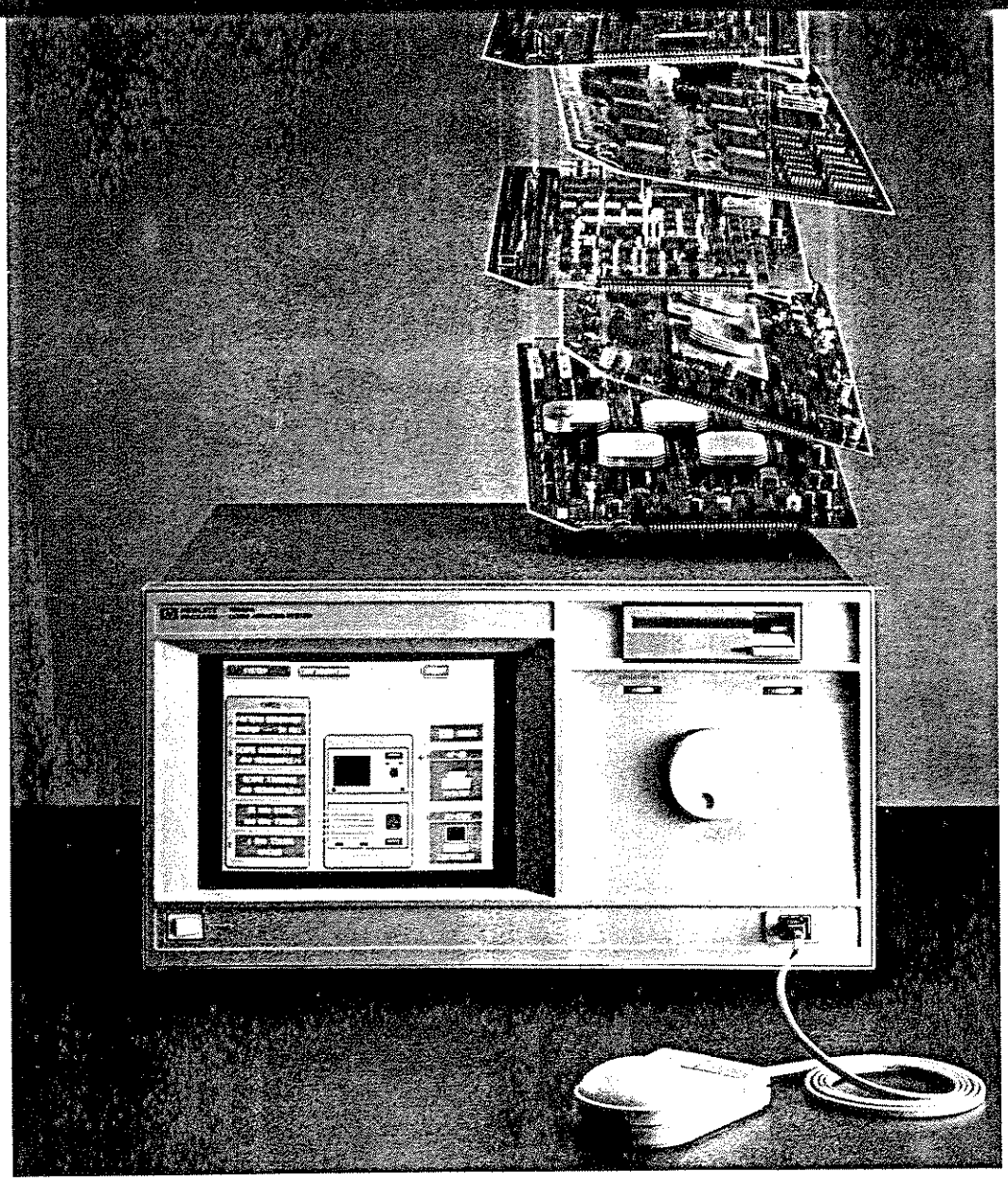
HP 16532A 1 GSa/s Digitizing Oscilloscope Module

for the HP 16500A Logic Analysis System

Programming Reference



HEWLETT PACKARD

Programming Reference

HP 16532A Oscilloscope Module

for the HP 16500A Logic Analysis System

© Copyright Hewlett-Packard Company 1991

Manual Part Number 16532-90903

Printed in U.S.A. June 1991

Product Warranty

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

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

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

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

Limitation of Warranty

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

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

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

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

For any assistance, contact your nearest Hewlett-Packard Sales and Service Office.

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

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

Table of Contents

Chapter 4: CHANnel Subsystem

Chapter 5: DISPlay Subsystem

Chapter 6: MARKer Subsystem

Chapter 7: MEASure Subsystem

Chapter 8: TIMebase Subsystem

Chapter 9: TRIGger Subsystem

Chapter 10: WAVeform Subsystem

Index

1

Programming the HP 16532A

Introduction

This manual combined with the *HP 16500A/16501A Programming Reference* manual provides you with the information needed to program the HP 16532A Oscilloscope Module. Each module has its own manual to supplement the mainframe manual since not all mainframes will be configured with the same modules.

About This Manual

This manual is organized into ten chapters. The first chapter contains:

- General information and instructions to help you get started
- Mainframe system commands that are frequently used with the oscilloscope module
- HP 16532A Oscilloscope command tree
- Alphabetic command-to-subsystem directory

Chapter two contains module level commands. Chapters three through ten contain the subsystem commands for the oscilloscope.

Error messages for the HP 16532A are included in generic system error messages and are in the *HP 16500A/16501A Programming Reference* manual.

Programming the HP 16532A Oscilloscope

This section introduces you to the basic command structure used to program the oscilloscope. Also included is an example program that displays a waveform and makes automatic parametric measurements.

Selecting the Module

Before you can program the oscilloscope, you must first "select" it, otherwise, there is no way to direct your commands to the oscilloscope.

To select the module, use the system command :SELect followed by the numeric reference for the slot location of the oscilloscope (1...10 refers to slot A...J respectively). For example, if the oscilloscope card is in slot E, then the command:

:SELect 5

would select this module. For a multi-card configuration you would select the topmost card slot of the multi-card configuration. For more information on the select command, refer to the *HP 16500A/16501A Programming* manual.

Setting up an Oscilloscope

The easiest and fastest way to set up the oscilloscope is to use the AUTOSCALE command. The AUTOSCALE command causes the oscilloscope to automatically select the vertical sensitivity, vertical offset, trigger source, trigger level and timebase settings for optimum viewing of any input signals. The trigger source is the lowest channel on which the trigger was found. If no signal is found, the oscilloscope defaults to auto-trigger.

To demonstrate a quick oscilloscope setup, we will use the AC CAL OUTPUT signal available at the rear panel of the card. This square wave is normally used for calibration and probe compensation.

Connect the AC CAL OUTPUT signal from the rear panel output connector to CHAN 1, also on the rear panel. Ensure that the mainframe is connected to a controller. Enter the program listed on the next page and execute it.

Example Program:

```
10 OUTPUT XXX;":SELECT 5"
20 OUTPUT XXX;":AUTOSCALE"
25 WAIT 5
30 DIM Me$[200]
40 OUTPUT ;":MEASURE:SOURCE CHANNEL1;ALL?"
50 ENTER XXX;Me$
60 PRINT Me$
70 END
```

Note

The three Xs (XXX) after the OUTPUT and ENTER statements in the above example refer to the device address required for programming over either HP-IB or RS-232-C. Refer to your controller manual and programming language reference manual for information on initializing the interface.

Program Comments

Line 10 selects the oscilloscope in slot E.
Line 20 causes the oscilloscope to execute the AUTOSCALE command.
Line 25 causes the oscilloscope to wait 5 seconds (the time you allow for the measurement to be complete)
Line 30 dimensions and reserves memory for the string array
Line 40 causes the oscilloscope to make all the parametric measurements of the Measure subsystem. The source for the measurements is channel 1.
Line 50 enters data from the oscilloscope.
Line 60 causes the data to be printed either on controller screen or hardcopy, depending on the output device chosen.

For more information on the specific oscilloscope commands, refer to chapters 2 through 10 of this manual.

Mainframe Commands

These commands are part of the HP 16500A/16501A mainframe system and are mentioned here only for reference. For more information on these commands, refer to the *HP 16500A/16501A Programming Reference* manual.

CARDcage? query

The CARDcage query returns a series of integers which identify the modules that are installed in the mainframe. The returned string is in two parts. The first five two-digit numbers identify the card type. The identification number for the HP 16532A oscilloscope is 13. A "-1" in the first part of the string indicates no card is installed in the slot.

The five single-digit numbers in the second part of the string indicate which slots have cards installed, which card has the controlling software for the module, and where the master card is located.

Example (for HP 16500A mainframe without the HP 16501A Expander):

-1,13,-1,-1,31,0,2,0,0,5

The first five numbers of the returned string of -1,13,-1,-1,31,0,2,0,0,5 means:

Slot A is empty (-1).
The oscilloscope (ID number 13) is loaded in slot B.
The next two slots (C and D) are empty (-1).
Slot E contains a logic analyzer module (ID number 31).

The next group of numbers (0,2,0,0,5) mean:

Either Slot A is empty (0), the module software is not recognized, or the module software is not loaded.
A single card module is installed in slot B.
Either slots C and D (0) are empty, the module software is not recognized, or the module software is not loaded.
The last digit (5) in this group indicates that a single module card is loaded in slot E.

Complete information for the CARDcage query is in the *HP 16500A/16501A Programming Reference* manual.

MENU command/query

The MENU command selects the menu to be displayed on the screen. The first parameter specifies the desired module. The optional second parameter specifies the desired menu in the module (defaults to 0 if not specified). The query returns the currently selected (and displayed) menu.

For the HP 16532A Oscilloscope:

- X,0 - Channel Menu
- X,1 - Trigger Menu
- X,2 - Display Menu
- X,3 - Auto-Measure Menu
- X,4 - Calibration Menu

X = slot number that contains the oscilloscope card

SELect command/query

The SELect command selects which module or intermodule will have parser control. SELect 0 selects system/intermodule, and assumming the HP 16501 extender is being used, SELect 1 through 10 selects modules A through J respectively. Parameters -1 and -2 select software options 1 and 2. The SELect query returns the currently selected module.

STARt command

The STARt command starts the selected module or intermodule. If the selected module is configured for intermodule, STARt will start all modules configured for intermodule.

STOP command

The STOP command stops the selected module or intermodule. If the selected module is configured for intermodule, STOP will stop all modules configured for intermodule.

RMODe command/query

The RMODe command specifies the run mode (either single or repetitive) for a module or intermodule. If the selected module is configured for intermodule, the intermodule run mode will be set by this command. The RMODe query returns the current setting.

SYSTem:ERRor? query

The SYSTem:ERRor query returns the oldest error in the error queue. In order to return all the errors in the error queue, a simple FOR/NEXT loop can be written to query the queue until all errors are returned. Once all errors are returned, the queue will return zeros.

SYSTem:PRINt command/query

The SYSTem:PRINt command initiates a print of the screen or listing buffer over the current printer communication interface. The SYSTem:PRINt query sends the screen or listing buffer data over the current controller communication interface.

MMEMory Subsystem

The MMEMory Subsystem commands provide access to both internal disc drives for loading and storing configurations.

INTermodule Subsystem

The INTermodule Subsystem commands are used to specify intermodule arming between multiple modules.

Command Set Organization

The command set for the HP 16532A is divided into module level commands and subsystem commands. Module level commands are listed in Chapter 2 and each of the subsystem commands are covered in their individual chapters starting with Chapter 3.

Each of these chapters contain a description of the subsystem, syntax diagrams and the commands in alphabetical order. The commands are shown in longform and shortform using upper and lowercase letters. For example, TRIGger indicates that the longform of the command is TRIGGER and the shortform is TRIG. Each of the commands contains a description of the command and its arguments, the command syntax, and a programming example.

Figure 1-1 is the command tree for the HP 16532A oscilloscope module.

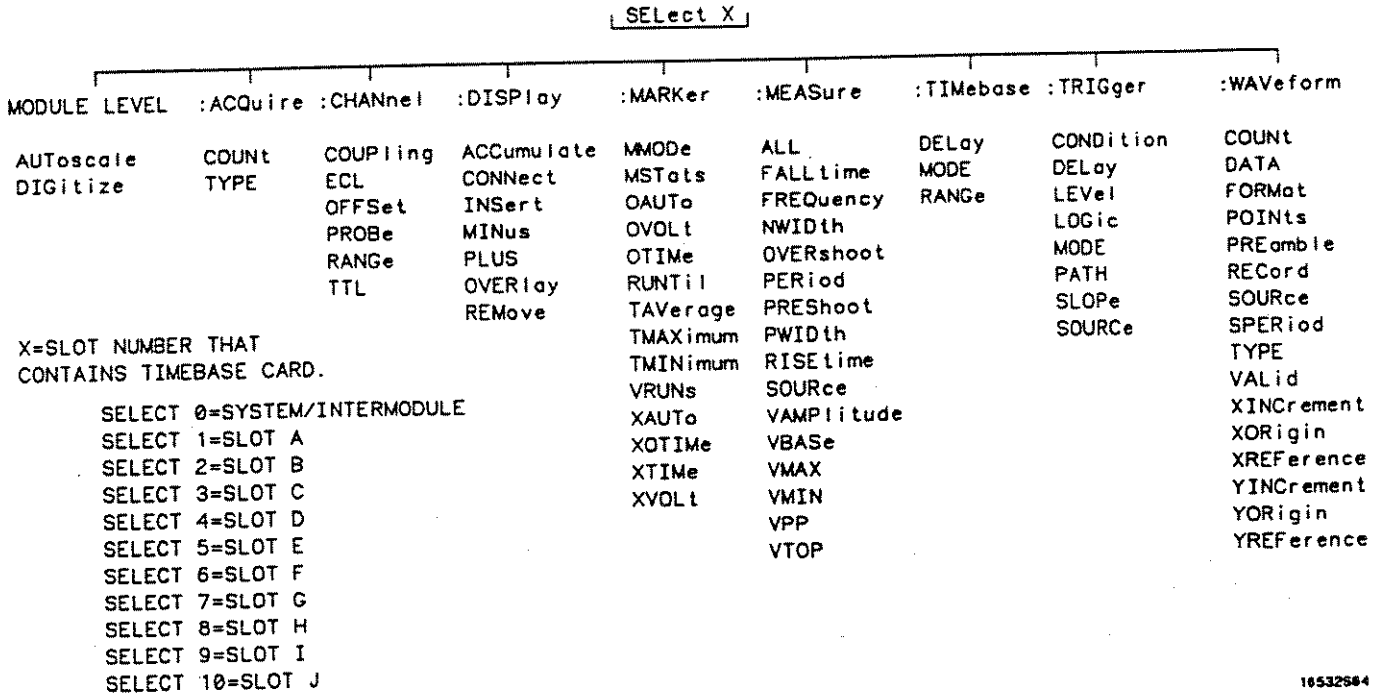


Figure 1-1. HP 16532A Command Tree

Table 1-1. Alphabetical Command to Subsystem Directory

COMMAND	WHERE USED
ACCumulate	DISPlay
AUToscale	MODULE LEVEL
ALL	MEASure
CONDition	TRIGger
CONNect	DISPlay
COUNt	ACQuire
	WAVeform
COUPling	CHANnel
DATA	WAVeform
DELay	TIMebase
	TRIGger
DIGitize	MODULE LEVEL
ECL	CHANnel
FALLtime	MEASure
FORMat	WAVeform
FREQuency	MEASure
INSert	DISPlay
LEVel	TRIGger
LOGic	TRIGger
MINus	DISPlay
MMODe	MARKer
MODE	TIMebase
	TRIGger
MSTats	MARKer
NWIDth	MEASure
OAUTo	MARKer
OFFSet	CHANnel
OTIMe	MARKer
OVERlay	DISPlay
OVERshoot	MEASure
OVOLt	MARKer
PATH	TRIGger
PERiod	MEASure
PLUS	DISPlay
POINts	WAVeform
PREamble	WAVeform
PREShoot	MEASure
PROBe	CHANnel
PWIDth	MEASure
RANGe	CHANnel
	TIMebase
RECord	WAVeform
REMove	DISPlay
RISetime	MEASure
RUNtil	MARKer
SLOPe	TRIGger
SOURce	MEASure
	TRIGger
	WAVeform
SPERiod	WAVeform
TAVerage	MARKer
TMAXimum	MARKer
TMINimum	MARKer
TTL	CHANnel
TYPE	ACQuire
	WAVeform
VALid	WAVeform
VAMPlitude	MEASure
VBASe	MEASure
VMAX	MEASure
VMIN	MEASure
VPP	MEASure
VRUNs	MARKer
VTOP	MEASure
XAUTo	MARKer
XINCrement	WAVeform
XORigin	WAVeform
XREFerence	WAVeform
XOTime	MARKer
XTIMe	MARKer
XVOLt	MARKer
YINCrement	WAVeform
YORigin	WAVeform
YREFerence	WAVeform

16532S05

Module Status Reporting

Each module reports its status to the Module Event Status Register (MESR<N>) which in turn reports to the Combined Event Status Register (CESR) in the HP 16500A/16501A mainframe (see *HP 16500A/16501A Programming Reference* manual Chapter 6). The Module Event Status Register is enabled by the Module Event Status Enable Register (MESE<N>).

