Agilent 5000 Series Oscilloscopes

User's Guide

Agilent Technologies

Notices

Manual Part Number

54574-97013

Edition

Second Edition, May 2007

Printed in Malaysia

Agilent Technologies, Inc.
395 Page Mill Road
Palo Alto, CA 94303 USA

A newer version of this manual may be available at

www.agilent.com/find/dso5000

Software Revision

This guide was written for version 04.10 of the Agilent 5000 Series Oscilloscope software.

Trademark Acknowledgments

Java is a U.S. trademark of Sun Microsystems, Inc.

Sun, Sun Microsystems, and the Sun Logo are trademarks or registered trademarks of Sun Microsystems, Inc. in the U.S. and other countries.

Windows and MS Windows are U.S. registered trademarks of Microsoft Corporation.

Warranty

The material contained in this document is provided "as is," and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied, with regard to this manual and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or of any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement shall control.

Technology Licenses

The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license.

Restricted Rights Legend

If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

Safety Notices

CAUTION

A **CAUTION** notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in damage to the product or loss of important data. Do not proceed beyond a **CAUTION** notice until the indicated conditions are fully understood and met.

WARNING

A WARNING notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in personal injury or death. Do not proceed beyond a WARNING notice until the indicated conditions are fully understood and met.

In This User's Guide...

This guide shows you how to use the 5000A Series oscilloscopes. It contains the following chapters and topics:

The Agilent 5000A Series oscilloscopes deliver powerful features and high performance:

- 100 MHz, 300 MHz, and 500 MHz bandwidth models.
- 2-channel and 4-channel Digital Storage Oscilloscope (DSO) models.
- Up to 4 GSa/s sample rate.
- Powerful triggering.
- USB, LAN, and GPIB ports make printing, saving and sharing data easy.
- Color XGA display.
- Secure environment mode option.

The 5000A Series oscilloscopes feature MegaZoom III technology:

- Most responsive deep memory.
- Highest definition color display.
- Fastest waveform update rates, uncompromised.

Table 1 5000A Series Oscilloscopes Model Numbers and Sampling Rates

Bandwidth	**100 MHz**	**300 MHz**	**500 MHz**
Maximum Sample Rate	**2 GSa/s**	**2 GSa/s**	**4 GSa/s**
2-Channel DSO	DSO5012A	DSO5032A	DSO5052A
4-Channel DSO	DSO5014A	DSO5034A	DSO5054A

Built-in Quick Help

A Quick Help system is built into the oscilloscope. Instructions for using the quick help system are given on **page 40**.

Abbreviated instructions for pressing a series of keys

Instructions for pressing a series of keys are written in an abbreviated manner. Instructions for pressing Key1, then pressing Key2, then pressing Key3 are abbreviated as follows:

Press **Key1** → **Key2** → **Key3**.

The keys may be front panel keys, or softkeys, which are located directly below the oscilloscope display.

Direct your Web browser to **www.agilent.com/find/dso5000** to

- Get software updates
- Download a newer version of this manual (if available)
- View or print the oscilloscope's data sheet
- Find out more about 5000A Series oscilloscopes

Contents

1 Getting Started

To get started using the oscilloscope:

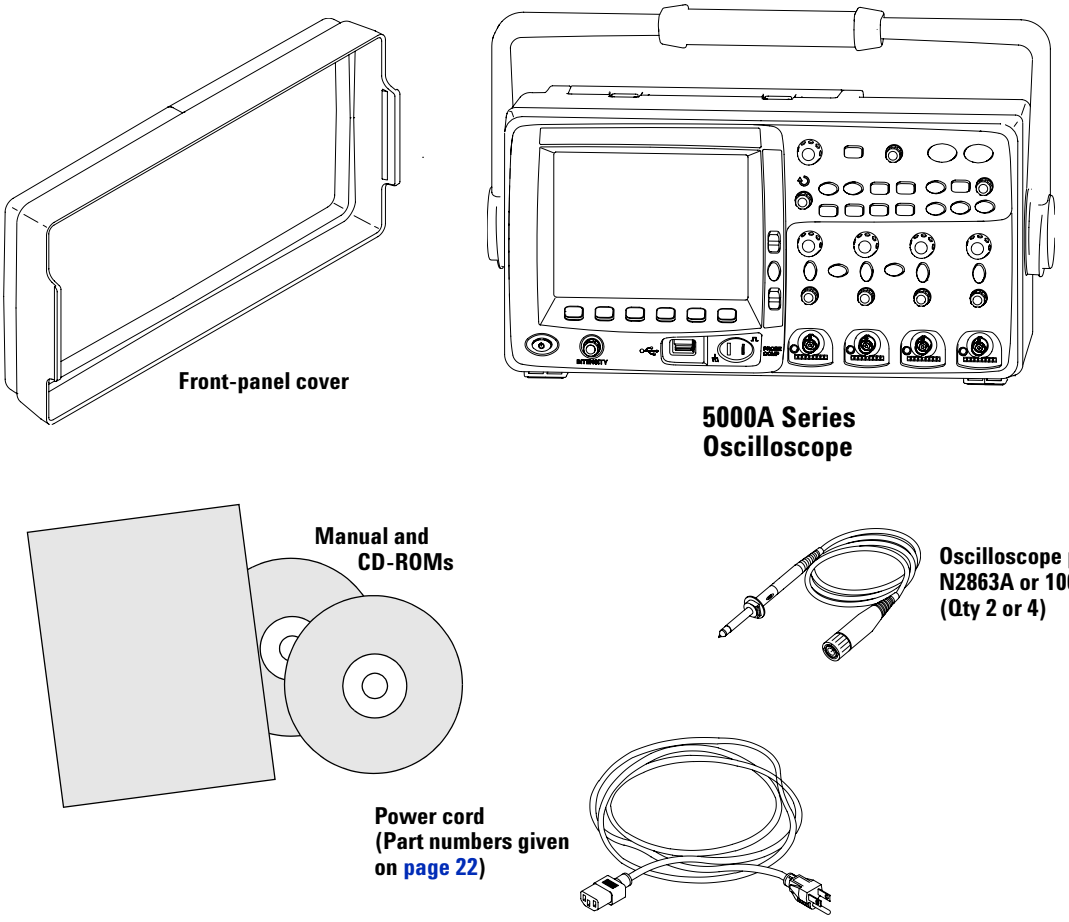
- ✔ Unpack the oscilloscope and verify the contents.
- ✔ Adjust the oscilloscope's handle position.
- ✔ Tilt the oscilloscope for easy viewing if desired.
- ✔ Apply power to the oscilloscope.
- ✔ Connect the probes to the oscilloscope.
- ✔ Verify basic oscilloscope operation and compensate the probes.

To inspect package contents

- ✔ Inspect the shipping container for damage.

  If your shipping container appears to be damaged, keep the shipping container or cushioning material until you have inspected the contents of the shipment for completeness and have checked the oscilloscope mechanically and electrically.

- ✔ Verify that you received the following items and any optional accessories you may have ordered:
  - 5000A Series Oscilloscope
  - Front-panel cover
  - Power cord (see Table 3 on page 22)
  - Oscilloscope probes
    - Two probes for 2-channel models
    - Four probes for 4-channel models
    - N2863A probes for 100 MHz and 300 MHz bandwidth models
    - 10073C probes for 500 MHz bandwidth models
  - User's Guide
  - CD-ROM containing the Programmer's Quick Start Guide, Programmer's Reference Guide, and Service Guide
  - Automation-Ready Software CD-ROM



Package contents for 5000A Series oscilloscopes

Table 2 Accessories available

Model	Description
N2760A	Soft carrying case
N2917B	Transit case
N2916B	Rackmount Kit
54684-44101	Front-panel cover
N2605A-097	USB cable
10833A	GPIB cable, 1 m long
5061-0701	LAN crossover cable
10074C	Passive probe, 10:1, 150 MHz, 1.5 m
N2863A	Passive probe, 10:1, 300 MHz, 1.2 m
10073C	Passive probe, 10:1, 500 MHz, 1.5 m
1130A	InfiniiMax probe amplifier (requires one or more probe heads)
1141A	200 MHz differential probe (with 1142A power supply)
1144A	800 MHz active probe (with 1142A power supply)
1145A	750 MHz 2-channel active probe (with 1142A power supply)
1156A	1.5 GHz active probe
1146A	100 kHz current probe, AC/DC
10070C	1:1 passive probe
10072A	Fine-pitch probe kit
10075A	0.5 mm IC clip kit
10076A	100:1, 4 kV 250 MHz probe
E2613B	0.5 mm Wedge probe adapter, 3-signal, qty 2
E2614A	0.5 mm Wedge probe adapter, 8-signal, qty 1
E2615B	0.65 mm Wedge probe adapter, 3-signal, qty 2
E2616A	0.65 mm Wedge probe adapter, 8-signal, qty 1
E2643A	0.5 mm Wedge probe adapter, 16-signal, qty 1
E2644A	0.65 mm Wedge probe adapter, 16-signal, qty 1
N2772A	20 MHz differential probe
N2773A	Power supply for N2772A
N2774A	50 MHz current probe AC/DC
N2775A	Power supply for N2774A

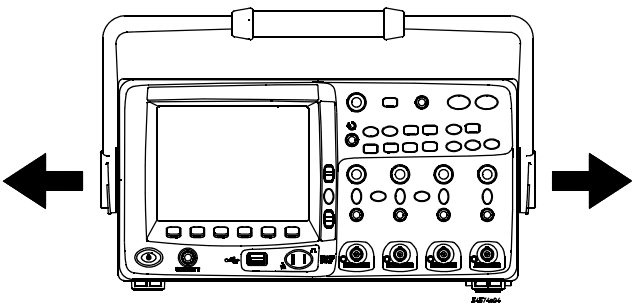
You can search for these parts at www.agilent.com or at www.parts.agilent.com.

To adjust the handle

The oscilloscope's handle can be locked in one of three positions:

- up, for carrying
- back, so the handle is out of the way
- down, so the oscilloscope can be tilted back for easy viewing when the oscilloscope is on the floor and you are standing above it.

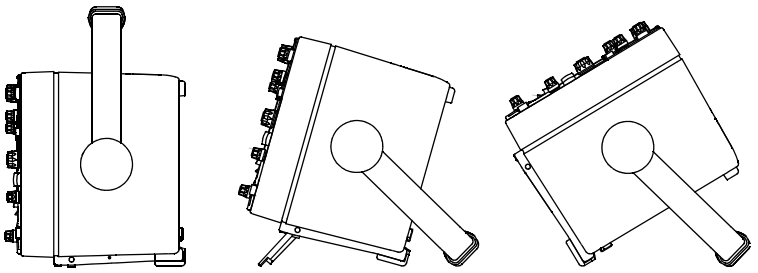
1 To rotate the handle, grasp the handle hubs on each side of the instrument and pull the hubs out until they stop.



2 Without releasing the hubs, rotate the handle to the desired position. Then release the hubs. Continue rotating the handle until it clicks into a set position.

To tilt the oscilloscope for easy viewing

The tilt tabs (underneath the oscilloscope) can be positioned as shown in the center picture below. The handle can be used as a stand when placing the oscilloscope on a floor, as shown in the picture on the right.



To mount the oscilloscope in a rack

The 5000A Series oscilloscopes can be mounted into Electronic Industries Association (EIA) standard 19-inch (487-mm) rack cabinets.

To mount the oscilloscope in a rack, purchase and install the N2916B rack mount kit. Instructions are included in the kit.

To power-on the oscilloscope

1 Connect the power cord to the rear of the oscilloscope, then to a suitable AC voltage source.

The oscilloscope automatically adjusts for input line voltages in the range 100 to 240 VAC. Ensure that you have the correct line cord. See Table 3 on page 22. The line cord provided is matched to the country of origin.

WARNING **Always use a grounded power cord. Do not defeat the power cord ground.**

2 Press the power switch.

The power switch is located on the lower left corner of the front panel. The front panel lights will come on and the oscilloscope will be operational in a few seconds.

Ventilation requirements

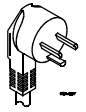
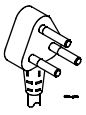
The air intake and exhaust areas must be free from obstructions. Unrestricted air flow is required for proper cooling.

Ventilation Requirements

The fan draws air in from underneath the oscilloscope and pushes it out behind the oscilloscope. Always ensure that the air intake and exhaust areas are free from obstructions.

When using the oscilloscope in a bench-top setting, provide at least 4" (100 mm) clearance behind and above the oscilloscope for proper cooling.

Table 3 Power Cords

Plug Type	Cable Part Number	Plug Type	Cable Part Number
Opt 900 (U.K.) 	8120-1703	Opt 918 (Japan) 	8120-4754
Opt 901 (Australia) 	8120-0696	Opt 919 (Israel) 	8120-6799
Opt 902 (Europe) 	8120-1692	Opt 920 (Argentina) 	8120-6871
Opt 903 (U.S.A.) 	8120-1521	Opt 921 (Chile) 	8120-6979
Opt 906 (Switzerland) 	8120-2296	Opt 922 (China) 	8120-8377
Opt 912 (Denmark) 	8120-2957	Opt 927 (Thailand) 	8120-8871
Opt 917 (South Africa) 	8120-4600		

The remote interface

You can communicate with all 5000A Series oscilloscopes using the front panel keys, or via LAN, USB, or GPIB.

The *Automation Ready CD-ROM* provided with the oscilloscope contains connectivity software to enable communication over these interfaces. Refer to the instructions provided on the CD-ROM to install this software on your PC.

Remote commands can be issued via LAN, USB, or GPIB. These commands are generally used when the oscilloscope is under program control for automated test and data acquisition. Information about controlling the oscilloscope through remote commands is contained in the *Programmer's Quick Start Guide*, which is included in the documentation CD-ROM supplied with this oscilloscope. You can also access this document online. Direct your web browser to www.agilent.com/find/dso5000 and select Technical Support, then select Manuals.

All 5000A Series oscilloscopes feature a built-in Web server (requires software version 4.0 or greater; see page 218 for software updating instructions). Using the Web browser you can set up measurements, monitor waveforms, capture screen images and operate the oscilloscope remotely.

Detailed Connectivity Information

For detailed connectivity information, refer to the Agilent Technologies USB/LAN/GPIB Connectivity Guide. For a printable electronic copy of the Connectivity Guide, direct your Web browser to www.agilent.com and search for Connectivity Guide.

To establish a LAN connection

1. If the controller PC isn't already connected to the local area network (LAN), do that first.
2. Get the oscilloscope's network parameters (hostname, domain, IP address, subnet mask, gateway IP, DNS IP, etc.) from your network administrator.
3. Connect the oscilloscope to the local area network (LAN) by inserting the LAN cable into the "LAN" port on the rear panel of the oscilloscope.
4. On the oscilloscope, ensure the controller interface is enabled:
   a. Press the **Utility** key.
   b. Using the softkeys, press **I/O** and **Control**.
   c. Use the Entry knob to select "LAN"; then, press the **Control** softkey again.
5. Configure the oscilloscope's LAN interface:
   a. Press the **Configure** softkey until "**LAN**" is selected.
   b. Press the **LAN Settings** softkey.
   c. Use the **Config** softkey and the Entry knob to select DHCP, AutoIP, or netBIOS and associated options.
   d. Press the **Addresses** softkey. Use the **Modify** softkey (and the other softkeys and the Entry knob) to enter the IP Address, Subnet Mask, Gateway IP, and DNS IP values. When you are done, press the return (up arrow) softkey.
   e. Press the **Domain** softkey. Use the **Modify** softkey (and the other softkeys and the Entry knob) to enter the Host name and the Domain name. When you are done, press the return (up arrow) softkey.
   f. Press the **Apply** softkey to apply the changes.

NOTE When you connect the oscilloscope to a LAN it is a good practice to limit access to the oscilloscope by setting a password. By default, the oscilloscope is not password protected. See page 31 to set a password.

For more information about connecting to the oscilloscope, refer to the Agilent Technologies Connectivity Guide for Agilent USB, LAN, and GPIB interfaces. For a printable electronic copy of the Connectivity Guide, direct your Web browser to www.agilent.com and search for Connectivity Guide.

To establish a point-to-point LAN connection

The following procedure describes how to establish a point-to-point (stand alone) connection to the oscilloscope. This is useful if you want to control the oscilloscope using a laptop computer or a stand-alone computer.

1. Install Agilent I/O Libraries Suite from the CD that was supplied with the oscilloscope. If you do not have the CD you can download the I/O Libraries Suite from www.agilent.com/find/iolib.
2. Connect your PC to the oscilloscope using a cross-over LAN cable, such as Agilent part number 5061-0701 (order separately).
3. Switch on the oscilloscope power.
4. Press **Utility** → **I/O**. The I/O status will be displayed. Wait for the **LAN Status** to indicate that the oscilloscope is "**configured**". This may take a few minutes.
5. Start the Agilent Connection Expert application from the Agilent I/O Libraries Suite program group.
6. When the Agilent Connection Expert application is displayed, select **Refresh All**.
7. Right Click **LAN** and select **Add Instrument**.
8. In the Add Instrument window, the LAN line should be highlighted; select **OK**.
9. In the LAN Instrument window, select **Find Instruments...**
10. In the Search for instruments on the LAN window, **LAN** and **Look up hostnames** should be checked.
11. Select the **Find Now** key. (NOTE: It may take up to three minutes before the instrument is found. If the instrument is not found the first time, wait about one minute and try again.)
12. When the instrument is found, select **OK** and **OK** to close the Add Instrument windows.

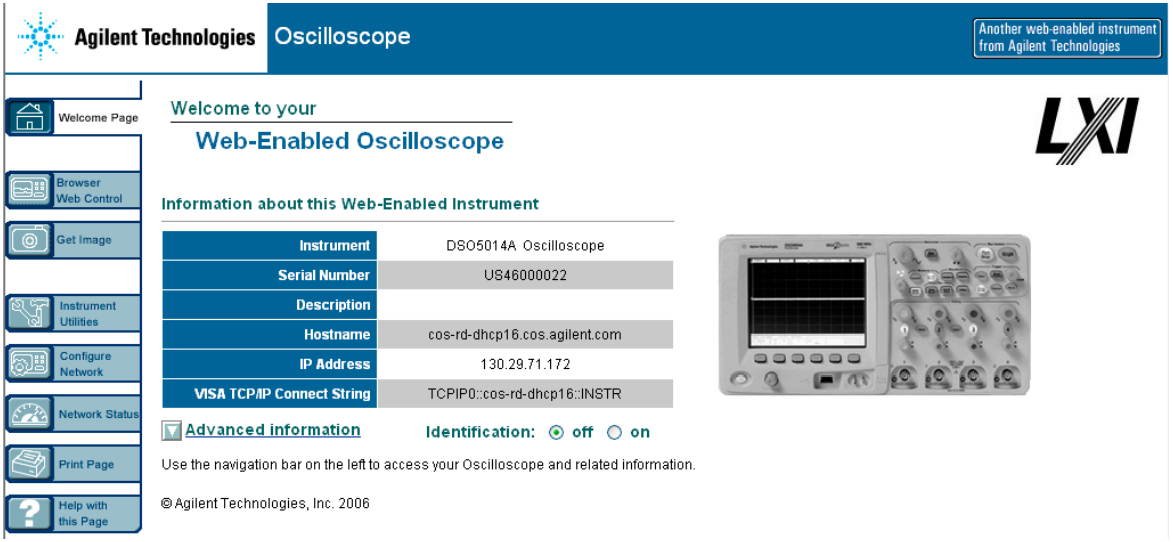
Now the instrument is connected and the instrument's Web interface may be used.

To use the Web interface

All 5000A Series oscilloscopes include a built-in Web server.

When you connect to the oscilloscope using a computer and web browser, you can:

- Control the oscilloscope using the Remote Front Panel function.
- Activate the Identify function (see page 30) to identify a particular instrument by causing it's front panel light to blink.
- View information about the oscilloscope like its model number, serial number, host name, IP address, and VISA address.
- View oscilloscope firmware version information and upload new firmware into the oscilloscope.
- View and modify the oscilloscope's network configuration and status information.



Controlling the oscilloscope using a Web browser

A built-in Web server allows communication and control via a Java™-enabled Web browser. Measurements can be set up, waveforms can be monitored, screen images can be captured, and the oscilloscope can be operated remotely. Also, SCPI (Standard Commands for Programmable Instrumentation) commands can be sent over the LAN.

The recommended Web browser for communication and control of the oscilloscope is Microsoft Internet Explorer 6 or higher. Other Web browsers may work but are not guaranteed to work with the oscilloscope. The Web browser must be Java-enabled with the Sun Microsystems™ Java Plug-in.

To operate the oscilloscope using a Web browser

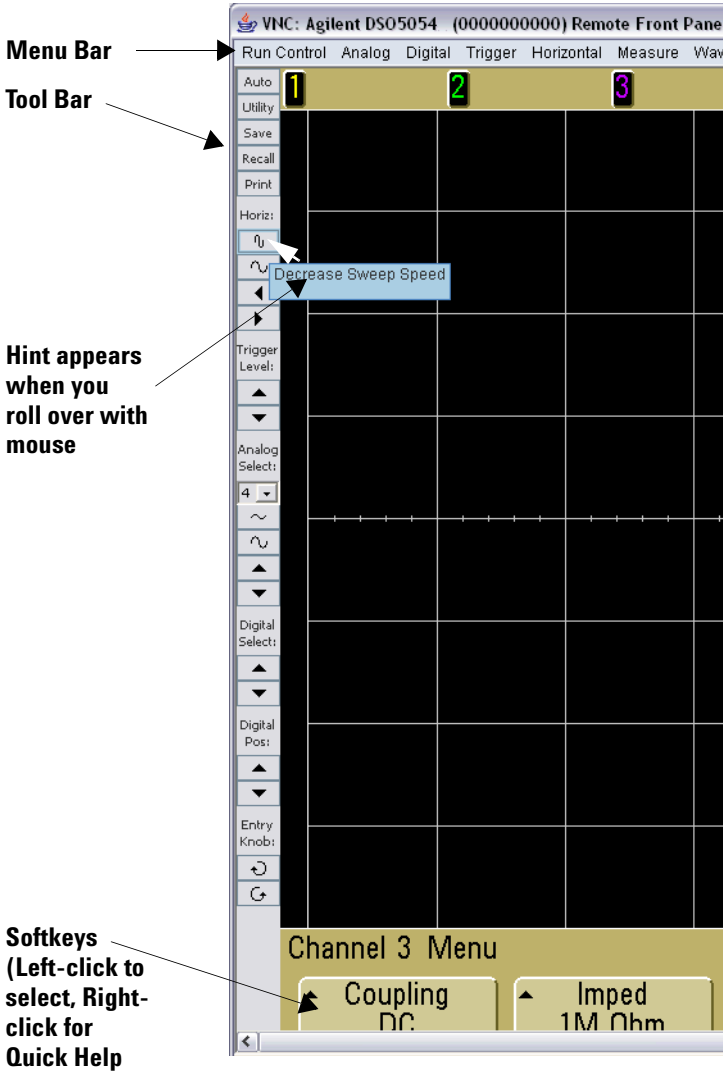
1 Connect the oscilloscope to your LAN (see page 24), or establish a point-to-point connection (see page 26). It is possible to use a point-to-point connection (see page 26), but using a LAN is the preferred method.

2 Type the oscilloscope's host name or IP address into your Web browser to navigate to the oscilloscope's Welcome page.

3 When the oscilloscope's Web page is displayed, select **Browser Web Control**, then select **Remote Front Panel**. After a few seconds the Remote Front Panel appears.

NOTE If Java is not installed on your PC, you will be prompted to install the Sun Microsystems Java Plug-in. This plug-in must be installed on the controlling PC for Remote Front Panel operation.

4 Use the Menu Bar and the Tool Bar to control the oscilloscope. This is a manual way to control an oscilloscope which is normally controlled by a remote program.

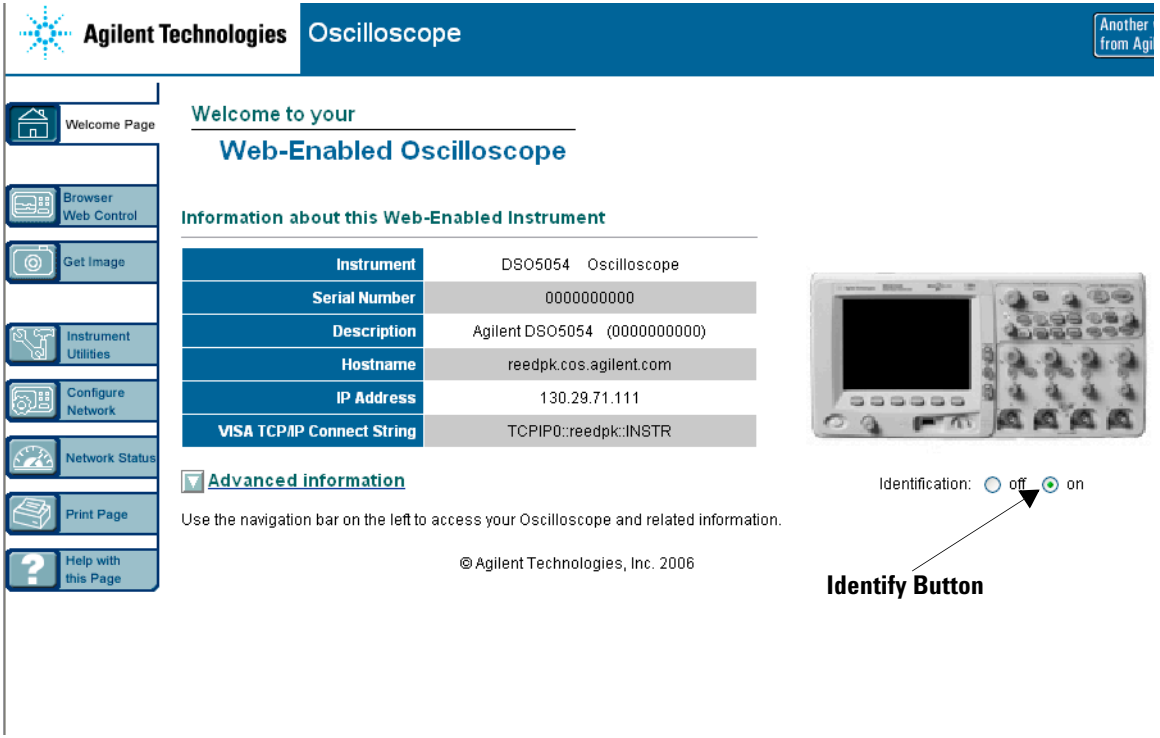

Menu Bar
Tool Bar
Hint appears when you roll over with mouse
Softkeys (Left-click to select, Right-click for Quick Help
VNC: Agilent DSO5054 (0000000000) Remote Front Pane
Run Control
Analog
Digital
Trigger
Horizontal
Measure
Auto
Utility
Save
Recall
Print
Horiz:
Decrease Sweep Speed
Trigger Level:
Analog Select:
Digital Select:
Digital Pos:
Entry Knob:
Channel 3 Menu
Coupling
DC
Imped
1M Ohm

Scrolling and Monitor Resolution

When using a monitor resolution of 1024 x 768 or less on the remote computer, you need to scroll to access the full remote front panel. To display the remote front panel without scroll bars, use a monitor resolution greater than 1024 x 768 on your computer's display.

Identify Function

Select the Identify **On** button (located below the picture of the oscilloscope) on the oscilloscope's Welcome page. An "**Identify**" message will be displayed and you will need to press the **OK** softkey or switch Identify off on the web page to continue. This feature is useful when trying to locate a specific instrument in a rack of equipment.

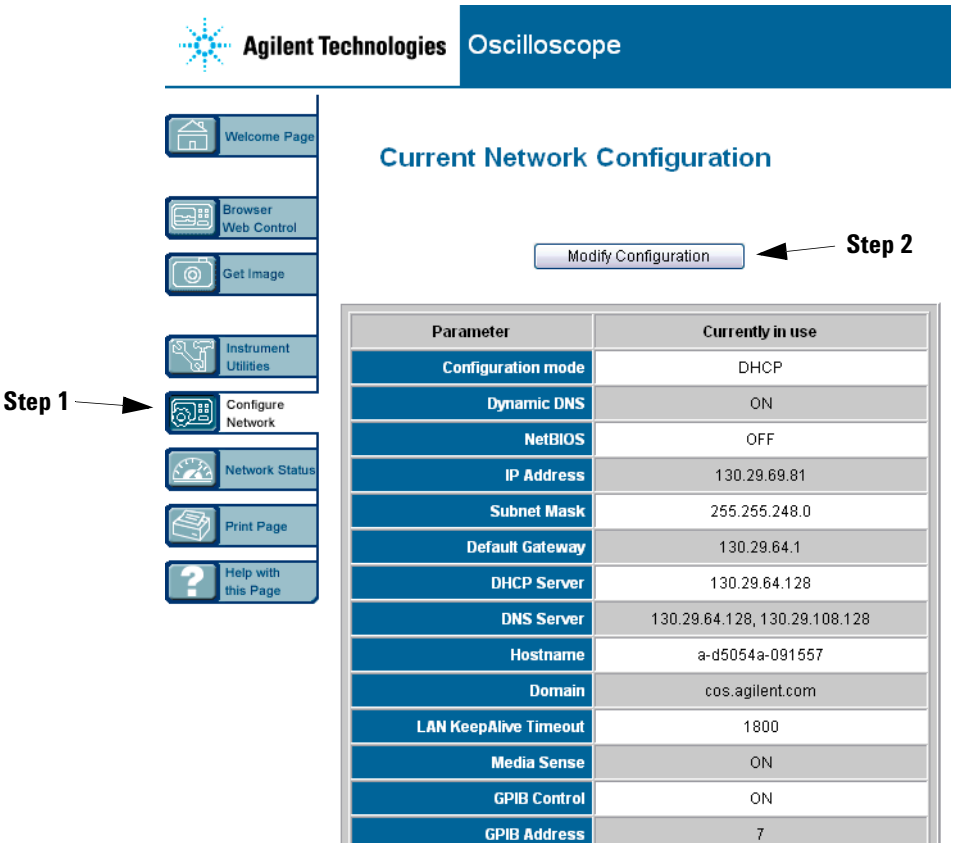


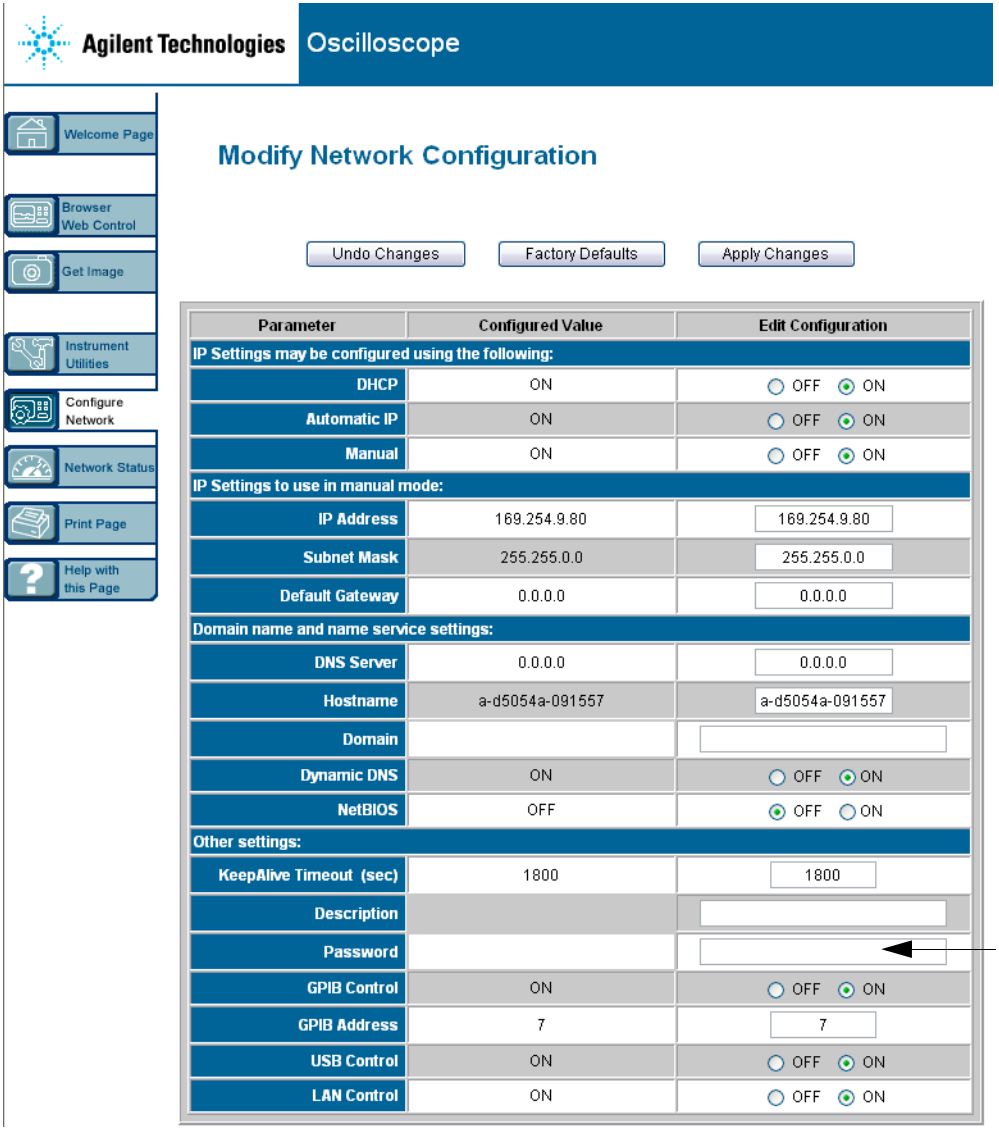
Setting a password

Whenever you connect the oscilloscope to a LAN, it is a good practice to set a password to prevent unauthorized access to the oscilloscope via Web browser.

1 Select the Configure Network tab from the instrument's Welcome page.

2 Select the Modify Configuration button.




Agilent Technologies
Oscilloscope
Welcome Page
Browser Web Control
Get Image
Instrument Utilities
Configure Network
Network Status
Print Page
Help with this Page
Modify Network Configuration
Undo Changes
Factory Defaults
Apply Changes
Parameter
Configured Value
Edit Configuration
IP Settings may be configured using the following:
DHCP
ON
OFF ON
Automatic IP
ON
OFF ON
Manual
ON
OFF ON
IP Settings to use in manual mode:
IP Address
169.254.9.80
169.254.9.80
Subnet Mask
255.255.0.0
255.255.0.0
Default Gateway
0.0.0.0
0.0.0.0
Domain name and name service settings:
DNS Server
0.0.0.0
0.0.0.0
Hostname
a-d5054a-091557
a-d5054a-091557
Domain
Dynamic DNS
ON
OFF ON
NetBIOS
OFF
OFF ON
Other settings:
KeepAlive Timeout (sec)
1800
1800
Description
Password
GPIB Control
ON
OFF ON
GPIB Address
7
7
USB Control
ON
OFF ON
LAN Control
ON
OFF ON
Step 3

3 Enter your desired password.

4 Select the “Apply Changes” button.

To reset the password:

1 Press **Utility** → **I/O** → **LAN Reset**.

For more information about connecting the oscilloscope to a LAN see the Agilent Technologies Connectivity Guide for Agilent USB, LAN, and GPIB interfaces. For a printable electronic copy of the Connectivity Guide, direct your Web browser to www.agilent.com and search for Connectivity Guide.

To connect the oscilloscope probes

Input impedance is selectable: 1 MΩ or 50 Ω Press the channel on/off key (see page 46), then press the **Imped** softkey to select the input impedance.

The 1 MΩ mode is for use with many passive probes and for general purpose measurements. The high impedance minimizes the loading effect of the oscilloscope on the circuit under test.

The 50 Ω mode matches 50 Ω cables and some active probes commonly used in making high frequency measurements. This impedance matching gives you the most accurate measurements since reflections are minimized along the signal path.

1. Connect the supplied oscilloscope probe to an oscilloscope channel BNC connector on the front panel of the oscilloscope.
2. Connect the retractable hook tip on the probe tip to the circuit point of interest. Be sure to connect the probe ground lead to a ground point on the circuit.

CAUTION

Do not exceed 5 Vrms at the BNC in 50 Ω mode on the Agilent 5000A Series oscilloscopes. Input protection is enabled in 50 Ω mode and the 50 Ω load will disconnect if greater than 5 Vrms is detected. However the inputs could still be damaged, depending on the time constant of the signal. The 50 Ω input protection mode on the Agilent 5000A Series oscilloscopes only functions when the oscilloscope is powered on.

