



# Programming Guide

**T3DSO1000(A) and T3DSO2000  
Digital Oscilloscopes Programming  
Guide**

## Version Declaration

This chapter indicates the modifications of commands in the most recent release of the programming guide version.

## Introduction

Manual version 1.3 describes all the currently available commands. Some of the commands vary between the oscilloscope series, and these will be annotated in the description of command.

The following are the main revisions:

- ⑩ Delete the **Table of Commands & Queries**, and all the instructions are classified according to the functional modules.
- ⑩ Removed incorrect instructions, added instructions for GEN and DIGITAL modules.
- ⑩ Add two new communication features: Telnet and Socket, visible in "Programming Overview-Remote Control".
- ⑩ Detailed programming instances for instructions (WF?/SCDP) to make it easier to understand.
- ⑩ Support obtaining waveform data of Digital channel and Math.

## Content

Programming Overview .....	6
Establishing Communications .....	6
Install NI-VISA .....	6
Connect the Instrument .....	10
Remote Control .....	11
User-defined Programming .....	11
Send SCPI Commands via NI-MAX .....	11
Using SCPI with Telnet .....	11
Using SCPI with Sockets.....	13
Introduction to the SCPI Language .....	14
About Commands & Queries.....	14
Description .....	14
Usage.....	14
Command Notation .....	15
Commands & Queries .....	16
COMMON (*) Commands .....	17
COMM_HEADER Commands .....	21
ACQUIRE Commands .....	23
AUTOSET Commands .....	37
CHANNEL Commands .....	39
CURSOR Commands .....	49
DIGITAL Commands.....	57
DISPLAY Commands .....	67
HISTORY Commands .....	73

**Digital Oscilloscope Series**

---

MATH Commands .....	78
MEASURE Commands .....	95
PASS/FAIL Commands .....	109
PRINT Commands .....	120
RECALL Commands .....	122
REFERENCE Commands.....	126
SAVE Commands .....	135
STATUS Commands .....	142
SYSTEM Commands.....	145
TIMEBASE Commands.....	150
TRIGGER Commands .....	158
WAVEFORM Commands .....	175
WGEN Commands.....	189
Programming Examples .....	196
VISA Examples .....	197
VC++ Example.....	197
VB Example.....	204
MATLAB Example.....	210
LabVIEW Example .....	212
C# Example .....	215
Examples of Using Sockets .....	218
Python Example .....	219
C Example.....	221
Common Command Examples.....	223
Read waveform data (WF) Example .....	223

**Digital Oscilloscope Series**

---

Screen Dump (SCDP) Example..... 225  
Index..... 226

## Programming Overview

This chapter introduces how to build communication between the instrument and the PC. It also introduces how to configure a system for remote instrument control. By using USB and LAN interfaces, in combination with NI-VISA and programming languages, users can remotely control the instruments. Through LAN interface, VXI-11, Sockets and Telnet protocols can be used to communicate with the instruments.

### Establishing Communications

#### Install NI-VISA

Before programming, you need to install the National Instruments NI-VISA library, which you can download from the National Instruments web site. Currently, NI-VISA is packaged in two versions: a full version and a Run-Time Engine version. The full version includes the NI device drivers and a tool named NI MAX which is a user interface to control and test remotely connected devices. The Run-Time Engine is recommended, as it is a much smaller download than the full version and includes the necessary tools for basic communication to instruments.

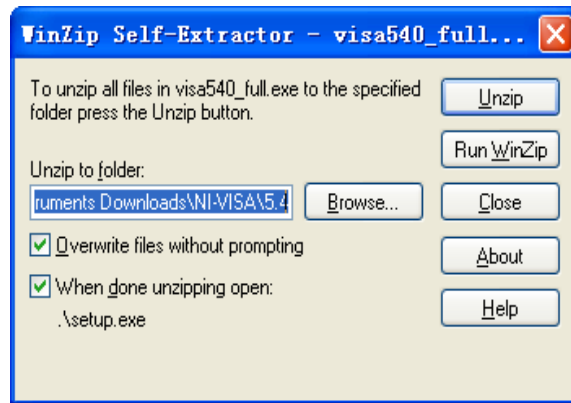
For example, you can get the NI-VISA 5.4 full version from:  
<http://www.ni.com/download/ni-visa-5.4/4230/en/>.

You also can download NI-VISA Run-Time Engine 5.4 to your PC and install it as the default selection. Its installation process is similar with the full version.

After you downloaded the file, follow these steps to install NI-VISA (The full version of NI-VISA 5.4 is used in this example. Newer versions are likely, and should be compatible with Teledyne Test Tools instrumentation. Download the latest version available for the operating system being used by the controlling computer):

- a. Double click the visa540\_full.exe, dialog shown as below:

## Digital Oscilloscope Series

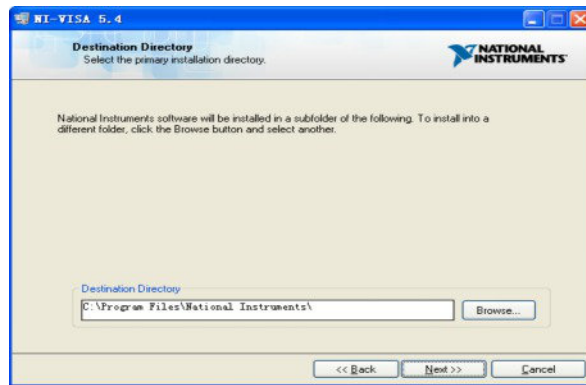


- b. Click Unzip, the installation process will automatically launch after unzipping files. If your computer needs to install .NET Framework 4, it may auto start.

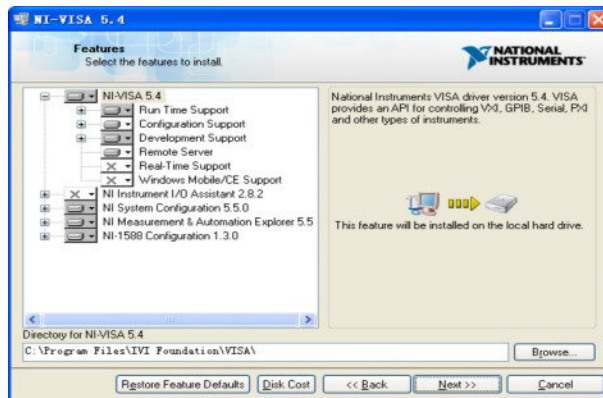


- c. The NI-VISA installing dialog is shown above. Click Next to start the installation process.

## Digital Oscilloscope Series



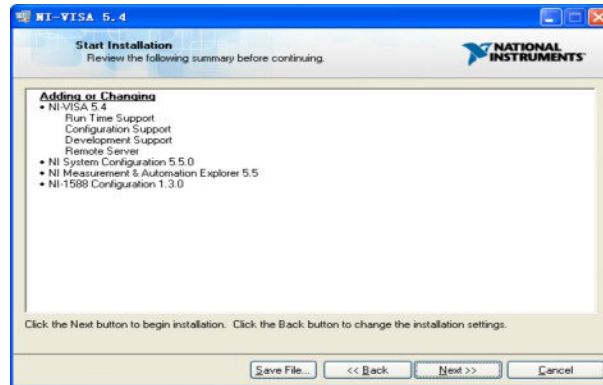
- d. Set the install path, default path is “C:\Program Files\National Instruments\”, you can change it. Click Next, dialog shown as above.



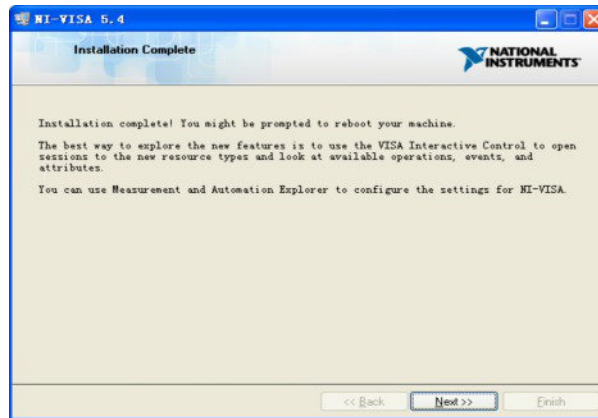
- e. Click Next twice, in the License Agreement dialog, select the “I accept the above 2 License Agreement(s).”, and click Next, dialog shown as below:



## Digital Oscilloscope Series



f. Click Next to begin installation.



g. Now the installation is complete. Reboot your PC.

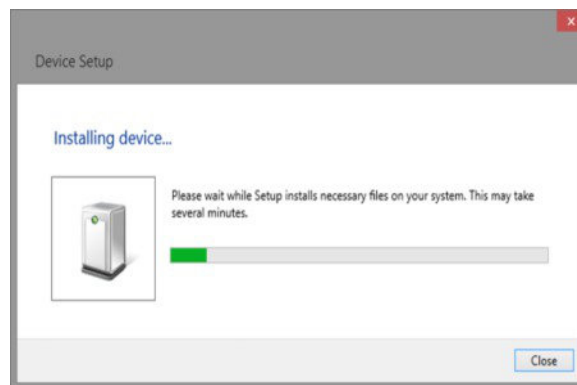
---

## Digital Oscilloscope Series

### Connect the Instrument

Depending on the specific model, your oscilloscope may be able to communicate with a PC through the USB or LAN interface.

Connect the instrument and the USB Host interface of the PC using a USB cable. Assuming your PC is already turned on, turn on your oscilloscope, and then the PC will display the “Device Setup” screen as it automatically installs the device driver as shown below.



Wait for the installation to complete and then proceed to the next step.

## Remote Control

### User-defined Programming

Users can use SCPI commands via a computer to program and control the digital oscilloscope. For details, refer to the introductions in "Programming Examples".

### Send SCPI Commands via NI-MAX

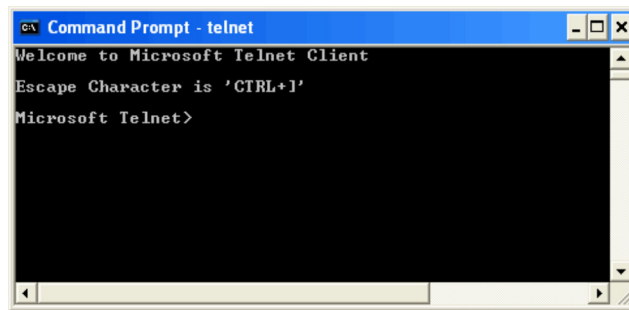
NI-Measurement and Automation eXplorer (NI-MAX) is a program created and maintained by National Instruments. It provides a basic remote control interface for VXI, LAN, USB, GPIB, and Serial communications. It is a utility that enables you to send commands one-at-a-time and also retrieve data from connected devices. It is a great tool for troubleshooting and testing command sequences. The oscilloscopes can be controlled remotely by sending SCPI commands via NI-MAX.

### Using SCPI with Telnet

Telnet provides a means of communicating with the oscilloscopes over a LAN connection. The Telnet protocol sends SCPI commands to the oscilloscopes from a PC and is similar to communicating with the oscilloscopes over USB. It sends and receives information interactively: one command at a time. Windows operating systems use a command prompt style interface for the Telnet client. The steps are as follows:

1. On your PC, click Start > All Programs > Accessories > Command Prompt.
2. At the command prompt, type in *telnet*.
3. Press the Enter key. The Telnet display screen will be displayed.

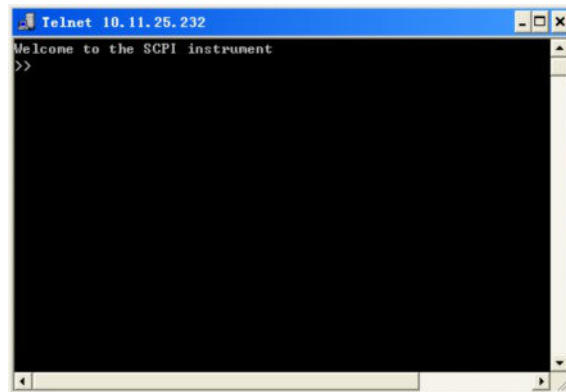
## Digital Oscilloscope Series



4. At the Telnet command line, type:

*open XXX.XXX.XXX.XXX 5024*

Where *XXX.XXX.XXX.XXX* is the instrument's IP address and 5024 is the port. You should see a response similar to the following:



5. At the SCPI> prompt, input the SCPI commands such as *\*IDN?* to return the company name, model number, serial number, and firmware version number.

6. To exit the SCPI> session, press the Ctrl+] keys simultaneously.

7. Type *quit* at the prompt or close the Telnet window to close the connection to the instrument and exit Telnet.

### **Using SCPI with Sockets**

Socket API can be used to control the T3DSO series via LAN without installing any other libraries. This can reduce the complexity of programming.

<b>SOCKET ADDRESS</b>	IP address + port number
<b>IP ADDRESS</b>	T3DSO IP address
<b>PORT NUMBER</b>	5024

Please see section "[Examples of Using Sockets](#)" for the details.

## Introduction to the SCPI Language

### About Commands & Queries

This section lists and describes the remote control commands and queries recognized by the instrument. All commands and queries can be executed in either local or remote state.

The description for each command or query, with syntax and other information, begins on a new page. The name (header) is given in both long and short form at the top of the page, and the subject is indicated as a command or query or both.

The commands are given in long format for the “**COMMAND SYNTAX**” and “**QUERY SYNTAX**” sections and they are used in a short form for the “**EXAMPLE**”.

Queries perform actions such as obtaining information, and are recognized by the question mark (?) following the header.

### Description

In the description, a brief explanation of the function performed is given. This is followed by a presentation of the formal syntax, with the header given in upper case characters and the short form derived from it. Where applicable, the syntax of the query is given with the format of its response.

### Usage

The commands and queries listed here can be used for the T3DSO1000(A) and T3DSO2000 Digital Oscilloscope Series.

## Command Notation

The following notations are used in the commands:

- <> Angular brackets enclose words that are used as placeholders, of which there are two types: the header path and the data parameter of a command.
- := A colon followed by an equals sign separates a placeholder from the description of the type and range of values that may be used in a command instead of the placeholder.
- { } Braces enclose a list of choices, one of which one must be made.
- [ ] Square brackets enclose optional items.
- ... An ellipsis indicates that the items both to its left and right may be repeated for a number of times.

As an example, consider the syntax notation for the command to set the vertical input sensitivity:

```
<channel>:VOLT_DIV <v_gain>  
<channel>:={C1,C2,C3,C4}  
<v_gain>:= 2 mV to 10 V
```

The first line shows the formal appearance of the command, with <channel> denoting the placeholder for the header path and <v\_gain> the placeholder for the data parameter specifying the desired vertical gain value. The second line indicates that one of four channels must be chosen for the header path. And the third explains that the actual vertical gain can be set to any value between 2mV and 10V.

## Commands & Queries

This chapter introduces each command subsystem of the Teledyne Test Tools Digital Oscilloscope Series command set. The contents of this chapter are shown as below:

- COMMON (\*) Commands
- COMM\_HEADER Commands
- ACQUIRE Commands
- AUTOSET Commands
- CHANNEL Commands
- CURSOR Commands
- DIGITAL Commands
- DISPLAY Commands
- HISTORY Commands
- MATH Commands
- MEASURE Commands
- PASS/FAIL Commands
- PRINT Commands
- RECALL Commands
- REFERENCE Commands
- SAVE Commands
- STATUS Commands
- SYSTEM Commands
- TIMEBASE Commands
- TRIGGER Commands
- WAVEFORM Commands
- WGEN Commands
- Obsolete Commands for Old Models



### **COMMON (\*) Commands**

The IEEE 488.2 standard defines some general commands for querying the basic information of an instrument or performing common basic operations. These commands usually start with \*, and the command key length is 3 characters.

\***IDN?** (Identification Number)

\***OPC** (Operation Complete)

\***RST** (Reset)

**COMMON (\*)**

**\*IDN?**

Query

**DESCRIPTION**

The \*IDN? query identifies the instrument type and software version. The response consists of four different fields providing information on the manufacturer, the scope model, the serial number and the firmware revision.

**QUERY SYNTAX**

\*IDN?

**RESPONSE FORMAT**

Teledyne Test Tools,<model>,<serial number>,<firmware>  
<model>:= the model number of the instrument.  
<serial number>:= A 14-digit decimal code.  
<firmware>:= the software revision of the instrument

**EXAMPLE**

The query identifies the instrument type and software version.

Command message:

*\*IDN?*

Response message:

*Teledyne Test Tools, T3DSO1204  
,NDS1EBAC0L0098,7.6.1.15*

---

## Digital Oscilloscope Series

### **COMMON (\*)**

### **\*OPC** Command/Query

#### **DESCRIPTION**

The \*OPC command sets the operation complete bit in the Standard Event Status Register when all pending device operations have finished.

The \*OPC? query places an ASCII "1" in the output queue when all pending device operations have completed. The interface hangs until this query returns.

#### **COMMAND SYNTAX**

\*OPC

#### **QUERY SYNTAX**

\*OPC?

#### **RESPONSE FORMAT**

\*OPC 1

---

## Digital Oscilloscope Series

### **COMMON (\*)**

### **\*RST** Command

#### **DESCRIPTION**

The \*RST command initiates a device reset. This is the same as pressing **[Default]** on the front panel.

#### **COMMAND SYNTAX**

\*RST

#### **EXAMPLE**

This example resets the oscilloscope.  
Command message:  
*\*RST*

## COMM\_HEADER Commands

### CHDR

**COMM\_HEADER**

**COMM\_HEADER | CHDR**

Command/ Query

**DESCRIPTION**

The COMM\_HEADER command controls the way the oscilloscope formats response to queries. This command does not affect the interpretation of messages sent to the oscilloscope. Headers can be sent in their long or short form regardless of the CHDR setting.

Examples of the three response formats to "C1:VDIV?":

CHDR	RESPONSE
LONG	C1:VOLT DIV 1.00E+01V
SHORT	C1:VDIV 1.00E+01V
OFF	1.00E+01

**COMMAND SYNTAX**

COMM\_HEADER <mode>

<mode>:={SHORT, LONG, OFF}

- SHORT — response starts with the short form of the header word.
- LONG — response starts with the long form of the header word.
- OFF — header is omitted from the response and units in numbers are suppressed.

**Note:**

Default is the SHORT response format.

**QUERY SYNTAX**

COMM\_HEADER?

**RESPONSE FORMAT**

COMM\_HEADER <mode>

**EXAMPLE**

The following command sets the response header format to SHORT.

Command message:

*CHDR SHORT*

## ACQUIRE Commands

The ACQUIRE subsystem controls the way in which waveforms are acquired. These commands set the parameters for acquiring and storing data.

**ARM**

**STOP**

**ACQW**

**AVGA**

**MSIZ**

**SAST?**

**SARA?**

**SANU?**

**SXSA**

**XYDS**

**ACQUIRE**

**ARM\_ACQUISITION | ARM**  
Command

**DESCRIPTION**

The ARM\_ACQUISITION command starts a new signal acquisition.

**COMMAND SYNTAX**

ARM\_ACQUISITION

**EXAMPLE**

The following steps show the effect of ARM.

**Note:**

INR bit 13 (8192) = Trigger is ready.  
INR bit 0 (1) = New Signal Acquired.

**Step 1:** Set the trigger mode to single, and input a signal which can be triggered. Once triggered, you can see the state of acquisition changes to stop. Send the query.

Query message:  
*INR?*

Response message:  
*INR 8193(trigger ready)*

**Step 2:** Send the query again to clear the register.

Query message:  
*INR?*

Response message:  
*INR 0*

**Step 3:** Now, send the command to start a new signal acquisition.

Command message:  
*ARM*

**Step 4:** Send the query to see the effect of ARM.

Query message:  
*INR?*



---

## Digital Oscilloscope Series

Response message:  
*INR 8193*

### RELATED COMMANDS

STOP  
TRMD  
INR?

**ACQUIRE**

**STOP**

Command

**DESCRIPTION**

The STOP command stops the acquisition. This is the same as pressing the Stop key on the front panel.

**COMMAND SYNTAX**

STOP

**EXAMPLE**

The following command stops the acquisition process.

Command message:

*STOP*

**RELATED COMMANDS**

ARM  
TRMD

**ACQUIRE**

**ACQUIRE\_WAY | ACQW**

Command /Query

**DESCRIPTION**

The ACQUIRE\_WAY command specifies the acquisition mode.

The ACQUIRE\_WAY? query returns the current acquisition mode.

**COMMAND SYNTAX**

ACQUIRE\_WAY <mode>[,<time>]

<mode>:={SAMPLING,PEAK\_DETECT,AVERAGE,HIGH\_RES}

<time>:={4,16,32,64,128,256,512,...}

- SAMPLING — sets the oscilloscope in the normal mode.
- PEAK\_DETECT — sets the oscilloscope in the peak detect mode.
- AVERAGE — sets the oscilloscope in the averaging mode.
- HIGH\_RES — sets the oscilloscope in the enhanced resolution mode (also known as smoothing). This is essentially a digital boxcar filter and is used to reduce noise at slower sweep speeds.

**Note:**

- The [HIGH\_RES] option is valid for T3DSO models. See models on [page 14](#).
- <time>:={4,16,32,64,128,256,512,...} when <mode> = AVERAGE.

Options vary from models. See the data sheet or the acquire menu of the oscilloscope.

**QUERY SYNTAX**

ACQUIRE\_WAY?

**RESPONSE FORMAT**

ACQUIRE\_WAY <mode>[,<time>]

## Digital Oscilloscope Series

---

### EXAMPLE

The following command sets the acquisition mode to average mode and also sets the average time to 16.

Command message:  
*ACQW AVERAGE,16*

### RELATED COMMANDS

AVGA

**ACQUIRE**

**AVERAGE\_ACQUIRE | AVGA**

Command /Query

**DESCRIPTION**

The AVERAGE\_ACQUIRE command selects the average times of average acquisition.

The AVERAGE\_ACQUIRE? query returns the currently selected count value for average mode.

**COMMAND SYNTAX**

AVERAGE\_ACQUIRE <time>

<time>:= {4,16,32,64,128,256,...}

**Note:**

Options of <time> vary from models. See the data sheet or the acquire menu of the oscilloscope for details.

**QUERY SYNTAX**

AVERAGE\_ACQUIRE?

**RESPONSE FORMAT**

AVERAGE\_ACQUIRE <time>

**EXAMPLE**

The following command turns the average times of average acquisition to 16.

Command message:

*AVGA 16*

**RELATED COMMANDS**

ACQW

**ACQUIRE**

**MEMORY\_SIZE | MSIZ**

Command /Query

**DESCRIPTION**

The MEMORY\_SIZE command sets the maximum depth of memory.

The MEMORY\_SIZE? query returns the maximum depth of memory.

**COMMAND SYNTAX**

MEMORY\_SIZE <size>

<size>:={7K,70K,700K,7M} for non-interleaved mode. Non-interleaved means a single channel is active per A/D converter. Most oscilloscopes feature two channels per A/D converter. .

<size>:={14K,140K,1.4M,14M} for interleave mode. Interleave mode means multiple active channels per A/D converter.

**Note:**

Options of <size> vary from models. See the data sheet or the acquire menu of the oscilloscope for details.

**QUERY SYNTAX**

MEMORY\_SIZE?

**RESPONSE FORMAT**

MEMORY\_SIZE <size>

**EXAMPLE**

The following command sets the maximum depth of memory to 14M in interleave mode.

Command message:

*MSIZ 14M*

---

## Digital Oscilloscope Series

### ***ACQUIRE***

### **SAMPLE\_STATUS? | SAST?**

Query

#### **DESCRIPTION**

The SAST? query returns the acquisition status of the scope.

#### **QUERY SYNTAX**

SAST?

#### **RESPONSE FORMAT**

SAST <status>

#### **EXAMPLE**

The following query returns the acquisition status of the scope.

