set of test routines or circuit simulators. There are two ways to handle re-

usable configurations.

- You can reload previously saved (see page 43) pattern generator configurations.
- You can import an HP 16522A ASCII file (see page 29)

See Also

"A Beginner's Exercise" on page 19

"Theory of Operation" on page 92

"Key Characteristics" on page 95

"Help - How to Navigate Quickly" on page 104

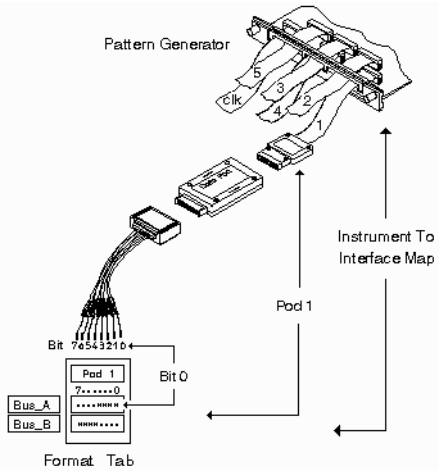
Mapping Probe Pods to the Interface

While the probes make the physical connection to your target circuit, a software connection is also made within the interface which routes generated signals to the proper probe output lines. This software connection is done within the Format tab and is called mapping.

The mapping process consists of logically grouping output signals that have a similar purpose to a label (see page 66) with a unique name. To add to or delete signals from a group, you simply turn On/Off the bits (see page 72) beside the label.

Example

This example shows eight channels (or bits) on probe pod 1 mapped to two labels in the interface. Bits 0-3 are assigned to label Bus_A, and bits 4-7 are assigned to label Bus_B. When a bit is assigned, an asterisk "*" appears in the bit assignment field, verifying the software connection.



NOTE: After you set up your first measurement, use the ribbon cable ID clips to mark the data cable numbers for future reference.

Vector Output Mode

The Vector Output Mode determines the channel width, available pods, and the frequency range for both the internal and external clock. The choice you make may be determined by trade-offs between clock speed and channel width.

Because the output mode affects clock frequency ranges, available pods, and channel width, keep your mode selection in mind when designing the circuit's hardware interface and when mapping probe connections between the test circuit and the labels of the pattern generator.

This table shows the difference between the Full-Channel 100 MBits/s mode and the Half-Channel 200 MBits/s mode.

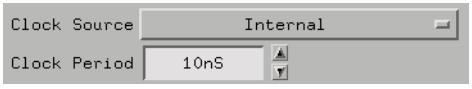
	Full Channel 100 MBits/s	Half Channel 200 MBits/s
Pods Available	Pods 1, 2, 3, 4, 5	Pods 1, 3, 5
Maximum Channels	40; eight per pod	20; eight on pods 1, 3, and lower 4 on pod 5
Maximum External Clock Frequency	100 MHz	200 MHz
Maximum Internal Clock Frequency	100 MHz	200 MHz
Minimum External Clock Frequency	DC	DC
Minimum Internal Clock Frequency	4 kHz	4 kHz

Clock Source

The Clock Source field toggles between internal and external. The internal clock source is supplied by the pattern generator and controls the frequency used to output the vectors to the system under test. The external clock is provided by the user or the system under test, and is input to the pattern generator through the CLK IN probe of a clock pod.

An advantage of using an external clock is that you synchronize the vector output of the pattern generator to the system under test. No matter which clock source is used, vectors are always output on the rising edge of the clock.

Internal Clock Source



Use an internal clock source when you want to have control over the frequency of the output vectors and it is not important for the output vectors to be synchronized to the system under test.

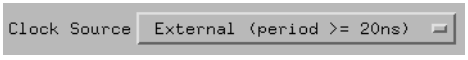
You select clock periods in steps of 1, 2, 2.5, 4, 5, 8, and 10. If you use

the keypad to select a value between the step intervals, the value is rounded to the nearest interval.

NOTE: If you use the keypad to change the the clock period value, you must press the *Enter key* to register the new value.

The minimum clock period available with Vector Output Mode at Full Channel 100Mbit/s is 10 ns. The minimum clock period available with Vector Output Mode at Half Channel 200Mbit/s is 5 ns. Maximum clock period for either mode is 250 us.

External Clock Source



Use an external clock source when you want to synchronize the frequency of the output vectors to the system under test. With this mode selected, you do not have direct control over the frequency of the output vectors. Output vector frequency will be the same as the external clock.

When using an external clock source, be sure to set the clock period range to match the period of the external clock source. If the Vector Output Mode is Full Channel 100Mbit/s, you are offered two clock period ranges:

- Greater than or equal to 20 ns
- Greater than or equal to 10 ns but less than 20 ns

If the Vector Output Mode is Half Channel 200 Mbit/s, you are offered three clock period ranges:

- Greater than or equal to 20 ns
- Greater than or equal to 10 ns but less than 20 ns
- Less than 10 ns

CAUTION: If the external clock is faster than the period range selected, the HP 16522A will produce erroneous output vectors.

Clock Out Delay

The clock out delay setting lets you position the output clock with respect to the data. The zero setting is uncalibrated and should be measured to determine the initial position with respect to the data. Each numerical change of one on the counter results in an approximate change of 1.3 ns.

See Also

"Clock Circuit PGTheory" on page 93

Building a Sequence of Test Vectors

Test vectors determine the pattern output at each clock cycle. Test vectors are positioned in a list called a sequence. When a sequence is run (see page 18), the list of vectors is executed in order of first vector to last vector. Vectors are always output on the rising edge of the clock.

In every pattern generator application, you have two sequences. An INIT SEQUENCE (initialization sequence) is used to place your circuit or subsystem in a known state. The initialization sequence is followed by the MAIN SEQUENCE. The main sequence is used for the actual pattern generation that stimulates your circuit under test. The INIT sequence is only run once, while the MAIN sequence loops if you select a repetitive run.

Using Hardware and Software Instructions

In addition to test vectors, both INIT and MAIN sequences can include predefined instruction (see page 56) elements. Instructions can create Breaks, Loops, Wait and If Events, and can even signal the Intermodule Bus. The most useful instruction is "*User Macro*". With a User Macro instruction, you can create reusable sequences that accept parameters. This flexibility is very useful in prototype turn-on and environmental testing.

For more information on INIT and MAIN sequences and how to create them, see the following topics.

"Building an Initialization Sequence" on page 22

"Building a Main Sequence" on page 24

"Building a User Macro" on page 26

"Working with Instruction Types" on page 56

Running the Pattern Generator

If you are not changing the run options, simply point to the *Group Run* field and click the *left mouse button* to run a measurement. To cancel a *Repetitive* run, click the *Cancel* field.

See Also

"Run/Group Run Function" on page 105

"What Happens when Run is Selected" on page 18

"What Happens when Stop is Selected" on page 18

What Happens when Run is Selected

In single run mode, the vectors are output from the first vector in the initialization sequence to the last vector of the main sequence. The last vector of the main sequence will be held at the outputs until you run again.

In repetitive run mode, the vectors in the initialization sequence will be output from first to last, one time, then the main sequence will repetitively output the vectors in that sequence until you press the *Stop* field.

What Happens when Stop is Selected

When the pattern generator acknowledges stop, the vector currently being output will be held at the outputs until you run again.

A Beginner's Exercise

This exercise begins with the pattern generator Format tab active. If it is not active, click *Format* in the pattern generator window now.

In this exercise, you will create a label with eight output channels assigned to it. You will then create a "Walking Ones" output pattern using one of the automatic pattern fill functions.

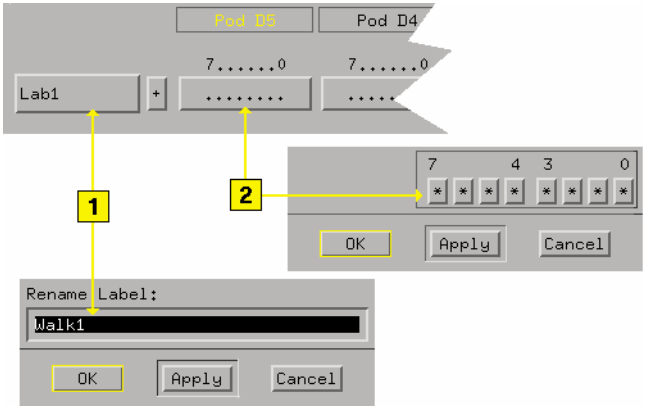
NOTE: This exercise does NOT require you to connect the probes or view the output. The intent of this exercise is to give you practice configuring the pattern generator interface. A timing analyzer display of the results is furnished for you.

1. Configure Format (see page 19) with a label called "Walk1" and all eight bits of Pod 5 assigned.
2. Click *Sequence*.
3. Configure Sequence (see page 20) with a Walking Ones sequence.
4. View the results. (see page 21)

See Also "Help - How to Navigate Quickly" on page 104

Configure Format

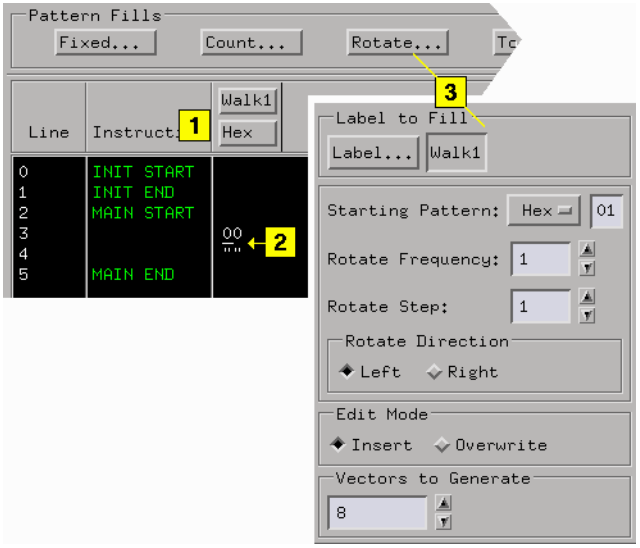
1. In the pattern generator's Format area, left-click the label *Lab1*, then select *Rename*. In the Rename dialog, type "Walk1", then click *OK*.
2. Left-click the bit assignment field under Pod 5 and set all eight bits to "*" (on) by clicking them, then click *OK*.



See Also "Help - How to Navigate Quickly" on page 104

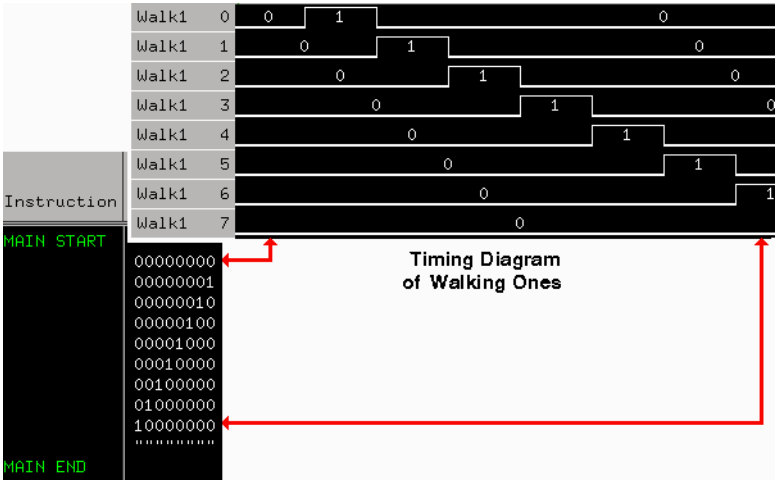
Configure Sequence

1. In the pattern generator's Sequence area, right-click *Hex* and set the numeric base to Binary.
2. Left-click the first sequence line. This positions the cursor.
3. Left-click *Rotate*, and configure the Rotate Pattern Fill dialog as shown. Click *Fill*.



See Also "Help - How to Navigate Quickly" on page 104

View the Results



See Also "Help - How to Navigate Quickly" on page 104

Building an Initialization Sequence

The initialization (INIT) sequence is the first of two vector sequences that appear in the Sequence display. Use the INIT sequence to put the circuit or subsystem into a known starting condition. You can also use the INIT sequence to arm a logic analyzer or oscilloscope with the Signal IMB instruction to begin a measurement when the MAIN sequence begins. If you leave the INIT sequence empty, it will be ignored.

What Happens when Run is Selected

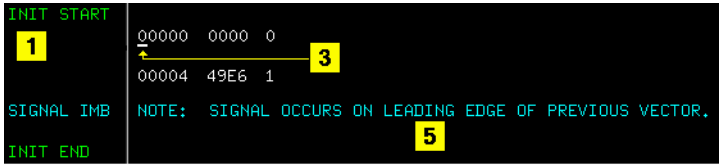
In single run mode, the vectors are output from the first vector in the initialization sequence to the last vector of the main sequence. The last vector of the main sequence will be held at the outputs until you execute run again.

In repetitive run mode, the vectors in the initialization sequence will be output from first to last, one time, then the main sequence will repetitively output the vectors in that sequence until you select *Stop*.

Building the INIT Sequence

The INIT sequence can contain hardware and software instructions (see page 56) as well as vector data. However, instructions are not allowed on the first two vector lines.

1. *Click* the *Sequence* tab, *point* to INIT START and left-click the mouse. This positions the cursor.
2. Press the Insert key on your keyboard one time for each new vector line you want to insert.
3. Point to the left-most character in the new vector line and left-click the mouse. This positions the cursor.
4. Type in the desired vector data. As you type, the default cursor wrap (see page 97) setting will roll the cursor left-to-right and top line to bottom line.
5. Optional - If applicable, insert an instruction (see page 56) instead of typing vector data.



NOTE: With the Vector Output Mode of *Half Channel 200 Mbit/s*, the INIT sequence must contain a number of vectors that is divisible by four. If this is not the case, the first vector of the INIT sequence is duplicated to create the correct number of vectors. With the Vector Output Mode of *Full Channel 100 Mbit/s* and vector frequencies greater than 50 MHz, the INIT sequence must contain an even number of vectors. Again, if this is not the case, the first vector of the INIT sequence is duplicated to create the correct number of vectors.

See Also

"Working with Labels and Pods" on page 66

"Editing Sequences" on page 50

"Working with Instruction Types" on page 56

"Building a User Macro" on page 26

"Automatic Cursor Wrap" on page 97

Building a Main Sequence

The MAIN sequence is the second of two vector sequences that appear in Sequence. Use the MAIN sequence as the primary signal generation sequence. The MAIN sequence must contain at least two vectors to output.

What Happens when Run is Selected

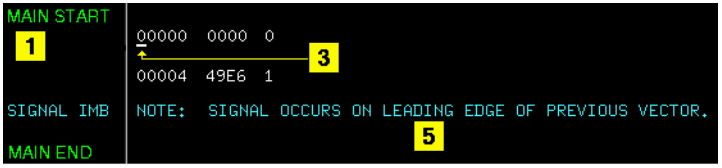
In single run mode, the vectors are output from the first vector in the initialization sequence to the last vector of the main sequence. The last vector of the main sequence will be held at the outputs until you select run again.

In repetitive run mode, the vectors in the initialization sequence will be output from first to last, one time, then the main sequence will repetitively output the vectors in that sequence until you select the *Stop* field.

Building the Main Sequence

The MAIN sequence can contain hardware and software instructions (see page 56) as well as vector data. However, instructions are not allowed on the first two vector lines or the last vector line.

1. *Click* the Sequence tab, *point* to MAIN START and left-click the mouse. This positions the cursor.
2. Press the Insert key on your keyboard one time for each new vector line you want to insert.
3. Point to the left-most character in the new vector line and left-click the mouse. This positions the cursor.
4. Type in the desired vector data. As you type, the default cursor wrap (see page 97) setting will roll the cursor left-to-right and top line to bottom line.
5. Optional - If applicable, insert an instruction (see page 56) instead of a typing vector data.