CAUTION

The probe ground lead is connected to the oscilloscope chassis and the ground wire in the power cord. If you need to measure between two live points, use a differential probe. Defeating the ground connection and “floating” the oscilloscope chassis will probably result in inaccurate measurements.

WARNING Do not negate the protective action of the ground connection to the oscilloscope. The oscilloscope must remain grounded through its power cord. Defeating the ground creates an electric shock hazard.

CAUTION

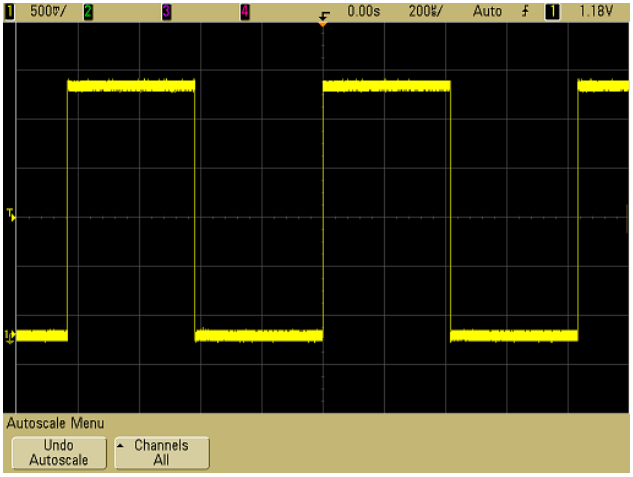
Maximum input voltage for analog inputs:

CAT I 300 Vrms, 400 Vpk; transient overvoltage 1.6 kVpk
CAT II 100 Vrms, 400 Vpk
with N2863A 10:1 probe: CAT I 600 V, CAT II 300 V (DC + peak AC)
with 10073C 10:1 probe: CAT I 500 Vpk, CAT II 400 Vpk

To verify basic oscilloscope operation

To verify that you can display a signal on the oscilloscope:

1. Press the **Save/Recall** key on the front panel, then press the **Default Setup** softkey. (The softkeys are located directly below the display on the front panel.) The oscilloscope is now configured to its default settings.
2. Connect an oscilloscope probe from channel 1 to the **Probe Comp** signal terminal on the front panel.
3. Connect the probe's ground lead to the ground terminal that is next to the **Probe Comp** terminal.
4. Press **AutoScale**.
5. You should see a waveform on the oscilloscope's display similar to this:



If you see the waveform, but the square wave is not shaped correctly as shown above, perform the procedure "To compensate the oscilloscope probes" on page 37.

If you do not see the waveform, ensure your power source is adequate, the oscilloscope is properly powered-on, and the probe is connected securely to the front-panel oscilloscope channel input BNC and to the Probe Comp terminal.

To compensate the oscilloscope probes

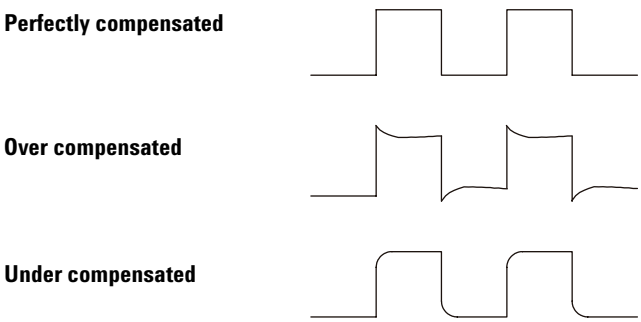
You should compensate your oscilloscope probes to match their characteristics to the oscilloscope's channels. A poorly compensated probe can introduce measurement errors.

To compensate the N2863A probes, follow the procedure that was supplied with the probes.

To compensate the 10073C probes, use the procedure that was supplied with the probes or the following procedure.

1 Perform the procedure "To verify basic oscilloscope operation" on page 36.

2 Use a nonmetallic tool to adjust the trimmer capacitor on the probe for the flattest pulse possible. The trimmer capacitor is located on the probe BNC connector.



3 Connect probes to all other oscilloscope channels (channel 2 of a 2-channel oscilloscope, or channels 2, 3, and 4 of a 4-channel oscilloscope). Repeat the procedure for each channel. This matches each probe to each channel.

The process of compensating the probes serves as a basic test to verify that the oscilloscope is functional.

To calibrate the probes

The oscilloscope can accurately calibrate its oscilloscope channels to certain active probes, such as InfiniiMax probes. Other probes, such as the 10073C and N2863A passive probes, do not require calibration. The Calibrate Probe softkey will be grayed-out (ghosted; displayed in faint text) when a connected probe does not require calibration.

When you connect a probe that can be calibrated (such as an InfiniiMax probe), the **Calibrate Probe** softkey in the channel's menu will become active. Connect the probe to the Probe Comp terminal, and the probe ground to the Probe Comp ground terminal. Press the **Calibrate Probe** softkey and follow the instructions on the display.

NOTE

When calibrating a differential probe, connect the positive lead to the Probe Comp terminal and the negative lead to the Probe Comp ground terminal. You may need to connect an alligator clip to the ground lug to allow a differential probe to span between the Probe Comp test point and ground. A good ground connection ensures the most accurate probe calibration.

Passive Probes Supported

The following passive probes can be used with the 5000A Series oscilloscopes. Any combination of passive probes can be used.

Table 4 Passive Probes

Passive Probes	Quantity Supported
N2863A	4
10070C	4
10073C	4
10074C	4
10076A	4

Active Probes Supported

Active probes that do not have their own external power supply require substantial power from the AutoProbe interface. “Quantity Supported” indicates the maximum number of each type of active probe that can be connected to the oscilloscope. If too much current is drawn from the AutoProbe interface, an error message will be displayed, indicating that you must momentarily disconnect all probes to reset the AutoProbe interface.

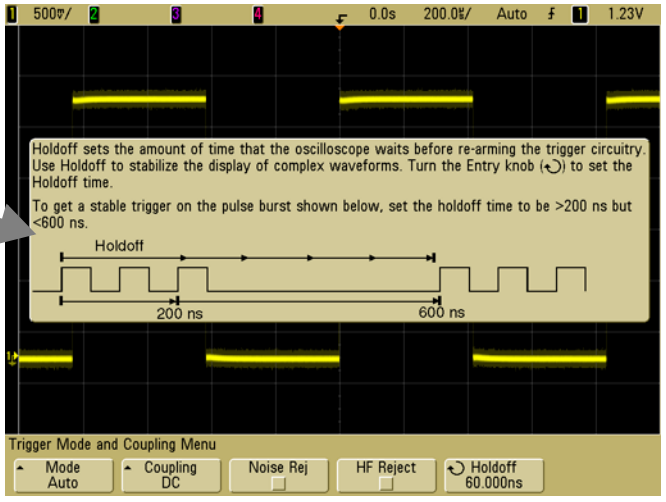
Table 5 Active Probes

Active Probes	Quantity Supported
1130A	2
1131A	2
1132A	2
1134A	2
1141A with 1142A power supply	4
1144A with 1142A power supply	4
1145A with 1142A power supply	2
1147A	2
1156A	4
1157A	4
1158A	4
N2772A with N2773A power supply	4
N2774A with N2775A power supply	4

Using Quick Help

To view Quick Help

1 Press and hold down the key or softkey for which you would like to view help.



Press and Hold Front Panel Key or Softkey
or Right-Click Softkey when using Web browser control

You can set Quick Help to close when you release the key (this is the default mode) or to remain on the screen until another key is pressed or a knob is turned. To select this mode, press the **Utility** key, then press the **Language** softkey, then press the **Help Remain**/**Help Close** softkey.

When Quick Help is used from Web browser control, the Quick Help remains visible until you click on the screen, regardless of whether **Help Remain** or **Help Close** is selected.

Quick Help Languages

To choose a Quick Help language in the oscilloscope:

1. Press **Utility**, then press the **Language** softkey.
2. Repeatedly press and release the **Language** softkey until the desired language is selected.

Quick Help Updates

Updated Quick Help for the 5000 Series oscilloscopes may be made available. If so, it will be available at www.agilent.com/find/dso5000.

1. Direct your web browser to www.agilent.com/find/dso5000sw.
2. Select **Quick Help Language Support** and follow the directions.

2
Front-Panel Controls

Front-Panel Controls

This is an introduction to the front-panel controls of the Agilent 5000A Series oscilloscope. Generally, you set up the front-panel controls and then make a measurement.

The keys on the front panel bring up softkey menus on the display that provide access to oscilloscope features. Many softkeys use the Entry knob to select values.

Six softkeys are located below the display. To understand the symbols used in the softkey menus and throughout this guide, see "Conventions" on page 45.

NOTE The simplest way to set up the oscilloscope is to connect it to the signals of interest and press the **AutoScale** key.

Conventions

Throughout this book, the front-panel keys and softkeys are denoted by a change in font. For example, the **Cursors** key is in the Measure section of the front panel. The **Acq Mode** softkey is the left-most softkey when the Acquire menu is displayed.

Instructions for pressing a series of keys are written in an abbreviated manner. Pressing the **Utility** key, then the **I/O** softkey, then the **Configure LAN** softkey is abbreviated as follows:

Press **Utility** → **I/O** → **Configure LAN**.

Graphic Symbols in Softkey Menus

The following graphic symbols appear in the oscilloscope's softkey menus. The softkey menus appear at the bottom of the display, just above the six softkeys.

Use the Entry knob to adjust the parameter. The Entry knob is located on the front panel. The symbol above the knob is illuminated when this control is active.

Press the softkey to display a pop up with a list of choices. Repeatedly press the softkey until your choice is selected.

Use the Entry knob labeled or press the softkey to adjust the parameter.

✓ Option is selected and operational.

Feature is on. Press the softkey again to turn the feature off.

Feature is off. Press the softkey again to turn the feature on.

Press the softkey to view the menu.

Press the softkey to return to the previous menu.

4-Channel 5000A Series Oscilloscope Front Panel

The following diagram shows the front panel of the 5000A Series 4-channel oscilloscopes. The controls of the 2-channel oscilloscopes are very similar. For a diagram showing the differences of the 2-channel oscilloscope, see page 51.

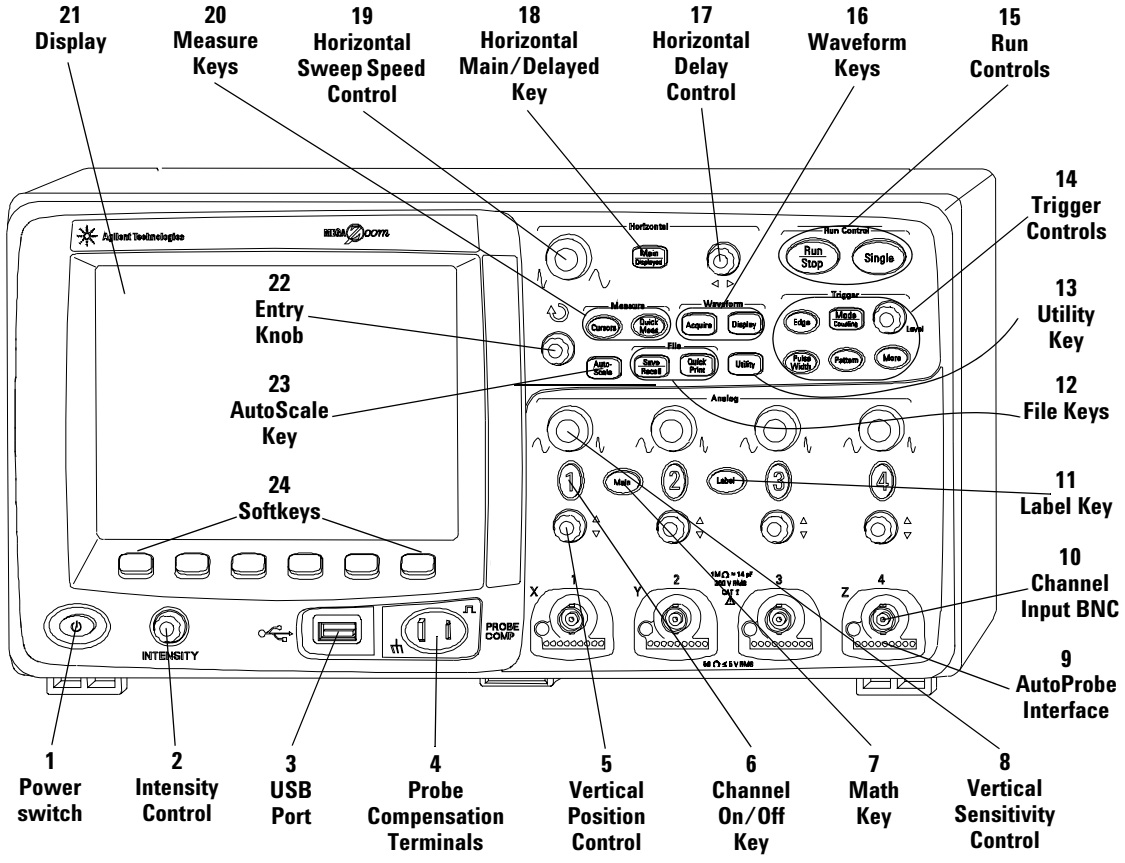


Figure 1 5000A Series 4-Channel Oscilloscope Front Panel

Front Panel Controls

1. Power Switch Press once to switch power on; press again to switch power off. See page 21.

2. Intensity Control Rotate clockwise to increase waveform intensity; counterclockwise to decrease. You can vary the intensity control to bring out signal detail, much like an analog oscilloscope. More details about using the Intensity control to view signal detail are on page 184.

3. USB Host Port Connect a USB compliant mass storage device to store or recall oscilloscope setup files or waveforms. You can also use the USB port to update the oscilloscope's system software or Quick Help language files if updates are available. You do not need to take special precautions before removing the USB mass storage device from the oscilloscope (you do not need to "eject" it). Simply unplug the USB mass storage device from the oscilloscope when the file operation is complete. More information on using the USB port is given in Chapter 6, "Saving and Printing Data," starting on page 199.

CAUTION Only connect USB devices to the USB host port. Do not attempt to connect a host computer to this port to control the oscilloscope. Use the USB device port if you want to connect a host (See the *Oscilloscope Programmer's Quick Start Guide* for details).

4. Probe Compensation Terminals Use the signal at these terminals to match each probe's characteristics to the oscilloscope channel to which it is connected. See page 37.

5. Vertical Position Control Use this knob to change the channel's vertical position on the display. There is one Vertical Position control for each channel. See "Using the channels" on page 60.

6. Channel On/Off Key Use this key to switch the channel on or off, or to access the channel's menu in the softkeys. There is one Channel On/Off key for each channel. See "Using the channels" on page 60.

7. Math Key The Math key provides access to FFT (Fast Fourier Transform), multiply, subtract, differentiate, and integrate functions. See "Math Functions" on page 131.

8. Vertical Sensitivity Use this knob to change the vertical sensitivity (gain) of the channel. See "Using the channels" on page 60.

9. AutoProbe Interface (When you connect a probe to the oscilloscope, the AutoProbe Interface attempts to determine the type of probe and set its parameters in the Probe menu accordingly. See page 58.

10. Channel Input BNC Connector Attach the oscilloscope probe or BNC cable to the BNC connector. This is the channel's input connector.

11. Label Key Press this key to access the Label menu, which lets you enter labels to identify each trace on the oscilloscope display. See page 74.

12. File Keys Press the File key to access file functions such as save or recall a waveform or setup. Or press the Quick Print key to print the waveform from the display. See "Saving and recalling traces and setups" on page 209.

13. Utility Key Press this key to access the Utility menu, which lets you configure the oscilloscope's I/O settings, printer configuration, file explorer, service menu, and other options.

14. Trigger Controls These controls determine how the oscilloscope triggers to capture data. See "Choosing Auto trigger mode or Normal trigger mode" on page 56 and Chapter 3, "Triggering the Oscilloscope," starting on page 89.

15. Run Controls Press Run/Stop to make the oscilloscope begin looking for a trigger. The Run/Stop key will illuminate in green. If the trigger mode is set to "Normal," the display will not update until a trigger is found. If the trigger mode is set to "Auto," the oscilloscope looks for a trigger, and if none is found, it will automatically trigger, and the display will immediately show the input signals. In this case, the background of the **Auto** indicator at the top of the display will flash, indicating that the oscilloscope is forcing triggers.

Press Run/Stop again to stop acquiring data. The key will illuminate in red. Now you can pan across and zoom-in on the acquired data.

Press Single to make a single acquisition of data. The key will illuminate in yellow until the oscilloscope triggers. See "To start and stop an acquisition" on page 53.

16. Waveform Keys The Acquire key lets you set the oscilloscope to acquire in Normal, Peak Detect, Averaging, or High Resolution modes (see "Acquisition Modes" on page 185), and lets you turn Realtime sampling off or on (see page 190). The Display key lets you access the menu where you can select infinite persistence (see page 181), switch vectors on or off (see page 182), and adjust the display grid (graticule) intensity (see page 182).

17. Horizontal Delay Control When the oscilloscope is running, this control lets you set the acquisition window relative to the trigger point. When the oscilloscope is stopped, you can turn this knob to pan through the data horizontally. This lets you see the captured waveform before the trigger (turn the knob clockwise) or after the trigger (turn the knob counterclockwise). See "To set up the Horizontal time base" on page 65.

18. Horizontal Main/Delayed Key Press this key to access the menu where you can split the oscilloscope display into Main and Delayed sections, and where you can select XY and Roll modes. You can also select horizontal time/division vernier and select the trigger time reference point on this menu. See "To set up the Horizontal time base" on page 65.

19. Horizontal Sweep Speed Control Turn this knob to adjust the sweep speed. This changes the time per horizontal division on the display. When adjusted after the waveform has been acquired and the oscilloscope is stopped, this has the effect of stretching out or squeezing the waveform horizontally. See "To set up the Horizontal time base" on page 65.

20. Measure Keys Press the Cursors key to switch on cursors that you can use for making measurements. Press the **Quick Meas** key to access a set of predefined measurements. See Chapter 4, "Making Measurements," starting on page 125.

21. Display The display shows captured waveforms using a different color for each channel. For more information about display modes see Chapter 5, "Displaying Data," starting on page 177. Signal detail is displayed using 256 levels of intensity. For more information about viewing signal detail see "Varying the intensity to view signal detail" on page 184.

22. Entry Knob The entry knob is used to select items from menus and to change values. Its function changes based upon which menu is displayed. Note that the curved arrow symbol  above the entry knob illuminates whenever the entry knob can be used to select a value. Use the entry knob to select among the choices that are shown on the softkeys.

23. AutoScale Key When you press the AutoScale key the oscilloscope will quickly determine which channels have activity, and it will turn these channels on and scale them to display the input signals. See "How AutoScale Works" on page 197

24. Softkeys The functions of these keys change based upon the menus shown on the display directly above the keys.

2-Channel 5000A Series Oscilloscope Front Panel (differences only)

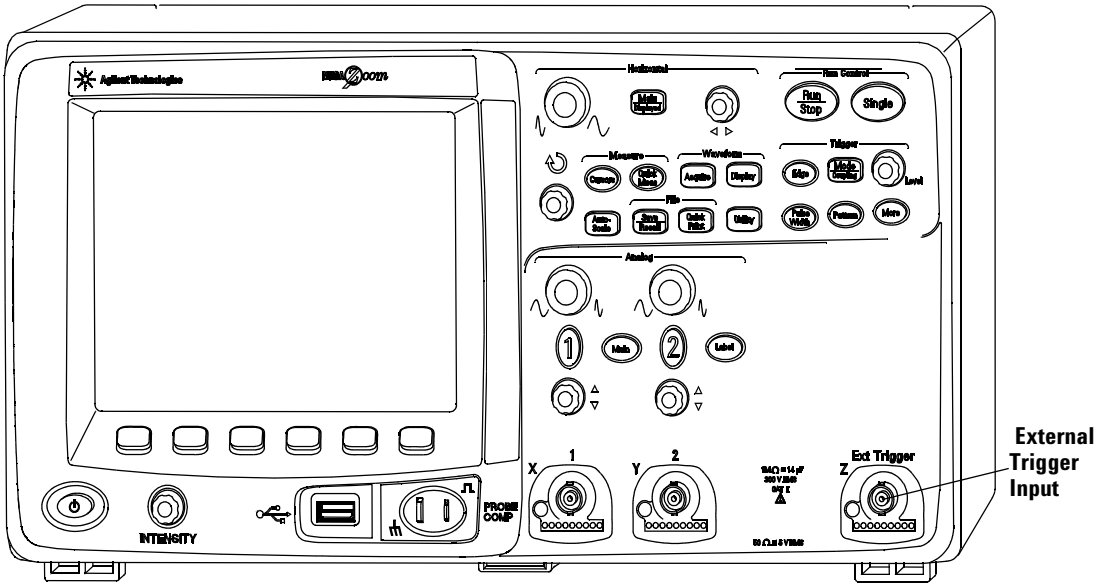


Figure 2 5000A Series 2-Channel Oscilloscope Front Panel

The differences between the front panel of the 4-channel oscilloscopes and the 2-channel oscilloscopes are:

- The 2-channel oscilloscope has two sets of channel controls
- The 2-channel oscilloscope's external trigger input is on the front panel instead of the rear panel. Some trigger features differ. See "The External Trigger input" on page 97.

Interpreting the display

The oscilloscope display contains acquired waveforms, setup information, measurement results, and softkeys for setting up parameters.

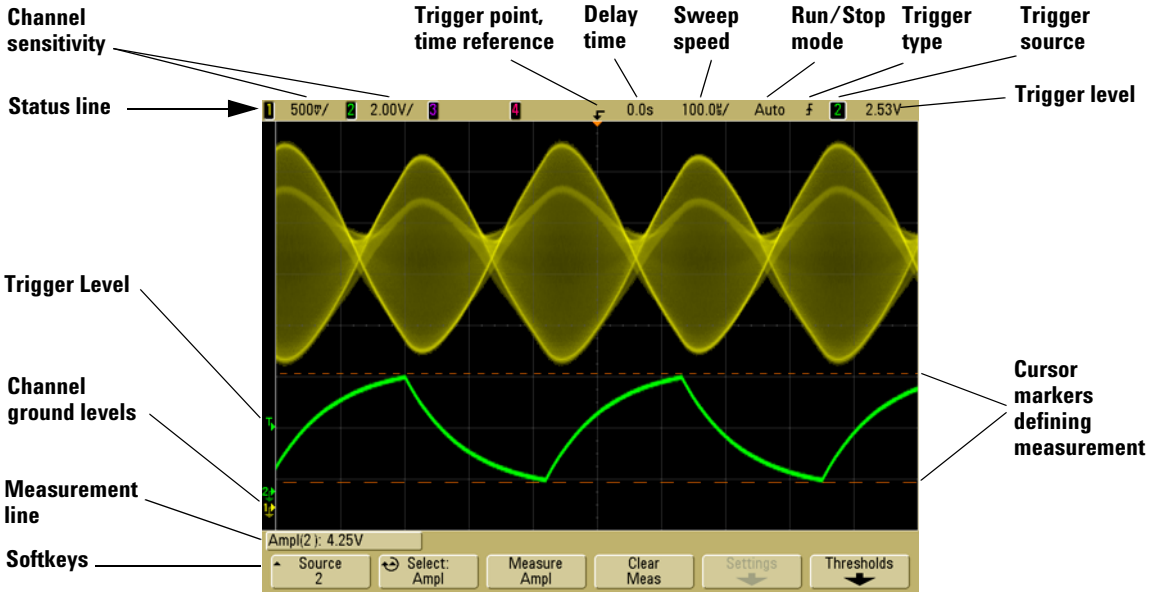


Figure 3 Interpreting the display

Status line The top line of the display contains vertical, horizontal, and trigger setup information.

Display area The display area contains the waveform acquisitions, channel identifiers, and trigger and ground level indicators. Each channel's information appears in a different color.

Measurement line This line normally contains automatic measurement and cursor results, but can also display advanced trigger setup data and menu information.

Softkeys The softkeys let you set up additional parameters for the selected mode or menu.

Front-Panel Operation

This section provides a brief overview of operating the front-panel controls. Detailed oscilloscope operating instructions are provided in later chapters.

To adjust the waveform intensity

The Intensity control is at the lower left corner of the front panel near the power switch.

- Rotate the **Intensity** control clockwise to increase the intensity of displayed waveforms, or counter-clockwise to decrease intensity. For more information see "Varying the intensity to view signal detail" on page 184.

To adjust the display grid (graticule) intensity

1. Press the **Display** key.
2. Turn the Entry knob to change the intensity of the displayed grid. The intensity level is shown in the **Grid** softkey and is adjustable from 0 to 100%.

   Each major vertical division in the grid corresponds to the vertical sensitivity shown in the status line at the top of the display.

   Each major horizontal division in the grid corresponds to the sweep speed time shown in the status line at the top of the display.

To start and stop an acquisition

- When you press the **Run/Stop** key, it illuminates in green and the oscilloscope is in continuous running mode.

  The oscilloscope examines the input voltage at each probe, and updates the display whenever the trigger conditions are met. Trigger processing and screen update rate are optimized based on the oscilloscope settings. The oscilloscope displays multiple acquisitions of the same signal similar to the way an analog oscilloscope displays waveforms.

- When you press the **Run/Stop** key again, it illuminates in red, and the oscilloscope is stopped.

  "Stop" is displayed in the trigger mode position in the status line at the top of the display. You can pan and zoom the stored waveform by turning the horizontal and vertical control knobs.

- When controlling the oscilloscope through its Web interface (see Remote Front Panel on page 28) select **Run Control** from the Main Menu or press ctrl+R for run/stop or ctrl+S for single.

When the oscilloscope is running and you press the **Run/Stop** key, it will flash until the current acquisition is completed. If the acquisition completes immediately, the **Run/Stop** key will not flash.

At slower sweep speeds, you may not want to wait for the acquisition to finish. Just press **Run/Stop** again. The acquisition will stop immediately and a partial waveform will be displayed.

You can display the results of multiple acquisitions using infinite persistence. See "Infinite persistence" on page 181.

Memory Depth/Record Length

Run/Stop versus Single

When the oscilloscope is running, the trigger processing and update rate are optimized over the memory depth.

Single

Single acquisitions always use the maximum memory available—at least twice as much memory as acquisitions captured in Run mode—and the oscilloscope stores at least twice as many samples. At slow sweep speeds, the oscilloscope operates at a higher sample rate when Single is used to capture an acquisition due to the increased memory available. To acquire data with the longest possible record length, press the **Single** key.

Running

When running, versus taking a single acquisition, the memory is divided in half. This allows the acquisition system to acquire one record while processing the previous acquisition, dramatically improving the number of waveforms per second processed by the oscilloscope. While running, maximizing the rate at which waveforms are drawn on the display provides the best picture of your input signal.

To make a single acquisition

When you press the **Single** key it illuminates in yellow and the oscilloscope starts the acquisition system, searching for the trigger condition. When the trigger condition is met, the captured waveform is displayed, the **Single** key is extinguished and the **Run/Stop** key illuminates in red.

- Use the **Single** key to view single-shot events without subsequent waveform data overwriting the display.

Use Single when you want the maximum sample rate and the maximum memory depth for pan and zoom. (See "To pan and zoom" on page 56)

1 Set the trigger mode to Normal (see "Trigger modes: Normal and Auto" on page 92 for instructions).

   This keeps the oscilloscope from automatically triggering immediately.

2 If you are triggering on oscilloscope channel events, turn the Trigger Level knob to the trigger threshold to a level that your waveform will cross.

3 To begin a single acquisition, press the **Single** key.

   When you press **Single**, the display is cleared, the trigger circuitry is armed, the **Single** key is illuminated yellow, and the oscilloscope will wait until a trigger condition occurs before it displays a waveform.

   When the oscilloscope triggers, the single acquisition is displayed and the oscilloscope is stopped (the **Run/Stop** key is illuminated in red).

4 To acquire another waveform, press **Single** again.

Auto Trigger Mode and Single

In Auto trigger mode the oscilloscope will generate a trigger for you if one is not found in the predetermined time (about 40 ms) after you press **Single**. If you want to make a single acquisition and you are not particularly interested in triggering the acquisition (for example, if you are probing a DC level), set the

trigger mode to Auto (see page 92) and press the **Single** key. If a trigger condition occurs within about 40 ms, it will be used; if a trigger doesn't occur, an untriggered acquisition will be taken.

To pan and zoom

You can pan across and zoom-in on a waveform even when the acquisition system is stopped.

1. Press the **Run/Stop** key to stop acquisitions (or press the **Single** key and allow the oscilloscope to acquire the waveform and stop). The **Run/Stop** key is illuminated red when the oscilloscope is stopped.
2. Turn the sweep speed knob to zoom horizontally and turn the volts/division knob to zoom vertically.

   The ∇ symbol at the top of the display indicates the time reference point where the zoom-in/zoom-out is referenced.
3. Turn the Delay Time knob (◂▸) to pan horizontally and turn the channel's vertical position knob (▴▾) to pan vertically.

The stopped display may contain several triggers worth of information, but only the last trigger acquisition is available for pan and zoom.

For more information about Pan and Zoom see page 178.

Choosing Auto trigger mode or Normal trigger mode

In Auto trigger mode the oscilloscope will generate a trigger for you if one is not found in a predetermined amount of time (based on the selected sweep speed) after you press **Run**. If you are probing a DC level and you want to see it displayed, set the trigger mode to Auto (see page 92). If a trigger condition occurs, it will be used; if a trigger doesn't occur, an untriggered acquisition will be taken.

If you press **Run** when the oscilloscope is in Normal trigger mode, a trigger must be detected before the oscilloscope displays an acquisition.

In many cases, a triggered display is not needed to check signal levels or activity. For these applications, use Auto trigger mode (which is the default setting). If you only want to acquire specific events as specified by the trigger settings, use Normal trigger mode.

You can select the trigger mode by pressing the **Mode/Coupling** key, then the **Mode** softkey.

For more detailed discussion of Auto trigger mode and Normal trigger mode, see "Trigger modes: Normal and Auto" on page 92.

Using AutoScale

To configure the oscilloscope quickly, press the **AutoScale** key to display the connected signal(s) that are active.

To undo the effects of AutoScale, press the **Undo AutoScale** softkey before pressing any other key. This is useful if you have unintentionally pressed the **AutoScale** key or do not like the settings AutoScale has selected and want to return to your previous settings.

To make the oscilloscope remain in the acquisition mode you have chosen, press the **AutoScale Acq Mode** softkey and select **Preserve Acquisition Mode**. Otherwise, the acquisition mode will default to **Normal** whenever the **AutoScale** key is pressed.

See also "How AutoScale Works" on page 197.

Example exercise

Example

Connect the oscilloscope probes for channels 1 and 2 to the Probe Comp output on the front panel of the instrument. Be sure to connect the probe ground leads to the ground lug beside the Probe Comp output. Set the instrument to the factory

default configuration by pressing the **Save/Recall** key, then the **Default Setup** softkey. Then press the **AutoScale** key. You should see a display similar to the one shown below.

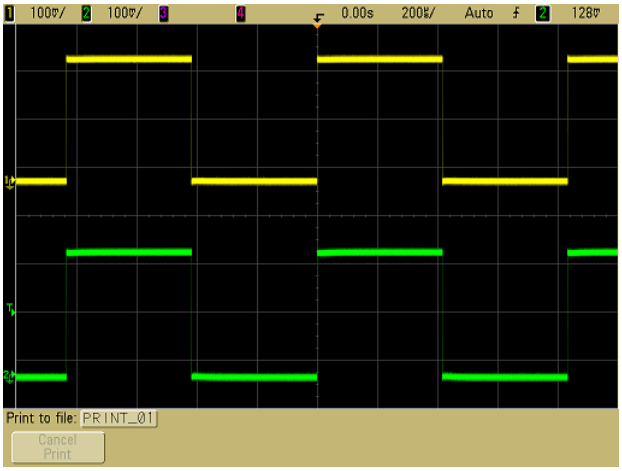


Figure 4 AutoScale of oscilloscope channels 1 and 2

If the waveforms are not perfectly square, you may need to set the probe compensation as described on page 37.

To set the probe attenuation factor

Passive Probes

5000A Series oscilloscopes recognize passive probes such as the N2863A, 10073C and 10074C. These probes have a pin on their connector that connects to the ring around the oscilloscope's BNC connector. Therefore, the oscilloscope will automatically set the attenuation factor for recognized Agilent passive probes.

Passive probes that do not have a pin that connects to the ring around the BNC connector will not be recognized by the oscilloscope, and you must set the probe attenuation factor manually.

Manually Setting the Probe Attenuation Factor

If you connect a probe that the oscilloscope does not automatically identify, you can set the attenuation factor as follows:

1. Press the channel key
2. Press the **Probe** softkey
3. Turn the Entry knob to set the attenuation factor for the connected probe.

The attenuation factor can be set from 0.1:1 to 1000:1 in a 1-2-5 sequence. The probe attenuation factor must be set properly for measurements to be made correctly.

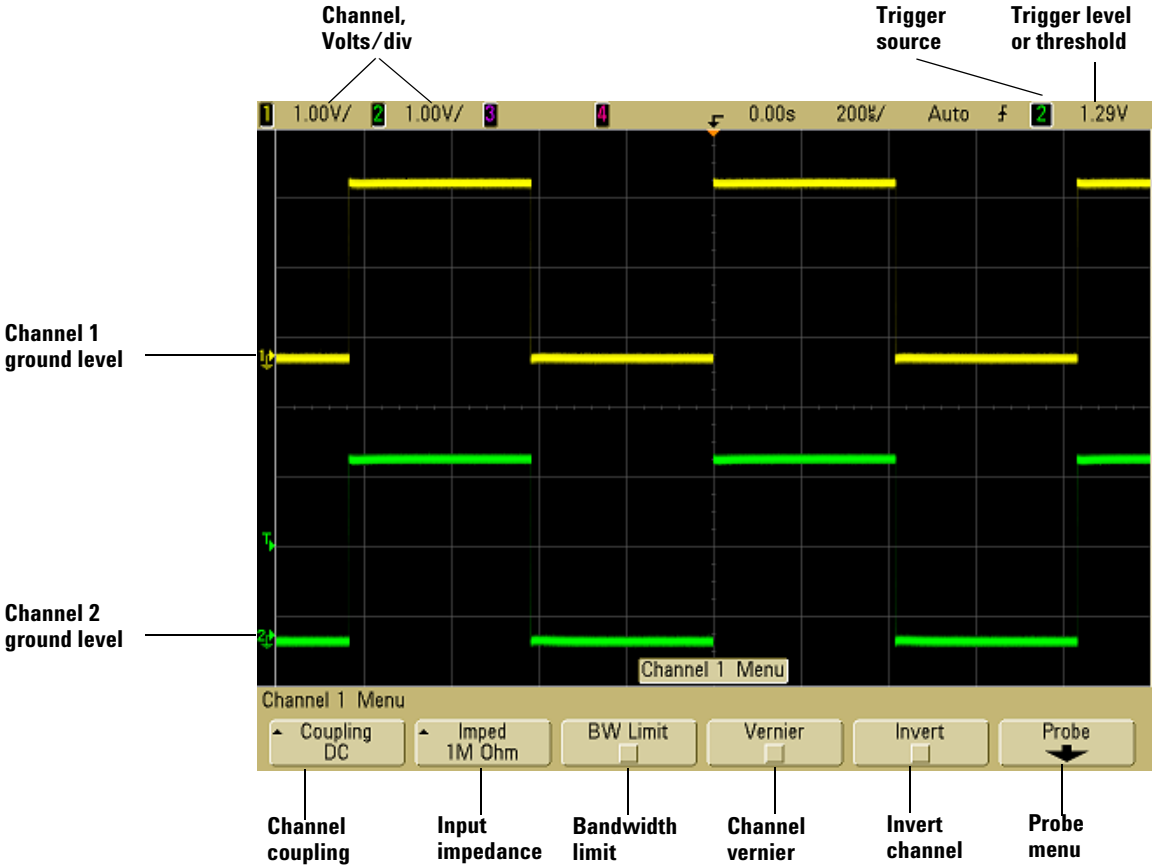
Active Probes

All 5000A Series oscilloscopes have an AutoProbe interface. Most Agilent active probes are compatible with the AutoProbe interface. The AutoProbe interface uses a series of contacts directly below the channel's BNC connector to transfer information between the oscilloscope and the probe. When you connect a compatible probe to the oscilloscope, the AutoProbe interface determines the type of probe and sets the oscilloscope's parameters (units, offset, attenuation, coupling, and impedance) accordingly.