Query message:

*SAST?*

Response message:

*SAST Trig'd*

---

## Digital Oscilloscope Series

---

### **ACQUIRE**

### **SAMPLE\_RATE? | SARA?**

Query

### **DESCRIPTION**

The SARA? query returns the sample rate of the scope.

### **QUERY SYNTAX**

SARA?  
DI:SARA?  
• DI — digital.

### **RESPONSE FORMAT**

SARA <value>  
DI:SARA <value>

Model	Format of <value>
T3DSO1000(A)	Numerical value in E-notation with SI unit, such as 5.00E+08Sa/s.
T3DSO2000	Numerical value with measurement unit and physical unit, such as 1.00GSa/s.

### **EXAMPLE**

- The following query returns the sample rate of the analog channel.

Query message:  
*SARA?*

Response message:  
*SARA 5.00E+05Sa/s*

- The following query returns the sample rate of the digital channel.

Query message:  
*DI:SARA?*

Response message:  
*DI:SARA 5.00E+05Sa/s*



## Digital Oscilloscope Series

---

**Note:**

The table shows the availability of “DI:SARA?” in each digital oscilloscope series.

Model	Valid?
T3DSO2000	yes
T3DSO1000(A)	yes

**ACQUIRE**

**SAMPLE\_NUM? | SANU?**

Query

**DESCRIPTION**

The SANU? query returns the number of data points that the hardware will acquire from the input signal. The number of points acquired is based on the horizontal scale and memory/acquisition depth selections and cannot be directly set.

**QUERY SYNTAX**

SANU? <channel>

<channel>:={C1,C2,C3,C4}

**RESPONSE FORMAT**

SANU <value>

Model	Format of <value>
T3DSO1000(A)	Numerical value in E-notation with SI unit, such as 7.00E+05pts.
T3DSO2000	Numerical value with measurement unit and physical unit, such as 28Mpts.

**EXAMPLE**

The following query returns the number of sampled points available from last acquisition from Channel 2.

Query message:  
*SANU? C2*

Response message:  
*SANU 7.00E+05pts*

**ACQUIRE**

**SINXX\_SAMPLE | SXSA**

Command/Query

**DESCRIPTION**

The SINXX\_SAMPLE command sets the way of interpolation.

The SINXX\_SAMPLE? query returns the way of interpolation.

**COMMAND SYNTAX**

SINXX\_SAMPLE <state>

<state>:= {ON,OFF}

- ON — sine interpolation.
- OFF — linear interpolation.

**QUERY SYNTAX**

SINXX\_SAMPLE?

**RESPONSE FORMAT**

SINXX\_SAMPLE <state>

**EXAMPLE**

The following command sets the way of the interpolation to sine interpolation. Command message:  
*SXS4 ON*

**ACQUIRE**

**XY\_DISPLAY | XYDS**

Command / Query

**DESCRIPTION**

The XY\_DISPLAY command enables or disables the display of XY mode. XY mode plots the voltage data of both channels with respect to one-another. For example, channel 1 vs. channel 2. This can be used to create Lissajous curves. The standard display mode plots voltage data vs. time.

The XY\_DISPLAY? query returns whether the XY format display is enabled.

**COMMAND SYNTAX**

XY\_DISPLAY <state>

<state>:= {ON,OFF}

**QUERY SYNTAX**

XY\_DISPLAY?

**RESPONSE FORMAT**

XY\_DISPLAY <state>

**EXAMPLE**

The following command enables the XY format.

Command message:

*XYDS ON*

### **AUTOSET Commands**

The AUTOSET subsystem commands control the function of automatic waveform setting. The oscilloscope will automatically adjust the vertical position, the horizontal time base and the trigger mode according to the input signal to make the waveform display to the best state.

#### **ASET**

***AUTOSET***

Command

**AUTO\_SETUP | ASET**

**DESCRIPTION**

The AUTO\_SETUP command attempts to identify the waveform type and automatically adjusts controls to produce a usable display of the input signal.

**COMMAND SYNTAX**

AUTO\_SETUP

**EXAMPLE**

The following command instructs the oscilloscope to perform an auto-setup.  
Command message:  
*ASET*

## CHANNEL Commands

The CHANNEL subsystem commands control the analog channels. Channels are independently programmable for offset, probe, coupling, bandwidth limit, inversion, and more functions. The channel index (1, 2, 3, or 4) specified in the command selects the analog channel that is affected by the command.

**ATTN**  
**BWL**  
**CPL**  
**OFST**  
**SKEW**  
**TRA**  
**UNIT**  
**VDIV**  
**INVS**

**CHANNEL**

Command /Query

**ATTENUATION | ATTN**

**DESCRIPTION**

The ATTENUATION command specifies the probe attenuation factor for the selected channel. The probe attenuation factor may be 0.1 to 10000. This command does not change the actual input sensitivity of the oscilloscope. It changes the reference constants for scaling the display factors, for making automatic measurements, and for setting trigger levels.

The ATTENUATION? query returns the current probe attenuation factor for the selected channel.

**COMMAND SYNTAX**

<channel>:ATTENUATION <attenuation>

<channel>:={C1,C2,C3,C4}

<attenuation>:={0.1,0.2,0.5,1,2,5,10,20,50,100,200,500,1000,2000,5000,10000}

**QUERY SYNTAX**

<channel>:ATTENUATION?

**RESPONSE FORMAT**

<channel>:ATTENUATION <attenuation>

**EXAMPLE**

The following command sets the attenuation factor of Channel 1 to 100:1. To ensure the data matches the true signal voltage values, the physical probe attenuation must match the scope attenuation values for that input channel.

Command message:

*C1:ATTN 100*

**RELATED COMMANDS**

VDIV  
OFST



**CHANNEL**

**BANDWIDTH\_LIMIT | BWL**

Command /Query

**DESCRIPTION**

BANDWIDTH\_LIMIT enables or disables the bandwidth-limiting low-pass filter. If the bandwidth filters are on, it will limit the bandwidth to reduce display noise. When you turn Bandwidth Limit ON, the Bandwidth Limit value is set to 20 MHz. It also filters the signal to reduce noise and other unwanted high frequency components.

The BANDWIDTH\_LIMIT? query returns whether the bandwidth filters are on.

**COMMAND SYNTAX**

BANDWIDTH\_LIMIT<channel>,<mode>  
[,<channel>,<mode>[,<channel>,<mode>[,  
<channel>,<mode>]]]

<channel>:={C1,C2,C3,C4}

<mode>:={ON,OFF}

**QUERY SYNTAX**

BANDWIDTH\_LIMIT?

**RESPONSE FORMAT**

BANDWIDTH\_LIMIT <channel>,<mode>  
[,<channel>,<mode>[,<channel>,<mode>[,  
<channel>,<mode>]]]

**EXAMPLE**

- The following command turns on the bandwidth filter for all channels.

Command message:

*BWL C1,ON,C2,ON,C3,ON,C4,ON*

- The following command turns the bandwidth filter on for Channel 1 only.

Command message:

*BWL C1,ON*

**CHANNEL**

**COUPLING | CPL**

Command /Query

**DESCRIPTION**

The COUPLING command selects the coupling mode of the specified input channel.

The COUPLING? query returns the coupling mode of the specified channel.

**COMMAND SYNTAX**

<channel>:COUPLING <coupling>

<channel>:={C1,C2,C3,C4}

<coupling>:={A1M,A50,D1M,D50,GND}

- A — alternating current.
- D — direct current.
- 1M — 1M $\Omega$  input impedance.
- 50 — 50 $\Omega$  input impedance.

**Note:**

Options of <coupling> vary from models. See the data sheet or the channel menu of oscilloscope for details.

**QUERY SYNTAX**

<channel>:COUPLING?

**RESPONSE FORMAT**

<channel>:COUPLING <coupling>

**EXAMPLE**

The following command sets the coupling of Channel 2 to 50  $\Omega$ , DC.

Command message:

*C2:CPL D50*

**CHANNEL**

**OFFSET | OFST**

Command/Query

**DESCRIPTION**

The OFFSET command allows adjustment of the vertical offset of the specified input channel. The maximum ranges depend on the fixed sensitivity setting.

The OFFSET? query returns the offset value of the specified channel.

**COMMAND SYNTAX**

<channel>:OFFSET <offset>

<channel>:={C1,C2,C3,C4}

<offset>:= vertical offset value with unit, see the data sheet for details.

**Note:**

- If there is no unit (V/mV/uV) added, it defaults to volts (V).
- If you set the offset to a value outside of the legal range, the offset value is automatically set to the nearest legal value. Legal values are affected by the probe attenuation setting.

**QUERY SYNTAX**

<channel>:OFFSET?

**RESPONSE FORMAT**

<channel>:OFFSET <offset>

<offset>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

- The following command sets the offset of Channel 2 to -3 V.  
Command message:  
*C2:OFST -3V*

- The following command sets the offset of Channel 1 to -50 mV.  
Command message:  
*C1:OFST -50mV*

**RELATED COMMANDS**

VDIV  
ATTN

**CHANNEL**

**SKEW**

Command/Query

**DESCRIPTION**

The SKEW command sets the channel-to-channel skew factor for the specified channel. Each analog channel can be adjusted + or -100 ns for a total of 200 ns difference between channels. You can use the oscilloscope's skew control to remove cable-delay errors between channels.

The SKEW? query returns the skew value of the specified trace.

**COMMAND SYNTAX**

<trace>:SKEW <skew>

<trace>:={C1,C2,C3,C4}

<skew>:=-100 ns to +100 ns.

**QUERY SYNTAX**

<trace>:SKEW?

**RESPONSE FORMAT**

<trace>:SKEW <skew>

Model	Format of <skew>
T3DSO1000(A)	Numerical value in E-notation with SI unit, such as 9.99E-08S.
T3DSO2000	Numerical value with measurement unit and physical unit, such as 0.00ns.

**EXAMPLE**

The following command sets skew value of Channel 1 to 3ns.

Command message:

*C1:SKEW 3NS*

**CHANNEL**

**TRACE | TRA**  
Command/Query

**DESCRIPTION**

The TRACE command turns the display of the specified channel on or off.

The TRACE? query returns the current display setting for the specified channel.

**COMMAND SYNTAX**

<trace>:TRACE <mode>

<trace>:={C1,C2,C3,C4}

<mode>:={ON,OFF}

**QUERY SYNTAX**

<trace>:TRACE?

**RESPONSE FORMAT**

<trace>:TRACE <mode>

**EXAMPLE**

The following command displays Channel 1.

Command message:

*C1:TRA ON*

**CHANNEL**

**UNIT**

Command / Query

**DESCRIPTION**

The UNIT command sets the unit of the specified trace. Measurement results, channel sensitivity, and trigger level will reflect the measurement units you select.

The UNIT? query returns the unit of the specified trace.

**COMMAND SYNTAX**

<channel>:UNIT <type>

<channel>:={C1,C2,C3,C4}

<type>:={V,A}

**QUERY SYNTAX**

<channel>:UNIT?

**RESPONSE FORMAT**

<channel>:UNIT <type>

**EXAMPLE**

The following command sets the unit of Channel 1 to V.

Command message:

*C1:UNIT V*

**CHANNEL**

**VOLT\_DIV | VDIV**  
Command /Query

**DESCRIPTION**

The VOLT\_DIV command sets the vertical sensitivity in Volts/div.  
If the probe attenuation is changed, the scale value is multiplied by the probe's attenuation factor.

The VOLT\_DIV? query returns the vertical sensitivity of the specified channel.

**COMMAND SYNTAX**

<channel>:VOLT\_DIV <v\_gain>

<channel>:={C1,C2,C3,C4}

<v\_gain>:= 500uV to 10V.

**Note:**

If there is no unit (V/mV/uV) added, it defaults to volts (V).

**QUERY SYNTAX**

<channel>:VOLT\_DIV?

**RESPONSE FORMAT**

<channel>:VOLT\_DIV <v\_gain>

<v\_gain>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

The following command sets the vertical sensitivity of Channel 1 to 50 mV/div.

Command message:

*C1:VDIV 50mV*

**RELATED COMMANDS**

ATTN

**CHANNEL**

**INVERTSET | INVS**

Command/Query

**DESCRIPTION**

The INVERTSET command mathematically inverts the specified traces or the math waveform.

The INVERTSET? query returns the current state of the channel inversion.

**COMMAND SYNTAX**

<trace>:INVERTSET <state>

<trace>:= {C1,C2,C3,C4,MATH}

<state>:= {ON,OFF}

**QUERY SYNTAX**

<trace>:INVERTSET?

**RESPONSE FORMAT**

<trace>:INVERTSET <state>

**EXAMPLE**

The following command inverts the trace of Channel 1.

Command message:

*C1:INVS ON*



## **CURSOR Commands**

The CURSOR subsystem commands set and query the settings of X-axis markers(X1 and X2 cursors) and the Y-axis markers (Y1 and Y2 cursors). You can set and query the marker mode and source, the position of X and Y cursors, and query delta X and delta Y cursor values.

**CRMS**  
**CRST**  
**CRTY**  
**CRVA?**

**CURSOR**

**CURSOR\_MEASURE | CRMS**

Command /Query

**DESCRIPTION**

The CURSOR\_MEASURE command specifies the type of cursor or parameter measurement to be displayed

The CURSOR\_MEASURE? query returns which cursors or parameter measurements are currently displayed.

**COMMAND SYNTAX**

CURSOR\_MEASURE <mode>

Format 1:

<mode>:={OFF,ON}

- OFF — manual mode.
- ON — track mode.

Format 2:

<mode>:={OFF,MANUAL,TRACK}

- OFF — close the cursors.
- MANUAL — manual mode.
- TRACK — track mode.

**Note:**

The table on next page shows the available command format in each oscilloscope series.

**QUERY SYNTAX**

CURSOR\_MEASURE?

**RESPONSE FORMAT**

CURSOR\_MEASURE <mode>

**EXAMPLE**

- The following command turns cursor function off on the T3DSO1000(A).

Command message:

*CRMS OFF*

- The following command sets cursor mode to track mode on the T3DSO1000(A).

Command message:

*CRMS ON*

**RELATED COMMANDS**

CRVA?  
CRST

---

## Digital Oscilloscope Series

---

### Format in Each Oscilloscope Series

Model	Command Format
T3DSO2000	Format 1
T3DSO1000(A)	Format 2

**CURSOR**

**CURSOR\_SET | CRST**

Command / Query

**DESCRIPTION**

The CURSOR\_SET command allows the user to position any one of the four independent cursors at a given screen location. The positions of the cursors can be modified or queried even if the required cursor is not currently displayed on the screen. When setting a cursor position, a trace must be specified, relative to which the cursor will be positioned.

The CURSOR\_SET? query returns the current position of the cursor(s). The values returned depend on the grid type selected.

**COMMAND SYNTAX**

<trace>:CURSOR\_SET

<cursor>,<position>[,<cursor>,<position>[,<cursor>,<position>[,<cursor>,<position>]]]

<trace>:={C1,C2,C3,C4}

<cursor>:={VREF,VDIF,TREF,TDIF,HRDF,HDIF}

- VREF — The voltage-value of Y1 (curA) under manual mode.
- VDIF — The voltage-value of Y2 (curB) under manual mode.
- TREF — The time value of X1 (curA) under manual mode.
- TDIF — The time value of X2 (curB) under manual mode.
- HREF — The time value of X1 (curA) under track mode.
- HDIF — The time value of X2 (curB) under track mode.

<position>:=- ( grid/2) \*DIV to (grid/2)\*DIV when <cursor> = {TREF, TDIF, HRDF, HDIF} (horizontal)

grid: The grid numbers in horizontal direction.  
 <position>:=- 4\*DIV to 4\*DIV when <cursor> = { VREF,VDIF} (vertical)

**Note:**

- The horizontal position range is related to the size of screen.
- You need to add the unit to the position value.

**QUERY SYNTAX**

```
<trace>:CURSOR_SET?  
<cursor>[,<cursor>[,<cursor>[,<cursor>]]]
```

```
<cursor>:={VREF,VDIF,TREF,TDIF,HREF,H  
DIF}
```

**RESPONSE FORMAT**

```
<trace>:CURSOR_SET <cursor>,<position>[,  
<cursor>,<position>[,<cursor>,<position>[,<cur  
sor>,<position>]]]
```

**EXAMPLE**

- When the current time base is 1 us, vdiv is 500 mV, the cursor mode is manual, the following command sets the X1 positions to -3 DIV, Y2 position to -1 DIV, using Channel 1 as a reference.  
Command message:  
*C1:CRST TREF,-3us,VDIF,-500mV*

- When the current time base is 1 us, the cursor mode is track, the following command sets the X1 positions to -1 DIV, X2 position to 2 DIV, using Channel 1 as a reference.  
Command message:  
*C1:CRST HREF,-1us,HDIF,2us*

**RELATED COMMANDS**

CRMS  
CRVA?

***CURSOR***

**CURSOR\_TYPE | CRTY**

Command /Query

**DESCRIPTION**

The CURSOR\_TYPE command specifies the type of cursor to be displayed when the cursor mode is manual.

The CURSOR\_TYPE query returns the current type of cursor.

**COMMAND SYNTAX**

CURSOR\_TYPE <type>

<mode>:={X,Y,X-Y}

**QUERY SYNTAX**

C U R S O R \_ T Y P E ?

**RESPONSE FORMAT**

CURSOR\_TYPE <type>

**EXAMPLE**

The following command sets cursor type to Y.

Command message:

*CRTY Y*

**RELATED COMMANDS**

CRMS

**CURSOR**

**CURSOR\_VALUE? | CRVA?**

Query

**DESCRIPTION**

The CURSOR\_VALUE? query returns the values measured by the specified cursors for a given trace.

**QUERY SYNTAX**

<trace>:CURSOR\_VALUE? <mode>

<trace>:= {C1, C2, C3, C4}

<mode>:= {HREL,VREL}

- HREL — return the delta time value, reciprocal of delta time value, X1 (curA) time value and X2 (curB) time value.
- VREL — return the delta volt value, Y1 (curA) volt value and Y2 (curB) volt value under manual mode.

**Note:**

For older models, VREL is the delta volt value under manual mode. See models on [page 14](#).

**RESPONSE FORMAT**

<trace>:CURSOR\_VALUE  
HREL,<delta>,<1/delta>,<value1>,<value2>

<trace>:CURSOR\_VALUE  
VREL,<delta>,<value1>,<value2>

**EXAMPLE**

When the cursor mode is manual, and the cursor type is Y, the following query returns the vertical value on channel 1.



Query message:

## Digital Oscilloscope Series

---

*C1:CRVA? VREL*

Response message:

*C1:CRVA VREL,-5.00E+00V,2.50E+00V,-  
2.50E+00V*

### RELATED COMMANDS

CRMS



## **DIGITAL Commands**

The DIGITAL subsystem commands control the viewing of digital channels. They also control threshold settings for groups of digital channels.

**DGCH**  
**DGST**  
**DGTH**  
**SW**  
**TRA**  
**TSM**  
**CUS**

**Note:**

These commands are only valid for models which have the MSO Option installed.

---

## Digital Oscilloscope Series

---

### **DIGITAL**

### **DIGITAL\_CHANNEL | DGCH**

Command / Query

#### **DESCRIPTION**

The DIGITAL\_CHANNEL command turns digital display on or off for the specified channel.

The DIGITAL\_CHANNEL? query returns the current digital display setting for the specified channel.

#### **COMMAND SYNTAX**

<digital>:DIGITAL\_STATE <state>

<digital>:={D0,D1,D2,D3,D4,D5,D6,D7,D8,D9,D10,D11,D12,D13,D14,D15}

<state>:={OFF,ON}

#### **QUERY SYNTAX**

<digital>:DIGITAL\_STATE?

#### **RESPONSE FORMAT**

<digital>:DIGITAL\_STATE <state>

#### **EXAMPLE**

The following command sets D8 display on.

Command message:

*D8:DGCH ON*

#### **Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	yes
T3DSO1000(A)	yes - except T3DSO1102

---

## Digital Oscilloscope Series

---

### **DIGITAL**

### **DIGITAL\_STATE | DGST**

Command /Query

#### **DESCRIPTION**

The DIGITAL\_STATE command is used to set the state of digital.

The DIGITAL\_STATE? query returns the state of digital.

#### **COMMAND SYNTAX**

DIGITAL\_STATE <state>

<state>:={OFF,ON}

#### **QUERY SYNTAX**

DIGITAL\_STATE?

#### **RESPONSE FORMAT**

DIGITAL\_STATE <state>

#### **EXAMPLE**

The following command sets Digital function on.  
Command message:  
*DGST ON*

#### **Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	yes
T3DSO1000(A)	yes - except T3DSO1102

**DIGITAL**

**DIGITAL\_THR | DGTH**

Command /Query

**DESCRIPTION**

The DIGITAL\_THR command sets the threshold for the specified group of channels. The threshold is used for triggering purposes and for displaying the digital data as high (above the threshold) or low (below the threshold).

The DIGITAL\_THR? query returns the threshold value for the specified group of channels.

**COMMAND SYNTAX**

<group>:DIGITAL\_THR <type>[,<level>]

<group>:= {C1,C2}

- C1 — D0-D7.
- C2 — D8-D15.

<type>:= {TTL,CMOS,CMOS3.3,CMOS2.5,CUSTOM}

<level>:= -5V to 5V when <type> is CUSTOM.

**Note:**

- If there is no unit(V) added to <level>, it defaults to be V.
- If you set the threshold to a value outside of the legal range, the threshold is automatically set to the nearest legal value.

**QUERY SYNTAX**

<group>:DIGITAL\_THR?

**RESPONSE FORMAT**

Format 1:  
DIGITAL\_THR <type>

Format 2:  
DIGITAL\_THR <group>:<level>

<type>	Response Format
TTL/CMOS/CMOS3.3/CMOS2.5	Format 1
CUSTOM	Format 2

## Digital Oscilloscope Series

### EXAMPLE

- For T3DSO1000(A) series, when the Digital function is on, the following command sets the threshold of D0-D7 to LVLCMOS3.3.

Command message:

*C1:DGTH CMOS3.3*

- For T3DSO1000(A) series, when the Digital function is on, the following command sets the threshold of D8-D15 to 3 V.

Command message:

*C2:DGTH CUSTOM,3V*

### Note:

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	yes
T3DSO1000(A)	yes - except T3DSO1102

---

## Digital Oscilloscope Series

---

### **DIGITAL**

### **SWITCH | SW**

Command / Query

#### **DESCRIPTION**

The SWITCH command is used to set the state of digital.

The SWITCH? query returns the state of digital.

#### **COMMAND SYNTAX**

<function>:SWITCH <state>

<function>:={DI}

<state>:={OFF,ON}

#### **QUERY SYNTAX**

<function>:SWITCH?

#### **RESPONSE FORMAT**

<function>:SWITCH <state>

#### **EXAMPLE**

For T3DSO1000(A) series, the following command sets Digital function on.

Command message:

*DI:SWITCH ON*

#### **Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes - except T3DSO1102

---

## Digital Oscilloscope Series

---

### **DIGITAL**

### **TRACE | TRA**

Command /Query

#### **DESCRIPTION**

The TRACE command turns digital display on or off for the specified channel.

The TRACE? query returns the current digital display setting for the specified channel.

#### **COMMAND SYNTAX**

<digital>:TRACE <state>

<digital>:={D0,D1,D2,D3,D4,D5,D6,D7,D8,D9,D10,D11,D12,D13,D14,D15}

<state>:={OFF,ON}

#### **QUERY SYNTAX**

<digital>:TRACE?

#### **RESPONSE FORMAT**

<digital>:TRACE <state>

#### **EXAMPLE**

For T3DSO1000(A) series, the following command sets D8 display on.

