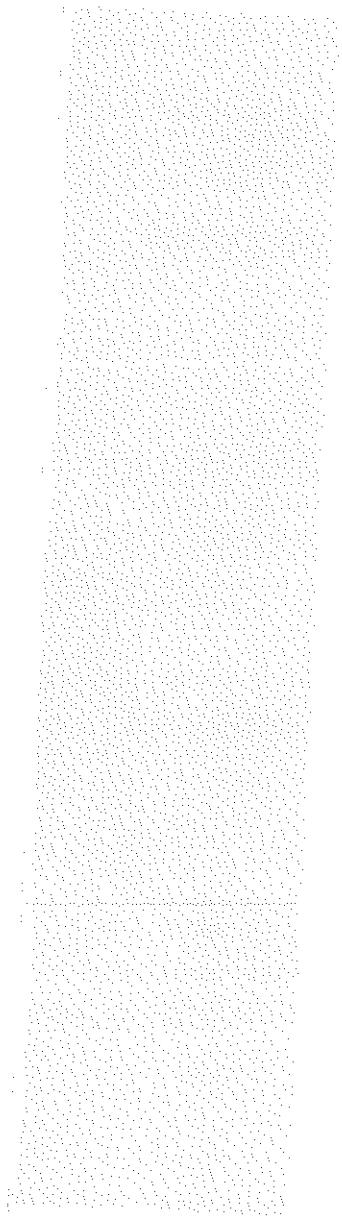
LOGIC ANALYZERS
PM 3580 / PM 3585
Getting started guide

PHILIPS

Dual logic analysis

PF8690/00 System Software
Software Version 1.0, English
I&E, Test & Measurement

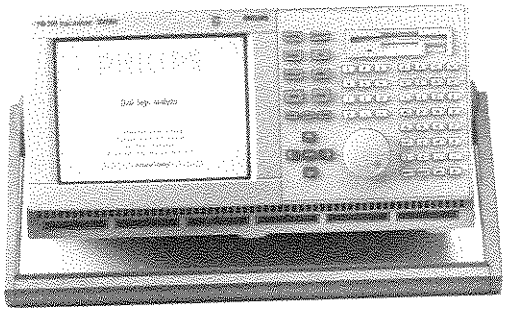
Publication Number 4022 104 90151



Printed in the Netherlands

Table of Contents

Introduction



Welcome to the new PM 3580 / PM 3585 family of logic analyzers from PHILIPS.

The PM 3580/PM 3585 logic analyzers were designed to give you easy and simple access to state of the art performance, and even more.

The family consists of four different models.

PM 3580/30: 32 dual-analysis channels with 100 MHz timing, 50 MHz state, and 1k deep acquisition memory.

PM 3580/60: 64 dual-analysis channels with 100 MHz timing, 50 MHz state, and 1k deep acquisition memory.

PM 3585/60: 64 dual-analysis channels with 200 MHz timing, 50 MHz state, and 2k deep acquisition memory.

PM 3585/90: 96 dual-analysis channels with 200 MHz timing, 50 MHz state, and 2k deep acquisition memory.

The key features of all these models are as follows:

- "Dual Analysis Per Pin (DAPP)" architecture (simultaneous correlated state and timing acquisition on all channels)
- Transitional timing on all channels
- Powerful triggering functionality integrating state & timing trigger functions in the same trigger sequence
- Eight-level sequencer with full conditional structure (If...Then...Else)
- Eight state trigger words
- One range recognizer
- Three timing trigger words
- One edge detector
- One glitch detector
- Absolute or relative time stamp, always at full speed (5 ns resolution)
- Powerful selective data acquisition functions.

Not all these features are covered in this manual. For additional information, please refer to the *PM 3580/PM 3585 User Manual* and *PM 3580/PM 3585 Reference Guide.*

This *Getting Started Guide* is intended for someone who is using a PM 3580/PM 3585 analyzer for the first time. It is written for both novice and experienced users of logic analyzers.

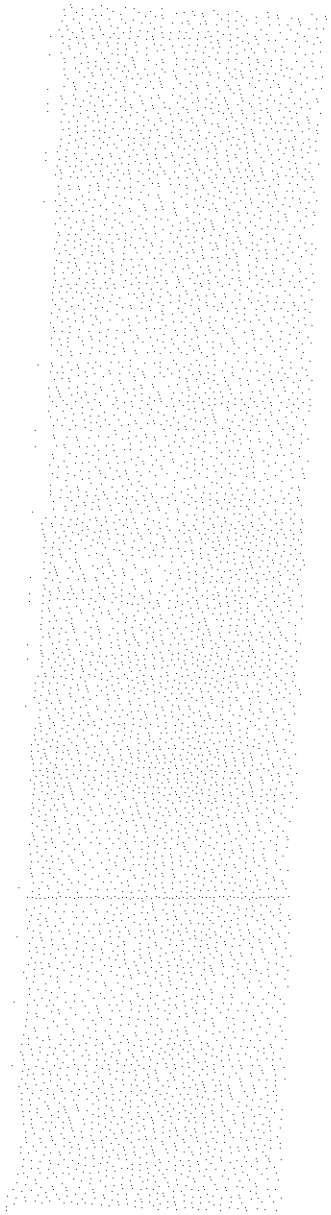
This manual will give you the basic knowledge on these instruments, based on real life examples. After reading this manual, you will be able to make measurements with the PM 3580/PM 3585 instruments.

The logical follow-up of the *Getting Started Guide* is the *User Manual.* This gives you complete information on what you can do with PM 3580/PM 3585, and it tells you how to do it.

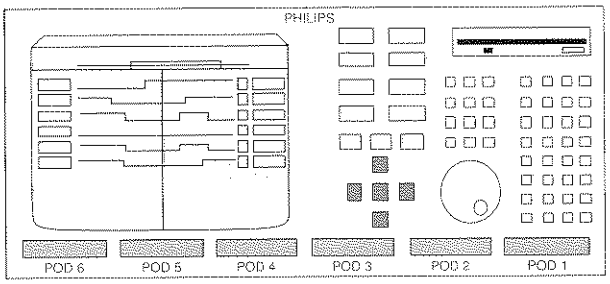
A *Reference Guide* is also supplied with the instrument. In this manual you can find an explanation of any option of any field in any menu of the analyzer.

If you are new to logic analyzers or if you just need a refresher, we think you will find that *The ABC's of logic analysis* is a good way to get a clear view of the applications of logic analyzers and how to get the most out of them.

The ABC's of logic analysis can be ordered from your local Fluke/Philips sales representative. (It has code number 9498 718 02611.)



Exploring the Instrument

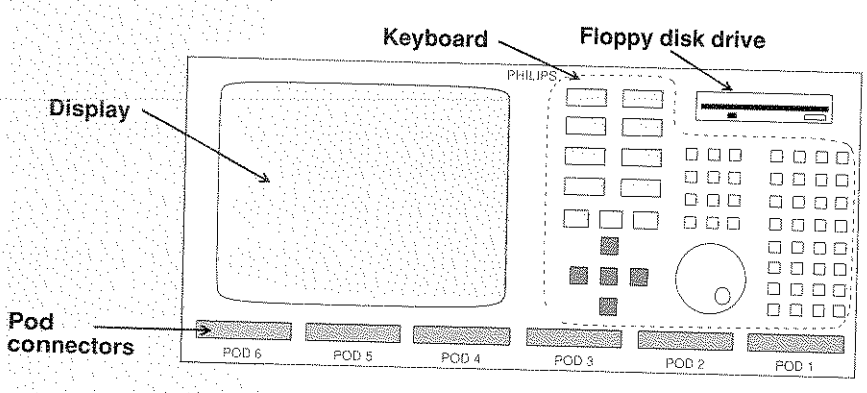


This chapter explains the layout of the instrument and shows you the basic principles of the User Interface.

Front Panel

The Front Panel of the Logic Analyzer (as shown at the bottom of this page) consists of four areas:

- The display screen on the left displays the menus, operating information, and acquisition results.
- The keyboard in the center and right is used to enter commands.
- The floppy disk drive in the upper right is used for the system floppy disk and for saving and restoring data.
- The pod connectors at the bottom are used to connect the signals to be measured to the analyzer. Each pod connector carries 16 signals. Depending on the channel width of your instrument, you can see two, four or six pod connectors at the bottom of the front panel.



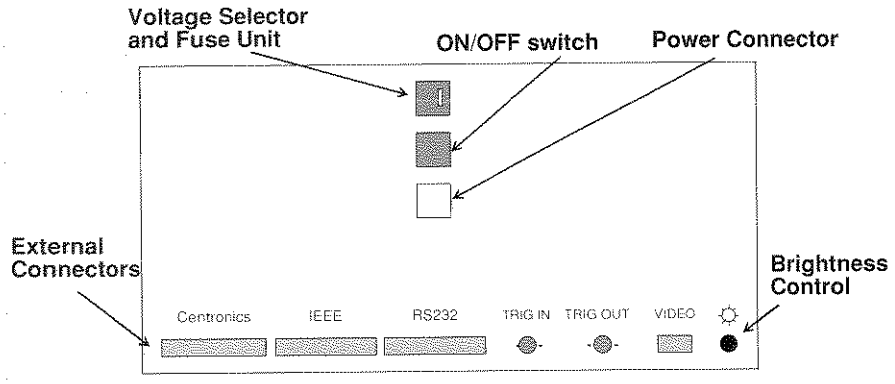
Rear Panel

On the rear panel shown below you will find:

- The voltage selector and fuse unit.

> **CAUTION**
>
> The selected voltage MUST match your line (mains) voltage: otherwise, you can damage the instrument. See the *Installation* procedures for instructions on selecting the correct voltage.

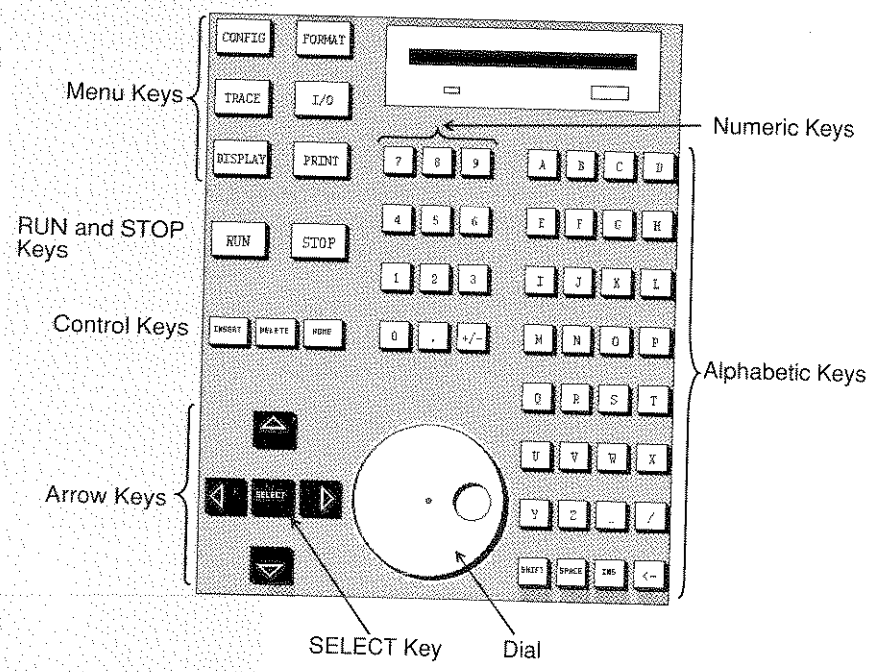
- The instrument ON/OFF switch.
- The connection for the power cable.
- External connectors for connecting a Centronics printer, for remote communication, for exchange of a trigger pulse with other instruments, and for connecting an external 32 kHz monochrome monitor (MVGA). For a complete description of these connectors, see the *User Manual.*
- Brightness control to increase or decrease the brightness level of the display.



Keyboard

You tell the instrument what to do via the keyboard, and the instrument tells you what is happening by the various displays on the screen.

The PM 3580/ PM 3585 Logic Analyzer keyboard is logically grouped as shown below.



The main controls are those on the left side of the keyboard, the six menu, *RUN*, *STOP*, three control, four arrow, and *SELECT* keys.

The six menu keys each select a major menu (screen display). Three of these, *CONFIG*, *FORMAT* and *TRACE*, are setup menus. The *DISPLAY* key selects the screen on which the measurement results are shown. The other two,

PRINT and *I/O*, are auxiliary screens. For more information, see the *PM 3580/PM 3585 User Manual.*

Now, you should connect the instrument and take a short tour around the menus.

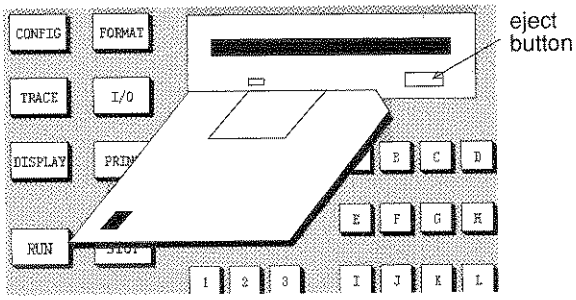
Power On

First, ensure that the instrument is set up for the correct voltage (see page 11 and the *Installation* procedures delivered with the instrument).

Ensure the power cable is not connected to the power supply. Plug the female end into the instrument, and ensure that the power switch is *OFF*.

Plug the power cable into an appropriate ***grounded*** power source.

Remove the transport protector (if any) from the floppy disk drive by pressing the eject button.



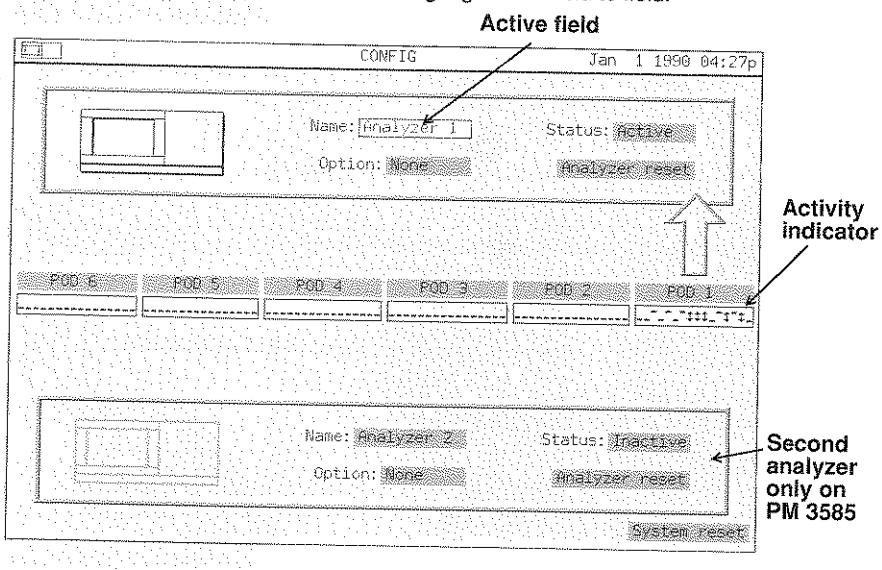
Set the power switch to *ON*. The light on the disk drive lights up, and a start-up message appears on the screen.

Now insert the system floppy disk (see the *Installation* procedures for details) fully into the floppy disk drive until it clicks.

The analyzer now loads the system software. Wait until the Configuration menu (similar to that shown on the next page) appears.

Experimenting with Fields

On each menu, there are a number of ***fields.*** These are the small white or gray boxes containing text. The currently active field is the one with a white (highlighted) background. That is the Analyzer name field in the Configuration menu shown below. Key press actions only affect highlighted fields. You use the arrow keys or the dial to move the highlight from field to field.



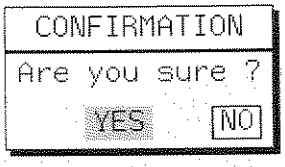
Status: Active

Press the right arrow key once. The Status field should be highlighted as shown at the left.

To do something with the instrument, you will select the appropriate menu, highlight the appropriate field, and then press the appropriate key to do the action you want done. During the learning phase, this will most often be the *SELECT* key. You can also think of the *SELECT* key as a kind of help function.

Press the *SELECT* key. The word *Active* is toggled to *Inactive*. Press the *SELECT* key again, and the Status is again *Active*. The status field is a simple toggle field.