See Also

"Working with Labels and Pods" on page 66

"Editing Sequences" on page 50

"Working with Instruction Types" on page 56

"Building a User Macro" on page 26

"Automatic Cursor Wrap" on page 97

Building a User Macro

A User Macro is a vector sequence defined by a custom name, then inserted by name into a sequence wherever the macro is needed. Macros may be inserted into the INIT or MAIN sequences of the vectors in Sequence, or into other macros. Using macros gives you the benefit of keeping INIT or MAIN sequences generic. By simply interchanging macros, you change the pattern generator output.

NOTE: Care should be taken to avoid infinite loops. For example, if macro 0 calls macro 1, and macro 1 calls macro 0, this will cause an infinite loop.

Macros can also accept parameters (see page 79). A major benefit in using parameters is that you keep a macro's functionality generic and still direct specific action identified by parameters. Think of a parameter as the only part of a macro that changes as the macro is reused. Each macro can accept a maximum of 10 unique parameters.

Typically, you create a macro first under the *Macro* tab, then insert it into sequences under the *Sequence* tab. You can create 100 different macros for use in one or more pattern generator sequences.

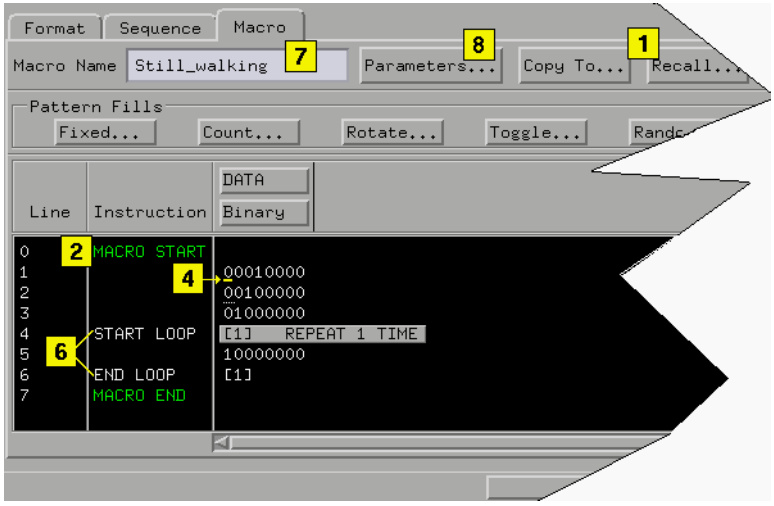
Differences between User Macros and the INIT and MAIN sequences are that macros cannot use the "If" instruction or any instruction that interacts with the intermodule bus (IMB). The reason is that these instructions can only be included once into the sequence. Since macros may be called as many times as desired, allowing these instructions within macros would violate this restriction. You remove macros from sequences by using the *Delete Line(s)* function.

Creating the Macro

A macro sequence can contain hardware and software instructions (see page 56) as well as vector data. However, instructions are not allowed on the first vector line.

1. In Macro, recall (see page 98) the macro that you want to create.
2. *Point* to MACRO START and left-click the mouse. This positions the cursor.

3. Press the Insert key on your keyboard one time for each new vector line you want to insert.
4. Point to the left-most character in the new vector line and left-click the mouse. This positions the cursor.
5. Type in the desired vector data. As you type, the default cursor wrap (see page 97) setting will roll the cursor left-to-right and top line to bottom line.
6. Optional - Insert an instruction (see page 56) instead of a typing vector data.
7. Type in a name for the new macro.
8. Optional - Click *Parameters* and turn on any parameters you plan to use.



Inserting the Macro

1. In *Sequence* or *Macro*, point to the vector line directly above where you want to insert the User Macro instruction.
2. Press and hold the right mouse button while pointing to *Insert After*. Release the right mouse button.
3. Point to the User Macro instruction and right-click.
4. From the Macro Selection dialog that appears, left-click the desired macro name you want to insert.

5. Click *OK*.

NOTE: See The User Macro Instruction (see page 63) for restrictions on User Macro instruction usage.

See Also

"Recalling Macros" on page 98

"Copying Macros" on page 100

"Working with Macro Parameters" on page 79

"Working with Instruction Types" on page 56

"Editing Sequences" on page 50

"Working with Labels and Pods" on page 66

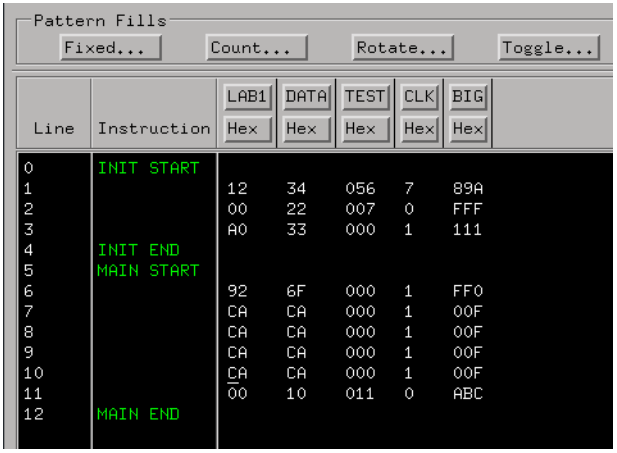
Importing HP 16522A ASCII Files

You can create an ASCII text file and import it into as a complete pattern generator program. In general, the ASCII file consists of a block of setup information, a block of label and channel information, and a block of pattern generator vector data. The file must be saved in ASCII format and organized as shown in step 1 of the procedure below.

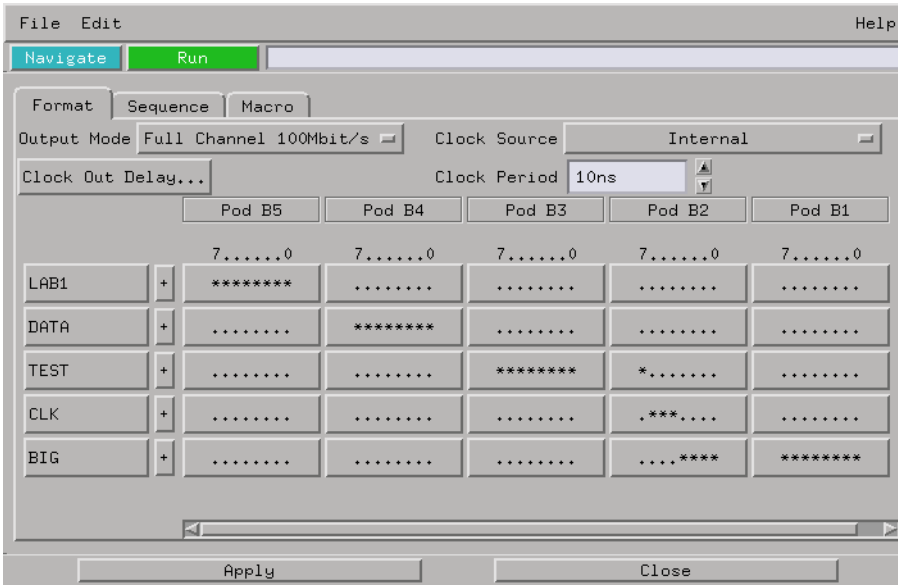
1. Create the ASCII file (see page 31) in a text editor.
2. Save the file as *Text Only* or *ASCII Format* in a directory on your analyzer's hard drive.
3. In Sequence's menu bar, select *File*, then *Import 16522A ASCII File*. See the caution below.
4. From the file selection dialog that appears, select the desired path and ASCII file name.
5. Click *Import*.

CAUTION: Importing an HP *16522A ASCII* file causes all current Format and Sequence information to be overwritten. Be sure to save the pattern generator configuration before you begin the import process.

This figure shows Sequence after the ASCII file example shown in step 1 was imported.



This figure shows Format after the ASCII file example shown in step 1 was imported.



Creating an ASCII File

You can create an ASCII file using any MS-DOS or UNIX text editor. An ASCII file consists of a file identifier and three blocks of information. Each block must follow the specified order.

- `ASCII 000000` - required file identifier ("ASCII" followed by 5 spaces and 6 zeros).
  `ASCDown` - optional. Retained for backwards compatibility.

- 1st block (optional)

  `FORMat` - clock, channel mode, and delay information.

- 2nd block

  `LABel` - names and number of channels.

- 3rd block

  `VECTor` - vector data and Repeat indicators.

File Requirements and Precautions

- The file must contain only specified pattern generator commands (see page 33), and in the order and format shown in the example below.
- The file must be saved in "ASCII" or "text only" format.
- Vector data is assumed to be entirely hexadecimal base.
- No pattern generator instructions are allowed in the data.
- No pattern generator macros are defined or invoked in the data.
- All labels consist of adjacent bits.
- The file must end with a line termination character (line feed "<lf>" or a carriage return and line feed "<CR><lf>".
- Comments can be included after the first line (ASCII 000000). Comments begin with a slash '/' and terminate at the end of the line.

ASCII File Example

NOTE: In this example, the underlined links are added for documentation purposes only, and would NOT be part of an actual disk file's text. Line feeds "<lf>" are shown for example purposes only, and depending on the computer and text editor used, could actually be a carriage return followed by a line feed "<CR><lf>".

```
ASCII 000000 (see page 33)<lf>
/
/ This is a test of the ascii file
/
ASCDown (see page 34)<lf>
FORMat:MODE FULL (see page 37)<lf>
FORMat:CLOCk INTernal, 10E-9 (see page 37)<lf>
LABel LAB1,8 (see page 34)<lf>
LABel DATA,8<lf>
LABel TEST,9<lf>
LABel CLK,3<lf>
LABel BIG,12<lf>
/*
/* This is the beginning of the vector part
/*
VECTor (see page 35)<lf>
12 34 056 7 89A<lf>      / Some vectors
0 22 007 0 FFF<lf>
A0 33 000 1 111<lf>      /* Some more vectors */
*M<lf>
92 6F 000 1 FF0<lf>
CA CA 000 1 00F<lf>      // Even more vectors
*R 3<lf>
00 10 011 0 ABC<lf>
```

NOTE: The *LABel* sequence specified in the 8th through 12th lines results in a specific bit assignment. A different ordering of the LABel commands would give a different ordering to the bits.

ASCII Disk File Identifier

The first line of a disk file must contain the text string "`ASCII 000000`". It consists of the text "`ASCII`" followed by 5 blanks, then 6 zeros. The purpose of this string is to uniquely identify the file as an ASCII disk file.

Example

```
ASC11      000000
ASCDOWN
FORMAT: MODE FULL
LABEL LAB,8
LABEL DATA,8
LABEL "TEST",9
LABEL CLK,3
LABEL BIG,12
VECTOR
12 34 56 7 89A
0 22 7 0 FFF
A0 33 00 1 111
*M
92 6F 00 1 FF0
CA CA 00 1 00F
00 10 11 0 ABC
```

ASCII File Commands

The following commands are used in the ASCII file to configure Sequence and Format. Commands are not case sensitive. Lowercase letters in an illustrated command simply show the long and short form difference.

The uppercase letters of the command show the mandatory portions of each command.

Commands

DELay (see page 38)

"LABel Command" on page 34

"VECTor Command" on page 35

ASCDown Command

The ASCDown command was formerly used to signal the start of an ASCII file load. It causes the current pattern generator label and sequence structures to be cleared and reset to a default state. The HP 16522A pattern generator now does this automatically.

Example

```
ASC11     000000
ASCDOWN
FORMAT: MODE FULL
LABEL LAB,8
LABEL DATA,8
LABEL "TEST",9
LABEL CLK,3
LABEL BIG,12
VECTOR
12 34 56 7 89A
0 22 7 0 FFF
A0 33 00 1 111
*M
92 6F 00 1 FF0
CA CA 00 1 00F
00 10 11 0 ABC
```

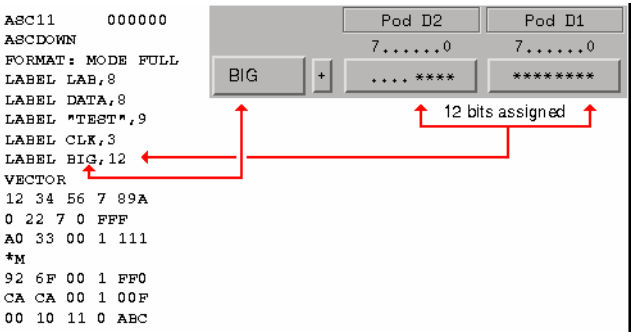
LABel Command

The LABel command is a special means of specifying labels for use by an ASCII file. The label bits are assigned from most to least significant bits across the output pods. You must specify the label string (quotation marks on the string are optional) and the width of the field. The label base defaults to hexadecimal. There are a maximum of 126 labels. No label may be more than 32 bits wide. If a label is too wide (too many bits) for the remaining unused pattern generator bits, it will be discarded.

Command Syntax

`LABel <name_str>,<width>`
`<name_str>` = label string a maximum of 20 characters in length.
`<width>` = integer number of bits in the label (1 through 32).

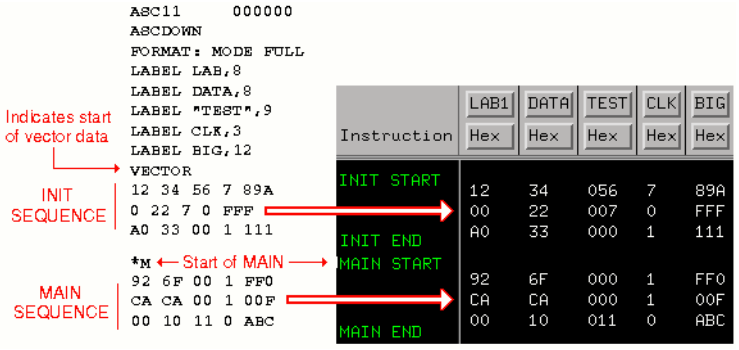
Example



VECTor Command

The VECTor command is used after the end of the header/setup commands to signal the start of the actual pattern generator data in an ASCII file. No data is allowed in the same line as the VECTor command.

Example



Vector Data

The data portion of the ASCII file is an array of hexadecimal data fields. Each row of the array corresponds to a single line of the main program. Each column of the array corresponds to a single label as defined under the pattern generator Format tab.

Data fields are separated by one or more blank characters. A line termination (line feed or carriage return + line feed) signals the end of

a line and the start of a new line. If a data field has more data than the label width would indicate, only the least significant bits of the data field are used. If there are more data fields in a row than there are labels, the extra data fields (last data fields in the row) are ignored. If there are fewer data fields in a row than there are labels, the data for the extra (right-most) labels will be zero.

The MAIN sequence must have at least two data lines. In the ASCII data file, a row consisting of only "*M" signals the start of the MAIN sequence. If there is to be no data in the INIT sequence, the first row of the file after the VECTor command must be "*M". Note that the quotation marks in "*M" are not really in the file.

CAUTION: Any character that is not a valid hexadecimal digit (that is, 0 through 9, or upper/lower case A through F) are ignored and treated as field separators. This could cause problems if a mistyped character appears in the middle of a data value. For example, "12R4" will be assigned to two labels as "12" and "4".

Either of the INIT or MAIN sequences can include multiple Repeat indicators specified by "*R <count>". Note that the quotation marks around the Repeat indicator are not part of the indicator, and like the "*M", the "*R <count>" must be located on a separate line.

The Repeat indicator specifies that the previous data vector is repeated <count> more times, where <count> is a positive integer. The Repeat indicator must follow a sequence line containing a data vector or another Repeat indicator. Placing the Repeat indicator after the VECTor command or the "*M" will cause an error message to appear.

The last data row of the file must end with a line termination character. The line termination character is the flag to load the data row into the data structure. Failure to do this will result in the last data row not being loaded.

The ASCII file import mechanism assumes correctness in the data file and any header commands. Error handling is basic, and treating unexpected characters as field separators could create bizarre results when parsing the file. Error messages point to the line number where the parser finds the error.

Serious problems will cause the default main program to be loaded in an effort to avoid locking up the logic analysis system.