The following descriptions of the MESE<N> and MESR<N> commands provide the module specific information needed to enable and interpret the contents of the registers.

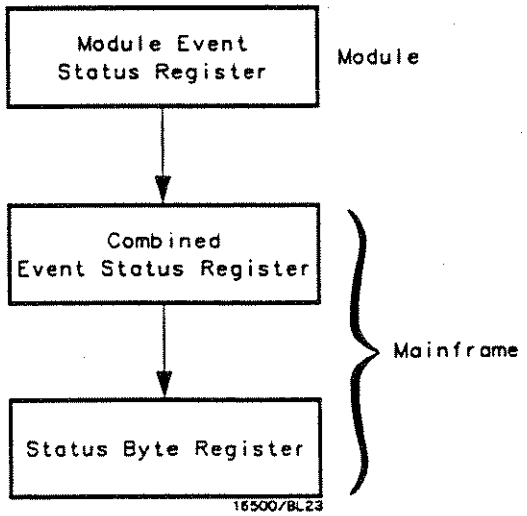


Figure 1-2. Module Status Reporting

MESE<N> command/query

The MESE<N> command sets the Module Event Status Enable register bits. The MESE register contains a mask value for the bits enabled in the MESR register. A one in the MESE register will enable the corresponding bit in the MESR, a zero will disable the bit.

The first parameter specifies the module, and the second parameter specifies the enable value. 1...10 refers to the module in slot A...J.

The MESE query returns the current setting.

Refer to table 1-2 for information about the Module Event Status Enable register bits, bit weights, and what each bit masks for the module.

Command Syntax: :MESE<N> <enable_mask>

where:

<N> ::= {1|2|3|4|5|6|7|8|9|10} number of slot where the module resides
<enable_mask> ::= integer 0 to 255

Example: OUTPUT XXX;":MESE5 2"

Query Syntax: :MESE<N>?

Returned Format: [MESE]<enable_mask><NL>

Example:

```
10 OUTPUT XXX;":MESE5?"
20 ENTER XXX; Mes
30 PRINT Mes
40 END
```

Table 1-2. Module Event Status Enable Register

Module Event Status Enable Register (A "1" enables the corresponding MESR bit)		
Bit	**Weight**	**Enables**
7	128	Not used
6	64	Not used
5	32	Not used
4	16	Number of averages met
3	8	Auto triggered
2	4	Trigger received
1	2	RNT-Run until satisified
0	1	MC-Measurement complete

The Module Event Status Enable Register contains a mask value for the bits to be enabled in the Module Event Status Register (MESR). A one in the MESE enables the corresponding bit in the MESR, a zero disables the bit.

MESR<N> query

The MESR<N> query returns the contents of the Module Event Status register.

Note Reading the register clears the Module Event Status Register.

Table 1-3 shows each bit in the Module Event Status Register and their bit weights for this module. When you read the MESR, the value returned is the total bit weights of all bits that are high at the time the register is read.

The parameter 1...10 refers to the module in slot A...J respectively.

Query Syntax: :MESR<N>?

Returned Format: [MESR]<status><NL>

where:

<N> ::= {1|2|3|4|5|6|7|8|9|10} number of slot where the module resides
<status> ::= integer 0 to 255

Example:

```
10 OUTPUT XXX;":MESR5?"
20 ENTER XXX; Mer
30 PRINT Mer
40 END
```

Table 1-3. Module Event Status Register

Module Event Status Register		
Bit	**Weight**	**Condition**
7	128	Not used
6	64	Not used
5	32	Not used
4	16	1 = Number of averages satisfied 0 = Number of averages not satisfied
3	8	1 = Auto trigger received 0 = Auto trigger not received
2	4	1 = Trigger received 0 = No trigger received
1	2	1 = Run until satisified 0 = Run until not satisified
0	1	1 = Measurement complete 0 = Measurement not complete

Notes

2

Module Level Commands

Introduction

Oscilloscope Module Level commands control the basic operation of the oscilloscope. Refer to figure 2-1 for the module level syntax command diagram. The HP 16532A Module Level commands are:

- AUToscale
- DIGitize

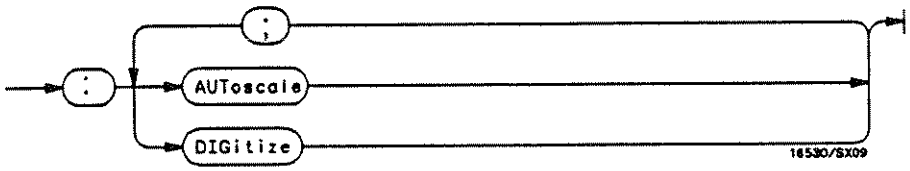


Figure 2-1. Module Level Command Syntax Diagram

AUToscale command

The AUToscale command causes the oscilloscope to automatically select the vertical sensitivity, vertical offset, trigger source, trigger level and timebase settings for optimum viewing of any input signals. The trigger source is the lowest channel on which the trigger was found. If no trigger is found, the oscilloscope defaults to auto-trigger. The display window configuration is not altered by AUToscale.

Command Syntax: :AUToscale

Example: OUTPUT XXX;":AUTOSCALE"

DIGitize command

The DIGitize command is used to acquire waveform data for transfer over HP-IB. The command initiates the Repetitive Run for the oscilloscope and any modules that are grouped together in Group Run through the Intermodule Bus. If a RUNtil condition has been specified in any module, the oscilloscope and the grouped modules will acquire data until the RUNtil conditions have been satisfied.

The Acquire subsystem commands may be used to set up conditions such as acquisition type and average count for the DIGitize command. See the Acquire subsystem for the description of these commands.

When a count number in the average acquisition type has been specified, the oscilloscope and all grouped modules will acquire data until these conditions have been satisfied.

When both the RUNtil and the ACQuire:COUNt have been satisfied, the acquisition will stop.

For a faster data transfer rate over the interface bus, display a menu that has no waveforms on screen.

Note 

The DIGitize command is an overlap command, thus ensure that all data has been acquired and stored in the channel buffers before executing any other commands. The MESE command and the MESR query may be used to check for run complete or a WAIt instruction may be inserted after the DIGitize command to ensure enough time for command execution.

Command Syntax: :DIGitize

Example:

```
10 CLEAR 707
20 OUTPUT XXX;":SELECT 4"
30 OUTPUT XXX;":SYSTEM:HEADER OFF;:EOI ON"
40 OUTPUT XXX;":ACQUIRE:TYPE AVERAGE"
50 OUTPUT XXX;":WAVEFORM:SOURCE CHANNEL1"
60 OUTPUT XXX;":WAVEFORM:FORMAT WORD"
70 OUTPUT XXX;":WAVEFORM:RECORD FULL"
80 OUTPUT XXX;":AUTOSCALE"
90 DIM Header$[20]
100 OUTPUT XXX;":DIGITIZE"
105 WAIT 10
110 Length = 8000
120 ALLOCATE INTEGER Waveform(1:Length)
130 OUTPUT XXX;":WAVEFORM:DATA?"
140 ENTER XXX USING "#,10A";Header$
150 ENTER XXX USING "#,W";Waveform(*)
160 END
```

3

ACQuire Subsystem

Introduction

The Acquire Subsystem commands are used to set up acquisition conditions for the DIGitize command. The subsystem contains commands to select the type of acquisition and the number of averages to be taken if the average type is chosen. Refer to Figure 3-1 for the ACQuire Subsystem Syntax Diagram. The ACQuire Subsystem commands are:

- COUNt
- TYPE

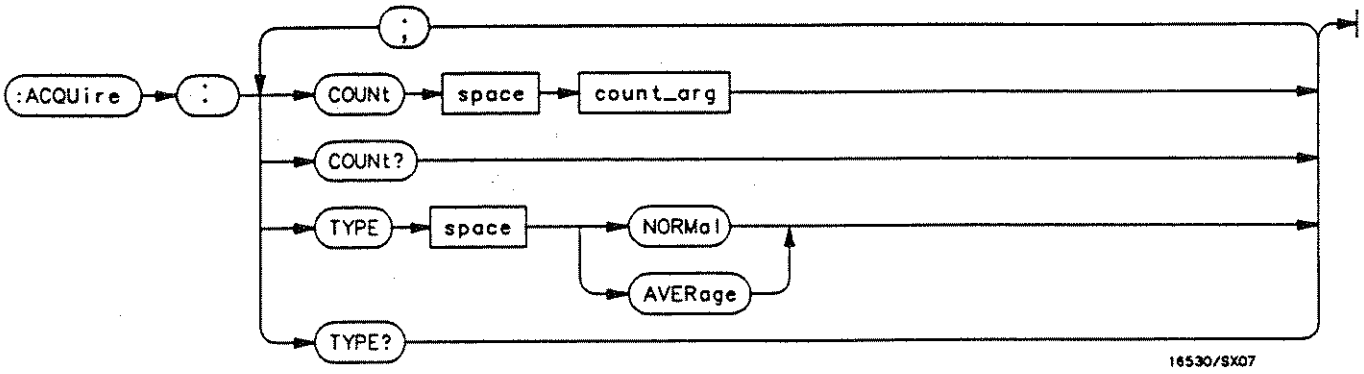


Figure 3-1. ACQuire Subsystem Syntax Diagram

count_arg = *An integer that specifies the number of averages to be taken of each time point. The choices are 2, 4, 8, 16, 32, 64, 128, or 256.*

Acquisition Type Normal

In the Normal mode, with the ACCumulate command OFF, the oscilloscope acquires waveform data and then displays the waveform. When the oscilloscope makes a new acquisition, the previously acquired waveform is erased from the display and replaced by the newly acquired waveform.

When the ACCumulate command is ON, the oscilloscope displays all the waveform acquisitions without erasing the previously acquired waveform.

Acquisition Type Average

In the Average mode, the oscilloscope averages the data points on the waveform with previously acquired data. Averaging helps eliminate random noise from the displayed waveform. In this mode the ACCumulate command is OFF. When Average mode is selected, the number of averages must also be specified using the COUNt command. Previously averaged waveform data is erased from the display and the newly averaged waveform is displayed.

COUNt command/query

The COUNt command specifies the number of acquisitions for the running weighted average. This command generates an error if Normal acquisition mode is specified. The query returns the last specified count.

Command Syntax: :ACQuire:COUNt <count>

where

<count> ::= {2|4|8|16|32|64|128|256}

Example OUTPUT XXX;":ACQUIRE:COUNT 16"

Query Syntax: :ACQuire:COUNt?

Returned Format [:ACQuire:COUNt] <count><NL>

Example:

```
10 DIM Ac$[100]
20 OUTPUT XXX;":ACQ:COUN?"
30 ENTER XXX;Ac$
40 PRINT Ac$
50 END
```

TYPE command/query

The TYPE command selects the type of acquisition that is to take place when a DIGitize or STARt command is executed. One of two acquisition types may be chosen: the NORMal or AVERage mode. The query returns the last specified type.

Command Syntax :ACQuire:TYPE {NORMal|AVERage}

Example: OUTPUT XXX;":ACQUIRE:TYPE NORMAL"

Query Syntax: :ACQuire:TYPE?

Returned Format: [:ACQuire:TYPE] {NORMal|AVERage}<NL>

Example:

```
10 DIM At$[100]
20 OUTPUT XXX;":ACQUIRE:TYPE?"
30 ENTER XXX;At$
40 PRINT At$
50 END
```

4

CHANnel Subsystem

Introduction

The Channel Subsystem commands control the channel display and the vertical axis of the oscilloscope. Each channel must be programmed independently for all offset, range and probe functions. When ECL or TTL commands are executed, the vertical range, offset and trigger levels are automatically set for optimum viewing. Refer to figure 4-1 for the CHANnel Subsystem Syntax Diagram. The CHANnel Subsystem commands are:

- COUPling
- ECL
- OFFSet
- PROBe
- RANGe
- TTL

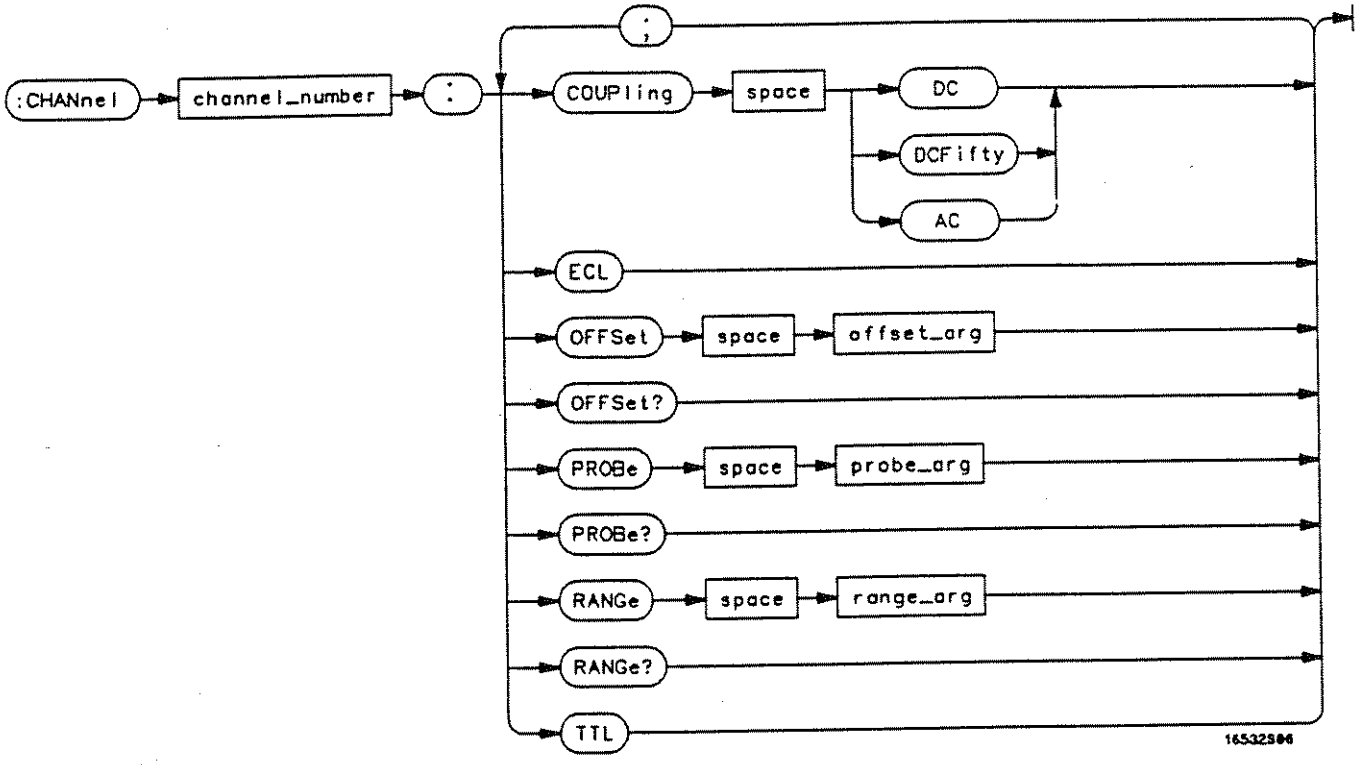


Figure 4-1. CHANnel Subsystem Syntax Diagram

channel_number = *An integer from 1 through 8, depending on how many oscilloscope cards are installed as a single unit (chained together).*

offset_arg = *a real number defining the voltage at the center of the display. The offset range is as follows (for a 1:1 probe setting):*