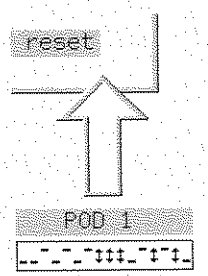
Confirmation Fields



Press the down arrow key. The *Analyzer reset* field should be highlighted. Press the *SELECT* key. A Confirmation menu, as shown at the left pops up in the middle of the screen. You can highlight the answer with the left or right arrow key, as appropriate, and select the highlighted answer by pressing the *SELECT* key.

The popup is then closed and the action is taken or not according to your answer. (Either answer is O.K. at this moment.) The *Yes* and *No* fields are function fields, i.e., the action is performed when they are selected.

Pod Field



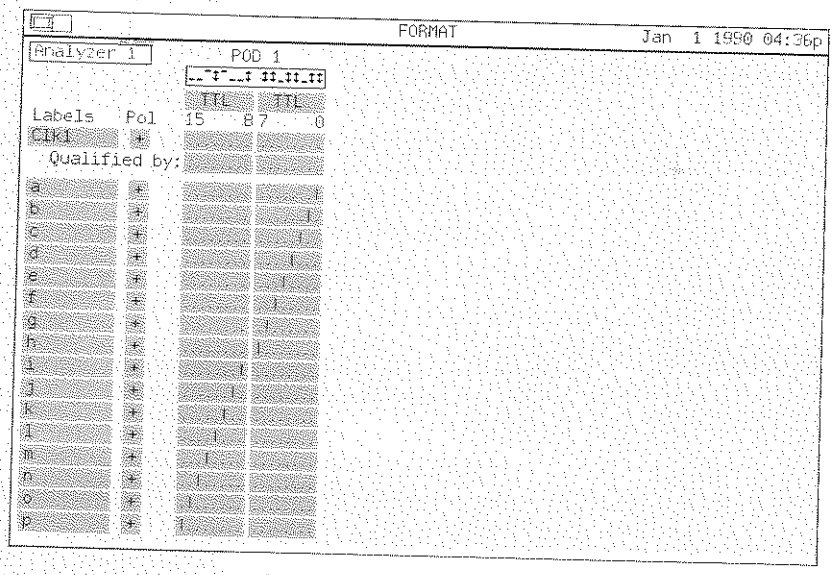
Now press the down arrow key again. The Pod field for Pod 1 should be highlighted. Press the left arrow key once to move to Pod 2.

Now press *SELECT*. You should see an arrow, as for Pod 1 in the figure at left, pointing from the Pod box to the analyzer (1) box.

On PM 3580 instruments, if you press the *SELECT* key again, the arrow is toggled off. On PM 3585 instruments the arrow is toggled to the second analyzer box, and a third press of the *SELECT* key toggles the arrow off.

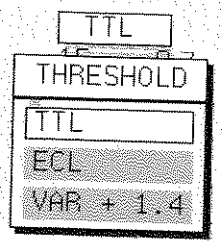
Other Field Types

To look at some other field types, press the *FORMAT* key. The Format Menu is shown.



As you will notice, the first time you come to a menu, the highlighted field is always that at the top left. However, once you have moved that highlight, as long as you do not remove power, the instrument remembers where you were. You can see that now by pressing the *CONFIG* key. Then come back to the Format menu.

A List Field



- Use the arrow keys or dial to move to the first threshold field (reading "TTL"). These fields show the threshold values selected.
- Press the *SELECT* key. A list appears showing you the different options available (TTL, ECL, or variable) as shown at left.
- Use the arrow keys or the dial to move to the option you want and press *SELECT*.

You may choose any option here.

There are other field types. These are for advanced use. See the *User Manual* chapter "Menu Overview" and the *Reference Guide* for more information.

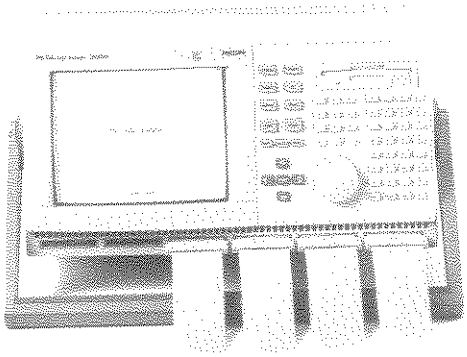
Connecting Signals to be Measured

You will use the pod system (pod cable plus front end) to connect the signals you want to measure to your analyzer. This section describes the pod system only briefly. For a detailed description of the pod system and probing, refer to the "Probing" chapter of the *User Manual*.

CAUTION

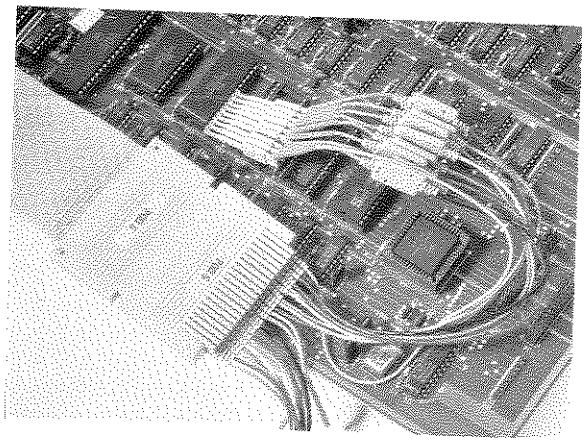
Do not use any other probe leads or cables than those delivered with your instrument, as damage may occur to your instrument if you do so.

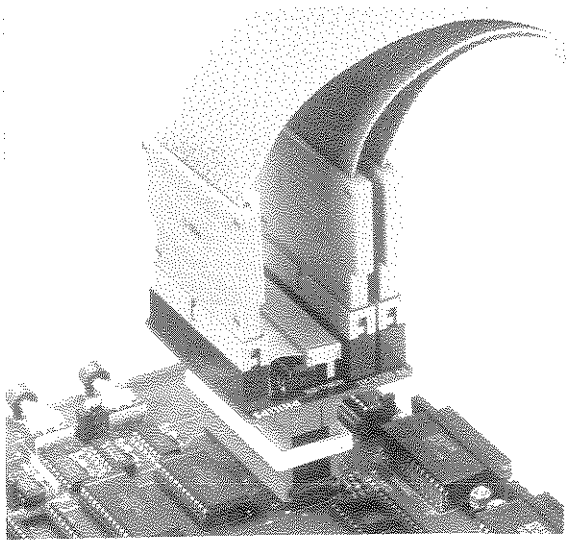
The specially-designed pod cable connects at one end to a pod connector on the Logic Analyzer.



The other end of the pod cable is connected to a target in one of three ways:

- Connect the standard front end (plug with detachable leads) to the pod cable. You can then use the grabbers delivered with the logic analyzer to attach the probe leads to those pins of the ICs you want to observe (see photo below). Alternatively, use only the small probe tips at the end of each lead and push them onto the wire wrap pins on your printed circuit board or onto the pins of a measuring clip.
- Connect the pod cable directly to an RC connector on a microprocessor adapter to measure microprocessor signals (see photo on next page).
- Connect the pod cable directly to an RC connector that has been designed-in as part of an application.





Summary

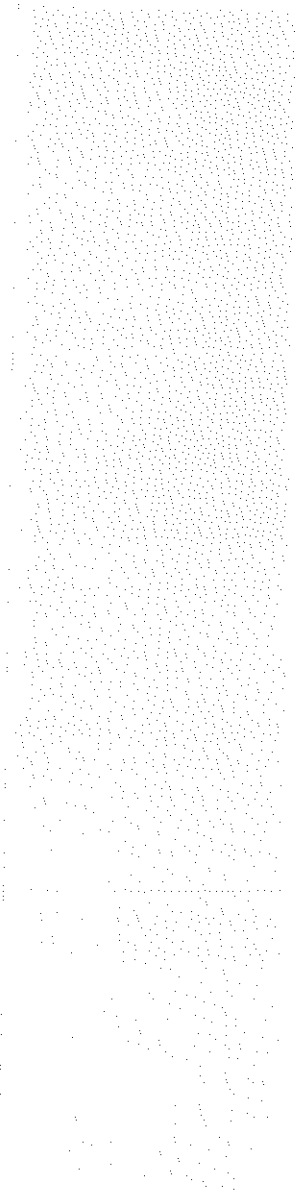
This has completed a preliminary look around the instrument. Now it is time to do some measurements.

Before moving on, you may want to look at other aspects of the instrument. If you do, we recommend that you avoid using the *PRINT* or *I/O* keys until you are familiar with their operation (see "Making a Printout" beginning on page 68 and "Getting Examples from the Disk" beginning on page 70).

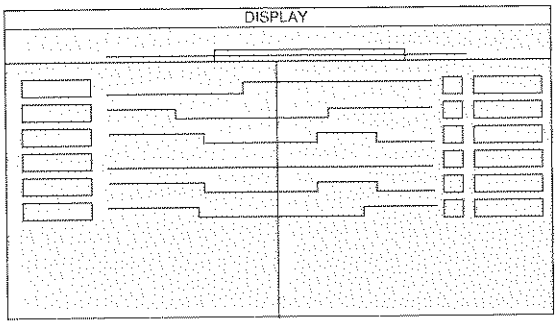
When you have finished, press the eject button to remove the floppy disk from the instrument, and power off the instrument.

It is good practice to remove the floppy disk whenever the instrument is not running, and to remove the power from the instrument when it is not required.

Note: By powering down the instrument before performing the operations in the next chapter, you will clear any of the settings made during your experiments in this chapter.



Timing Acquisition



The Purpose of Timing Acquisition

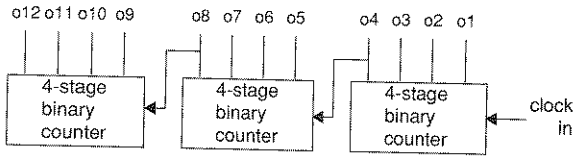
In timing acquisition, logic analyzers sample the input channels at a fast fixed rate.

In this mode, the analyzer behaves like a digital storage oscilloscope with a very limited vertical resolution (1 bit). Because of that, you lose the analog shape of the signals you capture, but you keep all the timing relationships between them.

> If you want to know more about timing analysis, please refer to *The ABC's of logic analysis.*

A Practical Problem

You are going to check if a 12-stage binary ripple counter works properly.



First you will use the logic analyzer to determine if the counter counts properly (frequency divided by 2 at each output). You will also measure the delay between the toggling of different outputs.

Logic Target PF 8669/20

The measurements described in this and the following chapters are all based on the logic target PF8669/20, which contains a 12-stage binary ripple counter.

The complete documentation for this logic target is supplied with it, and may be inserted at the back of the *User Manual.*

If you do not have this logic target, you can use the files containing the results of the measurements. These files are available on the system disk supplied with your instrument. The chapter "Getting Examples from the Disk" beginning on page 70, explains how you access these files.

The "Probing" chapter of the *User Manual* provides detailed information on the pod system and probing.

Connecting the Logic Target

The logic target has an RC connector designed-in. To connect the logic target to your analyzer, proceed as follows:

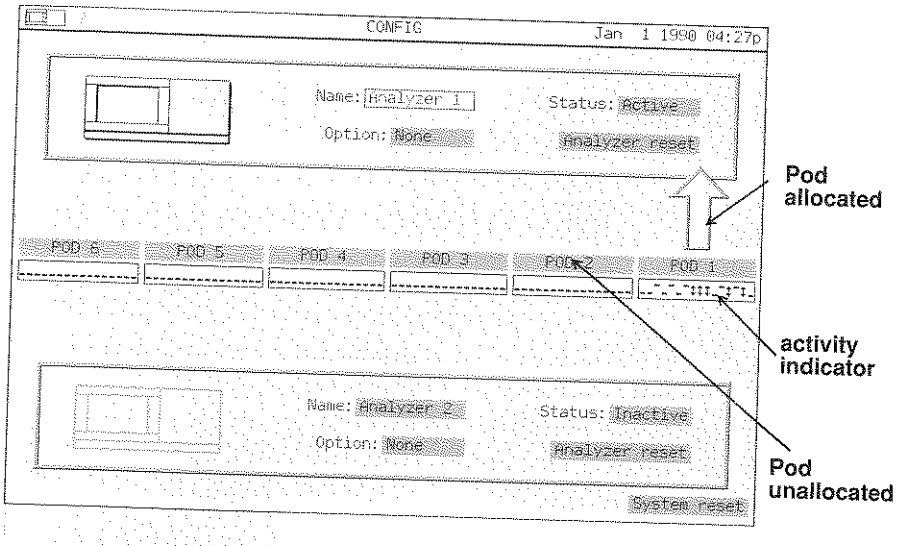
- Connect a pod cable directly to the RC connector on the logic target. If the standard front end is connected to the pod cable, first remove it.
- Connect the other end of the pod cable to pod connector 1 on your analyzer (bottom right on the Front Panel).
- Set the Mode switch of the logic target to position "I".
- Set the switch for the clock frequency into position "15 MHz".

Your logic target has now been properly set up and connected for the measurements you are going to do.

Setup and Acquisition

Once the analyzer is powered up, the first menu, called the Configuration menu, appears. This menu is used to set up the main configuration of a measurement session, i.e., the assignment of pods to an analyzer. Activity indicators are also displayed here, showing the activity of the input channels. Signal level is high (‾), low (_), or changing (‡).

Note that if you have a PM 3580, you have only Analyzer 1 and the box below the activity indicators relating to Analyzer 2 is not shown.



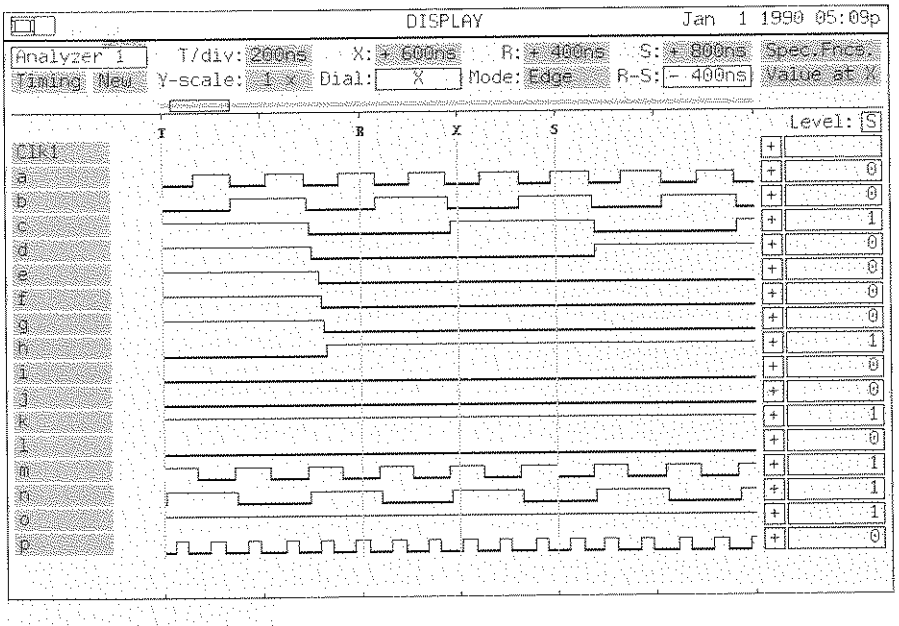
By default, only POD 1 is allocated to acquisition. This is sufficient for this test, because you need only 13 signals (12 output signals o1 .. o12, plus the clock signal) to characterize this target. So you can make an acquisition immediately and see signals displayed on the screen.

The Timing Display

To start the acquisition, just press the RUN key; almost immediately you will get a timing diagram of your signals on the screen similar to that shown below.

> If you want to experiment with the same data but you do not have the logic target available, you can get the data from the floppy disk under the file name TIMING.NEW. Please refer to "Getting Examples from the Disk" beginning on page 70 for how to load this file.

A first visual check shows you that the target behaves like a binary counter: each line toggles at a frequency that is half that of the previous line. So it seems that the target is made of cascaded dividers by two, i.e. a binary counter.