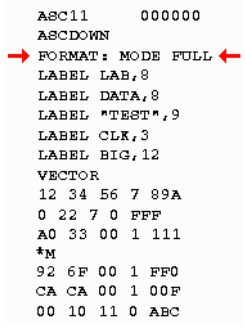
FORMat:MODe Command

The FORMat:MODe command is optional. The existing mode scheme is used if nothing is specified. FULL channel output mode limits the data rate to a maximum of 100 MHz, but allows 40 channels per card. HALF channel output mode allows an output rate of greater than 100 MHz, but limits the number of channels to 20 per card.

Command Syntax

```
FORMat:MODe [FULL|HALF]
```

Example



FORMat:CLOCk Command

The FORMat:CLOCk command is optional. The existing clock scheme is used if nothing is specified. The CLOCk command specifies the clock source for the pattern generator.

Command Syntax

`FORMat:CLOCk INTernal, <clk_period>`
= a real number value that corresponds to the interface selectable clock period values (example = 5E-9).

`FORMat:CLOCk EXTernal, [LEFifty|GTFifty|GTONe]`
`[LEFifty]` = Less than or equal to 50 MHz.
`[GTFifty]` = Greater than 50 MHz and less than or equal to 100 MHz.
`[GTONe]` = Greater than 100 MHz.

NOTE: The maximum clock rate is limited by the channel mode. See FORMat:MODe (see page 37) command.

Example

```
ASC11     000000
ASCDOWN
FORMAT: MODE FULL
FORMAT: CLOCK INTERNAL, 10E-9
LABEL LAB,8
LABEL DATA,8
LABEL "TEST",9
LABEL CLK,3
LABEL BIG,12
VECTOR
12 34 56 7 89A
0 22 7 0 FFF
A0 33 00 1 111
*M
92 6F 00 1 FF0
CA CA 00 1 00F
00 10 11 0 ABC
```

FORMat:DELay Command

The FORMat:DELay command is optional. The existing delay scheme is used if nothing is specified. The DELay command specifies the clock out delay. The clock out delay setting allows positioning of the clock with respect to the data. Delay setting range is 0 - 9 with each increment delaying the clock approximately 1.3 ns per step.

NOTE: The delay setting that corresponds to zero is uncalibrated. You must measure it to determine the basic clock/data timing.

Command Syntax

`FORMat:DELay <delay_arg>`
`<delay_arg>` = an integer from 0 to 9 with each increment delaying the clock by approximately 1.3 ns.

Example

```
ASC11     000000
ASCDOWN
FORMAT: MODE FULL
FORMAT: CLOCK INTERNAL, 10E-9
FORMAT: DELAY 2
LABEL LAB,8
LABEL DATA,8
LABEL "TEST",9
LABEL CLK,3
LABEL BIG,12
VECTOR
12 34 56 7 89A
0 22 7 0 FFF
A0 33 00 1 111
*M
92 6F 00 1 FF0
CA CA 00 1 00F
00 10 11 0 ABC
```

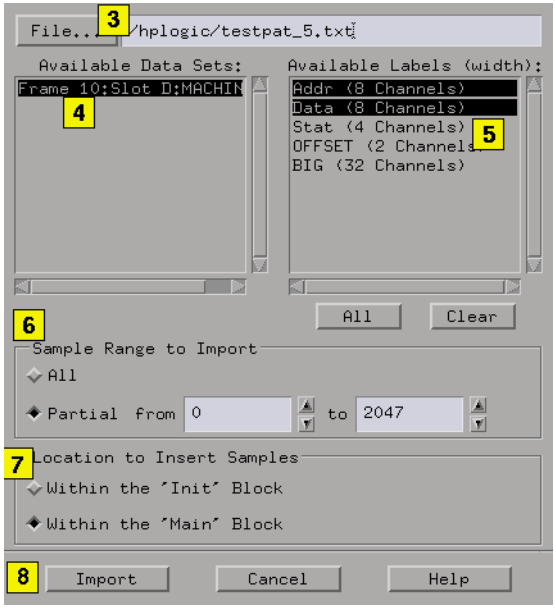
Importing System Data Files

If you store logic analyzer data using the File Out tool, in either Internal, ASCII, or Fast Binary format, you can import these files into the pattern generator's INIT or MAIN sequences.

The "Import System Data File" dialog will only load files saved using the File Out tool. For ASCII files created in a PC or UNIX text editor, use Import 16522A ASCII File (see page 29).

CAUTION: Importing a *File Out* tool file causes all current Format and Sequence information to be overwritten. Be sure to save the pattern generator configuration before you begin the import process.

1. *Click* the Sequence tab.
2. Select *File* from the menu bar, then *Import System Data File*.
3. From the Import System Data File dialog, load the desired File Out tool file. See the note below.
4. Select the desired data set (see page 41).
5. Select the desired labels (see page 41).
6. Select a data set range (see page 42) to import. If you select "Partial", the range integers you specify will correspond to state numbers found in a state listing of the same data set.
7. Select the location to insert the samples. Samples are inserted at the beginning of the selected sequence. See the caution below.
8. Click Import.



NOTE: If in step 3 you typed the file name into the text entry field, you must press the Return key on your keyboard to start the file import process instead of clicking the *Import* field.

Data Sets

A data set is defined as data captured from a single source. For example, the data captured from machine one of a logic analyzer is a single source. Because the File Out tool allows multiple machines worth of data to be stored within the same file, you pick only one data set to import into the pattern generator.

Data Set Labels

All labels in a single data set are listed along with each label's corresponding channel width. If a label is wider than the pattern

generator's maximum allowable channel width (32 channels), the label is marked "unavailable". Labels wider than 32 channels cannot be selected.

Data Set Range

When you select a data set, the "from" and "to" fields update to the minimum and maximum values of the state listing. Both positive and negative integers are valid.

CAUTION: If the range of data samples is too large for the pattern generator sequence (258048 vectors), a message will appear giving you the choice to continue or not. If you continue, data will be imported starting from the beginning of the file. At the point where you run out of vectors, data will be missing. To solve this problem, reduce the range of samples you are importing.

Loading and Saving Pattern Generator Configurations

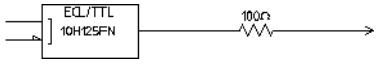
You can save pattern generator settings and data to a configuration file. You can also save any tools connected to the pattern generator. Later, you can restore your data and settings by loading the configuration file.

- Loading Configuration Files (see the *HP 16600A/16700A Logic Analysis System* help volume)
- Saving Configuration Files (see the *HP 16600A/16700A Logic Analysis System* help volume)

Selecting the Correct Probe Pod

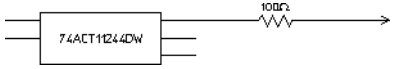
The following equivalent circuit information is provided to help you select the appropriate clock and data pods for your application.

HP 10461A TTL Data Pod



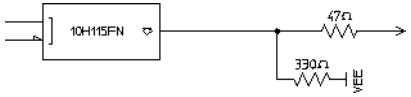
Output type	10H125 with 100 ohm in series
Maximum clock	200 MHz
Skew	Typical less than 2 ns; worst case 4 ns (note 1)
Recommended lead set	HP 10474A

HP 10462A 3-State TTL/CMOS Data Pod



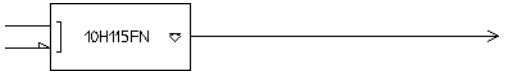
Output type	74ACT11244 with 100 ohm in series
3-state enable	10H125 on non 3-state channel 7 (note 2), negative true, 100K ohm to GND, enabled on no connect.
Maximum clock	100 MHz
Skew	Typical less than 4 ns; worst case 12 ns (note 1)
Recommended lead set	HP 10474A

HP 10464A ECL Data Pod (terminated)



Output type	10H115 with 330 ohm pulldown, 47 ohm in series
Maximum clock	200 MHz
Skew	Typical less than 1 ns; worst case 2 ns (note 1)
Recommended lead set	HP 10474A

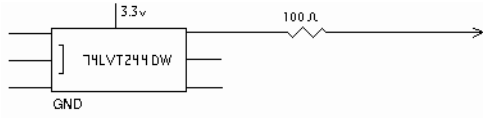
HP 10465A ECL Data Pod (unterminated)



Output type	10H115 (no termination)
Maximum clock	200 MHz

Skew	Typical less than 1 ns; worst case 2 ns (note 1)
Recommended lead set	HP 10347A

HP 10466A 3-State TTL/3.3 volt Data Pod



Output type	74LVT244 with 100 ohm in series 10H125 on non 3-state channel 7 (see note 2)
3-state enable	negative true, 100K ohm to GND, enabled on no connect.
Maximum clock	200 MHz
Skew	Typical less than 3 ns; worst case 7 ns (note 1)
Recommended lead set	HP 10474A

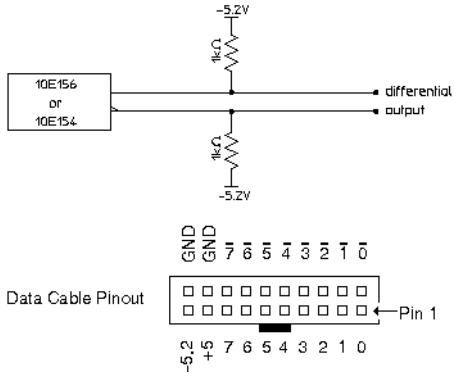
Note 1

Typical skew measurements made at the pod connector with approximately 10 pF/50K ohm load to GND; worst case skew numbers are a calculation of worst case conditions through circuits. Both numbers apply to any channel within a single or multiple module system.

Note 2

Channel 7 on the 3-state pods has been brought out in parallel as a non 3-state signal. By looping this output back into the 3-state enable line, the channel can be used as a 3-state enable.

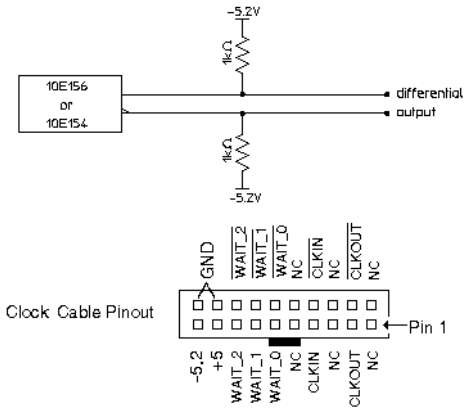
Data Cable Characteristics without a Data Pod



The HP 16522A data cables without a data pod provide an ECL-terminated (1K ohm; to -5.2V) differential signal. These signals are usable when received by a differential receiver, preferably with a 100

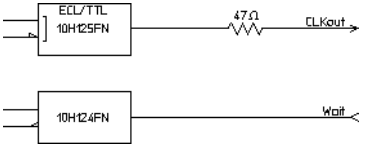
ohm termination across the lines. These signals should not be used single-ended due to the slow fall time and shifted voltage threshold; they are not ECL compatible.

Clock Cable Characteristics without a Clock Pod



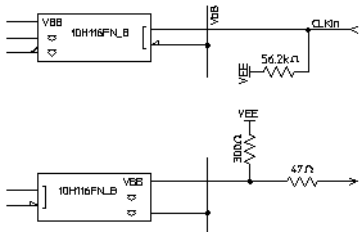
The clock out signals (CLKOUT and not-CLKOUT) without a clock pod provide an ECL-terminated (1K ohm to -5.2V) differential signal. These signals are usable when received by a differential receiver, preferably with a 100 ohm termination across the lines. These signals should not be used single-ended due to the slow fall time and shifted voltage threshold; they are not ECL compatible.

10460A TTL Clock Pod



Clock output type	10H125 with 47 ohm series; true & inverted
Clock output rate	100 MHz maximum
Clock out delay	11 ns maximum in 9 steps
Clock input type	TTL - 10H124
Clock input rate	DC to 100 MHz
Pattern input type	TTL - 10H124 (no connect is logic 1)
Clk-in to clk-out	Approx. 30 ns
Patt-in to recognition	Approx. 15 ns + 1 clk period
Recommended lead set	HP 10474A

10463A ECL Clock Pod



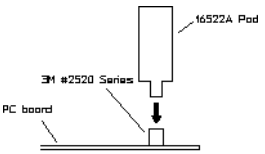
Clock output type	10H116 differential unterminated; differential with 330 ohm to -5.2v and 47 ohm series
Clock output rate	200 MHz maximum
Clock out delay	11 ns maximum in 9 steps
Clock input type	ECL - 10H116 with 50K ohm to -5.2v
Clock input rate	DC to 200 MHz
Pattern input type	ECL - 10H116 with 50K ohm (no connect is logic 0)
Clk-in to clk-out	Approx. 30 ns
Patt-in to recognition	Approx. 15 ns + 1 clk period
Recommended lead set	HP 10474A

See Also "Connecting the Probe Pods" on page 48

Connecting the Probe Pods

NOTE: Clock and Data pods are required for a proper signal interface. (see page 94) There are different types (see page 44) available, and they should match your target circuit characteristics. In addition, depending on the Vector Output Mode (see page 14) selected, some pods may not be available for use.

Direct Pod-to-Board Connection



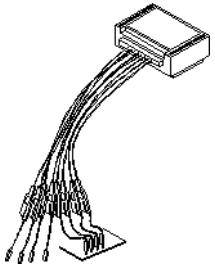
Plug the pod directly into the ©3M 2520-series, or similar alternative connector on the PC board.

Jumper Cable-to-Pod Connection

Use this method when you have clearance problems on the PC board. Construct a flat-ribbon cable and connect as shown above.

NOTE: You can obtain equivalent connectors from sources other than 3M.

Probe Lead Set to Board Pin Connection



Two probe lead assemblies are available for connecting to PC board pins.

- HP 10474A 8-channel probe lead set.
- HP 10347A 8-channel probe lead set, 50-ohm coaxial for unterminated signals.

The probe tips of both lead sets plug directly into any 0.1-inch grid with 0.026- to 0.033-inch diameter round pins or 0.025-inch square pins. These probe tips work with the HP 5090-4356 surface mount grabbers and the HP 5959-0288 through-hole grabbers.

Editing Sequences

Cutting, Copying, Pasting, and Deleting Sequence Lines

NOTE: When you use the *Cut* and *Copy* operations from the menu bar, you are placing the sequence lines in a temporary storage buffer. All subsequent *Paste* operations will insert sequence lines from the storage buffer until new lines are cut or copied. When you use the *Delete* operation from the menu bar, the sequence lines are not placed in the temporary buffer. They are just deleted.

1. Select the sequence lines to cut, copy, or delete by pointing to the first line.
2. Press and hold the left mouse button while dragging the cursor either up or down. To select the entire sequence, select *Edit* from the menu bar, then *Select All Lines*.
3. From the menu bar, select *Edit*, then *Cut Line(s)*, *Copy Line(s)*, or *Delete Lines(s)*.
4. Left-click the sequence line just above where you want to paste the sequence lines. This positions the cursor. When pasting sequence lines, the lines are placed *after* the cursor line.
5. If you are pasting cut or copied lines, select *Edit* from the menu bar, then *Paste Line(s)*.

NOTE: Cutting or deleting all the sequence lines causes the sequence to be reset to the power-up state.

Restrictions on Use

The above operations will not be allowed if the result of the operation places an instruction on one of the following vectors:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.
- The last vector of the MAIN sequence.
- The vector prior to the IF block.
- The first or second vector of the IF block.
- The last vector of the IF block.
- The first or second vector following the IF block.

In addition, the above operations will not be allowed if the result of the operation places a hardware instruction immediately after another instruction. Hardware instructions must follow data vectors.

Deleting Sequence Lines

CAUTION: When you delete sequence lines using the keyboard, they are permanently removed. If you want to place them in a temporary storage buffer, use the Cut (see page 50) operation from the menu bar.

Deleting Single Lines

- To delete single lines, left-click anywhere on the line to position the cursor, then press the *Delete* key on the keyboard.
- To delete multiple lines, select the lines to delete by pointing to the first line, then press and hold the left mouse button while dragging the cursor either up or down. Then press the *Delete* key on the keyboard.