Vertical Sensitivity	Vertical Range	Offset Voltage
4 mV - 100 mV/div	16 mV - 400 mV	±2 V
>100 mV - 400 mV/div	>400 mV - 1.6 V	±10 V
>400 mV - 2.5 V/div	>1.6 V - 10 V	±50 V
>2.5 V - 10 V/div	>10 V - 40 V	±250 V

probe_arg = *an integer from 1 through 1000, specifying the probe attenuation with respect to 1.*

range_arg = *a real number specifying vertical sensitivity. The allowable range is 16 mV to 40 V for a probe attenuation of 1. The specified range is equal to 4 times Volts/Div.*

Figure 4-1. CHANnel Subsystem Syntax Diagram (continued)

COUPling

command/query

The COUPling command sets the input impedance for the selected channel. The choices are 1M Ohm DC (DC), 1M Ohm AC (AC), or 50 Ohms DC (DCFifty). The query returns the current input impedance for the specified channel.

Command Syntax: :CHANnel<N>:COUPling {DC|AC|DCFifty}

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2.

Example:

OUTPUT XXX;":CHANNEL1:COUPLING DC"

Query Syntax: :CHANnel<N>:COUPling?

Returned Format: [:CHANnel<N>:COUPling:] {DC|AC|DCFifty}<NL>

Example:

```
10 DIM Cc$[100]
20 OUTPUT XXX;":CHANNEL1:COUPLING?"
30 ENTER XXX;Cc$
40 PRINT Cc$
50 END
```

ECL command

The ECL command sets the vertical range, offset, and trigger levels for the selected input channel for optimum viewing of ECL signals. The set ECL values are:

Range: 2.0 V (500 mV per division)
Offset: -1.3 V
Trigger level: -1.3 V

Command Syntax: :CHANnel<N>:ECL

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2.

Example: OUTPUT XXX;":CHANNEL1:ECL"

To return to "Preset User", change the CHANnel:RANGe, CHANnel:OFFSet, or TRIGger:LEVel value.

OFFSet

command/query

The OFFSet command sets the voltage that is represented at center screen for the selected channel. The allowable offset voltage <value> is shown in the table below. The table represents values for a Probe setting of 1:1. The offset value is recompensated whenever the probe attenuation factor is changed. The query returns the current value for the selected channel.

Command Syntax: :CHANnel<N>:OFFSet <value>

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2.

<value> ::= allowable offset voltage value shown in the table below

Vertical Range (4 times V/Div)	Offset Voltage
16 mV - 400 mV	±2 V
>400 mV - 1.6 V	±10 V
>1.6 V - 10 V	±50 V
>10 V - 40 V	±250 V

Example: OUTPUT XXX;":CHAN1:OFFS 1.5"

Query Syntax: :CHANnel<N>:OFFSet?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2.

Returned Format: [:CHANnel<N>:OFFSet] <value><NL>

Example:

```
10 DIM Co$[100]
20 OUTPUT XXX;":CHANNEL1:OFFSET?"
30 ENTER XXX;Co$
40 PRINT Co$
50 END
```

PROBe

command/query

The PROBe command specifies the attenuation factor for an external probe connected to a channel. The command changes the channel voltage references such as range, offset, trigger level and automatic measurements. The actual sensitivity is not changed at the channel input. The allowable probe attenuation factor is an integer from 1 to 1000. The query returns the probe attenuation factor for the selected channel.

Command Syntax: :CHANnel<N>:PROBe <atten>

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2.

<atten> ::= 1 to 1000 (Integer)

Example: OUTPUT XXX;":CHAN1:PROB 10"

Query Syntax: :CHANnel<N>:PROBe?

Returned Format: [:CHANnel<N>:PROBe]<atten><NL>

Example:

```
10 DIM Att$[100]
20 OUTPUT XXX;":CHANNEL1:PROBE?"
30 ENTER XXX;Att$
40 PRINT Att$
50 END
```

RANGe

command/query

The RANGe command defines the full-scale (4 • Volts/Div) vertical axis of the selected channel. The values for the RANGe command are dependent on the current probe attenuation factor for the selected channel. The allowable range for a probe attenuation factor of 1:1 is 16 mV to 40 V. For a larger probe attenuation factor, multiply the range limit by the probe attenuation factor. The query returns the current range setting.

Command Syntax: :CHANnel<N>:RANGe <range>

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2.

<range> ::= 16 mV to 40 V for a probe attenuation factor of 1:1

Example: OUTPUT XXX;":CHANNEL1:RANGE 4.8"

Query Syntax: :CHANnel<N>:RANGe?

Returned Format: [:CHANnel<N>:RANGe] <range><NL>

where:

<range> ::= 16 mV to 40 V when probe attenuation factor is set to 1:1

Example:

```
10 DIM Pr$[100]
20 OUTPUT XXX;":CHANNEL1:RANGE?"
30 ENTER XXX;Pr$
40 PRINT Pr$
50 END
```

TTL command

The TTL command sets the vertical range, offset, and trigger level for the selected input channel for optimum viewing of TTL signals. The set TTL values are:

Range: 6.0 V (1.50 V per division)
Offset: 2.5 V
Trigger Level: 1.62 V

Command Syntax: :CHANnel<N>:TTL

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2.

Example: OUTPUT XXX;":CHANNEL1:TTL"

To return to "Preset User" change the CHANnel:RANGe, CHANel:OFFSet, or TRIGger:LEVel value.

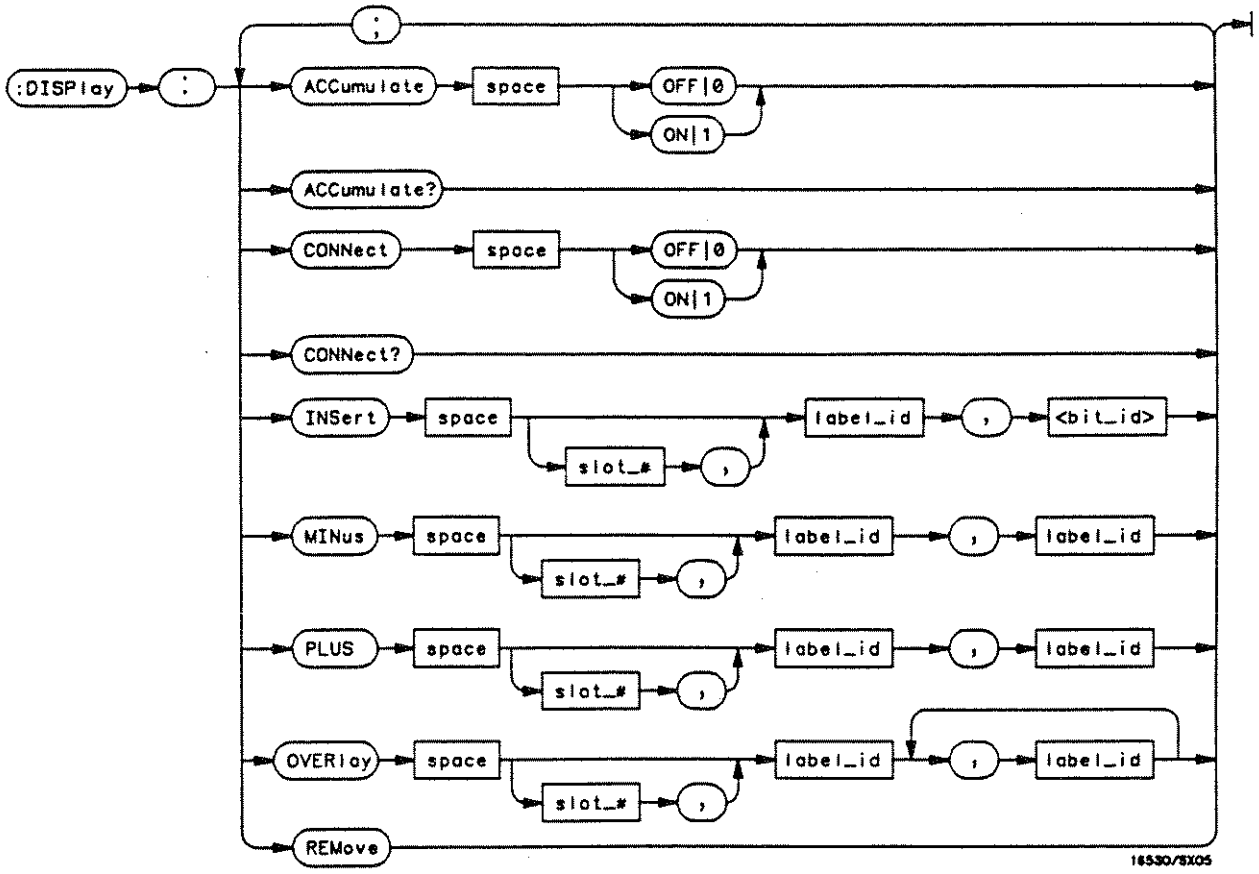
5

DISPlay Subsystem

Introduction

The Display Subsystem is used to control the display of data. Refer to Figure 5-1 for the DISPlay Subsystem Syntax Diagram. The DISPlay Subsystem commands are:

- ACCumulate
- CONNect
- INSert
- MINus
- OVERlay
- PLUS
- REMove



slot_# = *a number from 1 through 10 identifying the oscilloscope card slot*
label_identification = *a string of 1 alpha and 1 numeric character for the oscilloscope, or 6 characters for the timing modules.*

Figure 5-1. DISPlay Subsystem Syntax Diagram

ACCumulate command/query

The ACCumulate command works in conjunction with the commands in the Acquisition Subsystem. In the Normal mode, the ACCumulate command turns the infinite persistence on or off. The query reports if accumulate is turned on or off.

Command Syntax: :DISPLAY:ACCumulate {{ON|1}|{OFF|0}}

Example: OUTPUT XXX;":DISPLAY:ACC ON"

Query Syntax: :DISPLAY:ACCumulate?

Returned Format: [:DISPlay:ACCumulate] {1|0}<NL>

Example:

```
10 DIM Ac$[100]
20 OUTPUT XXX;":DISPLAY:ACCUMULATE?"
30 ENTER XXX;Ac$
40 PRINT Ac$
50 END
```

CONNect command/query

The CONNect command sets the Connect Dots mode. When ON, each displayed sample dot will be connected to the adjacent dot by a straight line. The waveform is easier to see in this mode. When OFF, only the sampling points will be displayed. The query reports if connect is on or off.

Command Syntax: :DISPlay:CONNect {{ON|1}|{OFF|0}}

Example: OUTPUT XXX;":DISPLAY:CONNECT ON"

Query Syntax: :DISPlay:CONNect?

Returned Format: [:DISPlay:CONNect] {1|0}<NL>

Example:

```
10 DIM Dots$[100]
20 OUTPUT XXX;":DISPLAY:CONNECT?"
30 ENTER XXX;Dots$
40 PRINT Dots$
50 END
```

INSert command

The INSert command inserts waveforms into the current display. Time-correlated waveforms from another oscilloscope module, logic analyzer or high speed timing module may also be added to the current display. The waveforms are added just below any currently displayed signals. Only eight oscilloscope waveforms can be displayed at any time.

The first parameter is optional and specifies the module from where the waveform is to be taken. The module number is the same as the slot number in which the master card is installed. If a module is not specified, the current module is assumed. The second parameter is the label of the waveform that is to be added to the current display. The label names depend on the slot in which the acquisition cards are installed. The example of figure 5-2 shows the labeling scheme for eight oscilloscope channels (four oscilloscope cards).

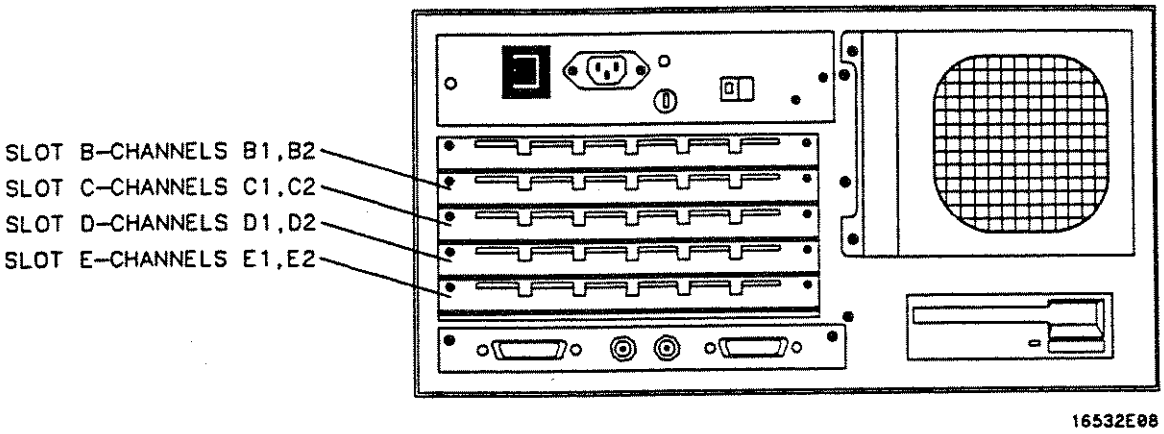


Figure 5-2. Oscilloscope Label Identification

INSert

Command Syntax: Inserting a waveform from the oscilloscope to the oscilloscope display:

:DISPlay:INSert [<module number>,]<label>

where:

<module number> ::= slot in which oscilloscope master card is installed.
<label> ::= string of 1 alpha and 1 numeric character enclosed by single quotes

Example: OUTPUT XXX;":DISPLAY:INSERT 'C1'"

Command Syntax: Inserting a waveform from a logic analyzer or high speed timing module to the oscilloscope display:

:DISPlay:INSert <slot no>,<label>,<bit-id>

where:

<slot no> ::= card slot number of the module from which waveform is to be taken
<label> ::= string of up to 6 alphanumeric characters enclosed by single quotes
<bit-id> ::= integer from 0 to 31

Example: :OUTPUT XXX;":DISPLAY:INSERT 4,'WAVE',10"

For a complete explanation of the label name and the <bit-id>, refer to the *HP 16510A or HP 16510B Logic Analyzer Programming Reference* manual, the *HP 16515A High Speed Timing Programming Reference* manual, or the *HP 16540Aor HP 16541A State/Timing Programming Reference* manual.