Specifying Signal Names

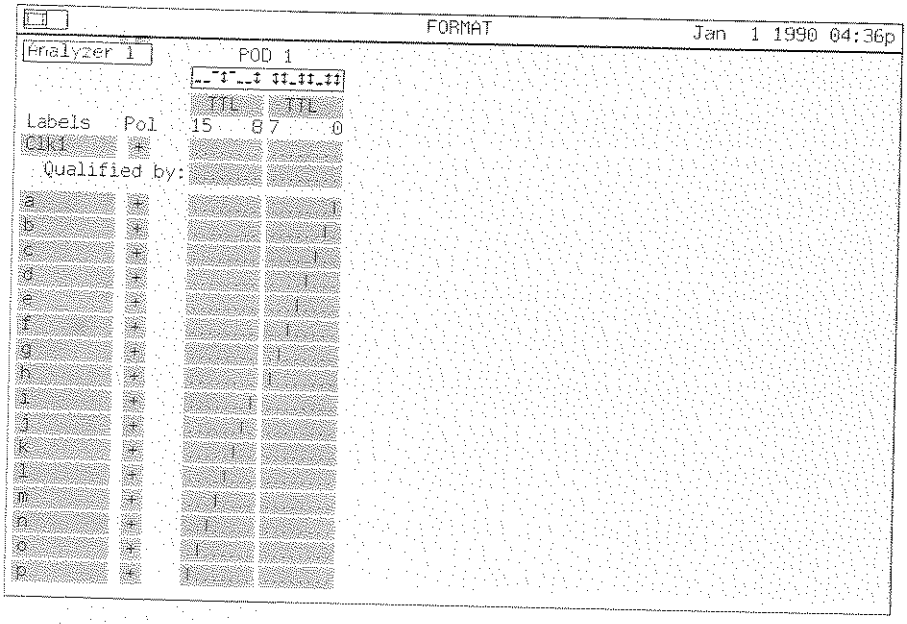
Before measuring the parameters of the signals, you can customize the display to your needs.

If you do not specify signal names, the analyzer automatically assigns default names to each channel.

The Format Menu

Using the Format menu, you can define your own labels for channels or groups of channels.

Now display the Format menu by pressing the *FORMAT* key. Note that the activity indicators you previously saw on the Configuration menu are also present on the Format menu.

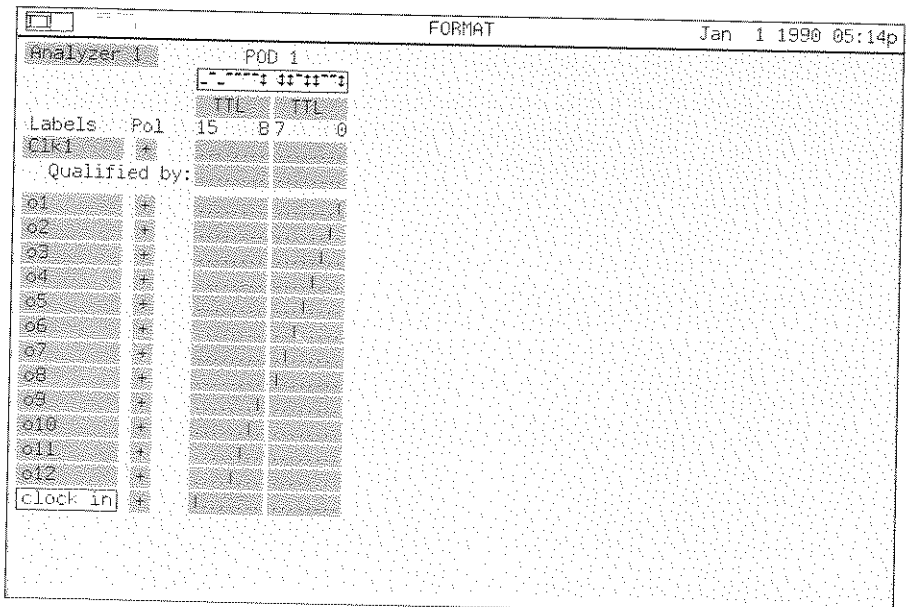


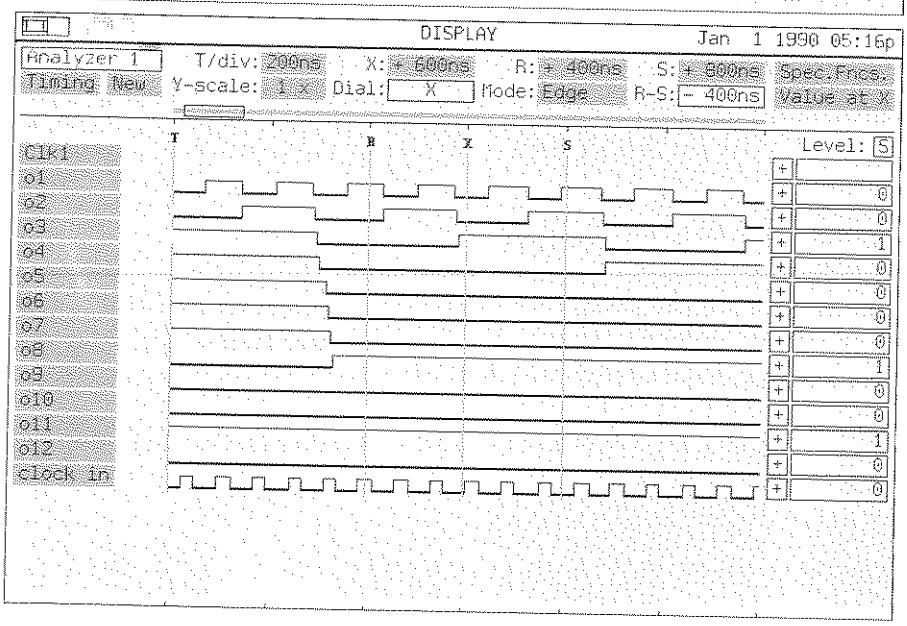
To re-label the channels, proceed as follows:

- Use the arrow keys or the dial to go to the first data label, called ***a*** by default.
- Type in the new name, *o1*, using the alphabetic and numeric keys.
- Do the same for the following labels, between ***b*** and ***l***, and replace them by *o2* to *o12*.
- The labels ***m*** through ***o*** are allocated to currently irrelevant input channels. Delete these unused labels:
  - Use the arrow keys or the dial to go to the label you want to delete.
  - Press the *DELETE* key; the label is removed.

- Label ***p*** is allocated to channel 15 by default. This channel carries the counter's clock signal. Rename this label to *clock in*.

Now that the Format menu fits your needs (see top of next page), you can go back to the Display menu (bottom of next page) to make some measurements.


FORMAT
Jan 1 1990 05:14p
Analyzer 1
POD 1
TTL
TTL
Labels
Pol
15 8 7 0
Clk1
Qualified by:
o1
o2
o3
o4
o5
o6
o7
o8
o9
o10
o11
o12
clock in

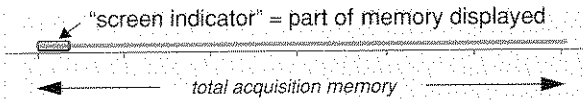

DISPLAY
Jan 1 1990 05:16p
Analyzer 1
T/div: 200ns
X: + 600ns
R: + 400ns
S: + 800ns
Spec.Fncs.
Timing New
Y-scale: 1 x
Dial: X
Mode: Edge
R-S: - 400ns
Value at X
Level: 5
Clk1
o1
o2
o3
o4
o5
o6
o7
o8
o9
o10
o11
o12
clock in

Measurements

First you measure the ripple time through the flip-flops of the counter.

To do this, you use four functions of the Timing Display menu: T/div, the X cursor, the R cursor, and the S cursor.

On top of the data area of the Timing Display menu, you can see a line with a screen indicator that shows which part of the memory is displayed on the screen.



X: + 11.1us

The X field allows you to display different parts of the data captured (change the position of the *screen indicator* in memory), i.e., to move along the time axis in the measurement.

T/Div: 10 us

T/div allows you to change the horizontal scale on the screen, and thus the size of the *screen indicator*.

Now look at the interesting "event" at the left of the screen: a lot of output toggling.

- Use the dial to move the screen indicator and get this event at the center of the screen. The dial is linked by default to changes of X.
- Then, use the arrow keys to highlight the scale field (T/div).

This enables the dial to change the scale of the screen divisions. Turning it clockwise expands the data on screen and the screen indicator gets smaller. You can also type in the value of the scale you want to get, for example 20n (n for nanosecond).

Measuring a Time Difference

Once the interesting part of the data is on the screen at the proper scale, you can easily measure timing parameters with the two cursors (R and S).

R: + 11.1us S: + 11.1us

R-S: + 0us

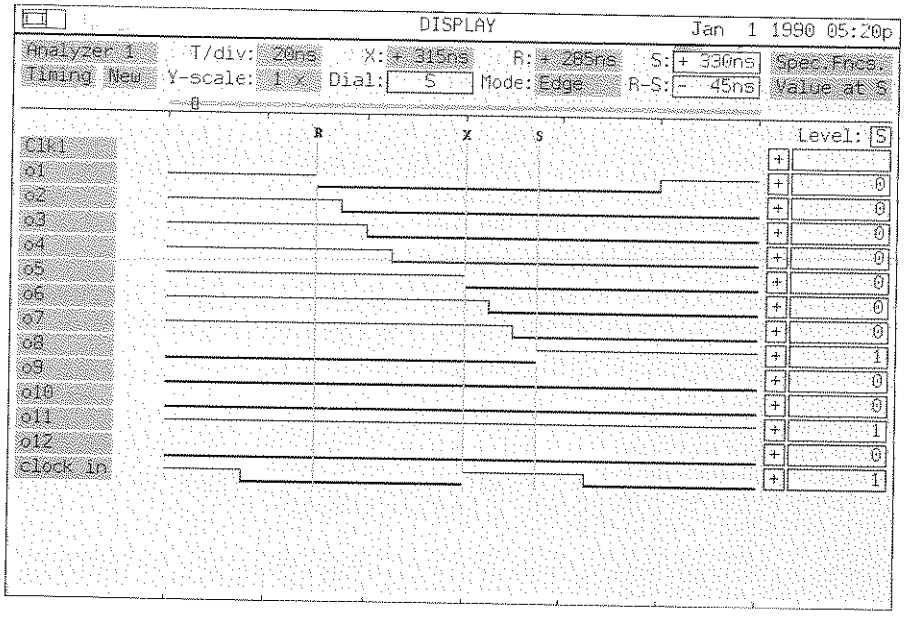
On the top line of the screen, right of center, there are two fields showing from left to right the exact position of the R and S cursors. (You will modify them later.) Below the S field, is a field (R-S) that shows the time interval between the R and S cursors.

To move the R or S cursor, first use the arrow keys to highlight the corresponding field. Then you can move the cursors:

- By using the dial.
- By making a direct numeric entry (as for T/div).
- By pressing *SELECT* to get a list on which you can select a predefined position (begin, center, end, etc.).
- By pressing the first character of one of the predefined positions on the list (e.g., X for the X position).

Back to the practical measurement:

- Use the arrow keys to highlight the R field.
- Type X to position the R cursor to the X cursor in the middle of the screen.



- Use the dial to move the R cursor to the falling edge of the first output (o1). .
- Use the same procedure to position the S cursor to the rising edge of the last changing signal (o8).

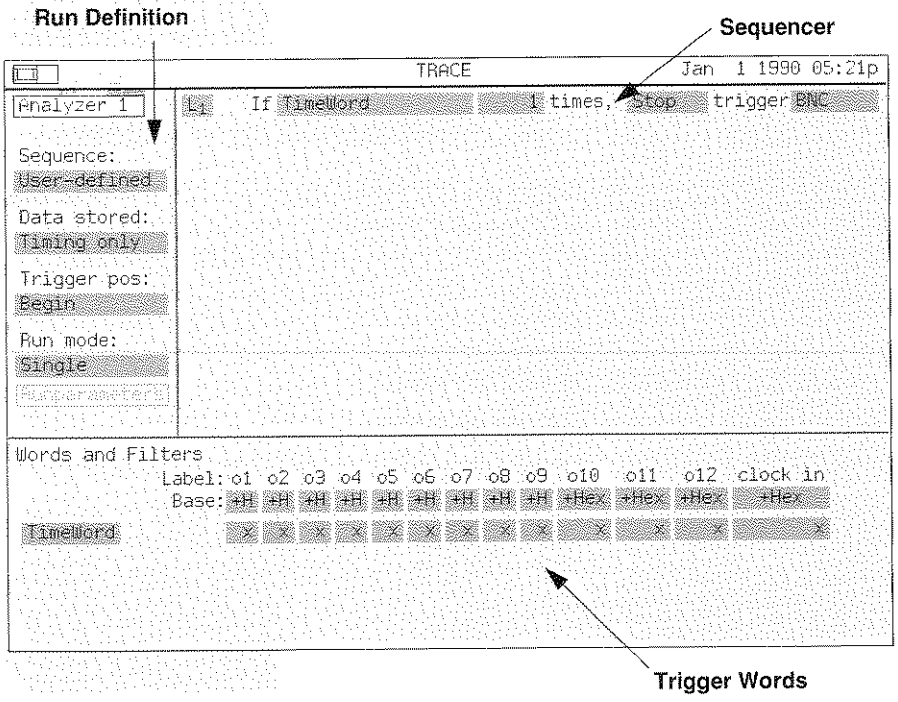
The time elapsed between these two events is displayed in the R-S field (45 ns):

The target needs at least 45 ns to toggle its first eight outputs.

Specifying a Trigger Condition

The acquisition you made before used the default trigger condition to trigger the analyzer. If you want the analyzer to trigger on a special condition, you must specify that condition. This is the purpose of the Trace menu.

- Press the *TRACE* key to access the Trace menu.



The Trace Menu

The Trace menu is divided into three menu areas:

- Run definition
- Sequencer
- Trigger words

Menu Areas of the Trace Menu

In the run definition area, you can choose various acquisition parameters, such as the position of the trigger point in the memory.

In the sequencer area, you can define the "triggering scenario".

In the trigger words area, you can define each of the events of the scenario; they are called trigger words.

An Example

For example: to see the end of one count cycle and the beginning of the next set the trigger condition such that the analyzer will trigger when all signals are high (i.e. at the end of a cycle).

The first thing to do is to choose the "scenario": the trigger sequence. The default trigger sequence (TimeWord) fits perfectly:

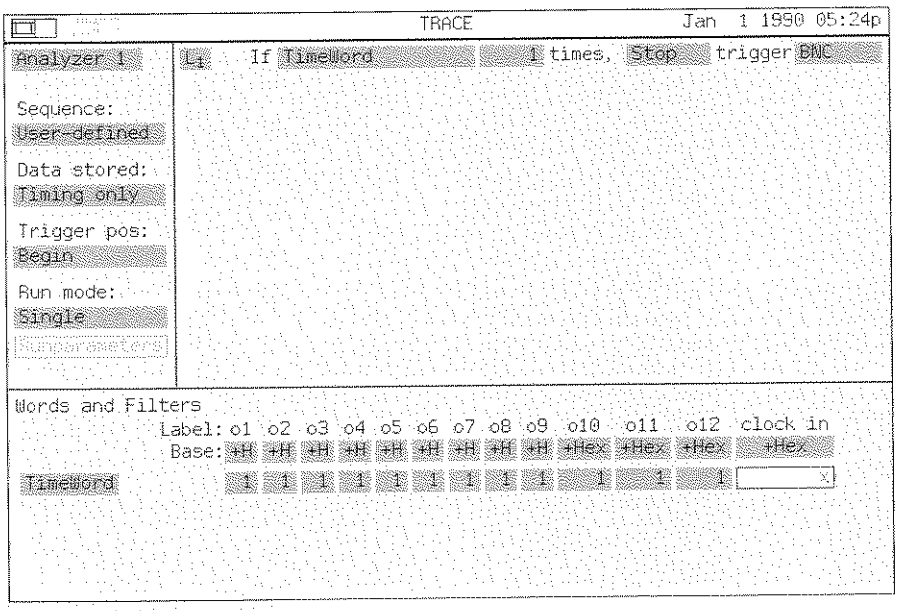
This means that the analyzer will sample and store data until the trigger condition is met. The trigger condition is met when the pattern is the same as the TimeWord.

Defining the Events

After you have chosen the sequence, you must define each of its events. In this case, it is simple, since only one trigger word is used, and the value of this trigger word is: "all output lines high".

To define the events:

- Use the arrow keys to go to the trigger words area.
- Go to the definition line of TimeWord.
- Type 1 in all the fields allocated to the outputs of the flip-flops (o1 - o12).