Delete All Lines

To delete all lines in a sequence, go to the menu bar and select *Edit -> Select All Lines*, then *Edit -> Delete Lines(s)*. Deleting all lines causes the sequence to be reset to the power-up state.

Restrictions on Use

You are not allowed to delete the following sequence lines:

- The INIT START line.
- The INIT END line.
- The MAIN START line.
- The MAIN END line.
- The MACRO START line.
- The MACRO END line.

A delete will not be performed if the result of the delete places an instruction on one of the following lines:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.
- The last vector of the MAIN sequence.
- The vector prior to the IF block.
- The first or second vector of the IF block.
- The last vector of the IF block.
- The first or second vector following the IF block.

A delete will not be performed if the result of the delete does either of the following:

- Places a hardware instruction immediately after another instruction.
- Causes the MAIN sequence to contain fewer than two data vectors.

Inserting Blank Sequence Lines

Use the Insert key on the keyboard to insert blank lines (see page 54) below the cursor. This is a very useful operation when starting a new sequence. After you insert a new line, you replace the ditto values with the desired normal data values.

NOTE: The new blank lines are inserted with ditto values.

1. *Point* to the vector line directly above where you want to insert lines and left-click the mouse. This positions the cursor.
2. Press the Insert key on your keyboard one time for each new vector line you want to insert.

NOTE: The "I" key on the keyboard works the same way as the *Insert* key on the keyboard.

In addition, you can insert blank sequence lines by using the mouse as follows:

1. Point to the vector line directly above where you want to insert the new data vector.
2. Press and hold the right mouse button, point to *Insert After*, then select *Vector*.

See Also

"Using Ditto " values" on page 54

Go to a Line Number

1. From within the *Sequence* or *Macro* tab, *point* to any vector line.
2. Press and hold the right mouse button, then select *Goto Line*.
3. In the Goto Line dialog that appears, enter the desired line number.
4. Click *OK*.

See Also

"Positioning the Sequence" on page 54

Positioning the Sequence

Among the keyboard keys that insert and delete data vectors, the pattern generator defines four other keys used to position the sequence when either the *Sequence* or *Macro* tab is active. Specifically, these additional keys are defined:

1. Home - move to the first line of the sequence.
2. End - move to the last line of the sequence.
3. Page Up - scroll upward by one screen's worth of data.
4. Page Down - scroll downward by one screen's worth of data.

To use the positioning keys, perform the following steps:

1. From within Sequence or Macro, *point* to any vector line.
2. Press one of the above keys on the keyboard.

See Also

"Go to a Line Number" on page 53

Using Ditto " values

In any existing data vector, with the exception of the first, you can replace normal data values with ditto values (designated by the double quotation marks).

Ditto values indicate that the previous data value should be repeated for the current data vector. Ditto values save you effort by requiring you to only type in the data values that change.

NOTE: When you insert blank lines, they are inserted as ditto values.

1. From either the *Sequence* or *Macro* tab, left-click the *data field* that you want to edit. This positions the cursor.
2. Type in the ditto values using the double quotation mark key on the keyboard.

NOTE: Ditto characters following an IF block always evaluate to the last vector in the IF block regardless of whether the block is executed or not.

See Also

"Inserting Blank Sequence Lines" on page 53

Working with Instruction Types

Instructions are like programming commands. They are inserted into a sequence for the purpose of executing their designated control over the flow of the sequence. There are two types of instructions: hardware and software. The differences between these types are described below.

Inserting Instructions into Sequences

1. *Point* to the vector line directly above where you want to insert the new instruction.
2. Press and hold the right mouse button while pointing to *Insert After*. Then release the button.
3. *Right-click* the desired instruction to insert.

Hardware Instruction Types

The following hardware instruction types are available. Each of these instructions can affect external hardware or the pattern generator hardware.

- "The Break Instruction" on page 57
- "The Signal IMB Instruction" on page 58
- "The Wait IMB Event Instruction" on page 58
- "The Wait External Event Instruction" on page 59
- "The If External Event Instruction" on page 60
- "The If IMB Event Instruction" on page 61

Software Instruction Types

The following software instruction types are available. Each of these instructions only affects the execution flow of the currently running

sequence. However, a User Macro software instruction can include hardware instructions.

- "The User Macro Instruction" on page 63
- "The Repeat Loop Instruction" on page 64

See Also

"Building a Main Sequence" on page 24

"Building an Initialization Sequence" on page 22

"Building a User Macro" on page 26

The Break Instruction

The Break instruction causes a break at the current vector. In single run mode, this instruction halts the sequence and holds the outputs at the value of the previously outputted data value. In repetitive run mode, this instruction pauses the sequence at the current vector momentarily, then continues. The duration of the pause depends on the activity of other modules in the frame.

Restrictions on Use

The Break instruction must follow data vectors. Also, the Break instruction is not allowed on the following vector lines of a sequence:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.
- The last vector of the MAIN sequence.
- The vector prior to the IF block.
- The first or second vector of the IF block.
- The last vector of the IF block.
- The first or second vector following the IF block.

The Signal IMB Instruction

The Signal IMB instruction creates an arming signal on the *intermodule bus* when the instruction is executed. This *arming* signal allows the pattern generator to *cross trigger* other modules in the frame.

Restrictions on Use

The Signal IMB instruction, as with all hardware instructions, must follow data vectors. Also, the Signal IMB instruction can only be used one time in a sequence, and consequently it is not allowed in a user macro or a repeat loop. The Signal IMB instruction is not allowed on the following vector lines of a sequence:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.
- The last vector of the MAIN sequence.
- The vector prior to the IF block.
- The first or second vector of the IF block.
- The last vector of the IF block.
- The first or second vector following the IF block.

See Also

Using the Intermodule Window (see the *HP 16600A/16700A Logic Analysis System* help volume)

The Wait IMB Event Instruction

The Wait IMB Event instruction halts the execution of the program sequence until an IMB signal is received by the pattern generator.

Restrictions on Use

The Wait IMB Event instruction, as with all hardware instructions, must follow data vectors. Also, the Wait IMB Event instruction can only

be used one time in a sequence, and consequently it is not allowed in a user macro or a repeat loop. The Wait IMB Event instruction is not allowed on the following vector lines of a sequence:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.
- The last vector of the MAIN sequence.
- The vector prior to the IF block.
- The first or second vector of the IF block.
- The last vector of the IF block.
- The first or second vector following the IF block.

The Wait External Event Instruction

The Wait External Event instruction halts the execution of the program sequence until one of four designated events (A-D) is received by the pattern generator.

Because the Wait External Event is an OR'ing function of the toggled "On" wait patterns, as soon as one of the selected wait patterns is satisfied, the sequence continues. The wait patterns are specified on the three input lines of the clock pod: Wait0, Wait1, and Wait2.

To choose and configure a particular event, right-click the Wait External Event instruction, then from the External Wait Pattern dialog, toggle the appropriate fields.

Restrictions on Use

The Wait External Event instruction, as with all hardware instructions, must follow data vectors. Wait External Events are not allowed on the following vector lines of a sequence:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.

- The last vector of the MAIN sequence.
- The vector prior to the IF block.
- The first or second vector of the IF block.
- The last vector of the IF block.
- The first or second vector following the IF block.

The If External Event Instruction

NOTE: The If External Event instruction is only available in Full Channel 100 Mbit/s mode with clock frequencies of 50 MHz and lower (or clock periods of 20 ns or longer).

The If External Event is an OR'ing function of the toggled "On" wait patterns. If one of the selected patterns is satisfied when the If External Event is evaluated, the If block is output. Otherwise, the If block is skipped.

The If External Event instruction uses the same three clock pod input lines (Wait0, Wait1, and Wait2) as the Wait External Event instruction. Right-click the instruction to bring up a dialog used to designate a three-bit data pattern for the pattern generator to compare against. Use the toggle buttons within the External If Pattern dialog to configure the "If" event.

The If External Event instruction inserts the following set of vector lines:

```
        blank data vector
IF EXTERNAL
        blank data vector
        blank data vector
END IF
        blank data vector
        blank data vector
```

NOTE: Ditto characters following an IF block always evaluate to the last vector in the IF block regardless of whether it is executed or not.

The default data vectors inserted with the instruction are restricted.

They can contain vector data, but not instructions. Any new instructions inserted into an If External Event must obey the restrictions listed below. The "If Event" can be removed by deleting either the start or end of the If block.

NOTE: With an If Event instruction (External or IMB) in a sequence, repetitive runs may have latency between the last and the first vectors of the MAIN SEQUENCE. This latency depends on the activity of other measurement modules in the frame, thus varying from run to run.

Restrictions on Use

The If External Event instruction, as with all hardware instructions, must follow data vectors. Also, the If External Event instruction can only be used one time in a sequence, and consequently it is not allowed in a user macro or a repeat loop. The If External Event instruction is not allowed on the following vector lines of a sequence:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.
- The last vector of the MAIN sequence.

The If IMB Event Instruction

NOTE: The If IMB Event instruction is only available in Full Channel 100 Mbit/s mode with clock frequencies of 50 MHz and lower (clock periods of 20 ns or longer).

If an IMB signal is present when the If IMB Event instruction is executed, the data in the If block is output, otherwise it is skipped.

The If IMB Event instruction inserts the following set of vector lines:

```
        blank data vector
IF IMB
        blank data vector
        blank data vector
END IF
        blank data vector
        blank data vector
```

NOTE: Ditto characters following an IF block always evaluate to the last vector in the IF block regardless of whether it is executed or not.

The default data vectors inserted with the instruction are restricted. They can contain vector data, but not instructions. Any new instructions inserted into an If IMB Event must obey the restrictions listed below. Remove the If IMB Event by deleting either the start or end of the If block.

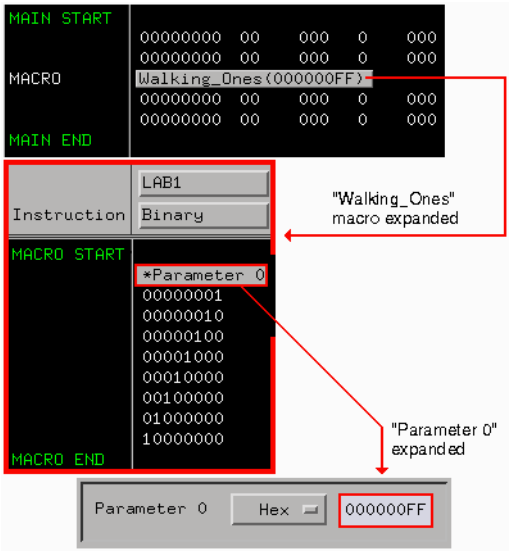
NOTE: With an If Event instruction (External or IMB) in a sequence, repetitive runs may have latency between the last and the first vectors of the MAIN SEQUENCE. This latency is dependent on the activity of other measurement modules in the frame, thus varying from run to run.

Restrictions on Use

The If IMB Event instruction, as with all hardware instructions, must follow data vectors. Also, the If IMB Event instruction can only be used one time in a sequence, and consequently it is not allowed in a user macro or a repeat loop. The If IMB Event instruction is not allowed on the following vector lines of a sequence:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.
- The last vector of the MAIN sequence.

The User Macro Instruction



The User Macro instruction lets you insert a group of vectors that have a specific function. At run time, the user macro is expanded into its corresponding vectors. A typical macro application would be a generic test stimulus that is used by multiple circuits. By containing the device or circuit specific test vectors within a macro, you can simply interchange the macro and reuse the same INIT and MAIN SEQUENCE.

Macros can contain parameters. Using parameters in a macro gives you the same benefit as using macros in a sequence. By passing parameters, you can reuse the same macro for other purposes.

Restrictions on Use

The User Macro instruction is not allowed on the following vector lines of a sequence:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.

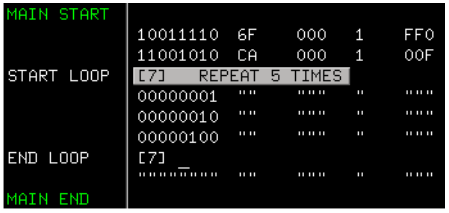
- The last vector of the MAIN sequence.
- The vector prior to the IF block.
- The first or second vector of the IF block.
- The last vector of the IF block.
- The first or second vector following the IF block.

See Also

"Building a User Macro" on page 26

"Working with Macro Parameters" on page 79

The Repeat Loop Instruction

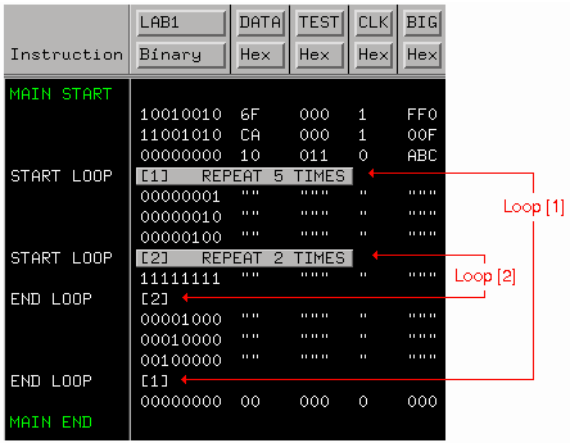


The Repeat Loop instruction inserts the start and end vectors of a repeat loop, along with one blank data vector row, below the selected vector row. Once the loop has been created, you can insert or copy vectors and instructions into the loop.

Set the number of loop repetitions (maximum 20,000) by right-clicking the "Start Loop" vector line and editing the *Loop Count* dialog. At run time, the loop count determines the number of times to expand the loop's body into individual vectors. Both the start and end vectors of a repeat loop are removed from the sequence if either one is included in a delete operation.

Nested Repeat Loop Instructions

This example shows a nested loop. Loop[1] is a walking ones pattern that is repeated five times. Loop[2] is a bit pattern of all ones that is repeated twice directly in the middle of the walking ones pattern of Loop[1].



Restrictions on Use

The following instructions can NOT be used in a repeat loop:

- The Signal IMB instruction.
- The If External Event instruction.
- The If IMB Event instruction.
- The Wait IMB Event instruction.

The Repeat Loop instruction is not allowed on the following vector lines of a sequence:

- The first or second vector of the INIT sequence.
- The first or second vector of the MAIN sequence.
- The last vector of the MAIN sequence.
- The vector prior to the IF block.
- The first or second vector of the IF block.
- The last vector of the IF block.
- The first or second vector following the IF block.

Working with Labels and Pods

Access label and pod operations from either the *Edit* pick in the menu bar, or under the label name field itself. When you select operations from the menu bar, they are global to all labels or pods in the window. When you select operations from under the label name, they are directed at the individual label.

In addition, label and pod operations in the *Sequence* and *Macro* displays are centered around adding or deleting. Operations in the *Format* display are centered around initial creation and configuration.

NOTE: Label operations are performed on labels assigned in Format. If labels are not created, assigned bits, and turned on in Format, they do not appear in Sequence or Macro. Also, depending on the Vector Output Mode (see page 14), some pods may not be available.

Operations in Format

- "Creating and Inserting New Labels" on page 67
- "Deleting Labels" on page 68
- "Renaming Existing Labels" on page 69
- "Reordering a Label's Pod Bits" on page 70
- "Turning Labels On/Off" on page 70
- "Clearing Format Labels" on page 71
- "Finding a label" on page 73
- "Swap Pods" on page 71
- "Clear Pods" on page 72
- "Assigning Bits to a Label" on page 72
- "Label Polarity" on page 73

Operations in Sequence and Macro

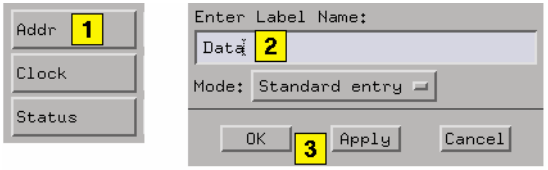
- "Inserting Pre-assigned Labels" on page 69

Creating and Inserting New Labels

Creating New Labels in Format

You create labels in Format to uniquely identify groups of assigned pod bits (see page 72) that have a common purpose or identity. Inserting and assigning labels are part of the mapping process (see page 13) for your application.