MINus command

The MINus command algebraically subtracts one channel from another and inserts the resultant waveform to the display. The first parameter is an optional module specifier. The module is identified by the slot number that contains the oscilloscope master card. If a module is not specified, the current module is assumed. The next two parameters are the label of the waveform selected to be added to the display. The label names are defined in the same manner as the INSert command.

Command Syntax: :DISPlay:MINus [<module number>,]<label>,<label>

where:

<module number> ::= slot number in which oscilloscope master card is installed
<label> ::= string of 1 alpha and 1 numeric character enclosed by single quotes

Example: OUTPUT XXX;":DISPLAY:MINUS 5,'C1','C2'"

REMove command

The REMove command removes all displayed waveforms from the current display.

Command Syntax: :DISPlay:REMove

Example: OUTPUT XXX;":DISPLAY:REMOVE"

6

MARKer Subsystem

Introduction

In addition to automatic parametric measurements, the oscilloscope has two markers for making time and voltage measurements. These measurements may be made automatically or manually. Another feature is the run until time (RUNTil) mode. This feature allows you to set a stop condition based on the time interval between the X marker and the O marker. When this condition is met, the oscilloscope will stop acquiring data. Refer to Figure 6-1 for the Marker Subsystem Syntax Diagram. The MARKer Subsystem commands are:

- MMODe
- MSTats
- OAUTo
- OTIMe
- OVOLt
- RUNTil
- TAVerage
- TMAXimum
- TMINimum
- VRUNs
- XAUTo
- XTIMe
- XOTime
- XVOLt

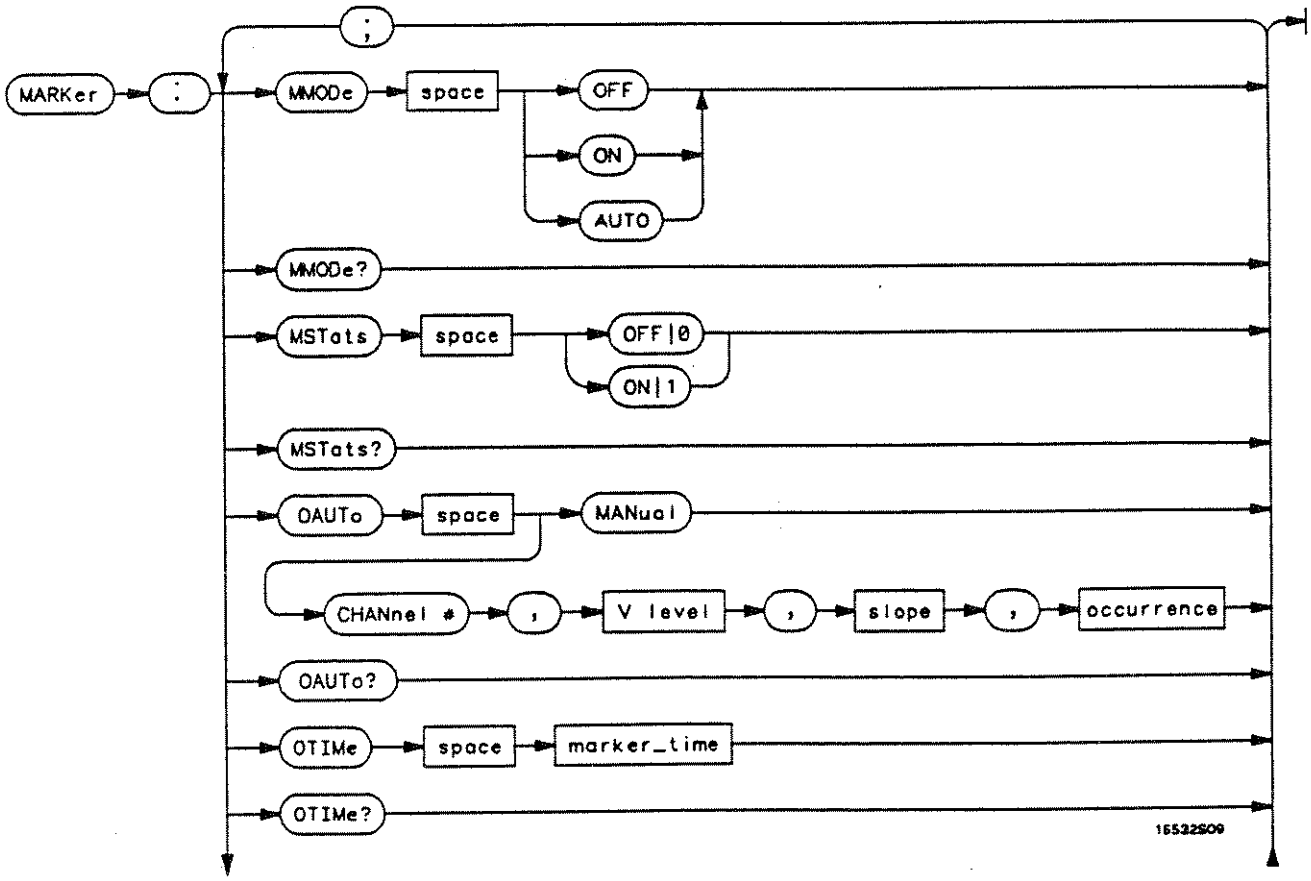


Figure 6-1. MARKer Subsystem Syntax Diagram

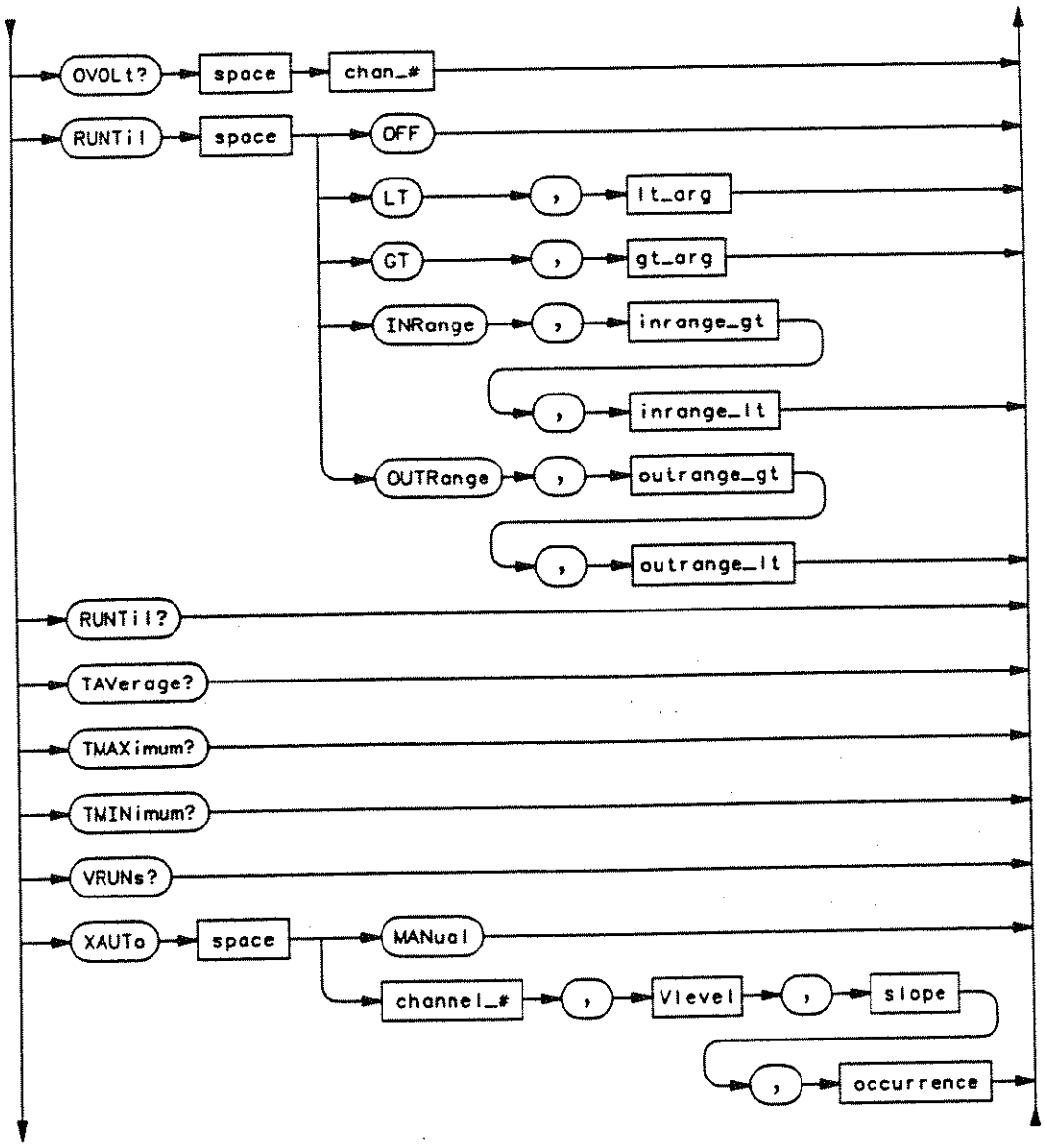
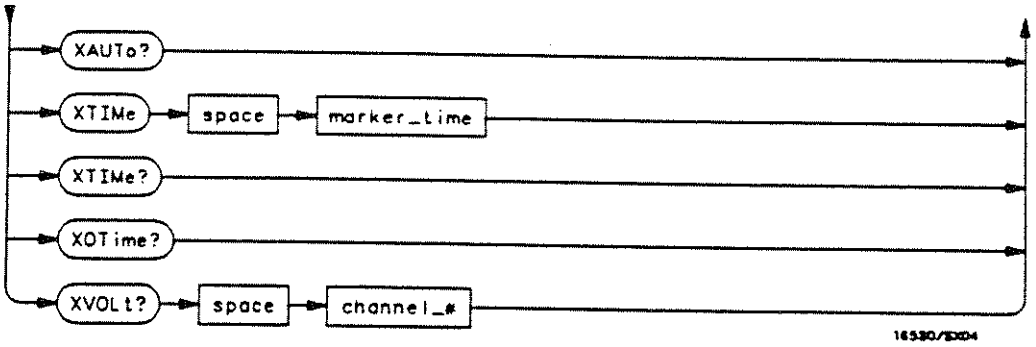


Figure 6-1. MARKer Subsystem Syntax Diagram (Cont'd)



channel_# = *An integer from 1 through 8, depending on how many oscilloscope cards are installed as a single unit (chained together).*
marker_time = *time in seconds from trigger marker to X or O marker*
lt_arg = *time in seconds that specifies the less than (lt) RUNTil time*
gt_arg = *time in seconds that specifies the greater than (gt) RUNTil time*
inrange_gt = *time in seconds specifying the lower limit of the INRange runtime*
inrange_lt = *time in seconds specifying the upper limit of the INRange runtime*
outrange_gt = *time in seconds specifying the lower limit of the OUTRange runtime*
outrange_lt = *time in seconds specifying the upper limit of the OUTRange runtime*
V level = *percentage of waveform voltage level, ranging from 10% to 90% of the* $V_{\mathbf{top}}$ *to* $V_{\mathbf{base}}$ *voltage*
slope = *positive or negative slope*
occurrence = *integer from 1 to 100*

Figure 6-1. MARKer Subsystem Syntax Diagram (Cont'd)

MMODe command/query

The MMODe command allows you to select the marker mode. The choices are: OFF, ON and AUTO. When OFF, marker measurements cannot be made. When the markers are turned on, the X and O markers can be moved to make time and voltage measurements. The AUTO mode allows you to make automatic marker placements by specifying channel, percentage of voltage level, slope and occurrence count for each marker. Also the Statistics mode may be used when AUTO is chosen. Statistics mode allows you to make minimum, maximum and mean time interval measurements from the X marker to the Omarker. The query returns the current marker mode choice.

Command Syntax: :MARKer:MMODe {OFF|ON|AUTO}

Example: OUTPUT XXX;":MARKER:MMODE ON"

Query Syntax: :MARKer:MMODe?

Returned Format: [:MARKer:MMODe] <state> <NL>

where:

<state> ::= ON or OFF or AUTO

Example:

```
10 DIM Mm$[100]
20 OUTPUT XXX;":MARKER:MMODE?"
30 ENTER XXX;Mm$
40 PRINT Mm$
50 END
```

MSTats command/query

The MSTats command allows you to turn statistics ON or OFF in the auto marker mode. When statistics is turned on, Min X-O, Max X-O, and Mean X-O times are displayed on screen. When off, X-O, Trig-X, and Trig-O times will be displayed on screen. The query returns the current setting.

Command Syntax: :MARKer:MSTats {{ON|1}|{OFF|0}}

Example:
```
OUTPUT XXX;":MARKER:MSTATS ON"
```

Query Syntax: :MARKer:MSTats?

Returned Format: [:MARKer:MSTats]{1|0}<NL>

Example:
```
10 DIM Time$[100]
20 OUTPUT XXX;":MARKER:MSTATS?"
30 ENTER XXX;Time$
40 PRINT Time$
50 END
```

OAUTo command/query

The OAUTo command specifies the automatic placement specification for the O marker. The first parameter specifies if automarker placement is to be in the manual mode or on a specified channel. If a channel is specified, three other parameters must be included in the command syntax. The three parameters are: the percentage of waveform voltage level, the slope, and the occurrence count. The query returns the current settings.

Command Syntax: :MARKer:OAUTo{ MANual|CHANnel<N>,<level>,<slope>,<occurrence>}

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

<level> ::= percentage of waveform voltage level, ranging from 10% to 90% of the V_{top} to V_{base} voltage

<slope> ::= POSitive or NEGative

<occurrence> ::= integer from 1 to 100

Example:
```
OUTPUT XXX;":MARKER:OAUTO CHANNEL1,50,POSITIVE,5"
```

Query Syntax: :MARKer:OAUTo?

Returned Format: [:MARKer:OAUTo] CHANnel<N>,<level>,<slope>,<occurrence><NL>

Example:
```
10 DIM Oam$[100]
20 OUTPUT XXX;":MARKER:OAUTO?"
30 ENTER XXX;Oam$
40 PRINT Oam$
50 END
```

OTIMe

command/query

The OTIMe command moves the O marker to the specified time with respect to the trigger marker. The query returns the time in seconds between the O marker and the trigger marker.

Command Syntax: :MARKer:OTIMe <O marker time>

where:

<O marker time> ::= time in seconds from trigger marker to O marker.

Example:
```
OUTPUT XXX;":MARKER:OTIME 1E-6"
```

Query Syntax: :MARKer:OTIMe?

Returned Format: [:MARKer:OTIMe] <O marker time> <NL>

Example:
```
10 DIM Otime$[100]
20 OUTPUT XXX;":MARKER:OTIME?"
30 ENTER XXX;Otime$
40 PRINT Otime$
50 END
```

OVOLt query

The OVOLt query returns the current voltage level of the selected source at the O marker.