Command message:

*D8:TRACE ON*

#### **Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes - except T3DSO1102

**DIGITAL**

**THRESHOLD\_MODE | TSM**

Command / Query

**DESCRIPTION**

The THRESHOLD\_MODE command sets the threshold type for the specified group of channels. The threshold is used for triggering purposes and for displaying the digital data as high (above the threshold) or low (below the threshold).

The THRESHOLD\_MODE? query returns the threshold type for the specified group of channels.

**COMMAND SYNTAX**

<group>:THRESHOLD\_MODE <type>

<group>:={H8,L8}

- H8 — D8-D15.
- L8 — D0-D7.

<type>:={TTL,CMOS,LVCMOS33,LVCMOS25,CUSTOM}

**QUERY SYNTAX**

<group>:THRESHOLD\_MODE?

**RESPONSE FORMAT**

<group>:THRESHOLD\_MODE <type>

**EXAMPLE**

For T3DSO1000(A) series, when the Digital function is on, the following command sets the threshold of D0-D7 to LVLCMOS3.3.

Command message:

*L8:TSM LVCMOS33*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes - except T3DSO1102



**DIGITAL**

**CUSTOM | CUS**

Command /Query

**DESCRIPTION**

The CUSTOM command sets the threshold value by customer for the specified group of channels. The threshold is used for triggering purposes and for displaying the digital data as high (above the threshold) or low (below the threshold).

The CUSTOM? query returns the threshold value set by customer for the specified group of channels.

**COMMAND SYNTAX**

<group>:CUSTOM <value>

<group>:={H8,L8}

- H8 — D8-D15.
- L8 — D0-D7.

<value>:= volt value with unit.

**Note:**

- You need to add the volt unit(V/mV) to the value. If there is no unit added, it defaults to volts (V).
- The range of value varies from models. See the data sheet for details.
- An out-of-range value will be adjusted to the closest legal value.

**QUERY SYNTAX**

<group>:CUSTOM?

**RESPONSE FORMAT**

<group>:CUSTOM <value>

**EXAMPLE**

For T3DSO1000(A) series, when the Digital function is on, the following command sets the threshold value of D8-D15 to 5V.

Command message:

*L8:CUSTOM 5V*

## Digital Oscilloscope Series

---

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes - except T3DSO1102

## DISPLAY Commands

The DISPLAY subsystem is used to control how waveforms, and the graticules are displayed on the screen.

**DTJN**  
**GRDS**  
**INTS**  
**MENU**  
**PESU**

**DISPLAY**

**DOT\_JOIN | DTJN**

Command /Query

**DESCRIPTION**

The DOT\_JOIN command sets the interpolation lines between data points.

**COMMAND SYNTAX**

DOT\_JOIN <state>

<state>:={ON,OFF}

- ON — dots. This mode displays data more quickly than vector mode but does not draw lines between sample points.
- OFF — vectors. This is the default mode and draws lines between points.

**QUERY SYNTAX**

DOT\_JOIN?

**RESPONSE FORMAT**

DOT\_JOIN <state>

**EXAMPLE**

The following command turns off the interpolation lines.

Command message:

*DTJN ON*

**DISPLAY**

**GRID\_DISPLAY | GRDS**

Command /Query

**DESCRIPTION**

The GRID\_DISPLAY command selects the type of the grid which is used to display.

The GRID\_DISPLAY? query returns the current type of grid.

**COMMAND SYNTAX**

GRID\_DISPLAY <type>

< type >:={FULL,HALF,OFF}

**QUERY SYNTAX**

GRID\_DISPLAY?

**RESPONSE FORMAT**

GRID\_DISPLAY <type>

**EXAMPLE**

The following command changes the type of grid to full grid.

Command message:

*GRDS FULL*

**DISPLAY**

**INTENSITY | INTS**

Command/Query

**DESCRIPTION**

The INTENSITY command sets the intensity level of the grid or the trace.

The INTENSITY? query returns the grid and trace intensity levels.

**COMMAND SYNTAX**

INTENSITY GRID,<value>,TRACE,<value>

<value>:= 0(or 30) to 100

**Note:**

You can also set the intensity level of the grid or trace using a key-value pair alone, see the example for details.

**QUERY SYNTAX**

INTENSITY?

**RESPONSE FORMAT**

INTENSITY TRACE,<value>,GRID,<value>

**EXAMPLE**

The following command changes the grid intensity level to 75%.

Command message:

*INTS GRID,75*

**DISPLAY**

**MENU**

Command/Query

**DESCRIPTION**

The MENU command enables or disables to display the menu.

The MENU? query returns whether the menu is displayed.

**COMMAND SYNTAX**

MENU <state>

<state>:={ON,OFF}

**QUERY SYNTAX**

MENU?

**RESPONSE FORMAT**

MENU <state>

**EXAMPLE**

The following command enables the display of the menu.

Command message:

*MENU ON*

**DISPLAY**

**PERSIST\_SETUP | PESU**

Command /Query

**DESCRIPTION**

The PERSIST\_SETUP command selects the persistence duration of the display, in seconds, in persistence mode.

The PERSIST\_SETUP? query returns the current status of the persistence.

**COMMAND SYNTAX**

PERSIST\_SETUP <time>

Models	<time>:=
T3DSO1000(A)	{OFF,INFINITE,1,5,10,30}
Others	{INFINITE,1,5,10,30}

**Note:**

- See models on [page 14](#).
- See the command PERS in Obsolete Commands for Old Models to set persist off .
- Options of <time> vary from models. See the data sheet or the display menu of the oscilloscope for details.

**QUERY SYNTAX**

PERSIST\_SETUP?

**RESPONSE FORMAT**

PERSIST\_SETUP <time>

**EXAMPLE**

The following command sets the variable persistence at 5 seconds.

Command message:

*PESU 5*



## **HISTORY Commands**

The HISTORY subsystem commands control the waveform recording function and the history waveform play function.

**FRAM**

**FTIM?**

**HSMD**

**HSLST**

**HISTORY**

**FRAME\_SET | FRAM**

Command/ Query

**DESCRIPTION**

The FRAME\_SET command is used to set history current frame number.

The FRAME\_SET? query returns the current frame number.

**COMMAND SYNTAX**

FRAM <frame\_num>

<frame\_num>:= 0 to the max frame number.

**Note:**

You can send the query FRAM? to get the max frame number when the history function is turned on for the first time.

**QUERY SYNTAX**

FRAM?

**RESPONSE FORMAT**

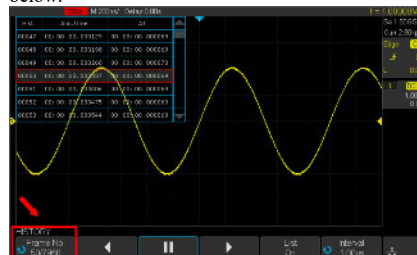
FRAM <frame\_num>

**Note:**

The query is only valid for T3DSO1000(A) series.

**EXAMPLE**

When the history function is on, the following command sets current frame number to 50. Then you can see the response on the screen as shown below.



Command message:

FRAM 50

**HISTORY**

**FRAME\_TIME? | FTIM?**

Query

**DESCRIPTION**

The FRAME\_TIME query returns the acquire timestamp of the current frame.

**QUERY SYNTAX**

FTIM?

**RESPONSE FORMAT**

Format 1:  
FTIM hour: minute: second. micro-second

Format 2:  
\\xFF\\x0F\\x03\\x01&\\xD5\\x02\\x00

**Note:**

- Format 2 is binary data and has no key word.
- The table below shows the available response format in each oscilloscope series.

**EXAMPLE**

For the T3DSO1000(A) series, when the history function is on, the following query returns the acquire time of the current frame.

Query message:  
*FTIM?*

Response message:  
*FTIM 00: 05: 12. 650814*

**Format in Each Oscilloscope Series**

Model	Response Format
T3DSO2000	Format 2
T3DSO1000(A)	Format 1

**HISTORY**

**HISTORY\_MODE | HSMD**

Command/ Query

**DESCRIPTION**

The HISTORY\_MODE command is used to set the state of history mode.

The HISTORY\_MODE? query returns the current state of history mode.

**COMMAND SYNTAX**

HSMD <state>

<state>:= {ON,OFF}

**QUERY SYNTAX**

HSMD?

**RESPONSE FORMAT**

HSMD <state>

**EXAMPLE**

The following command sets the state of history mode to ON.

Command message:

*HSMD ON*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**HISTORY**

**HISTORY\_LIST | HSLST**

Command/ Query

**DESCRIPTION**

The HISTORY\_LIST command is used to set the state of history list.

The HISTORY\_LIST? query returns the current state of history list.

**COMMAND SYNTAX**

HSLST <state>

<state>:={ON,OFF}

**Note:**

This command can only be used when History function is turned on.

**QUERY SYNTAX**

HSLST?

**RESPONSE FORMAT**

HSLST <state>

**EXAMPLE**

When History function is on, the following command sets the state of history list to ON.  
Command message:

*HSLST ON*

**RELATED COMMANDS**

HSMD

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

## MATH Commands

The MATH subsystem controls the math functions in the oscilloscope. As selected by the **DEF** command, these math functions are available:  
Operators: Add, Subtract, Multiply, Divide.  
Operators perform their function on two analog channel sources.  
Transforms: DIFF, Integrate, FFT, SQRT.

**DEF**  
**INVS**  
**MTVD**  
**MTVP**  
**FFTC**  
**FFTF**  
**FFTP**  
**FFTS**  
**FFTT?**  
**FFTU**  
**FFTW**

**MATH**

**DEFINE | DEF**  
Command /Query

**DESCRIPTION**

The DEFINE command sets the desired waveform math operation.

The DEFINE? query returns the current operation for the selected function.

**COMMAND SYNTAX**

DEFINE EQN, '<equation>'

**Note:**

<equation> is the mathematical expression, enclosed by single or double quotation marks.

Function Equations	
<source1> + <source2>	Addition
<source1> - <source2>	Subtraction
<source1>*<source2>	Multiplication
<source1>/<source2>	Ratio
FFT<source>	FFT
INTG<source>	Integral
DIFF<source>	Differentiator
SQRT<source>	Square Root

<source>:={C1,C2,C3,C4}

<source1>:={C1,C2,C3,C4}

<source2>:={C1,C2,C3,C4}

**QUERY SYNTAX**

DEFINE?

**RESPONSE FORMAT**

DEFINE EQN, '<equation>'

**EXAMPLE**

• When the Math function is on, and both Channel 1 and Channel 2 are on, the following command sets the math operation to Multiplication, source1 to C1, source2 to C2.

Command message:

*DEFINE EQN, 'C1\*C2'*

• When the Math function is on, and Channel 1

## Digital Oscilloscope Series

---

is on, the following command sets the math operation to Differentiator, source to C1.

Command message:

*DEFINE EQN,'DIFFC1'*



**MATH**

**INVERTSET | INVS**

Command/Query

**DESCRIPTION**

The INVERTSET command inverts the math waveform.

The INVERTSET? query returns whether the math waveform is inverted or not.

**Note:**

This command is only valid in add, subtract, multiply and divide operation.

**COMMAND SYNTAX**

<trace>:INVERTSET <state>

<trace>:= {MATH}

<state>:= {ON,OFF}

**QUERY SYNTAX**

<trace>:INVERTSET?

**RESPONSE FORMAT**

<trace>:INVERTSET <state>

**EXAMPLE**

When the Math function is on, and the operation is Add, the following command inverts the math waveform.

Command message:

*MATH:INVS ON*

**MATH**

Command/Query

**MATH\_VERT\_DIV | MTVD**

**DESCRIPTION**

The MATH\_VERT\_DIV command sets the vertical scale of the selected math operation. This command is only valid in add, subtract, multiply and divide operation.

The MATH\_VERT\_DIV? query returns the current scale value for the selected operation.

**COMMAND SYNTAX**

MATH\_VERT\_DIV <scale>

<scale>:={500uV,1mV,2mV,5mV,10mV,20mV,50mV,100mV,200mV,500mV,1V,2V,5V,10V,20V,50V,100V}(for add, subtract, multiply and divide)

**Note:**

Legal values for the scale depend on the selected operation. For details, please refer to the math menu of the oscilloscope as shown below.



**QUERY SYNTAX**

MATH\_VERT\_DIV?

**RESPONSE FORMAT**

MATH\_VERT\_DIV <scale>

Model	Format of <scale>
T3DSO1000(A)	Numerical value in E-notation with SI unit, such as 5.00E-01V.
others	Numerical value

---

## Digital Oscilloscope Series

	with measurement unit and physical unit, such as 500mV.
--	---

### EXAMPLE

When the Math function is on, and the operator is Add, the following command changes the vertical scale of the math waveform to 1 V.

Command message:  
*MTVD 1V*

**MATH**

Command/Query

**MATH\_VERT\_POS | MTVP**

**DESCRIPTION**

The MATH\_VERT\_POS command sets the vertical position of the math waveform with specified source.

The FFT waveform isn't included, but we have another command which called FFTP to set vertical position.

The MATH\_VERT\_POS? query returns the vertical position of the math waveform.

**COMMAND SYNTAX**

MATH\_VERT\_POS <point>

<point>:= -255 to 255.

**Note:**

The point represents the screen pixels and is related to the screen center. For example, if the point is 50. The math waveform will be displayed 1 grid above the vertical center of the screen. Namely one grid is 50.

**QUERY SYNTAX**

MATH\_VERT\_POS?

**RESPONSE FORMAT**

MATH\_VERT\_POS <point>

**EXAMPLE**

When the Math function is on, the following command sets the vertical position of the math waveform to 1 grid above the screen vertical center.

Command message:

*MTVP 50*

**RELATED COMMANDS**

FFTP

**MATH**

Command /Query

**FFT\_CENTER | FFTC**

**DESCRIPTION**

The FFT\_CENTER command sets the center frequency when FFT (Fast Fourier Transform) is selected.

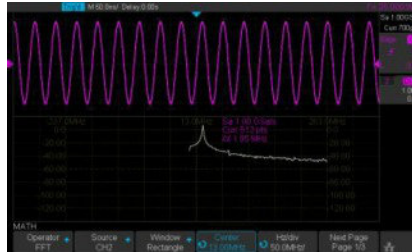
The FFT\_CENTER? query returns the current center frequency of FFT waveform.

**COMMAND SYNTAX**

FFT\_CENTER <center>  
<center>:= frequency value with unit (MHz/ kHz/ Hz).

**Note:**

- If you set the center to a value outside of the legal range, the center value is automatically set to the nearest legal value. Legal values are affected by the Hz/div setting.
- The range for center is related to the horizontal scale of FFT and varied by models. See the math menu of oscilloscope as shown below for details.



**QUERY SYNTAX**

FFT\_CENTER?

**RESPONSE FORMAT**

FFT\_CENTER <center>

**EXAMPLE**

When the Math function is on, the operator is FFT, and the horizontal scale is 100 MHz, the following command sets the center frequency of FFT to 58 MHz.

Command message:

*FFTC 58MHz*

---

## Digital Oscilloscope Series

---

### RELATED COMMANDS      FFFT?

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**MATH**

**FFT\_FULLSCREEN | FFTF**

Command / Query

**DESCRIPTION**

The FFT\_FULLSCREEN command sets the display mode of FFT waveform.

The FFT\_FULLSCREEN? query returns whether the FFT waveform is full screen displayed.

**COMMAND SYNTAX**

FFT\_FULLSCREEN <state>

<state>:= {OFF,ON, EXCLU}

- OFF — Split Screen.
- ON — Full Screen.
- EXCLU — Exclusive.

**QUERY SYNTAX**

FFT\_FULLSCREEN?

**RESPONSE FORMAT**

FFT\_FULLSCREEN <state>

**EXAMPLE**

When the Math function is on, and the operator is FFT, the following command sets the display mode of FFT waveform to Full Screen.

Command message:

*FFTF ON*

**MATH**

**FFT\_POSITION | FFTP**

Command /Query

**DESCRIPTION**

The FFT\_POSITION command sets the vertical offset of FFT waveform. The unit is related to the vertical scale type of the current FFT and the unit of the channel.

The FFT\_POSITION? query returns the current vertical offset of the FFT waveform.

**Note:**

- This command is only valid when the scale type is Vrms.

**COMMAND SYNTAX**

FFT\_POSITION <offset>

<offset>:= -24.4\*DIV to 15.6\*DIV.

**Note:**

- If there is no unit (V/mV/uV) added, it defaults to volts (V).
- If you set the offset to a value outside of the legal range, the center value is automatically set to the nearest legal value. Legal values are affected by the Scale setting.

**QUERY SYNTAX**

FFT\_POSITION?

**RESPONSE FORMAT**

FFT\_POSITION <offset>

<offset>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

- When the Math function is on, the operator is FFT, and the scale is 10 mV, the following steps set the offset of FFT waveform to 28 mV.

**Step 1:** Send command to set the scale unit to Vrms.

Command message:

*FFTU VRMS*

**Step 2:** Send command to set the offset to 28mV



---

## Digital Oscilloscope Series

Command message:

*FFTP 28mV*

- When the Math function is on, the operator is FFT, and the scale is 5 V, the following steps set the offset of FFT waveform to -13.5 V.

**Step 1:** Send command to set the scale unit to Vrms.

Command message:

*FFTU VRMS*

**Step 2:** Send command to set the offset to -13.5V

Command message:

*FFTP -13.5V*

### RELATED COMMANDS

FFTS  
FFTU

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**MATH**

**FFT\_SCALE | FFTS**

Command /Query

**DESCRIPTION**

The FFT\_SCALE command sets the vertical scale of FFT waveform. The unit is related to the vertical scale type of the current FFT and the unit of the channel.

The FFT\_SCALE? query returns the current vertical scale of FFT waveform.

**COMMAND SYNTAX**

FFT\_SCALE <scale>  
<scale>:= {0.1,0.2,0.5,1,2,5,10,20} when scale type is dBVrms or dBm.  
  
<scale>:= {0.001,0.002,0.005,0.01,0.02,0.05,0.1, 0. 2,0.5,1, 2,5,10,20} when scale type is Vrms.

**QUERY SYNTAX**

FFT\_SCALE?

**RESPONSE FORMAT**

FFT\_SCALE <scale>  
<scale>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

• When the Math function is on, and the operator is FFT, the following steps set the vertical scale of FFT to 5 dBVrms.

**Step 1:** Send command to set the scale unit to dBVrms.

Command message:  
*FFTU DBVRMS*

**Step 2:** Send command to set the scale to 5.

Command message:  
*FFTS 5*

• When the Math function is on, and the operator is FFT, the following steps set the vertical scale of FFT to 100 mVrms.

**Step 1:** Send command to set the scale unit to Vrms.

## Digital Oscilloscope Series

---

Command message:  
*FFTU VRMS*

**Step 2:** Send command to set the scale to 0.1.

Command message:  
*FFTS 0.1*

### RELATED COMMANDS

UNIT  
FFTU  
FFTP

---

## Digital Oscilloscope Series

---

### **MATH**

### **FFT\_TDIV? | FFTT?**

Query

#### **DESCRIPTION**

The FFT\_TDIV? query returns current horizontal scale of FFT waveform.

#### **QUERY SYNTAX**

FFT\_TDIV?

#### **RESPONSE FORMAT**

FFT\_TDIV <value>  
<value>:= Numerical value with measurement unit and physical unit.

#### **EXAMPLE**

The following query returns the horizontal scale unit of FFT.

Query message:

*FFTT?*

Response message:

*FFTT 100.00MHz*

#### **Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**MATH**

**FFT\_UNIT | FFTU**

Command /Query

**DESCRIPTION**

The FFT\_UNIT command sets the vertical scale type of FFT (Fast Fourier Transform algorithm).

The FFT\_UNIT? query returns the current vertical scale type of FFT waveform.

**COMMAND SYNTAX**

FFT\_UNIT <unit>  
<unit>:={VRMS,DBM,DBVRMS}

**QUERY SYNTAX**

FFT\_UNIT?

**RESPONSE FORMAT**

FFT\_UNIT <unit>

**EXAMPLE**

For T3DSO1000(A) series, when the Math function is on, and the operator is FFT, the following command sets the vertical scale unit of FFT to dBVrms.

Command message:

*FFU DBVRMS*

**RELATED COMMANDS**

FFTS  
FFTP

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**MATH**

Command /Query

**FFT\_WINDOW | FFTW**

**DESCRIPTION**

The FFT\_WINDOW command allows the selection of five different windowing transforms or operations for the FFT (Fast Fourier Transform) function. Each window is useful for certain classes of input signals.

The FFT\_WINDOW? query returns the current window of FFT.

**COMMAND SYNTAX**

FFT\_WINDOW <window>  
<window>:={RECT,BLAC,HANN,HAMM,FL  
ATTOP}

- RECT — Rectangle is useful for transient signals, and signals where there are an integral number of cycles in the time record.
- BLAC — Blackman reduces time resolution compared to the rectangular window, but it improves the capacity to detect smaller impulses due to lower secondary lobes (provides minimal spectral leakage).
- HANN — Hanning is useful for frequency resolution and general purpose use. It is good for resolving two frequencies that are close together, or for making frequency measurements.
- HAMM — Hamming.
- FLAT — Flattop is the best for making accurate amplitude measurements of frequency peaks.

**QUERY SYNTAX**

FFT\_WINDOW?

**RESPONSE FORMAT**

FFT\_WINDOW <window>

**EXAMPLE**

When the Math function is on, and the operator is FFT, the following command sets the FFT window to Hamming.

Command message:

*FTW HAMM*

## MEASURE Commands

The commands in the MEASURE subsystem are used to make parametric measurements on displayed waveforms.

To make a measurement, the portion of the waveform required for that measurement must be displayed on the oscilloscope screen.

**CYMT?**

**MEAD**

**PACU**

**PAVA?**

**MEASURE**

**CYMOMETER? | CYMT?**

Query

**DESCRIPTION**

The CYMOMETER? query measures and returns the frequency counter of the specified source. The counter measurement counts the trigger level crossings at the selected trigger slope and displays the results in MHz/kHz/Hz.

In the following picture, the content of the red box is the measured value of the cymometer.



**QUERY SYNTAX**

CYMOMETER?

**RESPONSE FORMAT**

CYMOMETER <freq>

Model	Format of <freq>
T3DSO1000(A)	Numerical value in E-notation with SI unit, such as 1.00E+03Hz.
others	Numerical value with measurement unit and physical unit, such as 1.00001kHz.

**Note:**

When the signal frequency is less than 10 Hz, it Returns 10 Hz| or <10Hz| .



**EXAMPLE**

- When the frequency of input signal is 1 Hz, the following returns the value of cymometer which displaying on the screen of the instrument.

Response message:

*CYMT 10Hz*

- When the frequency of input signal is 25.000137 MHz, the following returns the value of cymometer which displaying on the screen of the instrument.

Response message:

*CYMT 2.50E+07 Hz*

**MEASURE**

**MEASURE\_DELAY | MEAD**

Command/Query

**DESCRIPTION**

The MEASURE\_DELAY command places the instrument in the continuous measurement mode and starts a type of delay measurement.







The MEASURE\_DELAY? query returns the measured value of delay type.

**COMMAND SYNTAX**





MEASURE\_DELAY <type>,<sourceA-sourceB>

<sourceA-sourceB>:={C1-C2,C1-C3,C1-C4,C2-C3,C2-C4,C3-C4}

<type>:={PHA,FRR,FRF,FFR,FFF,LRR,LRF,LFR,LFF,SKEW}

Type	Description
 PHA	The phase difference between two channels. (rising edge - rising edge)
 FRR	Delay between two channels. (first rising edge - first rising edge)
 FRF	Delay between two channels. (first rising edge - first falling edge)
 FFR	Delay between two channels. (first falling edge - first rising edge)
 FFF	Delay between two channels. (first falling edge - first falling edge)
 LRR	Delay between two channels. (First rising edge - last rising edge)

## Digital Oscilloscope Series

	Delay between two channels. (first rising edge - last falling edge)
	Delay between two channels. (first falling edge - last rising edge)
	Delay between two channels. (first falling edge - last falling edge)
	Delay between two channels. (edge – edge of the same type)

### QUERY SYNTAX

<type>,<sourceA-sourceB>:MEASURE\_DELY?