1. Right-click the label where you want to insert the new label, then select *Insert before* or *Insert after*.
2. In the Enter Label Name dialog that appears, type in the new label name. If you do not provide a label name, a default name is assigned.
3. Click *OK* if you are done, or *Apply* if you have more labels to insert.

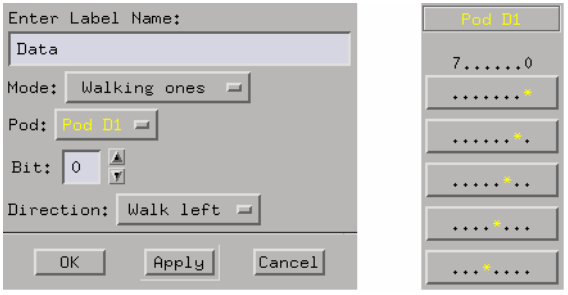


NOTE: Duplicate label names are not allowed.

Mode

If you are creating a series of new labels, you have the option to automatically assign bits in a "walking ones" pattern across the label series. You do this by toggling the *Mode* field to *Walking ones*.

Select the Pod, the starting Bit, and the walk direction. As you enter new label names and click *Apply*, a single bit is automatically assigned under each new label in a "walking ones" pattern.



Deleting Labels

1. Point to the label's name, then press and hold the right mouse button.
2. Select *Delete*, then release the mouse button.

See Also

"Clearing Format Labels" on page 71

"Delete All Labels" on page 76

Inserting Pre-assigned Labels

NOTE: If labels are not created, assigned bits, and turned on in *Format*, they do not appear in subsequent label operation dialogs in *Sequence* and *Macro*.

1. From the *Sequence* or *Macro* tab, right-click the label where you want to insert the new label, then select *Insert Before* or *Insert After*.
2. In the Select Label dialog that appears, select the pre-assigned label name to insert.
3. Click *OK* if you are done, or *Apply* if you have more labels to insert.

By holding down the *Control key* while using the mouse, you can select multiple label names in a discontinuous manner. By holding down the *Shift key* while using the mouse, you can select multiple label names in a continuous manner.

See Also "Searching for Labels" on page 71

Renaming Existing Labels

1. Under *Format*, right-click the label you want to rename, then select *Rename*.
2. In the Rename dialog, type in the new label name.

3. Click *OK* if you are done, or *Apply* if you have more labels to rename.

NOTE: When you click *Apply*, the next label is automatically highlighted. This lets you rename multiple labels very quickly.

Reordering a Label's Pod Bits

Use the *Reorder bits* feature to remap the physical probe connections to the interface without changing the actual probe connections. This feature allows the probe tips for each channel to be physically connected where convenient.

1. Under the *Format* tab, right-click the label you want to reorder bits on, then select *Reorder bits*.
2. Set the bit order by one of the following options:
   - To reorder bits individually, for each channel, type the number of the bit you want to map the channel to.
   - To arrange the bits sequentially, click the field at the top of the dialog, then select *Default Order*.
   - To reverse the order of the bytes, click the field at the top of the dialog, then select *Big Endian - Little Endian* mapping. This option is only available for labels that have either 16 or 32 channels.
3. Click *OK*.

NOTE: Reordering the bits of a label does not reorder the physical output signals of the channels assigned to that label. The bit order of a label is a display-orientation concept, useful only when entering data values.

Turning Labels On/Off

Turning labels off prevents output signals from appearing on the associated probe channels. When a label is turned off, it is removed from the *Sequence* and *Macro* areas. However, the label's name and bit

assignments are preserved.

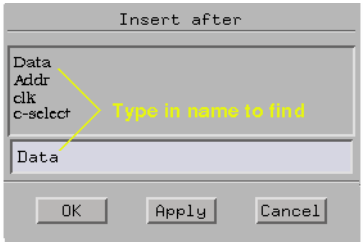
To toggle labels On/Off in Format, right-click the label that you want to modify and toggle the Label On/Off selection.

Clearing Format Labels

The *Clear Format Labels* command is located under *Edit* in the Format menu bar. When you select *Clear Format Labels*, all user-assigned labels are permanently removed, and the default label *Lab1* is reset.

Searching for Labels

In all label selection dialogs, where a list of label names appears, a text entry field is available for a quick search of label names. Simply type in a search string, and all labels that begin with that string appear in the list.



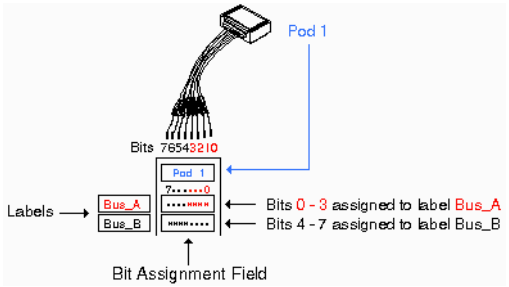
Swap Pods

The *Swap Pods* command is located under *Edit* in the Format menu bar. When you select *Swap Pods*, the bit assignments for the two designated pods are swapped across all labels.

Clear Pods

The *Clear Pods* command is located under *Edit* in the Format menu bar. When you select *Clear Pods*, all bit assignments for all labels under the designated pod are cleared.

Assigning Bits to a Label



The bits in a label correspond to the physical pattern generator probe channels. When you run the pattern generator, data is output on all bits (channels) that are assigned to labels. Unassigned bits are inactive.

- An asterisk "*" indicates an assigned bit.
- A period "." indicates an unassigned bit.

To Assign Bits

1. Select the bit assignment field to the right of the label name you want to define. Each bit assignment field corresponds to the data pod listed above it.
2. Click the bits you want to change, toggling them between an asterisk and a period. You can also hold the left mouse button and drag the mouse to

assign several bits at once.

3. When the bits are assigned as desired, close the dialog box.

Right-click in the bit assignment field or dialog to see a shortcut menu for assigning groups of bits.

NOTE: Labels can have a maximum of 32 channels assigned to them, however, you cannot assign any single output channel to more than one label.

Bits assigned to a label are numbered from right to left. The least significant assigned bit on the far right is numbered 0. The next assigned bit to the left is numbered 1, and so on. Labels can contain bits that are not consecutive; however, bits are always numbered consecutively within a label.

Label Polarity

The pattern generator can display each label's data in both positive and negative logic. The default polarity for all labels is positive. To toggle the polarity, left-click the label's polarity field under the *Format* tab.

NOTE: Toggling the polarity of a label does not toggle the physical output signal. The polarity of a label is a display orientation concept, useful only when entering data values.

Finding a label

Access the *Find Label* feature under *Edit* in the menu bar of all tabs. The *Find Label* feature locates specified labels, then displays them.

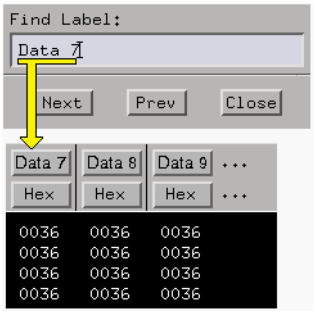
The search functionality is similar to a text based keyword search. After you type the label name and select Next or Prev, the list of labels is rolled, exposing the label whose text characters, left to right, match

the label name entered.

You can search for a complete label name as shown below in example 1, or type just the base name and use the *Next* or *Prev* fields to roll through the list as in example 2.

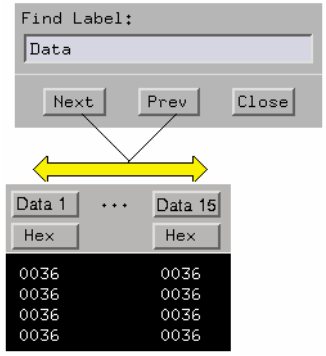
Example 1

You have a list of labels named *Data1* through *Data15*. Type the complete name *Data7* in the text entry field, then select *Next*. The label named *Data7* appears on the left-most side of the displayed labels.



Example 2

You have a list of labels named *Data1* through *Data15*. Type only the base name *Data* in the text entry field, then use the *Next* or *Prev* fields to roll the list of labels.

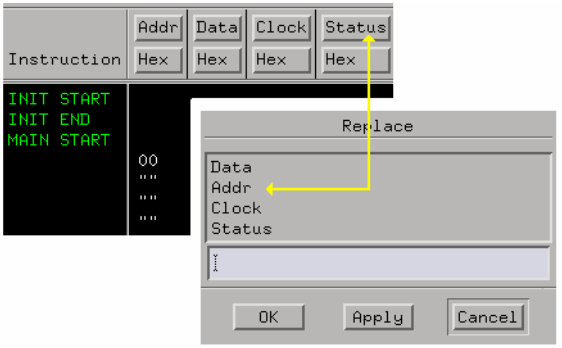


Replace Labels

The *Replace Label* command is available in the *Sequence* and *Macro* tabs.

1. Right-click the label to replace.
2. Select *Replace*.
3. From the Select Label dialog that appears, select one or more new labels to replace the existing label. See the mouse tip below.
4. Click *OK* if you are done, or *Apply* if you have more labels to replace.

By holding down the *Control key* while using the mouse, you can select multiple label names in a discontinuous manner. By holding down the *Shift key* while using the mouse, you can select multiple label names in a continuous manner.



See Also "Searching for Labels" on page 71

Appending Labels

The *Append Label* command is located under *Edit* in the Sequence

and Macro menu bar. The *Append Label* command adds labels to the right of the list of labels in Sequence and Macro.

1. In Sequence or Macro, left-click *Edit*, then select *Append Label*.
2. From the Select Label dialog that appears, select the new labels to append.
3. Click *OK* if you are done, or *Apply* if you have more labels to append.

See Also

"Searching for Labels" on page 71

Insert All Labels

The *Insert All Labels* command is located under *Edit* in the Sequence and Macro menu bar. Select this command to insert all valid labels under the Format tab into the sequence.

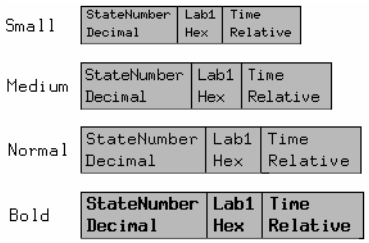
Delete All Labels

The *Delete All Labels* command is located under *Edit* in the Sequence and Macro menu bar. Select this command to delete all assigned labels in the sequence.

Setting the Label Font Size

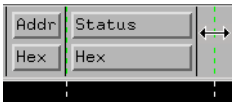
Font settings are global. The font size you select is applied to all text in the display area.

1. From the menu bar select *Options*, then select *Font*.
2. Select a font size from the list.



Example of Font Sizes

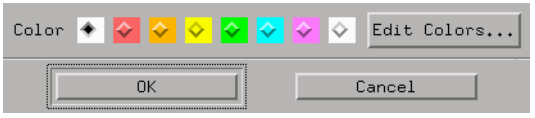
Adjusting Column Width



1. *Point* to the right edge of the label box.
2. Press and hold the *left mouse button*.
3. Drag the box edge to the desired width, then release the mouse button.

Setting Column Color

1. Click the *Sequence* or *Macro* tab.
2. Right-click a label, then select *Change Color*.
3. Click the desired color.
4. Click *OK*.

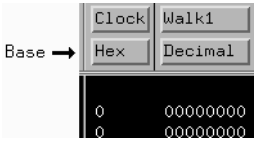


Setting Default Label Color

The *Default Label Color* command lets you set the label color for all new labels inserted into either Sequence or Macro.

1. Click the *Sequence* or *Macro* tab.
2. From the menu bar, select *Options*, then select *Default Label Color*.
3. From the Default Color dialog that appears, select the desired color.
4. Click *OK*.

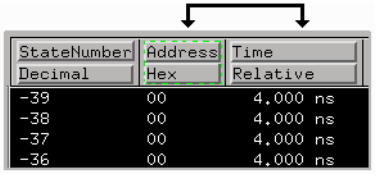
Setting the Numeric Base



1. *Point* to the label's base field, then press and hold the right mouse button.
2. Select the desired base type, then release the mouse button.

Rearranging the Label Order

1. *Point* to a label, then press and hold the left mouse button.
2. Drag the highlighted label to its new location, then release the mouse button.



Working with Macro Parameters

Parameters are used to pass values into macros. A major benefit in passing parameters is that you keep a macro's functionality generic and still direct specific action identified by parameters. Think of a parameter as the only part of a macro that changes as the macro is reused.

Use this procedure after creating your macro or recalling it in the *Macro* display:

1. From *Macro*, turn the desired parameters on. (see page 79)
2. Insert the parameters. (see page 80)
3. Assign values to the parameters. (see page 80)

NOTE: Macro parameters are passed as 32-bit values. If fewer than 32 bits are assigned to a label in *Format*, the most significant bits of the parameter value are truncated when the parameter is used as data for the given label.

See Also

"Building a User Macro" on page 26

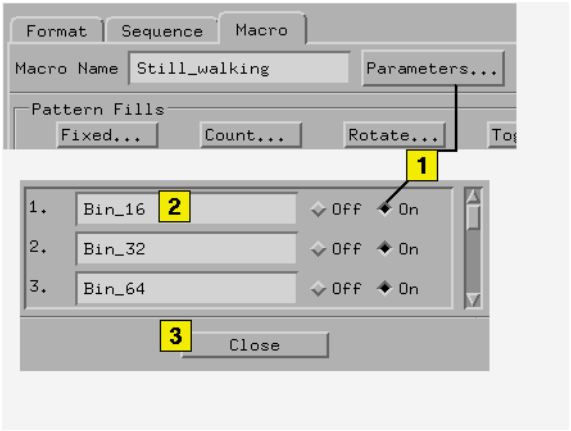
"Recalling Macros" on page 98

"Removing Parameters from a Macro" on page 81

Turning Parameters On

You have 10 available parameters per macro. You cannot use parameters until you turn them on.

1. From *Macro*, click *Parameters...*, then click *On*.
2. Optional - If you want a custom parameter name, left-click in the name field and type a name.
3. Click *Close*.



Inserting Parameters into a Macro

NOTE: Because parameters replace a *data field* within data vectors, you can only set them on an existing data vector. They cannot be set on instruction vectors.

The following procedure is performed after your macro is created, recalled into the Macro display, and the parameter is turned on. (see page 79)

1. From *Macro*, left-click the data field you want to replace. This positions the cursor.
2. Right-click over any vector line and select *Set Parameter*.
3. From the Set Parameter dialog, click the parameter you want to set.
4. Click *OK*.

See Also "Working with Macro Parameters" on page 79

Assigning Parameter Values

Parameter values are easily set or changed from the tab in which the

macro instruction was inserted. Assigned parameter values are displayed in parentheses next to their corresponding macro instruction. Parameter values are always displayed as hexadecimal values.

1. From the *Sequence* tab (or the *Macro* tab, if the macro is nested), right-click the macro instruction.
2. From the Set Parameters dialog that appears, type in the new parameter values.
3. Click *OK*.

NOTE: Macro parameters are passed as 32-bit values. If fewer than 32 bits are assigned to a label in Format, the most significant bits of the parameter value are truncated when the parameter is used as data for the given label.

Removing Parameters from a Macro

1. From *Macro*, left-click the *data field* that contains the parameter to be removed.
2. Right-click over any vector line and select *Clear Parameter*.

Working with Automatic Pattern Fills

There are five automatic pattern fill utilities available to quickly generate and insert signal patterns into your sequences and user macros. By using pattern fills, you not only save time generating generic patterns, but you also guarantee the accuracy of the sequence.

Pattern fills can either be inserted below the selected sequence line (flashing cursor), or configured to overwrite existing vectors starting at the selected sequence line.