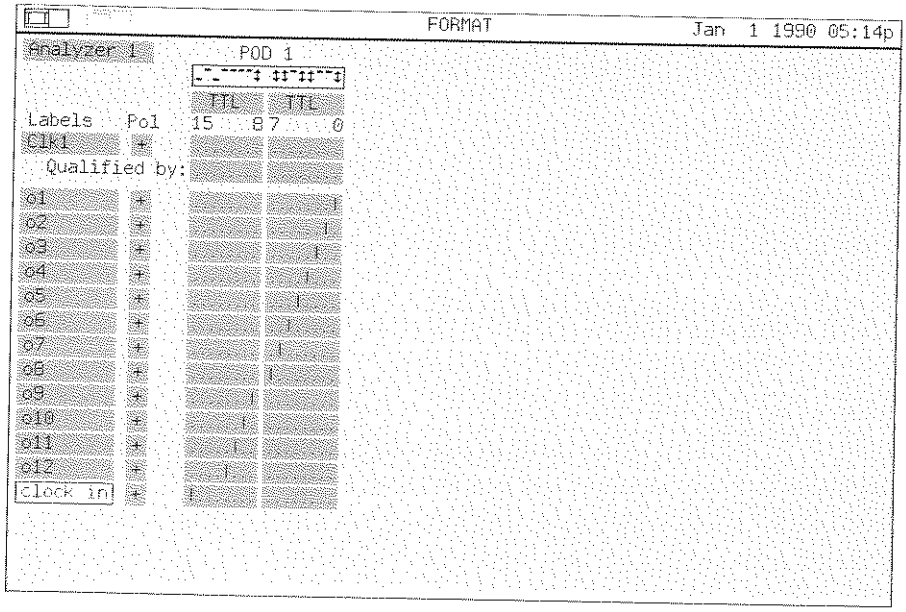
Now that the Trace menu is filled in with the right parameters, press the RUN key to make the acquisition.

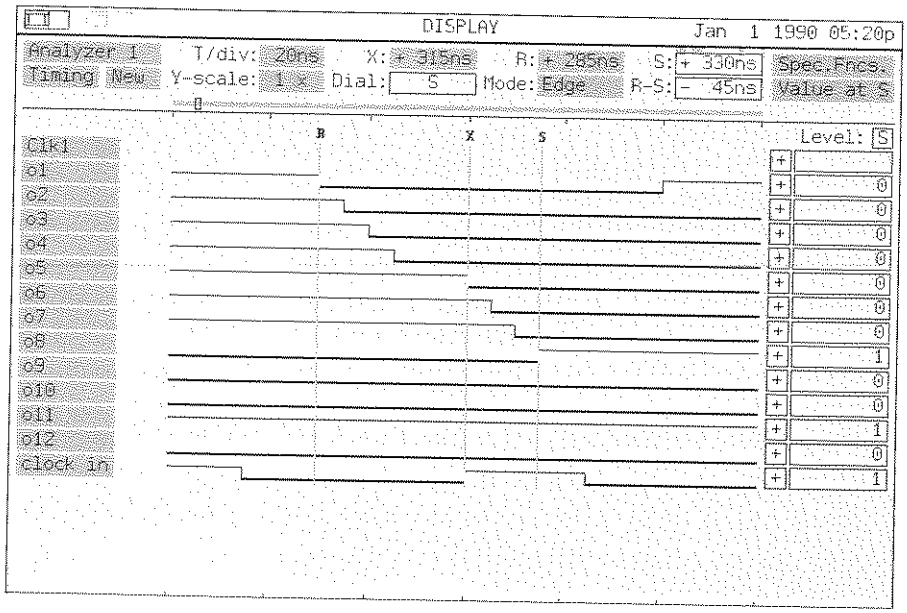
> An acquisition done with this trigger sequence is available on disk with the file name TIMETRIG.NEW. Please refer to "Getting Examples from the Disk" beginning on page 70 for instructions on how to recall this file.

Summary

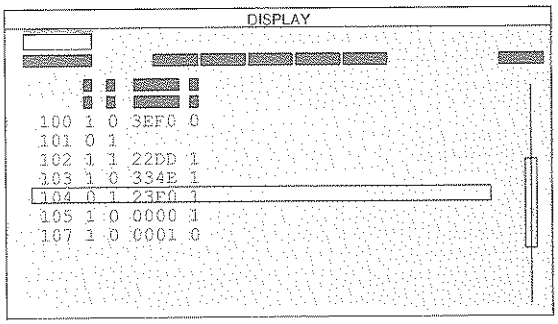
With timing acquisition, you can display and measure the timing relationships between digital signals. On the next page, you can see how the Format and Display menus were set up.

The next chapter explains another method of looking at the target: state acquisition.


FORMAT
Jan 1 1990 05:14p
Analyzer 1
POD 1
TTL
TTL
Labels
Pol
15 8 7 0
Clk1
Qualified by:
o1
o2
o3
o4
o5
o6
o7
o8
o9
o10
o11
o12
Clock in


DISPLAY
Jan 1 1990 05:20p
Analyzer 1
Timing New
T/div: 20ns
X: + 315ns
R: + 285ns
S: + 330ns
Spec.Fncs.
Y-scale: 1 x
Dial: 5
Mode: Edge
R-S: - 45ns
Value at S
B
X
S
Level: S
Clk1
o1
o2
o3
o4
o5
o6
o7
o8
o9
o10
o11
o12
Clock in

State Acquisition



The Purpose of State Acquisition

With state acquisition, logic analyzers store information strobed using an external signal coming from the application itself.

This clock signal is chosen such that, at the time it changes, all the important signals of the system have a stable and defined state.

Therefore, with state acquisition, you store only meaningful states of the system you are observing but you lose the continuous view you have with timing acquisition.

> If you want to know more about state analysis, please refer to *The ABC's of logic analysis.*

The Problem

After you have verified that the output lines of the counter toggle properly, you must check to see if it counts properly.

As mentioned before, to make a state acquisition, you use an external signal as a clock. In this case, it will be the clock input of the counter.

On each falling edge of this signal, the outputs change, incrementing the binary number they represent. The outputs must have stabilized before the next falling edge. Therefore you use the falling edge of the clock signal as the external clock for the analyzer.

If the target counts properly, the screen will show a list of sequential numbers (0,1,2,.....,FFF and back to 0).

Connecting the Target

Your logic target should still be connected to pod connector 1 as described in the previous chapter.

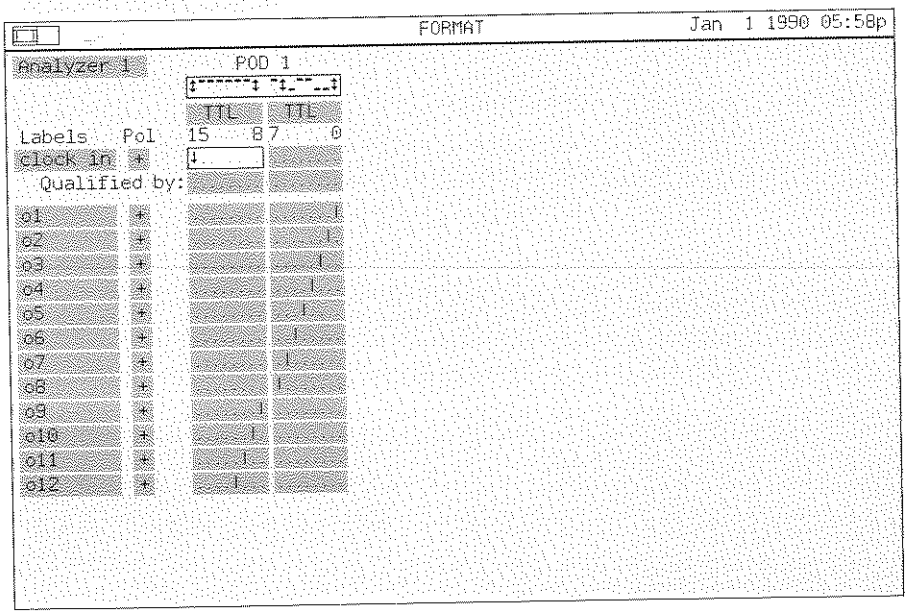
Specifying External Clocks

Once you have connected the signal used as an external clock, you must tell the analyzer which of its inputs is used as the external clock. Therefore we have to remove the

normal label we called *clock in*, and define the channel as an external clock instead. This allocation is done in the Format menu.

To make the allocation:

- Press the *FORMAT* key to go to the Format menu.
- Use the arrow keys or dial to move the cursor to the label *clock in* and press the *DELETE* key to remove it.
- Use the arrow keys or dial to move the cursor to the field called *Clk1*.
- Use the alphabetic keyboard to change the name to *clock in*.
- Use the arrow keys or dial to move the cursor to the right until you reach the allocation field containing channel 15.
- Press *SELECT* to enter edit mode.
- Use the arrow keys to move the cursor to channel 15.
- Press *SELECT* until the down arrow symbol (for falling edge) is displayed.



Specifying a Trigger Condition

Since you want to see a counting sequence from the beginning, you need to trigger the analyzer for example, on the last value of a sequence (i.e., all outputs set to 1). You can do that on the Trace menu. You can also see that the trigger scenario is almost the same as for the last timing example.

The only change is, instead of using TimeWord as the trigger word, you will use a trigger word synchronized with the input clock, like sw_1.

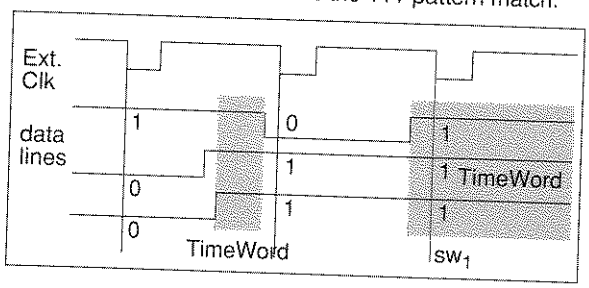
Synchronous and Asynchronous Trigger Words

What is the difference? Suppose that, if all the lines do not toggle between two stable states at the same time, you can have one or several transient states (thus of short duration).

If you specify a trigger condition with TimeWord, which is totally asynchronous from the external world (i.e., the target system), the analyzer would be able to trigger on one of these transients. A state trigger word, such as sw_1, is examined at the same time as the external clock you use on the target. This is a way to be sure to trigger on well-defined events.

The diagram below shows a typical situation where you are matching on a 111 signal.

The shaded areas show where the TimeWord would match. The vertical dotted lines are where data are sampled on the falling edge of the clock. Only at the third falling edge of the external clock does the 111 pattern match.



Setting Up a Trigger Sequence

To set up the desired trigger sequence, do the following:

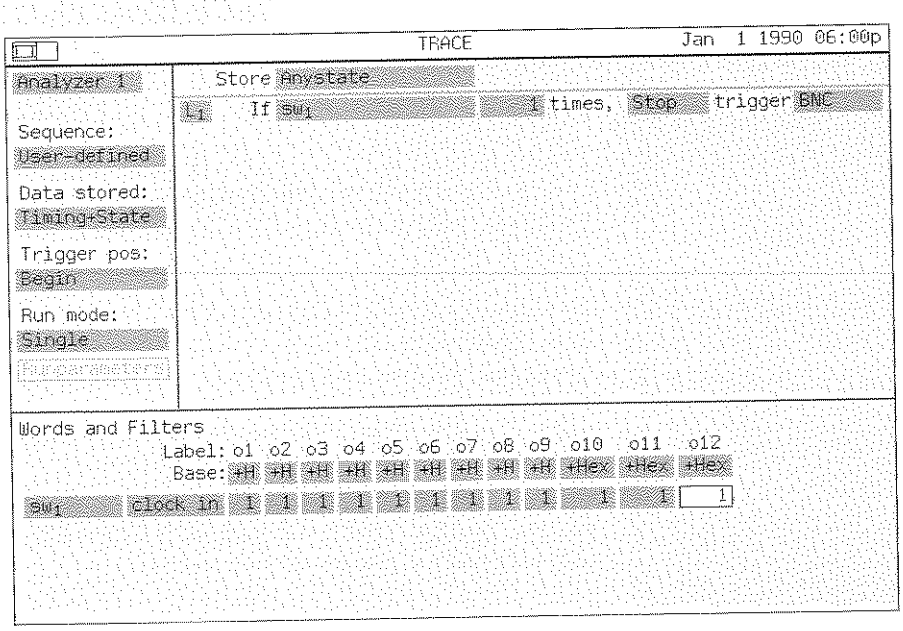
- Press the *TRACE* key to go to the Trace menu.
- Use the arrow keys to go to the trigger sequencer menu field with TimeWord in it:

  L_1 If [TimeWord] 1 times, Stop ...

Press the *S* key to indicate you want a *state word.* A list of state words is shown; press *1* or *SELECT* to select sw_1. You must also tell the instrument which value the trigger word sw_1 should recognize.

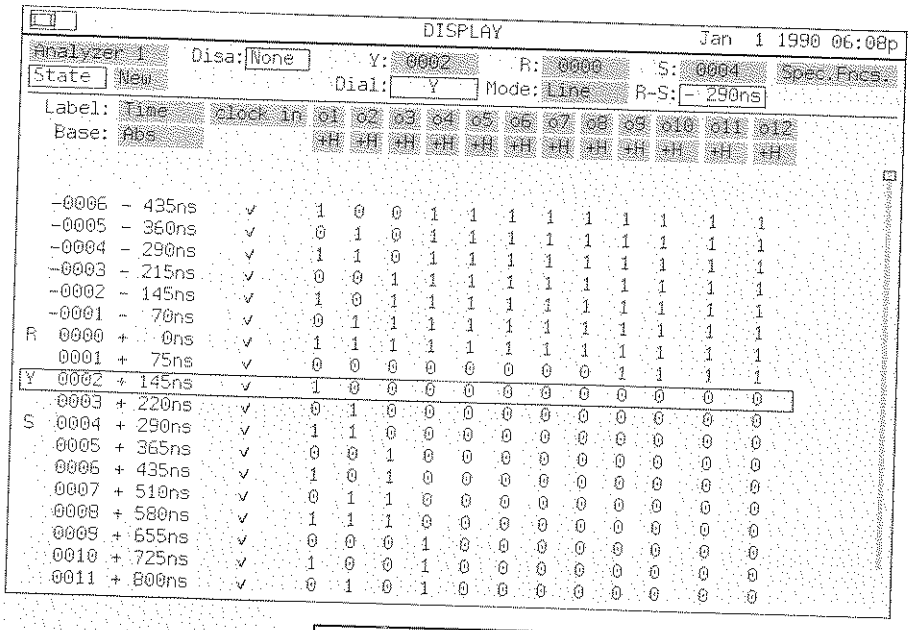
This is done as follows:

- Use the arrow keys to move to the trigger words area at the bottom of the Trace menu.
- Use the arrow keys to go to the definition line of sw_1.
- Type in the value needed in all the fields: 11111...11. The trigger words menu is updated as shown below:



The State Display

Now, press the RUN key. Immediately you see a state listing on the screen, showing the state of each individual output of the counter on each falling edge of the input clock.



DISPLAY — Jan 1 1990 06:08p

Analyzer 1 | Disa: None | Y: 0002 | R: 0000 | S: 0004 | Spec Encs.
State New | Dial: Y | Mode: Line | R-S: -290ns

Label: Time / Base: Abs	clock in	o1 +H	o2 +H	o3 +H	o4 +H	o5 +H	o6 +H	o7 +H	o8 +H	o9 +H	o10 +H	o11 +H	o12 +H
-0006 - 435ns	✓	1	0	0	1	1	1	1	1	1	1	1	1
-0005 - 360ns	✓	0	1	0	1	1	1	1	1	1	1	1	1
-0004 - 290ns	✓	1	1	0	1	1	1	1	1	1	1	1	1
-0003 - 215ns	✓	0	0	1	1	1	1	1	1	1	1	1	1
-0002 - 145ns	✓	1	0	1	1	1	1	1	1	1	1	1	1
-0001 - 70ns	✓	0	1	1	1	1	1	1	1	1	1	1	1
R 0000 + 0ns	✓	1	1	1	1	1	1	1	1	1	1	1	1
0001 + 75ns	✓	0	0	0	0	0	0	0	0	1	1	1	1
Y 0002 + 145ns	✓	1	0	0	0	0	0	0	0	0	0	0	0
0003 + 220ns	✓	0	1	0	0	0	0	0	0	0	0	0	0
S 0004 + 290ns	✓	1	1	0	0	0	0	0	0	0	0	0	0
0005 + 365ns	✓	0	0	1	0	0	0	0	0	0	0	0	0
0006 + 435ns	✓	1	0	1	0	0	0	0	0	0	0	0	0
0007 + 510ns	✓	0	1	1	0	0	0	0	0	0	0	0	0
0008 + 580ns	✓	1	1	1	0	0	0	0	0	0	0	0	0
0009 + 655ns	✓	0	0	0	1	0	0	0	0	0	0	0	0
0010 + 725ns	✓	1	0	0	1	0	0	0	0	0	0	0	0
0011 + 800ns	✓	0	1	0	1	0	0	0	0	0	0	0	0

> This acquisition is also available on disk with the file name STATE.NEW. Refer to "Getting Examples from the Disk" beginning on page 70 for instructions on how to load this file.