### RESPONSE FORMAT

<sourceA-sourceB>:MEAD <type>,<value>

Model	Format of <value>
T3DSO1000(A)	Numerical value in E-notation with SI unit, such as 1.24E-04S. Except for PHA, it returns as "44.65degree".
others	Numerical value in E-notation with SI unit, such as 2.06E+01degree.

### EXAMPLE

The following steps show how to get the measured value of phase between C2 and C4.

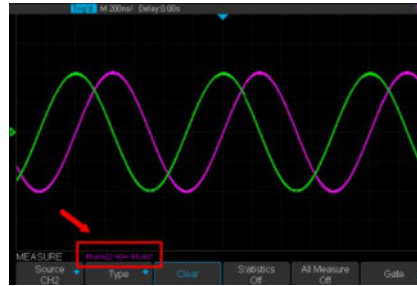
**Step 1:** Send the message to set the measurement to Phase between C2 and C4, and then there displays a phase measurement on the screen.

Command message:

*MEAD PHA,C2-C4*

---

## Digital Oscilloscope Series



**Step 2:** Send the message to get the measured value of phase.

Command message:  
*C2-C4:MEAD? PHA*

Response message:  
*C2-C4:MEAD PHA,-89.46degree*

**MEASURE**

**PARAMETER\_CUSTOM | PACU**  
Command

**DESCRIPTION**

The PARAMETER\_CUSTOM command installs a measurement and starts the specified measurement of the specified source.

See the command PAVA? to get the measured value of specified measurement.

See the command MEAD to install the measurement of delay class.

**COMMAND SYNTAX**

PARAMETER\_CUSTOM

<parameter>,<source>

<source>:= {C1,C2,C3,C4}

<parameter>:= {PKPK,MAX,MIN,AMPL,TOP, BASE,CMEAN,MEAN,RMS,CRMS,OVSN,FP RE,OVSP,RPRE,PER,FREQ,PWID,NWID,RIS E,FALL,WID,DUTY,NDUTY,ALL}

**Description of Parameter**

Parameter	Description
PKPK	vertical peak-to-peak
MAX	maximum vertical value
MIN	minimum vertical value
AMPL	vertical amplitude
TOP	waveform top value
BASE	waveform base value
CMEAN	average value in the first cycle
MEAN	average value
RMS	RMS value
CRMS	RMS value in the first cycle
OVSN	overshoot of a falling edge
FPRE	preshoot of a falling edge
OVSP	overshoot of a rising edge
RPRE	preshoot of a rising edge
PER	period
FREQ	frequency

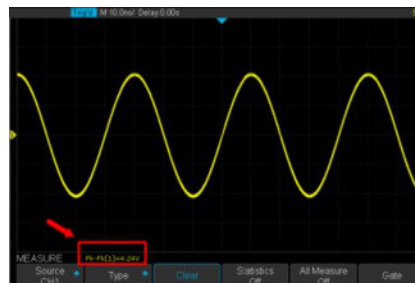
## Digital Oscilloscope Series

PWID	positive pulse width
NWID	negative pulse width
RISE	rise-time
FALL	fall-time
WID	Burst width
DUTY	positive duty cycle
NDUTY	negative duty cycle
ALL	All measurement

### EXAMPLE

- The following command sets the type of measure to PKPK of Channel 1. Command message:  
*PACU PKPK,C1*

Then, you can see the measurement on the screen.



- The following command sets the type of measure to ALL of Channel 2. Command message:  
*PACU ALL,C2*

Then, you can see a snapshot of all measurements on the screen.

## Digital Oscilloscope Series



RELATED COMMANDS

PAVA?  
MEAD

**MEASURE**

**PARAMETER\_VALUE? | PAVA?**

Query

**DESCRIPTION**

The PARAMETER\_VALUE query measures and returns the specified measurement value present on the selected waveform.

There are three uses for this command:

Usage	Description
Usage 1	Specify the source and the measurement. See the command "MEAD?" to get the measured value of delay measurement.
Usage 2	Use "PAVA? CUST<x>" to get customized.
Usage 3	Use "PAVA? STAT<x>" to get statistics.

**QUERY SYNTAX**

**Usage 1**

<source>:PARAMETER\_VALUE?  
<parameter>

<source>:= {C1,C2,C3,C4}

<parameter>:= {PKPK,MAX,MIN,AMPL,TOP, BASE,CMEAN,MEAN,RMS,CRMS,OVS,N,FP RE,OVS,P,RPRE,PER,FREQ,PWID,NWID,RIS E,FALL,WID,DUTY,NDUTY,ALL}

See the table **Description of Parameter** for details.

**RESPONSE FORMAT**

<source>:PARAMETER\_VALUE  
<parameter>,<value>  
<value>:= Numerical value in E-notation with SI unit.

**QUERY SYNTAX**

**Usage 2**

PARAMETER\_VALUE? CUST<x>  
<x>:= 1 to 5, and ALL



## Digital Oscilloscope Series

Custom Parameters	Description
CUST1	The first measure parameter specified by "PACU"
CUST2	The second measure parameter specified by "PACU"
CUST3	The third measure parameter specified by "PACU"
CUST4	The fourth measure parameter specified by "PACU"
CUST5	The fifth measure parameter specified by "PACU"
CUSTALL	All measure parameters specified by "PACU"

**Note:**

- Installing the measurement as CUST<x> by using command "PACU", before using usage 2.
- When the number of installed measurements is less than 5 and you send the command "PAVA? CUSTALL", it will return OFF as value for remaining custom parameters.

### RESPONSE FORMAT

PARAMETER\_VALUE  
 CUST<x>:<source>,<parameter>,<value>  
 <value>:= Numerical value in E-notation with SI unit.

### QUERY SYNTAX

Usage 3  
 PARAMETER\_VALUE? STAT<x>  
 <x>:= 1 to 5

Custom Parameters	Description
STAT1	Statistics of the first measure parameter specified by "PACU"
STAT2	Statistics of the second measure parameter specified by "PACU"
STAT3	Statistics of the third measure parameter

## Digital Oscilloscope Series

	specified by "PACU"
STAT4	Statistics of the fourth measure parameter specified by "PACU"
STAT5	Statistics of the fifth measure parameter specified by "PACU"

**Note:**

Installing the statistics of the measurement as STAT<x> by using command "PACU", before using usage 3.

### RESPONSE FORMAT

PARAMETER VALUE STAT<x> <source>  
 <parameter>:cur,<value1>,mean,<value2>,min,  
 <value3>,max,<value4>,std-  
 dev,<value5>,count,<value6>

Parameter	Description
cur	Current value of measurement
mean	Mean value of measurement
min	Minimum value of measurement
max	Maximum value of measurement
std-dev	Standard deviation of measurement
count	Measurement count

<value>:= Numerical value in E-notation with SI unit.

### EXAMPLE

- The following query returns the rise time of Channel 2.

Query message:  
*C2:PAVA? RISE*

Response message:  
*C2:PAVA RISE, 3.6E-9S*

- The following query returns all measurement of Channel 1.

## Digital Oscilloscope Series

---

Query message:

*CI:PAVA? ALL*

Response message:

*CI:PAVA MAX,2.04E+00V,MIN,-  
2.16E+00V,PKPK,4.20E+00V,TOP,2.00E+00V,  
BASE,-2.08E+00V,AMPL,4.08E+00V,MEAN,-  
1.95E-02V,CMEAN,-6.30E-  
03V,STDEV,1.46E+00V,VSTD,1.46E+00V,RMS,  
1.46E+00V,CRMS,1.46E+00V,OVSN,1.96%,FP  
RE,0.98%,OVSP,0.98%,RPRE,0.00%,LEVELX,0  
.00E+00V,PER,4.00E08S,FREQ,2.50E+07Hz,P  
WID,\*\*\*,NWID,\*\*\*,RISE,4.29E-  
01S,FALL,1.14E-08S,WID,9.99E-  
08S,DUTY,\*\*\*,NDUTY,\*\*\*,DELAY,-6.01E-  
08S,TIMEL,3.97E-08S*

- The following steps show how the user customize the measurement parameters and get the measured value.

**Step 1:** Send the command to set the measurement parameter.

Command message:

*PACU PKPK,CI*

**Step 2:** Send the query to get the measured value.

Query message:

*PAVA? CUST1*

Response message:

*PAVA CUST1:CI,PKPK,4.08E+00V*

**Step 3:** You can also send the query to get the measured value.

Command message:

*PAVA? CUSTALL*

Response message:

*PAVA  
CUST1:CI,PKPK,4.08E+00V;CUST2:OFF;CU  
ST3:OFF;CUST4:OFF;CUST5:OFF*

## Digital Oscilloscope Series

---

- The following steps show how to get the statistical values of user defined measurement parameters.

**Step 1:** Send the command to set the measurement parameter as the first customized parameter.

Command message:  
*PACU FREQ,C3*

**Step 2:** Send the query to get the statistical values of the first customized parameter.

Query message:  
*PAVA? STAT1*

Response message:  
*PAVA STAT1 C3*  
*FREQ:cur,1.00E+06Hz,mean,1.00E+06Hz,min,*  
*9.97E+05Hz,max,1.00E+06Hz,std-*  
*dev,1.41E+03Hz,count,171*

### RELATED COMMANDS

PACU  
MEAD

## PASS/FAIL Commands

The PASS/FAIL subsystem commands and queries control the mask test features.

- **PACL**
- **PFBF**
- **PFCM**
- **PFDD?**
- **PFDS**
- **PFEN**
- **PFFS**
- **PFOP**
- **PFSC**
- **PFST**

---

**Digital Oscilloscope Series**

***PASS/FAIL***

**PARAMETER\_CLR | PACL**  
Command

**DESCRIPTION**

The PARAMETER\_CLR command resets the P/F test statistics.

**COMMAND SYNTAX**

PARAMETER\_CLR

**RELATED COMMANDS**

PFDD?

**PASS/FAIL**

**PF\_BUFFER | PFBF**

Command/Query

**DESCRIPTION**

The PF\_BUFFER command sets the output mode when the test fails. This is the same as pressing the “Output” button on the menu of PASS/FAIL on the front panel.

The PF\_BUFFER? query returns the current output mode of the pass/fail.

**COMMAND SYNTAX**

PF\_BUFFER <state>  
<state>:= {ON,OFF}

- ON — The statistical result is displayed when the failed waveform is detected, and the buzzer alarm. (not related to the state of the sound switch)
- OFF — The statistical result is displayed when the failed waveform is detected, but the buzzer does not alarm.

**QUERY SYNTAX**

PF\_BUFFER?

**RESPONSE FORMAT**

PF\_BUFFER <state>

**EXAMPLE**

When the PASS/FAIL function is on, the following command sets “output” to “ON”.  
Command message:

*PFBF ON*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**PASS/FAIL**

**PF\_CREATEM | PFCM**  
Command

**DESCRIPTION**

The PF\_CREATEM command creates a pass/ fail test rule around the current selected channel, using the horizontal adjustment parameters and the vertical adjustment parameters defined by the PFST commands.

**Note:**

This command is valid only if the pass / fail test function has been opened (PFEN) and is not in operation (PFOP).

**COMMAND SYNTAX**

PF\_CREATEM

**EXAMPLE**

The following steps create the mask of the pass/fail.

**Step 1:** Send command to set the Pass/Fail test enable.

Command message:

*PFEN ON*

**Step 2:** Send command to stop the operation.

Command message:

*PFOP OFF*

**Step 3:** Send command to create the rule.

Command message:

*PFCM*

**RELATED COMMANDS**

PFST  
PFSC  
PFEN  
PFOP



---

## Digital Oscilloscope Series

### ***PASS/FAIL***

### **PF\_DATADIS? | PFDD?**

Query

#### **DESCRIPTION**

The PF\_DATADIS? query returns the number of the failed frames, passed frames and total frames which are shown on screen.

#### **COMMAND SYNTAX**

PF\_DATADIS?

#### **RESPONSE FORMAT**

PF\_DATADIS  
FAIL,<num>,PASS,<num>,TOTAL,<num>

#### **EXAMPLE**

The following query returns the number of the message display of the pass/fail.

Query message:

*PFDD?*

Response message:

*PFDD FAIL,0,PASS,0,TOTAL,0*

**PASS/FAIL**

**PF\_DISPLAY | PFDS**

Command /Query

**DESCRIPTION**

The PF\_DISPLAY command displays information in Pass/Fail test features.

The PF\_DISPLAY? query returns whether the message of Pass/Fail is displayed.

**COMMAND SYNTAX**

PF\_DISPLAY <state>

<state>:= {ON,OFF}

**QUERY SYNTAX**

PF\_DISPLAY?

**RESPONSE FORMAT**

PF\_DISPLAY <state>

**EXAMPL**

The following steps display the message of Pass/Fail.

**Step 1:** Send command to set the Pass/Fail test enable.

Command message:

*PFEN ON*

**Step 2:** Send command to display the message of Pass/Fail.

Command message:

*PFDS ON*

**RELATED COMMANDS**

PFEN

**PASS/FAIL**

**PF\_ENABLE | PFEN**

Command /Query

**DESCRIPTION**

The PF\_ENABLE command enables or disables the Pass/Fail test features.

The PF\_ENABLE? query returns the current state of mask test features.

**COMMAND SYNTAX**

PF\_ENABLE <state>

<state>:= {ON,OFF}

- ON — Enables the mask test features.
- OFF — Disables the mask test features.

**QUERY SYNTAX**

PF\_ENABLE?

**RESPONSE FORMAT**

PF\_ENABLE <state>

**EXAMPL**

The following command enables mask test features.

Command message:

*PFEN ON*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**PASS/FAIL**

**PF\_FAIL\_STOP | PFFS**

Command/Query

**DESCRIPTION**

The PF\_FAIL\_STOP command sets the switch of the “stop on fail” function. This is the same as pressing the “Stop on Fail” button on the menu of PASS/FAIL on the front panel.

The PF\_FAIL\_STOP? query returns the state of the “stop on fail” function.

**COMMAND SYNTAX**

PF\_FAIL\_STOP <state>

<state>:={ON,OFF}

- ON — To monitor the failure waveform, the oscilloscope stops testing and enters the "STOP" state. At this point, the screen displays the last statistical result.(if the display is already open)
- OFF — To monitor the failure waveform, the oscilloscope will continue to test and update the statistics on the screen immediately.

**QUERY SYNTAX**

PF\_FAIL\_STOP?

**RESPONSE FORMAT**

PF\_FAIL\_STOP <state>

**EXAMPLE**

The following command sets “stop on fail” to “off”.

Command message:  
*PFFS OFF*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**PASS/FAIL**

**PF\_OPERATION | PFOF**

Command/Query

**DESCRIPTION**

The PF\_OPERATION command controls to run or stop Pass/Fail test.

The PF\_OPERATION? query returns the operation state of Pass/Fail test.

**COMMAND SYNTAX**

PF\_OPERATION <state>

<state>:={ON,OFF}

**QUERY SYNTAX**

PF\_OPERATION?

**RESPONSE FORMAT**

PF\_OPERATION <state>

**EXAMPLE**

The following command controls to run Pass/Fail test.

Command message:

*PFOF ON*

**RELATED COMMANDS**

PFEN

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**PASS/FAIL**

**PF\_SOURCE | PFSC**

Command/Query

**DESCRIPTION**

The PF\_SOURCE command sets measurement sources for Pass/Fail test.

The PF\_SOURCE? query returns the measurement source for Pass/Fail test.

**COMMAND SYNTAX**

PF\_SOURCE <trace>

<trace>:={C1,C2,C3,C4}

**QUERY SYNTAX**

PF\_SOURCE?

**RESPONSE FORMAT**

PF\_SOURCE <trace>

**EXAMPLE**

The following command sets the measurement source to Channel 1 when Channel 1 is on.

Command message:

*PFSC C1*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**PASS/FAIL**

**PF\_SET | PFST**

Command /Query

**DESCRIPTION**

The PF\_SET command sets the tolerance in the X/Y direction around the selected waveform defined by PFSC for the Pass/Fail feature. The value of the tolerance will be added and subtracted to horizontal/Vertical values of the waveform to determine the boundaries of the mask.

The PF\_SET? query returns the current setting of the  $\Delta X$  tolerance and  $\Delta Y$  tolerance for Pass/Fail.

**COMMAND SYNTAX**

PF\_SET XMASK,<div>,YMASK,<div>

<div>:= 0.04 to 4.0.

**Note:**  
Step value is 0.04.

**QUERY SYNTAX**

PF\_SET?

**RESPONSE FORMAT**

PF\_SET XMASK,<div>,YMASK,<div>

**EXAMPLE**

The following command sets the X mask to 0.4 and the Y mask to 0.52.

Command message:  
*PFST XMASK,0.4,YMASK,0.52*

**RELATED COMMANDS**

PFSC

**PRINT Commands**

**SCDP**



**PRINT**

**SCREEN\_DUMP | SCDP**

Query

**DESCRIPTION**

The SCREEN\_DUMP command captures the screen and returns the data of bmp file.

**QUERY SYNTAX**

SCREEN\_DUMP

**RESPONSE FORMAT**

<bmp header>+<bmp screen data>

**Note:**

You only need to save the returned information in a BMP format file.

**EXAMPLE**

The following step shows how to transfers the screen information as a file named screen.bmp in a Python shell.

**Step 1:** Send the query to get the bmp data.

Query message: *SCDP*

**Step 2:** Create a new bmp file named "screen.bmp".

**Step 3:** Write the data to the file.

**Step 4:** Close the file.

```
>>> bmp_data=sds.ask("SCDP")
>>> bmp_file=open("F:\\screen.bmp", "w")
>>> bmp_file.write(bmp_data)
>>> bmp_file.close()
>>> |
```

See the code in Screen Dump (SCDP) Example

## RECALL Commands

Recall previously saved oscilloscope setups and reference waveforms.

**\*RCL**

**RCPN**

**RECALL**

**\*RCL**  
Command

**DESCRIPTION**

The \*RCL command recalls the complete front-panel setup of the instrument from internal memory, using one of the twenty non-volatile panel setups. This command is opposite to the command \*SAV.

See the command RCPN for recalling the setup from external.

**COMMAND SYNTAX**

\*RCL <setup\_num>

<setup\_num>:= 0 to 20.

**Note:**

- When setup\_num is 0, it will recall the default panel setup.
- As shown below, when the progress is finished, there will be a prompt message.



**EXAMPLE**

When you have stored the instrument setup in No.3, the following command recalls the setup 3.

Command message:

\*RCL 3

**RELATED COMMANDS**

RCPN  
\*SAV

**RECALL**

**RECALL\_PANEL | RCPN**

Command

**DESCRIPTION**

The RECALL\_PANEL command recalls a front-panel setup from the specified-DOS path directory in an external memory device.

See the command “\*RCL” for recalling from internal.

**COMMAND SYNTAX**

RECALL\_PANEL  
DISK,<device>,FILE,'<filename>'  
<device>:= {UDSK}

<filename>:= A waveform file under a legal DOS path.

Models	Description
T3DSO1000(A)	The filename string is up to eight characters, with the extension “.xml”.
Others	The filename string is up to eight characters, with the extension “.set”.

**Note:**

- See models on [page 14](#).
- For T3DSO1000(A) series, the ‘/’ character to define the root directory is not supported.
- As shown below, when the progress is finished, there will be a prompt message.



---

## Digital Oscilloscope Series

- As shown below, if the filename is wrong, there will be a prompt message.



### EXAMPLE

- For T3DSO1000(A) series, when you plug in an U-disk to the oscilloscope, the following command recalls the front-panel setup from a file called "TEST.xml" in root directory of the USB memory device.

Command message:

*RCPN DISK,UDSK,FILE,'TEST.xml'*

- For T3DSO1000(A) series, when you plug in an U-disk to the oscilloscope, the following command recalls the front-panel setup from a file called "TEST.xml" in specified-directory of the USB memory device.

Command message:

*RCPN  
DISK,UDSK,FILE,'/SAVE/TEST.xml'*

### RELATED COMMANDS

STPN  
\*RCL

## REFERENCE Commands

The REFERENCE system controls the reference waveforms.

**REFCL**  
**REFDS**  
**REFLA**  
**REFPO**  
**REFSA**  
**REFSC**  
**REFSR**

---

## Digital Oscilloscope Series

---

### **REFERENCE**

### **REF\_CLOSE | REFCL** Command

#### **DESCRIPTION**

The REF\_CLOSE command closes the Reference function.

#### **COMMAND SYNTAX**

REF\_CLOSE

#### **EXAMPLE**

The following command closes the Reference function.

Command message:

*REFCL*

#### **Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**REFERENCE**

**REF\_DISPLAY | REFDS**

Command /Query

**DESCRIPTION**

The REF\_DISPLAY command enables or disables the current reference channel shown on the screen.

The REF\_DISPLAY? query returns whether the current reference channel shows on the screen.

**COMMAND SYNTAX**

REF\_DISPLAY <state>

<state>:= {ON,OFF}

**Note:**

Only used when the current reference channel has been stored, and the Reference function is enable.

**QUERY SYNTAX**

REF\_DISPLAY?

**RESPONSE FORMAT**

REF\_DISPLAY <state>

**EXAMPLE**

The following command displays the waveform of the current reference channel.

Command message:

*REFDS ON*

**RELATED COMMANDS**

REFCL

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes



**REFERENCE**

**REF\_LOCATION | REFLA**

Command /Query

**DESCRIPTION**

The REF\_LOCATION command selects the current reference channel.

The REF\_LOCATION? query returns the current reference channel.

**COMMAND SYNTAX**

REF\_LOCATION <location>

<location>:= {REFA,REFB,REFC,REFD}

**QUERY SYNTAX**

REF\_LOCATION?

**RESPONSE FORMAT**

REF\_LOCATION <location>

**EXAMPLE**

The following command selects REFA as the current reference channel.

Command message:

*REFLA REFA*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**REFERENCE**

**REF\_POSITION | REFPO**

Command /Query

**DESCRIPTION**

The REF\_POSITION command sets the vertical offset of the current reference channel. This command is only used when the current reference channel has been saved, and the display state is on.

The REF\_POSITION? query returns the vertical offset of the current reference channel.

**COMMAND SYNTAX**

REF\_POSITION <offset>

<offset>:= vertical offset value with unit.

**Note:**

- If there is no unit(V/mV/uV) added, it defaults to be V.
- The range of legal offset varies with the value set by the REFSC command. If you set the offset to a value outside of the legal range, the offset value is automatically set to the nearest legal value.

**QUERY SYNTAX**

REF\_POSITION?

**RESPONSE FORMAT**

REF\_POSITION <offset>

<offset>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

When the Reference function is on, REFB has been saved and the scale is 2 V, the following command sets the current reference channel vertical offset to 0.2 V.

Command message:

*REFPO 0.2V*

**RELATED COMMANDS**

REFSC

**Note:**

The table below shows the availability of command in each oscilloscope series.

**Digital Oscilloscope Series**

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**REFERENCE**

**REF\_SAVE | REFSA**  
Command

**DESCRIPTION**

The REF\_SAVE command saves the waveform (screen range) of the specified source as the reference waveform of the current reference channel to the memory and displays it on the screen.

**COMMAND SYNTAX**

REF\_SAVE

**EXAMPLE**

When the Reference function is on, the REF source is Channel 2, and the REF location is REFA, the following command saves Channel 2 as REFA and displays REFA on screen.

Command message:  
*REFSA*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**REFERENCE**

**REF\_SCALE | REFSC**

Command /Query

**DESCRIPTION**

The REF\_SCALE command sets the vertical scale of the current reference channel. This command is only used when the current reference channel has been stored, and the display state is on.

The REF\_SCALE? query returns the vertical scale of the current reference channel.