Query Syntax: :MARKer:OVOLt? CHANNEL<N>

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MARKer:OVOLt]<level><NL>

where:

<level> ::= level in volts where the O marker crosses the waveform.

Example:

```
10 DIM Ovm$[100]
20 OUTPUT XXX;":MARKER:OVOLT? CHANNEL1?"
30 ENTER XXX;Ovm$
40 PRINT Ovm$
50 END
```

RUNTil

command/query

The RUNTil command allows you to set a stop condition based on the time interval between the X marker and the O marker. In repetitive runs, when the time specification is met, the oscilloscope stops acquiring data and the advisory "Stop condition satisfied" will be displayed on screen. The query will return the current Run Until Time X - O (RUNTil) setting.

Command Syntax: :MARKer:RUNTil {OFF|LT,<time>|GT,<time>|INRange,<time>, <time>|OUTRange,<time>, <time>}

where:

<time> ::= a real number specifying the time in seconds between the X and O markers

Example:
```
OUTPUT XXX;":MARKER:RUNTIL LT,1MS"
```

Query Syntax: :MARKer:RUNTil?

Returned Format: [:MARKer:RUNTil] {OFF|LT,<time>|GT,<time>|INRange,<time>, <time>|OUTRange,<time>,<time>}<NL>

Example:
```
10 DIM Run$[100]
20 OUTPUT XXX;":MARKER:RUNTIL?"
30 ENTER XXX;Run$
40 PRINT Run$
50 END
```

TAVerage query

The TAVerage query returns the average time between the X and O markers. If there is no valid data, the query returns 9.9E37.

Query Syntax: :MARKer:TAVerage?

Returned Format: [:MARKER:TAVERAGE] <time value><NL>

where:

<time value> ::= real number

Example:

```
10 DIM Tv$[100]
20 OUTPUT XXX;":MARKER:TAVERAGE?"
30 ENTER XXX;Tv$
40 PRINT Tv$
50 END
```

TMAXimum query

The TMAXimum query returns the value of the maximum time between the X and O markers. If there is no valid data, the query returns 9.9E37.

Query Syntax: :MARKer:TMAXimum?

Returned Format: [:MARKer:TMAXimum] <time value> <NL>

where:

<time value> ::= real number

Example:

```
10 DIM Tx$[100]
20 OUTPUT XXX;":MARKER:TMAXIMUM?"
30 ENTER XXX; Tx$
40 PRINT Tx$
50 END
```

TMINimum query

The TMINimum query returns the value of the minimum time between the X and O markers. If there is no valid data, the query returns 9.9E37.

Query Syntax: :MARKer:TMINimum?

Returned Format: [:MARKer:TMINimum] <time value> <NL>

where:

<time value> ::= real number

Example:

```
10 DIM Tm$[100]
20 OUTPUT XXX;":MARKER:TMINIMUM?"
30 ENTER XXX;Tm$
40 PRINT Tm$
50 END
```

VRUNs query

The VRUNs query returns the number of valid runs and the total number of runs made. Valid runs are those where the edge search for both the X and O markers was successful, resulting in valid marker time measurement.

Query Syntax: :MARKer:VRUNs?

Returned Format: [:MARKer:VRUNs] <valid runs>,<total runs><NL>

where:

<valid runs> ::= positive integer
<total runs> ::= positive integer

Example:

```
10 DIM Vr$[100]
20 OUTPUT XXX;":MARKER:VRUNS?"
30 ENTER XXX; Vr$
40 PRINT Vr$
50 END
```

XAUTo command/query

The XAUTo command specifies the automatic placement specification for the X marker. The first parameter specifies if automarker placement is to be in the Manual mode or on a specified channel. If a channel is specified, three other parameters must be included in the command syntax. The three parameters are: the percentage of waveform voltage level, the slope and the occurrence count. The query returns the current settings.

Command Syntax: :MARKer:XAUTo{ MANual | CHANnel <N>, <level>, <slope>, <occurrence>}

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

<level> ::= percentage of waveform voltage level, ranging from 10% to 90% of the V_{top} to V_{base} voltage

<slope> ::= POSitive or NEGative

<occurrence> ::= integer from 1 to 100

Example:
```
OUTPUT XXX;":MARKER:XAUTO CHANNEL1,50,POSITIVE,5"
```

Query Syntax: :MARKer:XAUTo?

Returned Format: [:MARKer:XAUTo] CHANnel <N>, <level>, <slope>, <occurrence> <NL>

Example:
```
10 DIM Xam$[100]
20 OUTPUT XXX;":MARKER:XAUTO?"
30 ENTER XXX;Xam$
40 PRINT Xam$
50 END
```

XOTime query

The XOTime query returns the time in seconds from the X marker to the O marker. If data is not valid, the query returns 9.9E37.

Query Syntax: :MARKer:XOTime?

Returned Format: [:MARKer:XOTime] <time> <NL>

where:

<time> ::= real number

Example:

```
10 DIM Xo$[100]
20 OUTPUT XXX;":MARKER:XOTIME?"
30 ENTER XXX;Xo$
40 PRINT Xo$
50 END
```

XTIMe command/query

The XTIMe command moves the X marker to the specified time with respect to the trigger marker. The query returns the time in seconds between the X marker and the trigger marker.

Command Syntax: :MARKer:XTIMe <X marker time>

where:

<X marker time> ::= time in seconds from trigger marker to X marker

Example:
```
OUTPUT XXX;":MARKER:XTIME 1E-6"
```

Query Syntax: :MARKer:XTIMe?

Returned Format: [:MARKer:XTIMe] <marker time> <NL>

Example:
```
10 DIM Xt$[100]
20 OUTPUT XXX;":MARKER:XTIME?"
30 ENTER XXX;Xt$
40 PRINT Xt$
50 END
```

XVOLt

query

The XVOLt query returns the current voltage level of the selected channel at the X marker.

Query Syntax: :MARKer:XVOLt? CHANnel <N>

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MARKer:XVOLt] <level> <NL>

where:

<level> ::= level in volts where the X marker crosses the waveform

Example:

```
10 DIM Xvm$[100]
20 OUTPUT XXX;":MARKER:XVOLT? CHANNEL1"
30 ENTER XXX;Xvm$
40 PRINT Xvm$
50 END
```

7

MEASure Subsystem

Introduction

The commands/queries in the Measure Subsystem are used to make automatic parametric measurements on displayed waveforms. Measurements are made on the displayed waveform(s) specified by the SOURce command. If the source is not specified, the last waveform source is assumed. Measurements are made in the following manner:

Frequency The frequency of the first complete cycle displayed is measured using the 50% level.

Period The period of the first complete cycle displayed is measured at the 50% level.

Peak-to-Peak The absolute minimum and the maximum voltages for the selected source are measured.

Positive Pulse Width Pulse width is measured at the 50% level of the first displayed positive pulse.

Negative Pulse Width Pulse width is measured at the 50% level of the first displayed negative pulse.

Risetime The risetime of the first displayed rising edge is measured. To obtain the best possible measurement accuracy, select the fastest sweep speed while keeping the rising edge on the display. The risetime is determined by measuring time at the 10% and the 90% voltage points of the rising edge.

Falltime Falltime is measured between the 10% and the 90% points of the first displayed falling edge. To obtain the best possible measurement

accuracy, select the fastest sweep speed possible while keeping the falling edge on the display.

Preshoot and Overshoot Preshoot and overshoot measure the perturbation on a waveform above or below the top and base voltages.

Preshoot is a perturbation before a rising or a falling edge and measured as a percentage of the top-base voltage.

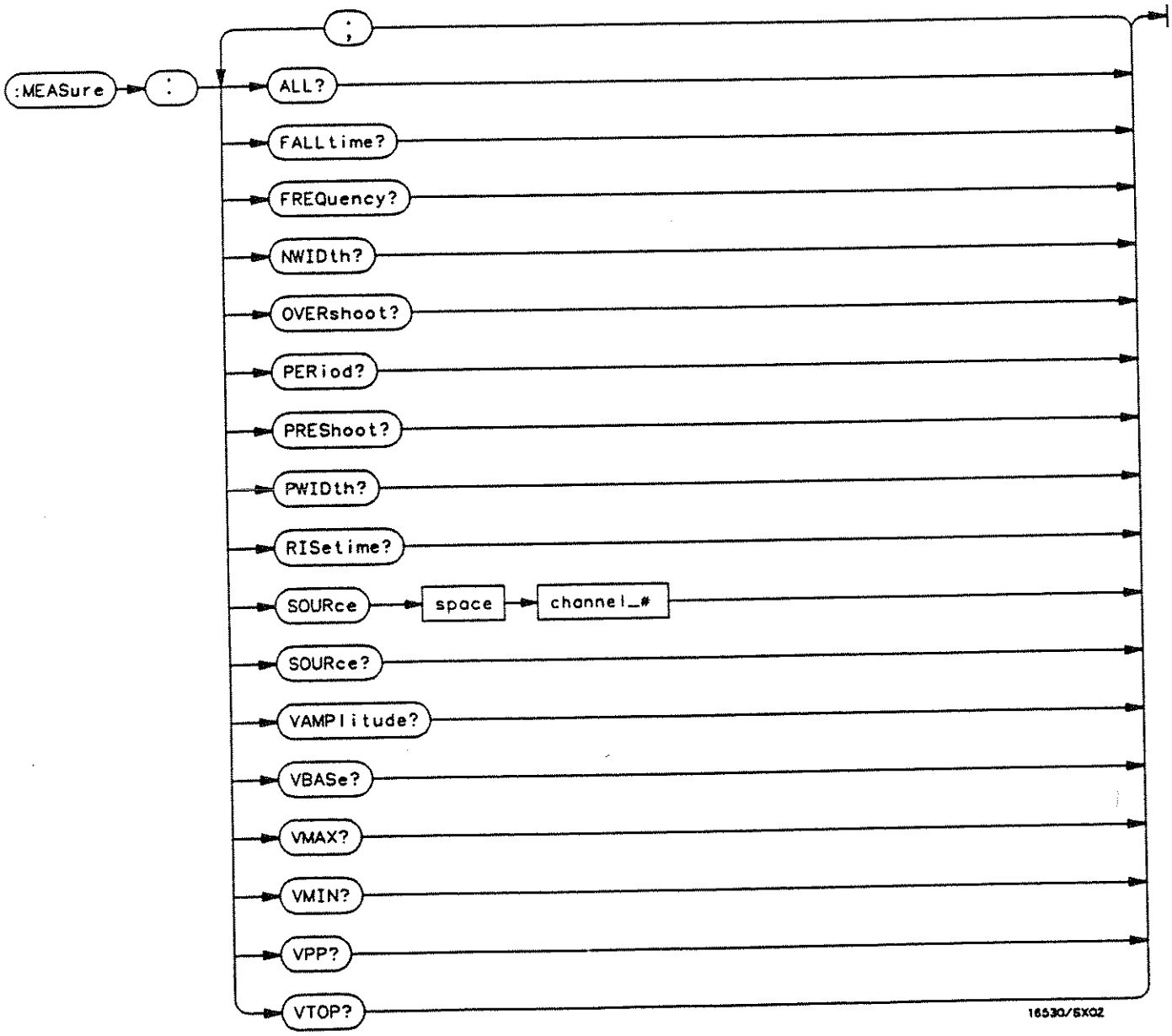
Overshoot is a perturbation after a rising or falling edge and is measured as a percentage of the top-base voltage.

For complete details of the measurement algorithms, refer to the User's Reference Manual.

Refer to Figure 7-1 for the MEASure Subsystem Syntax Diagram

Before using any of the Measure Subsystem queries, note that the SOURce command is part of every query of this subsystem. The SOURce command specifies the channel that is to be used for making the measurements.

If a parameter cannot be measured, the instrument responds with 9.9E37.



channel_# = *An integer from 1 through 8, depending on how many oscilloscope cards are installed as a single unit (chained together)*

Figure 7-1. MEASure Subsystem Syntax Diagram

ALL

query

The ALL query makes a set of measurements on the displayed waveform using the selected source.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]ALL?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:ALL PERiod] <real number>;
[RISetime] <real number>;
[FALLtime] <real number>;
[FREQuency] <real number>;
[PWIDth] <real number>;
[NWIDth] <real number>;
[VPP] <real number>;
[VAMPlitude] <real number>;
[PREShoot] <real number>;
[OVERshoot] <real number><NL>

Example:

```
10 DIM All$[300]
20 OUTPUT XXX;":MEASURE:SOURCE CHANNEL1;ALL?"
30 ENTER XXX;All$
40 PRINT All$
50 END
```

FALLtime query

The FALLtime query makes a fall time measurement on the selected channel. The measurement is made between the 90% to the 10% voltage point of the first falling edge displayed on screen.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]FALLtime?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:FALLtime] <value><NL>

where:

<value> ::= time in seconds between the 90% and 10% voltage points of the first falling edge displayed on the screen

Example:

```
10 DIM Ft$[100]
20 OUTPUT XXX;":MEASURE:SOUR CHAN2;FALLTIME?"
30 ENTER XXX;Ft$
40 PRINT Ft$
50 END
```

FREQuency query

The FREQency query makes a frequency measurement on the selected channel. The measurement is made using the first complete displayed cycle at the 50% voltage level.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]FREQuency?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format:

[:MEASure:FREQuency]<value><NL>

<value> ::= frequency in Hertz

Example:

```
10 DIM Frcy$[100]
20 OUTPUT XXX;":MEASURE:SOUR CHAN1;FREQ?"
30 ENTER XXX;Frcy$
40 PRINT Frcy$
50 END
```

NWIDth query

The NWIDth query makes a negative width time measurement on the selected channel. The measurement is made between the 50% points of the first falling and the next rising edge displayed on screen.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]NWIDth?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:NWIDth] <value><NL>

<value> ::= negative pulse width in seconds

Example:

```
10 DIM Nw$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHAN2;NWID?"
30 ENTER XXX;Nw$
40 PRINT Nw$
50 END
```

OVERshoot query

The OVERshoot query makes an overshoot measurement on the selected channel. The measurement is made by finding a distortion following the first major transition. The result is the ratio of OVERshoot vs. VAMPlitude.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]OVERshoot?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:OVERshoot]<value><NL>

<value> ::= ratio of overshoot to Vamplitude

Example:

```
10 DIM Ovs$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHAN1;OVER?"
30 ENTER XXX;Ovs$
40 PRINT Ovs$
50 END
```

PERiod

query

The PERiod query makes a period measurement on the selected channel. The measurement is equivalent to the inverse of the frequency.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]PERiod?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:PERiod] <value><NL>

where:

<value> ::= waveform period in seconds

Example:

```
10 DIM Pd$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHANNEL1;PERIOD?"
30 ENTER XXX;Pd$
40 PRINT Pd$
50 END
```

PREShoot

query

The PREShoot query makes the preshoot measurement on the selected channel. The measurement is made by finding a distortion which precedes the first major transition on screen. The result is the ratio of PREshoot vs. VAMPlitude.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]PREShoot?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:PREShoot] <value><NL>

<value> ::= ratio of preshoot to Vamplitude

Example:

```
10 DIM Prs$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHANNEL2;PRES?"
30 ENTER XXX;Prs$
40 PRINT Prs$
50 END
```

PWIDth query

The PWIDth query makes a positive pulse width measurement on the selected channel. The measurement is made by finding the time difference between the 50% points of the first rising and the next falling edge displayed on screen.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]PWIDth?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:PWIDth] <value><NL>

where:

<value> ::= positive pulse width in seconds

Example:

```
10 DIM Pw$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHANNEL2;PWIDTH?"
30 ENTER XXX;Pw$
40 PRINT Pw$
50 END
```

RISetime

query

The RISetime query makes a risetime measurement on the selected channel by finding the 10% and 90% voltage levels of the first rising edge displayed on screen.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]RISetime?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:RISetime] <value><NL>

where:

<value> ::= risetime in seconds

Example:

```
10 DIM Tr$[100]
20 OUTPUT XXX;":MEASURE:SOUR CHAN1;RISETIME?"
30 ENTER XXX;Tr$
40 PRINT Tr$
50 END
```

SOURce command/query

The SOURce command specifies the source to be used for subsequent measurements. If the source is not specified, the last waveform source is assumed. The query returns the presently specified channel.