Using the channels

Connect the oscilloscope probes for channels 1 and 2 to the Probe Comp output on the front panel of the instrument.

1 Press the **1** key on the oscilloscope's front panel to display the Channel 1 menu.



Pressing a channel key displays the channel's menu and turns the display of the channel on or off. The channel is displayed when the key is illuminated.

Turning channels off

You must be viewing the menu for a channel before you can turn it off. For example, if channel 1 and channel 2 are turned on and the menu for channel 2 is being displayed, to turn channel 1 off, press **1** to display channel 1 menu, then press **1** again to turn channel 1 off.

Vertical sensitivity Turn the large knob above the channel key to set the sensitivity (volts/division) for the channel. The vertical sensitivity knob changes the channel sensitivity in a 1-2-5 step sequence (with a 1:1 probe attached). The channel Volts/Div value is displayed in the status line.

Vernier Press the **Vernier** softkey to turn on vernier for the selected channel. When Vernier is selected, you can change the channel's vertical sensitivity in smaller increments. The channel sensitivity remains fully calibrated when Vernier is on. The sensitivity value is displayed in the status line at the top of the display.

When Vernier is turned off, turning the volts/division knob changes the channel sensitivity in a 1-2-5 step sequence.

Vertical expansion The default mode for expanding the signal when you turn the volts/division knob is vertical expansion about the ground level of the channel. To set the expansion mode to expand about the center of the screen instead, press **Expand** in the **Utility→Options→Preferences** menu and select **Center**. See also page 80.

Ground level The ground level of the signal for each displayed channel is identified by the position of the icon at the far-left side of the display.

Vertical position Turn the small vertical position knob ( ) to move the channel's waveform up or down on the display. The voltage value momentarily displayed in the upper right portion of the display represents the voltage difference between the vertical center of the display and the ground level ( ) icon. It also represents the voltage at the vertical center of the display if vertical expansion is set to expand about ground.

Measurement Hints

If the channel is DC coupled, you can quickly measure the DC component of the signal by simply noting its distance from the ground symbol.
If the channel is AC coupled, the DC component of the signal is removed, allowing you to use greater sensitivity to display the AC component of the signal.

2 Press the channel's on/off key, then press the **Coupling** softkey to select the input channel coupling.

Coupling changes the channel's input coupling to either **AC** (alternating current) or **DC** (direct current). AC coupling places a 3.5 Hz high-pass filter in series with the input waveform that removes any DC offset voltage from the waveform. When **AC** is selected, "AC" is illuminated on the front panel next to the channel position knob (▲▼).

- DC coupling is useful for viewing waveforms as low as 0 Hz that do not have large DC offsets.
- AC coupling is useful for viewing waveforms with large DC offsets. When AC coupling is chosen, you cannot select 50Ω mode. This is done to prevent damage to the oscilloscope.

Note that Channel Coupling is independent of Trigger Coupling. To change Trigger coupling see page 94.

3 Press the **Imped** (impedance) softkey.

NOTE When you connect an AutoProbe, self-sensing probe, or a compatible InfiniiMax probe, the oscilloscope will automatically configure the oscilloscope to the correct impedance.

The oscilloscope channel input impedance can be set to either **1M Ohm** or **50 Ohm** by pressing the **Imped** softkey.

- **50 Ohm** mode matches 50-ohm cables commonly used in making high frequency measurements, and 50-ohm active probes. This impedance matching gives you the most accurate measurements since reflections are minimized along the signal path. When **50 Ohm** is selected, "50Ω" is illuminated on the front panel next to the channel position knob. The oscilloscope automatically switches to **1 M Ohm** mode to prevent possible damage if AC coupling is selected.
- **1M Ohm** mode is for use with many passive probes and for general-purpose measurements. The higher impedance minimizes the loading effect of the oscilloscope on the circuit under test.

4 Press the **BW Limit** softkey to turn on bandwidth limiting.

Pressing the **BW Limit** softkey turns the bandwidth limit on or off for the selected channel. When bandwidth limit is on, the maximum bandwidth for the channel is approximately 25 MHz. For waveforms with frequencies below this, turning bandwidth limit on removes unwanted high frequency noise from the waveform. The bandwidth limit also limits the trigger signal path of any channel that has **BW Limit** turned on.

When **BW Limit** is selected, "BW" is illuminated on the front panel next to the channel position knob (▲▼).

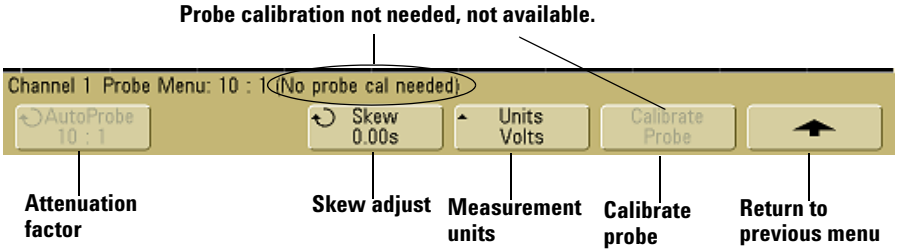
5 Press the **Invert** softkey to invert the selected channel.

When Invert is selected, the voltage values of the displayed waveform are inverted. Invert affects how a channel is displayed, but does not affect triggering. If the oscilloscope is set to trigger on a rising edge, it remains set to trigger on a same edge (triggers at the same point on the waveform) after the channel is inverted.

Inverting a channel will also change the result of any function selected in the Math menu or any measurement.

6 Press the **Probe** softkey to display the channel probe menu.

This menu lets you select additional probe parameters such as attenuation factor and units of measurement for the connected probe.



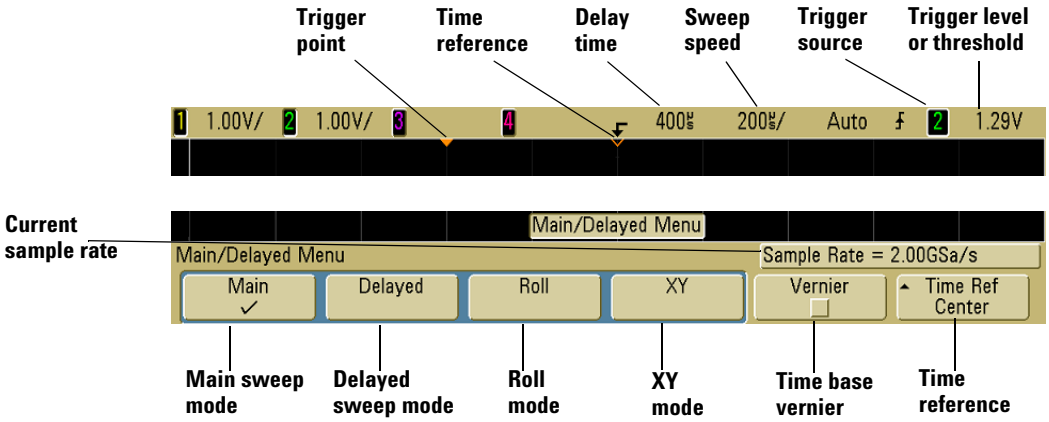
- **Probe Attenuation** — See "To set the probe attenuation factor" on page 58.
- **Skew** — When measuring time intervals in the ns range, small differences in cable length can affect the measurement. Use **Skew** to remove cable-delay errors between any two channels.

  Probe the same point with both probes, then press **Skew** and turn the Entry knob to enter the amount of skew between channels. Each channel can be adjusted ±100 ns in 10 ps increments for a total of 200 ns difference.

  The skew setting is reset to zero when you press **Save/Recall→Default Setup**.
- **Probe Units** — Press the **Units** softkey to select the proper measurement unit for the connected probe. Select **Volts** for a voltage probe and select **Amps** for a current probe. Channel sensitivity, trigger level, measurement results, and math functions will reflect the measurement units you have selected.
- **Calibrate Probe** — See "To calibrate the probes" on page 38.

To set up the Horizontal time base

1 Press the **Main/Delayed** key in the Horizontal section of the front panel.



The Main/Delayed menu lets you select the horizontal mode (Main, Delayed, Roll, or XY), and set the time base vernier and time reference.

The current sample rate is displayed above the **Vernier** and **Time Ref** softkeys.

Main mode

1 Press the **Main** softkey to select Main horizontal mode.

Main horizontal mode is the normal viewing mode for the oscilloscope.When the oscilloscope is stopped, you can use the Horizontal knobs to pan and zoom the waveform.

2 Turn the large knob (time/division) in the Horizontal section and notice the change it makes to the status line.

When the oscilloscope is running in Main mode, use the large Horizontal knob to change sweep speed and use the small knob (◀▶) to set the delay time. When the oscilloscope is

stopped, use these knobs to pan and zoom your waveform. The sweep speed (seconds/division) value is displayed in the status line.

3 Press the **Vernier** softkey to turn on the time base vernier.

The **Vernier** softkey lets you change the sweep speed in smaller increments with the time/division knob. The sweep speed remains fully calibrated when Vernier is on. The value is displayed in the status line at the top of the display.

When Vernier is turned off, the Horizontal sweep speed knob changes the time base sweep speed in a 1-2-5 step sequence.

4 Note the setting of the **Time Ref** (time reference) softkey.

Time reference is the reference point on the display for delay time. Time reference can be set to one major division from the left or right edge, or to the center of the display.

A small hollow triangle (∇) at the top of the display grid marks the position of the time reference. When delay time is set to zero, the trigger point indicator (▼) overlays the time reference indicator.

Turning the Horizontal sweep speed knob will expand or contract the waveform about the time reference point (∇). Turning the Horizontal delay time (◀▶) knob in Main mode will move the trigger point indicator (▼) to the left or right of the time reference point (∇).

The time reference position sets the initial position of the trigger event within acquisition memory and on the display, with delay set to 0. The delay setting sets the specific location of the trigger event with respect to the time reference position. The time reference setting affects the delayed sweep as described in the following ways:

- When the horizontal mode is set to Main, the delay knob positions the main sweep relative to the trigger. This delay is a fixed number. Changing this delay value does not affect the sweep speed.

- When the horizontal mode is set to Delayed, the delay knob controls the position of the delayed sweep window inside the main sweep display. This delay value is independent of sampling interval and sweep speed. Changing this delay value does not effect the position of the main window.

5 Turn the delay knob (◀▶) and notice that its value is displayed in the status line.

The delay knob moves the main sweep horizontally, and it pauses at 0.00 s, mimicking a mechanical detent. Changing the delay time moves the sweep horizontally and indicates how far the trigger point (solid inverted triangle) is from the time reference point (hollow inverted triangle ∇). These reference points are indicated along the top of the display grid. The previous figure shows the trigger point with the delay time set to 400 µs. The delay time number tells you how far the time reference point is located from the trigger point. When delay time is set to zero, the delay time indicator overlays the time reference indicator.

All events displayed left of the trigger point happened before the trigger occurred, and these events are called pre-trigger information. You will find this feature very useful because you can now see the events that led up to the trigger point. Everything to the right of the trigger point is called post-trigger information. The amount of delay range (pre-trigger and post-trigger information) available depends on the sweep speed selected and memory depth.

Delayed mode

Delayed sweep is an expanded version of main sweep. When Delayed mode is selected, the display divides in half and the delayed sweep icon displays in the middle of the line at the top of the display. The top half of the display shows the main sweep and the bottom half displays the delayed sweep.

Delayed sweep is a magnified portion of the main sweep. You can use delayed sweep to locate and horizontally expand part of the main sweep for a more detailed (higher-resolution) analysis of signals.

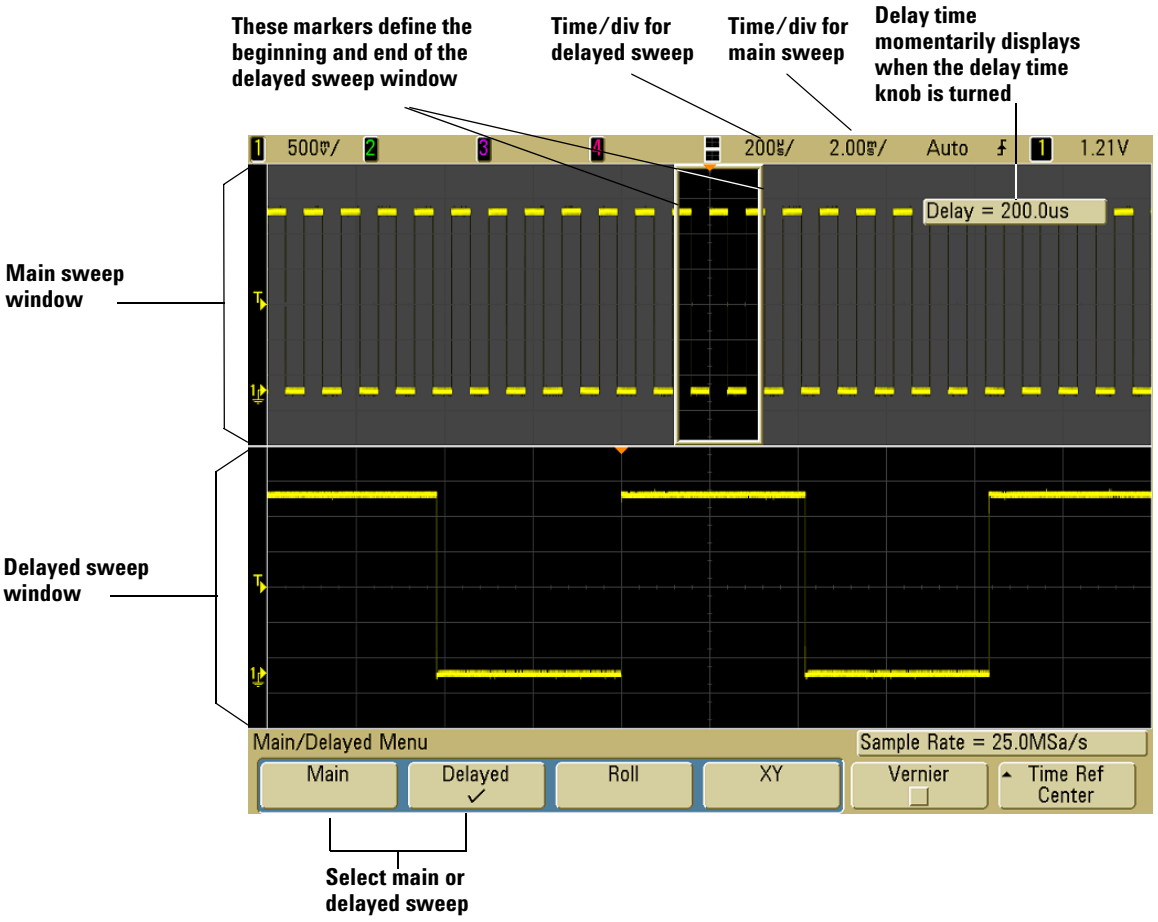
The following steps show you how to use delayed sweep. Notice that the steps are very similar to operating the delayed sweep in analog oscilloscopes.

1. Connect a signal to the oscilloscope and obtain a stable display.
2. Press **Main/Delayed**.
3. Press the **Delayed** softkey.

To change the sweep speed for the delayed sweep window, turn the sweep speed knob. As you turn the knob, the sweep speed is highlighted in the status line above the waveform display area.

The area of the main display that is expanded is intensified and marked on each end with a vertical marker. These markers show what portion of the main sweep is expanded in the lower half. The Horizontal knobs control the size and position of the delayed sweep. The delay value is momentarily displayed in the upper-right portion of the display when the delay time (◀▶) knob is turned.

To change the sweep speed for the main sweep window, press the **Main** softkey, then turn the sweep speed knob.



The area of the main display that is expanded is intensified and marked on each end with a vertical marker. These markers show what portion of the main sweep is expanded in the lower half. The Horizontal knobs control the size and position of the delayed sweep. The delay value is momentarily displayed in the upper-right portion of the display when the delay time (◀▶) knob is turned.

To change the sweep speed for the delayed sweep window, turn the sweep speed knob. As you turn the knob, the sweep speed is highlighted in the status line above the waveform display area.

The time reference position sets the initial position of the trigger event within acquisition memory and on the display, with delay set to 0. The delay setting sets the specific location of the trigger event with respect to the time reference position. The time reference setting affects the delayed sweep in the following ways.

When the horizontal mode is set to Main, the delay positions the main sweep relative to the trigger. This delay is a fixed number. Changing this delay value does not affect the sweep speed. When the horizontal mode is set to Delayed, the delay controls the position of the delayed sweep window inside the main sweep display. This delay value is independent of sampling interval and sweep speed.

To change the sweep speed for the main sweep window, press the **Main** softkey, then turn the sweep speed knob.

For information about using delayed mode for measurements, refer to Chapter 4, “Making Measurements,” starting on page 125.

Roll mode

- Press the **Main/Delayed** key, then press the **Roll** softkey.
- Roll mode causes the waveform to move slowly across the screen from right to left. It only operates on time base settings of 500 ms/div and slower. If the current time base setting is faster than the 500 ms/div limit, it will be set to 500 ms/div when Roll mode is entered.
- In Normal horizontal mode, signal events occurring before the trigger are plotted to the left of the trigger point (t) and signal events after the trigger plotted to the right of the trigger point.

- In Roll mode there is no trigger. The fixed reference point on the screen is the right edge of the screen and refers to the current moment in time. Events that have occurred are scrolled to the left of the reference point. Since there is no trigger, no pre-trigger information is available.

If you would like to pause the display in Roll mode press the **Single** key. To clear the display and restart an acquisition in Roll mode, press the **Single** key again.

Use Roll mode on low-frequency waveforms to yield a display much like a strip chart recorder. It allows the waveform to roll across the display.

XY mode

XY mode changes the display from a volts-versus-time display to a volts-versus-volts display. The time base is turned off. Channel 1 amplitude is plotted on the X-axis and Channel 2 amplitude is plotted on the Y-axis.

You can use XY mode to compare frequency and phase relationships between two signals. XY mode can also be used with transducers to display strain versus displacement, flow versus pressure, volts versus current, or voltage versus frequency.

Use the cursors to make measurements on XY mode waveforms.

For more information about using XY mode for measurements, refer to "To use the XY horizontal mode" on page 126.

Z-Axis Input in XY Display Mode (Blanking)

When you select the XY display mode, the time base is turned off. Channel 1 is the X-axis input, channel 2 is the Y-axis input, and channel 4 (or the External trigger on 2-channel models) is the Z-axis input. If you only want to see portions of the Y versus X display, use the Z-axis input. Z-axis turns the trace on and off (analog oscilloscopes called this Z-axis blanking because it turned the beam on and off). When Z is low (<1.4 V), Y versus X is displayed; when Z is high (>1.4 V), the trace is turned off.

To make cursor measurements

You can use the cursors to make custom voltage or time measurements on oscilloscope signals.

1 Connect a signal to the oscilloscope and obtain a stable display.

2 Press the **Cursors** key. View the cursor functions in the softkey menu:

- **Mode** — Set the cursors to measure voltage and time (Normal), or display the binary or hexadecimal logic value of the displayed waveforms.
- **Source** — selects a channel or math function for the cursor measurements.
- **X Y** — Select either the X cursors or the Y cursors for adjustment with the Entry knob.
- **X1** and **X2** — adjust horizontally and normally measure time.
- **Y1** and **Y2** — adjust vertically and normally measure voltage.
- **X1 X2** and **Y1 Y2** — move the cursors together when turning the Entry knob.

NOTE If you intend to make cursor measurements on a trace that you recall from memory, be sure to recall both setup and trace. See "To recall traces and setups" on page 214.

For more information about using the cursors, see "To make cursor measurements" on page 150.

To make automatic measurements

You can use automatic measurements on any channel source or any running math function. Cursors are turned on to focus on the most recently selected measurement (right-most on the measurement line above the softkeys on the display).

1. Press the **Quick Meas** key to display the automatic measurement menu.
2. Press the **Source** softkey to select the channel or running math function to be measured.

   Only channels or math functions that are displayed are available for measurements. If you choose an invalid source channel for a measurement, the measurement will default to the nearest in the list that makes the source valid.

   If a portion of the waveform required for a measurement is not displayed or does not display enough resolution to make the measurement, the result will be displayed with a message such as greater than a value, less than a value, not enough edges, not enough amplitude, incomplete, or waveform is clipped to indicate that the measurement may not be reliable.
3. Choose a type of measurement by pressing the **Select** softkey, then turn the Entry knob to select the desired measurement from the popup list.
4. Press the **Measure** softkey to make the selected measurement.
5. To stop making measurements and erase the measurement results from the measurement line above the softkeys, press the **Clear Meas** softkey.

For more information about automatic measurements, see "Automatic Measurements" on page 157.

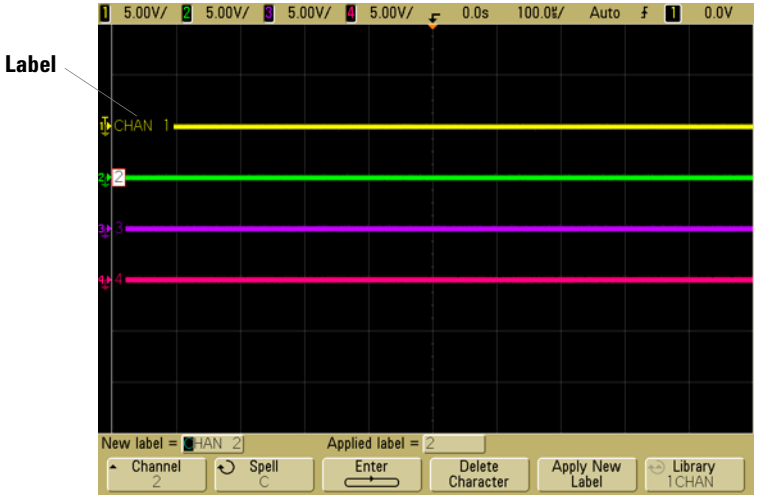
Using Labels

You can define labels and assign them to each input channel, or you can turn labels off to increase the waveform display area.

To turn the label display on or off

1 Press the **Label** key on the front panel.

This turns on display labels for the channels. When the **Label** key is illuminated, the labels for displayed channels are displayed at the left edge of the displayed traces. The channel labels default to their channel number. The figure below shows an example of the label “CHAN 1” assigned to Channel 1.

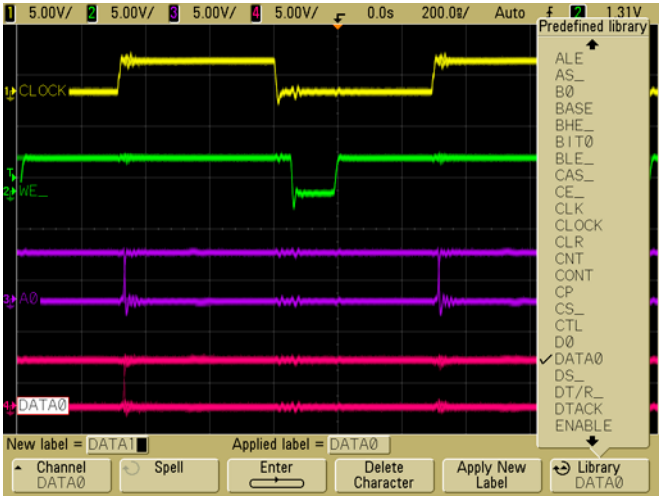


2 To turn the labels off, press the **Label** key until it is not illuminated.

To assign a predefined label to a channel

1 Press the **Label** key.

2 Press the **Channel** softkey, then turn the Entry knob or successively press the **Channel** softkey to select a channel for label assignment.



The channel does not have to be turned on to have a label assigned to it.

3 Press the **Library** softkey, then turn the Entry knob or successively press the **Library** softkey to select a predefined label from the library.

4 Press the **Apply New Label** softkey to assign the label to your selected channel.

5 Repeat the above procedure for each predefined label you want to assign to a channel.

To define a new label

1 Press the **Label** key.

2 Press the **Channel** softkey, then turn the Entry knob or successively press the softkey to select a channel for label assignment.

The channel does not have to be turned on to have a label assigned to it. If the channel is turned on, its current label will be highlighted.

3 Press the **Spell** softkey, then turn the Entry knob to select the first character in the new label.

Turning the Entry knob selects a character to enter into the highlighted position shown in the "`New label =`" line above the softkeys and in the **Spell** softkey. Labels can be up to six characters in length.

4 Press the **Enter** softkey to enter the selected character and to go to the next character position.

You may position the highlight on any character in the label name by successively pressing the **Enter** softkey.

5 To delete a character from the label, press the **Enter** softkey until the letter you want to delete is highlighted, then press the **Delete Character** softkey.

6 When you are done entering characters for the label, press the **Apply New Label** softkey to assign the label to the selected channel.

When you define a new label, it is added to the nonvolatile label list.

Label Assignment Auto-Increment Features

When you assign a label ending in a digit, such as ADDR0 or DATA0, the oscilloscope automatically increments the digit and displays the modified label in the "New label" field after you press the **Apply New Label** softkey. Therefore, you only need to select a new channel and press the **Apply New Label** softkey again to assign the label to the channel. Only the original label is saved in the label list. This feature makes it easier to assign successive labels to numbered control lines and data bus lines.

Label List Management

When you press the **Library** softkey, you will see a list of the last 75 labels used. The list does not save duplicate labels. Labels can end in any number of trailing digits. As long as the base string is the same as an existing label in the library, the new label will not be put in the library. For example, if label A0 is in the library and you make a new label called A12345, the new label is not added to the library.
When you save a new user-defined label, the new label will replace the oldest label in the list. Oldest is defined as the longest time since the label was last assigned to a channel. Any time you assign any label to a channel, that label will move to the newest in the list. Thus, after you use the label list for a while, your labels will predominate, making it easier to customize the instrument display for your needs.
When you reset the label library list (see next topic), all of your custom labels will be deleted, and the label list will be returned to its factory configuration.

To reset the label library to the factory default

1 Press **Utility→Options→Preferences**.

CAUTION Pressing the Default Library softkey will remove all user-defined labels from the library and set the labels back to the factory default. Once deleted, these user-defined labels cannot be recovered.

2 Press the **Default Library** softkey.

This will delete all user-defined labels from the library and set the labels in the library back to the factory default. However, this does not default the labels currently assigned to the channels (those labels that appear in the waveform area).

Defaulting labels without erasing the default library.

Selecting **Default Setup** in the **Save/Recall** menu will set all channel labels back to the default labels but does not erase the list of user-defined labels in the library.

To print the display

You can print the complete display, including the status line and softkeys, to a USB printer or to a USB mass storage device by pressing the **Quick Print** key. You can stop printing by pressing the **Cancel Print** softkey.

To set up your printer, press **Utility→Print Config**.

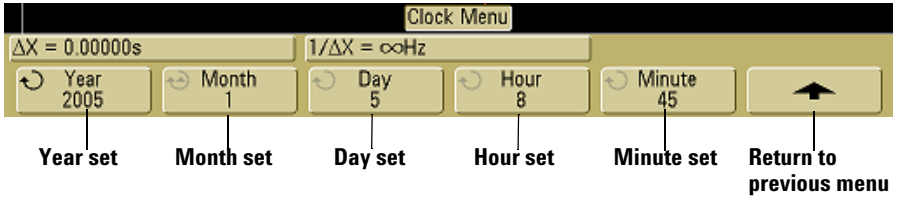
For more information on printing, see "To configure printing" on page 200.

To set the clock

The Clock menu lets you set the current date and time of day (24-hour format). This time/date stamp will appear on hardcopy prints and directory information on the USB mass storage device.

To set the date and time, or to view the current date and time:

1 Press **Utility→Options→Clock**.



2 Press the **Year**, **Month**, **Day**, **Hour** or **Minute** softkey and rotate the Entry knob to set to the desired number.

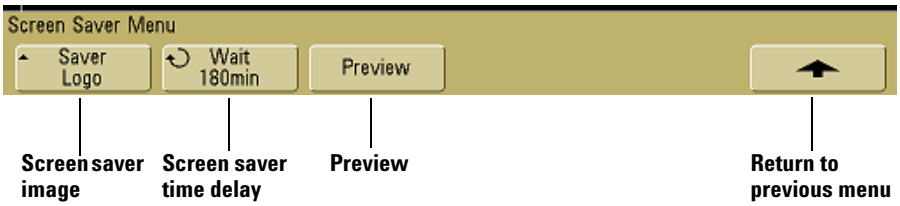
The hours are shown in the 24-hour format. So 1:00 PM is hour 13.

The real-time clock only allows selection of valid dates. If a day is selected and the month or year is changed so the day is invalid, the day is automatically adjusted.

To set up the screen saver

The oscilloscope can be configured to turn on a display screen saver when the oscilloscope has been idle for a specified length of time.

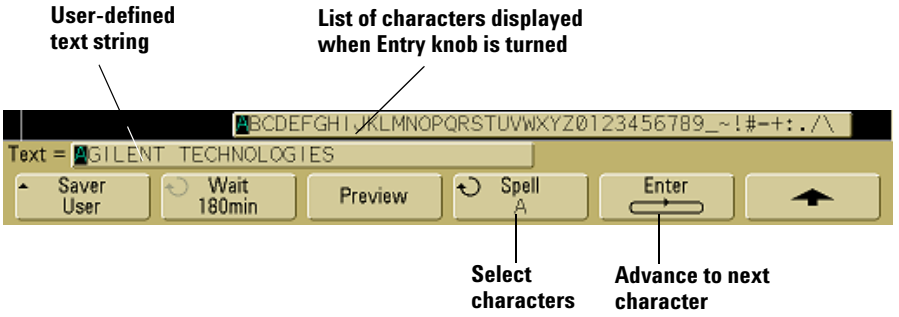
1 Press **Utility→Options→Preferences→Screen Saver** to display the screen saver menu.



2 Press the **Saver** softkey to select the screen saver type.

The screen saver can be set to **Off**, to display any of the images shown in the list, or can display a user-defined text string.

If **User** is selected, press the **Spell** softkey to select the first character of the text string. Use the Entry knob to choose a character. Then press the **Enter** softkey to advance to the next character and repeat the process. The resultant string is displayed in the "`Text =`" line above the softkeys.



3 Turn the Entry knob to select the number of minutes to wait before the selected screen saver activates.

When you turn the Entry knob, the number of minutes is displayed in the **Wait** softkey. The default time is 180 minutes (3 hours).

4 Press the **Preview** softkey to preview the screen saver you have selected with the **Saver** softkey.

5 To view the normal display after the screen saver has started, press any key or turn any knob.

To set the waveform expansion reference point

- Press **Utility→Options→Preferences→Expand** and select **Ground** or **Center**.

When you change a channel's volts/division setting, the waveform display can be set to expand (or compress) about the signal ground level or the center of the display.

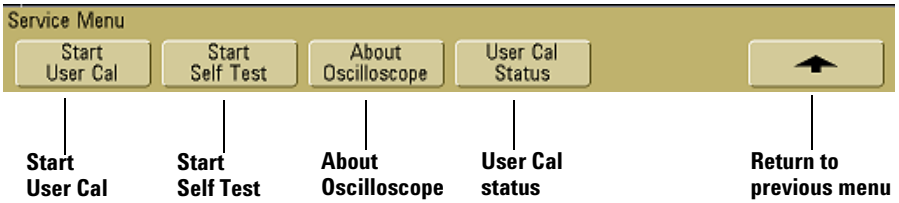
Expand About Ground The displayed waveform will expand about the position of the channel's ground. This is the default setting. The ground level of the signal is identified by the position of the ground level ( ) icon at the far-left side of the display. The ground level will not move when you adjust the vertical sensitivity (volts/division) control.

If the ground level is off screen, the waveform will expand about the top or bottom edge of the screen based on where the ground is off screen.

Expand About Center The displayed waveform will expand about the center of the display.

To perform service functions

- Press the **Utility→Service** to display the service menu.



The Service Menu lets you:

- Perform User Cal on the oscilloscope.
- View User Cal status.
- Perform instrument Self Test.
- View information about your oscilloscope model number, code revision information, and User Cal status.

User Calibration

Perform user-calibration:

- Each year or after 2000 hours of operation.
- If the ambient temperature is >10° C from the calibration temperature.
- If you want to maximize the measurement accuracy.

The amount of use, environmental conditions, and experience with other instruments help determine if you need shorter User Cal intervals.

User Cal performs an internal self-alignment routine to optimize the signal path in the oscilloscope. The routine uses internally generated signals to optimize circuits that affect channel sensitivity, offset, and trigger parameters. Disconnect all inputs and allow the oscilloscope to warm up before performing this procedure.

Performing User Cal will invalidate your Certificate of Calibration. If NIST (National Institute of Standards and Technology) traceability is required, perform the Performance Verification procedure in the *Agilent 5000A Series Oscilloscopes Service Guide* using traceable sources.

To perform User Cal

1. Set the rear-panel CALIBRATION switch to UNPROTECTED.
2. Connect short (12 inch maximum) equal length cables to each oscilloscope channel's BNC connector on the front of the oscilloscope. You will need two equal-length cables for a 2-channel oscilloscope or four equal-length cables for a 4-channel oscilloscope.

   Use 50Ω RG58AU or equivalent BNC cables when performing User Cal.

   For a 2-channel oscilloscope, connect a BNC tee to the equal length cables. Then connect a BNC(f)-to-BNC(f) (also called a barrel connector) to the tee as shown below.

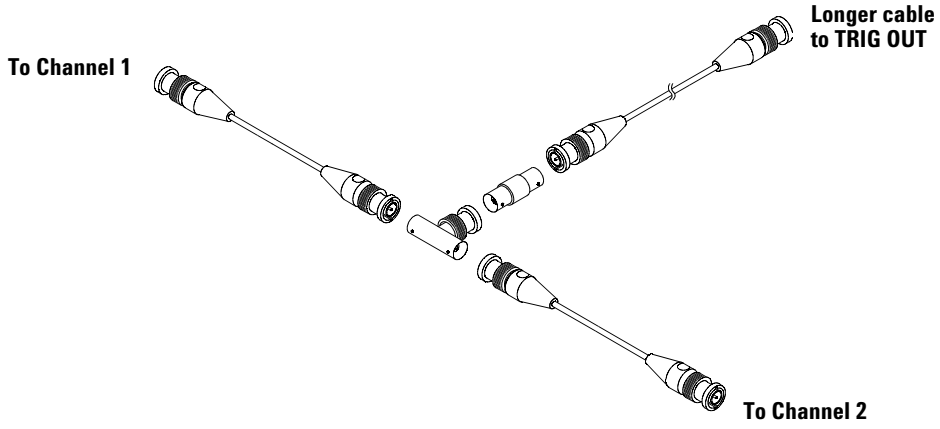


Figure 5 User Calibration cable for 2-channel oscilloscope

For a 4-channel oscilloscope, connect BNC tees to the equal-length cables as shown below. Then connect a BNC(f)-to-BNC(f) (barrel connector) to the tee as shown below.

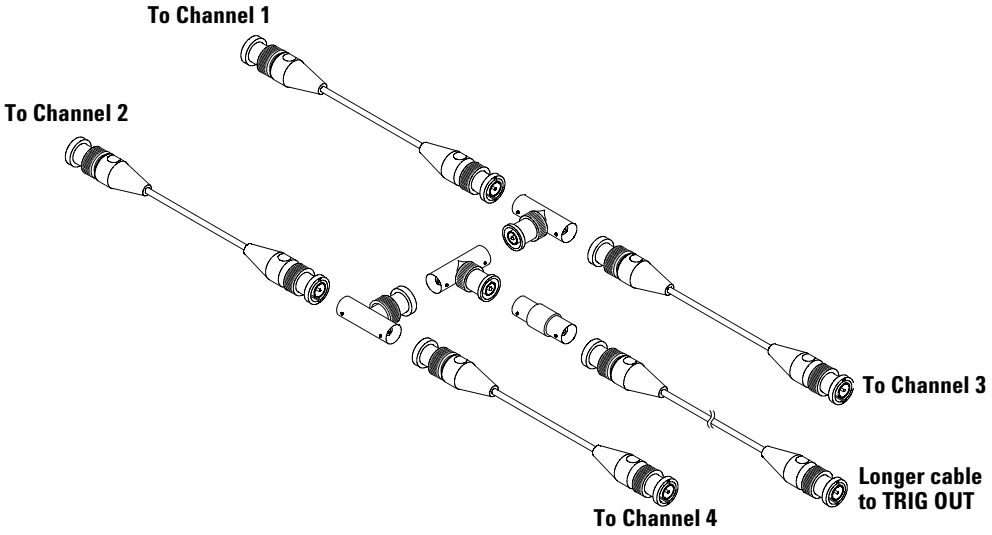


Figure 6 User Calibration cable for 4-channel oscilloscope

1. Connect a BNC cable (40 inches maximum) from the TRIG OUT connector on the rear panel to the BNC barrel connector.
2. Press the **Utility** key, then press the **Service** softkey.
3. Begin the Self Cal by pressing the **Start User Cal** softkey.
4. When the User Cal is completed, set the rear-panel CALIBRATION switch to PROTECTED.