**COMMAND SYNTAX**

REF\_SCALE <scale>

<scale>:= 500uV to 10V.

**Note:**

If there is no unit(V/mV/uV) added, it defaults to be V.

**QUERY SYNTAX**

REF\_SCALE?

**RESPONSE FORMAT**

REF\_SCALE <scale>

<scale>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

When the Reference function is on, and REFA has been saved, the following command sets the vertical scale of REFA to 100 mV.

Command message:

*REFSC 100mV*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

**REFERENCE**

**REF\_SOURCE | REFSR**

Command /Query

**DESCRIPTION**

The REF\_SOURCE command sets the reference waveform source.

The REF\_SOURCE? query returns the source of the current reference channel.

**COMMAND SYNTAX**

REF\_SOURCE <source>

<source>:= {C1,C2,C3,C4,MATH}

**QUERY SYNTAX**

REF\_SOURCE?

**RESPONSE FORMAT**

REF\_SOURCE <source>

**EXAMPLE**

When Channel 1 is on, the following command selects Channel 1 as the source of current reference channel.

Command message:  
*REFSR C1*

**Note:**

The table below shows the availability of command in each oscilloscope series.

Model	Valid?
T3DSO2000	no
T3DSO1000(A)	yes

## SAVE Commands

Save oscilloscope setups and waveform data.

**\*SAV**  
**PNSU**  
**STPN**

---

## Digital Oscilloscope Series

---

### **SAVE**

### **\*SAV** Command

#### **DESCRIPTION**

The \*SAV command stores the complete front-panel setup of the instrument in internal memory.

This instruction does not support storing to external temporarily. See the command STPN for external storage.

#### **COMMAND SYNTAX**

\*SAV <setup\_num>

<setup\_num>:= 1 to 20.

#### **Note:**

If there is already a file in the specified location, it will overwrite the original file.

#### **EXAMPLE**

When you want to save the current setup in panel as shown below, the following command saves it to setup No.3.



Command message:

*\*SAV 3*

If you want to recall this setup, send the following command.

Command message:

*\*RCL 3*

#### **RELATED COMMANDS**

STPN  
\*RCL

---



**SAVE**

**PANEL\_SETUP | PNSU**

Command /Query

**DESCRIPTION**

The PANEL\_SETUP command use the encoded data get from “PNSU?” to set the panel setup.

The PNSU? query return the panel setup in binary format from scope.

**COMMAND SYNTAX**

PANEL\_SETUP <binary data>

<binary data>:= A setup previously read by PNSU?

**QUERY SYNTAX**

PANEL\_SETUP?

**RESPONSE FORMAT**

PANEL\_SETUP <binary data>

**EXAMPLE**

The following steps show how to use the query and command to set the panel setup.

**Step 1:** Send the command to set the response format.

Command message:

*CHDR OFF*

**Step 2:** Send the query to get the binary data of setup.

Command message:

*PNSU?*

Response message:

*<binary data>*

**Step 3:** Change the panel setup, and then send the command to restore setup get from step2.

Command message:

*PNSU <binary data>*

**Step 4:** You can also save the data in step 2 to a text file and make it easier to recall later. The following program is used as a reference.

## Digital Oscilloscope Series

---

```
import visa

def main():
    ads = visa.instrument("USB0::0xF4EC::0xEE3B::0123456789::INSTR")
    ads.write("chdr off")
    data=ads.ask("PNSU?")
    f=open("F:\Setup.txt","a")
    f.write(data)
    f.close()
    f=open("F:\Setup.txt","r")
    data = f.read()
    ads.write("PNSU "+str(data))

if __name__=='__main__':
    main()
```

**SAVE**

**STORE\_PANEL | STPN**

Command

**DESCRIPTION**

The STORE\_PANEL command stores the complete front-panel setup of the instrument into a file on the specified-DOS path directory in a USB memory device.

See the command “\*SAV” for internal storage.

**COMMAND SYNTAX**

STORE\_PANEL  
DISK,<device>,FILE,<filename>'

<device>:= {UDSK}

<filename>:= A waveform file under a legal DOS path.

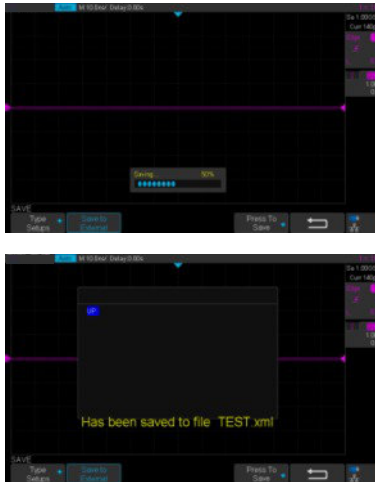
Models	Description
T3DSO1000(A)	The filename string is up to eight characters, with the extension “.xml”.
Others	The filename string is up to eight characters, with the extension “.set”.

**Note:**

- See models on [page 14](#).
- For T3DSO1000(A) series, the ‘/’ character to define the root directory is not supported.
- As shown below, during the execution of the command, a progress bar will appear on the interface. When the progress is finished, there will be a prompt message.

---

## Digital Oscilloscope Series

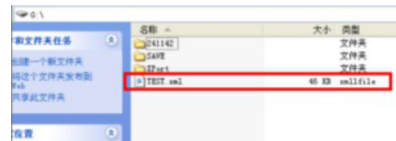


### EXAMPLE

- For T3DSO1000(A) series, the following command saves the current setup to root directory of the USB memory device in a file called “TEST.xml”.

Command message:

*STPN DISK,UDSK,FILE,'TEST.xml'*



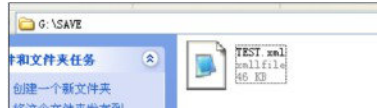
- For T3DSO1000(A) series, the following command saves the current setup to the specified-directory of the USB memory device in a file called “TEST.xml”.

Command message:

*STPN DISK,UDSK,FILE,'/SAVE/TEST.xml'*

## Digital Oscilloscope Series

---



### RELATED COMMANDS

\*SAV  
RCPN

## **STATUS Commands**

IEEE 488.2 defines data structures, commands, and common bit definitions for status reporting. There are also instrument-defined structures and bits. An overview of the oscilloscope's status reporting structure is shown in the following commands.

### **INR?**

**STATUS**

**INR?**

Query

**DESCRIPTION**

The INR? query reads and clears the contents of INternal state change Register (INR). The INR register records the completion of various internal operations and state transitions.

**QUERY SYNTAX**

INR?

**RESPONSE FORMAT**

INR <value>

<value>:= 0 to 65535.

**Note :**

- This query only returns 0 bit and 13 bit.
- See the table **INternal State Register (INR) Structure** as shown below for details.

**EXAMPLE**

The following steps show the change of INR.

**Step 1:** When the trigger mode is single, and there is no signal input, send the query.

Response message:

*INR 0*

**Step 2:** Now, input a signal to trigger. The acquisition mode is Stop. Then, send the query.

Response message:

*INR 1*

**Step 3:** Now, change the trigger mode to Auto. Then, send the query.

Response message:

*INR 8193*

**Step 4:** Now, change the trigger mode to Single. The acquisition mode changes to be Stop. And then, send the query.

Response message:

*INR 8193*

## Digital Oscilloscope Series

---

**Step 5:** After sending the query in step 4, send the query again.

Response message:  
*INR 0*

**Step 6:** After step 2, not to input the signal, change the trigger mode to single. And then, send the query.

Response message:  
*INR 8192*

### Internal State Register (INR) Structure

Bit	Bit value	Description
15	---	Not used (always 0)
14	---	Not used (always 0)
13	8192	Trigger is ready
12	4096	Pass/Fail test detected desired outcome
11	2048	Waveform processing has terminated in Trace D
10	1024	Waveform processing has terminated in Trace C
9	512	Waveform processing has terminated in Trace B
8	256	Waveform processing has terminated in Trace A
7	128	A memory card, floppy or hard disk exchange has been detected
6	64	Memory card, floppy or hard disk has become full in -AutoStore Fill mode
5	---	Not use(always 0)
4	16	A segment of a sequence waveform has been acquired
3	8	A time-out has occurred in a data block transfer
2	4	A return to the local state is detected
1	2	A screen dump has terminated
0	1	A new signal has been acquired



## SYSTEM Commands

The SYSTEM subsystem commands control basic system functions of the oscilloscope.

**\*CAL?**  
**BUZZ**  
**CONET**  
**SCSV**

**SYSTEM**

**\*CAL?**  
Query

**DESCRIPTION**

The \*CAL? query starts the user calibration procedure and return a response.

The user calibration can quickly make the oscilloscope achieve the best working state, in order to obtain the most accurate measurement value.

All function keys have been disabled during the self calibration process.

Before starting the user calibration procedure, you must disconnect anything from inputs.

**QUERY SYNTAX**

\*CAL?

**RESPONSE FORMAT**

\*CAL 0

- 0 — Calibration successful.

**EXAMPLE**

The following query starts a self-calibration.

Query message:

*\*CAL?*

Response message:

*\*CAL 0*

**SYSTEM**

**BUZZER | BUZZ**

Command /Query

**DESCRIPTION**

The BUZZER command enables or disables the buzzer.

The BUZZER? query returns the switch state of the buzzer.

**COMMAND SYNTAX**

BUZZER <state>

<state>:= {ON,OFF}

**QUERY SYNTAX**

BUZZER?

**RESPONSE FORMAT**

BUZZER <state>

**EXAMPLE**

The following command enables the oscilloscope buzzer.

Command message:

*BUZZ ON*

**SYSTEM**

**COMM\_NET | CONET**

Command /Query

**DESCRIPTION**

The COMM\_NET command sets the IP address of the oscilloscope's internal network interface.

When using this command, DHCP should be off.

The COMM\_NET? query returns the IP address of the oscilloscope's internal network interface.

**COMMAND SYNTAX**

COMM\_NET  
<ip\_add0>,<ip\_add1>,<ip\_add2>,<ip\_add3>  
< ip\_add0 >:= 1 to 223(except 127).  
< ip\_add1 >:= 0 to 255.  
< ip\_add2 >:= 0 to 255.  
< ip\_add3 >:= 0 to 255.

**QUERY SYNTAX**

COMM\_NET?

**RESPONSE FORMAT**

COMM\_NET  
<ip\_add0>,<ip\_add1>,<ip\_add2>,<ip\_add3>

**EXAMPLE**

The following command sets the IP address to 10.11.0.230.

Command message:  
*CONET 10,11,0,230*

**SYSTEM**

**SCREEN\_SAVE | SCSV**

Command/Query

**DESCRIPTION**

The SCREEN\_SAVE command controls the automatic screen saver, which automatically shuts down the internal color monitor after a preset time.

The SCREEN\_SAVE? query returns whether the automatic screen saver feature is on.

**Note:**

When the screen saver is enabled, the oscilloscope is still fully functional.

**COMMAND SYNTAX**

SCREEN\_SAVE <time>

<time>:= {OFF,1MIN,5MIN,10MIN,30MIN,60MIN}

- OFF — Do not use screen saver.
- Others — When the oscilloscope enters the idle state and holds for the specified time, screen saver will be enabled.

**QUERY SYNTAX**

SCREEN\_SAVE?

**RESPONSE FORMAT**

SCREEN\_SAVE <time>

**EXAMPLE**

The following command sets the automatic screen saver to 10 minutes.

Command message:

*SCSV 10MIN*

## **TIMEBASE Commands**

The TIMEBASE subsystem commands control the horizontal (X-axis) functions. The time per division, delay, and reference can be controlled for the main and window (zoomed) time bases.

**TDIV**  
**TRDL**  
**HMAG**  
**HPOS**

<b>TIMEBASE</b>	<b>TIME_DIV   TDIV</b> Command/Query
<b>DESCRIPTION</b>	<p>The TIME_DIV command sets the horizontal scale per division for the main window.</p> <p>The TIME_DIV? query returns the current horizontal scale setting in seconds per division for the main window.</p>
<b>COMMAND SYNTAX</b>	<p>TIME_DIV &lt;value&gt;</p> <p>&lt;value&gt;:={1NS,2NS,5NS,10NS,20NS,50NS,100NS,200NS,500NS,1US,2US,5US,10US,20US,50US,100US,200US,500US,1MS,2MS,5MS,10MS,20MS,50MS,100MS,200MS,500MS,1S,2S,5S,10S,20S,50S,100S}</p> <ul style="list-style-type: none"><li>• NS — for nanoseconds.</li><li>• US — for microseconds.</li><li>• MS — for milliseconds.</li><li>• S — for seconds.</li></ul> <p><b>Note:</b> The range of value varies from the models. See the data sheet for details.</p>
<b>QUERY SYNTAX</b>	TIME_DIV?
<b>RESPONSE FORMAT</b>	TIME_DIV <value> <value>:= Numerical value in E-notation with SI unit.
<b>EXAMPLE</b>	<p>The following command sets the horizontal scale to 500 <math>\mu</math>s.</p> <p>Command message: <i>TDIV 500US</i></p>
<b>RELATED COMMANDS</b>	TRDL HMAG HPOS

**TIMEBASE**

**TRIG\_DELAY | TRDL**

Command /Query

**DESCRIPTION**

The TRIG\_DELAY command sets the time interval between the trigger event and the horizontal center point on the screen. The maximum position value depends on the time/division settings.

- Pre-trigger acquisition — Data acquired before the trigger occurs. Negative trigger delays must be given in seconds.
- Post-trigger acquisition — Data acquired after the trigger has occurred.

The TRIG\_DELAY? query returns the current time from the trigger to the horizontal center point in seconds.

**COMMAND SYNTAX**

TRIG\_DELAY <delay>

<delay>:= time value with unit.

**Note:**

- The range of delay is related to the time base. See the data sheet for details.
- If you set the delay to a value outside of the legal range, the delay value is automatically set to the nearest legal value.

**QUERY SYNTAX**

TRIG\_DELAY?

**RESPONSE FORMAT**

TRIG\_DELAY <value>

Model	Format of <skew>
T3DSO1000(A)	Numerical value in E-notation with SI unit, such as 1.00E-04S.
others	Numerical value with measurement unit and physical



## Digital Oscilloscope Series

---

### EXAMPLE

	unit, such as 3.58ns.
--	-----------------------

When the time base is 1us/div, the following command sets the trigger delay to -4.8us (pre trigger).

Command message:

*TRDL -4.8US*

### RELATED COMMANDS

TDIV

**TIMEBASE**

**HOR\_MAGNIFY | HMAG**

Command /Query

**DESCRIPTION**

The HOR\_MAGNIFY command sets the zoomed (delayed) window horizontal scale (seconds/div). The main sweep scale determines the range for this command. The maximum value is the TDIV value.

The HOR\_MAGNIFY? query returns the current zoomed window scale setting.

**COMMAND SYNTAX**

Format 1:  
HOR\_MAGNIFY <value>

<value >:= {1NS,2NS,5NS,10NS,20NS,50NS,100NS,200NS,500NS,1US,2US,5US,10US,20US,50US,100US,200US,500US,1MS,2MS,5MS,10MS,20MS}

The range of value is related to the current time base. It is from 1NS to the current time base.

Format 2:  
HOR\_MAGNIFY <factor>

<factor>:= 1 to 2,000,000.

The range of <factor> is related to the current time base and the range of the time base.

**Note:**

The table on next page shows the available format in each oscilloscope series.

**QUERY SYNTAX**

HOR\_MAGNIFY?

**RESPONSE FORMAT**

HOR\_MAGNIFY <value>  
<value>:= Numerical value in E-notation with SI unit.

HOR\_MAGNIFY <factor>

**EXAMPLE**

For T3DSO1000(A) series, when the time base is 1ms/div, and Zoom function is on, the following

## Digital Oscilloscope Series

---

command sets the zoomed (delayed) window horizontal scale to 1US.

Command message:

*HMAG 1US*

### RELATED COMMANDS

TDIV

### Format in Each Oscilloscope Series

Model	Command Format
T3DSO2000	Format 2
T3DSO1000(A)	Format 1

**TIMEBASE**

Command /Query

**HOR\_POSITION | HPOS**

**DESCRIPTION**

The HOR\_POSITION command sets the horizontal position in the zoomed (delayed) view of the main sweep. The main sweep range and the main sweep horizontal position determine the range for this command. The value for this command must keep the zoomed view window within the main sweep range.

The HOR\_POSITION? query the current horizontal window position setting in the zoomed view.

**COMMAND SYNTAX**

Format 1:  
HOR\_POSITION <position>

<position>:= time value with unit.

**Note:**

- You need add the time unit(s/ms/us/ns) to the position. If there is no unit added, it defaults to be S.

- The range of position is related to the main sweep range and the main sweep horizontal position. The range after magnifying which beyond the screen could display, and it will be adjusted to the proper value.

Format 2:  
HOR\_POSITION <factor\_div>

< factor\_div>:= the factor of zoomed time base.

**Note:**

The table on next page shows the available format in each oscilloscope series.

**QUERY SYNTAX**

HOR\_POSITION?

**RESPONSE FORMAT**

HOR\_POSITION <position>  
<position>:= Numerical value in E-notation with SI unit.

## Digital Oscilloscope Series

---

HOR\_POSITION <factor\_div>

### EXAMPLE

For T3DSO1000(A) series, when the time base is 10 us/div, the horizontal position is 0, Zoom function is on, and the zoomed scale is 5 us. The range of zoom position is from -35 us to 35 us. The following command sets the zoom position to 100 ns.

Command message:  
*HPOS 100ns*

### RELATED COMMANDS

HMAG  
TDIV  
TRDL

### Format in Each Oscilloscope Series

Model	Command Format
T3DSO2000	Format 2
T3DSO1000(A)	Format 1

## TRIGGER Commands

The TRIGGER subsystem controls the trigger modes and parameters for each trigger type.

**SET50**  
**TRCP**  
**TRLV**  
**TRLV2**  
**TRMD**  
**TRPA**  
**TRSE**  
**TRSL**  
**TRWI**

**TRIGGER**

**SET50**  
Command

**DESCRIPTION**

The SET50 command automatically sets the trigger levels to center of the trigger source waveform.

When High and Low (dual) trigger levels are used (as Runt triggers, for example), this command has no effect.

**COMMAND SYNTAX**

SET50

**EXAMPLE**

When the trigger type is edge and the trigger source is Channel 1, the following command sets the trigger level to the center of Channel 1.

Command message:  
*SET50*

**RELATED COMMANDS**

TRLV

<b>TRIGGER</b>	<b>TRIG_COUPLING   TRCP</b> Command /Query
<b>DESCRIPTION</b>	<p>The TRIG_COUPLING command sets the input coupling for the selected trigger sources.</p> <p>The TRIG_COUPLING? query returns the trigger coupling of the selected source.</p>
<b>COMMAND SYNTAX</b>	<pre>&lt;trig_source&gt;:TRIG_COUPLING &lt;trig_coupling&gt;</pre> <pre>&lt;trig_source&gt;:={C1,C2,C3,C4,EX,EX5}</pre> <pre>&lt;trig_coupling&gt;:={AC,DC,HFREJ,LFREJ}</pre> <ul style="list-style-type: none"> <li>• AC — AC coupling block DC component in the trigger path, removing dc offset voltage from the trigger waveform. Use AC coupling to get a stable edge trigger when your waveform has a large dc offset.</li> <li>• DC — DC coupling allows dc and ac signals into the trigger path.</li> <li>• HFREJ — HFREJ coupling places a low-pass filter in the trigger path.</li> <li>• LFREJ — LFREJ coupling places a high-pass filter in the trigger path.</li> </ul>
<b>QUERY SYNTAX</b>	<pre>&lt;trig_source&gt;:TRIG_COUPLING?</pre>
<b>RESPONSE FORMAT</b>	<pre>&lt;trig_source&gt;:TRIG_COUPLING &lt;trig_coupling&gt;</pre>
<b>EXAMPLE</b>	<p>The following command sets the coupling mode of the trigger source Channel 2 to AC.</p> <p>Command message: <i>C2:TRCP AC</i></p>
<b>RELATED COMMANDS</b>	TRSE



**TRIGGER**

**TRIG\_LEVEL | TRLV**

Command /Query

**DESCRIPTION**

The TRIG\_LEVEL command sets the trigger level voltage for the active trigger source.

When there are two trigger levels to set, this command is used to set the higher trigger level voltage for the specified source. TRLV2 is used to set the lower trigger level voltage.

The TRIG\_LEVEL? query returns the trigger level of the current trigger source.

**COMMAND SYNTAX**

<trig\_source>:TRIG\_LEVEL <trig\_level>

<trig\_source>:={C1,C2,C3,C4,EX,EX5}

<trig\_level>:= -4.5\*DIV to 4.5\*DIV for internal triggers.

<trig\_level>:= -3\*DIV to 3\*DIV for external triggers.

**Note:**

- You need to add the volt unit(V/mV) to the trig\_level. If there is no unit added, it defaults to volts (V).
- An out-of-range value will be adjusted to the closest legal value.

**QUERY SYNTAX**

<trig\_source>:TRIG\_LEVEL?

**RESPONSE FORMAT**

<trig\_source>:TRIG\_LEVEL <trig\_level>

<trig\_level>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

When the vertical scale of Channel 3 is 200 mV, and the trigger source is Channel 3, the following command sets the trigger level of Channel 3 to 52.00 mV.

Command message:

*C3:TRLV52MV*

**Digital Oscilloscope Series**

---

**RELATED COMMANDS**

TRSE  
TRLV2

**TRIGGER**

**TRIG\_LEVEL2 | TRLV2**

Command /Query

**DESCRIPTION**

The TRIG\_LEVEL2 command sets the lower trigger level voltage for the specified source.

Higher and lower trigger levels are used with runt /slope triggers.

The TRIG\_LEVEL2? query returns the lower trigger level voltage for the specified source.

**COMMAND SYNTAX**

<trig\_source>:TRIG\_LEVEL2 <trig\_level>

<trig\_source>:= {C1,C2,C3,C4}

<trig\_level>:= -4.5\*DIV to 4.5\*DIV.

**Note:**

- You need add the volt unit(V/mV) to the trig\_level. If there is no unit added, it defaults to volts (V).
- An out-of-range value will be adjusted to the closest legal value.

**QUERY SYNTAX**

<trig\_source>:TRIG\_LEVEL2?

**RESPONSE FORMAT**

<trig\_source>:TRIG\_LEVEL2 <trig\_level>

<trig\_level>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

When the trigger type is slope, the following steps set the high trigger level of Channel 2 to 3.5 V, and the low trigger level of Channel 2 to 800 mV.

**Step 1:** Send the command to set high trigger level.

Command message:

*C2:TRLV 3.5V*

**Step 2:** Send the command to set low trigger level.

Command message:

---

**Digital Oscilloscope Series**

*C2:TRLV2 800mV*

**RELATED COMMANDS**

TRSE  
TRLV

**TRIGGER**

Command /Query

**TRIG\_MODE | TRMD**

**DESCRIPTION**

The TRIG\_MODE command selects the trigger sweep mode.

The TRIG\_MODE? query returns the current trigger sweep mode.

**COMMAND SYNTAX**

TRIG\_MODE <mode>

<mode>:= {AUTO,NORM,SINGLE,STOP}

- **AUTO** — When AUTO sweep mode is selected, the oscilloscope begins to search for the trigger signal that meets the conditions. If the trigger signal is satisfied, the running state on the top left corner of the user interface shows Trig'd, and the interface shows stable waveform.

Otherwise, the running state always shows Auto, and the interface shows unstable waveform.

- **NORM** — When NORMAL sweep mode is selected, the oscilloscope enters the wait trigger state and begins to search for trigger signals that meet the conditions.

If the trigger signal is satisfied, the running state shows Trig'd, and the interface shows stable waveform.

Otherwise, the running state shows Ready, and the interface displays the last triggered waveform (previous trigger) or does not display the waveform (no previous trigger).