Command Syntax: :MEASure:SOURce CHAN<N>

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Example: OUTPUT XXX;":MEASURE:SOURCE CHAN1"

Query Syntax: :MEASure:SOURce?

Returned Format:

[:MEASure:SOURce] CHANnel<N><NL>

Example:

```
10 DIM So$[100]
20 OUTPUT XXX;":MEASURE:SOURCE?"
30 ENTER XXX;So$
40 PRINT So$
50 END
```

VAMPlitude

query

The VAMPlitude query makes a voltage measurement on the selected channel. The measurement is made by finding the relative maximum (VTOP) and minimum (VBASe) points on screen.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]VAMPlitude?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:VAMPlitude] <value><NL>

where:

<value> ::= difference between top and base voltage

Example:

```
10 DIM Va$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHANNEL2;VAMP?"
30 ENTER XXX;Va$
40 PRINT Va$
50 END
```

VBASe

query

The VBASe query returns the base voltage (relative minimum) of a displayed waveform. The measurement is made on the selected source.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]VBASe?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:VBASe] <value><NL>

<value> ::= voltage at base (relative minimum) of selected waveform

Example:

```
10 DIM Vb$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHAN1;VBAS?"
30 ENTER XXX;Vb$
40 PRINT Vb$
50 END
```

VMAX

query

The VMAX query returns the absolute maximum voltage of the selected source.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]VMAX?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:VMAX] <value><NL>

where:

<value> ::= maximum voltage of selected waveform

Example:

```
10 DIM Vma$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHAN2;VMAX?"
30 ENTER XXX;Vma$
40 PRINT Vma$
50 END
```

VMIN

query

The VMIN query returns the absolute minimum voltage present on the selected source.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]VMIN?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure VMIN] <value><NL>

where:

<value> ::= minimum voltage of selected waveform

Example:

```
10 DIM Vmi$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHAN1;VMIN?"
30 ENTER XXX;Vmi$
40 PRINT Vmi$
50 END
```

VPP query

The VPP query makes a peak to peak voltage measurement on the selected source. The measurement is made by finding the absolute maximum (VMAX) and minimum (VMIN) points on the displayed waveform.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]VPP?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:VPP]<value><NL>

where:

<value> ::= peak to peak voltage of selected waveform

Example:

```
10 DIM Vpp$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHAN1;VPP?"
30 ENTER XXX;Vpp$
40 PRINT Vpp$
50 END
```

VTOP

query

The VTOP query returns the voltage at the top (relative maximum) of the waveform on the selected source.

Query Syntax: :MEASure:[SOURce CHANnel<N>;]VTOP?

where:

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:MEASure:VTOP] <value><NL>

where:

<value> ::= voltage at the top (relative maximum) of the selected waveform

Example:

```
10 DIM Vt$[100]
20 OUTPUT XXX;":MEASURE:SOURCE CHAN2;VTOP?"
30 ENTER XXX;Vt$
40 PRINT Vt$
50 END
```

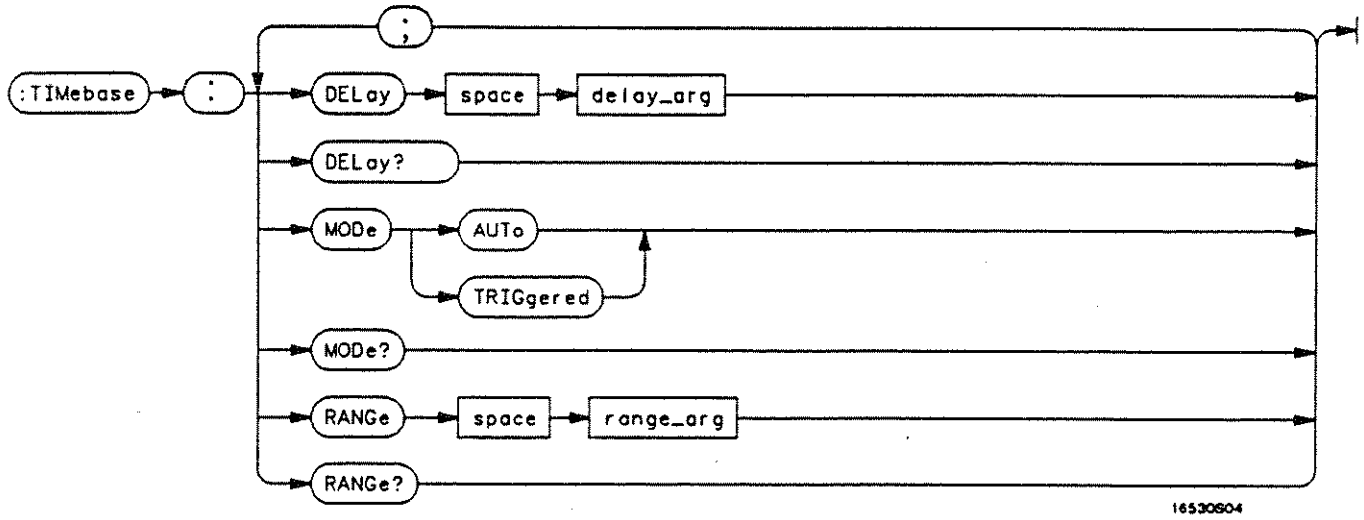
Notes

8

TIMebase Subsystem

Introduction

The commands of the Timebase Subsystem control the Timebase, Trigger Delay Time, and the Timebase Mode. If TRIGgered mode is to be used, ensure that the trigger specifications of the Trigger Subsystem have been set. Refer to Figure 8-1 for the TIMebase Subsystem Syntax Diagram.



delay_arg = *delay time in seconds, from -2500 seconds through +2500 seconds. The full range is available for panning the waveform when acquisition is stopped. Refer to the User's Reference Manual for a list of the available Delay Pre-trigger and Delay Post-trigger ranges while running and making acquisitions.*
range_arg = *a real number from 1 ns through 5 s*

Figure 8-1. TIMebase Subsystem Syntax Diagram

DELay command/query

The DELay command sets the time between the trigger and the center of the screen. The query returns the current delay setting.

Command Syntax: :TIMebase:DELay < delay time >

where:

< delay time > ::= delay time in seconds, from -2500 seconds through +2500 seconds. The full range is available for panning the waveform when acquisition is stopped. Refer to the *HP 16532A 1 GSa/s Oscilloscope User's Reference* manual for a list of the available Delay Pre-trigger and Delay Post-trigger ranges while running and making acquisitions.

Example:

```
OUTPUT XXX;":TIM:DEL 2US"
```

Query Syntax: :TIMebase:DELay?

Returned Format:

[:TIMebase DELay] < value > < NL >

Example:

```
10 DIM Dt$[100]
20 OUTPUT XXX;":TIM:DEL?"
30 ENTER XXX;Dt$
40 PRINT Dt$
50 END
```

MODE command/query

The MODE command sets the oscilloscope timebase to either Auto or Triggered mode. When the AUTO mode is chosen, the oscilloscope waits approximately 50 ms for a trigger to occur. If a trigger is not generated within that time, then auto trigger is executed. If a signal is not applied to the input, a baseline is displayed. If there is a signal at the input and the specified trigger conditions have not been met within 50 ms, the waveform display will not be synchronized to a trigger.

When the TRIGgered mode is chosen, the oscilloscope waits until a trigger is received before data is acquired. The TRIGgered mode should be used when the trigger source signal has less than a 20 Hz repetition rate, or when the trigger events counter is set so that the number of trigger events would not occur before 50 ms.

The Auto-Trig On field in the trigger menu is the same as the AUTO mode over HP-IB or RS-232-C. The TRIGgered command is the same as the Auto-Trig Off on the front panel. The query returns the current Timebase mode.

Command Syntax: :TIMebase:MODE {TRIGgered|AUTO}

Example:

```
OUTPUT XXX;":TIME:MODE AUTO"
```

Query Syntax: :TIMebase:MODE?

Returned Format:

[:TIMebase:MODE] {AUTO|TRIGGERED}<NL>

Example:

```
10 DIM Tm$[100]
20 OUTPUT XXX;":TIMebase:MODE?"
30 ENTER XXX;Tm$
40 PRINT Tm$
50 END
```

RANGe

command/query

The RANGE command sets the full-scale horizontal time in seconds. The RANGE value is ten times the value in the s/Div field.

Command syntax: :TIMebase:RANGe <range>

where:

<range> ::= time in seconds

Example: OUTPUT XXX;":TIMEBASE:RANGE 2US"

Query Syntax: :TIMebase:RANGe?

Returned Format:

[:TIMebase:RANGe] <range><NL>

Example:

```
10 DIM Tr$[100]
20 OUTPUT XXX;":TIMEBASE:RANGE?"
30 ENTER XXX;Tr$
40 PRINT Tr$
50 END
```

9

TRIGger Subsystem

Introduction

The commands of the Trigger Subsystem allow you to set all the trigger conditions necessary for generating a trigger. Many of the commands in the Trigger subsystem may be used in either the EDGE or the PATTern trigger mode. If a command is a valid command for the chosen trigger mode, then that setting will be accepted by the oscilloscope. However, if the command is not valid for the trigger mode, an error will be generated. None of the commands of this subsystem (except Mode) are used in conjunction with Immediate trigger mode. See Figure 9-1 for the TRIGger Subsystem Syntax Diagram.

The EDGE Trigger Mode

In the EDGE trigger mode, the oscilloscope triggers on an edge of a waveform, specified by the SOURce, DELay, LEVel, and SLOPe commands. If a source is not specified, then the current source is assumed. If EXTERNAL source is specified, the input signal is assumed to be ECL. The DELay value corresponds to the Count field displayed on the TRIGger menu.

The PATTern Trigger Mode

In the pattern trigger mode, the oscilloscope triggers when a pattern is generated using the CONDition, DELay, LEVel, LOGic and PATH commands. The CONDition command allows the oscilloscope to trigger when entering the specified pattern or exiting the pattern. The DELay value corresponds to the Count field displayed on the TRIGger menu. The LOGic command defines the pattern. The PATH command is used to change the trigger pattern and level. The path consists of two channels and the external trigger input.

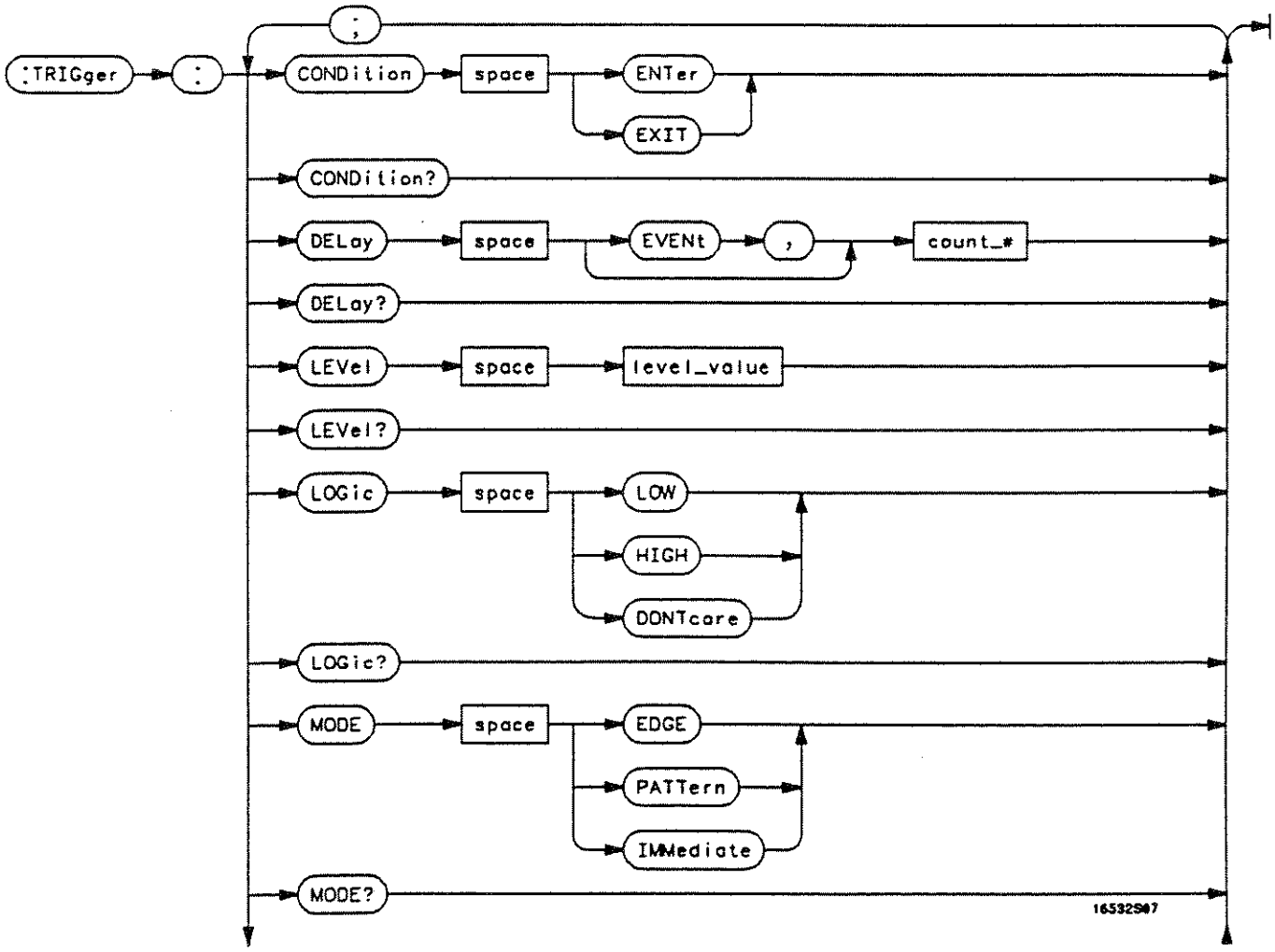
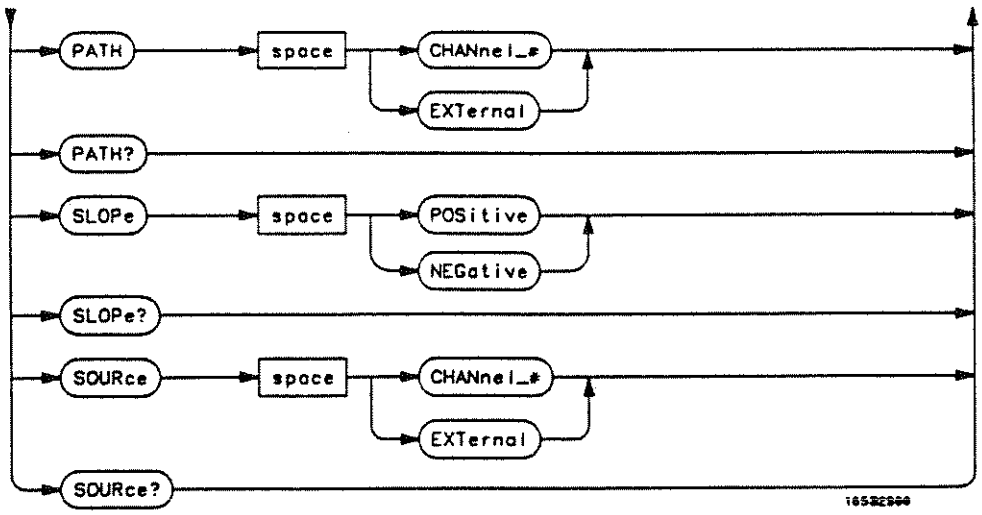


Figure 9-1. TRIGger Subsystem Syntax Diagram



count_# = *an integer from 1 through 32000*
level_value = *trigger level in volts*
channel_# = *an integer from 1 to 2*

Figure 9-1. TRIGger Subsystem Syntax Diagram (Cont'd)

CONDition command/query

The CONDition command specifies if a trigger is to be generated on entry (ENTer) to a specific logic pattern or when exiting (EXIT) the specified pattern. The specified pattern is defined by using the LOGic command.