The left column (Time) on the State Display menu is always available to the user. It gives very valuable information: The time elapsed between a certain cycle and the trigger point. For example, the value 290ns at the S cursor (line 004) means that, between the trigger event (value 0ns) and cycle 4, there was a delay of 290ns.

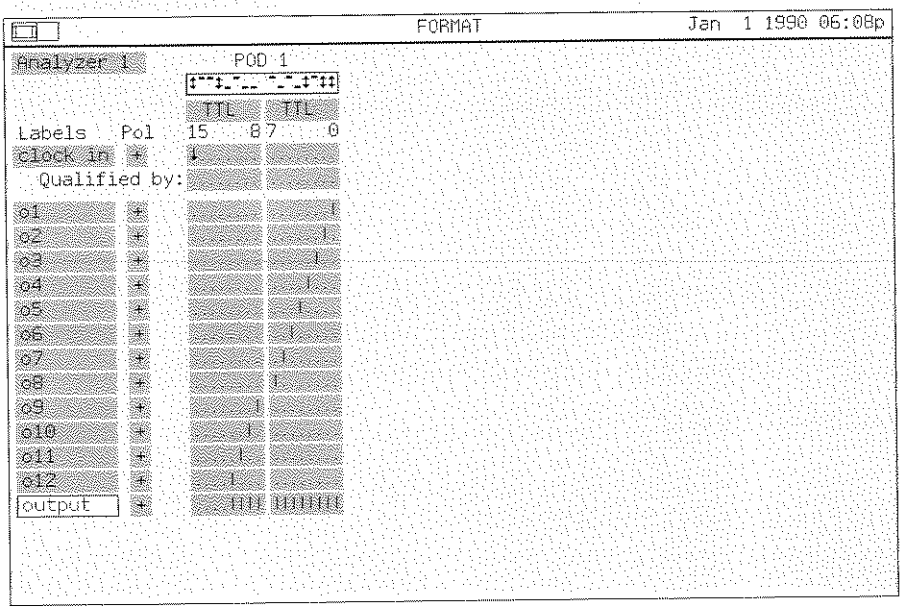
A tick (✓) in the *clock in* column indicates that the data on that line was captured with that clock signal. This is more significant when you have more than one clock.

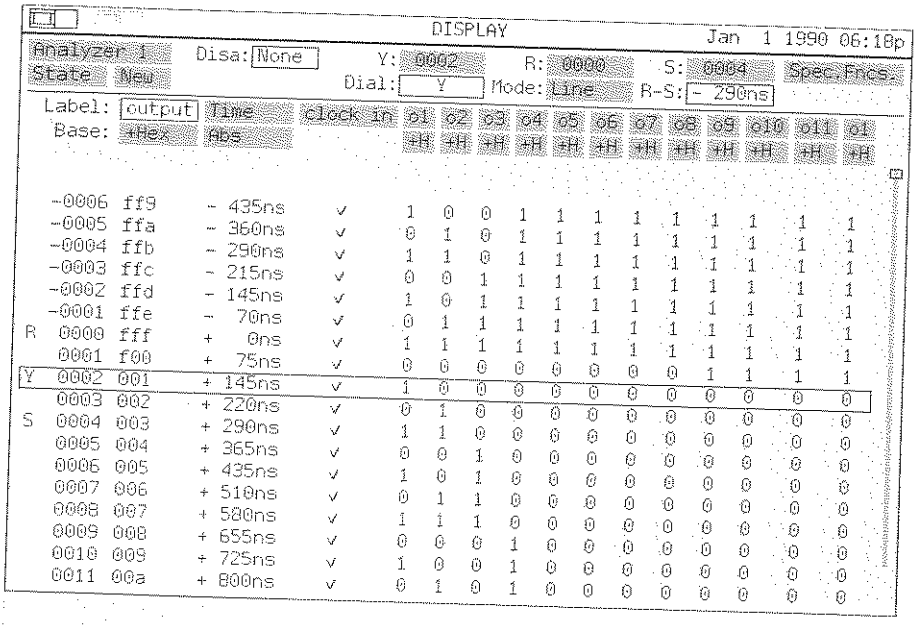
You can scroll through this list by turning the dial (by default allocated to that task – Y for vertical scroll).

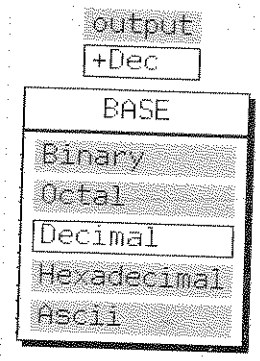
Grouping Signals Under a Label

It would, however, be more useful to have all the outputs re-grouped under a single label name and their combined value shown in hexadecimal format. To do that, you must define a new label in the Format menu:

- Press the *FORMAT* key, and the Format menu appears.
- Use the arrow keys or dial to go to the last label (*o12*).
- Press the *INSERT* key, and a new label appears under the last one. This will have the default name ***a***.
- Type in the new name of this label, for example *output*.
- In the allocation field, use the *SELECT* key to assign your 12 outputs to the label.





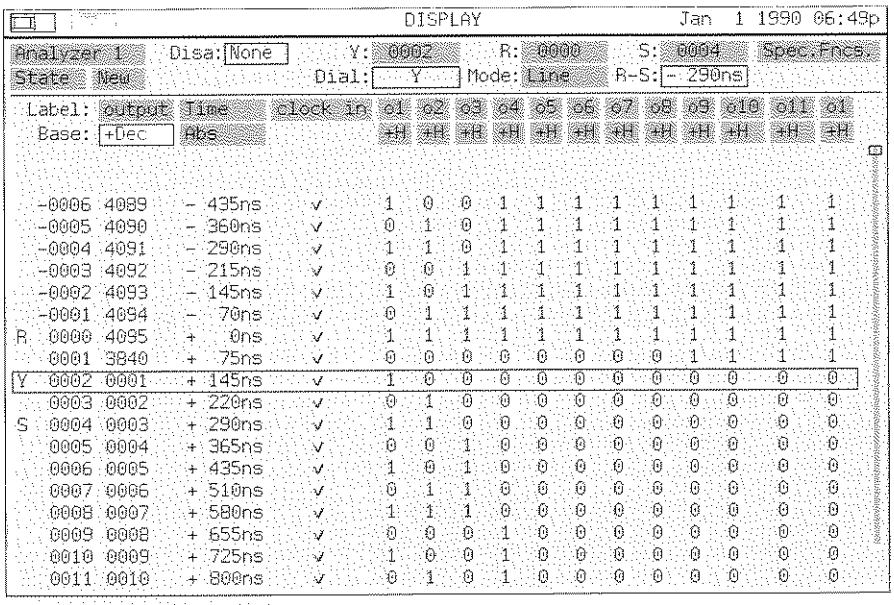


Return to the Display menu. You now see the hexadecimal values of the combined signals in the column under *output*.

To change the radix (the *Base)* in which the *output* column is displayed, do the following:

- Use the arrow keys to move to the field under the *output* label on the line called *Base*.
- Press the *SELECT* key to get a list with the different bases available.
- Use the arrow keys or dial to choose one of them. In the example at the left, Decimal was chosen.
- Then press the *SELECT* key to close the list and change the base. The result is shown in the figure on the next page.

Now switch back to the hexadecimal base and inspect the state list in detail. You can see that one of the numbers is not correct in the list: F00 appears instead of 000 on line 0001.



DISPLAY Jan 1 1990 06:49p

Analyzer 1 Disa: None Y: 0002 R: 0000 S: 0004 Spec.Fncs.
State New Dial: Y Mode: Line R-S: - 290ns

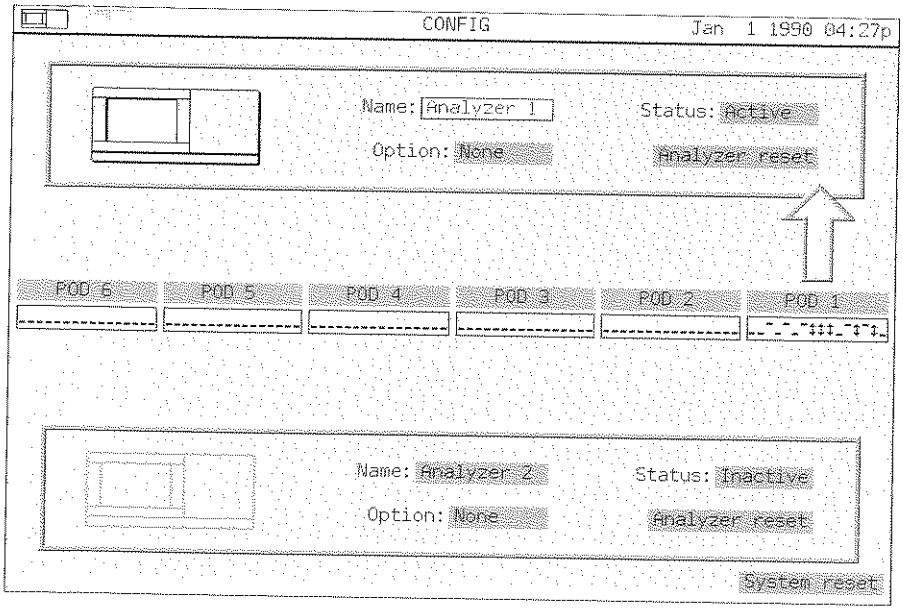
	Label:	output	Time	clock in	o1	o2	o3	o4	o5	o6	o7	o8	o9	o10	o11	o1
	Base:	+Dec	Abs		+H	+H	+H	+H	+H	+H	+H	+H	+H	+H	+H	+H
	-0006	4089	- 435ns	v	1	0	0	1	1	1	1	1	1	1	1	1
	-0005	4090	- 360ns	v	0	1	0	1	1	1	1	1	1	1	1	1
	-0004	4091	- 290ns	v	1	1	0	1	1	1	1	1	1	1	1	1
	-0003	4092	- 215ns	v	0	0	1	1	1	1	1	1	1	1	1	1
	-0002	4093	- 145ns	v	1	0	1	1	1	1	1	1	1	1	1	1
	-0001	4094	- 70ns	v	0	1	1	1	1	1	1	1	1	1	1	1
R	0000	4095	+ 0ns	v	1	1	1	1	1	1	1	1	1	1	1	1
	0001	3840	+ 75ns	v	0	0	0	0	0	0	0	0	1	1	1	1
Y	0002	0001	+ 145ns	v	1	0	0	0	0	0	0	0	0	0	0	0
	0003	0002	+ 220ns	v	0	1	0	0	0	0	0	0	0	0	0	0
S	0004	0003	+ 290ns	v	1	1	0	0	0	0	0	0	0	0	0	0
	0005	0004	+ 365ns	v	0	0	1	0	0	0	0	0	0	0	0	0
	0006	0005	+ 435ns	v	1	0	1	0	0	0	0	0	0	0	0	0
	0007	0006	+ 510ns	v	0	1	1	0	0	0	0	0	0	0	0	0
	0008	0007	+ 580ns	v	1	1	1	0	0	0	0	0	0	0	0	0
	0009	0008	+ 655ns	v	0	0	0	1	0	0	0	0	0	0	0	0
	0010	0009	+ 725ns	v	1	0	0	1	0	0	0	0	0	0	0	0
	0011	0010	+ 800ns	v	0	1	0	1	0	0	0	0	0	0	0	0

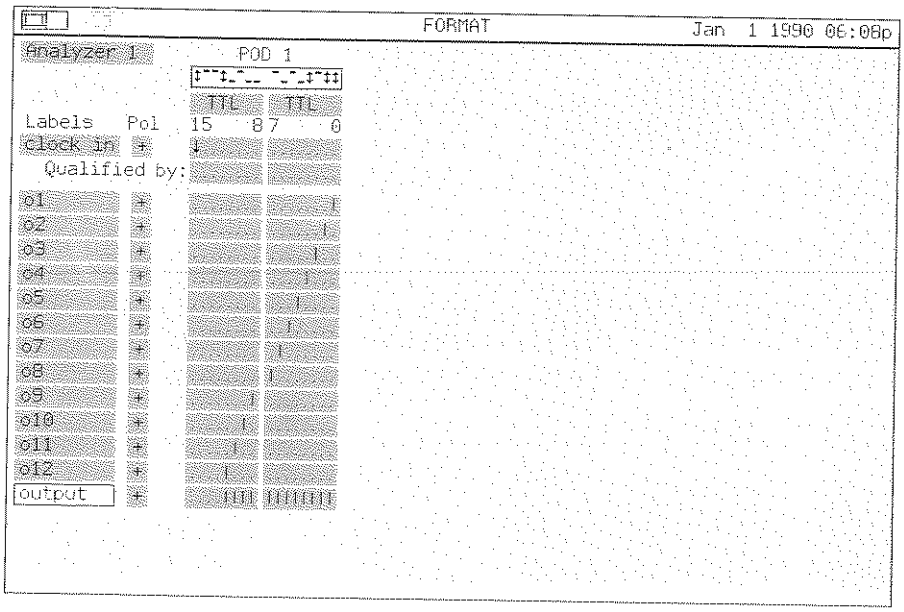
So, it seems that the target system has some problems. You know when it occurs, but you do not know why. In the next chapter, you will use another mode of these logic analyzers to find out why.

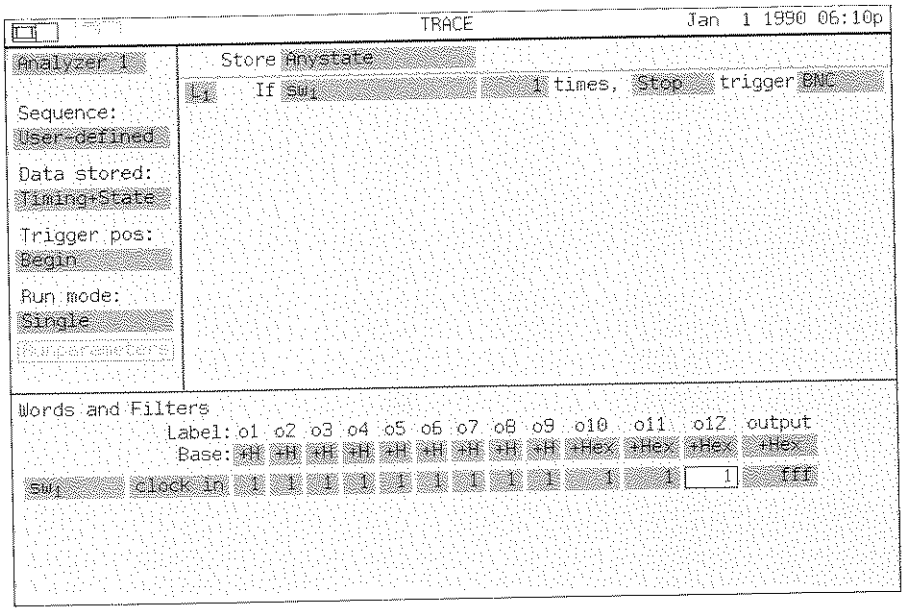
Summary

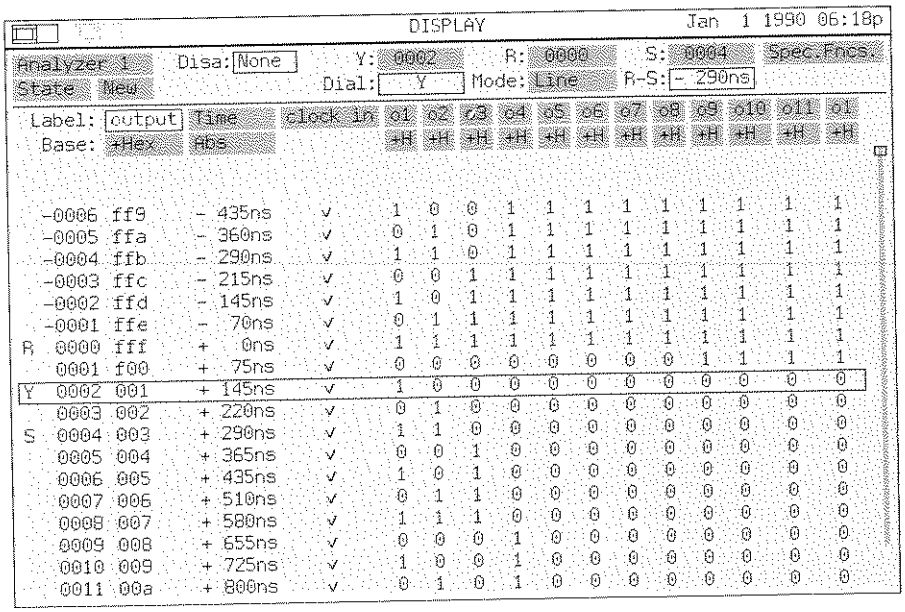
With state acquisition, you tested, fast and efficiently, the functionality of the clock-driven target. On the next two pages, you can see the four menus that you used to do this. On the other hand, you did not see most of the time-related information you got with timing acquisition which could be useful in understanding the "why" of a problem.