- **SINGLE** — When SINGLE sweep mode is selected, the backlight of SINGLE key lights up, the oscilloscope enters the waiting trigger state and begins to search for the trigger signal that meets the conditions.

If the trigger signal is satisfied, the running state shows Trig'd, and the interface shows stable waveform. Then, the oscilloscope stops scanning, the RUN/STOP key is red light, and the running status shows Stop.

---

## Digital Oscilloscope Series

Otherwise, the running state shows Ready, and the interface does not display the waveform.

- STOP — STOP is a part of the option of this command, but not a trigger mode of the oscilloscope.

### QUERY SYNTAX

TRIG\_MODE?

### RESPONSE FORMAT

TRIG\_MODE <mode>

### EXAMPLE

The following command sets the trigger mode to Normal.

Command message:

*TRMD NORM*

### RELATED COMMANDS

ARM  
STOP

**TRIGGER**

**TRIG\_PATTERN | TRPA**

Command /Query

**DESCRIPTION**

The TRIG\_PATTERN command specifies the channel values to be used in the pattern trigger and sets the condition of the pattern trigger.

The TRIG\_PATTERN? query returns channel values and the condition of the pattern trigger.

**COMMAND SYNTAX**

TRIG\_PATTERN  
<source>,<status>[,<source>,<status>[,<source>,<status>[,<source>,<status>[,<source>,<status>]]]],STATE,<condition>

< source >:={C1,C2,C3,C4}

<status>:={X,L,H}

< condition >:={AND,OR,NAND,OR}

- X — Ignore this channel. When all channels are set to X, the oscilloscope will not trigger.
- L — Low level.(lower than the threshold level of the channel)
- H — High level.(higher than the threshold level of the channel)

**Note:**

The status of source can only be set when the source is on.

**QUERY SYNTAX**

TRIG\_PATTERN?

**RESPONSE FORMAT**

TRIG\_PATTERN  
<source>,<status>,<source>,<status>,<source>,<status>,<source>,<status>,<source>,<status>,STATE,<condition>

**EXAMPLE**

When the trigger type is Pattern, and Channel 2 & Channel 3 are on, the following command sets the Channel 2 and Channel 3 to low and the condition to AND.

Command message:

*TRPA C2,L,C3,L,STATE,AND*

**TRIGGER**

**TRIG\_SELECT | TRSE**

Command /Query

**DESCRIPTION**

The TRIG\_SELECT command selects the condition that will trigger the acquisition of waveforms.

Depending on the trigger type, additional parameters must be specified. These additional parameters are grouped in pairs. The first in the pair names the variable to be modified, while the second gives the new value to be assigned. Pairs may be given in any order and restricted to those variables to be changed.

The TRIG\_SELECT? query returns the current trigger condition.

Parameter		description	
SLEW	Slope	IL	Interval larger
GLIT	Glitch/ Pulse	IS	Interval smaller
INTV	Interval	I2	Interval in range
DROP	Dropout	I1	Interval out of range
SR	Source	PL	Pulse larger
TI	Time	PS	Pulse smaller
HT	Hold type/ Limit range	P2	Pulse in range
HV	Hold value/ Limit value	P1	Pulse out of range

**COMMAND SYNTAX  
(FOR ALL BUT TV)**

TRIG\_SELECT  
<trig\_type>,SR,<source>,HT,<hold\_type>,H  
V,<hold\_value1>[.HV2,<hold\_value2>]



<trig\_type>:={EDGE,SLEW,GLIT,INTV,RUN  
T,DROP}

<source>:={C1,C2,C3,C4,LINE,EX,EX5}

**Note:**

LINE/EX/EX5 can only be selected when the trigger type is Edge.

<hold\_type>:={TI,OFF} for EDGE trigger.

<hold\_type>:={TI} for DROP trigger.

<hold\_type>:={PS,PL,P2,P1} for GLIT/RUNT  
trigger.

<hold\_type>:={IS,IL,I2,I1} for SLEW/INTV  
trigger.

<hold\_value1>:= a time value with unit.

<hold\_value2>:= a time value with unit.

**Note:**

- If there is no unit(S/mS/uS/nS) added, it defaults to be S.
- The range of hold\_values varies from trigger types. [80nS, 1.5S] for Edge trigger, and [2nS, 4.2S] for others.

**QUERY SYNTAX**

TRIG\_SELECT?

**RESPONSE FORMAT**

TRIG\_SELECT

<trig\_type>,SR,<source>,HT,<hold\_type>,H  
V,<hold\_value1>[,HV2,<hold\_value2>]

**EXAMPLE**

- When you want to set trigger type to Edge, trigger source to Channel 1, hold type to TIME, and the time value to 1.43uS, the following comes true.

Command message:

*TRSE EDGE,SR,C1,HT,TI,HV,1.43uS*

- When you want to set trigger type to Pulse, trigger source to Channel 2, limit range to [5nS, 1uS], the following comes true.

Command message:

## Digital Oscilloscope Series

*TRSE GLIT,SR,C2,HT,P2,HV,5nS,HV2,1uS*

- When you want to set trigger type to Dropout, trigger source to Channel 4, overtime value to 2.8 mS , the following comes true.

Command message:

*TRSE DROP,SR,C4,HT,TI,HV,2.8mS*

### TV COMMAND SYNTAX

TRIG\_SELECT  
 <trig\_type>,SR,<source>,STAN,<standard>,SYNC,<sync\_type>[,LINE,<line>[,FLD,<field>]]

Parameter description	
STAN	Standard
FLD	field
CUST	Custom

<trig\_type>:= {TV}

<source>:= {C1,C2,C3,C4}

<standard>:= {NTSC,PAL,720P/50,720P/60,1080P/50,1080P/60,1080I/50,1080I/60, CUST}

<line>:=allows triggering on a specific line of video. The line number limits vary with the standard and mode, as shown in the following table.

#### TV Trigger Line Number Limits

stand ard	Mode		
	Line	Field 1	Field 2
NTSC		1~26 3	1 to 262
PAL		1 to 313	1 to 312
720P/ 50	1 to 750		
720P/ 60	1 to 750		
1080P /50	1- 1125		
1080P /60	1- 1125		

## Digital Oscilloscope Series

1080I/50		1 to 563	1 to 562
1080I/60		1 to 563	1 to 562
CUST	1 to number of Lines		

<field>:= [1,2]  
for NTSC/PAL/1080I/50/1080I/60

<field>:=1 to <field\_count>for CUST.

<field\_count>:=1 to 8 depending on the interlace.

**Note:**  
Field can only be selected when the standard is NTSC/PAL/1080I/50/1080I/60/CUST.

### TV QUERY SYNTAX

TRIG\_SELECT?

### TV RESPONSE FORMAT

TRIG\_SELECT

<trig\_type>,<SR>,<source>,<STAN>,<standard>,<SYNC>,<sync\_type>[,<LINE>,<line>[,<FLD>,<field>]]

### TV EXAMPLE

- When you want to set trigger type to Video, trigger source to Channel 1, standard to NTSC, and SYNC to ANY, the following comes true.

Command message:

*TRSE TV,SR,C1,STAN,NTSC,SYNC,ANY*

- When you want to set trigger type to Video, trigger source to Channel 1, standard to PAL, Line to 300, and Field to 2, the following comes true.

Command message:

*TRSE TV,SR,C1,STAN,PAL,SYNC,SELECT,LINE,300,FLD,2*

- When you want to set trigger type to Video, trigger source to Channel 2, standard to 1080P/50, and Line to 200, the following

## Digital Oscilloscope Series

---

comes true.

Command message:

*TRSE*

*TV,SR,C2,STAN,1080P/50,SYNC,SELECT,LI*

*NE,200*

**TRIGGER**

**TRIG\_SLOPE | TRSL**

Command /Query

**DESCRIPTION**

The TRIG\_SLOPE command sets the trigger slope of the specified trigger source.

The TRIG\_SLOPE? query returns the trigger slope of the selected source.

**COMMAND SYNTAX**

<trig\_source>:TRIG\_SLOPE <trig\_slope>

<trig\_source>:={C1,C2,C3,C4,EX,EX5}

<trig\_slope>:={NEG,POS,WINDOW} for edge trigger.

<trig\_slope>:={NEG,POS} for other trigger.

- NEG — falling edge.
- POS — rising edge.
- WINDOW — altering edge.

**QUERY SYNTAX**

<trig\_source>:TRIG\_SLOPE?

**RESPONSE FORMAT**

<trig\_source>:TRIG\_SLOPE <trig\_slope>

**EXAMPLE**

The following command sets the trigger slope of Channel 2 to negative.

Command message:

*C2:TRSL NEG*

**RELATED COMMANDS**

TRSE

**TRIGGER**

**TRIG\_WINDOW | TRWI**

Command /Query

**DESCRIPTION**

The TRIG\_WINDOW command sets the relative height of the two trigger line of the trigger window type.

**Note:**

This command is only valid when the window type is relative.

The TRIG\_WINDOW? query returns relative height of the two trigger line of the trigger window type.

**COMMAND SYNTAX**

TRIG\_WINDOW <value>

<value>: 0 to 9\*DIV when the center level is 0.

**Note:**

- You need add the volt unit(V/mV) to the value. If there is no unit added, it defaults to be V.
- The range of value is related to the center value of the level.

**QUERY SYNTAX**

TRIG\_WINDOW?

**RESPONSE FORMAT**

TRIG\_WINDOW <value>  
<value>:= Numerical value in E-notation with SI unit.

**EXAMPLE**

When the window type is relative, and the center level is 1 V, the following command sets the relative height of the two trigger line to 2 V.

Command message:

*TRWI 2V*

**RELATED COMMANDS**

TRLV  
TRLV2

## WAVEFORM Commands

The WAVEFORM subsystem is used to transfer data to a controller from the oscilloscope waveform memory.

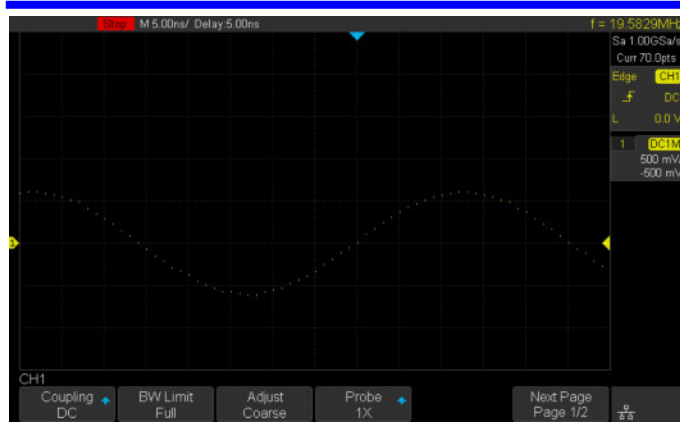
The waveform record is actually contained in two portions: the preamble and waveform data. The waveform record must be read from the oscilloscope by the controller using two separate commands. The waveform data is the actual data acquired for each point in the specified source. The preamble contains the information for interpreting the waveform data.

**WF?**  
**WFSU**

<b>WAVEFORM</b>	<b>WAVEFORM?   WF?</b> Query
<b>DESCRIPTION</b>	<p>The WAVEFORM? query transfers a waveform from the oscilloscope to the controller.</p> <p><b>Note:</b> The format of the waveform data depends on the current settings specified by the last WFSU command.</p>
<b>QUERY SYNTAX</b>	<p><code>&lt;trace&gt;:WAVEFORM? &lt;section&gt;</code></p> <p><code>&lt;trace&gt;:={C1,C2,C3,C4,MATH,D0,D1,D2,D3,D4,D5,D6,D7,D8,D9,D10,D11,D12,D13,D14,D15}</code></p> <ul style="list-style-type: none"><li>• C[X] — Analog channel.</li><li>• D[X] — Digital channel. Only valid for T3DSO1000(A) series.</li><li>• MATH — Valid except for the FFT waveform and only valid for T3DSO1000(A) series.</li></ul> <p><code>&lt;section&gt;:={DAT2}</code></p> <ul style="list-style-type: none"><li>• DAT2 — Return the main data include the head, the wave data and the ending flag. The length of data is current memory depth.</li></ul>
<b>RESPONSE FORMAT</b>	<code>&lt;trace&gt;:WAVEFORM &lt;data block&gt;</code>
<b>RELATED COMMANDS</b>	WFSU
<b>EXAMPLE</b>	<p>For T3DSO1000(A) series, the following steps show how to use the command to reconstitute the display of waveform.</p> <p>For analog channel waveform:</p>



## Digital Oscilloscope Series



**Step 1:** Send the query to get the data of waveform.

Query message:

*C1:WF? DAT2*

Response message:

The head of message: C1:WF ALL. These are followed by the string #9000000070, the beginning of a binary block in which nine ASCII integers are used to give the length of the block (70 bytes). After the length of block, is beginning of the wave data. "0A 0A" means the end of data.

Data	Description
43 31 3A 57 46 20 41 4C 4C 2C 23 39 30 30 30 30	C1:WF ALL, #90000
30 30 30 37 30 02 03 03 03 03 03 01 00 FE FC F9	00070.....
F7 F3 F0 ED E9 E6 E3 DF DC D9 D6 D3 D1 CF CE CD	.....
CC CC CC CD CE CF D1 D4 D6 D9 DC E0 E2 E6 EA ED	.....
F1 F4 F7 FA FC FE 00 02 02 03 03 03 02 01 00 FE	.....
FC F9 F6 F3 F0 ED EA E6 E2 DF DC 0A 0A	.....

**Step 2:** Calculate the voltage value corresponding to the data point.

Using the formula: voltage value (V) = code value \*(vdiv /25) - voffset.

code value: The decimal of wave data.

**Note:** If the decimal is greater than “127”, it should minus 255. Then the value is code value. Such as the wave data is “FC” convert to decimal is “252”. So the code value is  $252-255 = -3$ .

vdiv: The Volts/div value.

voffset: The voltage position value.

The picture above as an example:

Send command *CI:VDIV?*  
Return *CI:VDIV 5.00E-01V*.

Get the current Volts/div values:  $vdiv = 0.5V$ .

Send command *CI:OFST?*  
Return *CI:OFST -5.00E-01V*.

Get the current voltage position values:  $voffset = -0.5V$ .

According to the wave data, we can know the first point of waveform is the 22th data “02”, convert to decimal is “2” (Hexadecimal converted to decimal).

The first point of wave data voltage value =  $2*(0.5/25)-(-0.5) = 0.54V$ .

**Step 3:** Calculate the time value of the data point.

Using the formula:  $\text{time value(S)} = -(\text{timebase}*\text{grid}/2)$ .

timebase: The timebase value.

grid: The grid numbers in horizontal direction.

The picture above as an example:

## Digital Oscilloscope Series

Send command *TDIV?*  
Return *TDIV 5.00E-09S.*

Get the current timebase: timebase = 5.00E-09S.

The time value of the first data point: time value = - (5.00E-09\*14/2) = -35.00E-09(s) = -35(ns).

Send command *SARA?*  
Return *SARA 1.00E+09Sa/s.*

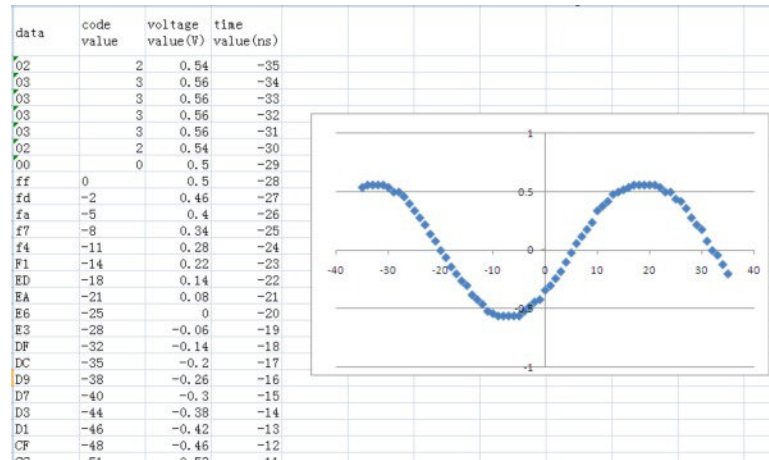
Get the current sampling rate: sampling rate= 1.00GSa/s.

The time interval: time inter = 1/ sampling rate = 1ns.

So the time value of the second data point: value = -35ns+1ns = -34ns.

The following are two ways of waveform reconstruction:

Use Excel to reconstruct the waveform:

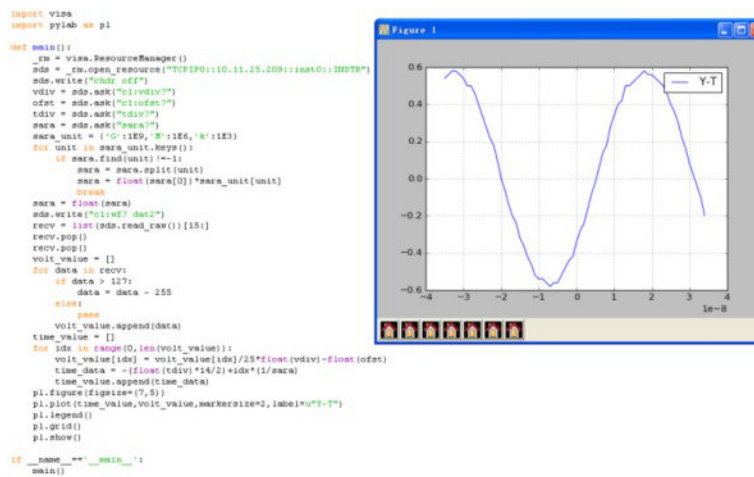


Use python to reconstruct the waveform: (See the code in [Read Waveform](#))

## Digital Oscilloscope Series

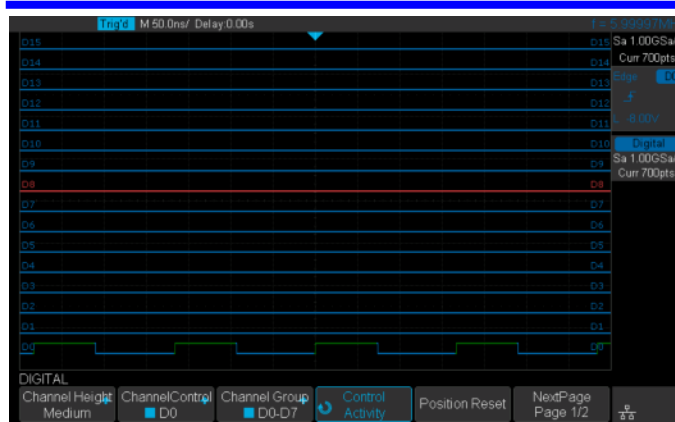
### Data (WF) Example)

Note: If you want the command return the “numerical” data type only (i.e. return as “1.00E+09” when send the command “SARA?”), send the command “CHDR OFF” at the first. See CHDR for details.



For digital channel waveform:

## Digital Oscilloscope Series



**Step 1:** Send the query to get the data of waveform.

Query message:  
*D0:WF? DAT2*

Response message:

The head of message: *D0:WF ALL, #900000700*. These are followed by the string *#900000700*, the beginning of a binary block in which nine ASCII integers are used to give the length of the data (700 points). For digital, one bit represents a data point, so there are 88 bytes. After the length of block, is beginning of the wave data. "0A 0A" means the end of data.

Data	Description
44 30 3A 57 46 20 41 4C 4C 2C 23 39 30 30 30 30	D0:WF ALL, #900000
30 30 37 30 30 00 00 80 FF FF FF FF FF FF FF FF	00700.....
FF FF 00 00 00 00 00 00 00 00 00 00 C0 FF FF FF	.....
FF FF FF FF FF FF 7F 00 00 00 00 00 00 00 00	.....
00 E0 FF FF FF FF FF FF FF FF FF 3F 00 00 00 00	.....
00 00 00 00 00 00 F8 FF FF FF FF FF FF FF FF	.....
0F 00 00 00 00 00 00 00 00 00 FC FF 0A 0A	.....

**Step 2:** Covert to the high (1) and low (0) corresponding to the data point.

According to the wave data, we can know the first eight points of

## Digital Oscilloscope Series

---

waveform is the 22th byte “00”, convert to binary is “00000000” (Hexadecimal converted to binary (LSB)).

**Step 3:** Calculate the time value of the data point.

Using the formula:  $\text{time value(S)} = - (\text{timebase} * \text{grid} / 2)$ .

timebase: The timebase value.

grid: The grid numbers in horizontal direction.

The picture above as an example:

Send command *TDIV?*

Return *TDIV 5.00E-08S*.

Get the current timebase:  $\text{timebase} = 5.00\text{E-}08\text{S}$ .

The time value of the first data point:  $\text{time value} = - (5.00\text{E-}08 * 14 / 2) = - 3.50\text{E-}07(\text{s}) = -350(\text{ns})$ .

Send command *DI:SARA?*

Return *DI:SARA 1.00E+09Sa/s*.

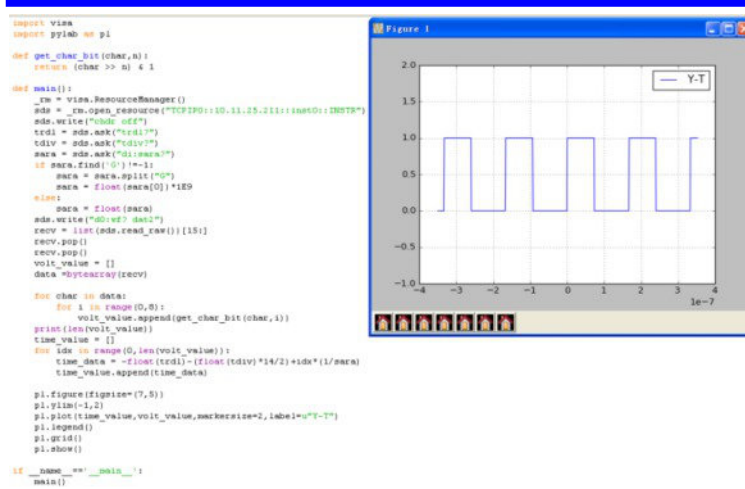
Get the current sampling rate:  $\text{sampling rate} = 1\text{GSa/s}$ .

The time interval:  $\text{time inter} = 1 / \text{sampling rate} = 1\text{ns}$ .

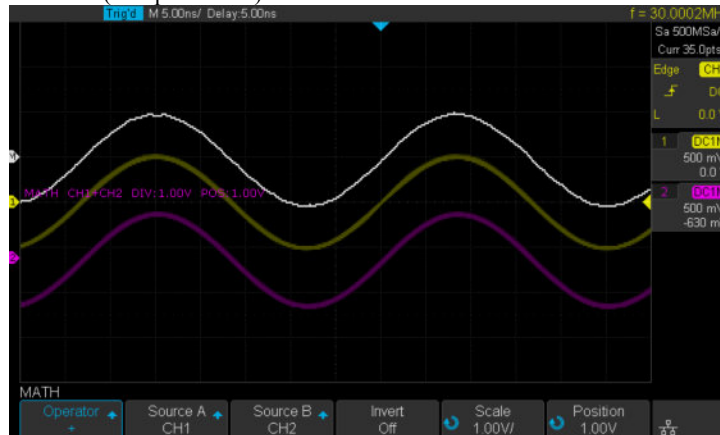
So the time value of the second data point:  $\text{value} = -350\text{ns} + 1\text{ns} = -349\text{ns}$ .

Use python to reconstruct the waveform: (See the code in [Read Waveform Data of Digital Example](#))

## Digital Oscilloscope Series



For math (except for FFT) waveform:



Step 1: Send the query to get the data of waveform.

Query message:

*MATH:WF? DAT2*

Response message:

The head of message: *MATH:WF ALL*. These are followed by the string #900000700, the beginning of a binary block in which nine ASCII integers are used to give the length of the block (700 bytes). The point number is 700 with interpolation. After the length of block, is beginning of the wave data. "0A 0A" means the end of data.