- "Generating a Fixed Pattern Fill" on page 82
- "Generating a Count Pattern Fill" on page 83
- "Generating a Rotate Pattern Fill" on page 85
- "Generating a Toggle Pattern Fill" on page 86
- "Generating a Random Pattern Fill" on page 87

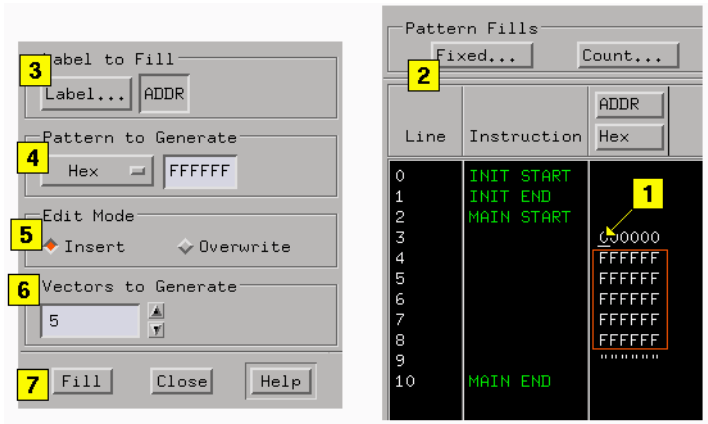
Generating a Fixed Pattern Fill

1. Position the cursor for one of the following operations.
   - To *insert* lines, *point* the cursor to the vector line directly above where you want to insert the new lines, then left-click the mouse.
   - To *overwrite* lines, point to the vector line where you want the overwrite to begin, then left-click the mouse.
2. From either *Sequence* or *Macro*, select *Fixed...*.
3. From the Fixed Pattern Fill dialog, click *Label...*, then select the desired label to fill.
4. Assign a pattern by filling in the pattern entry field and selecting a numeric base. For a shortcut, see the mouse tip below.
5. Select the Edit Mode.
   - Insert - new vector lines are inserted after the flashing cursor.

- Overwrite - existing vector lines are overwritten, beginning with the line containing the flashing cursor.

6. Select the number of pattern fill vectors to generate.
7. Click *Fill*.

Right-click in the pattern entry field to see a shortcut menu for assigning common patterns.

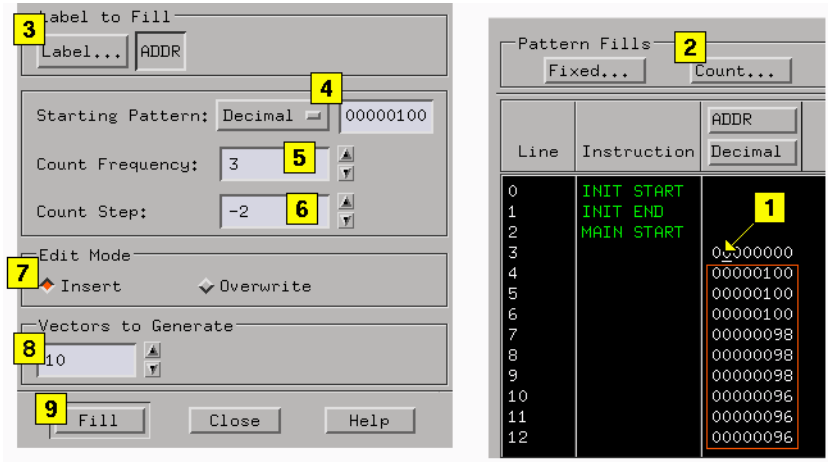


Generating a Count Pattern Fill

1. Position the cursor for one of the following operations.
   - To *insert* lines, *point* the cursor to the vector line directly above where you want to insert the new lines, then left-click the mouse.
   - To *overwrite* lines, point to the vector line where you want the overwrite to begin, then left-click the mouse.
2. From either *Sequence* or *Macro*, select *Count...*.
3. From the Count Pattern Fill dialog, click *Label...*, then select the desired label to fill.

4. Assign a starting pattern by filling in the pattern entry field and selecting a numeric base. For a shortcut, see the mouse tip below.
5. Select the Count Frequency. The count frequency specifies how often the counting pattern changes. For example, if count frequency is set to 2, the counting pattern increments after every 2 vectors (that is, 1,1,2,2,3,3,...).
6. Select the Count Step. The count step specifies whether the pattern counts up or down (by using a positive or negative value) and the unit of step. For example, with count step set to -2, the pattern counts downward, skipping every other value (that is, 100, 98,96,...).
7. Select the Edit Mode.
   - Insert - new vector lines are inserted after the flashing cursor.
   - Overwrite - existing vector lines are overwritten, beginning with the line containing the flashing cursor.
8. Select the number of pattern fill vectors to generate.
9. Click *Fill*.

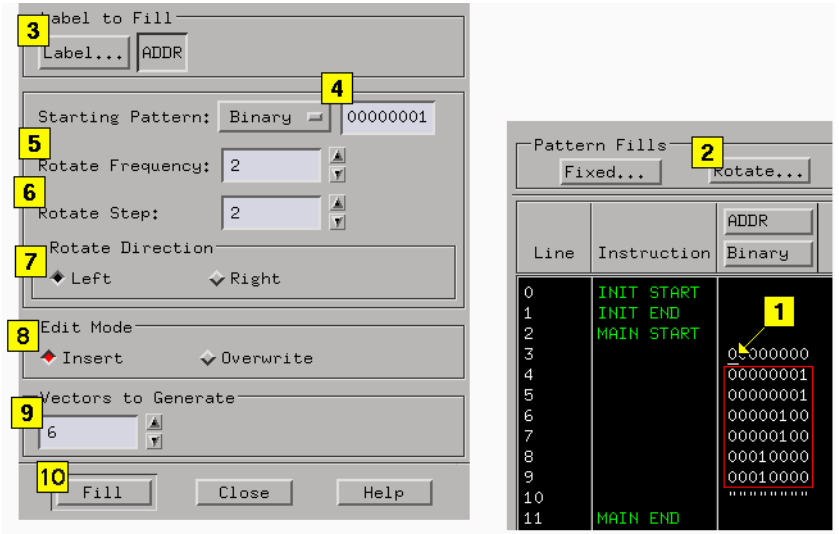
Right-click in the pattern entry field to see a shortcut menu for assigning common patterns.



Generating a Rotate Pattern Fill

1. Position the cursor for one of the following operations.
   - To *insert* lines, *point* the cursor to the vector line directly above where you want to insert the new lines, then left-click the mouse.
   - To *overwrite* lines, point to the vector line where you want the overwrite to begin, then left-click the mouse.
2. From either *Sequence* or *Macro*, select *Rotate...*.
3. From the Rotate Pattern Fill dialog, click *Label...*, then select the desired label to fill.
4. Assign a starting pattern by filling in the pattern entry field and selecting a numeric base. For a shortcut, see the mouse tip below.
5. Select the Rotate Frequency. The rotate frequency specifies how often the rotating pattern changes. For example, if rotate frequency is set to 2, the pattern rotates after every 2 vectors (that is, 0001, 0001, 0010, 0010, 0100, 0100,...).
6. Select the Rotate Step. The rotate step specifies how many units the pattern rotates. For example, with rotate step set to 2, the pattern rotates 2 units (that is, 00001, 00100, 10000,...).
7. Select the Rotate Direction.
   - Left - rotates pattern to the left.
   - Right - rotates pattern to the right.
8. Select the Edit Mode.
   - Insert - new vector lines are inserted after the flashing cursor.
   - Overwrite - existing vector lines are overwritten, beginning with the line containing the flashing cursor.
9. Select the number of pattern fill vectors to generate.
10. Click *Fill*.

Right-click in the pattern entry field to see a shortcut menu for assigning common patterns.



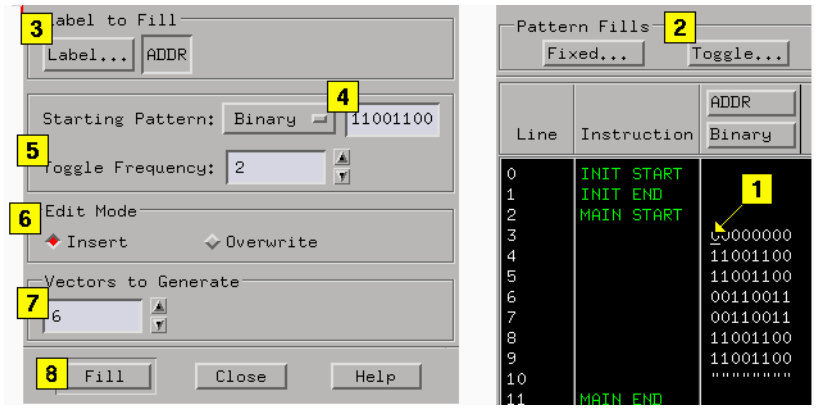
Generating a Toggle Pattern Fill

1. Position the cursor for one of the following operations.
   - To *insert* lines, *point* the cursor to the vector line directly above where you want to insert the new lines, then left-click the mouse.
   - To *overwrite* lines, point to the vector line where you want the overwrite to begin, then left-click the mouse.
2. From either *Sequence* or *Macro*, select *Toggle...*.
3. From the Toggle Pattern Fill dialog, click *Label...*, then select the desired label to fill.
4. Assign a Starting Pattern by filling in the pattern entry field and selecting a numeric base. For a shortcut, see the mouse tip below.
5. Select the Toggle Frequency. The toggle frequency specifies how often the pattern changes. For example, if toggle frequency is set to 2, the pattern toggles after every 2 vectors (that is, 0011, 0011, 1100, 1100, 0011,

0011,...).

6. Select the Edit Mode.
   - Insert - new vector lines are inserted after the flashing cursor.
   - Overwrite - existing vector lines are overwritten, beginning with the line containing the flashing cursor.
7. Select the number of pattern fill vectors to generate.
8. Click *Fill*.

Right-click in the pattern entry field to see a shortcut menu for assigning common patterns.

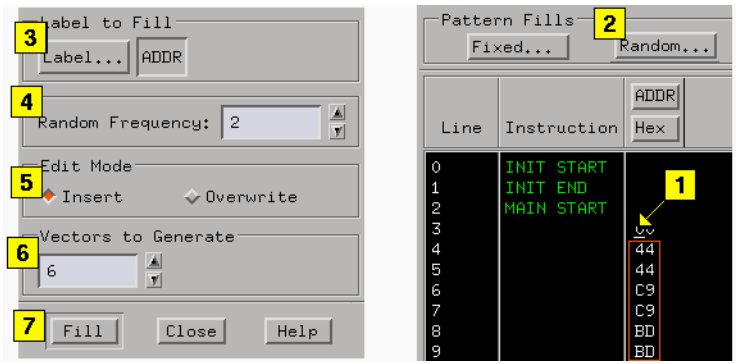


Generating a Random Pattern Fill

1. Position the cursor for one of the following operations.
   - To *insert* lines, *point* the cursor to the vector line directly above where you want to insert the new lines, then left-click the mouse.
   - To *overwrite* lines, point to the vector line where you want the overwrite to begin, then left-click the mouse.

2. From either *Sequence* or *Macro*, select *Random...*.
3. From the Random Pattern Fill dialog, click *Label...*, then select the desired label to fill.
4. Select the Random Frequency. The Random frequency specifies how often the random pattern changes. For example, if random frequency is set to 2, the pattern changes after every 2 vectors (that is, 86, 86, 74, 74, F3, F3,...).
5. Select the Edit Mode.
   - Insert - new vector lines are inserted after the flashing cursor.
   - Overwrite - existing vector lines are overwritten, beginning with the line containing the flashing cursor.
6. Select the number of pattern fill vectors to generate.
7. Click *Fill*.

Right-click in the pattern entry field to see a shortcut menu for assigning common patterns.



Printing the Pattern Generator Window

The *Print This Window* operation lets you print just the pattern generator window. Use this operation if you want a hardcopy or electronic record of configurations and data currently displayed in the viewing area of the pattern generator window.

NOTE: Only the currently displayed viewing area of the pattern generator window is printed. If any data or configuration fields appear offscreen, scroll the desired data or configuration fields into the viewing area before printing.

1. Optional - configure the Print Options (see the *HP 16600A/16700A Logic Analysis System* help volume) if desired.
   Print Options include print destination, file format type, filename autoincrement, and color/b&w; pixel mapping.
2. In the Pattern Generator tool menu bar, click *File*, then select *Print This Window*. The print output will be as configured in the Print Options in step 1.

See Also

Setting Print Options (see the *HP 16600A/16700A Logic Analysis System* help volume)

Set Up the Printer (see the *HP 16600A/16700A Logic Analysis System* help volume)

Printing Vector Sequences to a File

Printing sequences to a file is useful when you want a filecopy or hardcopy of your programs for documentation purposes. The procedure is the same for sequences and macros.

1. From the menu bar, left-click *File*.
2. Left-click *Print Sequence To File* or *Print Macro To File*. The Print to File dialog appears.
3. Select which label to print. If you want to print all of the labels, select *All*. If you want to print selected labels, click *Selected*, then select the desired labels. See the mouse tip below.
4. Select either *All* or *Partial* as the range of sequence lines to print. If you select Partial, specify the range of line numbers.
5. Select the path and filename to print the sequence to. Use either the *File...* dialog to build a path and filename, or type the path and filename into the text entry field.
6. Optional - type a file description in the text entry area. File descriptions are printed at the top of the output file.
7. Click *Save*.

By holding down the *Control key* while using the mouse, you can select multiple label names in a discontinuous manner. By holding down the *Shift key* while using the mouse, you can select multiple label names in a continuous manner

Viewing Saved Vector Sequence Files

The following procedure shows you how to connect to the logic analysis system through *File Transfer Protocol* (FTP), then execute the FTP "Get" command to place files into your home directory for viewing.

NOTE: The logic analysis system stores the vector sequence files in binary format. Use the FTP binary mode to preserve the data stored in these files.

1. From the command line of the remote machine, type *ftp "machine_name"*, where *"machine_name"* can be either the IP address, or the hostname with an assigned alias for the logic analysis system.
2. For the User Name: type *"anonymous"*.
3. For the Password: type *"anonymous"*.
4. From the FTP prompt, type *"get filename"*.

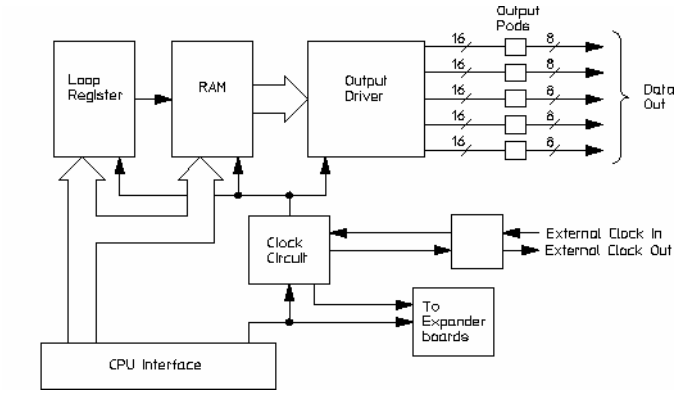
At this point, the sequence file should be saved in your home directory, and be ready for you to open and view it.

Example of Saved Sequence

```
                          DATA        ADDR
Line      Instruction     Binary      Hex
_____     ___________     ________    ____

0         INIT START
1         INIT END
2         MAIN START
3                         00000000    00
4                         00000001    01
5                         00000010    02
6                         00000100    03
7                         00001000    04
8                         00010000    05
9                         00100000    06
10                        01000000    07
11                        10000000    08
12                        00000000    00
13        MAIN END
```

Theory of Operation



Pattern Generator Hardware Block Diagram

Loop Register PGTheory (see page 92)

RAM PGTheory (see page 93)

Output Driver PGTheory (see page 93)

Clock Circuit PGTheory (see page 93)

CPU Interface PGTheory (see page 94)

Pod PGTheory (see page 94)

Loop Register PGTheory

The loop register holds the programmable vector flow information. When the module reaches the end of the vector listing, the loop register is queried for the RAM address location of the next user-programmed vector. In many cases, the next vector address location would be the start of the vector listing. Consequently, the vectors would continue to loop from the end of the listing back to the beginning until you instruct the pattern generator to stop.