User Cal Status

Pressing **Utility→Service→User Cal Status** displays the summary results of the previous User Cal, and the status of probe calibrations for probes that can be calibrated. Note that passive

probes do not need to be calibrated, but InfiniiMax probes can be calibrated. For more information about calibrating probes see page 38.

Results:
User Cal date:
Change in temperature since last User Cal:
Failure:
Comments:
Probe Cal Status:

Self Test

Pressing **Utility→Service→Start Self Test** performs a series of internal procedures to verify that the oscilloscope is operating properly.

It is recommended you run Self Test:

- after experiencing abnormal operation.
- for additional information to better describe an oscilloscope failure.
- to verify proper operation after the oscilloscope has been repaired.

Successfully passing Self Test does not guarantee 100% of the oscilloscope's functionality. Self Test is designed to provide an 80% confidence level that the oscilloscope is operating properly.

About Oscilloscope

Pressing **Utility→Service→About Oscilloscope** displays information about your oscilloscope model number, serial number, software version, boot version, graphics version, and installed licenses.

Installed licenses:

This line in the About This Oscilloscope dialog contains information about the licenses that have been installed on the oscilloscope. For example, it can show:

- **SEC** — Secure environment mode.
- **None** — No license installed.

To restore the oscilloscope to its default configuration

- Press the **Save/Recall** key, then press the **Default Setup** softkey.

The default configuration returns the oscilloscope to its default settings. This places the oscilloscope in a known operating condition. The major default settings are:

Horizontal main mode, 100 µs/div scale, 0 s delay, center time reference.

Vertical Channel 1 on, 5 V/div scale, DC coupling, 0 V position, 1 MΩ impedance, probe factor to 1.0 if an AutoProbe probe is not connected to the channel.

Trigger Edge trigger, Auto sweep mode, 0 V level, channel 1 source, DC coupling, rising edge slope, 60 ns holdoff time.

Display Vectors on, 33% grid intensity, infinite persistence off.

Other Acquire mode normal, Run/Stop to Run, cursors and measurements off.

Labels All custom labels that you have created in the Label Library are preserved (not erased), but all channel labels will be set to their original names.

3
Triggering the Oscilloscope

The Agilent 5000A Series oscilloscopes provide a full set of features to help automate your measurement tasks. MegaZoom technology lets you capture and examine untriggered waveforms. With these oscilloscopes you can:

- Modify the way the oscilloscope acquires data.
- Set up simple or complex trigger conditions as needed, to capture only the sequence of events you want to examine.

Triggering Features

- Trigger modes:
  - Auto
  - Normal
  - Coupling (DC, AC, low frequency rejection)
  - Noise rejection
  - High frequency rejection
- Holdoff
- Trigger Level
- External Trigger input
- Trigger types:
  - Edge (slope)
  - Pulse width (glitch)
  - Pattern
  - Duration
  - TV
- Trigger Out connector

Selecting Trigger Modes and Conditions

The trigger mode affects the way in which the oscilloscope searches for the trigger. The figure below shows the conceptual representation of acquisition memory. Think of the trigger event as dividing acquisition memory into a pre-trigger and post-trigger buffer. The position of the trigger event in acquisition memory is defined by the time reference point and the delay setting.

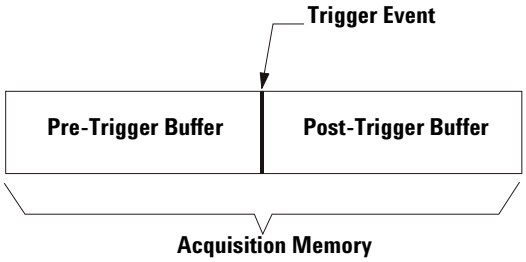
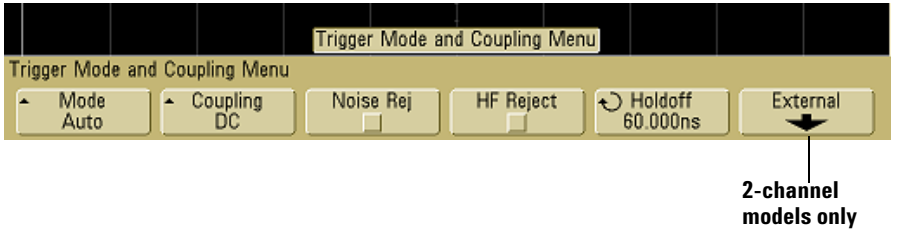


Figure 7 Acquisition Memory

To select the Mode and Coupling menu

- Press the **Mode/Coupling** key in the Trigger section of the front panel.



Trigger modes: Normal and Auto

An introduction to Normal and Auto trigger modes is given on page 56.

1 Press the **Mode/Coupling** key.

2 Press the **Mode** softkey, then select **Normal** or **Auto**.

- **Normal** mode displays a waveform when the trigger conditions are met, otherwise the oscilloscope does not trigger and the display is not updated.
- **Auto** mode is the same as Normal mode, except it forces the oscilloscope to trigger if the trigger conditions are not met.

Auto mode

Use the auto trigger modes for signals other than low-repetitive-rate signals and for unknown signal levels. To display a DC signal, you must use auto trigger mode since there are no edges on which to trigger.

When you select **Run**, the oscilloscope operates by first filling the pre-trigger buffer. It starts searching for a trigger after the pre-trigger buffer is filled, and continues to flow data through this buffer while it searches for the trigger. While searching for the trigger, the oscilloscope overflows the pre-trigger buffer; the first data put into the buffer is the first pushed out (FIFO). When a trigger is found, the pre-trigger buffer will contain the events that occurred just before the trigger. If no trigger is found, the oscilloscope generates a trigger and displays the data as though a trigger had occurred. In this case, the background of the **Auto** indicator at the top of the display will flash, indicating that the oscilloscope is forcing triggers.

When you press the **Single** key, the oscilloscope will fill pre-trigger buffer memory, and continue flowing data through the pre-trigger buffer until the auto trigger overrides the searching and forces a trigger. At the end of the trace, the oscilloscope will stop and display the results.

Normal mode

Use Normal trigger mode for low repetitive-rate signals or when Auto trigger is not required.

In Normal mode the oscilloscope must fill the pre-trigger buffer with data before it will begin searching for a trigger event. The trigger mode indicator on the status line flashes to indicate the oscilloscope is filling the pre-trigger buffer. While searching for the trigger, the oscilloscope overflows the pre-trigger buffer; the first data put into the buffer is the first pushed out (FIFO).

When the trigger event is found, the oscilloscope will fill the post-trigger buffer and display the acquisition memory. If the acquisition was initiated by **Run/Stop**, the process repeats. If the acquisition was initiated by pressing **Single**, then the acquisition stops and you can Pan and Zoom the waveform.

In either Auto or Normal mode, the trigger may be missed completely under certain conditions. This is because the oscilloscope will not recognize a trigger event until the pre-trigger buffer is full. Suppose you set the Time/Div knob to a slow sweep speed, such as 500 ms/div. If the trigger condition occurs before the oscilloscope has filled the pre-trigger buffer, the trigger will not be found. If you use Normal mode and wait for the trigger condition indicator to flash before causing the action in the circuit, the oscilloscope will always find the trigger condition.

Some measurements you want to make will require you to take some action in the circuit under test to cause the trigger event. Usually, these are single-shot acquisitions, where you will use the **Single** key.

To select trigger Coupling

1. Press the **Mode/Coupling** key.
2. Press the **Coupling** softkey, then select **DC, AC,** or **LF Reject** coupling.
   - **DC** coupling allows DC and AC signals into the trigger path.
   - **AC** coupling places a 10 Hz high-pass filter in the trigger path removing any DC offset voltage from the trigger waveform. The high-pass filter in the External Trigger input path is 3.5 Hz for all models. Use AC coupling to get a stable edge trigger when your waveform has a large DC offset.
   - **LF** (low frequency) **Reject** coupling places a 50 kHz high-pass filter in series with the trigger waveform. Low frequency reject removes any unwanted low frequency components from a trigger waveform, such as power line frequencies, that can interfere with proper triggering. Use this coupling to get a stable edge trigger when your waveform has low frequency noise.
   - **TV** coupling is normally grayed-out, but is automatically selected when TV trigger is enabled in the Trigger More menu.

Note that Trigger Coupling is independent of Channel Coupling. To change Channel Coupling see page 62.

To select trigger Noise Rejection and HF rejection

1. Press the **Mode/Coupling** key.
2. Press the **Noise Rej** softkey to select noise reject or press the **HF Reject** softkey to select high frequency reject.
   - **Noise Rej** adds additional hysteresis to the trigger circuitry. When noise reject is on, the trigger circuitry is less sensitive to noise but may require a greater amplitude waveform to trigger the oscilloscope.

- **HF Reject** adds a 50 kHz low-pass filter in the trigger path to remove high frequency components from the trigger waveform. You can use HF Reject to remove high-frequency noise, such as AM or FM broadcast stations or noise from fast system clocks, from the trigger path.

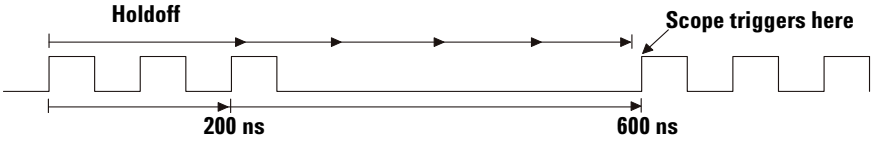
To set Holdoff

1 Press the **Mode/Coupling** key.

2 Turn the Entry knob to increase or decrease the trigger holdoff time shown in the **Holdoff** softkey.

Holdoff sets the amount of time that the oscilloscope waits before re-arming the trigger circuitry. Use Holdoff to stabilize the display of complex waveforms.

To get a stable trigger on the pulse burst shown below, set the holdoff time to be >200 ns but <600 ns.



By setting the Holdoff, you can synchronize triggers. The oscilloscope will trigger on one edge of the waveform, and ignore further edges until the holdoff time expires. The oscilloscope will then re-arm the trigger circuit to search for the next edge trigger. This allows the oscilloscope to trigger on a repeating pattern in a waveform.

Holdoff Operating Hints

Holdoff keeps a trigger from occurring until after a certain amount of time has passed since the last trigger. This feature is valuable when a waveform crosses the trigger level multiple times during one period of the waveform.

Without holdoff, the oscilloscope could trigger on each of the crossings, producing a confusing waveform. With holdoff set correctly, the oscilloscope always triggers on the same crossing. The correct holdoff setting is typically slightly less than one period. Set the holdoff to this time to generate a unique trigger point. This action works even though many waveform periods pass between triggers, because the holdoff circuit operates on the input signal continuously.

Changing the time base settings does not affect the holdoff number. In contrast, the holdoff in analog oscilloscopes is a function of the time base setting, making it necessary to readjust the holdoff each time you change the time base setting.

With Agilent's MegaZoom technology, you can press **Stop**, then pan and zoom through the data to find where it repeats. Measure this time using the cursors, then set the holdoff.

The External Trigger input

The External Trigger input can be used as a source in several of the trigger types.

On 2-channel oscilloscopes, the external trigger BNC input is on the front panel and is labeled **Ext Trigger**.

On 4-channel oscilloscopes, the external trigger BNC input is on the rear panel and is labeled **Ext Trig**.

For trigger system specifications, see page 235.

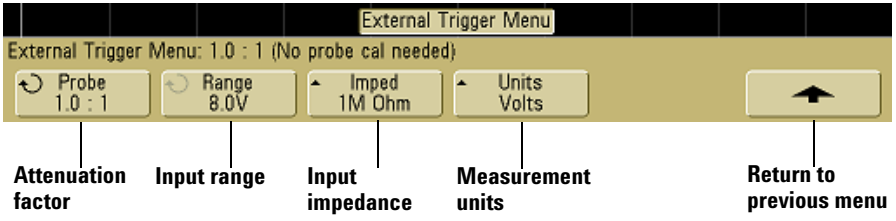
2-Channel oscilloscope External Trigger input

External Trigger Probe Settings

You can set external trigger probe parameters as described below.

1 Press the **Mode/Coupling** key in the Trigger section of the front panel.

2 Press the **External** softkey to display the external trigger probe menu.



Probe Attenuation Turn the Entry knob to set the attenuation factor displayed in the **Probe** softkey for the connected probe. The attenuation factor can be set from 0.1:1 to 1000:1 in a 1-2-5 sequence.

When you connect an AutoProbe self-sensing probe, the oscilloscope will automatically configure your probe to the correct attenuation factor.

The probe correction factor must be set properly for measurements to be made correctly.

Range The input voltage range can be set to 1.0 Volts or 8.0 Volts. When in current mode, the range is fixed at 1.0 Amps. Range is automatically scaled according to the probe's attenuation factor.

Maximum input voltage for the external trigger input of the 2-channel oscilloscope:

CAUTION

Maximum input voltage for analog inputs:

CAT I 300 Vrms, 400 Vpk; transient overvoltage 1.6 kVpk
CAT II 100 Vrms, 400 Vpk
with N2863A 10:1 probe: CAT I 600 V, CAT II 300 V (DC + peak AC)
with 10073C 10:1 probe: CAT I 500 Vpk, CAT II 400 Vpk

CAUTION

Do not exceed 5 Vrms in 50 Ω mode on the 2-channel models. Input protection is enabled in 50 Ω mode and the 50 Ω load will disconnect if greater than 5 Vrms is detected. However the input could still be damaged, depending on the time constant of the signal.

CAUTION

The 50 Ω input protection mode only functions when the oscilloscope is powered on.

Input Impedance The external trigger input impedance can be set to either **1M Ohm** or **50 Ohm** by pressing the **Imped** softkey.

- **50 Ohm** mode matches 50-ohm cables commonly used in making high frequency measurements. This impedance matching gives you the most accurate measurements since reflections are minimized along the signal path.
- **1M Ohm** mode is for use with many passive probes and for general-purpose measurements. The higher impedance minimizes the loading effect of the oscilloscope on the circuit under test.

Probe Units Press the **Units** softkey to select the proper measurement unit for the connected probe. Select **Volts** for a voltage probe and select **Amps** for a current probe. Measurement results, channel sensitivity, and trigger level will reflect the measurement units you have selected.

4-Channel oscilloscope External Trigger input

Input Impedance The external trigger input impedance for the 4-channel oscilloscope is approximately 1.015 kΩ

Input Voltage The input voltage sensitivity is 500 mV, from DC to 100 MHz. The input voltage range is ±15 V.

CAUTION Do not exceed 15 Vrms at rear panel External Trigger input or damage to the oscilloscope may occur.

There are no range or units settings for the External Trigger input of the 4-channel oscilloscope.

Trigger Types

The oscilloscope lets you synchronize the display to the actions of the circuit under test by defining a trigger condition. You can use any input channel or the Ext Trigger BNC as the source for most trigger types.

MegaZoom Technology Simplifies Triggering

With the built-in MegaZoom technology, you can simply AutoScale the waveforms, then stop the oscilloscope to capture a waveform. You can then pan and zoom through the data using the Horizontal and Vertical knobs to find a stable trigger point. AutoScale often produces a triggered display.

These trigger types are available and are presented in the following order in this chapter:

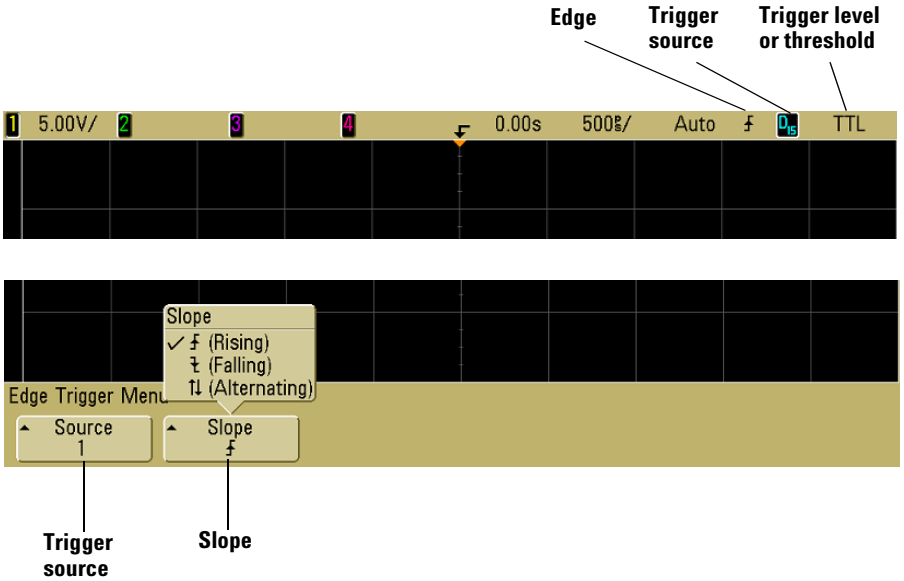
- Edge trigger
- Pulse Width (glitch) trigger
- Pattern trigger
- Duration trigger
- TV trigger

Changes to the trigger specification are applied when you make them. If the oscilloscope is stopped when you change a trigger specification, the oscilloscope will use the new specification when you press **Run/Stop** or **Single**. If the oscilloscope is running when you change a triggering specification, it uses the new trigger definition when it starts the next acquisition.

To use Edge triggering

The Edge trigger type identifies a trigger by looking for a specified edge (slope) and voltage level on a waveform. You can define the trigger source and slope in this menu. The slope can be set to rising edge, falling edge, or alternating edges on all sources except Line. The trigger type, source, and level are displayed in the upper-right corner of the display.

1 Press the **Edge** key in the Trigger section of the front panel to display the Edge trigger menu.



2 Press the **Slope** softkey and select rising edge, falling edge, or alternating edges. The selected slope is displayed in the upper-right corner of the display.

NOTE Alternating edge mode is useful when you want to trigger on both edges of a clock (for example, DDR signals). All modes operate up to the bandwidth of the oscilloscope.

3 Select the trigger source.

You can select oscilloscope channel 1 or 2, Ext, or Line as the trigger source on any Agilent 5000A Series oscilloscope. The trigger source can also be set to channel 3 and 4 on 4-channel oscilloscopes. You can choose a channel that is turned off (not displayed) as the source for the edge trigger.

The selected trigger source is indicated in the upper-right corner of the display next to the slope symbol:

1 through **4** = oscilloscope channels

E = External trigger

L = Line trigger

Trigger level adjustment

You can adjust the trigger level for a selected oscilloscope channel by turning the Trigger Level knob. The position of the trigger level for the channel is indicated by the trigger level icon T▸ (if the channel is on) at the far left side of the display when DC coupling is selected. The value of the oscilloscope channel trigger level is displayed in the upper-right corner of the display.

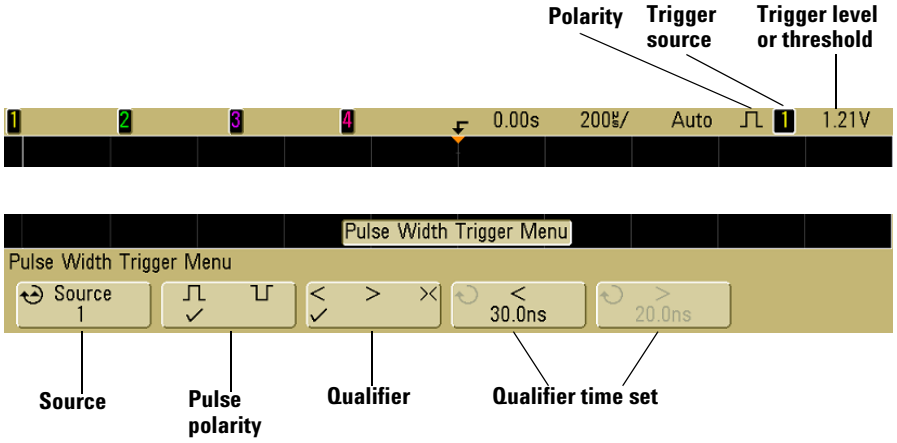
When **Ext** (External Trigger) is selected, its level can be adjusted using the **Level** knob in the Trigger section of the front panel. The trigger level is displayed in the upper right corner of the display.

The line trigger level is not adjustable. This trigger is synchronized with the power line supplied to the oscilloscope.

To use Pulse Width triggering

Pulse Width (glitch) triggering sets the oscilloscope to trigger on a positive or negative pulse of a specified width. If you want to trigger on a specific timeout value, use **Duration** trigger in the Trigger **More** menu.

1 Press the **Pulse Width** key in the Trigger section of the front panel to display the Pulse Width trigger menu.



2 Press the **Source** softkey (or rotate the Entry knob on mixed-signal oscilloscopes) to select a channel source for the trigger.

The channel you select is shown in the upper-right corner of the display next to the polarity symbol.

The source can be any channel available on your oscilloscope. External trigger may also be specified as a source when using a 2-channel oscilloscope.

Adjust the trigger level for the selected channel by turning the Trigger Level knob. The value of the trigger level is displayed in the upper-right corner of the display.

3 Press the pulse polarity softkey to select positive ( )or negative ( ) polarity for the pulse width you want to capture.

The selected pulse polarity is displayed in the upper-right corner of the display. A positive pulse is higher than the current trigger level or threshold and a negative pulse is lower than the current trigger level or threshold.

When triggering on a positive pulse, the trigger will occur on the high to low transition of the pulse if the qualifying condition is true. When triggering on a negative pulse, the trigger will occur on the low to high transition of the pulse if the qualifying condition is true.

4 Press the qualifier softkey (**< > ><**) to select the time qualifier.

The Qualifier softkey can set the oscilloscope to trigger on a pulse width that is:

- less than a time value (**<**).

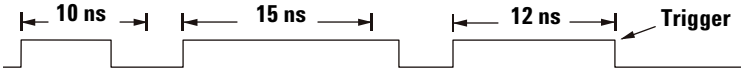
  For example, for a positive pulse, if you set t<10 ns:

- greater than a time value (**>**).

  For example, for a positive pulse, if you set t>10 ns:

- within a range of time values (**><**).

  For example, for a positive pulse, if you set t>10 ns and t<15 ns:



5 Select the qualifier time set softkey (**<** or **>**), then rotate the Entry knob to set the pulse width qualifier time.

The qualifiers can be set as follows:

- 2 ns to 10 s for > or < qualifier (5 ns to 10 s for 100 MHz and 300 MHz bandwidth models)
- 10 ns to 10 s for >< qualifier, with minimum difference of 5 ns between upper and lower settings

< qualifier time set softkey

- When the less than (<) qualifier is selected, the Entry knob sets the oscilloscope to trigger on a pulse width less than the time value displayed on the softkey.
- When the time range (><) qualifier is selected, the Entry knob sets the upper time range value.

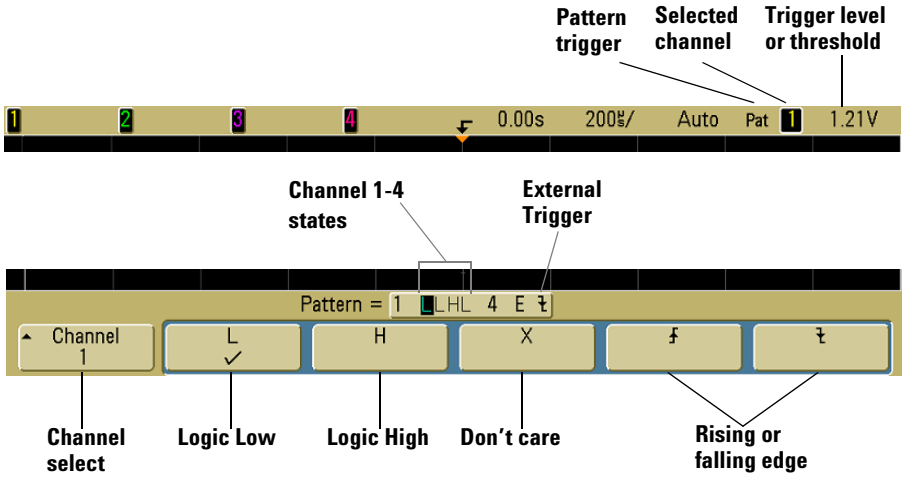
> qualifier time set softkey

- When the greater than (>) qualifier is selected, the Entry knob sets the oscilloscope to trigger on a pulse width greater than the time value displayed on the softkey.
- When the time range (><) qualifier is selected, the Entry knob sets the lower time range value.

To use Pattern triggering

The Pattern trigger identifies a trigger condition by looking for a specified pattern. This pattern is a logical AND combination of the channels. Each channel can have a value of high (H), low (L), don't care (X). A rising or falling edge can be specified for one channel included in the pattern.

1 Press the **Pattern** key in the Trigger section of the front panel to display the Pattern trigger menu.



2 For each oscilloscope channel you want to include in the desired pattern, press the **Channel** softkey to select the channel.

This is the channel source for the H, L, X, or edge condition. As you press the **Channel** softkey (or rotate the Entry knob on mixed-signal oscilloscopes), the channel you select is highlighted in the `Pattern =` line directly above the softkeys and in the upper-right corner of the display next to "`Pat`". External trigger may also be specified as a channel in the pattern when using the 2-channel and 4-channel oscilloscopes.

Adjust the trigger level for the selected channel by turning the Trigger Level knob. The value of the trigger level is displayed in the upper-right corner of the display.

3 For each channel you select, press one of the condition softkeys to set the condition for that channel in the pattern.

- **H** sets the pattern to high on the selected channel. A high is a voltage level that is greater than the channel's trigger level or threshold level.
- **L** sets the pattern to low on the selected channel. A low is a voltage level that is less than the channel's trigger level or threshold level.
- **X** sets the pattern to don't care on the selected channel. Any channel set to don't care is ignored and is not used as part of the pattern. However, if all channels in the pattern are set to don't care, the oscilloscope will not trigger.
- The rising edge () or falling edge () softkey sets the pattern to an edge on the selected channel. Only one rising or falling edge can be specified in the pattern. When an edge is specified, the oscilloscope will trigger at the edge specified if the pattern set for the other channels is true.

  If no edge is specified, the oscilloscope will trigger on the last edge that makes the pattern true.

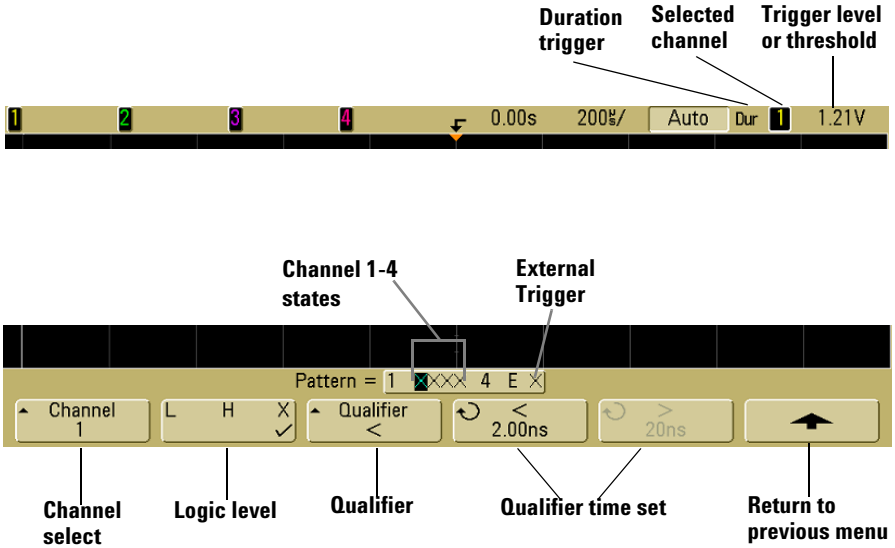
Specifying an Edge in a Pattern

You are allowed to specify only one rising or falling edge term in the pattern. If you define an edge term, then select a different channel in the pattern and define another edge term, the previous edge definition is changed to a don't care.

To use Duration triggering

Duration trigger lets you define a pattern, then trigger on a specified time duration of this logical AND combination of the channels.

1 Press the **More** key in the Trigger section of the front panel, rotate the Entry knob until **Duration** is displayed in the **Trigger** softkey, then press the **Settings** softkey to display the Duration trigger menu.



2 For each channel you want to include in the desired pattern, press the **Channel** softkey to select the channel.

This is the channel source for the H, L, or X condition. As you press the **Channel** softkey (or rotate the Entry knob on mixed-signal oscilloscopes), the channel you select is highlighted in the `Pattern =` line directly above the softkeys and in the upper-right corner of the display next to "`Dur`". External trigger may also be specified as a channel in the pattern when using the 2-channel and 4-channel oscilloscopes.

Adjust the trigger level for the selected channel by turning the Trigger Level knob. The value of the trigger level is displayed in the upper-right corner of the display.

3 For each channel you select, press the logic level softkey to set the condition for that channel in the pattern.

- **H** sets the pattern to high on the selected channel. A high is a voltage level that is greater than the channel's trigger level or threshold level.
- **L** sets the pattern to low on the selected channel. A low is a voltage level that is less than the channel's trigger level or threshold level.
- **X** sets the pattern to don't care on the selected channel. Any channel set to don't care is ignored and is not used as part of the pattern. If all channels in the pattern are set to don't care, the oscilloscope will not trigger.

4 Press the **Qualifier** softkey to set the time duration qualifier for the pattern.

The time qualifier can set the oscilloscope to trigger on a channel pattern whose time duration is:

- less than a time value (**<**)
- greater than a time value (**>**)
- greater than a time value, but with timeout (**Timeout**). A trigger will be forced at the timeout value, rather than occurring when the pattern is exited.
- within a range of time values (**><**)
- outside a range of time values (**<>**)

The time values for the selected qualifier are set using the qualifier time set softkeys (**<** and **>**) and the Entry knob.

5 Select a qualifier time set softkey (**<** or **>**), then rotate the Entry knob to set the duration qualifier time.

< qualifier time set softkey

- When the less than (**<**) qualifier is selected, the Entry knob sets the oscilloscope to trigger on a pattern duration less than the time value displayed on the softkey.

- When the within time range (><) qualifier is selected, the Entry knob sets the upper time range value.
- When the out of time range (<>) qualifier is selected, the Entry knob sets the lower time range value.

> qualifier time set softkey

- When the greater than (>) qualifier is selected, the Entry knob sets the oscilloscope to trigger on a pattern duration greater than the time value displayed on the softkey.
- When the within time range (><) qualifier is selected, the Entry knob sets the lower time range value.
- When the out of time range (<>) qualifier is selected, the Entry knob sets the upper time range value.
- When **Timeout** qualifier is selected, the Entry knob sets the timeout value.

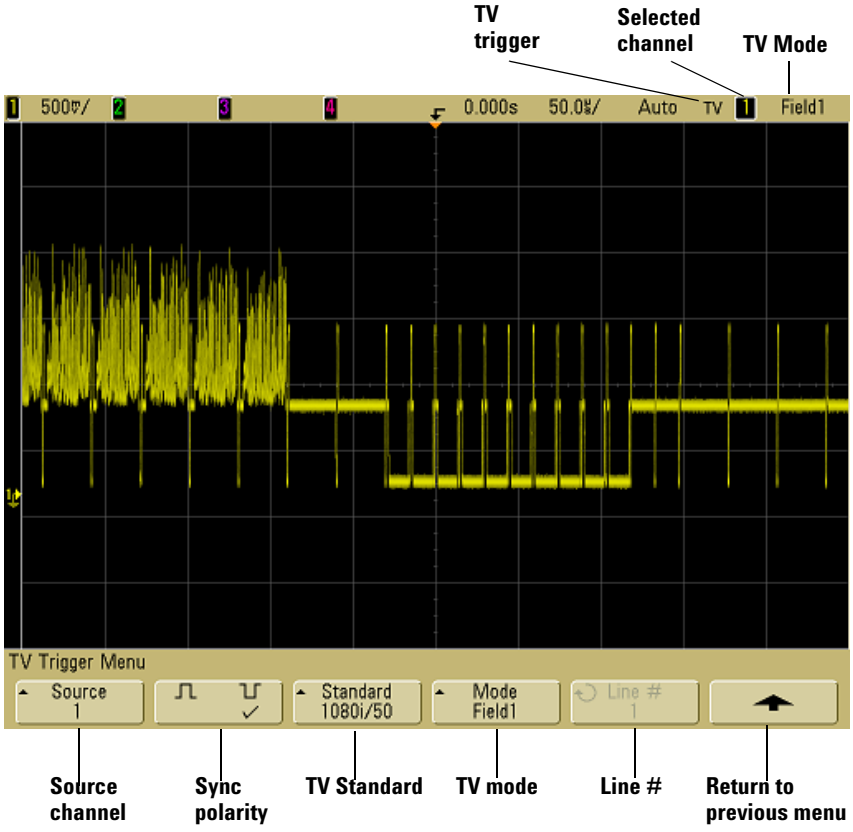
When the duration trigger occurs
The timer starts on the last edge that makes the pattern (logical AND) true. The trigger will occur on the first edge that makes the pattern false if the time qualifier criteria of the pattern has been met, except in Timeout mode. In Timeout, the trigger occurs when the timeout value is reached while the pattern is true.

To use TV triggering

TV triggering can be used to capture the complicated waveforms of most standard and high-definition analog video signals. The trigger circuitry detects the vertical and horizontal interval of the waveform and produces triggers based on the TV trigger settings you have selected.

The oscilloscope's MegaZoom III technology gives you bright, easily viewed displays of any part of the video waveform. Analysis of video waveforms is simplified by the oscilloscope's ability to trigger on any selected line of the video signal.

1 Press the **More** key in the Trigger section of the front panel. If **TV** is not selected, rotate the Entry knob until **TV** is displayed in the **Trigger** softkey, then press the **Settings** softkey to display TV trigger menu.



2 Press the **Source** softkey and select any oscilloscope channel as the TV trigger source.

The selected trigger source is displayed in the upper-right corner of the display. Turning the Trigger **Level** knob does not change the trigger level because the trigger level is automatically set to the sync pulse. Trigger coupling is automatically set to **TV** in the Trigger **Mode/Coupling** menu.

Provide Correct Matching

Many TV signals are produced from 75 Ω sources. To provide correct matching to these sources, a 75 Ω terminator (such as an Agilent 11094B) should be connected to the oscilloscope input.

3 Press the sync polarity softkey to set the TV trigger to either positive (⎍) or negative (⎏) sync polarity.

4 Press the **Standard** softkey to set the TV standard.

The oscilloscope supports triggering on the following television (TV) and video standards.

Standard	Type	Sync Pulse
NTSC	Interlaced	Bi-level
PAL	Interlaced	Bi-level
PAL-M	Interlaced	Bi-level
SECAM	Interlaced	Bi-level
Generic	Interlaced/Progressive	Bi-level/Tri-level
EDTV 480p/60	Progressive	Bi-level
HDTV 720p/60	Progressive	Tri-level
HDTV 1080p/24	Progressive	Tri-level
HDTV 1080p/25	Progressive	Tri-level
HDTV 1080i/50	Interlaced	Tri-level
HDTV 1080i/60	Interlaced	Tri-level

5 Press the **Mode** softkey to select the portion of the video signal that you would like to trigger on.

The TV trigger modes available are:

- **Field1** and **Field2** — Trigger on the rising edge of the first serration pulse of field 1 or field 2 (interlaced standards only).
- **All Fields** — Trigger on the rising edge of the first pulse in the vertical sync interval (not available in Generic mode).
- **All Lines** — Trigger on all horizontal sync pulses.
- **Line** — Trigger on the selected line # (EDTV and HDTV standards only).
- **Line: Field1** and **Line:Field2** — Trigger on the selected line # in field 1 or field 2 (interlaced standards only except 1080i).
- **Line: Alternate** — Alternately trigger on the selected line # in field 1 and field 2 (NTSC, PAL, PAL-M, and SECAM only).

- **Vertical** — Trigger on the rising edge of the first serration pulse or approximately 70 µs after the start of vertical sync, whichever occurs first (only available in Generic mode).
- **Count: Vertical** — Counts falling edges of sync pulses; triggers on the selected count number (only available in Generic mode).

6 If you select a line # mode, press the **Line #** softkey, then rotate the Entry knob to select the line number on which you want to trigger.