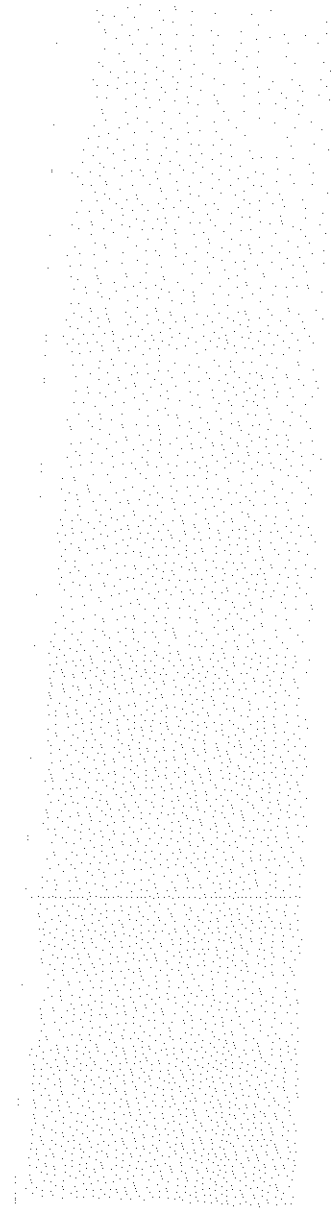
In the next chapter, you will combine these two modes to see both views of the target together.


CONFIG
Jan 1 1990 04:27p
Name: Analyzer 1
Status: Active
Option: None
Analyzer reset
POD 6
POD 5
POD 4
POD 3
POD 2
POD 1
Name: Analyzer 2
Status: Inactive
Option: None
Analyzer reset
System reset

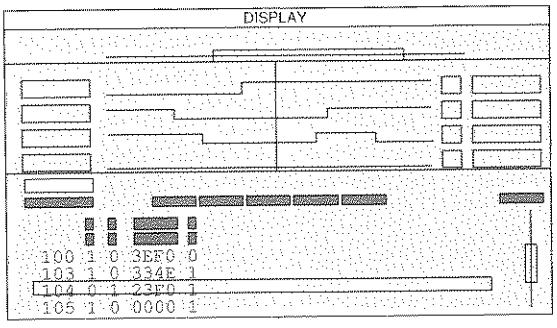

FORMAT
Jan 1 1990 06:08p
Analyzer 1
POD 1
TTL TTL
Labels Pol 15 8 7 0
Clock in +
Qualified by:
o1 +
o2 +
o3 +
o4 +
o5 +
o6 +
o7 +
o8 +
o9 +
o10 +
o11 +
o12 +
Output +


TRACE
Jan 1 1990 06:10p
Analyzer 1
Store Anystate
Sequence:
User-defined
Data stored:
Timing+State
Trigger pos:
Begin
Run mode:
Single
L1 If sw1 1 times, Stop trigger BNC
Words and Filters
Label: o1 o2 o3 o4 o5 o6 o7 o8 o9 o10 o11 o12 output
Base: +H +H +H +H +H +H +H +H +H +Hex +Hex +Hex +Hex
sw1 clock in 1 1 1 1 1 1 1 1 1 1 1 1 fff


DISPLAY
Jan 1 1990 06:18p
Analyzer 1
Disa: None
Y: 0002
R: 0000
S: 0004
Spec.Fncs.
State Mem
Dial: Y
Mode: Line
R-S: - 290ns
Label: output Time clock in o1 o2 o3 o4 o5 o6 o7 o8 o9 o10 o11 o1
Base: +Hex Abs +H +H +H +H +H +H +H +H +H +H +H +H
-0006 ff9 - 435ns v 1 0 0 1 1 1 1 1 1 1 1 1
-0005 ffa - 360ns v 0 1 0 1 1 1 1 1 1 1 1 1
-0004 ffb - 290ns v 1 1 0 1 1 1 1 1 1 1 1 1
-0003 ffc - 215ns v 0 0 1 1 1 1 1 1 1 1 1 1
-0002 ffd - 145ns v 1 0 1 1 1 1 1 1 1 1 1 1
-0001 ffe - 70ns v 0 1 1 1 1 1 1 1 1 1 1 1
R 0000 fff + 0ns v 1 1 1 1 1 1 1 1 1 1 1 1
0001 f00 + 75ns v 0 0 0 0 0 0 0 0 1 1 1 1
Y 0002 001 + 145ns v 1 0 0 0 0 0 0 0 0 0 0 0
0003 002 + 220ns v 0 1 0 0 0 0 0 0 0 0 0 0
S 0004 003 + 290ns v 1 1 0 0 0 0 0 0 0 0 0 0
0005 004 + 365ns v 0 0 1 0 0 0 0 0 0 0 0 0
0006 005 + 435ns v 1 0 1 0 0 0 0 0 0 0 0 0
0007 006 + 510ns v 0 1 1 0 0 0 0 0 0 0 0 0
0008 007 + 580ns v 1 1 1 0 0 0 0 0 0 0 0 0
0009 008 + 655ns v 0 0 0 1 0 0 0 0 0 0 0 0
0010 009 + 725ns v 1 0 0 1 0 0 0 0 0 0 0 0
0011 00a + 800ns v 0 1 0 1 0 0 0 0 0 0 0 0



Dual Analysis Per Pin (DAPP) Mode



Why DAPP?

In the previous two chapters, you have seen that timing and state acquisition are complementary to exploring and testing a digital system.

The Dual Analysis Per Pin (DAPP) mode, an exclusive feature of the PHILIPS PM 3580 logic analyzer family, allows you to get timing and state acquisition on all channels *at the same time* with a common triggering sequence.

> If you want to know more about timing and/or state analysis, please refer to *The ABC's of logic analysis.*

Back to the Practical Problem

With state acquisition, you checked the binary counter. You saw that in some cases, the counter did not count properly.

If there had been a timing measurement at the same time as the state acquisition, you would have been able to see immediately what really happened on each channel for each unexpected state in the state list.

Connection

Since all the signals were previously connected for timing and state acquisition, you do not need to do anything special, because the *DAPP* architecture allows you to make a state and a timing acquisition simultaneously on all pins.

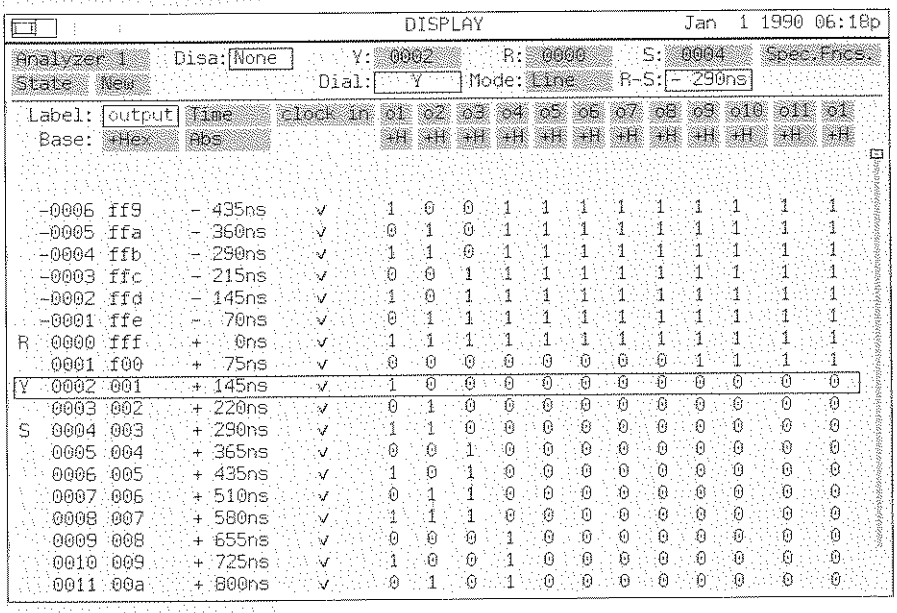
Setup of the Logic Analyzer

The default acquisition mode for PM 3580/PM 3585 logic analyzers is *DAPP* mode, as soon as an external clock is defined. So, the last acquisition you did was already made in *DAPP* mode even though you perhaps did not realize it.

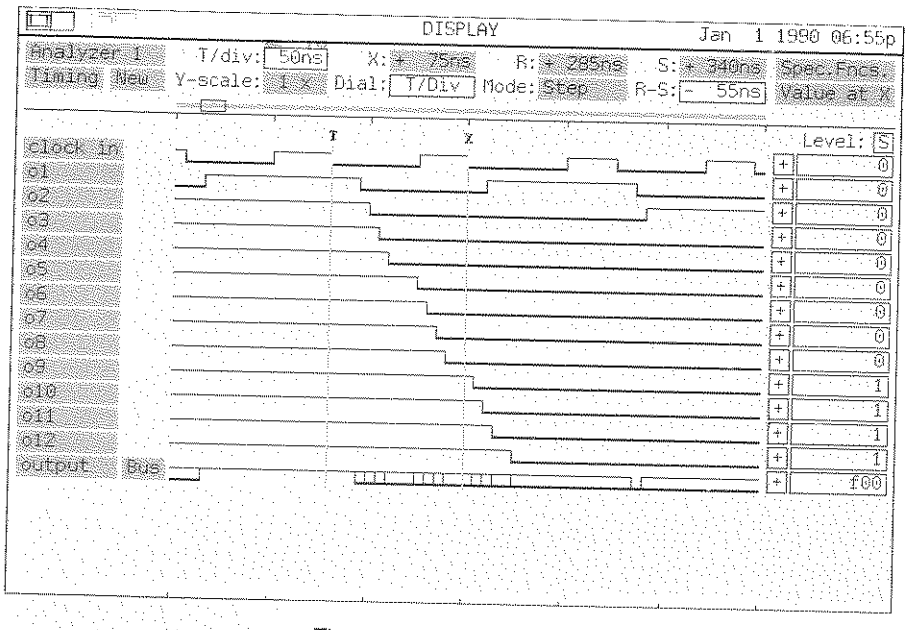
So, how do you display all this information? Since the information you need is already in acquisition memory, you

do not need to do a new acquisition or read in a file, you simply do the following:

- Press the *DISPLAY* key to return to the state list of the last acquisition (STATE.NEW).



- Scroll the data with the dial until the first bad state (F00 instead of 000) is in the center of the screen in the small rectangle.
- Go to the field called "State" (beginning of the 2nd line).
- Press the *SELECT* key to switch from the state to the timing acquisition display.
- Go to the X field and press *Y* to set the X cursor position to be the same as the Y cursor position of the state list.



The center of the display window (X) is now aligned with the time when the state in the rectangle was sampled. It is now clear that, because of too long a propagation delay in the flip-flops of the counter, the value "000" is never stable on the falling edge of the clock.

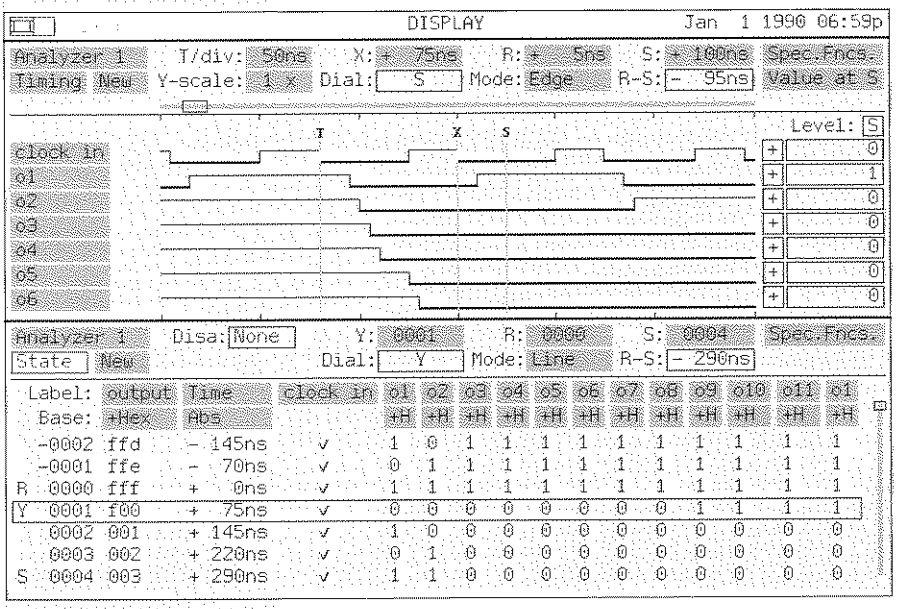
Determining the Maximum Clock Speed

You can go further and try to determine the maximum clock speed that you can use on the target to avoid a false count.

To do this, just do the same kind of measurement you did in "Timing Acquisition", "Measuring a Time Difference" on page 29, to measure the time delay between the falling edge of the clock and the toggling of the last signal (o12).

The toggling time is 95 ns. To always have proper counting, the count information should be stable before the next falling edge of the clock; therefore, the clock period should be greater than the time you measured. Thus the clock frequency should be less than10 MHz.

You can also see both the timing and state displays on the same screen, as shown below. All you do is press the *HOME* key to ensure you are on the Analyzer field, then press *INSERT*. This displays the two windows as shown below.



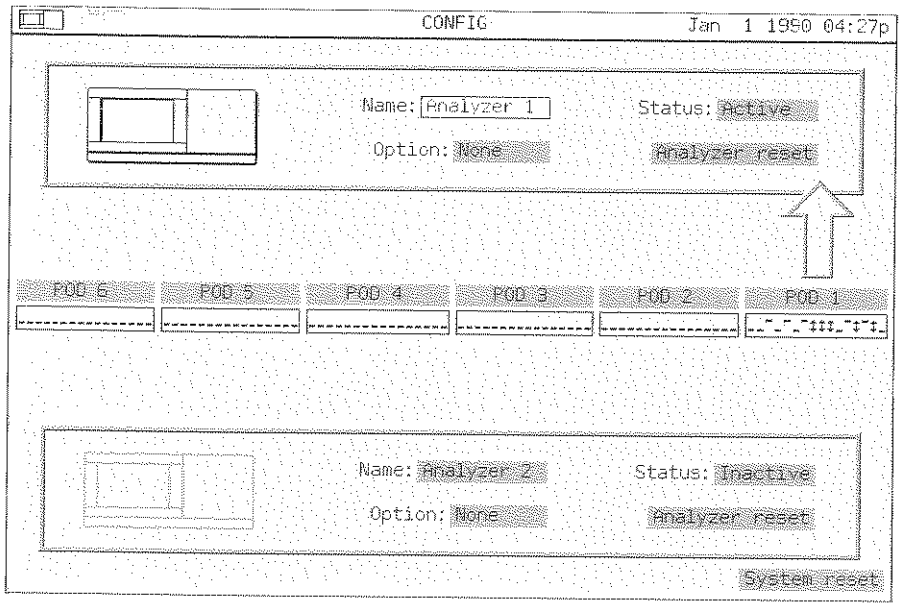
To switch between the two windows, use the *DISPLAY* key. To remove one of the windows, press the *DELETE* key on the Analyzer field of the window you wish to delete.