RAM PGTheory

Consisting of six 256Kx16 VRAM ICs and RAM addressing circuitry, the RAM stores the desired patterns that appear at the module output. The RAM addressing circuitry is merely a counter which addresses the pattern locations in RAM. When the end of the vector listing is reached, the addressing circuitry is loaded from the loop register with the address of the first vector of the listing to provide an uninterrupted vector loop. The RAM output is sent to the Output Driver circuit where the patterns are presented in a logic configuration that is usable by the output pods.

Output Driver PGTheory

The output driver circuit is made up of a series of latch/logic translators and multiplexers. The latch/translators convert the working-level TTL signals to output-level ECL signals for each channel. The ECL-level signals are then directed to the multiplexers.

The multiplexers, one per channel, direct the programmed data patterns to the output channels. The single-ended ECL-level signals are converted to differential signals which are routed to the output cables and to the pods.

NOTE: The differential ECL output signal of the pattern generator module is not suitable to directly drive ECL circuitry.

Clock Circuit PGTheory

The clock circuit paces the loop register, the RAM address circuitry, and the multiplexers in the output driver according to the desired data rate. A 200-MHz clock source is directed through a divider circuit which provides a 100-MHz and 50-MHz clock in addition to a 200-MHz clock. The 200-MHz, 100-MHz, 50-MHz, and external clock signals are routed to a clock select multiplexer. The output of the multiplexer,

which represents the user-selected clocking rate, is distributed to the above listed subcircuits on both the *master card* and all expander cards that are configured with the master card.

The output of the clock select multiplexer is also distributed to an external clock-out circuit. The clock signal is routed to a bank of external clock delays, then to an external clock delay select multiplexer. The output of this multiplexer, which represents the desired clock delay, is directed to the external clock out pin on the clock pod. Consequently, either the internal clock or external clock is redirected to the clock out pin on the clock pod with a user-selected clock delay.

CPU Interface PGTheory

The CPU interface is a single programmable-logic device which interprets the logic analysis system backplane logic and translates the logic into signals to drive and program the pattern generator module.

Clock and Data Pod PGTheory

Both the clock and data pods convert the differential output ECL signal to the logic levels of interest. Because the output of the pattern generator module cannot directly drive ECL circuitry, the Clock and Data Pod are required to interface the pattern generator with the target system.

Key Characteristics

Output Channnels:

20 channels at 200 MHz clock; 40 channels at 100 MHz clock.

Memory Depth:

258,048 vectors.

Logic Levels (data pods):

TTL, 3-state TTL/3.3v, 3-state TTL/CMOS, ECL terminated, ECL unterminated, and differential ECL (without pod).

Data Inputs:

3-bit pattern level sensing (clock pod).

Clock Output:

Synchronized to output data, delay of 11 ns in 9 steps (clock pod).

Clock Input:

DC to 200 MHz (clock pod).

Internal Clock Period:

Programmable from 5 ns to 250 us in a 1, 2, 2.5, 4, 5, 8 sequence.

External Clock Period:

DC to 200 MHz.

External Clock Duty Cycle:

2 ns minimum high time.

Maximum Number of "IF Condition" Blocks at 50 MHz:

1

Maximum Number of Different Macros:

100

Maximum Number of Lines in a Macro:

4096

Maximum Number of "Wait" Event Patterns:

4

Automatic Cursor Wrap

The Cursor Wrap mode specifies how the cursor should line wrap when the cursor comes to the end of a *data field* during data entry from the keyboard.

The automatic cursor wrap has two modes:

- Line (default) - enters all values for all labels within a single line before proceeding to the next line below.
- Column - enters all values for a single label only.

Setting the Cursor Wrap

1. In *Sequence* or *Macro*, left-click *Options*, then select Cursor Wrap.
2. Select either Line or Column.

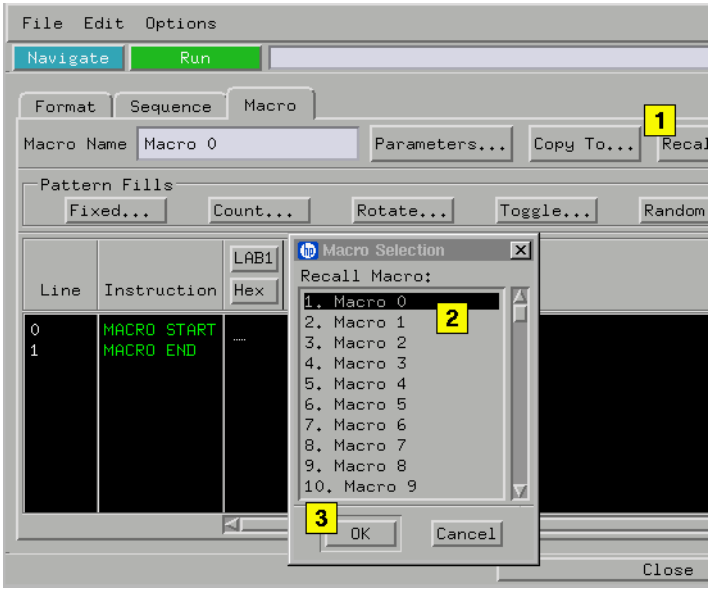
NOTE: If you are entering many data vectors, it is quicker to insert all blank data vectors first, and then enter the data while letting the cursor wrap feature guide you.

Recalling Macros

The pattern generator contains 100 macros. Each macro is initially empty at power-up. To create a new macro or modify a particular macro, you must first recall that macro in the *Macro* display for editing.

To Recall a User Macro

1. Click the *Macro* tab.
2. Left-click *Recall*.
3. From the Macro Selection dialog that appears, left-click the desired macro name.
4. Click *OK*.



See Also

"Working with Macro Parameters" on page 79

"Working with Instruction Types" on page 56

"Editing Sequences" on page 50

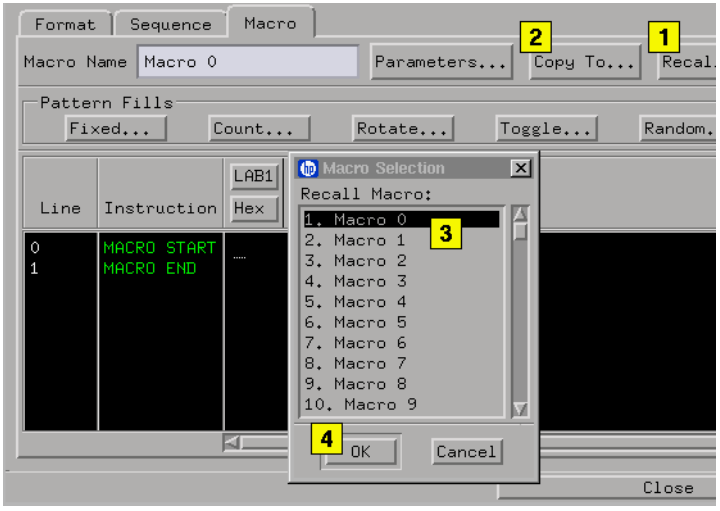
"Working with Labels and Pods" on page 66

Copying Macros

You can copy the currently displayed macro in the Macro display to any of the other available macros. Use this feature to create a new, but slightly different macro from the original macro.

To Copy a User Macro

1. From Macro, recall (see page 98) the macro you want to copy.
2. Left-click *Copy To*.
3. From the Macro Selection dialog that appears, left-click the desired macro name you want to copy to.
4. Click *OK*.



See Also

"Working with Macro Parameters" on page 79

"Working with Instruction Types" on page 56

"Editing Sequences" on page 50

"Working with Labels and Pods" on page 66

Viewing a Compiled Sequence

When a display tool is connected to the Pattern Generator icon, it displays the *compiled* sequence, rather than the actual sequence output at run time.

The compiled view of the sequence can be helpful when trying to understand the behavior of software instructions. The compiled view is not helpful when trying to understand the behavior of hardware instructions, since their behavior changes from run to run.

For more information on hardware and software instruction types, see Working with Instruction Types. (see page 56)

Testing the Pattern Generator Hardware

In order to verify that the pattern generator hardware is operational, run the Self Test utility. The Self Test function of the HP 16600A/16700A logic analysis system performs functional tests on both the system and any installed modules.

1. Disconnect any inputs from the pattern generator.
2. From the system window, click the *System Admin* icon, then select *Self Test*.
3. Click *Master Frame*.
   If the module is in an expansion frame, click *Expansion Frame*.
4. Click *16522A 200Mvector/s Pattern Generator*.
5. In the Self Test dialog box, click *Test All*.
   You can also run individual tests by clicking on them. The *Output Stimulus Vectors* test must be run this way.
6. To verify the output, run the *Output Stimulus Vectors* test.
   For this test, you need an oscilloscope with a bandwidth of at least 500 MHz, an oscilloscope probe with a bandwidth of at least 500 MHz, and the HP 10460A-series output data pod.
   a. Connect the output data pod to the end of the pod 1 cable.
   b. Click *Output Stimulus Vectors*.
   c. Click *Checkerboard Pattern*.
   d. Use the oscilloscope to test each channel. The logic levels should correspond to the logic levels of the output data pod.
   e. Repeat step c for each of the pattern generator data pods.
   f. Connect an HP 10460-series clock pod to the end of the pattern generator clock cable.
   g. Repeat step c for the clock pod.
   h. Click *Stop*.
7. Click *Close*.

If any test fails, contact your local Hewlett-Packard Sales Office or Service Center for assistance.

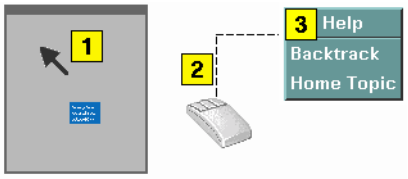
See Also

Self Test (see the *HP 16600A/16700A Logic Analysis System* help volume)

HP 16522A 200Mvector/S Pattern Generator Service Guide

Help - How to Navigate Quickly

1. Place mouse cursor anywhere in a help window.
2. Press the right mouse button.
3. Select desired destination.



You can also access all navigation and search commands from the help window menu bar.

Run/Group Run Function

- Setting a tool for independent or Group Run (see page 106)
- Setting Single or Repetitive Run (see page 107)
- “Checking Run Status” on page 107

Understanding Run/Run All/Group Run

The Run/Run All/Group Run buttons initiate data capture in the instrument tools you have configured. When an instrument tool is connected to analysis or display tools, any of the tools can initiate a run. When two or more instrument tools are configured, you can run them independently or as a group. Two or more instruments running as a group is called an Intermodule measurement.

Use the Intermodule Window (see the *HP 16600A/16700A Logic Analysis System* help volume) to coordinate the run function of multiple instruments as a "Group Run". A common "Group Run" configuration is to run the instrument tools at the same time. A more advanced measurement is to configure one instrument to arm another instrument, each with their own trigger conditions.

- Run appears in the setup dialog and icon menu of an instrument if it is not part of an Intermodule measurement.
- Group Run appears in the setup dialog and icon menu of each tool if two or more instruments are configured for an Intermodule measurement.
- Run All always appears in the System, Workspace and Run Status windows, and initiates a run in all configured instruments, whether they are run independently or are part of a Group Run.

Intermodule measurements are configured between individual instruments. Arming between two machines that belong to one analyzer is configured in the *Arming Info...* dialog found in the *Trigger* window of the analyzer.

Understanding Stop/Stop_All/Cancel

- Stop will terminate an individual instrument measurement that is running.

(perhaps waiting for a trigger condition)

- Stop All, when selected from the Workspace, will terminate running measurements from all instruments currently on the Workspace.
- Cancel will terminate the processing of trace data from an instrument to an analysis or display tool connected to its output.

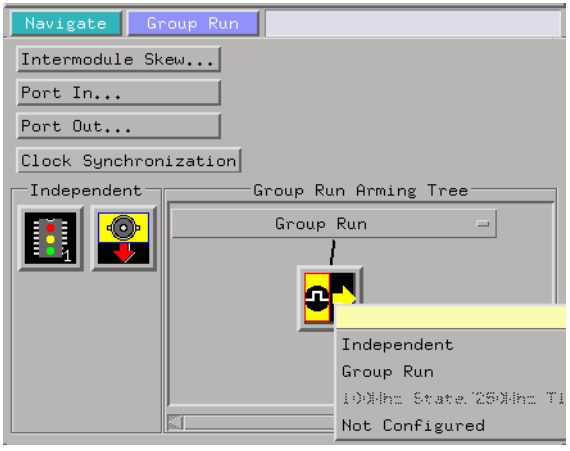
See Also “Demand Driven Data” on page 108

Setting a tool for independent or Group Run

Use the Intermodule Window to change between Group Run and independent Run.

- Click the Intermodule icon in the System Window, OR
- Use Navigate->System->Intermodule

In the Intermodule window, move instruments between independent Run and Group Run by clicking the icon and selecting the desired arming source. All instruments in "Group Run" will run simultaneously.

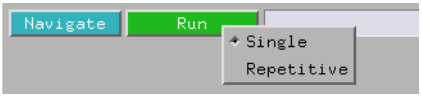


Setting Single or Repetitive Run

A single measurement will stop after memory is full or a store qualification is met. A repetitive measurement executes successive Single measurements until Stop is selected.

When a single or repetitive measurement is stopped, only data that has been captured to that point is available for viewing.

Select single or repetitive by right-clicking on the Run button in the tool's setup window.



If you have problems displaying trace data when running Repetitive measurements, see "Demand Driven Data" on page 108.

Checking Run Status

The *Run Status* dialog provides status information about the currently configured instruments, and the status of the run with respect to the trigger specification.

To access the *Run Status* dialog:

- The Run Status icon in the System Window, OR
- Navigate->System->Run Status

Demand Driven Data

When an analyzer measurement occurs, acquisition memory is filled with data that is then transferred to the display memory of the analysis or display tools you are using, as needed by those tools. In normal use, this *demand driven data* approach saves time by not transferring unnecessary data.

Since acquisition memory is cleared at the beginning of a measurement, stopping a run may create a discrepancy between acquisition memory and the memory buffer of connected tools. Without a complete trace of acquisition memory, the display memory will appear to have 'holes' in it which appear as filtered data.

This situation will occur in these cases:

- If you stop a repetitive measurement after analyzer data has been cleared and before the measurement is complete.
- If a trigger is not found by the analyzer and the run must be stopped to regain control.

To make sure all of the data in a repetitive run is available for viewing:

- In the workspace, attach a Filter tool to the output of the analyzer.
- In the Filter, select "Pass Matching Data"
- In the filter terms, assure the default pattern of all "Don't Cares" (Xs).