7 When using the Generic standard and you select a line # mode or **Count:Vertical**, press the **Count #** softkey and rotate the Entry knob to select the desired count number.

Listed below are the line (or count) numbers per field for each video standard.

Table 6 Line (or count for Generic) numbers per field for each non-HDTV/EDTV video standard

Video standard	Field 1	Field 2	Alt Field
NTSC	1 to 263	1 to 262	1 to 262
PAL	1 to 313	314 to 625	1 to 312
PAL-M	1 to 263	264 to 525	1 to 262
SECAM	1 to 313	314 to 625	1 to 312
Generic	1 to 1024	1 to 1024	1 to 1024 (vertical)

Line Number Represents Count

In **Generic** mode, the line number represents the number of a count instead of a real line number. This is reflected in the label in the softkey changing from **Line** to **Count**. In the **Mode** softkey selections, **Line:Field 1**, **Line:Field 2** and **Count:Vertical** are used to indicate where the counting starts. For an interlaced video signal, the counting starts from the rising edge of the first vertical serration pulse of Field 1 and/or Field 2. For a non-interlaced video signal, the counting starts after the rising edge of the vertical sync pulse.

Table 7 Line numbers for each EDTV/HDTV video standard

EDTV 480p/60	1 to 525
HDTV 720p/60	1 to 750
HDTV 1080p/24	1 to 1125
HDTV 1080p/25	1 to 1125
HDTV 1080i/50	1 to 1125
HDTV 1080i/60	1 to 1125

Example exercises

The following are exercises to familiarize you with TV triggering. These exercises use the NTSC video standard.

To trigger on a specific line of video

TV triggering requires greater than 1/2 division of sync amplitude with any oscilloscope channel as the trigger source. Turning the trigger **Level** knob in TV trigger does not change the trigger level because the trigger level is automatically set to the sync pulse tips.

One example of triggering on a specific line of video is looking at the vertical interval test signals (VITS), which are typically in line 18. Another example is closed captioning, which is typically in line 21.

1 Press the Trigger **More** key, then press the **TV** softkey.

2 Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard (NTSC).

3 Press the **Mode** softkey and select the TV field of the line you want to trigger on. You can choose **Line:Field1**, **Line:Field2**, or **Line:Alternate**.

4 Press the **Line #** softkey and select the number of the line you want to examine.

Alternate Triggering

If **Line:Alternate** is selected, the oscilloscope will alternately trigger on the selected line number in Field 1 and Field 2. This is a quick way to compare the Field 1 VITS and Field 2 VITS or to check for the correct insertion of the half line at the end of Field 1.

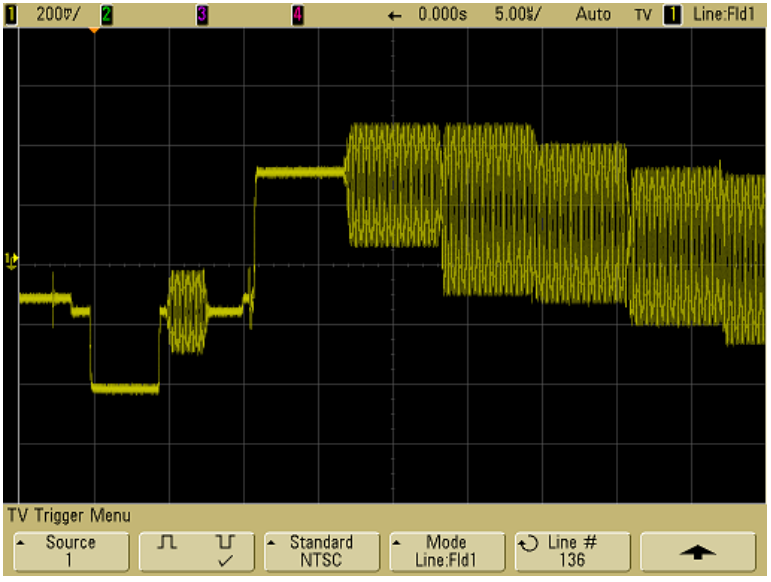


Figure 8 Example: Triggering on Line 136

To trigger on all sync pulses

To quickly find maximum video levels, you could trigger on all sync pulses. When **All Lines** is selected as the TV trigger mode, the oscilloscope will trigger on all horizontal sync pulses.

1 Press the Trigger **More** key, then press the **TV** softkey.

2 Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard.

3 Press the **Mode** softkey and select **All Lines**.

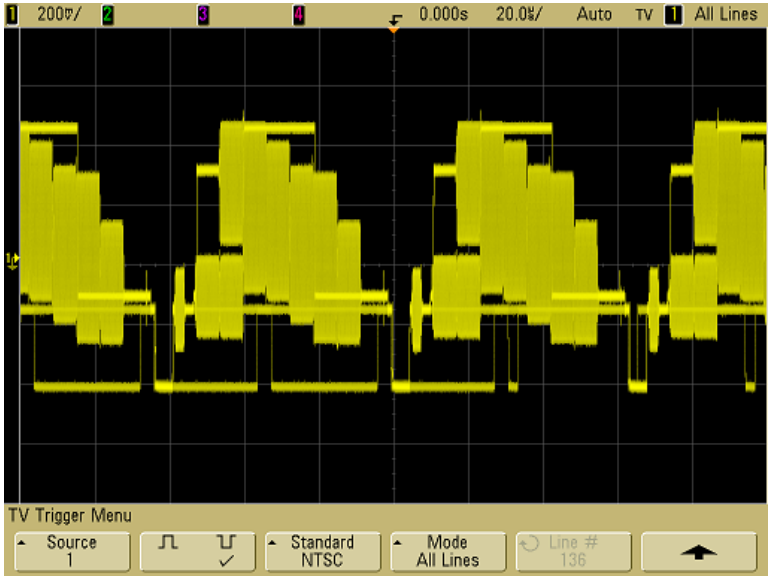


Figure 9 Triggering on All Lines

To trigger on a specific field of the video signal

To examine the components of a video signal, trigger on either Field 1 or Field 2 (available for interleaved standards). When a specific field is selected, the oscilloscope triggers on the rising edge of the first serration pulse in the vertical sync interval in the specified field (1 or 2).

1 Press the Trigger **More** key, then press the **TV** softkey.

2 Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard.

3 Press the **Mode** softkey and select **Field1** or **Field2**.

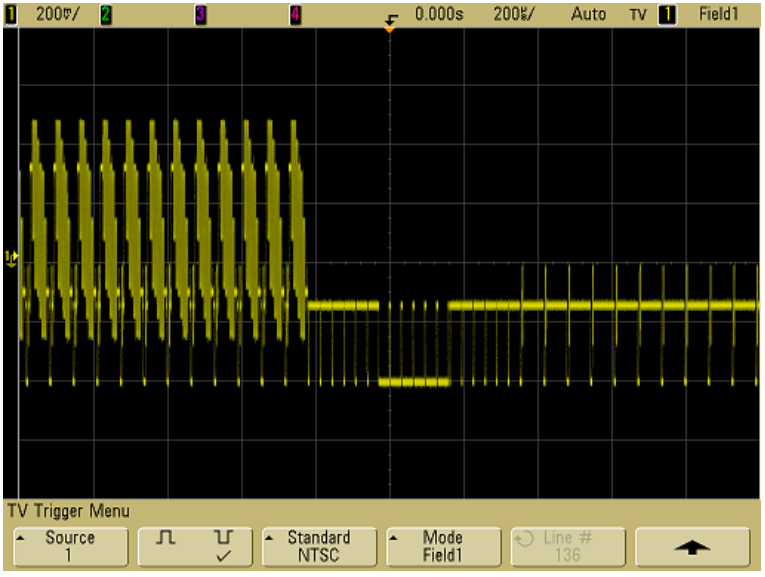


Figure 10 Triggering on Field 1

To trigger on all fields of the video signal

To quickly and easily view transitions between fields, or to find the amplitude differences between the fields, use the All Fields trigger mode.

1 Press the Trigger **More** key, then press the **TV** softkey.

2 Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard.

3 Press the **Mode** softkey and select **All Fields**.

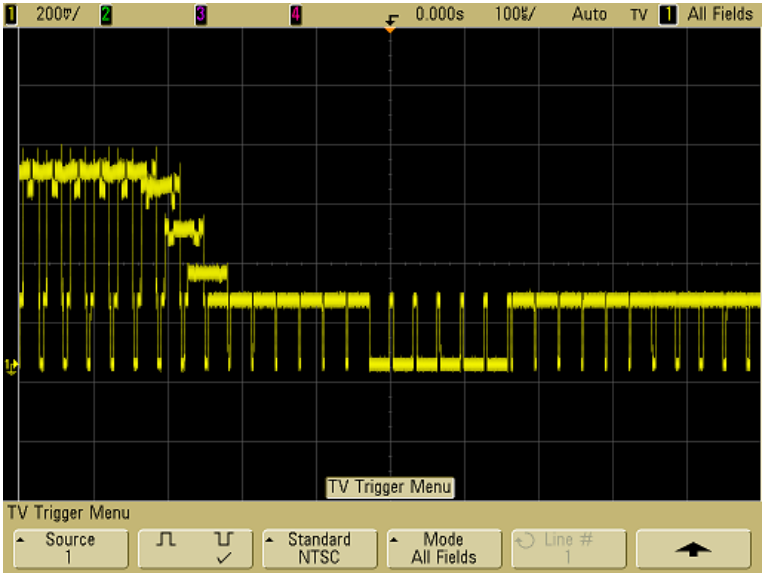


Figure 11 Triggering on All Fields

To trigger on odd or even fields

To check the envelope of your video signals, or to measure worst case distortion, trigger on the odd or even fields. When Field 1 is selected, the oscilloscope triggers on color fields 1 or 3. When Field 2 is selected, the oscilloscope triggers on color fields 2 or 4.

1. Press the Trigger **More** key, then press the **TV** softkey.
2. Press the **Settings** softkey, then press the **Standard** softkey to select the appropriate TV standard.
3. Press the **Mode** softkey and select **Field1** or **Field2**.

The trigger circuits look for the position of the start of Vertical Sync to determine the field. But this definition of field does not take into consideration the phase of the reference subcarrier. When Field 1 is selected, the trigger system will find any field where the vertical sync starts on Line 4. In the case of NTSC video, the oscilloscope will trigger on color field 1 alternating with color field 3 (see the following figure). This setup can be used to measure the envelope of the reference burst.

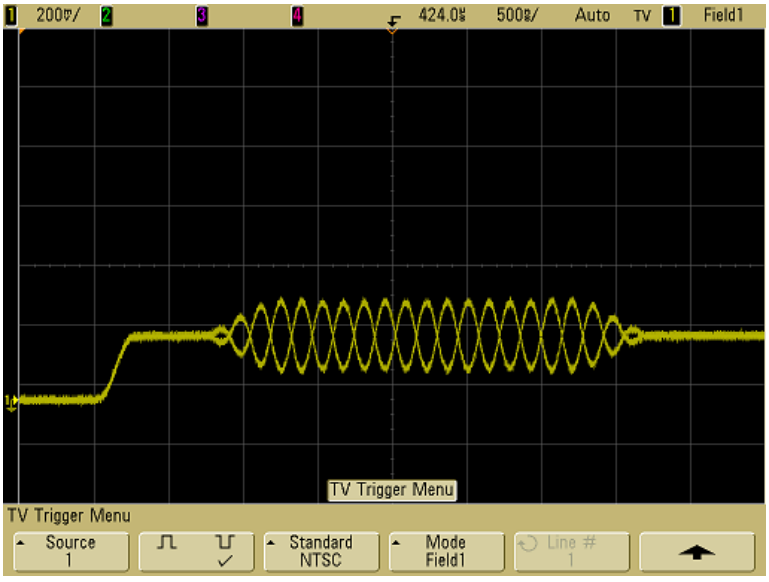


Figure 12 Triggering on Color Field 1 Alternating with Color Field 3

If a more detailed analysis is required, then only one color field should be selected to be the trigger. You can do this by using the **TV Holdoff** softkey in the trigger More Trigger Menu when the trigger type is set to **TV**. Press the **TV Holdoff** softkey and use the Entry knob to adjust the holdoff in half-field increments until the oscilloscope triggers on only one phase of the color burst.

A quick way to synchronize to the other phase is to briefly disconnect the signal and then reconnect it. Repeat until the correct phase is displayed.

When holdoff is adjusted using the **TV Holdoff** softkey and the Entry knob, the corresponding holdoff time will be displayed in the **Mode/Coupling** menu.

Table 8 Half-field holdoff time

Standard	Time
NTSC	8.35 ms
PAL	10 ms
PAL-M	10 ms
SECAM	10 ms
Generic	8.35 ms
EDTV 480p/60	8.35 ms
HDTV 720p/60	8.35 ms
HDTV 1080p/24	20.835 ms
HDTV 1080p/25	20 ms
HDTV 1080i/50	10 ms
HDTV 1080i/60	8.35 ms

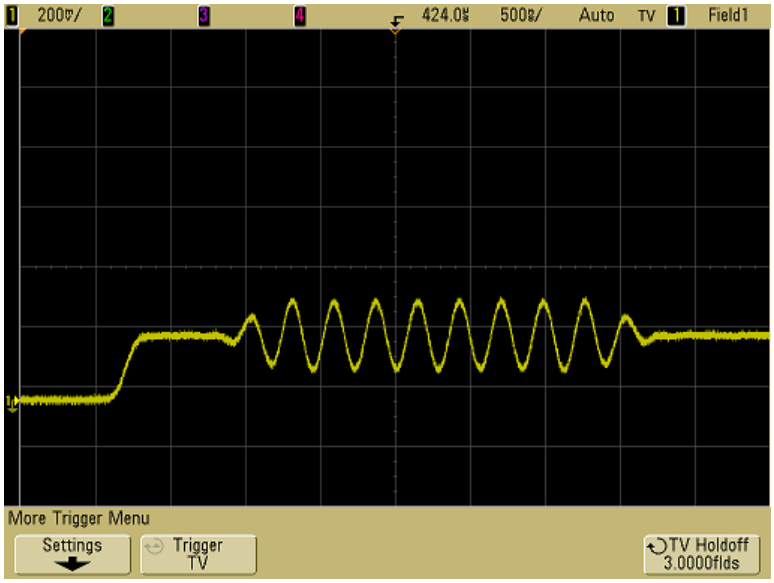


Figure 13 Using TV Holdoff to Synchronize to Color Field 1 or 3 (Field 1 mode)

The Trigger Out connector

Each time the oscilloscope triggers, a rising edge is output at the TRIG OUT connector on the rear panel of the oscilloscope. This rising edge is delayed 17 ns from the oscilloscope's trigger point. The output level is 0-5 V into an open circuit, or 0-2.5 V into 50 Ω

The Trigger Out connector also provides the User Cal signal. See "User Calibration" on page 81.

4 Making Measurements

Post Acquisition Processing

In addition to changing display parameters after the acquisition, you can perform all of the measurements and math functions after the acquisition. Measurements and math functions will be recalculated as you pan and zoom and turn channels on and off. As you zoom in and out on a signal using the horizontal sweep speed knob and vertical volts/division knob, you affect the resolution of the display. Because measurements and math functions are performed on displayed data, you affect the resolution of functions and measurements.

To use the XY horizontal mode

The XY horizontal mode converts the oscilloscope from a volts-versus-time display to a volts-versus-volts display using two input channels. Channel 1 is the X-axis input, channel 2 is the Y-axis input. You can use various transducers so the display could show strain versus displacement, flow versus pressure, volts versus current, or voltage versus frequency. This exercise shows a common use of the XY display mode by measuring the phase difference between two signals of the same frequency with the Lissajous method.

1. Connect a sine wave signal to channel 1, and a sine wave signal of the same frequency but out of phase to channel 2.
2. Press the **AutoScale** key, press the **Main/Delayed** key, then press the **XY** softkey.
3. Center the signal on the display with the channel 1 and 2 position (▲▼) knobs. Use the channel 1 and 2 volts/div knobs and the channel 1 and 2 **Vernier** softkeys to expand the signal for convenient viewing.

   The phase difference angle (θ) can be calculated using the following formula (assuming the amplitude is the same on both channels):

   $$\sin\theta = \frac{A}{B} \text{ or } \frac{C}{D}$$

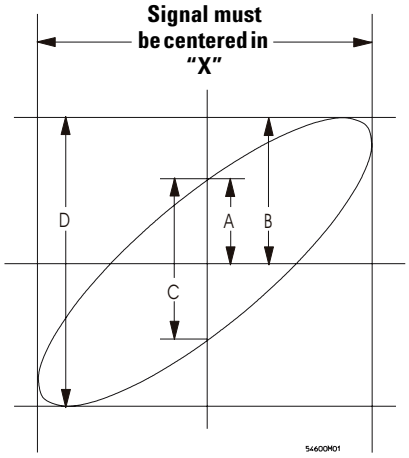


Figure 14 Example of centering a signal on the display

Figure 15 Signal centered on the display

4 Press the **Cursors** key.

5 Set the Y2 cursor to the top of the signal, and set Y1 to the bottom of the signal.

Note the ΔY value at the bottom of the display. In this example, we are using the Y cursors, but you could have used the X cursors instead.

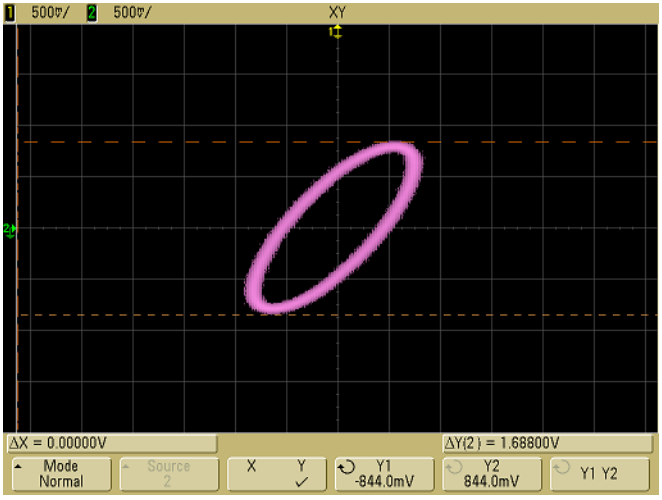


Figure 16 Cursors set on displayed signal

6 Move the Y1 and Y2 cursors to the intersection of the signal and the Y axis.

Again, note the ΔY value.

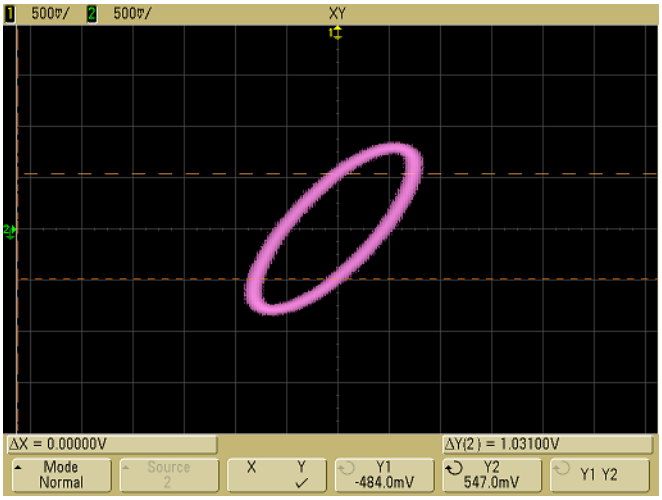


Figure 17 Cursors set to center of signal

7 Calculate the phase difference using the formula below.

$$\sin\theta = \frac{\text{second } \Delta Y}{\text{first } \Delta Y} = \frac{1.031}{1.688}\ ;\ \theta = 37.65 \text{ degrees of phase shift}$$

Z-Axis Input in XY Display Mode (Blanking)

When you select the XY display mode, the time base is turned off. Channel 1 is the X-axis input, channel 2 is the Y-axis input, and channel 4 (or the External trigger on 2-channel models) is the Z-axis input. If you only want to see portions of the Y versus X display, use the Z-axis input. Z-axis turns the trace on and off (analog oscilloscopes called this Z-axis blanking because it turned the beam on and off). When Z is low (<1.4 V), Y versus X is displayed; when Z is high (>1.4 V), the trace is turned off.

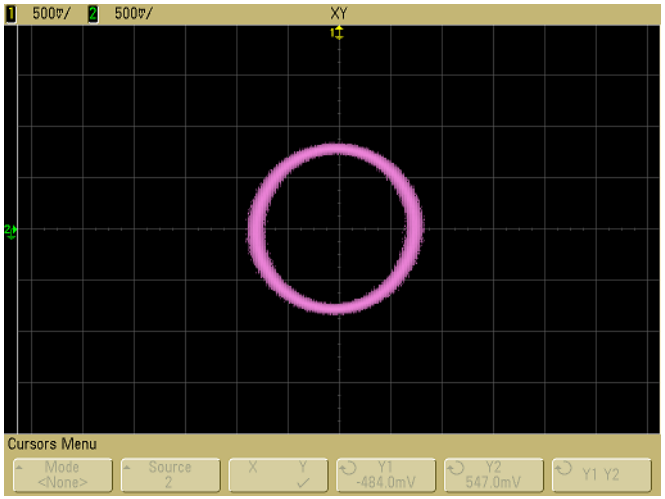


Figure 18 Signals are 90 out of phase

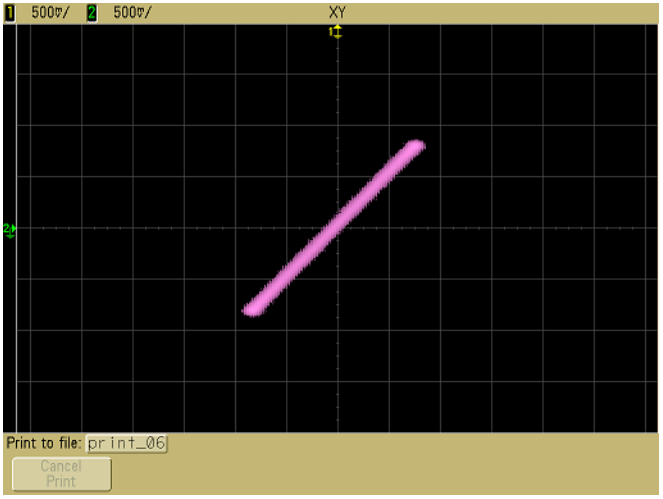


Figure 19 Signals are in phase

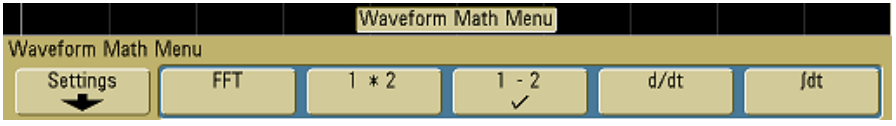
Math Functions

The Math menu lets you display math functions on oscilloscope channels. You can:

- subtract (–) or multiply (*) the signals acquired on oscilloscope channels 1 and 2, then display the result.
- integrate, differentiate, perform an FFT, or the square root function on the signal acquired on any channel or on math functions 1 * 2, 1 – 2, or 1 + 2, then display the result.

To access math functions:

1 Press the **Math** key on the front panel to display the Math menu. After selecting a math function, press the **Settings** softkey to display settings for the selected math function if you want to change the Y scaling.



Math Operating Hints

If the oscilloscope channel or math function is clipped (not fully displayed on screen,) the resulting displayed math function will also be clipped.

Once the function is displayed, the oscilloscope channel(s) may be turned off for better viewing.

The vertical scaling and offset of each math function can be adjusted for ease of viewing and measurement considerations.

Each function can be measured in the Cursors and Quick Meas menus.

Math scale and offset

Any math function may be manually scaled by pressing the **Settings** softkey, and then adjusting the Scale or Offset value.

Math Scale and Offset are Set Automatically

Any time the currently displayed math function definition is changed, the function is automatically scaled for optimum vertical scale and offset. If you manually set scale and offset for a function, select a new function, then select the original function, the original function will be automatically rescaled.

1 In the Waveform Math Menu for the selected math function, press the **Scale** or **Offset** sofkeys to set your own scale factors (units/division) or offset (units) for the selected math function.

Units for each input channel can be set to Volts or Amps using the channel **Probe Units** softkey. Scale and offset units are:

Math function	Units
FFT	dB* (decibels)
1*2	V^2, A^2, or W (Volt-Amp)
1-2	V or A
d/dt	V/s or A/s (V/second or A/second)
$\int$ dt	Vs or As (V-seconds or A-seconds)
$\sqrt{}$(square root)	$V^{1/2}$, $A^{1/2}$, or $W^{1/2}$ (Volt-Amp)

* When the FFT source is channel 1, 2, 3 or 4, FFT units will be displayed in dBV when channel units is set to Volts and channel impedance is set to 1 MΩ FFT units will be displayed in dBm when channel units is set to Volts and channel impedance is set to 50Ω FFT units will be displayed as dB for all other FFT sources or when a source channel's units has been set to Amps.

A scale unit of **U** (undefined) will be displayed for math function 1-2 and for d/dt, and $\int$ dt when 1-2 or 1+2 is the selected source if channel 1 and channel 2 are set to dissimilar units in the channel **Probe Units** softkey.

2 Press the **Scale** or **Offset** softkey, then turn the Entry knob to rescale or change the offset value for your math function.

Multiply

When you select **1 * 2**, channel 1 and channel 2 voltage values are multiplied point by point, and the result is displayed. 1 * 2 is useful for seeing power relationships when one of the channels is proportional to the current.

1 Press the **Math** key, press the **1 * 2** softkey, then press the **Settings** softkey if you want to change the scaling or offset for the multiply function.

- **Scale** — lets you set your own vertical scale factors for multiply expressed as V^2/div (Volts-squared/division), A^2/div (Amps-squared/division), or W/div (Watts/division or Volt-Amps/division). Units are set in the channel **Probe** menu. Press the **Scale** softkey, then turn the Entry knob to rescale 1 * 2.
- **Offset** — lets you set your own offset for the multiply math function. The offset value is in V^2 (Volts-squared), A^2 (Amps-squared), or W (Watts) and is represented by the center horizontal grid line of the display. Press the **Offset** softkey, then turn the Entry knob to change the offset for 1 * 2.

The figure below shows an example of multiply.

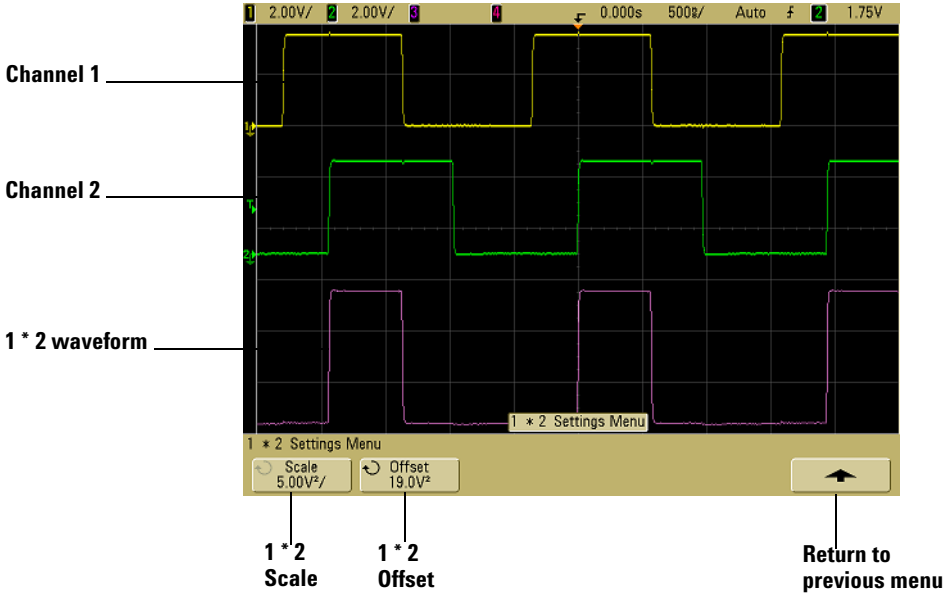


Figure 20 Multiply

Subtract

When you select **1 – 2**, channel 2 voltage values are subtracted from channel 1 voltage values point by point, and the result is displayed.

You can use 1 – 2 to make a differential measurement or to compare two waveforms. You may need to use a true differential probe if your waveforms have DC offsets larger than the dynamic range of the oscilloscope's input channel.

To perform the addition of channel 1 and channel 2, select **Invert** in the Channel 2 menu and perform the 1 – 2 math function.

1 Press the **Math** key, press the **1 – 2** softkey, then press the **Settings** softkey if you want to change the scaling or offset for the subtract function.

- **Scale** — lets you set your own vertical scale factors for subtract, expressed as V/div (Volts/division) or A/div (Amps/division). Units are set in the channel **Probe** menu. Press the **Scale** softkey, then turn the Entry knob to rescale 1 – 2.
- **Offset** — lets you set your own offset for the 1 – 2 math function. The offset value is in Volts or Amps and is represented by the center horizontal grid line of the display. Press the **Offset** softkey, then turn the Entry knob to change the offset for 1 – 2.

A scale unit of **U** (undefined) will be displayed for scale and offset if channel 1 and channel 2 are set to dissimilar units in the channel **Probe Units** softkey.

The figure below shows an example of subtract.

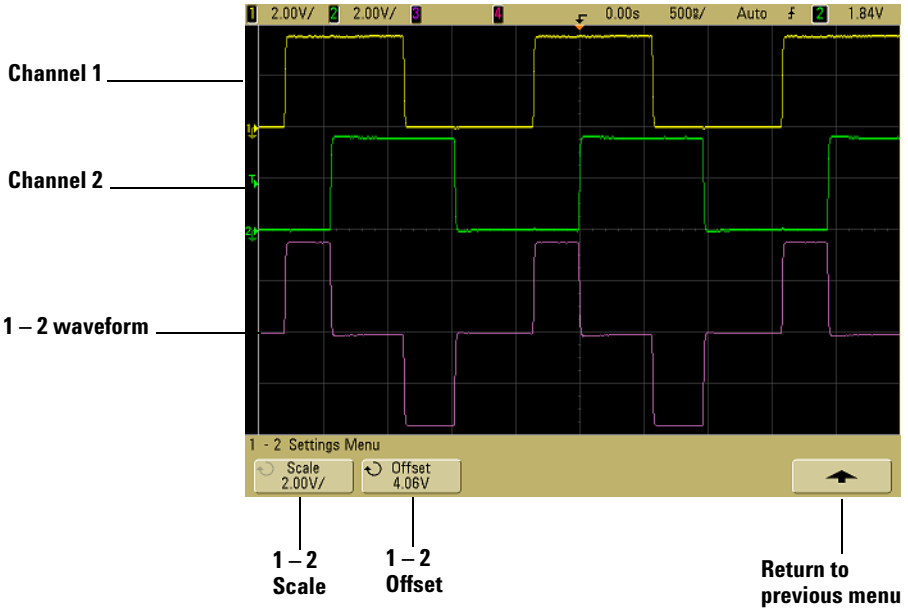


Figure 21 Subtract

Differentiate

d/dt (differentiate) calculates the discrete time derivative of the selected source. You can use differentiate to measure the instantaneous slope of a waveform. For example, the slew rate of an operational amplifier may be measured using the differentiate function.

Because differentiation is very sensitive to noise, it is helpful to set acquisition mode to **Averaging** in the **Acquire** menu.

d/dt plots the derivative of the selected source using the "average slope estimate at 4 points" formula. The equation is:

$$d_i = \frac{y_{i+4} + 2y_{i+2} - 2y_{i-2} - y_{i-4}}{8\Delta t}$$

Where

d = differential waveform
y = channel 1, 2, or function 1 + 2, 1 – 2, and 1 * 2 data points
i = data point index
Δt = point-to-point time difference

In Delayed sweep horizontal mode, the d/ dt function does not display in the delayed portion of the display.

1 Press the **Math** key, press the **d/dt** softkey, then press the **Settings** softkey if you want to change the source, scaling, or offset for the differentiate function.

- **Source** — selects the source for d/dt. The source can be any oscilloscope channel, or math functions 1 + 2, 1 – 2, and 1 * 2.
- **Scale** — lets you set your own vertical scale factors for d/dt expressed in units/second/division, where units can be V (Volts), A (Amps), or W (Watts). Units are set in the channel **Probe** menu. Press the **Scale** softkey, then turn the Entry knob to rescale d/dt.

- **Offset** —lets you set your own offset for the dV/dt math function. The offset value is in units/second where units can be V (Volts), A (Amps), or W (Watts) and is represented by the center horizontal grid line of the display. Press the **Offset** softkey, then turn the Entry knob to change the offset for d/dt.

A scale unit of **U** (undefined) will be displayed for scale and offset when 1-2 or 1+2 is the selected source if channel 1 and channel 2 are set to dissimilar units in the channel **Probe Units** softkey.

The figure below shows an example of differentiate.

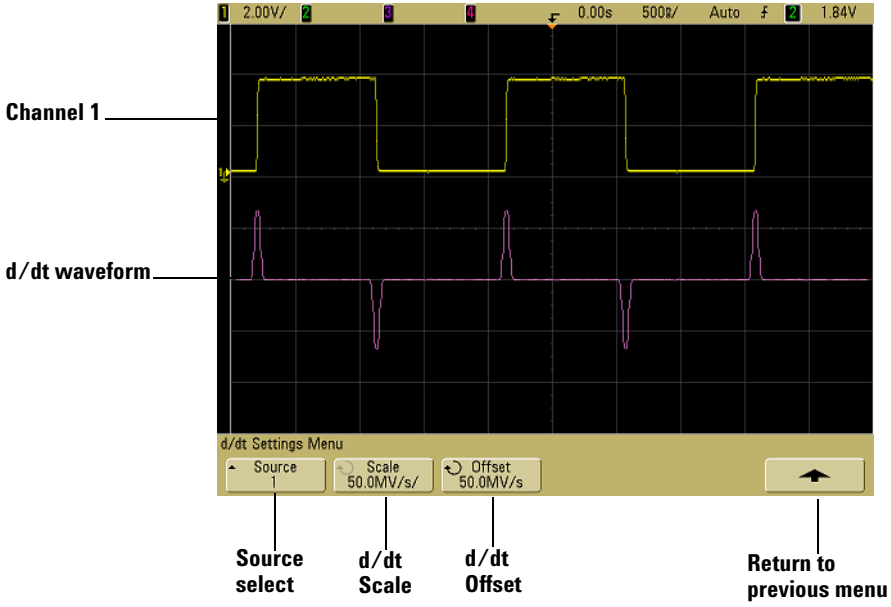


Figure 22 Differentiate

Integrate

∫ dt (integrate) calculates the integral of the selected source. You can use integrate to calculate the energy of a pulse in volt-seconds or measure the area under a waveform.

∫ dt plots the integral of the source using the "Trapezoidal Rule". The equation is:

$$I_n = c_o + \Delta t \sum_{i=0}^{n} y_i$$

Where

I = integrated waveform
Δt = point-to-point time difference
y = channel 1, 2, or function 1 + 2, 1 – 2, and 1 * 2 data points
c_o = arbitrary constant
i = data point index

In Delayed sweep horizontal mode, the ∫ dt function does not display in the delayed portion of the display.

1 Press the **Math** key, press the ∫ **dt** softkey, then press the **Settings** softkey if you want to change the source, scaling, or offset for the integrate function.

- **Source** — selects the source for ∫ dt. The source can be any oscilloscope channel, or math functions 1 + 2, 1 – 2, and 1 * 2.
- **Scale** — lets you set your own vertical scale factors for ∫ dt expressed in unit-seconds/division, where units can be V (Volts), A (Amps), or W (Watts). Units are set in the channel **Probe** menu. Press the **Scale** softkey, then turn the Entry knob to rescale ∫ dt.
- **Offset** — lets you set your own offset for the ∫ Vdt math function. The offset value is in unit-seconds where units can be V (Volts), A (Amps), or W (Watts) and is represented by the center horizontal grid line of the display. Press the **Offset** softkey, then turn the Entry knob to change the offset for ∫ dt. The integrate calculation is relative to the source signal's offset. The following examples illustrate the effects of signal offset.