Data	Description
4D 41 54 48 3A 57 46 20 41 4C 4C 2C 23 39 30 30	MATH:WF ALL, #900
30 30 30 30 37 30 30 FF FF FF 00 00 00 00 00 01	0000700.....
01 01 01 02 02 02 03 03 03 03 04 04 05 05 06 06	.....
06 06 06 07 07 07 07 08 08 08 09 09 0A 0A 0B 0B	.....
0C 0C 0D 0D 0D 0E 0E 0E 0F 0F 10 10 10 11 11 12	.....
12 13 13 14 14 15 16 16 16 .....	.....
03 03 03 03 04 04 04 04 05 05 05 06 06 07 07 07	.....
08 0A 0A	...

**Step 2:** Calculate the voltage value corresponding to the data point.

Using the formula: voltage value (V) = code value \*(vdiv /25).

code value: The decimal of wave data. Different from the code of analog channel waveform, it contains the offset.

**Note:** If the decimal is greater than "127", it should minus 255. Then the value is code value. Such as the wave data is "FC" convert to decimal is "252". So the code value is 252-255 = -3.

vdiv: The Volts/div value of math.

The picture above is an example:

Send command *MTVD?*

Return *MTVD 1.00E+00V*.

Get the current Volts/div values: vdiv = 1V.

According to the wave data, we can know the first point of waveform is



## Digital Oscilloscope Series

---

the 24th data "FF", convert to decimal is "255" (Hexadecimal converted to decimal). Then minus 255, the code value is 0.

The first point of wave data voltage value =  $0 \cdot (1/25) = 0V$ .

**Step 3:** Calculate the time value of the data point.

Using the formula: time value(S) = - (timebase\*grid/2).

timebase: The timebase value.

grid: The grid numbers in horizontal direction.

The picture above as an example:

Send command *TDIV?*  
Return *TDIV 5.00E-09S*.

Get the current timebase: timebase = 5.00E-09S.

The time value of the first data point: time value = - (5.00E-09\*14/2) = -35.00E-09(s) = -35(ns).

Send command *SARA?*  
Return *SARA 5.00E+08Sa/s*.

Get the current sampling rate: sampling rate= 500MSa/s.

Send command *SANU? CI*  
Return *SANU 3.50E+01pts*.

Get the number of sampling points: points number = 35pts.

The interpolation multiplier: interpolation multiplier = length of the block / points number = 700/35 = 20

The time interval: time inter = 1/ sampling rate/ interpolation multiplier = 0.1ns.



**WAVEFORM**

**WAVEFORM\_SETUP |WFSU**

Command/Query

**DESCRIPTION**

The WAVEFORM\_SETUP command specifies the amount of data in a waveform to be transmitted to the controller.

The WAVEFORM\_SETUP? query returns the transfer parameters currently in use.

**COMMAND SYNTAX**

WAVEFORM\_SETUP  
SP,<sparsing>,NP,<number>,FP,<point>

- SP — Sparse point. It defines the interval between data points.

For example:

SP = 0 sends all data points.

SP = 4 sends every 4 data points.

- NP — The number of points. It indicates how many points should be transmitted. For example:

NP = 0 sends all data points.

NP = 50 sends a maximum of 50 data points.

- FP — First point. It specifies the address of the first data point to be sent. For waveforms acquired in sequence mode, this refers to the relative address in the given segment.

For example:

FP = 0 corresponds to the first data point.

FP = 1 corresponds to the second data point.

**Note:**

- You can set the sparse point or number of points or the first point using key-value pairs alone. See the example for details.
- After power on, SP is set to 0, NP is set to 0, and FP is set to 0.

## Digital Oscilloscope Series

---

<b>QUERY SYNTAX</b>	WAVEFORM_SETUP?
<b>RESPONSE FORMAT</b>	WAVEFORM_SETUP SP,<sparsing>,NP,<number>,FP,<point>
<b>EXAMPLE</b>	The following command specifies that every 3 data points (SP=3) starting at the 200 <sup>th</sup> point should be transferred.  Command message: <i>WFSU SP,3,FP,200</i>
<b>RELATED COMMANDS</b>	WF?

## WGEN Commands

When the built-in waveform generator is licensed (Option AWG), you can use it to output sine, square, ramp, pulse, DC, noise, exponential rise, exponential fall, cardiac, Gaussian pulse and arbitrary waveforms. The WGEN commands are used to select the waveform function and parameters.

**ARWV**  
**PROD?**  
**STL?**  
**WGEN**  
**WVPR?**

**Note:**

These commands are only valid for the model which has installed AWG option.

### Availability of WGEN Commands in Each Oscilloscope Series

Model	Valid?
T3DSO2000	yes
T3DSO1000(A)	yes - except T3DSO1102

---

## Digital Oscilloscope Series

---

### ***WGEN***

### **ARBWAVE | ARWV** Command

#### **DESCRIPTION**

The ARBWAVE command sets the basic waveform type.

#### **COMMAND SYNTAX**

ARBWAVE INDEX,<index>

<index>:= {0,1,2,3,4,5,6,7,8,9}.

#### **EXAMPLE**

For T3DSO2000 series, when the AWG option is installed, the following command set the index of waveform type to 3.

Command message:  
*ARWV INDEX,3*

#### **Note:**

See the table **Availability of WGEN Commands in Each Oscilloscope Series** for details.

**WGEN**

**PRODUCT? | PROD?**

Query

**DESCRIPTION**

The PRODUCT? query returns the product model or the upper limit of frequency of the output signal.

**QUERY SYNTAX**

PRODUCT? <parameter>

<parameter>:={MODEL,BAND}

- MODEL — return the product model.
- BAND — return the upper limit of frequency of the output signal.

**RESPONSE FORMAT**

PRODUCT <parameter>,<value>

**EXAMPLE**

For T3DSO2000 series, when the AWG option is installed, the following query returns the upper limit of frequency of the output signal.

Query message:

*PROD? BAND*

Response message:

*PROD BAND,25MHz*

**Note:**

See the table **Availability of WGEN Commands in Each Oscilloscope Series** for details.

**WGEN**

**STORELIST? | STL?**

Query

**DESCRIPTION**

The STORELIST? query returns the stored arbitrary waveforms list with indexes and names. If the store unit is empty, the command will return "EMPTY" string.

**QUERY SYNTAX**

STORELIST? <type>

<type>:={DEBUG,RELEASE}

- **DEBUG** — return built-in waveforms. (include sine, noise, cardiac, gaus\_pulse, exp\_rise, exp\_fall, and four waveforms defined by user)
- **RELEASE** — return four waveforms defined by user.

**RESPONSE FORMAT**

STORELIST <list>

**EXAMPLE**

For T3DSO2000 series, when the AWG option is installed, the following query returns the waveform storage list.

Query message:

*STL? DEBUG*

Response message:

*STL  
M0,SINE,M1,NOISE,M2,CARDIAC,M3,GA  
US\_PULSE,M4,EXP\_RISE,M5,EXP\_FALL,  
M6,EMPTY,M7,EMPTY,M8,EMPTY,M9,E  
MPTY*

**Note:**

See the table **Availability of WGEN Commands in Each Oscilloscope Series** for details.



**WGEN**

**WAVEGENERATOR | WGEN**

Command/Query

**DESCRIPTION**

The WAVEGENERATOR command sets parameters of basic waveform.

The WAVEGENERATOR? query returns the waveform parameters.

**COMMAND SYNTAX**

WAVEGENERATOR  
<parameter>,<value>

<parameter>:= {a parameter from the table below}.

<value>:= {value of the corresponding parameter}.

Parameters	Value	Description
OUTP	<state>	:= {ON,OFF}.
WVTP	<type>	:= {SINE,SQUARE,RAMP,PULSE,DC,NOISE,CARDIAC,GAUS_PULSE,EXP_RISE,EXP_FALL,ARB1,ARB2,ARB3,ARB4}. If the command doesn't set basic waveform type, WVTP will be set to the current waveform.
FREQ	<frequency>	:= 0.000001 Hz to 25000000 Hz. Not valid when WVTP is NOISE or DC.
AMPL	<amplitude>	:= 0.004V to 6 V. Not valid when WVTP is NOISE or DC.
OFST	<offset>	:= -(6 - AMP)/2 to (6 - AMP)/2(V). Not valid when WVTP is NOISE.
DCOFST	<dc_offset>	:= -3V to 3 V. Only valid when WVTP is DC.
DUTY	<duty>	:= 20% to 80%. Only valid when WVTP is SQUARE.
SYMM	<sym>	:= 0 to 100%. Only valid when WVTP is RAMP.
WIDTH	<width>	:= 0.00000048s to 0.001s. Only valid when WVTP is PULSE.
STDEV	<std>	:= 0.0003V to 0.45V. Only valid when WVTP is NOISE.
MEAN	<mean>	:= -(0.45-STD)*(20/3)-(0.45-STD)*(20/3) (V). Only valid when WVTP is NOISE.

## Digital Oscilloscope Series

LOAD	<load>	:= { HZ, 50 }.
------	--------	----------------

### QUERY SYNTAX

WAVEGENERATOR? <parameter>

<parameter>:= {OUTP,WVTP,FREQ,AMPL,OFST,DCOFST,DUTY,SYMM,WIDTH,STDEV,MEAN,LOAD,ALL}

### RESPONSE FORMAT

WAVEGENERATOR  
<parameter>,<value>

### EXAMPLE

- For T3DSO2000 series, when the AWG option is installed, the following command set the type to square, amplitude to 2.5 V, frequency to 10 kHz and duty to 45%.

Command message:

```
WGEN
TYPE,SQUARE,FREQ,10000Hz,AMPL,2.5
V,DUTY,45%
```

- For T3DSO2000 series, when the AWG option is installed, the following command set the type to noise, stdev to 0.2 V, mean to 1 V.

Command message:

```
WGEN
TYPE,NOISE,STDEV,0.2V,MEAN,1V
```

- For T3DSO2000 series, when the AWG option is installed, the following command set the output to off.

Command message:

```
WGEN OUP,OFF
```

#### Note:

See the table **Availability of WGEN Commands in Each Oscilloscope Series** for details.

**WGEN**

**WAVE\_PARA? | WVPR?**

Query

**DESCRIPTION**

The WAVE\_PARA? query returns the location, name, frequency, amplitude, and offset of four arbitrary waveforms.

**QUERY SYNTAX**

WAVE\_PARA? <index>

**RESPONSE FORMAT**

WAVE\_PARA  
POS,<index>,WVNM,<name>,FREQ,<freq>,  
>,AMPL,<ampl>,OFST,<ofst>

**EXAMPLE**

For T3DSO2000 series, when the AWG option is installed, the following query returns the parameters of M0.

Query message:

*WVPR? M0*

Response message:

*WVPR*

*POS,M0,WVNM,SINE,FREQ,1.000000e+03,AMPL,6.000000e+00,OFST,0.000000e+00*

**RELATED COMMANDS**

STL?

**Note:**

See the table **Availability of WGEN Commands in Each Oscilloscope Series** for details.

## Programming Examples

This chapter gives some examples for the programmer. In these examples you can see how to use VISA or sockets, in combination with the commands described above to control the oscilloscope. By following these examples, you can develop many more applications.

### VISA Examples

- VC++ Example

- VB Example

- MATLAB Example

- LabVIEW Example

- C# Example

### Examples of Using Sockets

- Python Example

- C Example

### Common Command Examples

- Read Waveform Data (WF) Example

- Screen Dump (SCDP) Example

## VISA Examples

### VC++ Example

**Environment:** Win7 32-bit, Visual Studio.

**Description:** Use National Instruments VISA to control the device with USBTMC or TCP/IP access. Perform a write and read operation.

**Steps:**

1. Open Visual Studio, create a new VC++ win32 project.
2. Set the project environment to use the NI-VISA library. There are two ways to use NI-VISA, static or automatic:

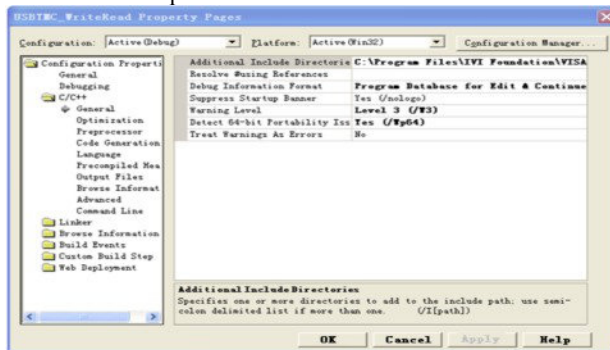
a) Static:

Find the files visa.h, visatype.h, visa32.lib in the NI-VISA installation path, copy them to your project, and add them into the project. In the projectname.cpp file, add the follow two lines:

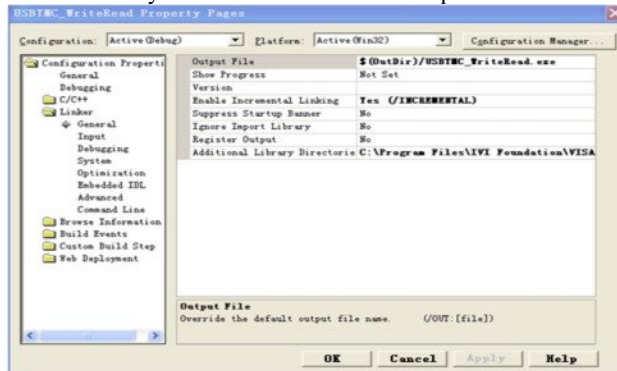
```
#include "visa.h"
#pragma comment(lib,"visa32.lib")
```

b) Automatic:

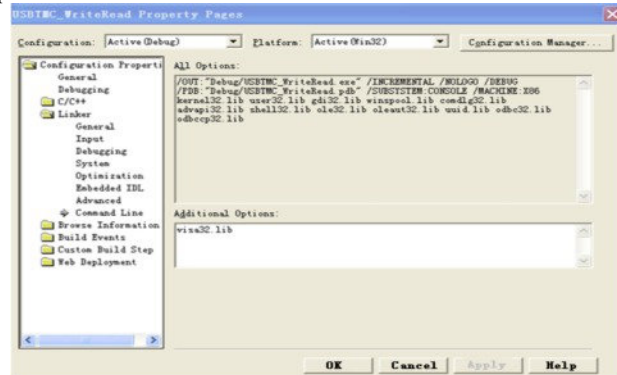
Set the .h file include directory, the NI-VISA install path, in our computer we set the path is: C:\Program Files\IVI Foundation\IVISA\WinNT\include. Set this path to project---properties---c/c++---General---Additional Include Directories: See the picture.



Set lib path set lib file:  
Set lib path: the NI-VISA install path, in our computer we set the path is :  
C:\Program Files\IVI Foundation\VISA\WinNT  
\lib\msc. Set this path to project---properties---Linker---General---  
Additional Library Directories: as shown in the pictures below.



Set lib file:project---properties---Linker---Command Line---Additional  
Options: visa32.lib



Include visa.h file in the projectname.cpp file:  
`#include <visa.h>`

3. Coding:  
a) USBTMC:

```
IntUsbtmc_test() {  
    /* This code demonstrates sending synchronous read & write commands */  
    /* to an USB Test & Measurement Class (USBTMC) instrument using */  
    /* NI-VISA */  
    /* The example writes the "*IDN?\n" string to all the USBTMC */  
    /* devices connected to the system and attempts to read back */  
    /* results using the write and read functions. */  
    /* The general flow of the code is*/  
    /* Open Resource Manager */  
    /* Open VISA Session to an Instrument */  
    /* Write the Identification Query Using viPrintf */  
    /* Try to Read a Response With viScanf */  
    /* Close the VISA Session */  
    /******  
    ViSession defaultRM;  
    ViSession instr;  
    ViUInt32 numInstrs;  
    ViFindList findList;  
    ViUInt32 retCount;  
    ViUInt32 writeCount;  
    ViStatus status;  
    Char instrResourceString[VI_FIND_BUFLLEN];  
    Unsignedchar buffer[100];  
    Charstringinput[512];  
    Int i;  
    /** First we must call viOpenDefaultRM to get the manager *  
    handle. We will store this handle in defaultRM.*/  
    status=viOpenDefaultRM (&defaultRM);
```

```
if (status<VI_SUCCESS) {
    printf ("Could not open a session to the VISA Resource Manager!\n");
    returnstatus;
}
/* Find all the USB TMC VISA resources in our system and store the number of
resources in the system in numInstrs. */
status = viFindRsrc (defaultRM, "USB?*INSTR", &findList, &numInstrs,
instrResourceString);
if (status<VI_SUCCESS)
{
    printf ("An error occurred while finding resources.\nHit enter to continue.");
    fflush(stdin);
    getchar();
    viClose (defaultRM);
    returnstatus;
}
/** Now we will open VISA sessions to all USB TMC instruments. *
We must use the handle from viOpenDefaultRM and we must
* also use a string that indicates which instrument to open. This
* is called the instrument descriptor. The format for this string
* can be found in the function panel by right clicking on the
* descriptor parameter. After opening a session to the
* device, we will get a handle to the instrument which we
* will use in later VISA functions. The AccessMode and Timeout
* parameters in this function are reserved for future
* functionality. These two parameters are given the value VI_NULL.*/
for (i=0; i<numInstrs; i++) {
    if (i> 0)
    {
```



## Digital Oscilloscope Series

---

```
viFindNext (findList, instrResourceString);
}
status = viOpen (defaultRM, instrResourceString, VI_NULL, VI_NULL, &instr);
if (status<VI_SUCCESS)
{
    printf ("Cannot open a session to the device %d.\n", i+1);
    continue;
}
/* * At this point we now have a session open to the USB TMC instrument.
* We will now use the viPrintf function to send the device the string "*IDN?\n", *
asking for the device's identification. */
char * cmmmand ="*IDN?\n";
status = viPrintf (instr, cmmmand);
if (status<VI_SUCCESS)
{
    printf ("Error writing to the device %d.\n", i+1);
    status = viClose (instr);
    continue;
}
/** Now we will attempt to read back a response from the device to
* the identification query that was sent. We will use the viScanf *
function to acquire the data.
* After the data has been read the response is displayed.*/
status = viScanf(instr, "%t", buffer); if
(status<VI_SUCCESS)
{
    printf ("Error reading a response from the device %d.\n", i+1);
} else
{
    printf ("\nDevice %d: %*s\n", i+1,retCount, buffer); }
```

```
        status = viClose (instr);
    }
    /** Now we will close the session to the instrument using
    * viClose. This operation frees all system resources.          */
    status = viClose (defaultRM);
    printf("Press 'Enter' to exit.");
    fflush(stdin);
    getchar();
    return 0;
}
```

b) TCP/IP:

```
intTCP_IP_Test(char *pIP) {
    char outputBuffer[VI_FIND_BUFLen];
    ViSession defaultRM, instr;
    ViStatus status;
    ViUInt32 count;
    ViUInt16 portNo;
    /** First we will need to open the default resource manager. */
    status = viOpenDefaultRM (&defaultRM);
    if (status<VI_SUCCESS)
    {
        printf("Could not open a session to the VISA Resource Manager!\n");
    }
    /** Now we will open a session via TCP/IP device */
    charhead[256] ="TCPIP0:.";
    chartail[] = "::INSTR";
    charresource [256];
    streat(head,pIP);
}
```

```
    streat(head,tail);
    status = viOpen (defaultRM, head, VI_LOAD_CONFIG, VI_NULL, &instr);
    if (status<VI_SUCCESS) {
        printf ("An error occurred opening the session\n");
        viClose(defaultRM); }
    status = viPrintf(instr, "%idn?\n");
    status = viScanf(instr, "%t", outputBuffer); if
    (status<VI_SUCCESS)
        {
            printf("viRead failed with error code: %x \n",status);
            viClose(defaultRM);
        } else
        {
            printf ("\ndata read from device: %*s\n", 0,outputBuffer); }
    status = viClose (instr);
    status = viClose (defaultRM);
    printf("Press 'Enter' to exit.");
    fflush(stdin);
    getchar();
    return 0;
}
}
```

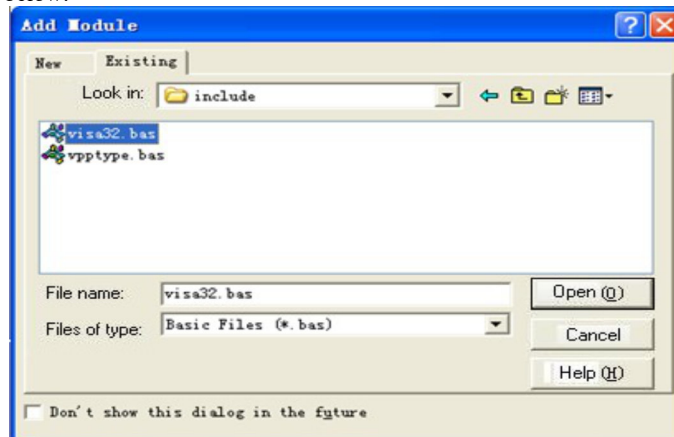
### VB Example

**Environment:** Windows7 32-bit, Microsoft Visual Basic 6.0

**Description:** The function of this example: Use the NI-VISA, to control the device with USBTMC and TCP/IP access to do a write and read.

**Steps:**

1. Open Visual Basic, and build a standard application program project.
2. Set the project environment to use the NI-VISA lib: Click the Existing tab of Project>>Add Module, search the visa32.bas file in the “include” folder under the NI-VISA installation path and add the file, as shown in the figure below:



3. Coding:

a) USBTMC:

`Private Function Usbtmc_test() As Long`

`'This code demonstrates sending synchronous read & write commands`

```
' to an USB Test & Measurement Class (USBTMC) instrument using '  
NI-VISA  
' The example writes the "*IDN?\n" string to all the USBTMC '  
devices connected to the system and attempts to read back  
' results using the write and read functions.  
' The general flow of the code is  
' Open Resource Manager  
' Open VISA Session to an Instrument  
' Write the Identification Query Using viWrite '  
Try to Read a Response With viRead  
' Close the VISA Session  
Const MAX_CNT = 200  
  
Dim defaultRM As Long  
Dim instrsesn As Long  
Dim numInstrs As Long  
Dim findList As Long Dim  
retCount As Long Dim  
writeCount As Long Dim  
status As Long  
Dim instrResourceString As String * VI_FIND_BUFLEN  
Dim buffer As String * MAX_CNT  
Dim i As Integer  
' First we must call viOpenDefaultRM to get the manager '  
handle. We will store this handle in defaultRM.  
status = viOpenDefaultRM(defaultRM)  
If (status < VI_SUCCESS) Then  
    Debug.Print "Could not open a session to the VISA Resource Manager!"  
    Usbtmc_test = status  
    ExitFunction  
End If
```

## Digital Oscilloscope Series

---

```
' Find all the USB TMC VISA resources in our system and store the
' number of resources in the system in numInstrs.
status=viFindRsrc(defaultRM,"USB?*INSTR",findList,numInstrs,instrResourceString)
If (status < VI_SUCCESS) Then
    Debug.Print "An error occurred while finding resources."
    viClose (defaultRM)
    Usbtmc_test = status
    Exit Function
End If

' Now we will open VISA sessions to all USB TMC instruments.
' We must use the handle from viOpenDefaultRM and we must
' also use a string that indicates which instrument to open. This
' is called the instrument descriptor. The format for this string
' can be found in the function panel by right clicking on the
' descriptor parameter. After opening a session to the
' device, we will get a handle to the instrument which we
' will use in later VISA functions. The AccessMode and Timeout '
parameters in this function are reserved for future
' functionality. These two parameters are given the value VI_NULL.
For i = 0 To numInstrs If
    (i > 0) Then
        status = viFindNext(findList, instrResourceString)
    End If
    status = viOpen(defaultRM, instrResourceString, VI_NULL, VI_NULL, instrsesn)
    If (status < VI_SUCCESS) Then
        Debug.Print "Cannot open a session to the device ", i + 1
        GoTo NextFind
    End If

    ' At this point we now have a session open to the USB TMC instrument.
```

---

## Digital Oscilloscope Series

---

' We will now use the viWrite function to send the device the string "\*IDN?", ' asking for the device's identification.

```
status = viWrite(instrsesn, "*IDN?", 5, retCount)
```

```
If (status < VI_SUCCESS) Then
```

```
    Debug.Print "Error writing to the device."
```

```
    status = viClose(instrsesn)
```

```
    GoTo NextFind
```

```
End If
```

' Now we will attempt to read back a response from the device to ' the identification query that was sent. We will use the viRead ' function to acquire the data.