When ENTer is chosen, the oscilloscope will trigger on the first transition that makes the pattern specification true for every input the number of times specified by the trigger event count (DELay command).

When EXIT is selected, the oscilloscope will trigger on the first transition that causes the pattern specification to be false after the pattern has been true for the number of times specified by the trigger event count (DELay command). The query returns the present condition.

Command Syntax: :TRIGger:[MODE PATTern;]CONDition {ENTer|EXIT}

Example:

```
OUTPUT XXX;":TRIG:COND ENT"
```

Query Syntax: :TRIGger:CONDition?

Returned Format:

[:TRIGger CONDition] {ENTer|EXIT}<NL>

Example:

```
10 DIM Ep$[100]
20 OUTPUT XXX;":TRIG:COND?"
30 ENTER XXX;Ep$
40 PRINT Ep$
50 END
```

DELay

command/query

The DELay command is used to specify the number of events at which trigger occurs. The time delay (see TIMe:DELay) is counted after the events delay. The DELay command cannot be used in the IMMediate trigger mode. The query returns the current trigger events count.

Command Syntax: :TRIGger:DELay [EVENt,] < count >

where:

< count > ::= integer from 1 to 32000

Example: OUTPUT XXX;":TRIGGER:DELAY 5"

Query Syntax: :TRIGger:DELay?

Returned Format:

[:TRIGger:DELay] < count > < NL >

Example:

```
10 DIM Td$[100]
20 OUTPUT XXX;":TRIG:DEL?"
30 ENTER XXX;Td$
40 PRINT Td$
50 END
```

LEVel command/query

The LEVel command sets the trigger level voltage for the selected source or path. This command cannot be used in the IMMediate trigger mode. In EDGE trigger mode, the SOURce command is used; in PATTern mode, the trigger PATH is used for the trigger level source. The LEVel command in PATTern trigger mode sets the high/low threshold for the pattern. The query returns the trigger level for the current path or source. LEVel for EXT cannot be specified, LEVel is fixed at ECL levels.

Command Syntax:

For EDGE trigger mode:

:TRIGger:[MODE EDGE;SOURce {CHANnel<N>|EXTernal};]
LEVel<value>

where:

<value>::= Trigger level in volts

For PATTern trigger mode:

:TRIGger:[MODE PATTern;PATH {CHANnel<N>|EXTernal};]
LEVel<value>

where:

<N>::= 1 or 2
<value>::= Trigger level in volts

Example: For EDGE trigger mode:

OUTPUT XXX;":TRIG:MODE EDGE;SOUR CHAN1;LEV 1.0"

Example: For PATTern trigger mode:

OUTPUT XXX;":TRIG:MODE PATTERN;PATH CHANNEL2;LEVEL 1.0"

LEVel

Query Syntax:

For EDGE trigger mode:

:TRIGger:[MODE EDGE;SOURce
{CHANnel<N>|EXTernal};]LEVel?

For PATTern trigger mode:

:TRIGger:[MODE PATTern;PATH
{CHANnel<N>|EXTernal};]LEVel?

Returned Format:

[:TRIGger:LEVel] <value><NL>

Example: For EDGE trigger mode

```
10 DIM El$[100]
20 OUTPUT XXX;":TRIGGER:SOURCE CHANNEL1;LEVEL?"
30 ENTER XXX;El$
40 PRINT El$
50 END
```

Example: For PATTern trigger mode

```
10 DIM Pl$[100]
20 OUTPUT XXX;":TRIGGER:PATH CHANNEL1;LEVEL?"
30 ENTER XXX;Pl$
40 PRINT Pl$
50 END
```

LOGic command/query

The LOGic command sets the logic for each trigger path in the PATTern trigger mode.The choices are HIGH, LOW and DONTcare. The trigger level set by the LEVel command determines logic high and low threshold levels. Any voltage higher than the present edge trigger level is considered a logic high for that trigger path; any voltage lower than the trigger level is considered a logic low for that trigger path. The query returns the current logic of the previously selected trigger or path.

Command Syntax: :TRIGger:[MODE PATTern;PATH {CHANnel<N>|EXTernal};] LOGic {HIGH|LOW|DONTcare}

where:

<N>::= 1 or 2

Example: OUTPUT XXX;":TRIG:PATH CHAN1;LOG HIGH"

Query Syntax: :TRIGger:LOGic?

Returned Format:

[:TRIGger:LOGic] {HIGH|LOW|DONTcare}<NL>

Example:

```
10 DIM Li$[100]
20 OUTPUT XXX;":TRIG:MODE PATT;PATH CHAN1;LOG?"
30 ENTER XXX;Li$
40 PRINT Li$
50 END
```

MODE

command/query

The MODE command allows you to select the trigger mode for the oscilloscope. The EDGE mode will trigger the oscilloscope on an edge whose slope is determined by the SLOPe command at a voltage set by the LEVel command. The PATTern mode will trigger the oscilloscope on entering or exiting a specified pattern of the two internal channels and external trigger. In the IMMediate trigger mode, the oscilloscope goes to a freerun mode and does not wait for a trigger. This mode is used in intermodule applications.

Command Syntax: :TRIGger:MODE {EDGE|PATTern|IMMediate}

Example:

```
OUTPUT XXX;":TRIGGER:MODE PATTERN"
```

Query Syntax: :TRIGger:MODE?

Returned Format:

[:TRIGger:MODE] {EDGE|PATTern|IMMediate}<NL>

Example:

```
10 DIM Md$[100]
20 OUTPUT XXX;":TRIGGER:MODE?"
30 ENTER XXX;Md$
40 PRINT Md$
50 END
```

PATH

command/query

The PATH command is used to select a trigger path for the subsequent LOGic and LEVel commands. This command can only be used in the PATTern trigger mode. The query returns the current trigger path.

Command Syntax: :TRIGger:[MODE PATTern;]PATH {CHANnel<N>|EXTernal}

where:

<N> ::= 1 or 2

Example: OUTPUT XXX;":TRIGGER:PATH EXTERNAL"

Query Syntax: :TRIGger:PATH?

Returned Format:

[:TRIGger PATH] {CHANnel<N>|EXTernal}<NL>

Example:

```
10 DIM Tp$[100]
20 OUTPUT XXX;":TRIGGER:PATH?"
30 ENTER XXX;Tp$
40 PRINT Tp$
50 END
```

SLOPe command/query

The SLOPe command selects the trigger slope for the specified trigger source. This command can only be used in the EDGE trigger mode. The query returns the slope of the current trigger source.

Command Syntax: :TRIGger:[MODE EDGE;SOURce {CHANnel<N>|EXTernal};]SLOPe {POSitive|NEGative}

where:

<N>::= 1 or 2

Example: OUTPUT XXX;":TRIG:SOUR CHAN1;SLOP POS"

Query Syntax: :TRIGger:SLOPe?

Returned Format:

[:TRIGger:SLOPe] {POSitive|NEGative}<NL>

Example:

```
10 DIM Ts$[100]
20 OUTPUT XXX;":TRIG:SOUR CHAN1;SLOP?"
30 ENTER XXX;Ts$
40 PRINT Ts$
50 END
```

SOURce

command/query

The SOURce command is used to select the trigger source and is used for any subsequent SLOPe and LEVel commands. This command can only be used in the EDGE trigger mode. It is the equivalent to the PATH command for the PATTern trigger mode. The query returns the current trigger source.

Command Syntax: :TRIGger:[MODE EDGE;]SOURce {CHANnel<N>|EXTernal}

where:

<N>::= 1 or 2

Example: OUTPUT XXX;":TRIG:SOUR CHAN1"

Query Syntax: :TRIGger:SOURce?

Returned Format:

[:TRIGger:SOURce] {CHANnel<N>|EXTernal}<NL>

Example:

```
10 DIM Tso$[100}
20 OUTPUT XXX;":TRIGGER:SOURCE?"
30 ENTER XXX;Tso$
40 PRINT Tso$
50 END
```

10

WAVeform Subsystem

Introduction

The commands of the Waveform subsystem are used to transfer waveform data from the oscilloscope to a controller. The waveform record is actually contained in two portions; the waveform data and preamble. The waveform data is the actual data acquired for each point when a DIGitize command is executed. The preamble contains the information for interpreting waveform data. Data in the preamble includes number of points acquired, format of acquired data, average count and the type of acquired data. The preamble also contains the X and Y increments, origins, and references for the acquired data for translation to time and voltage values.

The values set in the preamble are based on the settings of the variables in the Acquire, Waveform, Channel, and Timebase subsystems. The Acquire subsystem determines the acquisition type and the average count, the Waveform subsystem sets the number of points and format mode for sending waveform data over the remote interface and the Channel and Timebase subsystems set all the X - Y parameters.

Refer to Figure 10-3 for the Waveform Subsystem Syntax Diagram.

Data Acquisition Types

The two acquisition types that may be chosen are Normal or Average.

Normal Mode

In the Normal mode, with ACCumulate command OFF, the oscilloscope acquires waveform data and then displays the waveform. When the oscilloscope takes a new acquisition, the previously acquired waveform is erased from the display and replaced by the newly acquired waveform.

When the ACCumulate is set ON, the oscilloscope displays all the waveform acquisitions without erasing the previously acquired waveform.

Average Mode

In the Average mode, the oscilloscope averages the data points on the waveform with previously acquired data. Averaging helps eliminate random noise from the displayed waveform. In this mode ACCumulate is set to OFF. When Average mode is selected the number of averages must also be specified using the COUNt command. Previously displayed waveform data is erased from the display and the newly averaged waveform is displayed.

Format for Data Transfer

There are three formats for transferring waveform data over the remote interface. These formats are WORD, BYTE, or ASCII.

WORD and BYTE formatted waveform records are transmitted using the arbitrary block program data format specified in IEEE-488.2. When you use this format, the ASCII character string "# <8> <DD...D>" is sent before the actual data.

The <D>'s are ASCII numbers which indicate how many data bytes will follow.

For example, if 8192 points of data are to be transmitted, the ASCII string #800008192 would be sent.

BYTE Format

In BYTE format, the seven least significant bits represent the waveform data. This means that the possible range of data is divided into 128 vertical increments. The most significant bit is not used. If all "1"s are returned in the seven least significant bits, the waveform is clipped at the top of the screen. If all "0"s are returned, the waveform is clipped at the bottom of the screen (see figure 10-1).

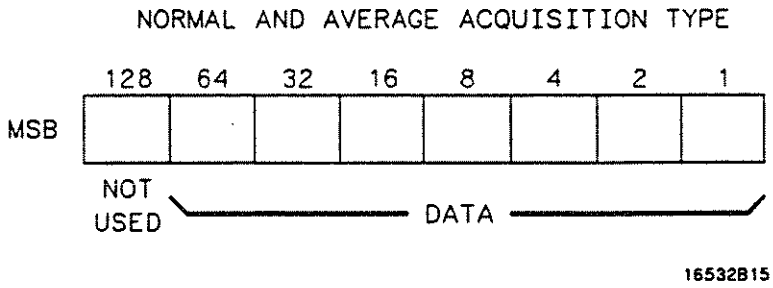


Figure 10-1. Byte Data Structure

The data returned in BYTE format is the same for either Normal or Average acquisition types. The data transfer rate in this format is faster than the other two formats.

WORD Format

Word data is two bytes wide with the most significant byte of each word being transmitted first. In WORD format, the 15 least significant bits represent the waveform data. The possible range of data is divided into 32768 vertical increments. The WORD data structure for normal and average acquisition types are shown in figure 10-2. If all "1's are returned in the 15 least significant bits, the waveform is clipped at the top of the screen.If all "0's are returned in the 15 least significant bits, the waveform is clipped at the bottom of the screen.

WORD (and ASCII) format data is more accurate than BYTE format data. BYTE format simply truncates the 8 least significant bits of WORD format data.

NORMAL AND AVERAGE ACQUISITION TYPE

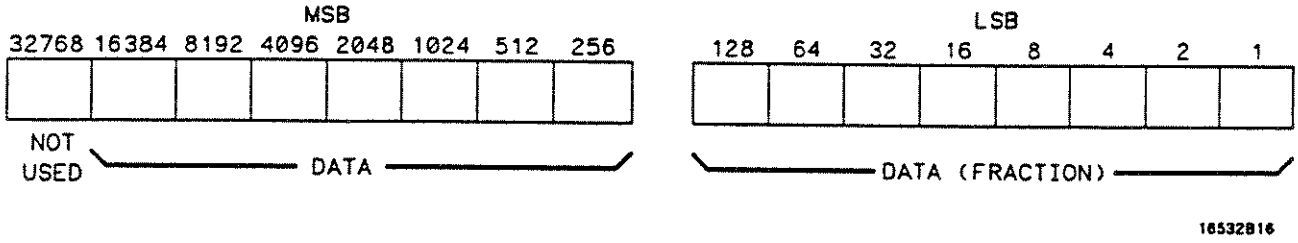


Figure 10-2. Word Data Structure

ASCII Format

ASCII formatted waveform records are transmitted one value at a time, separated by a comma. The data values transmitted are the same as would be sent in the WORD format except that they are converted to an integer ASCII format (six or less characters) before being transmitted. The header before the data is not included in this format.

Data Conversion

Data sent from the HP 16532A is raw data and must be scaled for useful interpretation. The values used to interpret the data are the X and Y references, X and Y origins, and X and Y increments. These values are read from the waveform preamble (see the PREamble command) or by the queries of these values.

Conversion from Data Value to Voltage

The formula to convert a data value returned by the instrument to a voltage is:

voltage = [(data value - yreference) * yincrement] + yorigin

Conversion from Data Value to Time

The time value of a data point can be determined by the position of the data point. As an example, the third data point sent with XORIGIN = 16ns, XREFERENCE = 0 and XINCREMENT = 2ns. Using the formula:

time = [(data point number - xreference) * xincrement] + xorigin

would result in the following calculation:

time = [(3 - 0) * 2ns] + 16ns = 22ns.

Conversion from Data Value to Trigger Point

The trigger data point can be determined by calculating the closest data point to time 0.