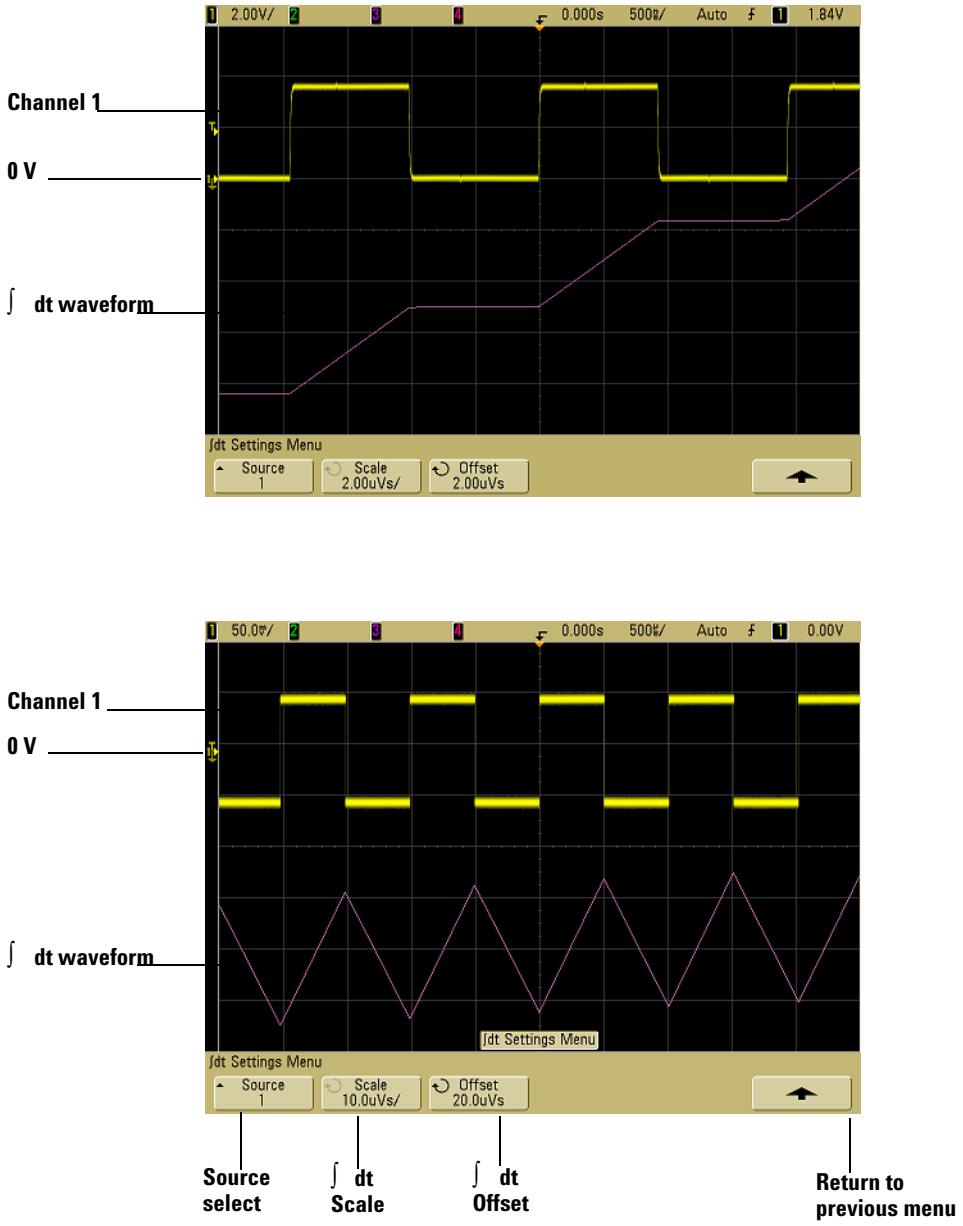


Figure 23 Integrate and Signal Offset

FFT Measurement

FFT is used to compute the fast Fourier transform using oscilloscope input channels or math functions 1 + 2, 1 – 2, or 1 * 2. FFT takes the digitized time record of the specified source and transforms it to the frequency domain. When the FFT function is selected, the FFT spectrum is plotted on the oscilloscope display as magnitude in dBV versus frequency. The readout for the horizontal axis changes from time to frequency (Hertz) and the vertical readout changes from volts to dB.

Use the FFT function to find crosstalk problems, to find distortion problems in analog waveforms caused by amplifier non-linearity, or for adjusting analog filters.

FFT Units

0 dBV is the amplitude of a 1 Vrms sinusoid. When the FFT source is channel 1 or channel 2 (or channel 3 or 4 on 4-channel models), FFT units will be displayed in dBV when channel units is set to Volts and channel impedance is set to 1 MΩ

FFT units will be displayed in dBm when channel units is set to Volts and channel impedance is set to 50Ω

FFT units will be displayed as dB for all other FFT sources or when a source channel's units has been set to Amps.

DC Value

The FFT computation produces a DC value that is incorrect. It does not take the offset at center screen into account. The DC value is not corrected in order to accurately represent frequency components near DC.

Aliasing

When using FFTs, it is important to be aware of frequency aliasing. This requires that the operator have some knowledge as to what the frequency domain should contain, and also consider the sampling rate, frequency span, and oscilloscope

vertical bandwidth when making FFT measurements. The FFT sample rate is displayed directly above the softkeys when the FFT menu is displayed.

Aliasing happens when there are frequency components in the signal higher than half the sample rate. Since the FFT spectrum is limited by this frequency, any higher components are displayed at a lower (aliased) frequency.

The following figure illustrates aliasing. This is the spectrum of a 990 Hz square wave, which has many harmonics. The FFT sample rate is set to 100 kSa/s, and the oscilloscope displays the spectrum. The displayed waveform shows the components of the input signal above the Nyquist frequency to be mirrored (aliased) on the display and reflected off the right edge.

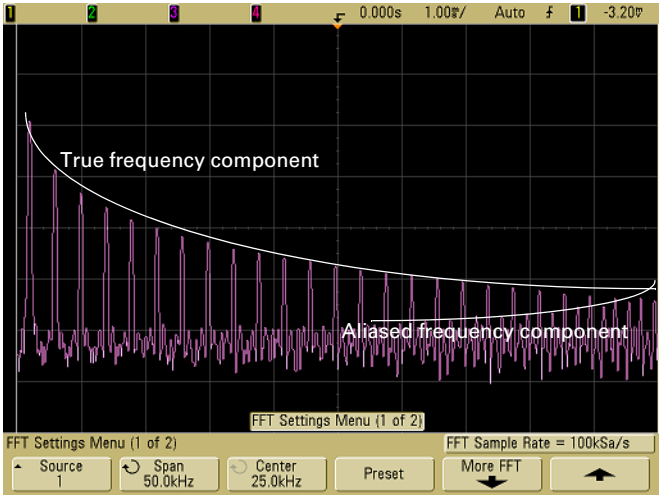


Figure 24 Aliasing

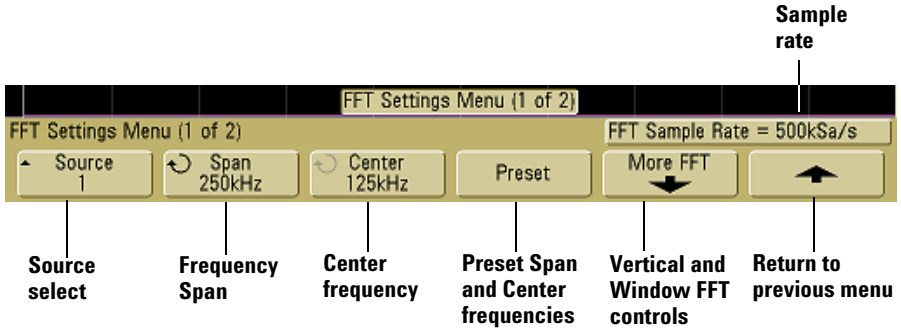
Since the frequency span goes from ≈ 0 to the Nyquist frequency, the best way to prevent aliasing is to make sure that the frequency span is greater than the frequencies of significant energy present in the input signal.

Spectral Leakage

The FFT operation assumes that the time record repeats. Unless there is an integral number of cycles of the sampled waveform in the record, a discontinuity is created at the end of the record. This is referred to as leakage. In order to minimize spectral leakage, windows that approach zero smoothly at the beginning and end of the signal are employed as filters to the FFT. The FFT menu provides three windows: Hanning, flattop, and rectangular. For more information on leakage, see Agilent Application Note 243, "The Fundamentals of Signal Analysis" at http://cp.literature.agilent.com/litweb/pdf/5952-8898E.pdf.

FFT Operation

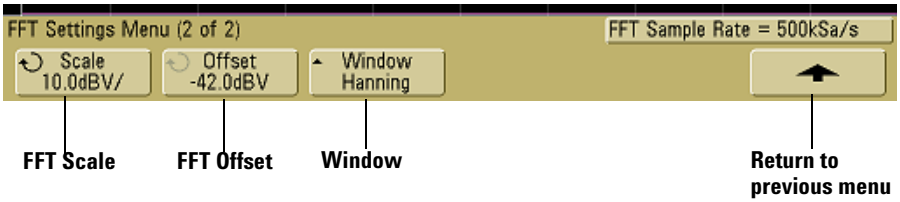
1 Press the **Math** key, press the **FFT** softkey, then press the **Settings** softkey to display the FFT menu.



- **Source** — selects the source for the FFT. The source can be any oscilloscope channel, or math functions 1 + 2, 1 – 2, and 1 * 2.
- **Span** — sets the overall width of the FFT spectrum that you see on the display (left to right). Divide span by 10 to calculate the number of Hertz per division. It is possible to set Span above the maximum available frequency, in which case the displayed spectrum will not take up the whole screen. Press the **Span** softkey, then turn the Entry knob to set the desired frequency span of the display.

- **Center** — sets the FFT spectrum frequency represented at the center vertical grid line of the display. It is possible to set the Center to values below half the span or above the maximum available frequency, in which case the displayed spectrum will not take up the whole screen. Press the **Center** softkey, then turn the Entry knob to set the desired center frequency of the display.
- **Preset** — sets the frequency Span and Center to values that will cause the entire available spectrum to be displayed. The maximum available frequency is half the effective FFT sample rate, which is a function of the time per division setting. The current FFT sample rate is displayed above the softkeys.

2 Press the More FFT softkey to display additional FFT settings.



- **Scale** — lets you set your own vertical scale factors for FFT expressed in dB/div (decibels/division). Press the **Scale** softkey, then turn the Entry knob to rescale your math function.
- **Offset** — lets you set your own offset for the FFT. The offset value is in dB and is represented by the center horizontal grid line of the display. Press the **Offset** softkey, then turn the Entry knob to change the offset of your math function

.

Scale and offset considerations

If you do not manually change the FFT scale or offset settings, when you turn the horizontal sweep speed knob, the span and center frequency settings will automatically change to allow optimum viewing of the full spectrum. If you do manually set scale or offset, turning the sweep speed knob will not change the span or center frequency settings, allowing you see better detail around a specific frequency. Pressing the FFT **Preset** softkey will automatically rescale the waveform and span and center will again automatically track the horizontal sweep speed setting.

- **Window** — selects a window to apply to your FFT input signal:
- **Hanning** — window for making accurate frequency measurements or for resolving two frequencies that are close together.
- **Flat Top** — window for making accurate amplitude measurements of frequency peaks.
- **Rectangular** — good frequency resolution and amplitude accuracy, but use only where there will be no leakage effects. Use on self-windowing waveforms such as pseudo-random noise, impulses, sine bursts, and decaying sinusoids.

3 To make cursors measurements, press the **Cursors** key and set the **Source** softkey to **Math**.

Use the X1 and X2 cursors to measure frequency values and difference between two frequency values (ΔX). Use the Y1 and Y2 cursors to measure amplitude in dB and difference in amplitude (ΔY).

4 To make other measurements, press the **Quick Meas** key and set the **Source** softkey to **Math**.

You can make peak-to-peak, maximum, minimum, and average dB measurements on the FFT waveform. You can also find the frequency value at the first occurrence of the waveform maximum by using the X at Max measurement.

The following FFT spectrum was obtained by connecting the front panel Probe Comp signal (~1.2 kHz) to channel 1. Set sweep speed to 5 ms/div, vertical sensitivity to 500 mV/div, Units/div to 10 dBV, Offset to -34.0 dBV, Center frequency to 5.00 kHz, frequency Span to 10.0 kHz, and window to Hanning.

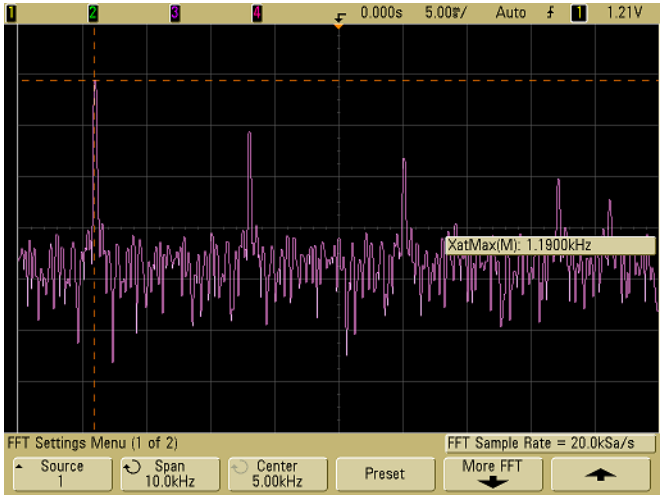


Figure 25 FFT measurements

FFT Measurement Hints

The number of points acquired for the FFT record is 1000 and when frequency span is at maximum, all points are displayed. Once the FFT spectrum is displayed, the frequency span and center frequency controls are used much like the controls of a spectrum analyzer to examine the frequency of interest in greater detail. Place the desired part of the waveform at the center of the screen and decrease frequency span to increase the display resolution. As frequency span is decreased, the number of points shown is reduced, and the display is magnified.

While the FFT spectrum is displayed, use the **Math** and **Cursors** keys to switch between measurement functions and frequency domain controls in FFT menu.

Decreasing the effective sampling rate by selecting a slower sweep speed will increase the low frequency resolution of the FFT display and also increase the chance that an alias will be displayed. The resolution of the FFT is the effective sample rate divided by the number of points in the FFT. The actual resolution of the display will not be this fine as the shape of the window will be the actual limiting factor in the FFTs ability to resolve two closely space frequencies. A good way to test the ability of the FFT to resolve two closely spaced frequencies is to examine the sidebands of an amplitude modulated sine wave.

For the best vertical accuracy on peak measurements:

- Make sure the probe attenuation is set correctly. The probe attenuation is set from the Channel menu if the operand is a channel.
- Set the source sensitivity so that the input signal is near full screen, but not clipped.
- Use the Flat Top window.
- Set the FFT sensitivity to a sensitive range, such as 2 dB/division.

For best frequency accuracy on peaks:

- Use the Hanning window.
- Use Cursors to place an X cursor on the frequency of interest.
- Adjust frequency span for better cursor placement.
- Return to the Cursors menu to fine tune the X cursor.

For more information on the use of FFTs please refer to Agilent Application Note 243, "The Fundamentals of Signal Analysis" at http://cp.literature.agilent.com/litweb/pdf/5952-8898E.pdf. Additional information can be obtained from Chapter 4 of the book "Spectrum and Network Measurements" by Robert A. Witte.

Square Root

√(square root) calculates the square root of the selected source.

In Delayed sweep horizontal mode, the √(square root) function does not display in the delayed portion of the display.

1 Press the **Math** key, press the √(square root) softkey, then press the **Source**, **Scale**, or **Offset** sofkeys if you want to change the source, scaling, or offset for the square root function.

- **Source** — selects the source for √(square root). The source can be any analog channel, or math functions 1 + 2, 1 – 2, and 1 * 2.
- **Scale** — lets you set your own vertical scale factors for √(square root) expressed as $V^{1/2}$/div (Volts-square-root/division), $A^{1/2}$/div (Amps-square-root/division), or $W^{1/2}$/div (Watts-square-root/division or Volt-Amps-square-root/division). Units are set in the channel **Probe** menu. Press the **Scale** softkey, then turn the Entry knob to rescale √(square root).
- **Offset** — lets you set your own offset for the multiply math function. The offset value is in $V^{1/2}$ (Volts-square-root), $A^{1/2}$ (Amps-square-root), or $W^{1/2}$ (Watts-square-root) and is represented by the center horizontal grid line of the display. Press the **Offset** softkey, then turn the Entry knob to change the offset for √(square root).

A scale unit of **U** (undefined) will be displayed for scale and offset when 1-2 or 1+2 is the selected source if channel 1 and channel 2 are set to dissimilar units in the channel **Probe Units** softkey.

The figure below shows an example of √(square root).

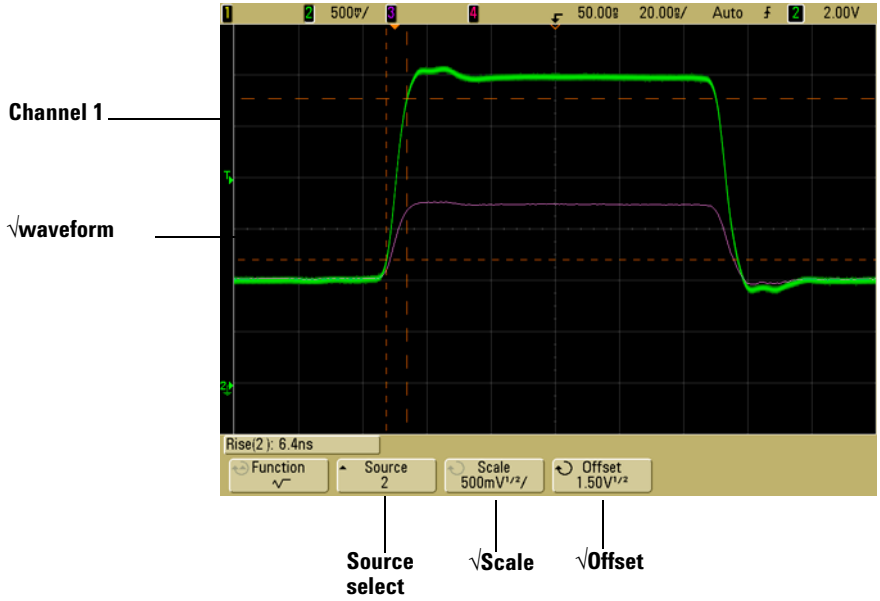


Figure 26 Square Root

Cursor Measurements

You can measure waveform data using cursors. Cursors are horizontal and vertical markers that indicate X-axis values (usually time) and Y-axis values (usually voltage) on a selected waveform source. The position of the cursors can be moved by turning the Entry knob. When you press the **Cursors** key, it will illuminate and the cursors will turn on. To turn cursors off, press this key again until it is not illuminated, or press the **Quick Meas** key.

Cursors are not always limited to the visible display. If you set a cursor, then pan and zoom the waveform until the cursor is off screen, its value will not be changed, and if you pan the waveform back again it will have the cursor in the original place.

To make cursor measurements

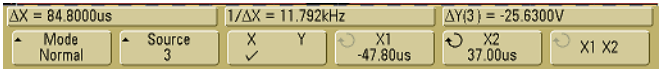
A brief description of how to make cursor measurements is given on page 72.

The following steps guide you through using the front-panel **Cursors** key. You can use the cursors to make custom voltage or time measurements on the signal.

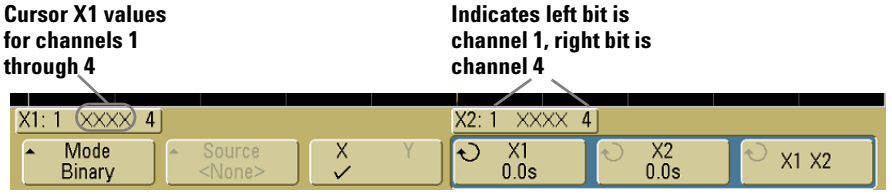
1. Connect a signal to the oscilloscope and obtain a stable display.
2. Press the **Cursors** key, then press the **Mode** softkey.

   X and Y cursor information is displayed on the softkeys. ΔX, 1/ΔX, ΔY, and binary and hexadecimal values are displayed on the line above the softkeys. The three cursors modes are:

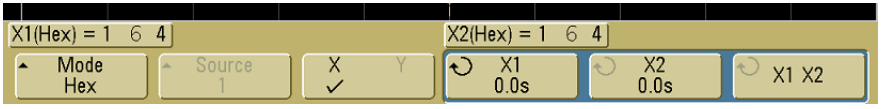
   - **Normal** – ΔX, 1/ΔX, and ΔY values are displayed. ΔX is the difference between the X1 and X2 cursors and ΔY is the difference between the Y1 and Y2 cursors.



- **Binary** — Binary logic levels are displayed directly above the softkeys for the current X1 and X2 cursor positions for all displayed channels.



- **Hex** — Hexadecimal logic levels are displayed directly above the softkeys for the current X1 and X2 cursor positions for all displayed channels.



In hexadecimal and binary mode, a level can display as 1 (higher than trigger level), 0 (lower than trigger level), indeterminate state (↕), or X (don't care). In binary mode, X is displayed if the channel is turned off. In hex mode, the channel is interpreted as a 0 if turned off.

3 Press the **Source** softkey to select the oscilloscope channel or math source on which the Y cursors will indicate measurements.

The source in **Normal** cursor mode can be any oscilloscope channel or math source. If you select binary or hexadecimal mode, the **Source** softkey is disabled since you are displaying binary or hexadecimal levels for all channels.

4 Select the X and Y softkeys to make a measurement.

- **X Y** — Press this softkey to select either X cursors or Y cursors for adjustment. The cursor currently assigned to the Entry knob displays brighter than the other cursors.

  X cursors are vertical dashed lines that adjust horizontally and normally indicate time relative to the trigger point. When used with the FFT math function as a source, the X cursors indicate frequency.

  Y cursors are horizontal dashed lines that adjust vertically and normally indicate Volts or Amps, dependent on the channel **Probe Units** setting. When math functions are used as a source, the measurement units correspond to that math function.

- **X1** and **X2** — The X1 cursor (short-dashed vertical line) and X2 cursor (long-dashed-vertical line) adjust horizontally and indicate time relative to the trigger point for all sources except math FFT (frequency is indicated). In XY horizontal mode, the X cursors display channel 1 values (Volts or Amps). The cursor values for the selected waveform source are displayed within the X1 and X2 softkeys.

  The difference between X1 and X2 (ΔX) and 1/ΔX are displayed on the dedicated line above the softkeys or in the display area when some menus are selected.

  Turn the Entry knob to adjust the X1 or X2 cursor when its softkey is selected.

- **Y1** and **Y2** — The Y1 cursor (short-dashed horizontal line) and Y2 cursor (long-dashed horizontal line) adjust vertically and indicate values relative to the waveform's ground point, except math FFT where the values are relative to 0 dB. In XY horizontal mode, the Y cursors display channel 2 values (Volts or Amps). The cursor values for the selected waveform source are displayed within the Y1 and Y2 softkeys.

  The difference between Y1 and Y2 (ΔY) is displayed on the dedicated line above the softkeys or in the display area when some menus are selected.

Turn the Entry knob to adjust the Y1 or Y2 cursor when its softkey is selected.

- **X1 X2** — Press this softkey to adjust the X1 and X2 cursors together by turning the Entry knob. The ΔX value will remain the same since the cursors adjust together.

  You can adjust the X cursors together to check for pulse width variations in a pulse train.

- **Y1 Y2** — Press this softkey to adjust the Y1 and Y2 cursors together by turning the Entry knob. The ΔY value will remain the same since the cursors adjust together.

Cursor Examples

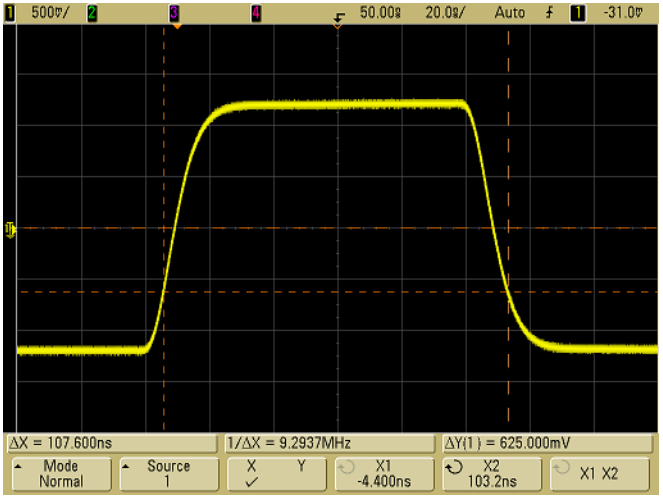


Figure 27 Cursors measure pulse widths other than middle threshold points

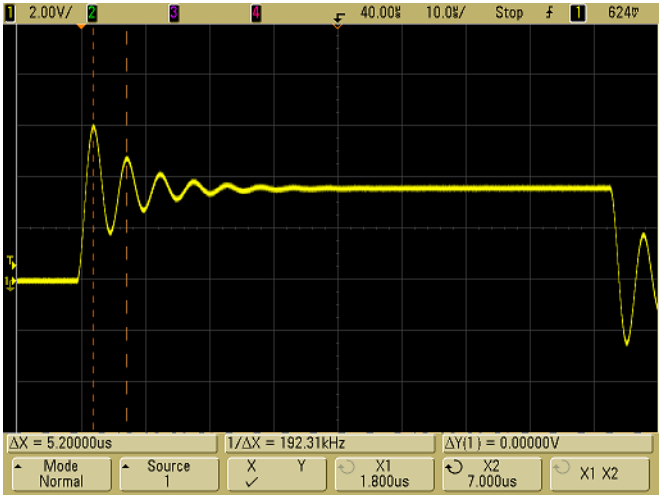


Figure 28 Cursors measure frequency of pulse ringing

Expand the display with delayed sweep, then characterize the event of interest with the cursors.

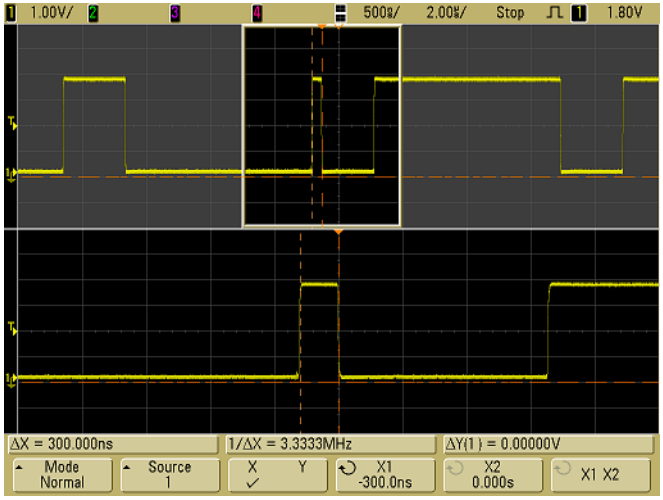


Figure 29 Cursors track delayed sweep

Put the **X1** cursor on one side of a pulse and the **X2** cursor on the other side of the pulse.

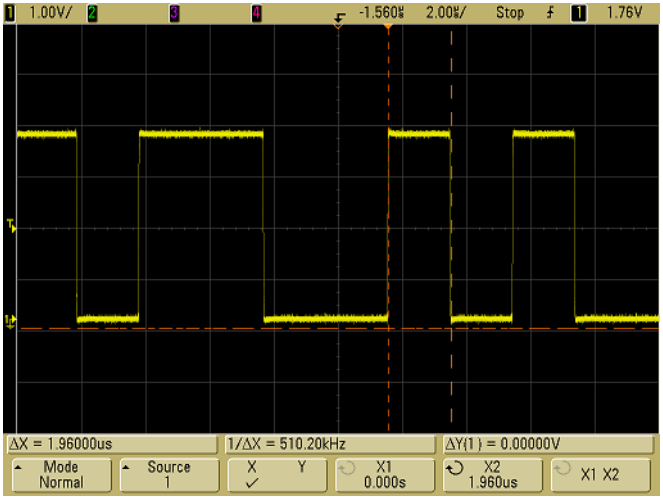


Figure 30 Measuring pulse width with cursors

Press the **X1 X2** softkey and move the cursors together to check for pulse width variations in a pulse train.

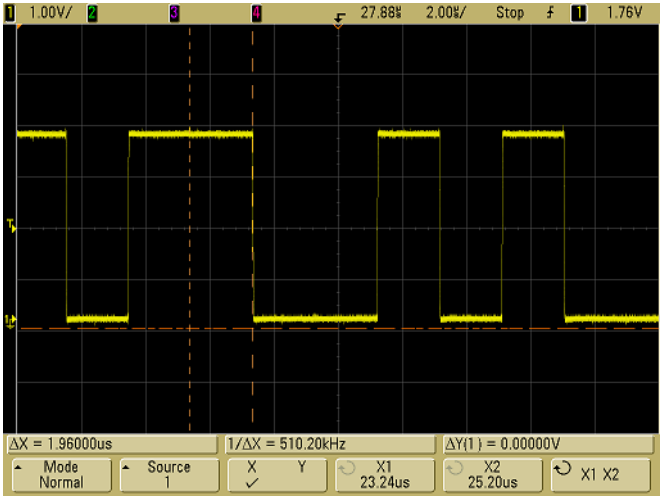


Figure 31 Moving the cursors together to check pulse width variations

Automatic Measurements

The following automatic measurements can be made in the **Quick Meas** menu.

Time Measurements

- Counter
- Duty Cycle
- Frequency
- Period
- Rise Time
- Fall Time
- + Width
- – Width
- X at Max
- X at Min

Phase and Delay

- Phase
- Delay

Voltage Measurements

- Average
- Amplitude
- Base
- Maximum
- Minimum
- Peak-to-Peak
- RMS
- Std Deviation
- Top

Preshoot and Overshoot

- Preshoot
- Overshoot

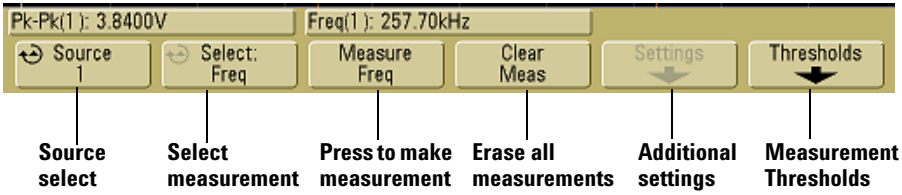
To make an automatic measurement

A brief description of how to make automatic measurements is given on page 73.

Quick Meas makes automatic measurements on any channel source or any running math function. The results of the last four measurements selected are displayed on the dedicated line above the softkeys, or in the display area when some menus are selected. Quick Meas also makes measurements on stopped waveforms when you are panning and zooming.

Cursors are turned on to show the portion of the waveform being measured for the most recently selected measurement (right-most on the measurement line).

1 Press the **Quick Meas** key to display the automatic measurement menu.



2 Press the **Source** softkey to select the channel or running math function on which the quick measurements will be made.

Only channels or math functions that are displayed are available for measurements. If you choose an invalid source channel for a measurement, the measurement will default to the nearest in the list that makes the source valid.

If a portion of the waveform required for a measurement is not displayed or does not display enough resolution to make the measurement, the result will display “No Edges”, “Clipped”, “Low Signal”, “< value”, or “> value”, or a similar message to indicate that the measurement may not be reliable.

3 Press the **Clear Meas** softkey to stop making measurements and to erase the measurement results from the display line above the softkeys.

When **Quick Meas** is pressed again, the default measurements will be Frequency and Peak-Peak.

4 Press the **Select** softkey then rotate the Entry knob to select a measurement to be made.

5 The **Settings** softkey will be available to make additional measurement settings on some measurements.

6 Press the **Measure** softkey to make the measurement.

7 To turn off **Quick Meas**, press the **Quick Meas** key again until it is not illuminated.

To set measurement thresholds

Setting measurement thresholds defines the vertical levels where measurements will be taken on an analog channel. or math waveform

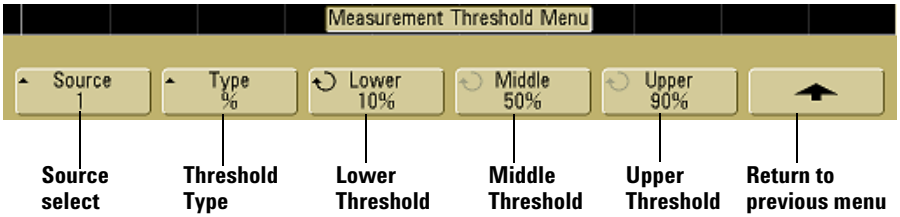
Changing default thresholds may change measurement results

The default lower, middle, and upper threshold values are 10%, 50%, and 90% of the value between Top and Base. Changing these threshold definitions from the default values may change the returned measurement results for Average, Delay, Duty Cycle, Fall Time, Frequency, Overshoot, Period, Phase, Preshoot, Rise Time, RMS, +Width, and -Width.

1 Press the **Thresholds** softkey in the **Quick Meas** menu to set oscilloscope channel measurement thresholds.

2 Press the **Source** softkey to select the oscilloscope channel source for which you want to change measurement

thresholds. Each oscilloscope channel can be assigned unique threshold values.



3 Press the **Type** softkey to set the measurement threshold to **%** (percentage of Top and Base value) or to **Absolute** (absolute value.)

- Percentage thresholds can be set from 5% to 95%.
- The units for absolute threshold for each channel is set in the channel probe menu.
- When the **Source** is set to **Math**, the threshold **Type** can only be set to **Percent**.

Absolute threshold hints

- Absolute thresholds are dependent on channel scaling, probe attenuation, and probe units. Always set these values first before setting absolute thresholds.
- The minimum and maximum threshold values are limited to on-screen values.
- If any of the absolute threshold values are above or below the minimum or maximum waveform values, the measurement may not be valid.

4 Press the **Lower** softkey, then turn the Entry knob to set the lower measurement threshold value.

Increasing the lower value beyond the set middle value will automatically increase the middle value to be more than the lower value. The default lower threshold is 10% or 800 mV.

If threshold **Type** is set to **%**, the lower threshold value can be set from 5% to 93%.

5 Press the **Middle** softkey, then turn the Entry knob to set the middle measurement threshold value.

The middle value is bounded by the values set for lower and upper thresholds. The default middle threshold is 50% or 1.20 V.

- If threshold **Type** is set to **%**, the middle threshold value can be set from 6% to 94%.

6 Press the **Upper** softkey, then turn the Entry knob to set the upper measurement threshold value.

Decreasing the upper value below the set middle value will automatically decrease the middle value to be less than the upper value. The default upper threshold is 90% or 1.50 V.

- If threshold **Type** is set to **%**, the upper threshold value can be set from 7% to 95%.

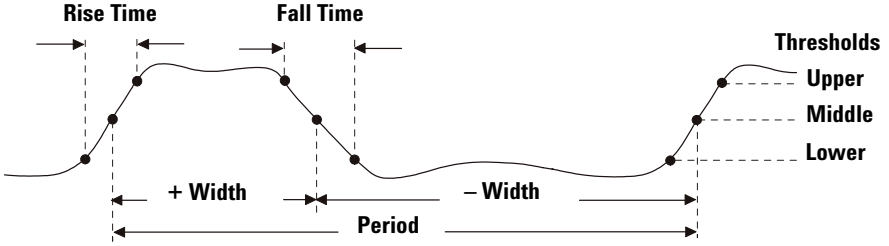
Time Measurements

FFT measurements

When you make an **X at Max** or **X at Min** measurement on a math FFT function, the resultant units will be in Hertz. No other time related automatic measurement can be made on the FFT math function. Use the cursors to make other measurements on FFT.

The default lower, middle, and upper measurement thresholds are 10%, 50%, and 90% between Top and Base values. See "To set measurement thresholds" on page 160 for other percentage threshold and absolute value threshold settings.

The following figure shows time measurement points.



Counter

The 5000A Series oscilloscopes have an integrated 5-digit hardware frequency counter which counts the number of cycles that occur within a period of time (known as the *gate time*) to measure the frequency of a signal.

The gate time for the Counter measurement is automatically adjusted to be 100 ms or twice the current time window, whichever is longer, up to 1 second.

The Counter can measure frequencies up to the bandwidth of the oscilloscope. The minimum frequency supported is 1/(2 X gate time).

The hardware counter uses the trigger comparator output. Therefore, the counted channel's trigger level must be set correctly. The Y cursor shows the threshold level used in the measurement.

Any channel except Math can be selected as the source.

Only one Counter measurement can be displayed at a time.

Duty Cycle

The duty cycle of a repetitive pulse train is the ratio of the positive pulse width to the period, expressed as a percentage. The X cursors show the time period being measured. The Y cursor shows the middle threshold point.

$$\text{Duty cycle} = \frac{+\text{Width}}{\text{Period}} \text{ X } 100$$

Frequency

Frequency is defined as 1/Period. Period is defined as the time between the middle threshold crossings of two consecutive, like-polarity edges. A middle threshold crossing must also travel through the lower and upper threshold levels which eliminates runt pulses. The X cursors show what portion of the waveform is being measured. The Y cursor shows the middle threshold point.