This configuration will always transfer all data from acquisition memory. While this configuration will increase the time of each run, it will guarantee that repetitive run data is available regardless of when it is stopped.

absolute Denotes the time period or count of states between a captured state and the trigger state. An absolute count of -10 indicates the state was captured ten states before the trigger state was captured.

acquisition Denotes one complete cycle of data gathering by a measurement module. For example, if you are using an analyzer with 128K memory depth, one complete acquisition will capture and store 128K states in acquisition memory.

analysis probe A probe connected to the target microprocessor. It provides an interface between the signals of the target microprocessor and the inputs of the logic analyzer. Also called a "preprocessor".

analyzer 1 In a logic analyzer with two *machines*, refers to the machine that is on by default. The default name is *Analyzer<N>*, where N is the slot letter.

analyzer 2 In a logic analyzer with two *machines*, refers to the machine that is off by default. The default name is *Analyzer<N2>*, where N is the slot letter.

arming An instrument tool must be armed before it can search for its trigger condition. Typically, instruments are armed immediately when *Run* or *Group Run* is selected. You can set up one instrument to arm another using the *Intermodule Window*. In these setups, the second instrument cannot search for its trigger condition until it receives the arming signal from the first instrument. In some analyzer instruments, you can set up one analyzer *machine* to arm the other analyzer machine in the *Trigger Window*.

asterisk (*) See *edge terms*, *glitch*, and *labels*.

bits Bits represent the physical logic analyzer channels. A bit is a *channel* that has or can be assigned to a *label*. A bit is also a position in a label.

card This refers to a single instrument intended for use in the HP 16600A-series or HP 16700A mainframe. One card fills one slot in the mainframe. A module may comprise a single card or multiple cards cabled together.

channel The entire signal path from the probe tip, through the cable and module, up to the label grouping.

click To click an item, position the cursor over the item. Then quickly press and release the *left mouse*

button.

clock channel A logic analyzer *channel* that can be used to carry the clock signal. When it is not needed for clock signals, it can be used as a *data channel*, except in the HP 16517A.

context record A context record is a small segment of analyzer memory that stores an event of interest along with the states that immediately preceded it and the states that immediately followed it.

context store If your analyzer can perform context store measurements, you will see a button labeled *Context Store* under the Trigger tab. Typical context store measurements are used to capture writes to a variable or calls to a subroutine, along with the activity preceding and following the events. A context store measurement divides analyzer memory into a series of context records. If you have a 64K analyzer memory and select a 16-state context, the analyzer memory is divided into 4K 16-state context records. If you have a 64K analyzer memory and select a 64-state context, the analyzer memory will be divided into 1K 64-state records.

count The count function records periods of time or numbers of state transactions between states stored in memory. You can set up the analyzer count function to count occurrences of a selected event during the trace, such as counting how many times a variable is read between each of the writes to the variable. The analyzer can also be set up to count elapsed time, such as counting the time spent executing within a particular function during a run of your target program.

cross triggering Using intermodule capabilities to have measurement modules trigger each other. For example, you can have an external instrument arm a logic analyzer, which subsequently triggers an oscilloscope when it finds the trigger state.

data channel A *channel* that carries data. Data channels cannot be used to clock logic analyzers.

data field A data field in the pattern generator is the data value associated with a single label within a particular data vector.

data set A data set is made up of all labels and data stored in memory of any single analyzer machine or instrument tool. Multiple data sets can be displayed together when sourced into a single display tool. The

Filter tool is used to pass on partial data sets to analysis or display tools.

debug mode See *monitor*.

delay The delay function sets the horizontal position of the waveform on the screen for the oscilloscope and timing analyzer. Delay time is measured from the trigger point in seconds or states.

demo mode An emulation control session which is not connected to a real target system. All windows can be viewed, but the data displayed is simulated. To start demo mode, select *Start User Session* from the Emulation Control Interface and enter the demo name in the *Processor Probe LAN Name* field. Click *Help* in the *Start User Session* window for details.

deskewing To cancel or nullify the effects of differences between two different internal delay paths for a signal. Deskewing is normally done by routing a single test signal to the inputs of two different modules, then adjusting the Intermodule Skew so that both modules recognize the signal at the same time.

don't care For *terms*, a "don't care" means that the state of the signal (high or low) is not relevant to the measurement. The analyzer ignores the state of this signal when determining whether a match occurs on an input label. "Don't care" signals are still sampled and their values can be displayed with the rest of the data. Don't cares are represented by the *X* character in numeric values and the dot (.) in timing edge specifications.

dot (.) See *edge terms*, *glitch*, *labels*, and *don't care*.

double-click To double-click an item, position the cursor over the item, and then quickly press and release the *left mouse button* twice.

drag and drop To drag and drop an item, position the cursor over the item, and then press and hold the *left mouse button*. While holding the left mouse button down, move the mouse to drag the item to a new location. When the item is positioned where you want it, release the mouse button.

edge mode In an oscilloscope, this is the trigger mode that causes a trigger based on a single channel edge, either rising or falling.

edge terms Logic analyzer trigger resources that allow detection of transitions on a signal. An edge term can be set to detect a rising edge,

falling edge, or either edge. Some logic analyzers can also detect no edge or a *glitch* on an input signal. Edges are specified by selecting arrows. The dot (.) ignores the bit. The asterisk (*) specifies a glitch on the bit.

emulation module A module within the logic analysis system mainframe that provides an emulation connection to the debug port of a microprocessor. An E5901A emulation module is used with a target interface module (TIM) or an analysis probe. An E5901B emulation module is used with an E5900A emulation probe.

emulation probe The stand-alone equivalent of an *emulation module*. Most of the tasks which can be performed using an emulation module can also be performed using an emulation probe connected to your logic analysis system via a LAN.

emulator An *emulation module* or an *emulation probe*.

Ethernet address See *link-level address*.

events Events are the things you are looking for in your target system. In the logic analyzer interface, they take a single line. Examples of events are *Label1 = XX* and *Timer 1 > 400 ns*.

filter expression The filter expression is the logical *OR* combination of all of the filter terms. States in your data that match the filter expression can be filtered out or passed through the Pattern Filter.

filter term A variable that you define in order to specify which states to filter out or pass through. Filter terms are logically OR'ed together to create the filter expression.

Format The selections under the logic analyzer *Format* tab tell the logic analyzer what data you want to collect, such as which channels represent buses (labels) and what logic threshold your signals use.

frame The HP 16600A-series or HP 16700A logic analysis system mainframe. See also *logic analysis system*.

gateway address An IP address entered in integer dot notation. The default gateway address is 0.0.0.0, which allows all connections on the local network or subnet. If connections are to be made across networks or subnets, this address must be set to the address of the

gateway machine.

glitch A glitch occurs when two or more transitions cross the logic threshold between consecutive timing analyzer samples. You can specify glitch detection by choosing the asterisk (*) for *edge terms* under the timing analyzer Trigger tab.

grouped event A grouped event is a list of *events* that you have grouped, and optionally named. It can be reused in other trigger sequence levels. Only available in HP 16715A, 16716A, and 16717A logic analyzers.

held value A value that is held until the next sample. A held value can exist in multiple data sets.

immediate mode In an oscilloscope, the trigger mode that does not require a specific trigger condition such as an edge or a pattern. Use immediate mode when the oscilloscope is armed by another instrument.

interconnect cable Short name for *module/probe interconnect cable*.

intermodule Intermodule is a term used when multiple instrument tools are connected together for the purpose of one instrument arming another. In such a configuration, an arming tree is developed and the group run function is designated to start all instrument tools. Multiple instrument configurations are done in the Intermodule window.

intermodule bus The intermodule bus (IMB) is a bus in the frame that allows the measurement modules to communicate with each other. Using the IMB, you can set up one instrument to *arm* another. Data acquired by instruments using the IMB is time-correlated.

internet address Also called Internet Protocol address or IP address. A 32-bit network address. It is usually represented as decimal numbers separated by periods; for example, 192.35.12.6. Ask your LAN administrator if you need an internet address.

labels Labels are used to group and identify logic analyzer channels. A label consists of a name and an associated bit or group of bits. Labels are created in the Format tab.

line numbers A line number (Line #s) is a special use of *symbols*. Line numbers represent lines in your source file, typically lines that have no unique symbols defined to represent them.

link-level address Also referred to as the Ethernet address, this is the unique address of the LAN interface. This value is set at the factory and cannot be changed. The link-level address of a particular piece of equipment is often printed on a label above the LAN connector. An example of a link-level address in hexadecimal: 0800090012AB.

local session A local session is when you run the logic analysis system using the local display connected to the product hardware.

logic analysis system The HP 16600A-series or HP 16700A mainframe, and all tools designed to work with it. Usually used to mean the specific system and tools you are working with right now.

machine Some logic analyzers allow you to set up two measurements at the same time. Each measurement is handled by a different machine. This is represented in the Workspace window by two icons, differentiated by a *1* and a *2* in the upper right-hand corner of the icon. Logic analyzer resources such as pods and trigger terms cannot be shared by the machines.

markers Markers are the green and yellow lines in the display that are labeled *x*, *o*, *G1*, and *G2*. Use them to measure time intervals or sample intervals. Markers are assigned to patterns in order to find patterns or track sequences of states in the data. The x and o markers are local to the immediate display, while G1 and G2 are global between time correlated displays.

master card In a module, the master card controls the data acquisition or output. The logic analysis system references the module by the slot in which the master card is plugged. For example, a 5-card HP 16555D would be referred to as *Slot C: machine* because the master card is in slot C of the mainframe. The other cards of the module are called *expansion cards*.

menu bar The menu bar is located at the top of all windows. Use it to select *File* operations, tool or system *Options*, and tool or system level *Help*.

message bar The message bar displays mouse button functions for the window area or field directly beneath the mouse cursor. Use the mouse and message bar together to prompt yourself to functions and shortcuts.

module An instrument that uses a single timebase in its operation. Modules can have from one to five cards functioning as a single instrument. When a module has more than one card, system window will show the instrument icon in the slot of the *master card*.

module/probe interconnect cable

The module/probe interconnect cable connects an E5901B emulation module to an E5900B emulation probe. It provides power and a serial connection. A LAN connection is also required to use the emulation probe.

monitor When using the Emulation Control Interface, running the monitor means the processor is in debug mode (that is, executing the debug exception) instead of executing the user program.

panning The action of moving the waveform along the timebase by varying the delay value in the Delay field. This action allows you to control the portion of acquisition memory that will be displayed on the screen.

pattern mode In an oscilloscope, the trigger mode that allows you to set the oscilloscope to trigger on a specified combination of input signal levels.

pattern terms Logic analyzer resources that represent single states to be found on labeled sets of bits; for example, an address on the address bus or a status on the status lines.

period (.) See *edge terms*, *glitch*, *labels*, and *don't care*.

pod See *pod pair*

pod pair A group of two pods containing 16 channels each, used to physically connect data and clock signals from the unit under test to the analyzer. Pods are assigned by pairs in the analyzer interface. The number of pod pairs avalaible is determined by the channel width of the instrument.

point To point to an item, move the mouse cursor over the item.

preprocessor See *analysis probe*.

primary branch The primary branch is indicated in the *Trigger sequence step* dialog box as either the *Then find* or *Trigger on* selection. The destination of the primary branch is always the next state in the sequence, except for the HP 16517A. The primary branch has an optional occurrence count field

that can be used to count a number of occurrences of the branch condition. See also *secondary branch*.

probe A device to connect the various instruments of the logic analysis system to the target system. There are many types of probes and the one you should use depends on the instrument and your data requirements. As a verb, "to probe" means to attach a probe to the target system.

processor probe See *emulation probe*.

range terms Logic analyzer resources that represent ranges of values to be found on labeled sets of bits. For example, range terms could identify a range of addresses to be found on the address bus or a range of data values to be found on the data bus. In the trigger sequence, range terms are considered to be true when any value within the range occurs.

relative Denotes time period or count of states between the current state and the previous state.

remote display A remote display is a display other than the one connected to the product hardware. Remote displays must be identified to the network through an address location.

remote session A remote session is when you run the logic analyzer using a display that is located away from the product hardware.

right-click To right-click an item, position the cursor over the item, and then quickly press and release the *right mouse button*.

sample A data sample is a portion of a *data set*, sometimes just one point. When an instrument samples the target system, it is taking a single measurement as part of its data acquisition cycle.

Sampling Use the selections under the logic analyzer Sampling tab to tell the logic analyzer how you want to make measurements, such as State vs. Timing.

secondary branch The secondary branch is indicated in the *Trigger sequence step* dialog box as the *Else on* selection. The destination of the secondary branch can be specified as any other active sequence state. See also *primary branch*.

session A session begins when you start a *local session* or *remote session* from the session manager, and ends when you select *Exit* from

the main window. Exiting a session returns all tools to their initial configurations.

skew Skew is the difference in channel delays between measurement channels. Typically, skew between modules is caused by differences in designs of measurement channels, and differences in characteristics of the electronic components within those channels. You should adjust measurement modules to eliminate as much skew as possible so that it does not affect the accuracy of your measurements.

state measurement In a state measurement, the logic analyzer is clocked by a signal from the system under test. Each time the clock signal becomes valid, the analyzer samples data from the system under test. Since the analyzer is clocked by the system, state measurements are *synchronous* with the test system.

store qualification Store qualification is only available in a *state measurement*, not *timing measurements*. Store qualification allows you to specify the type of information (all samples, no samples, or selected states) to be stored in memory. Use store qualification to prevent memory from being filled with unwanted activity such as no-ops or wait-loops. To set up store qualification, use the *While storing* field in a logic analyzer trigger sequence dialog.

subnet mask A subnet mask blocks out part of an IP address so that the networking software can determine whether the destination host is on a local or remote network. It is usually represented as decimal numbers separated by periods; for example, 255.255.255.0. Ask your LAN administrator if you need a the subnet mask for your network.

symbols Symbols represent patterns and ranges of values found on labeled sets of bits. Two kinds of symbols are available:

- Object file symbols - Symbols from your source code, and symbols generated by your compiler. Object file symbols may represent global variables, functions, labels, and source line numbers.
- User-defined symbols - Symbols you create.

Symbols can be used as *pattern* and *range* terms for:

- Searches in the listing display.

- Triggering in logic analyzers and in the source correlation trigger setup.
- Qualifying data in the filter tool and system performance analysis tool set.

system administrator The system administrator is a person who manages your system, taking care of such tasks as adding peripheral devices, adding new users, and doing system backup. In general, the system administrator is the person you go to with questions about implementing your software.

target system The system under test, which contains the microprocessor you are probing.

terms Terms are variables that can be used in trigger sequences. A term can be a single value on a label or set of labels, any value within a range of values on a label or set of labels, or a glitch or edge transition on bits within a label or set of labels.

TIM A TIM (Target Interface Module) makes connections between the cable from the emulation module or emulation probe and the cable to the debug port on the system under test.

timer terms Logic analyzer resources that are used to measure the time the trigger sequence remains within one sequence step, or a set of sequence steps. Timers can be used to detect when a condition lasts too long or not long enough. They can be used to measure pulse duration, or duration of a wait loop. A single timer term can be used to delay trigger until a period of time after detection of a significant event.

time-correlated Time correlated measurements are measurements involving more than one instrument in which all instruments have a common time or trigger reference.

timing measurement In a timing measurement, the logic analyzer samples data at regular intervals according to a clock signal internal to the timing analyzer. Since the analyzer is clocked by a signal that is not related to the system under test, timing measurements capture traces of electrical activity over time. These measurements are *asynchronous* with the test system.

tools A tool is a stand-alone piece of functionality. A tool can be an instrument that acquires data, a display for viewing data, or a post-processing analysis helper. Tools are represented as icons in the main

window of the interface.

toolbox The Toolbox is located on the left side of the main window. It is used to display the available hardware and software tools. As you add new tools to your system, their icons will appear in the Toolbox.

tool icon Tool icons that appear in the workspace are representations of the hardware and software tools selected from the toolbox. If they are placed directly over a current measurement, the tools automatically connect to that measurement. If they are placed on an open area of the main window, you must connect them to a measurement using the mouse.

trace See *acquisition*.

trigger Trigger is an event that occurs immediately after the instrument recognizes a match between the incoming data and the trigger specification. Once trigger occurs, the instrument completes its *acquisition*, including any store qualification that may be specified.

trigger sequence A trigger sequence is a sequence of events that you specify. The logic analyzer compares this sequence with the samples it is collecting to determine when to *trigger*.

trigger specification A trigger specification is a set of conditions that must be true before the instrument triggers.

workspace The workspace is the large area under the message bar and to the right of the toolbox. The workspace is where you place the different instrument, display, and analysis tools. Once in the workspace, the tool icons graphically represent a complete picture of the measurements.

zooming In the oscilloscope or timing analyzer, to expand and contract the waveform along the time base by varying the value in the s/Div field. This action allows you to select specific portions of a particular waveform in acquisition memory that will be displayed on the screen. You can view any portion of the waveform record in acquisition memory.

Index

Index

Index