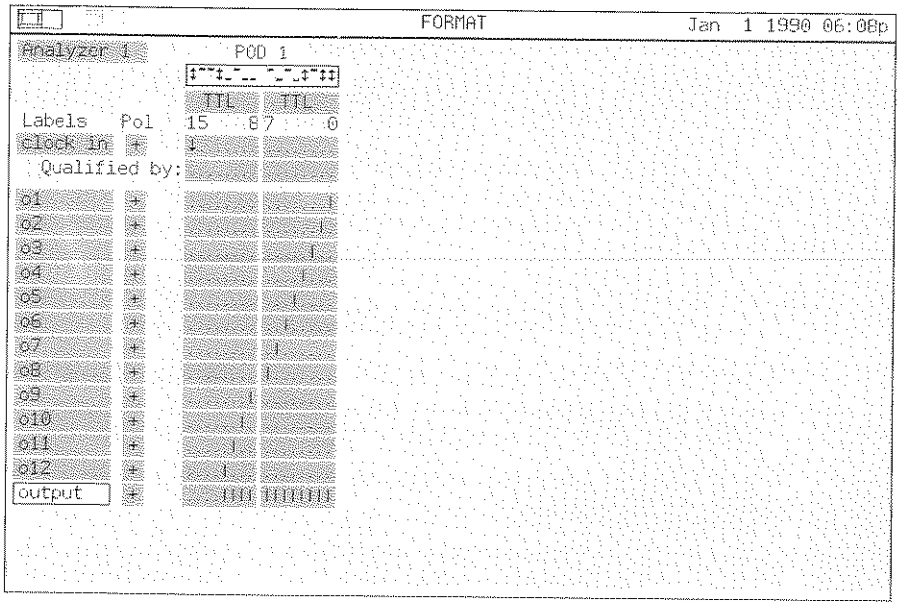
Summary

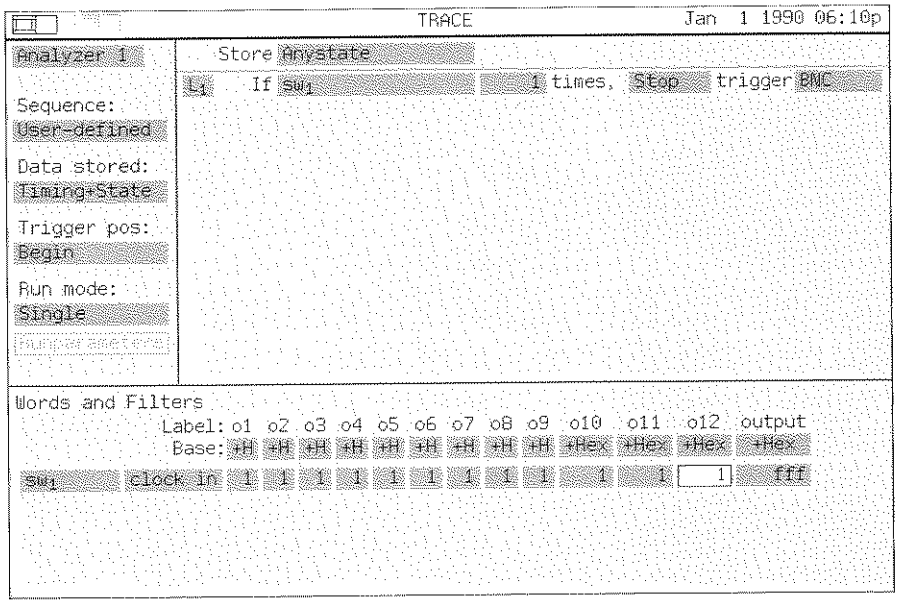
The last three chapters illustrated the respective advantages of state and timing acquisition, and the benefits of the exclusive Dual Analysis Per Pin (DAPP) mode.

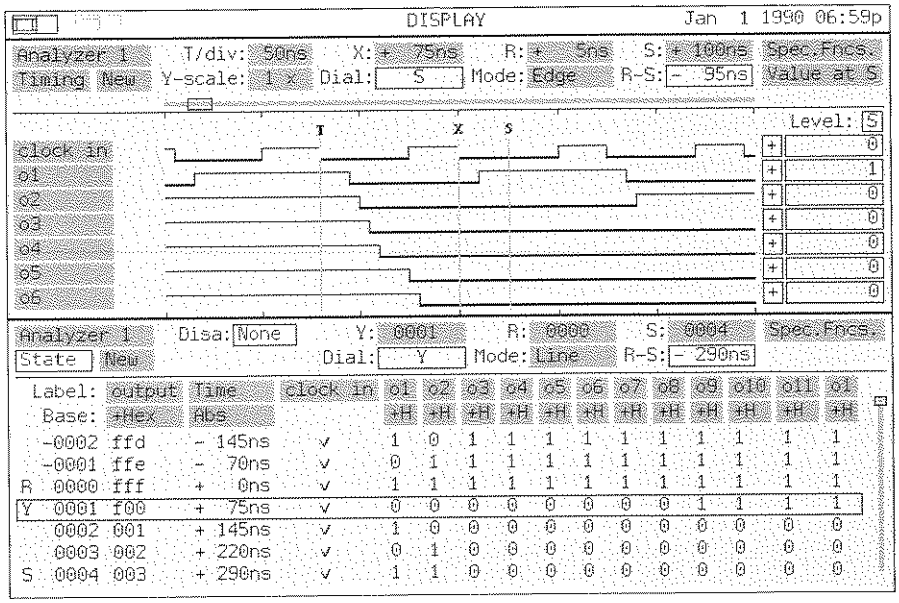
On the next two pages, the four contributing menus are shown.

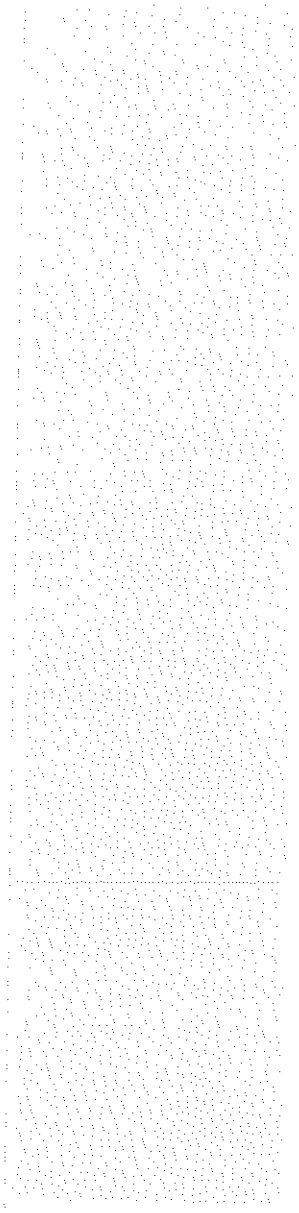
The next chapter introduces another feature of this range of logic analyzers: microprocessor disassembly support.


CONFIG
Jan 1 1990 04:27p
Name: Analyzer 1
Status: Active
Option: None
Analyzer reset
POD 6
POD 5
POD 4
POD 3
POD 2
POD 1
Name: Analyzer 2
Status: Inactive
Option: None
Analyzer reset
System reset

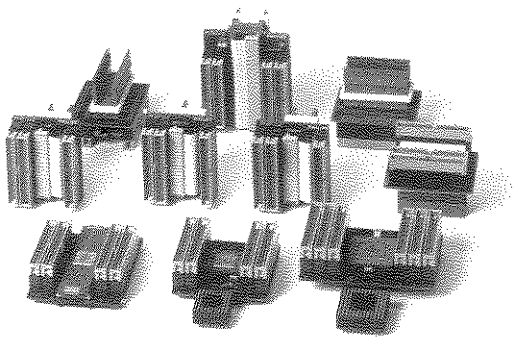

FORMAT
Jan 1 1990 06:08p
Analyzer 1
POD 1
TTL
TTL
Labels
Pol
15
8 7
0
Clock in
Qualified by:
o1
o2
o3
o4
o5
o6
o7
o8
o9
o10
o11
o12
output


TRACE
Jan 1 1990 06:10p
Analyzer 1
Store Anystate
L1 If sw1 1 times, Stop trigger BMC
Sequence:
User-defined
Data stored:
Timing+State
Trigger pos:
Begin
Run mode:
Single
Words and Filters
Label: o1 o2 o3 o4 o5 o6 o7 o8 o9 o10 o11 o12 output
Base: +H +H +H +H +H +H +H +H +H +Hex +Hex +Hex +Hex
sw1 clock in 1 1 1 1 1 1 1 1 1 1 1 1 fff


DISPLAY
Jan 1 1990 06:59p
Analyzer 1 Timing New
T/div: 50ns X: + 75ns R: + 5ns S: + 100ns Spec.Fncs.
Y-scale: 1 X Dial: S Mode: Edge R-S: - 95ns Value at S
Level: S
clock in
o1
o2
o3
o4
o5
o6
Analyzer 1 State New
Disa: None Y: 0001 R: 0000 S: 0004 Spec.Fncs.
Dial: Y Mode: Line R-S: - 290ns
Label: output Time clock in o1 o2 o3 o4 o5 o6 o7 o8 o9 o10 o11 o1
Base: +Hex Abs +H +H +H +H +H +H +H +H +H +H +H +H
-0002 ffd - 145ns v 1 0 1 1 1 1 1 1 1 1 1 1
-0001 ffe - 70ns v 0 1 1 1 1 1 1 1 1 1 1 1
R 0000 fff + 0ns v 1 1 1 1 1 1 1 1 1 1 1 1
Y 0001 f00 + 75ns v 0 0 0 0 0 0 0 0 1 1 1 1
0002 001 + 145ns v 1 0 0 0 0 0 0 0 0 0 0 0
0003 002 + 220ns v 0 1 0 0 0 0 0 0 0 0 0 0
S 0004 003 + 290ns v 1 1 0 0 0 0 0 0 0 0 0 0



Use of Microprocessor Disassembly



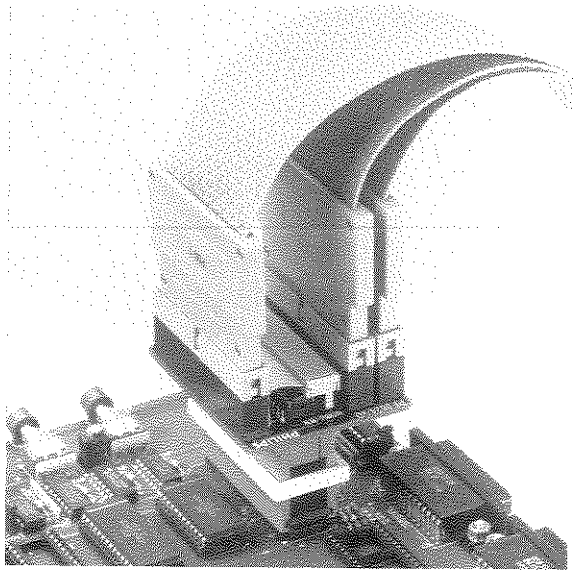
What is a Disassembler?

When you work on simple clock-driven digital circuitry, timing and state information usually are enough to let you understand what is going on.

However, if you are testing a microprocessor-controlled board, the task is more difficult.

In addition to observing the signals, you must also understand and trace the program executed by the microprocessor. This means you must translate the state data into a more understandable form, especially for the software engineer. Disassembly software is a tool that can be used to translate the numbers of the state display back into a list of assembly instructions.

A microprocessor adapter is supplied with each disassembler. This simplifies the connection between the logic analyzer and the microprocessor under test.

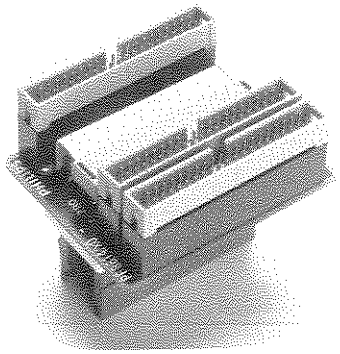


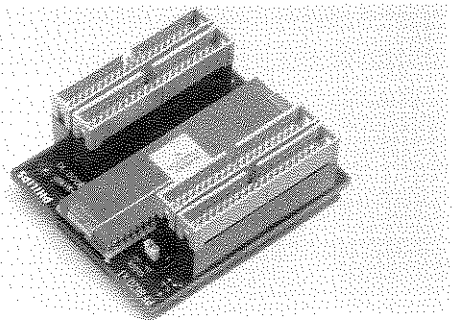
The Problem

The problem is to check which part of a program is executed after a board with a 16-bit microprocessor is reset. To do this, you will use the logic analyzer with the disassembly support dedicated to that microprocessor (in the example, a MC 68000 from MOTOROLA).

Connection

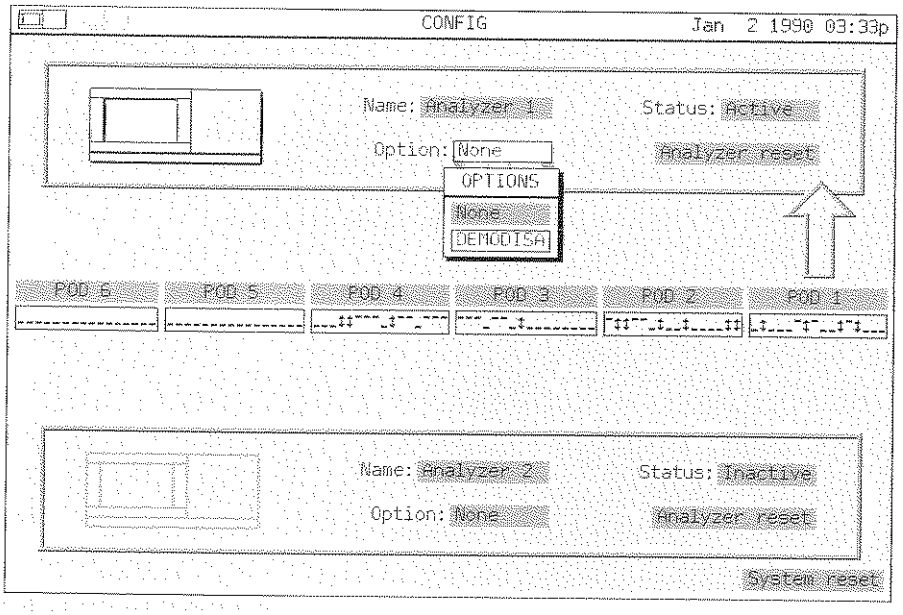
The microprocessor adapter makes it easy to connect to a microprocessor. Simply clip the clip version of the microprocessor adapter onto the chip or insert the socket version between the chip and its socket.





Setup of the Logic Analyzer

The setup of the analyzer with a disassembler is even easier than for normal logic observation. The Configuration menu has a field called "Option". Its default value is "None". By pressing *SELECT*, you get a list of all the disassemblers available on the floppy disk.

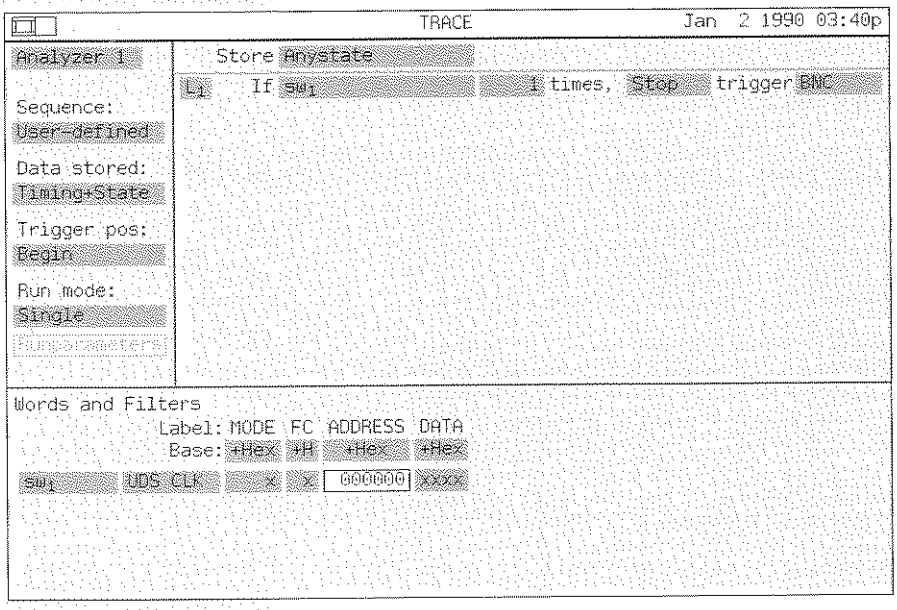


To choose the right disassembler (here called *Demodisa*): highlight the selection with the arrow keys or dial. Press *SELECT*. The disassembler translation software and the associated setup are then loaded.

Note: a confirmation popup may appear asking you if you want to remove settings (i.e., previous settings) if you have not just switched on. Answer yes.