To isolate an event for frequency measurement The following figure shows how to use delayed sweep to isolate an event for a frequency measurement. If the measurement is not possible in the delayed time base mode, then the main time base is used. If the waveform is clipped, it may not be possible to make the measurement.

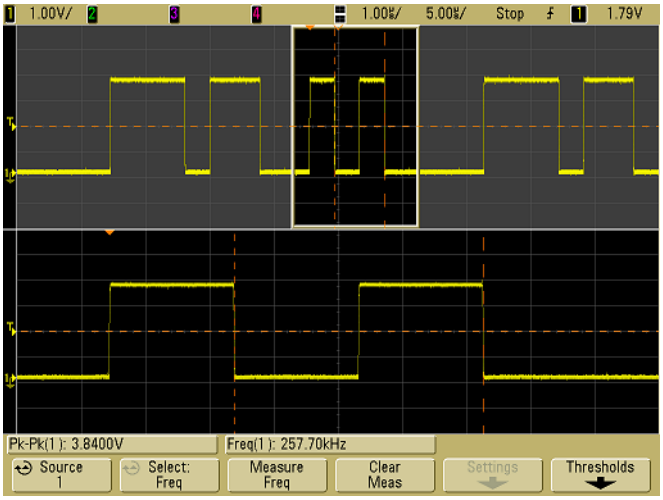


Figure 32 Isolating event for Frequency measurement

Period

Period is the time period of the complete waveform cycle. The time is measured between the middle threshold points of two consecutive, like-polarity edges. A middle threshold crossing must also travel through the lower and upper threshold levels which eliminates runt pulses. The X cursors show what portion of the waveform is being measured. The Y cursor shows the middle threshold point.

Fall Time

The fall time of a signal is the time difference between the crossing of the upper threshold and the crossing of the lower threshold for a negative-going edge. The X cursor shows the edge being measured. For maximum measurement accuracy, set

the sweep speed as fast as possible while leaving the complete falling edge of the waveform on the display. The Y cursors show the lower and upper threshold points.

Rise Time

The rise time of a signal is the time difference between the crossing of the lower threshold and the crossing of the upper threshold for a positive-going edge. The X cursor shows the edge being measured. For maximum measurement accuracy, set the sweep speed as fast as possible while leaving the complete rising edge of the waveform on the display. The Y cursors show the lower and upper threshold points.

+ Width

+ Width is the time from the middle threshold of the rising edge to the middle threshold of the next falling edge. The X cursors show the pulse being measured. The Y cursor shows the middle threshold point.

– Width

– Width is the time from the middle threshold of the falling edge to the middle threshold of the next rising edge. The X cursors show the pulse being measured. The Y cursor shows the middle threshold point.

X at Max

X at Max is the X axis value (usually time) at the first displayed occurrence of the waveform Maximum, starting from the left-side of the display. For periodic signals, the position of the maximum may vary throughout the waveform. The X cursor shows where the current X at Max value is being measured.

To measure the peak of an FFT:

1. Select **FFT** as the math function in the **Math** menu.
2. Choose **Math** as the source in the **Quick Meas** menu.
3. Choose **Maximum** and **X at Max** measurements.

Maximum units are in dB and **X at Max** units are in Hertz for FFT.

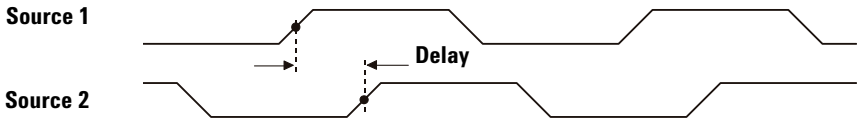
X at Min

X at Min is the X axis value (usually time) at the first displayed occurrence of the waveform Minimum, starting from the left-side of the display. For periodic signals, the position of the minimum may vary throughout the waveform. The X cursor shows where the current X at Min value is being measured.

Delay and Phase Measurements

Delay

Delay measures the time difference from the selected edge on source 1 and the selected edge on source 2 closest to the trigger reference point at the middle threshold points on the waveforms. Negative delay values indicate that the selected edge of source 1 occurred after the selected edge of source 2.



1 Press the **Quick Meas→Select** and select **Delay**. Press the **Settings** softkey to select the source channels and slope for the delay measurement.

   The default Delay settings measure from the rising edge of channel 1 to the rising edge of channel 2.

2 Press the **Measure Delay** softkey to make the measurement.

The following example shows a delay measurement between the rising edge of channel 1 and the rising edge of channel 2.

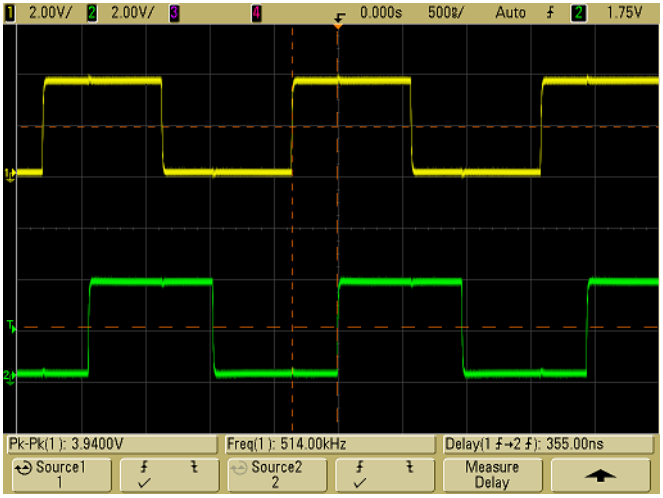
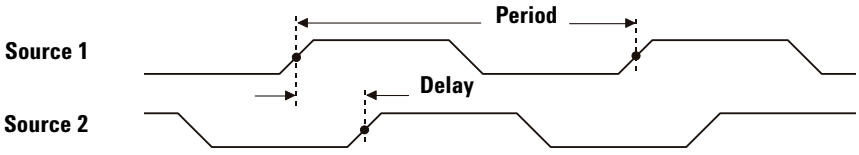


Figure 33 Delay Measurement

Phase

Phase is the calculated phase shift from source 1 to source 2, expressed in degrees. Negative phase shift values indicate that the rising edge of source 1 occurred after the rising edge of source 2.

$$\text{Phase} = \frac{\text{Delay}}{\text{Source 1 Period}} \text{ X } 360$$



1 Press the **Settings** softkey to select the source 1 and source 2 channels for the phase measurement.

The default Phase settings measure from channel 1 to channel 2.

The example below shows a phase measurement between the channel 1 and the math d/dt function on channel 1.

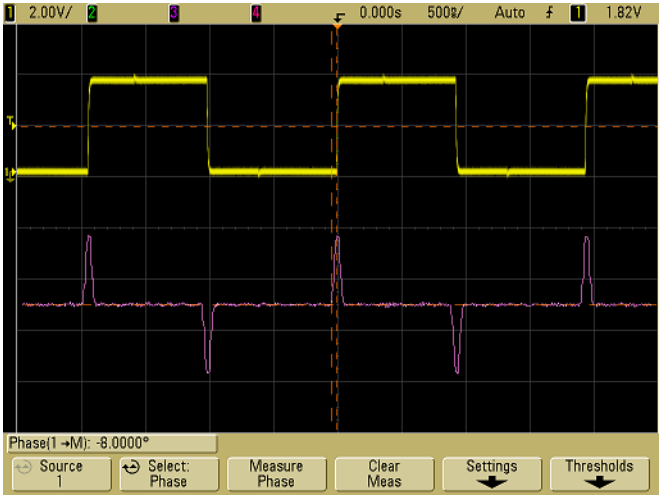


Figure 34 Phase Measurement

Voltage Measurements

Measurement units for each input channel can be set to Volts or Amps using the channel **Probe Units** softkey. A scale unit of **U** (undefined) will be displayed for math function 1-2 and for d/dt, and $\int$ dt when 1-2 or 1+2 is the selected source if channel 1 and channel 2 are set to dissimilar units in the channel **Probe Units** softkey.

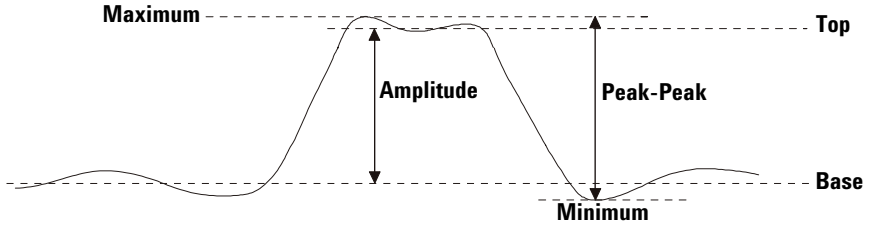
Math Measurements and Units

Only Peak-Peak, Maximum, Minimum, Average, X at Min, and X at Max automatic measurements may be made on a math FFT function. See "Making time measurements automatically" for the FFT X at Max and X at Min measurement. Use the cursors to make other measurements on FFT. All voltage measurements may be made on other math functions. The resultant units are:

FFT:	dB* (decibels)
1 * 2:	V^2, A^2 or W (Volt-Amp)
1 – 2:	V (Volts) or A (Amps)
d/dt:	V/s or A/s (V/second A/second)
$\int$ dt:	Vs or As (V-seconds or A-seconds)

* When the FFT source is channel 1, 2, 3, or 4, FFT units will be displayed in dBV when channel units is set to Volts and channel impedance is set to 1 MΩ FFT units will be displayed in dBm when channel units is set to Volts and channel impedance is set to 50Ω FFT units will be displayed as dB for all other FFT sources or when a source channel's units has been set to Amps.

The following figure shows the voltage measurement points.



Amplitude

The Amplitude of a waveform is the difference between its Top and Base values. The Y cursors show the values being measured.

Average

Average is the sum of the waveform samples divided by the number of samples over one or more full periods. If less than one period is displayed, Average is calculated on the full width of the display. The X cursors show what portion of the displayed waveform is being measured.

$$\text{Average} = \frac{\sum x_i}{n}$$

where x_i = value at ith point being measured
n = number of points in measurement interval

Base

The Base of a waveform is the mode (most common value) of the lower part of the waveform, or if the mode is not well defined, the base is the same as Minimum. The Y cursor shows the value being measured.

Maximum

Maximum is the highest value in the waveform display. The Y cursor shows the value being measured.

Minimum

Minimum is the lowest value in the waveform display. The Y cursor shows the value being measured.

Peak-Peak

The peak-to-peak value is the difference between Maximum and Minimum values. The Y cursors show the values being measured.

RMS

RMS (DC) is the root-mean-square value of the waveform over one or more full periods. If less than one period is displayed, RMS (DC) average is calculated on the full width of the display. The X cursors show what interval of the waveform is being measured.

$$\text{RMS (dc)} = \sqrt{\frac{\sum_{i=1}^{n} x_i^2}{n}}$$

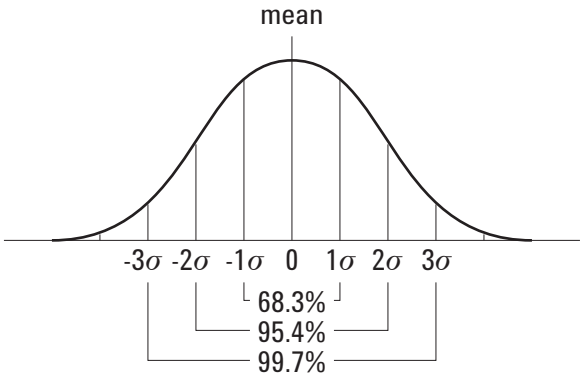
where x_i = value at ith point being measured
n = number of points in measurement interval

Std Deviation

The Std Deviation measurement shows the standard deviation of the displayed voltage values. It is an RMS measurement across the full screen with the DC component removed. It is useful, for example, for measuring power supply noise.

The standard deviation of a measurement is the amount that a measurement varies from the mean value. The Mean value of a measurement is the statistical average of the measurement.

The following figure graphically shows the mean and standard deviation. Standard deviation is represented by the Greek letter sigma: σ. For a Gaussian distribution, two sigma (± 1σ) from the mean, is where 68.3 percent of the measurement results reside. Six sigma (± 3σ) from is where 99.7 percent of the measurement results reside.



The mean is calculated as follows:

$$\bar{x} = \frac{\sum_{i=1}^{N} x_i}{N}$$

where:

$\bar{x}$ = the mean.
N = the number of measurements taken.
x_i = the i^{th} measurement result.

The standard deviation is calculated as follows:

$$\sigma = \sqrt{\frac{\sum_{i=1}^{N} (x_i - \bar{x})^2}{N-1}}$$

where:

σ= the standard deviation.
N = the number of measurements taken.
x_i = the i^{th} measurement result.
$\bar{x}$ = the mean.

Top

The Top of a waveform is the mode (most common value) of the upper part of the waveform, or if the mode is not well defined, the top is the same as Maximum. The Y cursor shows the value being measured.

To isolate a pulse for Top measurement The following figure shows how to use delayed sweep to isolate a pulse for a **Top** measurement.

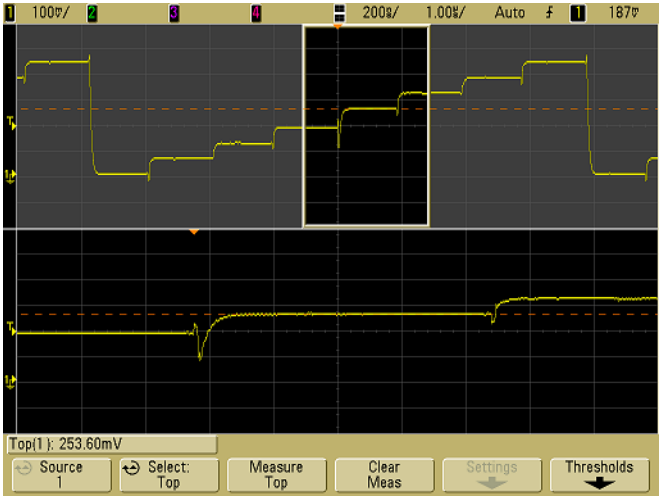


Figure 35 Isolating area for Top measurement

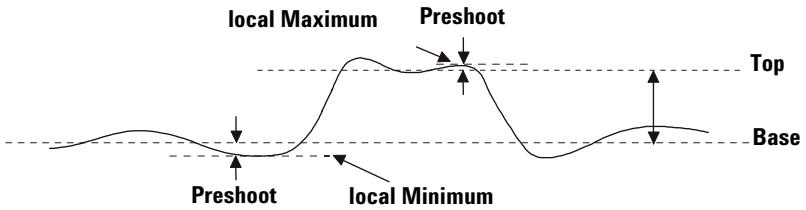
Overshoot and Preshoot Measurements

Preshoot

Preshoot is distortion that precedes a major edge transition expressed as a percentage of Amplitude. The X cursors show which edge is being measured (edge closest to the trigger reference point).

$$\text{Rising edge preshoot} = \frac{\text{Base - D local Minimum}}{\text{Amplitude}} \text{ X } 100$$

$$\text{Falling edge preshoot} = \frac{\text{local Maximum - D Top}}{\text{Amplitude}} \text{ X } 100$$

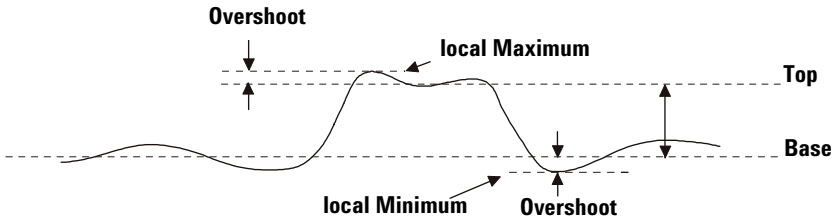


Overshoot

Overshoot is distortion that follows a major edge transition expressed as a percentage of Amplitude. The X cursors show which edge is being measured (edge closest to the trigger reference point).

$$\text{Rising edge overshoot} = \frac{\text{local Maximum - D Top}}{\text{Amplitude}} \text{ X } 100$$

$$\text{Falling edge overshoot} = \frac{\text{Base - D local Minimum}}{\text{Amplitude}} \text{ X } 100$$



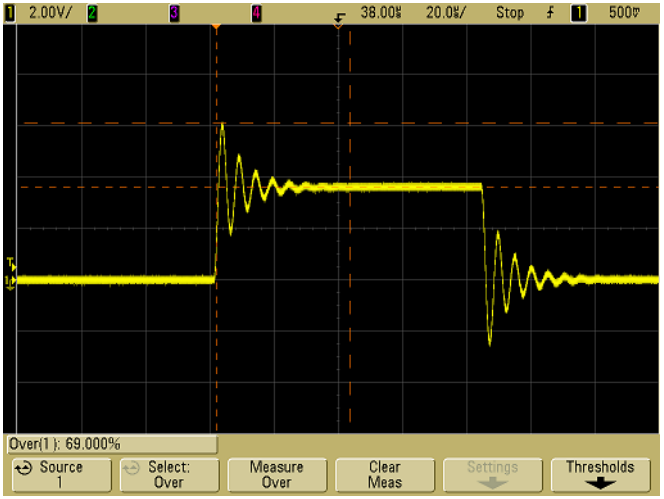


Figure 36 Automatic Overshoot measurement

5 Displaying Data

Pan and Zoom

The ability to pan (move horizontally) and zoom (expand or compress horizontally) an acquired waveform is important because of the additional insight it can reveal about the captured waveform. This additional insight is often gained from seeing the waveform at different levels of abstraction. You may want to view both the big picture and the specific little picture details.

The ability to examine waveform detail after the waveform has been acquired is a benefit generally associated with digital oscilloscopes. Often this is simply the ability to freeze the display for the purpose of measuring with cursors or printing the screen. Some digital oscilloscopes go one step further by including the ability to further examine the signal details after acquiring them by panning and zooming through the waveform.

There is no limit imposed on the zoom ratio between the sweep speed used to acquire the data and the sweep speed used to view the data. There is, however, a useful limit. This useful limit is somewhat a function of the signal you are analyzing.

In normal display mode, with vectors (connect-the-dots) off, you can zoom in to the point where there are no samples on the screen. Obviously, this is far beyond the useful limit. Likewise, with vectors on, you can see the linear interpolation between the points, but again, this is of very limited value.

Zoom

The screen will still contain a relatively good display if you zoom in horizontally by a factor of 1000 and zoom in vertically by a factor of 10 to display the information from where it was acquired. Remember that you can only make automatic measurements on displayed data.

To pan and zoom a waveform

1. Press the **Run/Stop** key to stop acquisitions. The **Run/Stop** key is illuminated red when the oscilloscope is stopped.
2. Turn the sweep speed knob to zoom horizontally and turn the volts/division knob to zoom vertically.

The ∇ symbol at the top of the display indicates the time reference point where the zoom-in/zoom-out is referenced.

3 Turn the Delay Time knob (◀▶) to pan horizontally and turn the channel's vertical position knob (▲▼) to pan vertically.

The stopped display may contain several triggers worth of information, but only the last trigger acquisition is available for pan and zoom.

To set the waveform expansion reference point

When you change a channel's volts/division setting, the waveform display can be set to expand (or compress) about the signal ground level or the center of the display.

Expand About Ground The displayed waveform will expand about the position of the channel's ground. This is the default setting. The ground level of the signal is identified by the position of the ground level (⏚▶) icon at the far-left side of the display. The ground level will not move when you adjust the vertical sensitivity (volts/division) control.

If the ground level is off screen, the waveform will expand about the top or bottom edge of the screen based on where the ground is off screen.

Expand About Center The displayed waveform will expand about the center of the display.

To set the waveform expansion reference point

Press **Utility→Options→Preferences→Expand** and select **Ground** or **Center**.

Antialiasing

At slower sweep speeds, the sample rate is reduced and a proprietary display algorithm is used to minimize the likelihood of aliasing.

By default, Antialiasing is enabled. You should leave Antialiasing enabled unless there is a specific reason to switch it off.

If you need to switch Antialiasing off, press **Utilities→Options→Preferences** and press the **Antialiasing** softkey to switch the feature off. The displayed waveforms will be more susceptible to aliasing.

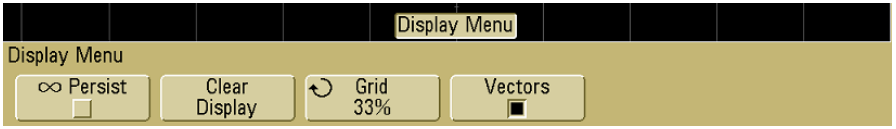
Using the XGA video output

A standard XGA video output connector is provided on the rear panel. You can connect a monitor to provide a larger display, or to provide a display at a viewing position away from the oscilloscope.

The oscilloscope's built-in display will remain on even when an external display is connected.

Display Settings

- Press the **Display** key to view the Display menu.



Infinite persistence

With infinite persistence, the oscilloscope updates the display with new acquisitions, but does not erase the results of previous acquisitions. All previous acquisitions are displayed in gray with reduced intensity, New acquisitions are shown in normal color with normal intensity. Waveform persistence is not kept beyond the display area boundary.

Use infinite persistence to measure noise and jitter, to see the worst-case extremes of varying waveforms, to look for timing violations, or to capture events that occur infrequently.

To use infinite persistence to display multiple repetitive events

1 Connect a signal to the oscilloscope.

2 Press the **Display** key, then press ∞**Persist** to turn on infinite persistence. The display will begin accumulating multiple acquisitions. The accumulated waveforms are shown in gray with reduced intensity.

3 Press the **Clear Display** softkey to erase previous acquisitions.

The oscilloscope will again start to accumulate acquisitions.

4 Turn off infinite persistence, then press the **Clear Display** key to return the oscilloscope to the normal display mode.

Accumulating multiple acquisitions

Turning off infinite persistence does not clear the display. This allows you to accumulate multiple acquisitions, stop acquisitions, and then compare future acquisitions to the stored waveforms.

Clearing stored infinite persistence waveforms

In addition to clearing the display by pressing the **Clear Display** softkey, the display is also cleared of previous acquisitions if you press the **AutoScale** key.

Grid intensity

To adjust the grid (graticule) intensity, press **Display→Grid** and use the Entry knob ↻ to adjust the intensity.

Vectors (connect the dots)

The oscilloscopes are designed to operate optimally with vectors on. This mode gives the most realistic waveforms in most situations.

When enabled, **Vectors** draws a line between consecutive waveform data points.

- Vectors give an analog look to a digitized waveform.
- Vectors allow you to see steep edges on waveforms, such as square waves.
- Vectors allow subtle detail of complex waveforms to be viewed, much like an analog oscilloscope trace, even when the detail is just a small number of pixels in size.

The oscilloscope switches vectors on whenever the acquisition system stops.

Using Vectors (Display menu)

One of the most fundamental choices you must make about your display is whether to draw vectors (connect the dots) between the samples, or simply let the samples fill in the waveform. To some degree, this is a matter of personal preference, but it also depends on the waveform.

- You will probably operate the oscilloscope most often with vectors on. Complex analog signals like video and modulated signals show analog-like intensity information with vectors on.
- Turn vectors off when highly complex or multivalued waveforms are displayed. Turning vectors off may aid the display of multivalued waveforms such as eye diagrams.
- Having vectors on does not slow down the display rate.

Varying the intensity to view signal detail

The **Intensity** knob lets you adjust the plotted waveforms to account for various signal characteristics, such as fast sweep speeds and low trigger rates. Increasing the intensity lets you see the maximum amount of noise and infrequently occurring events. Reducing the intensity can expose more detail in complex signals as shown in the following figures.

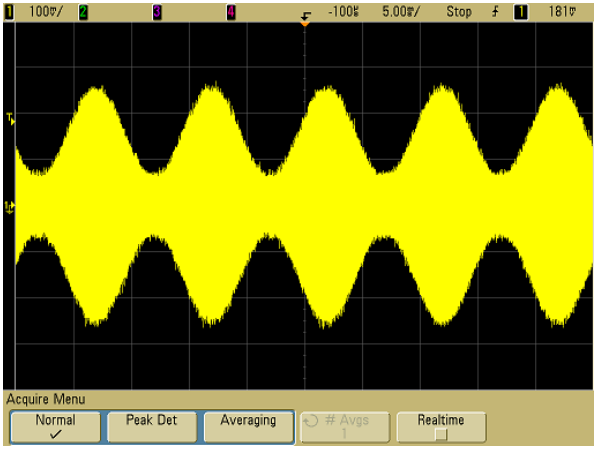


Figure 37 Amplitude Modulation with Noise Shown at 100% Intensity

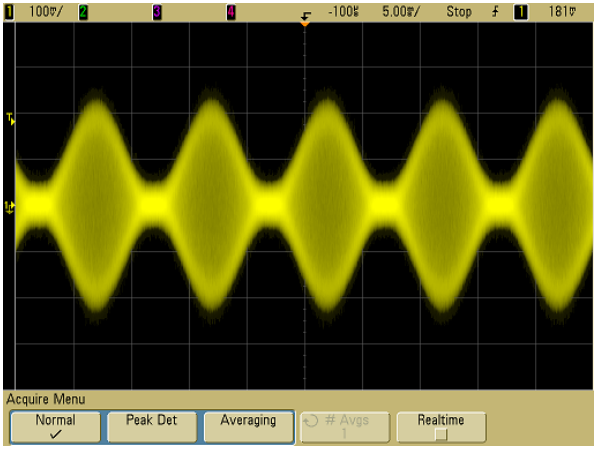


Figure 38 Amplitude Modulation with Noise Shown at 40% Intensity

Acquisition Modes

The 5000A Series oscilloscopes have the following acquisition modes:

- **Normal** — for most waveforms (with normal decimating at slower sweep speeds, no averaging).
- **Peak Detect** — for displaying narrow pulses that occur infrequently (at slower sweep speeds).
- **Averaging** — for reducing noise and increasing resolution (at all sweep speeds, without bandwidth or rise time degradation).
- **High Resolution** — for reducing random noise (at slower sweep speeds).

Realtime sampling (where the oscilloscope produces the waveform display from samples collected during one trigger event) can be turned off or on in the Normal, Peak Detect, and High Resolution modes.

At Slower Sweep Speeds

At slower sweep speeds, the sample rate drops because the acquisition time increases and the oscilloscope's digitizer is sampling faster than it must to fill memory.

For example, suppose an oscilloscope's digitizer has a sample period of 1 ns (maximum sample rate of 1 GSa/s) and a 1 M memory depth. At that rate, memory is filled in 1 ms. If the acquisition time is 100 ms (10 ms/div), only 1 of every 100 samples is needed to fill memory.

Selecting the Acquisition mode

To select the acquisition mode press the Acquire key on the front panel.

Normal Mode

In Normal mode at slower sweep speeds, extra samples are decimated (in other words, some are thrown away). This mode yields the best display for most waveforms.

Peak Detect Mode

In Peak Detect mode at slower sweep speeds, minimum and maximum samples are kept in order to capture infrequent and narrow events (at the expense of exaggerating any noise). This mode displays all pulses that are at least as wide as the sample period (see Table 9).

Table 9 Agilent 5000A Series Model Numbers and Sampling Rates

Bandwidth	**100 MHz**	**300 MHz**	**500 MHz**
Maximum Sample Rate	**2 GSa/s**	**2 GSa/s**	**4 GSa/s**
A sample is taken every (sample period)	**500 ps**	**500 ps**	**250 ps**
2-Channel DSO	DSO5012A	DSO5032A	DSO5052A
4-Channel DSO	DSO5014A	DSO5034A	DSO5054A

High Resolution Mode

In High Resolution mode, at slower sweep speeds extra samples are averaged in order to reduce random noise, produce a smoother trace on the screen, and effectively increase vertical resolution.

High Resolution mode averages sequential sample points within the same acquisition together. An extra bit of vertical resolution is produced for every factor of 4 averages. The number of extra bits of vertical resolution is dependent on the time per division setting (sweep speed) of the scope.

The slower the sweep speed, the greater the number of samples that are averaged together for each display point.

High Resolution mode is equivalent to the Averaging mode with #Averages=1; however, you can turn on Realtime sampling in the High Resolution mode.

High Resolution mode can be used on both single-shot and repetitive signals and it does not slow waveform update because the computation is done in the MegaZoom custom ASIC. High Resolution mode limits the scope's real-time bandwidth because it effectively acts like a low-pass filter.

Table 10 Sample Rate, Sweep Speed, and Bits of Resolution

2 GSa/s Sample Rate	4 GSa/s Sample Rate	Bits of resolution (# Avgs=1)
≤ 50 ns/div	≤ 50 ns/div	8
200 ns/div	100 ns/div	9
1 us/div	500 ns/div	10
5 us/div	2 us/div	11
≥ 20 us/div	≥ 10 us/div	12

Averaging Mode

The Averaging mode lets you average multiple acquisitions together to reduce noise and increase vertical resolution (at all sweep speeds). Averaging requires a stable trigger.

The number of averages can be set from 1 to 65536 in powers-of-2 increments.

A higher number of averages reduces noise more and increases vertical resolution.

Table 11 Number of Averages, Bits of Resolution

# Avgs	Bits of resolution
2	8
4	9
16	10
64	11
≥ 256	12

The higher the number of averages, the slower the displayed waveform responds to waveform changes. You must compromise between how quickly the waveform responds to changes and how much you want to reduce the displayed noise on the signal.

To use the Averaging mode

1 Press the **Acquire** key, then press the **Acq Mode** softkey until the Averaging mode is selected.

2 Press the **#Avgs** softkey and turn the Entry knob to set the number of averages that best eliminates the noise from the displayed waveform. The number of acquisitions being averaged is displayed in the **# Avgs** softkey.

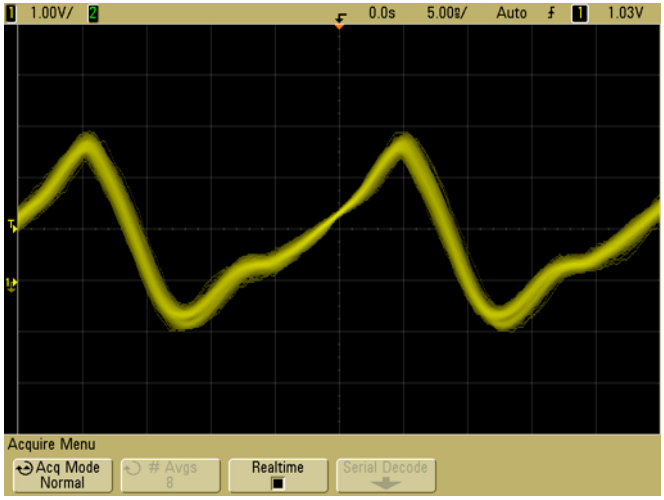


Figure 39 Random noise on the displayed waveform

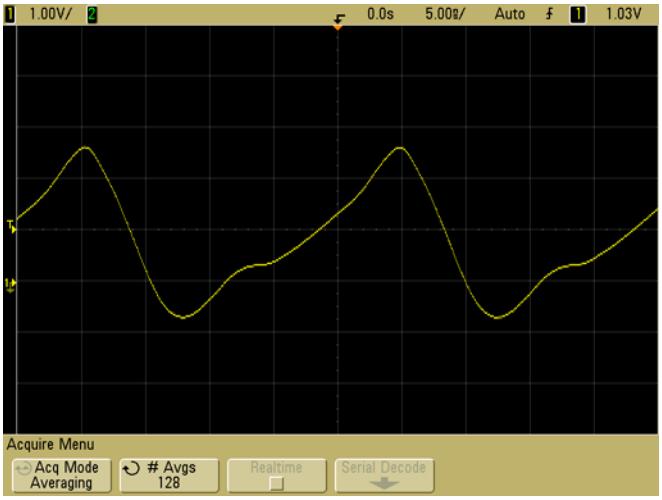


Figure 40 128 Averages used to reduce random noise

Realtime Sampling Option

Realtime sampling specifies that the oscilloscope produce the waveform display from samples collected during one trigger event (that is, one acquisition).

Use Realtime sampling to capture infrequent triggers, unstable triggers, or complex changing waveforms, such as eye diagrams.

Realtime sampling can be turned on in **Normal**, **Peak Detect**, or **High Resolution** acquisition modes. It cannot be turned on when the acquisition mode is **Averaging**.

When Realtime sampling is on (as in the default setup):

- When less than 1000 samples can be collected in the time spanned by the screen, a sophisticated reconstruction filter is used to fill in and enhance the waveform display.
- If you press the **Stop** key, and pan and zoom through the waveform using the Horizontal and Vertical controls, only the last trigger's acquisition will be displayed.

When Realtime sampling is off:

- The oscilloscope produces the waveform display from samples collected from multiple acquisitions. In this case, the reconstruction filter is not used.

Realtime Sampling and Oscilloscope Bandwidth

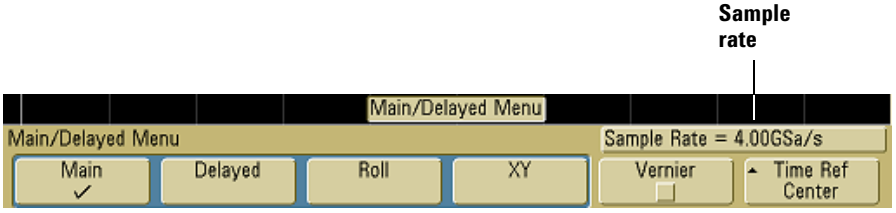
To accurately reproduce a sampled waveform, the sample rate should be at least four times the highest frequency component of the waveform. If not, it is possible for the reconstructed waveform to be distorted or aliased. Aliasing is most commonly seen as jitter on fast edges.

The maximum sample rate for 100 MHz and 300 MHz bandwidth oscilloscopes is 2 GSa/s.

The maximum sample rate for and 500 MHz bandwidth oscilloscopes is 4 GSa/s for a single channel in a channel pair. Channels 1 and 2 constitute a channel pair, and channels 3 and 4 constitute another channel pair. For example, the sample rate of a 4-channel oscilloscope is 4 GSa/s when channels 1 and 3, 1 and 4, 2 and 3, or 2 and 4 are on.

Whenever both channels in a channel pair are on, the sample rate for all channels is halved. For example, when channels 1, 2, and 3 are on, the sample rate for all channels is 2 GSa/s.

To see the sample rate, press the **Main/Delayed** key on the front panel. The sample rate is displayed in the line just above the softkeys.



To reduce the random noise on a signal

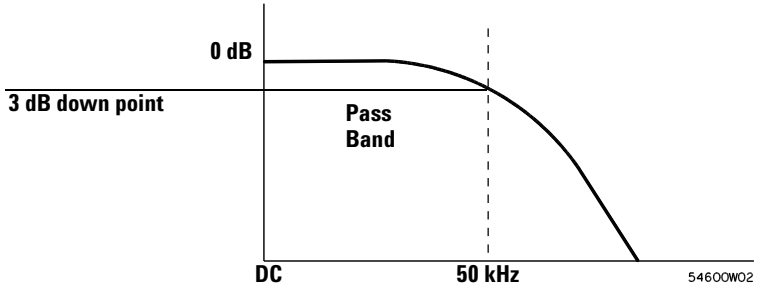
If the signal you are probing is noisy, you can set up the oscilloscope to reduce the noise on the displayed waveform. First, you stabilize the displayed waveform by removing the noise from the trigger path. Second, you reduce the noise on the displayed waveform.

1. Connect a signal to the oscilloscope and obtain a stable display.
2. Remove the noise from the trigger path by turning on high-frequency rejection (HF reject), low-frequency rejection (LF reject), or noise rejection (see the following pages).
3. Use averaging (see page 187) to reduce noise on the displayed waveform.

HF Reject

High-frequency reject (HF reject) adds a low-pass filter with the 3-dB point at 54 kHz. HF reject removes high-frequency noise, such as AM or FM broadcast stations, from the trigger path.

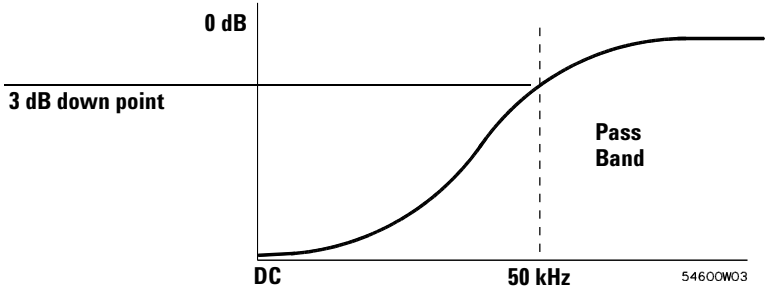
- Press **Mode/Coupling→HF Reject**.