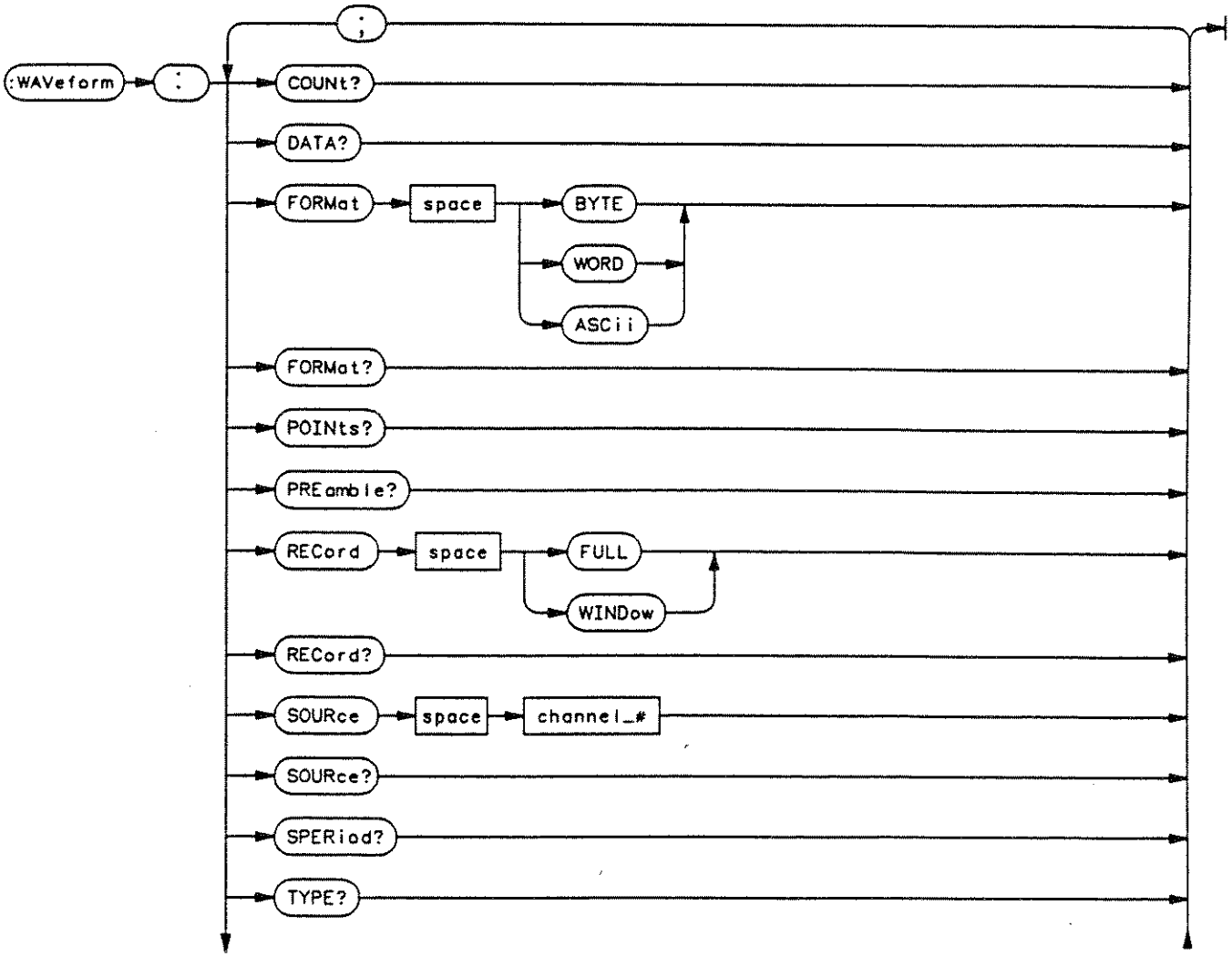
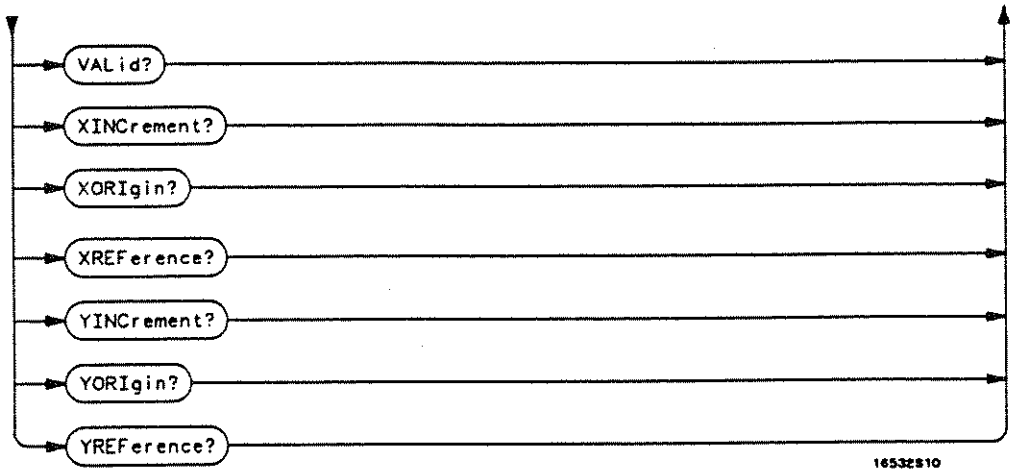


Figure 10-3. WAVeform Subsystem Syntax Diagram



channel_# = *an integer from 1 through 8 depending on how many oscilloscope cards are installed in the mainframe*

Figure 10-3. WAVeform Subsystem Syntax Diagram (Cont'd)

COUNt query

The COUNt query returns the count that was last specified in the ACQuire Subsystem.

Query Syntax: :WAVeform:COUNt?

Returned Format:

[:WAVeform:COUNt] <count><NL>

where:

<count> ::= {2|4|8|16|32|64|128|256}

Example:

```
10 DIM Ac$[100]
20 OUTPUT XXX;":WAVEFORM:COUNT?"
30 ENTER XXX;Ac$
40 PRINT Ac$
50 END
```

DATA query

The DATA query returns the waveform record stored in a specified channel buffer. The SOURce command of this subsystem has to be used to select the specified channel. The data is transferred based on the FORMAT (BYTE, WORD or ASCII) chosen and the RECORD specified (FULL or WINDOW). Since WAVeform:DATA is a query only, it cannot be used to send a waveform record back to the oscilloscope from the controller. If a waveform record is to be saved for later reloading into the oscilloscope, the SYSTem:DATA command should be used.

Query Syntax: :WAVeform:[SOURce CHANnel<N>;]DATA?

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format:

[:WAVeform:DATA]#800008000 <block data><NL>

The following example program moves data from the HP 16532A to a controller.

Example:

```
 10 CLEAR XXX
 20 OUTPUT XXX;":SELECT 4"
 30 OUTPUT XXX;":SYSTEM:HEADER OFF;:EOI ON"
 40 OUTPUT XXX;":ACQUIRE:TYPE AVERAGE"
 50 OUTPUT XXX;":WAVEFORM:SOURCE CH ANNEL1"
 60 OUTPUT XXX;":WAVEFORM:FORMAT WORD"
 70 OUTPUT XXX;":WAVEFORM:RECORD FULL"
 80 OUTPUT XXX;":AUTOSCALE"
 90 DIM Header$[20]
100 OUTPUT XXX;":DIGITIZE"
105 WAIT 10
110 Length = 8000
120 ALLOCATE INTEGER WAVEFORM(1:Length)
130 OUTPUT XXX;":WAVEFORM:DATA?"
140 ENTER XXX USING "#,10A";Header$
150 ENTER XXX USING "#,W";Waveform(*)
160 END
```

FORMat command/query

The FORMat command specifies the data transmission mode of waveform data over the remote interface. The query returns the currently specified format.

Command Syntax: :WAVeform:FORMat {BYTE|WORD|ASCii}

Example:

OUTPUT XXX;":WAV:FORM WORD"

Query Syntax: :WAVeform:FORMat?"

Returned Format:

[:WAVeform:FORMat]{BYTE|WORD|ASCii}<NL>

Example:

```
10 DIM Fo$[100]
20 OUTPUT XXX;":WAVEFORM:FORMAT?"
30 ENTER XXX;Fo$
40 PRINT Fo$
50 END
```

POINts query

When WAVeform RECord is set to FULL, the POINts query always returns a value of 8000 points. When WAVeform RECord is set to WINdow, then the query returns the number of points displayed on screen.

Query Syntax: :WAVeform:POINts?

Returned Format:

[:WAVeform:POINts] <points> <NL>

where:

<points> ::= number of points depending on the setting of the WAVeform RECord command

Example:

```
10 DIM Po$[100]
20 OUTPUT XXX;":WAVEFORM:POINTS?"
30 ENTER XXX;Po$
40 PRINT Po$
50 END
```

PREamble query

The PREamble query returns the preamble of the specified channel. The channel is specified using the SOURCE command.

Query Syntax: :WAVeform:SOURce CHANnel<N>;PREamble?

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:WAVeform:PREamble]

<format>, (0 = ASCII, 1 = BYTE, 2 = WORD,)
<type>, (1 = Normal, 2 = Average)
<points >,
<count >,
<Xincrement >,
<Xorigin >,
<Xreference >,
<Yincrement >,
<Yorigin >,
<Yreference ><NL>

Example:

```
10 DIM Pr$[300]
20 OUTPUT XXX;":WAVEFORM:PREAMBLE?"
30 ENTER XXX;Pr$
40 PRINT Pr$
50 END
```

For more information on the fields in PREamble, see the commands which query the individual fields. For example, see the FORmat command for an explanation of the format field.

RECord command/query

The RECord command specifies the data you want to receive over the bus. The choices are FULL or WINdow. When FULL is chosen, the entire 8000 point record of the specified channel is transmitted over the bus. In WINdow mode, only the data displayed on screen will be returned. The query returns the present mode chosen.

Command Syntax: :WAVeform:RECord {FULL|WINDow}

Example:

```
OUTPUT XXX;":WAV:SOUR CHAN1;REC FULL"
```

Query Syntax: :WAVeform:RECord?

Returned Format:

[:WAVeform:RECord] {FULL|WINDow}<NL>

Example:

```
10 DIM Wr$[100]
20 OUTPUT XXX;":WAVEFORM:RECORD?"
30 ENTER XXX;Wr$
40 PRINT Wr$
50 END
```

SOURce command/query

The SOURce command specifies the channel that is to be used for all subsequent waveform commands. The query returns the presently selected channel.

Command Syntax: :WAVeform:SOURce CHANnel<N>

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Example: OUTPUT XXX;":WAVEFORM:SOURCE CHANNEL1"

Query Syntax: :WAVeform:SOURce?

Returned Format:

[:WAVeform:SOURce] ChANnel<N><NL>

Example:

```
10 DIM Ws$[100]
20 OUTPUT XXX;":WAVEFORM:SOURCE?"
30 ENTER XXX;Ws$
40 PRINT Ws$
50 END
```

SPERiod query

The SPERiod query returns the present sampling period. The sample period is determined by the DELay and the RANGe commands of the TIMEbase subsystem.

Query Syntax: :WAVeform:SPERiod?

Returned Format:

[:WAVeform:SPERiod] <period> <NL>

where:

<period> ::= time in seconds

Example:

```
10 DIM Sp$[100]
20 OUTPUT XXX;":WAVEFORM:SPERIOD?"
30 ENTER XXX;Sp$
40 PRINT Sp$
50 END
```

TYPE

query

The TYPE query returns the presently acquisition type (normal or average). The acquisition type is specified in the ACQuire Subsystem using the ACQuire TYPE command.

Query Syntax :WAVeform:TYPE?

Returned Format:

[:WAVeform:TYPE]{NORMal|AVERage}<NL>

Example:

```
10 DIM Wt$[100]
20 OUTPUT XXX;":WAVEFORM:TYPE?"
30 ENTER XXX;Wt$
40 PRINT Wt$
50 END
```

VALid query

The VALid query checks the oscilloscope for acquired data. If a measurement is completed, and data has been acquired by all channels, then the query reports a 1. A 0 is reported if no data has been acquired for the last acquisition.

Query Syntax: :WAVeform:VALid?

Returned Format:

[:WAVeform:VALid] {0|1}<NL>

0 ::= No data acquired
1 ::= Data has been acquired

Example:

```
10 DIM Da$[100]
20 OUTPUT XXX;":WAVEFORM:VALID?"
30 ENTER XXX;Da$
40 PRINT Da$
50 END
```

XINCrement query

The XINCrement query returns the X-increment currently in the preamble. This value is the time difference between the consecutive data points.

Query Syntax: :WAVeform:XINCrement?

Returned Format:

[:WAVeform:XINCrement] <value> <NL>

<value> ::= X-increment value currently in preamble

Example:

```
10 DIM Xi$[100]
20 OUTPUT XXX;":WAVEFORM:XINCREMENT?"
30 ENTER XXX;Xi$
40 PRINT Xi$
50 END
```

XORigin

XORigin query

The XORigin query returns the X-origin value currently in the preamble. The value represents the time of the first data point in memory with respect to the trigger point.

Query Syntax: :WAVeform:[SOURce CHANnel<N>;]XORigin?

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format: [:WAVeform:XORigin]<value><NL>

<value> ::= X-origin currently in preamble

Example:

```
10 DIM Xo$[100]
20 OUTPUT XXX;":WAV:XOR?"
30 ENTER XXX;Xo$
40 PRINT Xo$
50 END
```

XREFerence query

The XREFerence query returns the current X- reference value in the preamble. This value specifies the X-value of the first data point in memory and is always 0.

Query Syntax: :WAVeform:XREFerence?

Returned Format:

[:WAVeform:XREFerence] <value> <NL>

<value> ::= X-reference value in the preamble

Example:

```
10 DIM Xr$[100]
20 OUTPUT XXX;":WAVEFORM:XREFERENCE?"
30 ENTER XXX;Xr$
40 PRINT Xr$
50 END
```

YINCrement

YINCrement query

The YINCrement query returns the Y-increment value currently in the preamble. This value is the voltage difference between consecutive data values.

Query Syntax: :WAVeform:[SOURce CHANnel<N>;]YINCrement?

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format:

[:WAVeform:YINCrement]<value><NL>

<value> ::= Y-increment value in preamble

Example:

```
10 DIM Yi$[100]
20 OUTPUT XXX;":WAVEFORM:YINCREMENT?"
30 ENTER XXX;Yi$
40 PRINT Yi$
50 END
```

YORigin query

The YORigin query returns the Y-origin value currently in the preamble. This value is the voltage at center screen.

Query Syntax: :WAVeform:[SOURce CHANnel<N>;]YORigin?

<N> ::= 1 through the number of channels in the oscilloscope connected as one unit (maximum of 8 channels), otherwise the maximum number of channels is 2

Returned Format:

[:WAVeform:YORigin]<value><NL>

<value> ::= Y-origin value in preamble

Example:

```
10 DIM Yo$[100]
20 OUTPUT XXX;":WAVEFORM:YORIGIN?"
30 ENTER XXX;Yo$
40 PRINT Yo$
50 END
```

YREFerence query

The YREFerence query returns the Y-reference value currently in the preamble. This value specifies the data value at center screen where Y-origin occurs.

Query Syntax: :WAVeform:YREFerence?

Returned Format:

[:WAVeform:YREFerence] <value> <NL>

<value> ::= Y-reference data value in preamble

Example:

```
10 DIM Yr$[100]
20 OUTPUT XXX;":WAVEFORM:YREFERENCE?"
30 ENTER XXX;Yr$
40 PRINT Yr$
50 END
```

Index

Q

R

S

T

V

W

X

Y

HP PART NUMBER: 16532-90903

Print Date: June 91

Printed in U.S.A.