Now all you must define are the events that will stop the acquisition. To do this:

- Press the *TRACE* key to go to the Trace menu.
- In the sequencer area, change the trigger sequence from TimeWord to sw_1, as in "State Acquisition", "Setting Up a Trigger Sequence" on page 40.



The first thing that the 68000 microprocessor does after reset is read addresses 0H, 2H, 4H, and 6H to initialize some registers.

To capture data starting after Reset, you just specify address 0H as the trigger event. As the default trigger sequence is appropriate (*if sw_1 1 times, stop*), you just have to fill in sw_1 with the correct pattern, i.e., 000000 Hex for the address.

Once the trace is specified, press *RUN* and wait for an occurrence of RESET.

While the analyzer is waiting for the trigger condition, a special window displays the present status of the sequencer, and the box at the top left of the screen flashes.

```
ACQUISITION PROGRESS
Analyzer 1:  1

Analyzer 2:  Inactive
```

As soon as the trigger sequence is met, the window disappears and the state display menu is shown.

```
                                DISPLAY                          Jan  2 1990 05:53p
Analyzer 1   Disa: On      Y: 0009    R: 0008    S: 0014    Spec.Facs.
State  New   Parameters  Dial:  Y   Mode: Line   R-S: -10.3us
  Label: MODE FC ADDRESS DATA 68000 Instructions                    Time   L
   Base: +Hex +H +Hex   +Hex                                        Abs
 -0001 4    6  000460  534e SUBQ.W   #+1,A6                        - 272ms
  0000 4    6  000000  0000                RESET                   +   0ns
  0001 4    6  000002  b000                                        +1560ns
  0002 4    6  000004  0000                                        +3125ns
  0003 4    6  000006  0700                                        +4690ns
  0004 4    6  000700  4eb8 JSR      0430.W                       +6255ns
  0005 4    6  000702  0430                                        +8445ns
  0007 0    5  00affc  0000           mw                          +12.5us
R 0008 0    5  00affe  0704           mw                          +14.4us
  0006 4    6  000430  327c MOVEA.W  #4007,A1                     +10.6us
Y 0009 4    6  000432  4007                                        +16.0us
  0010 4    6  000434  123c MOVE.B   #98,D1                       +17.5us
  0011 4    6  000436  0098                                        +19.1us
  0012 4    6  000438  1281 MOVE.B   D1,(A1)                      +20.6us
S 0014 Z    5  004006    98           mw                          +24.1us
  0013 4    6  00043a  4e75 RTS                                   +22.2us
  0016 4    5  00affc  0000           mr                          +27.2us
  0017 4    5  00affe  0704           mr                          +28.8us
  0018 4    6  000704  347c MOVEA.W  #4001,A2                     +30.3us
  0019 4    6  000706  4001                                        +31.9us
```

This acquisition is also available on disk with the file name DEMODISA.NEW. Refer to "Getting Examples from the Disk" beginning on page 70 for instructions on how to load this file.

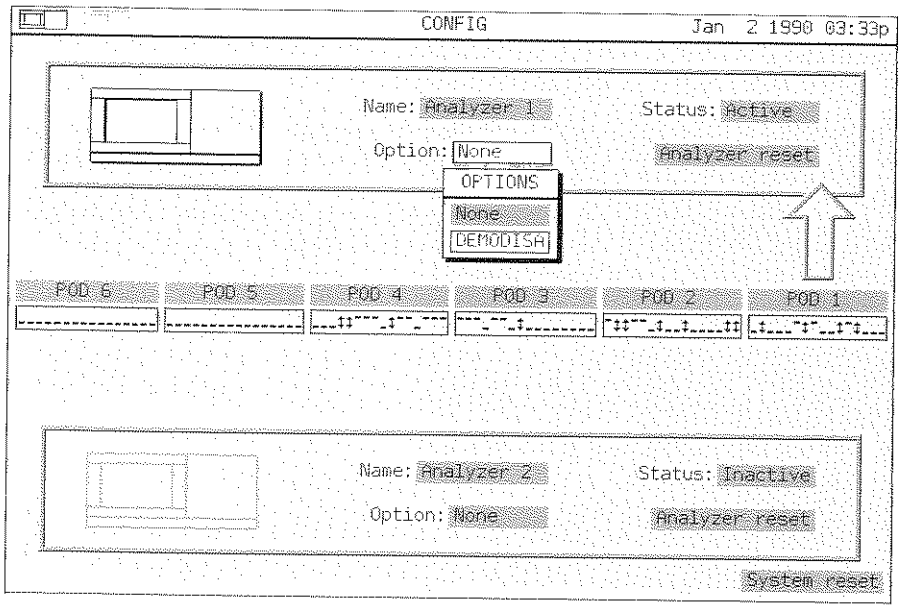
This list is formatted according to the setup loaded with the disassembly software.

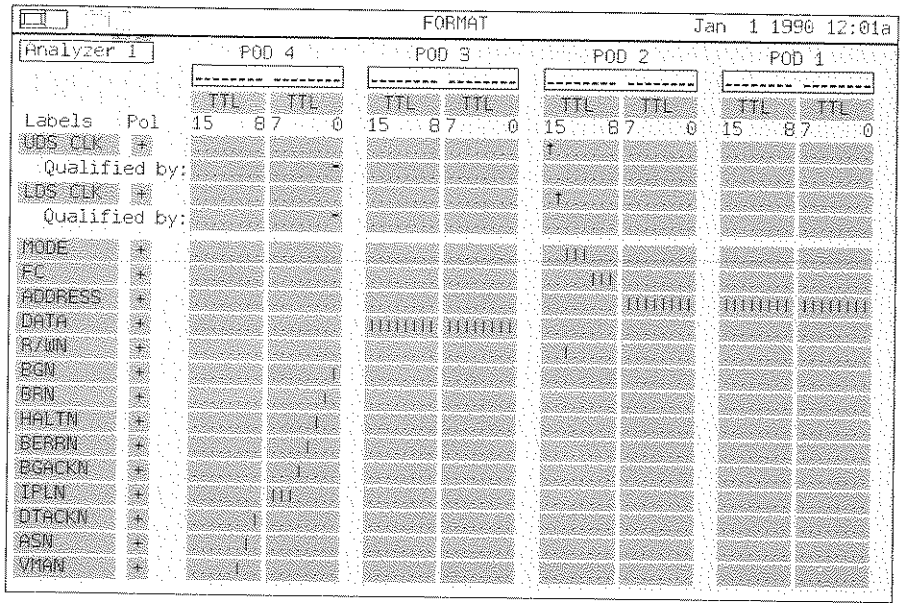
With this disassembled list, you can do exactly the same you did before: scrolling, setting cursors, etc..

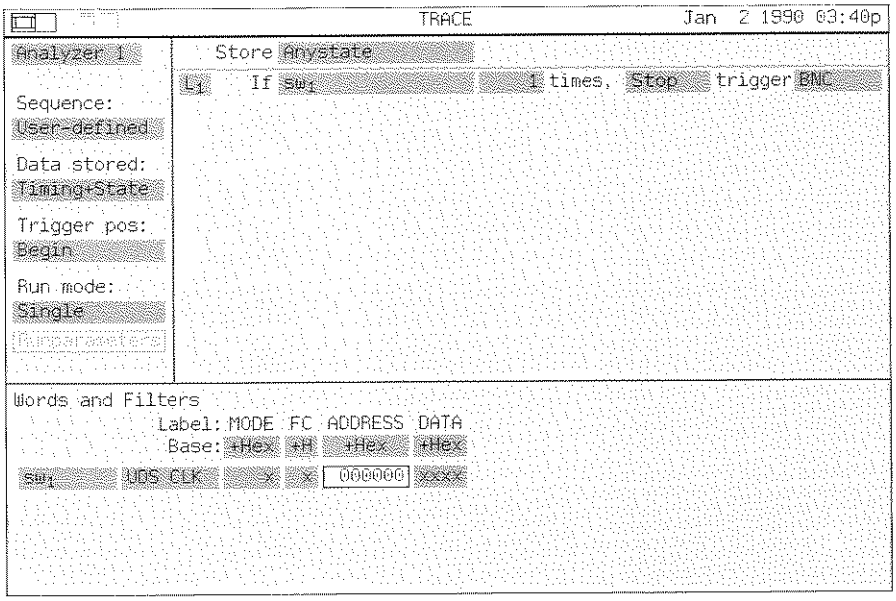
Summary

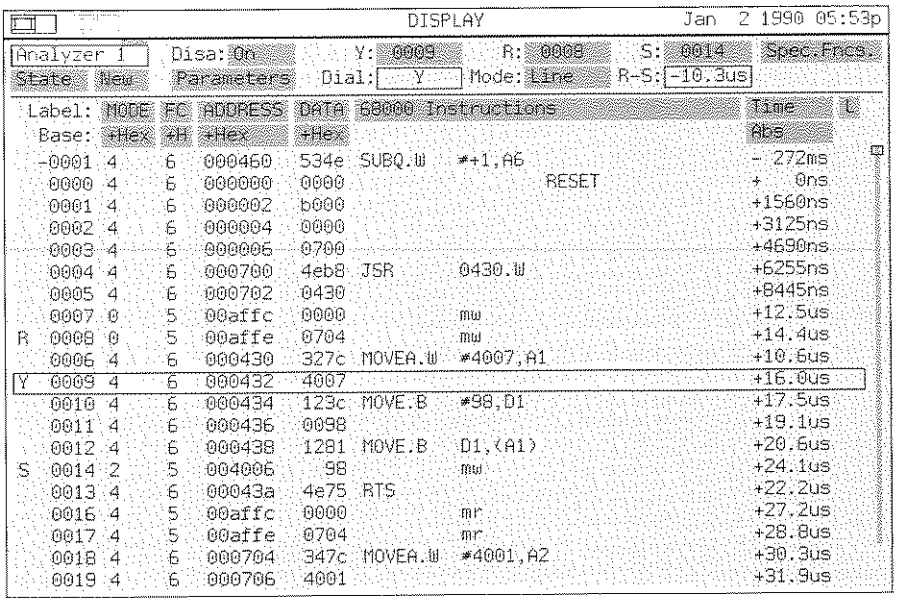
State acquisition with disassembly software allows you to follow the program executed on a microprocessor easily. The microprocessor adapters associated with this software give you simple access to the signals of the microprocessor for disassembly or state-only purposes or for timing applications.

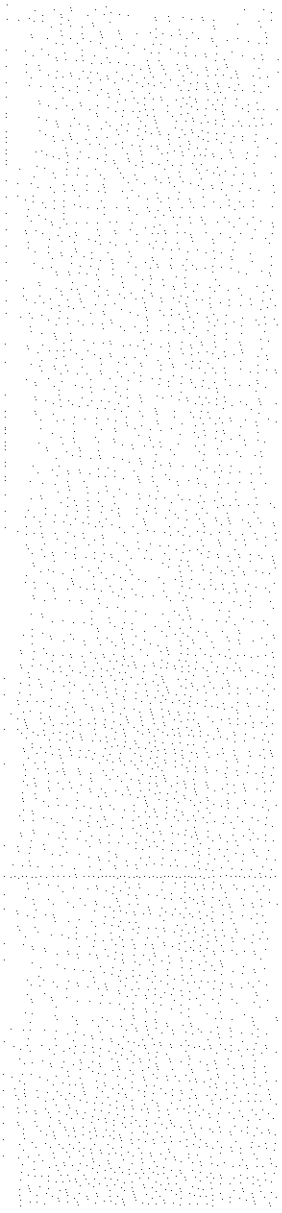
The next two pages show the four menus used.


CONFIG
Jan 2 1990 03:33p
Name: Analyzer 1
Status: Active
Option: None
Analyzer reset
OPTIONS
None
DEMODISA
POD 6
POD 5
POD 4
POD 3
POD 2
POD 1
Name: Analyzer 2
Status: Inactive
Option: None
Analyzer reset
System reset


FORMAT
Jan 1 1990 12:01a
Analyzer 1
POD 4
POD 3
POD 2
POD 1
TTL
Labels Pol 15 8 7 0
UDS CLK
Qualified by:
LDS CLK
Qualified by:
MODE
FC
ADDRESS
DATA
R/WN
BGN
BRN
HALTN
BERRN
BGACKN
IPLN
DTACKN
ASN
VMAN


TRACE
Jan 2 1990 03:40p
Analyzer 1
Store Anystate
L1 If sw1 1 times, Stop trigger BNC
Sequence:
User-defined
Data stored:
Timing+State
Trigger pos:
Begin
Run mode:
Single
Parameters
Words and Filters
Label: MODE FC ADDRESS DATA
Base: +Hex +H +Hex +Hex
sw1 UDS CLK X X 000000 xxxx


DISPLAY
Jan 2 1990 05:53p
Analyzer 1
Disa: On
Y: 0009
R: 0008
S: 0014
Spec.Fncs.
State New
Parameters
Dial: Y
Mode: Line
R-S: -10.3us
Label: MODE FC ADDRESS DATA 68000 Instructions Time L
Base: +Hex +H +Hex +Hex Abs
-0001 4 6 000460 534e SUBQ.W #+1,A6 - 272ms
0000 4 6 000000 0000 RESET + 0ns
0001 4 6 000002 b000 +1560ns
0002 4 6 000004 0000 +3125ns
0003 4 6 000006 0700 +4690ns
0004 4 6 000700 4eb8 JSR 0430.W +6255ns
0005 4 6 000702 0430 +8445ns
0007 0 5 00affc 0000 mw +12.5us
R 0008 0 5 00affe 0704 mw +14.4us
0006 4 6 000430 327c MOVEA.W #4007,A1 +10.6us
Y 0009 4 6 000432 4007 +16.0us
0010 4 6 000434 123c MOVE.B #98,D1 +17.5us
0011 4 6 000436 0098 +19.1us
0012 4 6 000438 1281 MOVE.B D1,(A1) +20.6us
S 0014 2 5 004006 98 mw +24.1us
0013 4 6 00043a 4e75 RTS +22.2us
0016 4 5 00affc 0000 mr +27.2us
0017 4 5 00affe 0704 mr +28.8us
0018 4 6 000704 347c MOVEA.W #4001,A2 +30.3us
0019 4 6 000706 4001 +31.9us



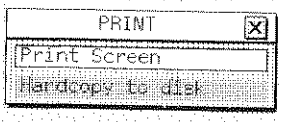
Making a Printout

Using the printout facility of your logic analyzer allows you to make hardcopies of settings and acquisition results.

To use it, you need an "Epson-compatible" printer (most printers used with PCs are of this type) with a Centronics interface, and the cable used to link a PC to this printer (male 25-pin 'D' to Amphenol 36-pin male).

Then link your printer to the logic analyzer via the cable using the 25-pin 'D' Centronics connector on the rear panel of your logic analyzer. The installation is now complete.

To start a printout at any point, just press the *PRINT* key.



The print pop-up menu appears as at left. Just press *SELECT* to start the print-out. To stop the print at any time, press any key. A confirmation pop-up appears so that you can continue or abort the print-out.

To cancel the print from the print popup menu, select the box with the 'x' (the cancel field).

The other option on the print menu ("Hardcopy to disk") allows you to 'print' to a floppy disk in a special format. See the *User Manual* for more information.

Getting Examples from the Disk

All the acquisitions on which the examples in this manual are based are stored on the system floppy disk delivered with the instrument.

If you want to practice "real life" examples you simply recall them from disk.

TIMING.NEW is the timing example from "Timing Acquisition" beginning on page 21,

TIMETRIG.NEWis the second timing example from "Timing Acquisition" beginning on page 21.

STATE.NEW is the state example from "State Acquisition" beginning on page 36 and "Dual Analysis Per Pin (DAPP) Mode" beginning on page 50.

DEMODISA.NEW
is the disassembly state example from "Use of Microprocessor Disassembly" beginning on page 58. It is associated with the disassembly software file DEMODISA.DIS.

To recall one of these examples, do the following:

- press the I/O key to see the screen at the top of the next page.

The I/O screen is divided in two parts:

1. The top of the menu where you can invoke all disk utilities (load, store, rename, copy, delete, format).
2. The large lower area showing the list of all the files available on disk.

- Use the arrow keys to move to the field "Load".
- Press the *SELECT* key, and a file selection window appears.
- In the file entry field, use the arrow keys to go to the name of the file you want to load (for example TIMING.NEW),
- When you are sure that the name you highlighted is correct, press the *SELECT* key to accept it; then select the return (✓) button to start loading.