' After the data has been read the response is displayed.

```
status = viRead(instrsesn, buffer, MAX_CNT, retCount)
```

```
If (status < VI_SUCCESS) Then
```

```
    Debug.Print "Error reading a response from the device.", i + 1
```

```
Else
```

```
    Debug.Print i + 1, retCount, buffer
```

```
End If
```

```
status = viClose(instrsesn)
```

```
Next i
```

' Now we will close the session to the instrument using '

viClose. This operation frees all system resources.

```
status = viClose(defaultRM)
```

```
Usbtmc_test = 0
```

```
End Function
```

b) TCP/IP:

```
Private Function TCP_IP_Test(ip As String) As Long
```

## Digital Oscilloscope Series

---

```
Dim outputBuffer As String * VI_FIND_BUFLen
Dim defaultRM As Long
Dim instrsesn As Long
Dim status As Long
Dim count As Long

'First we will need to open the default resource manager.
status = viOpenDefaultRM (defaultRM)
If (status < VI_SUCCESS) Then
    Debug.Print "Could not open a session to the VISA Resource Manager!"
    TCP_IP_Test = status
    Exit Function
End If

'Now we will open a session via TCP/IP device
status = viOpen(defaultRM, "TCPIP0:." + ip + ":", VI_LOAD_CONFIG,
VI_NULL, instrsesn)
If (status < VI_SUCCESS) Then
    Debug.Print "An error occurred opening the session"
    viClose (defaultRM)
    TCP_IP_Test = status
    Exit Function
End If

status = viWrite(instrsesn, "*IDN?", 5, count) If
(status < VI_SUCCESS) Then
    Debug.Print "Error writing to the device."
End If

status = viRead(instrsesn, outputBuffer, VI_FIND_BUFLen, count) If
(status < VI_SUCCESS) Then
    Debug.Print "Error reading a response from the device.", i + 1
```



## Digital Oscilloscope Series

---

```
Else
    Debug.Print "read from device:", outputBuffer
End If
status = viClose(instrsesn)
status = viClose(defaultRM)
TCP_IP_Test = 0
End Function
```

## MATLAB Example

**Environment:** Windows7 32-bit, MATLAB R2010b

**Description:** The function of this example: Use the NI-VISA, to control the device with USBTMC or TCP/IP access to do a write and read.

**Steps:**

1. Open MATLAB, and modify the current directory. In this demo, the current directory is modified to D:\USBTMC\_TCPIP\_Demo.
2. Click File>>New>>Script in the Matlab interface to create an empty M file.
3. Coding:
  - a) USBTMC:

```
function USBTMC_test()
% This code demonstrates sending synchronous read & write commands
% to an USB Test & Measurement Class (USBTMC) instrument using %
NI-VISA

%Create a VISA-USB object connected to a USB instrument
vu = visa('ni','USB0::0xF4EC::0xEE38::0123456789::INSTR');

%Open the VISA object created
fopen(vu);

%Send the string "*IDN?",asking for the device's identification.
fprintf(vu, '*IDN?');

%Request the data
outputbuffer = fscanf(vu);
disp(outputbuffer);

%Close the VISA object
fclose(vu);
delete(vu);
clear vu;

end
```

- b) TCP/IP:

---

## Digital Oscilloscope Series

---

```
function TCP_IP_test( IPstr )
% This code demonstrates sending synchronous read & write commands
% to an TCP/IP instrument using NI-VISA

%Create a VISA-TCPIP object connected to an instrument
%configured with IP address.
vt = visa('ni',['TCPIP0:;',IPstr,':INSTR']);

%Open the VISA object created
fopen(vt);

%Send the string "*IDN?", asking for the device's identification.
fprintf(vt, '*IDN?');

%Request the data
outputbuffer = fscanf(vt);
disp(outputbuffer);

%Close the VISA object
fclose(vt);
delete(vt);
clear vt;

end
```

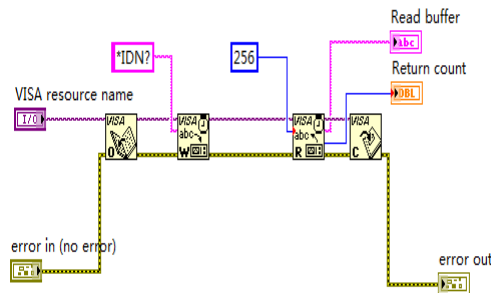
### LabVIEW Example

**Environment:** Windows7 32-bit, LabVIEW 2011

**Description:** The functions of this example: use the NI-VISA, to control the device with USBTMC and TCP/IP access to do a write and read.

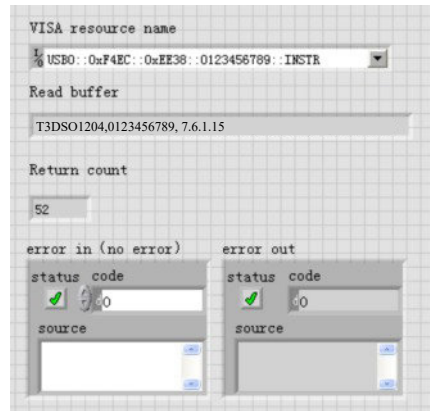
**Steps:**

1. Open LabVIEW, create a VI file.
2. Add controls. Right-click in the **Front Panel** interface, select and add **VISA resource name**, error in, error out and some indicators from the Controls column.
3. Open the **Block Diagram** interface. Right-click on the **VISA resource name** and you can select and add the following functions from VISA Palette from the pop-up menu: **VISA Write**, **VISA Read**, **VISA Open** and **VISA Close**.
4. The connection is as shown in the figure below:



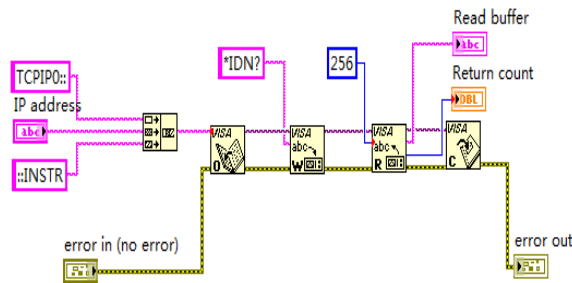
5. Select the device resource from the VISA Resource Name list box and run the program.

## Digital Oscilloscope Series



In this example, the VI opens a VISA session to a USBTMC device, writes a command to the device, and reads back the response. After all communication is complete, the VI closes the VISA session.

6. Communicating with the device via TCP/IP is similar to USBTMC. But you need to change VISA Write and VISA Read Function to Synchronous I/O. The LabVIEW default is asynchronous I/O. Right-click the node and select Synchronous I/O Mod>>Synchronous from the shortcut menu to write or read data synchronously.
7. The connection is as shown in the figure below:



8. Input the IP address and run the program.

## Digital Oscilloscope Series

---

IP address	
10.11.25.232	
Read buffer	
T3DSOI204,0123456789, 7.6.1.15	
Return count	
52	
error in (no error)	error out
status code	status code
✓ 0	✓ 0
source	source

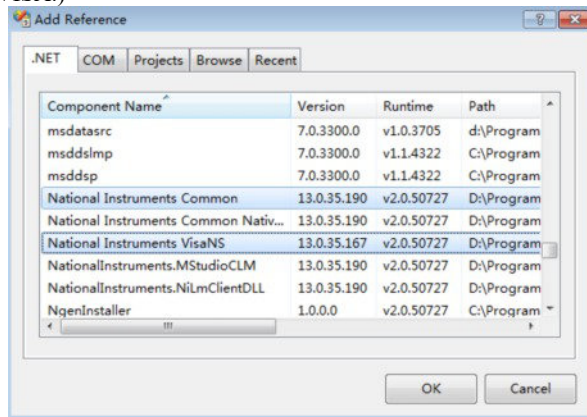
### C# Example

**Environment:** Windows7 32-bit, Visual Studio

**Description:** The functions of this example: use the NI-VISA, to control the device with USBTMC or TCP/IP access to do a write and read.

**Steps:**

1. Open Visual Studio, create a new C# project.
2. Add References. Add NationalInstruments.Common.dll and NationalInstruments.VisaNS.dll to the project. (Notice: you must install the .NET Framework 3.5/4.0/4.5 Languages support when you install the NI-VISA.)



3. Coding:

```
using System;  
using System.Collections.Generic;  
using System.Linq;  
using System.Text;  
using NationalInstruments.VisaNS;  
  
namespace TestVisa
```

```
{
  class Program
  {
    static void Main(string[] args)
    {
      // Find all the USBTMC resources
      string[]
      usbRsrcStrings=ResourceManager.GetLocalManager().FindResources("
      USB?*INSTR");
      if (usbRsrcStrings.Length <= 0)
      {
        Console.WriteLine("Can not find USBTMC Device!");
        return;
      }

      //Choose the first resource string to connect the device.
      //You can input the address manually
      //USBTMC:
      //MessageBasedSession
      mbSession=(MessageBasedSession)ResourceManager.GetLocalManager
      ().Open("USB0::0xF4EC::0xEE38::0123456789::INSTR");
      //TCP IP:
      //MessageBasedSession
      mbSession=(MessageBasedSession)ResourceManager.GetLocalManager
      ().Open("TCPIP0::192.168.1.100::INSTR");
      MessageBasedSession
      mbSession=(MessageBasedSession)ResourceManager.GetLocalManager
      ().Open(usbRsrcStrings[0]);
      mbSession.Write("*IDN?");
      string result = mbSession.ReadString();
      mbSession.Dispose();
    }
  }
}
```



## Digital Oscilloscope Series

---

```
        Console.WriteLine(result);  
    }  
}  
}
```

## Examples of Using Sockets

Socket communication is a basic communication technology in computer network. It allows applications to communicate through the standard network protocol mechanism built by network hardware and operation system. This method is a two-way communication between the instrument and the computer through a fixed port number.

Note that SCPI strings are terminated with a “\n” (new line) character.

### Python Example

Python has a low-level networking module that provides access to the socket interface. Python scripts can be written for sockets to do a variety of test and measurement tasks.

**Environment:** Windows 7 32-bit, Python v2.7.5

**Description:** Open a socket, send a query, and repeat this loop for 10 times, finally close the socket.

Below is the code of the script:

```
#!/usr/bin/env python
#-*- coding:utf-8 -*-
#----- #
The short script is a example that open a socket, sends a query, #
print the return message and closes the socket.
#----- #
import socket # for sockets
import sys # for exit
import time # for sleep
#----- #
remote_ip = "10.11.13.32" # should match the instrument's IP address
port = 5024 # the port number of the instrument service
count = 0
```

```
def SocketConnect():
    try:
        #create an AF_INET, STREAM socket (TCP)
        s = socket.socket(socket.AF_INET, socket.SOCK_STREAM)
    except socket.error:
        print ('Failed to create socket.')
        sys.exit();
    try:
        #Connect to remote server
        s.connect((remote_ip , port))
        info = s.recv(4096)
        print (info)
    except socket.error:
        print ('failed to connect to ip ' + remote_ip)
    return s

def SocketQuery(Sock, cmd):
    try :
        #Send cmd string
        Sock.sendall(cmd)
        time.sleep(1)
    except socket.error:
        #Send failed
        print ('Send failed')
        sys.exit()
    reply = Sock.recv(4096)
    return reply

def SocketClose(Sock):
    #close the socket
    Sock.close()
```

```
time.sleep(.300)

def main():
    global remote_ip
    global port global
    count

    # Body: send the SCPI commands *IDN? 10 times and print the return message
    s = SocketConnect()
    for i in range(10):
        qStr = SocketQuery(s, b'*IDN?')
        print (str(count) + " :: " + str(qStr))
        count = count + 1
    SocketClose(s)
    input('Press "Enter" to exit')

if __name__ == '__main__':
    proc = main()
```

## C Example

```
int MySocket;
if((MySocket=socket(PF_INET,SOCK_STREAM,0))==-1)
{
    exit(1);
}
struct in_addr
{
    unsigned long s_addr;
};
struct sockaddr_in
{
    short int sin_family; // Address family
    unsigned short int sin_port; // Port number
    struct in_addr sin_addr; // Internet address
    unsigned char sin_zero[8]; // Padding
};
struct sockaddr_in MyAddress;

// Initialize the whole structure to zero
memset(&MyAddress,0,sizeof(struct sockaddr_in)); //
Then set the individual fields
MyAddress.sin_family=PF_INET; // IPv4
MyAddress.sin_port=htons(5025); // Port number used by most instruments
MyAddress.sin_addr.s_addr=inet_addr("169.254.9.801"); // IP Address

// Establish TCP connection
if(connect(MySocket,(struct sockaddr*)&MyAddress,sizeof(struct sockaddr_in))==-1)
{
    exit(1);
}

// Send SCPI command
if(send(MySocket,"*IDN?\n",6,0))==-1)
{
    exit(1);
}

// Read response
char buffer[200];
int actual;
if((actual=recv(MySocket,&buffer[0],200,0))==-1)
{
    exit(1);
}
```

## Digital Oscilloscope Series

---

```
buffer[actual]=0; // Add zero character (C string)
printf("Instrument ID: %s\n",buffer);

// Close socket
if(close(MySocket)==-1)
{
    exit(1);
}
```

## Common Command Examples

This section lists the programming instances of common commands.

**Environment:** Windows 7 32-bit, Python v3.4.3, pyvisa-1.7, Matplotlib-1.5.1

## Read Waveform Data (WF) Example

```
import visa
import pylab as pl

def main():
    _rm = visa.ResourceManager()
    dso = _rm.open_resource("USB0::0xF4EC::0xEE38::0123456789::INSTR")
    dso.write("chdr off")
    vdiv = dso.query("c1:vdiv?")
    ofst = dso.query("c1:ofst?")
    tdiv = dso.query("tdiv?")
    sara = dso.query("sara?")
    sara_unit = {'G':1E9,'M':1E6,'k':1E3}
    for unit in sara_unit.keys():
        if sara.find(unit)!=-1:
            sara = sara.split(unit)
            sara = float(sara[0])*sara_unit[unit]
            break
    sara = float(sara)
    dso.timeout = 30000 #default value is 2000(2s)
    dso.chunk_size = 20*1024*1024 #default value is 20*1024(20k bytes)
    dso.write("c1:wf? dat2")
    recv = list(dso.read_raw())[15:]
    recv.pop()
    recv.pop()
    volt_value = []
    for data in recv:
        if data > 127:
            data = data - 255
        else:
            pass
        volt_value.append(data)
    time_value = []
    for idx in range(0,len(volt_value)):
        volt_value[idx] = volt_value[idx]/25*float(vdiv)-float(ofst)
        time_data = -(float(tdiv)*14/2)+idx*(1/sara)
        time_value.append(time_data)
```

```
pl.figure(figsize=(7,5))
pl.plot(time_value,volt_value,markersize=2,label=u"Y-T")
pl.legend()
pl.grid()
pl.show()

if __name__ == '__main__':
    main()
```

### Read Waveform Data of Digital Example

```
import visa
import pylab as pl

def get_char_bit(char,n):
    return (char >> n) & 1

def main():
    _rm = visa.ResourceManager()
    dso = _rm.open_resource("USB0::0xF4EC::0xEE38::0123456789::INSTR")
    dso.write("chdr off")
    tdiv = dso.query("tdiv?")
    sara = dso.query("di:sara?")
    sara_unit = {'G':1E9,'M':1E6,'k':1E3}
    for unit in sara_unit.keys():
        if sara.find(unit)!=-1:
            sara = sara.split(unit)
            sara = float(sara[0])*sara_unit[unit]
            break
    sara = float(sara)
    dso.timeout = 30000 #default value is 2000(2s)
    dso.chunk_size = 20*1024*1024 #default value is 20*1024(20k bytes)
    dso.write("d0:wf? dat2")
    recv = list(dso.read_raw())[15:]
    recv.pop()
    recv.pop()
    volt_value = []
    data = bytearray(recv)

    for char in data:
        for i in range(0,8):
            volt_value.append(get_char_bit(char,i))
    print(len(volt_value))
    time_value = []
    for idx in range(0,len(volt_value)):
```



## Digital Oscilloscope Series

```
time_data = -(float(tdiv)*14/2)+idx*(1/sara)
time_value.append(time_data)

pl.figure(figsize=(7,5))
pl.ylim(-1,2)
pl.plot(time_value,volt_value,markersize=2,label=u"Y-T")
pl.legend()
pl.grid()
pl.show()

if __name__=='__main__':
    main()
```

## Screen Dump (SCDP) Example

```
import visa

def main():
    _rm = visa.ResourceManager()
    dso = _rm.open_resource("USB0::0xF4EC::0xEE38::0123456789::INSTR ")
    file_name = "F:\\SCDP.bmp"
    dso.write("SCDP")
    result_str = dso.read_raw()
    f = open(file_name,'wb')
    f.write(result_str)
    f.flush()
    f.close()

if __name__=='__main__':
    main()
```

Then you can open the file as shown below:



## Index

### A

ACQUIRE\_WAY, ACQW, Command/Query, 27  
ARBWAVE, ARWV, Command, 190  
ARM\_ACQUISITION, ARM, Command, 24  
ATTENUATION, ATTN, Command/Query, 40  
AUTO\_SETUP, ASET, Command, 38  
AVERAGE\_ACQUIRE, AVGA, Command/Query, 29

### B

BANDWIDTH\_LIMIT, BWL, Command/Query, 41  
BUZZER, BUZZ, Command/Query, 147

### C

CAL?, Query, 146  
COMM\_HEADER, CHDR, Command/Query, 22  
COMM\_NET, CONET, Command/Query, 148  
COUPLING, CPL, Command/Query, 42  
CURSOR\_MEASURE, CRMS, Command/Query, 50  
CURSOR\_SET, CRST, Command/Query, 52  
CURSOR\_TYPE, CRTY, Command/Query, 54  
CURSOR\_VALUE?, CRVA?, Query, 55  
CUSTOM, CUS, Command/Query, 65  
CYMOMETER, CYMT, Query, 96

### D

DEFINE, DEF, Command/Query, 79  
DIGITAL\_CHANNEL, DGCH, Command/Query, 58  
DIGITAL\_STATE, DGST, Command/Query, 59  
DIGITAL\_THR, DGTH, Command/Query, 60  
DOT\_JOIN, DTJN, Command/Query, 68

### F

FFT\_CENTER, FFTC, Command/Query, 85  
FFT\_FULLSCREEN, FFTF, Command/Query, 87  
FFT\_POSITION, FFTP, Command/Query, 88

## Digital Oscilloscope Series

---

FFT\_SCALE, FFTS, Command/Query, 90  
FFT\_TDIV?, FFTT?, Query, 92  
FFT\_UNIT, FFTU, Command/Query, 93  
FFT\_WINDOW, FFTW, Command/Query, 94  
FRAME\_SET, FRAM, Command, 74  
FRAME\_TIME?, FTIM?, Query, 75

### G

GRID\_DISPLAY, GRDS, Command/Query, 69

### H

HOR\_MAGNIFY, HMAG, Command/Query, 154  
HOR\_POSITION, HPOS, Command/Query, 156  
HISTORY\_LIST, HLST, Command/Query, 77  
HISTORY\_MODE, HSMD, Command/Query, 76

### I

IDN?, Query, 18  
INTENSITY, INTS, Command/Query, 70  
INR?, Query, 143  
INVERT\_SET, INVS, Command/Query, 48/81

### M

MEASURE\_DELY, MEAD, Command/Query, 98  
MENU, MÈNU, Command/Query, 71  
MEMORY\_SIZE, MSIZ, Command/Query, 30  
MATH\_VERT\_DIV, MTVD, Command/Query, 82  
MATH\_VERT\_POS, MTVP, Command/Query, 84

### O

OFFSET, OFST, Command/Query, 43  
OPC, Command, 19

### P

PANEL\_SETUP, PNSU, Command/Query, 137  
PARAMETER\_CLR, PACL, Command, 110  
PARAMETER\_CUSTOM, PACU, Command, 101  
PARAMETER\_VALUE?, PAVA?, Query, 104  
PERSIST\_SETUP, PESU, Command/Query, 72

PF\_BUFFER, PFBF, Command/Query, 111  
PF\_CREATEM, PFCM, Command, 112  
PF\_DATEDIS?, PFDD?, Query, 113  
PF\_DISPLAY, PFDS, Command/Query, 114  
PF\_ENABLE, PFEN, Command/Query, 115  
PF\_FAIL\_STOP, PFFS, Command/Query, 116  
PF\_OPERATION, PPOP, Command/Query, 117  
PF\_SET, PFST, Command/Query, 119  
PF\_SOURCE, PFSC, Command/Query, 118  
PRODUCT?, PROD?, Query, 194

## R

RCL, Command, 123  
RECALL\_PANEL, RCPN, Command, 124  
REF\_CLOSE, REFCL, Command, 127  
REF\_DISPALY, REFDS, Command/Query, 128  
REF\_LOCATION, REFLA, Command/Query, 129  
REF\_POISITION, REFPO, Command/Query, 130  
REF\_SAVE, REFSA, Command, 132  
REF\_SCLALE, REFSC, Command/Query, 133  
REF\_SOURCE, REFSR, Command/Query, 134  
RST, Command, 20

## S

SAMPLE\_NUM?, SANU?, Query, 34  
SAMPLE\_RATE?, SARA?, Query, 32  
SAMPLE\_STATUS?, SAST?, Query, 31  
SAV, Command, 136  
SCREEN\_DUMP, SCDP, Query, 121  
SCREEN\_SAVE, SCSV, Command/Query, 149  
SET50, Command, 159  
SINXX\_SAMPLE, SXSA, Command/Query, 35  
SKEW, Command, 44  
STORELIST?, STL?, Query, 192  
STOP, Command, 26  
STORE\_PANEL, STPN, Command, 139  
SWITCH, SW, Command/Query, 62

## T

THRESHOLD\_MODE, TSM, Command/Query, 64  
TIME\_DIV, TDIV, Command/Query, 151  
TRACE, TRA, Command/Query, 45/63

## Digital Oscilloscope Series

---

TRIG\_COUPLING, TRCP, Command/Query, 160  
TRIG\_DELAY, TRDL, Command/Query, 152  
TRIG\_LEVEL, TRLV, Command/Query, 161  
TRIG\_LEVEL2, TRLV2, Command/Query, 163  
TRIG\_MODE, TRMD, Command/Query, 165  
TRIG\_PATTERN, TRPA, Command/Query, 167  
TRIG\_SELECT, TRSE, Command/Query, 168  
TRIG\_SLOPE, TRSL, Command/Query, 173  
TRIG\_WINDOW, TRWI, Command/Query, 174

### U

UNIT, UNIT, Command/Query, 46

### V

VOLT\_DIV, VDIV, Command/Query, 47

### W

WAVEFORM?, WF?, Query, 176  
WAVEFORM\_SETUP, WFSU, Command/Query, 187  
WAVEGENERATOR, WGEN, Command/Query, 193  
WAVE\_PARA?, WVPR?, Query, 195

### X

XY\_DISPLAY, XYDS, Command/Query, 36