LF Reject

Low-frequency reject (LF reject) adds a high-pass filter with the 3-dB point at 50 kHz. LF reject removes low-frequency signals, such as power line noise, from the trigger path.

- Press **Mode/Coupling→Coupling→LF Reject**.



Noise rejection

Noise reject increases the trigger hysteresis band. By increasing the trigger hysteresis band, you reduce the possibility of triggering on noise. However, this also decreases the trigger sensitivity so that a slightly larger signal is required to trigger the oscilloscope.

- Press **Mode/Coupling→HF Reject**.

To capture glitches or narrow pulses with peak detect and infinite persistence

A glitch is a rapid change in the waveform that is usually narrow as compared to the waveform. Peak detect mode can be used to more easily view glitches or narrow pulses. In peak detect mode, narrow glitches and sharp edges are displayed more brightly than when in Normal acquire mode, making them easier to see.

To characterize the glitch, use the cursors or the automatic measurement capabilities of the oscilloscope.

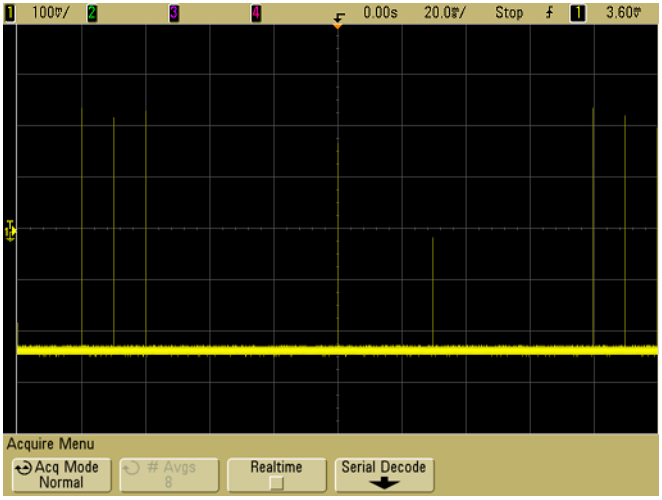


Figure 41 15 ns Narrow Pulse, 20 ms/div, Normal Mode

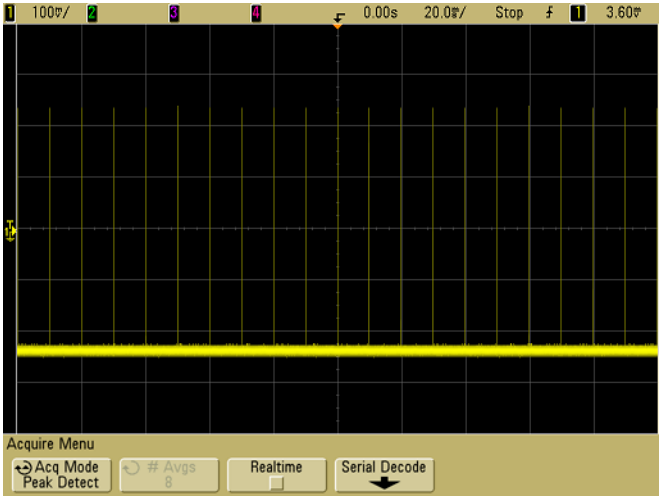


Figure 42 15 ns Narrow Pulse, 20 ms/div, Peak Detect Mode

Using peak detect mode to find a glitch

1. Connect a signal to the oscilloscope and obtain a stable display.
2. To find the glitch, press the **Acquire** key, then, press the **Acq Mode** softkey until **Peak Detect** is selected.
3. Press the **Display** key then press the ∞**Persist** (infinite persistence) softkey.

   Infinite persistence updates the display with new acquisitions but does not erase previous acquisitions. New sample points are shown at normal intensity while previous acquisitions are displayed in gray at lower intensity. Waveform persistence is not kept beyond the display area boundary.

   Press the **Clear Display** softkey to erase previously acquired points. The display will accumulate points until ∞**Persist** is turned off.
4. Characterize the glitch with delayed sweep:
   a. Press the **Main/Delayed** key, then press the **Delayed** softkey.
   b. To obtain a better resolution of the glitch, expand the time base.
   c. Use the horizontal delay time knob (◀▶) to pan through the waveform to set the expanded portion of the main sweep around the glitch.

How AutoScale Works

AutoScale automatically configures the oscilloscope to best display the input signals by analyzing any waveforms present at each channel and at the external trigger input.

AutoScale finds, turns on, and scales any channel with a repetitive waveform that has a frequency of at least 50 Hz, a duty cycle greater than 0.5%, and an amplitude of at least 10 mV peak-to-peak. Any channels that do not meet these requirements are turned off.

The trigger source is selected by looking for the first valid waveform starting with external trigger, then continuing with the highest number channel down to the lowest number channel.

During AutoScale, the delay is set to 0.0 seconds, the sweep speed setting is a function of the input signal (about 2 periods of the triggered signal on the screen), and the triggering mode is set to edge. Vectors remain in the state they were before the AutoScale.

Undo AutoScale

Press the **Undo AutoScale** softkey to return the oscilloscope to the settings that existed before you pressed the **AutoScale** key.

This is useful if you have unintentionally pressed the **AutoScale** key or do not like the settings AutoScale has selected and want to return to your previous settings.

Specifying the Channels Displayed After AutoScale

The **Channels** softkey selection determines which channels will be displayed on subsequent AutoScales.

- **All Channels** — The next time you press **AutoScale**, all channels that meet the requirements of AutoScale will be displayed.
- **Only Displayed Channels** — The next time you press **AutoScale**, only the channels that are turned on will be examined for signal activity. This is useful if you only want to view specific active channels after pressing **AutoScale**.

Preserving the Acquisition Mode During AutoScale

The acquisition mode is normally switched to Normal when AutoScale is performed. AutoScale can be set to leave the acquisition mode unchanged if you prefer this option.

Choose Normal to make the oscilloscope switch to Normal acquisition mode whenever the **AutoScale** key is pressed. This is the default mode.

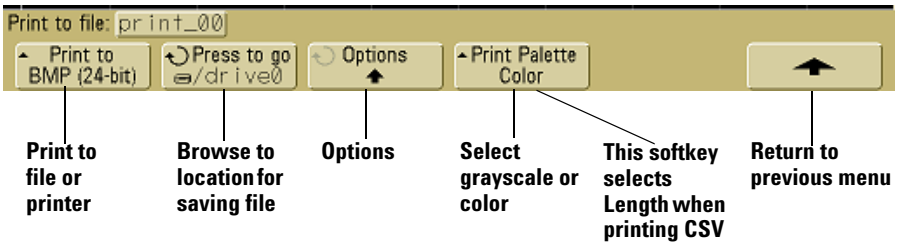
Choose Preserve to make the oscilloscope remain in the acquisition mode you have chosen when the **AutoScale** key is pressed.

6
Saving and Printing Data

To configure printing

You can print to a file or you can print to a USB printer. Use the **Print Config** menu to choose the type of image file you would like to create or to set up your printer.

You can print scale factors, print in color or grayscale, and choose whether to print each waveform on a separate sheet of paper (form feed). To conserve printer ink, you can invert the graticule colors so that the background is white instead of black.



Selecting a print file format

To choose a file format, press **Utility→Print Config→Print to**. You can create an image file in one of the following file formats:

- **BMP (8-bit) image file** — The screen image is converted to a smaller, lower resolution bitmap file of the complete screen, including the status line and softkeys.
- **BMP (24-bit) image file** — This is a larger, high-resolution bitmap file of the complete screen, including the status line and softkeys.
- **PNG (24-bit) image file** — This is a high-resolution PNG format bitmap file of the complete screen, including the status line and softkeys.
- **CSV data file** — This creates a file of comma-separated variable values of displayed channels and math waveforms. This format is suitable for spreadsheet analysis.

- **ASCII XY data file** — Gives a separate file for each channel's output. For example: print_nn_channel1.csv. Maximum record length is obtained in single shot mode.
- **BIN data file** — Saves waveform data to a binary format file (see "Binary Data (.bin)" on page 221).

Length Control

The **Length** softkey appears when the CSV, ASCII XY, or BIN formats are selected. **Length** sets the number of data points that will be output to the file. **Length** can be set to 100, 250, 500, or 1000 when the acquisition is running or more when the acquisition is stopped.

Only displayed data points are output. Therefore, you need to adjust the Horizontal controls to display the data you want to save.

The Length control will perform a "1 of n" decimation of the data when necessary. For example: if the **Length** is set to 1000, and you are displaying a record that is 5000 data points in length, four of each five data points will be decimated, creating an output file 1000 data points in length.

Minimum and Maximum Values in CSV Files

If you are running a Quick Measurement Minimum or Maximum measurement, the minimum and maximum values shown in the Quick Measurement display may not appear in the CSV file.

Explanation:

When the oscilloscope's sample rate is 4 GSa/s, a sample will be taken every 250 ps. If the sweep speed is set to 100 ns/div, there will be 1000 ns of data displayed (because there are ten divisions across the screen). To find the total number of samples the oscilloscope will take:

$$1000ns \times 4Gsa/\ s\ = 4000samples$$

For measurement data, the oscilloscope will decimate the 4000 points to 1000 points that will fit on the display. This decimation will not lose track of the min and max values at each of the 1000 horizontal data points, and min and max values will

be displayed on screen. However, the oversampled data is also processed to provide a best-estimate value at each of the 1000 horizontal points. The data in the CSV file will be the best-estimate value at each of the 1000 horizontal points. Therefore, the min and max values may not appear in the CSV file.

This occurs when oversampling occurs
(10 * seconds per division * maximum sample rate > 1000).

NOTE

To save the CSV, ASCII XY, or BIN data, or BMP or PNG images to the USB storage device, it is necessary to press the **Quick Print** key (see "To print the display" on page 78).

Selecting print options

Press **Utility→Print Config→Options**.

- **Factors** — Select **Factors** if you want the oscilloscope scale factors to be included on your print. If you print to an image file, the scale factors will be sent to a separate file named print_nn.txt. If you print to a CSV file, the factors will be appended to the end of the file. Oscilloscope scale factors include vertical, horizontal, trigger, acquisition, math, and display settings.
- **Invert Graticule Colors** — The **Invert Graticule Colors** option can be used to reduce the amount of black ink it takes to print oscilloscope images by changing the black background to white.
- **Form Feed** — The **Form Feed** option can be selected to send a form feed command to the printer when the print is complete. Use this if you want only one print per sheet of paper. Switch **Form Feed** off when you want to put more than one print on a sheet of paper. The **Form Feed** option is grayed-out and unavailable when printing to a file.

Print Palette

- **Color** — When **Color** printing is selected, the traces are printed in color. Color printing is not applicable to CSV format.
- **Grayscale** — When **Grayscale** printing is selected, the traces are printed in shades of gray rather than in color. Grayscale printing is not applicable to CSV format.

To print the display to a file

1. To print to a file, connect a USB mass storage device to the USB port on the front or the rear of the oscilloscope.
2. Access the Print Config menu by pressing **Utility→Print Config**.
3. Choose a format using the **Print to** softkey.
4. Press the second softkey from the left and use the Entry knob to browse to a location where you would like to save the image file. You can select among connected USB mass storage devices, and you can choose a subdirectory if desired.
5. Press the **Quick Print** key on the front panel
6. To make subsequent printouts, simply press the **Quick Print** key.

NOTE If two USB mass storage devices are connected to the oscilloscope, the first one is designated "drive0"" and the second one is designated "drive5," not "drive1." This numbering method is normal; it is inherent in the USB driver.

To print the display to a USB printer

A USB printer can be connected to the oscilloscope through a USB host port on the front or rear of the oscilloscope. (The USB host ports are rectangular; the USB device port is square.) You need a USB cable to connect the printer.

1 Connect the printer to the USB port on either the front or the rear of the oscilloscope. A list of supported printers is given on page 206.

2 To access the Print Config menu, press **Utility→Print Config**.

3 Press the **Print to** softkey and select the printer.

   If the oscilloscope identifies the printer you have connected, the oscilloscope will choose the correct driver.

   If the oscilloscope does not automatically choose the driver for your printer, use the **Driver** softkey and the Entry knob to select the correct driver for your printer. Choose **Generic** if you don't know which driver to use.

4 Press the **Quick Print** key on the front panel.

5 To make subsequent printouts, simply press the **Quick Print** key.

Supported printers

Printers

The following HP printers, available in stores at the time this manual was written, have been tested and found to be compatible with the 6000 Series oscilloscopes.

Deskjet 672C
Deskjet 694C
Deskjet 840C
Deskjet 935C
Deskjet 952C
Deskjet 6940
Deskjet 6980
Deskjet 9800
Officejet 7310
Officejet 7410
PhotoSmart C5180
PhotoSmart C6180
PhotoSmart C7100
PhotoSmart D7160
PhotoSmart D7360
PhotoSmart 8750

The following HP printers will likely work with the oscillocope, but have not been tested for compatibility.

Deskjet 350C
Deskjet 610C & 612C
Deskjet 630C & 632C
Deskjet 656
Deskjet 825
Deskjet 845C
Deskjet 648C
Deskjet 810C & 812C & 815C & 816C
Deskjet 842C
Deskjet 920
Deskjet 932C & 935C
Deskjet 940 & 948
Deskjet 952C
Deskjet 960
Deskjet 970C

Deskjet 980
Deskjet 990C
Deskjet 995
Deskjet 1220C & 1125C
Deskjet 3816 & 3820
Deskjet 5550 & 5551
Deskjet 6122 & 6127
Deskjet 5600 & 5100 & 5800
Deskjet 6540 & 6520
Deskjet 5740
Deskjet 6840
Deskjet 3740 & 3840
Deskjet CP1160 & CP1700
Deskjet 9300 & 9600
PhotoSmart PS100 & PS130 & PS230 & PS140 & PS240 & 1000 & 1100
PhotoSmart 320 & 370
PhotoSmart P2500 & P2600
PhotoSmart PS1115 & PS1215 & PS12818 & PS1315
PhotoSmart PS7150 & PS7350 & PS7550
PhotoSmart PS7960 & PS7760 & & PS7660 & PS7260 & PS7268
PhotoSmart 7400 & 8100 & 8400
PhotoSmart 2600 & 2700
PSC 1100 & 1200 & 1300 & 1310
PSC 2100 & 2150 & 2200 & 2300 & 2400 & 2500 & 2170
PSC 1600 & 2350
Officejet 5100 & 6100 & 6150 & 7100 & 4100 & 4105 & 4200 & 5500 & 9100
Officejet 7200 & 7300 & 7400 & 6200
Apollo P2100 & P2150
Apollo P2200 & P2250
E-Printer e20
Business InkJet 2200 & 2230 & 2250 & 2280 & 3000 & 1100 & 2300
Business InkJet 1200
Deskjet 600
Deskjet 640 & 642 & 644
Deskjet 660C
Deskjet 670 & 670TV & 672TV & 672C
Deskjet 680C & 682C
Deskjet 690C & 692C & 693C & 694C & 695C & 697C
Deskjet 830C & 832C
Deskjet 840C & 843
Deskjet 880 & 882C
Deskjet 895C
Deskjet 930C
Deskjet 950C & 955 & 957
Deskjet 975C

Secure Environment Mode Option

Secure environment mode is compliant to National Industrial Security Program Operating Manual (NISPOM) Chapter 8 requirements.

When you order your oscilloscope with the secure environment mode option, trace and setup data are stored to internal volatile memory (rather than internal non-volatile memory). Oscilloscope setup, waveform, and trace data are erased when the power is shut off. This ensures that any setup, waveform, and trace data will not be visible to the next user upon power-up. The clock, LAN, and GPIB settings are not discarded upon power-down.

To permanently store data, you can save it to an external device through one of the oscilloscope's USB ports.

Secure environment mode can not be disabled.

Oscilloscopes equipped with the secure environment mode option will have "SEC" displayed in the Installed Licenses line of the "About Oscilloscope" display. To access the About Oscilloscope display, press the **Utility** key, then press the **Service** softkey, then press the **About Oscilloscope** softkey.

Saving and recalling traces and setups

You can save the oscilloscope's current setup and waveform trace to the oscilloscope's internal memory. In 5000A Series oscilloscopes that do not have the Secure Environment Mode option the data are stored to non-volatile memory. In 5000A Series oscilloscopes that are equipped with Secure Environment mode the data are stored to volatile memory.

In all 5000A Series oscilloscopes, you can save the setup and waveform trace to a USB mass storage device (for example, a USB flash drive), then recall the setup, waveform trace, or both later.

Do not connect USB "hub" devices or USB devices that identify themselves as hardware type "CD" because these devices are not compatible with the 5000A Series oscilloscopes.

When you save a setup, all settings including measurements, cursors, math functions, and horizontal, vertical, and trigger settings are saved to the file you have selected.

Saving a trace lets you save the visible portion of the acquisition (the displayed waveform) for later recall and comparison with other measurements. Recalled traces appear on the display in blue.

A recalled trace is typically used for a quick comparison of measurement results. For example, you might make a measurement on a known good system, save the result in internal memory or to a USB mass storage device, then make the same measurement on a test system and recall trace to see the differences.

- Press the **Save/Recall** key to display the Save/Recall menu.

To AutoSave traces and setups

1 Connect a signal to the oscilloscope and obtain a stable display.

2 Connect a USB mass storage device to the USB port on the front panel or the rear panel.

NOTE

Regarding USB Ports:

The USB port on the front panel and the USB port on the rear panel labeled "HOST" are USB Series A receptacles. These are the receptacles to which you can connect USB mass storage devices and printers.

The square receptacle on the rear panel labeled "DEVICE" is provided for controlling the oscilloscope over USB. See the *Oscilloscopes Programmer's Quick Start Guide* or the *Oscilloscopes Programmer's Reference* for more information. To access these documents online, point your web browser to www.agilent.com/find/dso5000 and select Technical Support, then select Manuals.

If two USB mass storage devices are connected to the oscilloscope, the first one is designated "drive0" and the second one is designated "drive5," not "drive1." This numbering method is normal; it is inherent in the USB driver.

3 Press the **Save/Recall** key.

4 Turn the Entry knob and then press the far left softkey to select a directory on a USB mass storage device.

5 Press the **Press to AutoSave** softkey.

Your current setup and waveform trace will be saved to files using automatically generated file names (`QFILE_nn`) on the USB mass storage device. The file name is shown in the line above the softkeys.

The `nn` number in the `QFILE_nn` file name will automatically increment (starting at `00`) each time you save a new file to the USB mass storage device.

When viewed from the File Explorer Menu (**Utility→File Explorer**) the trace file will have a `TRC` file extension and the setup file will have a `SCP` file extension.

To save traces and setups to internal memory or to overwrite an existing USB mass storage device file

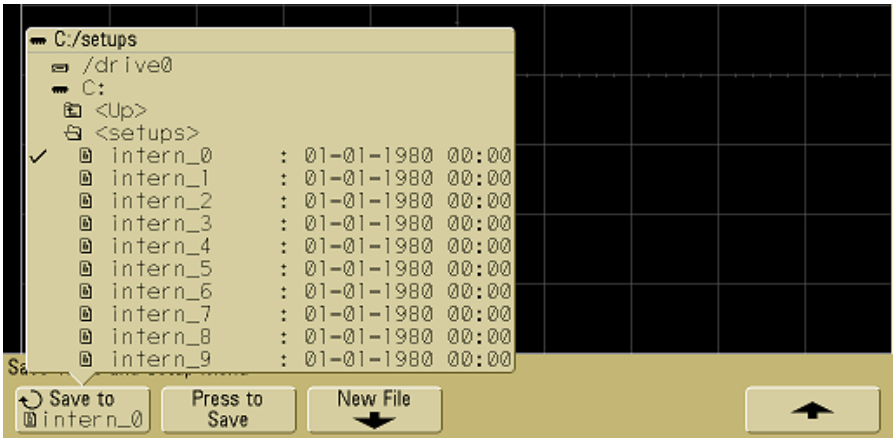
1. If you will be saving a trace and/or setup to a USB mass storage device, connect the device to the oscilloscope.
2. Press the **Save/Recall** key.
3. Press the **Save** softkey to display the Save menu.

4. Turn the Entry knob and press the far left softkey to select an internal memory file or USB mass storage device file to be overwritten.

In the following screen image:

- **drive0** is a USB mass storage device which has been connected to the oscilloscope.
- **C:** is the root directory of the oscilloscope's internal memory.
- **intern0** through **intern9** are the internal nonvolatile memory locations that can be used for storing setups and traces.
- Use the **<up>** selection to move up one level in the directory structure.

You cannot create new file names in the oscilloscope's internal memory, you can only overwrite the existing files.



5 When you have selected the file name to be overwritten, press the **Press to Save** softkey to save your current setup and waveform trace to the file.

To save traces and setups to a new file on the USB mass storage device

1 Follow steps 1-3 in the procedure on page 211.

2 Turn the Entry knob and press the far left softkey to select a directory on a USB mass storage.

3 To create a new file name, press the **New File** softkey.

New file names can only be written to a USB mass storage device, not the internal memory.

4 Turn the Entry knob to select the first character in the file name.

Turning the Entry knob selects a character to enter into the highlighted position shown in the "**`New file name =`**" line above the softkeys and in the **Spell** softkey.

5 Press the **Enter** softkey to enter the selected character and go to the next character position.

You may position the highlight on any character in the file name by successively pressing the **Enter** softkey.

6 To delete a character from the file name, press the **Enter** softkey until the letter you want to delete is highlighted, then press the **Delete Character** softkey.

7 When you are done entering characters for the file name, press the **Press to Save** softkey to save the file.

Two files will be saved to the USB mass storage device. In the example above, **`SCOPE1.TRC`** will be the trace file and **`SCOPE1.SCP`** will be the setup file. You do not need to remember these file extensions because you can select trace, setup, or both when you recall this information using the Recall menu.

To recall traces and setups

1 If you will be recalling a trace and/or setup from a USB mass storage device, connect the device to the oscilloscope.

2 Press the **Save/Recall** key to display the Save/Recall menu.

3 Press the **Recall** softkey to display the Recall menu.

4 Press the **Recall:** softkey and select the type of information to recall.

You can recall a waveform **Trace**, an oscilloscope **Setup**, or **Trace and Setup**.

NOTE Be sure to recall both setup and trace if you want to measure the recalled trace with cursors.

5 Select the directory and choose a file to recall by turning the Entry knob and pressing the associated softkey.

`INTERN_n` are internal nonvolatile oscilloscope memory files. All other files in the list are stored on the USB mass storage device.

NOTE

Recalling overwrites current settings

Because recalling a setup will overwrite the oscilloscope's current settings, you may want to save the existing setup first.

6 Recall the selected file by pressing the **Press to Recall** softkey.

7 The recalled trace will be displayed in blue.

8 To clear the display of any recalled trace, press **Display→Clear Display**.

To use the file explorer

The File Explorer menu lets you load or delete files from a USB mass storage device.

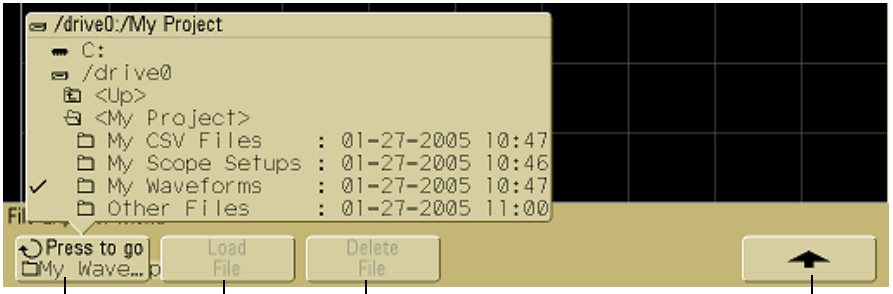
1 Connect a USB mass storage device to the USB port on the front or the rear of the oscilloscope. A small colored circle icon will be displayed as the USB device is read.

NOTE Do not connect USB "hub" devices or USB devices that identify themselves as hardware type "CD" because these devices are not compatible with the 5000A Series oscilloscopes.

2 Press **Utility→File Explorer**.

3 Press the far left softkey and turn the Entry knob to select the USB mass storage device, and a directory and file on the USB mass storage device.

You can create directories on a USB mass storage device using your PC or other instrument. You can navigate to any directory that you have made using the Entry knob and the far left softkey.



Navigate using Entry knob, select using this key

Load selected file

Delete selected file

Return to previous menu

NOTE If two USB mass storage devices are connected to the oscilloscope, the first one is designated "drive0" and the second one is designated "drive5," not "drive1." This numbering method is normal; it is inherent in the USB driver.

4 To load a file into the oscilloscope press the **Load File** softkey.

Files that can be loaded into the oscilloscope:

- `QFILE_nn.SCP` setup files, `QFILE_nn.TRC` trace files, and other user-defined setup or trace files that were created using the **Save/Recall** key on the front panel of the oscilloscope.
- Localized language pack files (`LANGPACK.JZP`).
- System software files (`*.BIN` and `*.JZP`).

Files that cannot be loaded into the oscilloscope:

- Any `PRINT_nn.xxx` printer file.
- Any other file not created by the oscilloscope.

5 To delete a file from the USB mass storage device, press the **Delete File** softkey.

NOTE **Deleted files cannot be recovered**

A file that has been deleted from the USB mass storage device cannot be recovered by the oscilloscope.

7 Reference

Software and firmware updates

From time to time Agilent Technologies releases software and firmware updates for its products. To search for firmware updates for your oscilloscope, point your web browser to www.agilent.com/find/dso5000 and select **Technical Support**, then select **Software Downloads & Utilities**.

To view the currently installed software and firmware press **Utility→Service→About Oscilloscope**.

To set up the I/O port

The oscilloscope can be controlled via GPIB, LAN, or USB. By default, all three are active, although they can be selected or disabled using the **Control** softkey in the I/O menu (press **Utility→I/O**).

The oscilloscope's I/O configuration, including its IP address and hostname, can be viewed by pressing **Utility→I/O**.

To change the I/O controller settings, press the **Configure** softkey, and choose the I/O connection type (GPIB, LAN, or USB).

For instructions on setting up the oscilloscope to run from a controller via LAN, GPIB, or USB, see the *Programmer's Quick Start Guide*.

To check warranty and extended services status

To learn the warranty status of your oscilloscope:

1 Point your web browser to: www.agilent.com/find/warrantystatus

2 Enter your product's model number and serial number. The system will search for the warranty status of your product and display the results. If the system cannot find your product's warranty status, select **Contact Us** and speak with an Agilent Technologies representative.

To return the instrument

Before shipping the oscilloscope to Agilent Technologies, contact your nearest Agilent Technologies sales or service office for additional details. Information on contacting Agilent Technologies can be found at www.agilent.com/find/contactus.

1. Write the following information on a tag and attach it to the oscilloscope.
   - Name and address of owner
   - Model number
   - Serial number
   - Description of service required or failure indication
2. Remove accessories from the oscilloscope.

   Only return accessories to Agilent Technologies if they are associated with the failure symptoms.
3. Package the oscilloscope.

   You can use the original shipping container, or provide your own materials sufficient to protect the instrument during shipping.
4. Seal the shipping container securely, and mark it FRAGILE.

To clean the oscilloscope

1. Remove power from the instrument.
2. Clean the external surfaces of the oscilloscope with a soft cloth dampened with a mixture of mild detergent and water.
3. Make sure that the instrument is completely dry before reconnecting it to a power source.

Binary Data (.bin)

The binary data format stores waveform data in binary format and provides data headers that describe that data.

Because the data is in binary format, the size of the file is approximately 5 times smaller than the XYPairs format.

If more than one source is on, you will save all displayed sources, except math functions, to a file.

When the oscilloscope is in the Peak Detect acquisition mode, the minimum and maximum value waveform data points are saved to the file in separate waveform buffers. The minimum value data points are saved first; then, the maximum value data points are saved.

Binary Data in MATLAB

Binary data from the 5000A Series oscilloscope can be imported to The MathWorks MATLAB®. You can download the appropriate MATLAB functions from the Agilent Technologies web site at www.agilent.com/find/dso5000sw.

Agilent provides the .m files, which need to be copied into the work directory for MATLAB. The default work directory is C:\MATLAB7\work.

Binary Header Format

File Header

There is only one file header in a binary file. The file header consists of the following information.

Cookie Two byte characters, AG, that indicate the file is in the Agilent Binary Data file format.

Version Two bytes that represent the file version.

File Size A 32-bit integer that is the number of bytes that are in the file.

Number of Waveforms A 32-bit integer that is the number of waveforms that are stored in the file.

Waveform Header

It is possible to store more than one waveform in the file, and each waveform stored will have a waveform header. The waveform header contains information about the type of waveform data that is stored following the waveform data header.

Header Size A 32-bit integer that is the number of bytes in the header.

Waveform Type A 32-bit integer that is the type of waveform stored in the file:

- 0 = Unknown.
- 1 = Normal.
- 2 = Peak Detect.
- 3 = Average.
- 4 = Not used in 5000A Series oscilloscopes.
- 5 = Not used in 5000A Series oscilloscopes.
- 6 = Not used in 5000A Series oscilloscopes.

Number of Waveform Buffers A 32-bit integer that is the number of waveform buffers required to read the data.

Points A 32-bit integer that is the number of waveform points in the data.

Count A 32-bit integer that is the number of hits at each time bucket in the waveform record when the waveform was created using an acquisition mode like averaging. For example, when averaging, a count of four would mean every waveform data point in the waveform record has been averaged at least four times. The default value is 0.

X Display Range A 32-bit float that is the X-axis duration of the waveform that is displayed. For time domain waveforms, it is the duration of time across the display. If the value is zero then no data has been acquired.

X Display Origin A 64-bit double that is the X-axis value at the left edge of the display. For time domain waveforms, it is the time at the start of the display. This value is treated as a double precision 64-bit floating point number. If the value is zero then no data has been acquired.

X Increment A 64-bit double that is the duration between data points on the X axis. For time domain waveforms, this is the time between points. If the value is zero then no data has been acquired.

X Origin A 64-bit double that is the X-axis value of the first data point in the data record. For time domain waveforms, it is the time of the first point. This value is treated as a double precision 64-bit floating point number. If the value is zero then no data has been acquired.

X Units A 32-bit integer that identifies the unit of measure for X values in the acquired data:

- 0 = Unknown.
- 1 = Volts.
- 2 = Seconds.
- 3 = Constant.
- 4 = Amps.
- 5 = dB.
- 6 = Hz.

Y Units A 32-bit integer that identifies the unit of measure for Y values in the acquired data. The possible values are listed above under "X Units".

Date A16-bit character array, left blank in the 5000A Series oscilloscope.

Time A16-bit character array, left blank in the 5000A Series oscilloscope.

Frame A 24 byte character array that is the model number and serial number of the oscilloscope in the format of: MODEL#:SERIAL#.

Waveform Label A 16 byte character array that contains the label assigned to the waveform.

Time Tags A 64-bit double, not used in the 5000A Series oscilloscope.

Segment Index A 32-bit unsigned integer, not used in the 5000A Series oscilloscope.

Waveform Data Header

A waveform may have more than one data set. Each waveform data set will have a waveform data header. The waveform data header consists of information about the waveform data set. This header is stored immediately before the data set.

Waveform Data Header Size A 32-bit integer that is the size of the waveform data header.

Buffer Type A 16-bit short that is the type of waveform data stored in the file:

- 0 = Unknown data.
- 1 = Normal 32-bit float data.
- 2 = Maximum float data.
- 3 = Minimum float data.
- 4 = Not used in 5000A Series oscilloscopes.
- 5 = Not used in 5000A Series oscilloscopes.
- 6 = Not used in 5000A Series oscilloscopes.

Bytes Per Point A 16-bit short that is the number of bytes per data point.

Buffer Size A 32-bit integer that is the size of the buffer required to hold the data points.

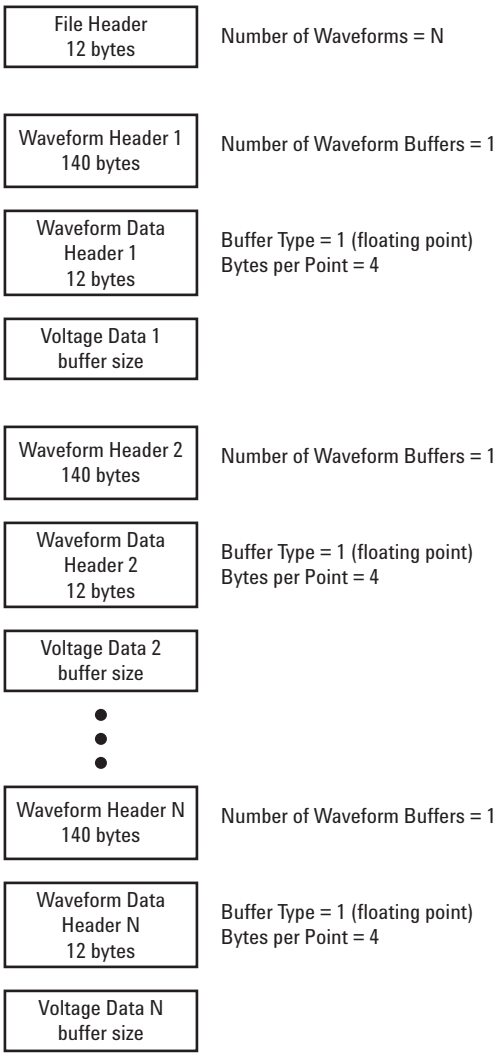
Example Program for Reading Binary Data

To find an example program for reading binary data, direct your web browser to www.agilent.com/find/dso5000, select Technical Support, then select Drivers & Software. Then select “Example Program for Reading Binary Data”.

Examples of Binary Files

Single Acquisition Multiple Channels

The following picture shows a binary file of a single acquisition with multiple oscilloscope channels.



8 Characteristics and Specifications

This chapter lists specifications, characteristics, environmental conditions, and measurement category for the Agilent 5000 Series oscilloscopes.

Environmental Conditions

Overvoltage Category

This product is intended to be powered by MAINS that comply to Overvoltage Category II, which is typical of cord-and-plug connected equipment.

Pollution Degree

The 5000A Series Oscilloscope may be operated in environments of Pollution Degree 2 (or Pollution Degree 1).

Pollution Degree Definitions

Pollution Degree 1: No pollution or only dry, non-conductive pollution occurs. The pollution has no influence. Example: A clean room or climate controlled office environment.

Pollution Degree 2. Normally only dry non-conductive pollution occurs. Occasionally a temporary conductivity caused by condensation may occur. Example: General indoor environment.

Pollution Degree 3: Conductive pollution occurs, or dry, non-conductive pollution occurs which becomes conductive due to condensation which is expected. Example: Sheltered outdoor environment.

Measurement Category

Measurement Category

The 5000A Series oscilloscope is intended to be used for measurements in Measurement Category I.

Measurement Category Definitions

Measurement category I is for measurements performed on circuits not directly connected to MAINS. Examples are measurements on circuits not derived from MAINS, and specially protected (internal) MAINS derived circuits. In the latter case, transient stresses are variable; for that reason, the transient withstand capability of the equipment is made known to the user.

Measurement category II is for measurements performed on circuits directly connected to the low voltage installation. Examples are measurements on household appliances, portable tools and similar equipment.

Measurement category III is for measurements performed in the building installation. Examples are measurements on distribution boards, circuit-breakers, wiring, including cables, bus-bars, junction boxes, switches, socket-outlets in the fixed installation, and equipment for industrial use and some other equipment, for example, stationary motors with permanent connection to the fixed installation.

Measurement category IV is for measurements performed at the source of the low-voltage installation. Examples are electricity meters and measurements on primary overcurrent protection devices and ripple control units.

Transient Withstand Capability

CAUTION

Maximum input voltage for analog inputs:

CAT I 300 Vrms, 400 Vpk; transient overvoltage 1.6 kVpk
CAT II 100 Vrms, 400 Vpk
with N2863A 10:1 probe: CAT I 600 V, CAT II 300 V (DC + peak AC)
with 10073C 10:1 probe: CAT I 500 Vpk, CAT II 400 Vpk

CAUTION

Do not exceed 5 Vrms in 50 Ω mode on the 2-channel models. Input protection is enabled in 50 Ω mode, and the 50 Ω load will disconnect if greater than 5 Vrms is detected. However, the input could still be damaged, depending on the time constant of the signal.

CAUTION

The 50 Ω input protection mode only functions when the oscilloscope is powered on.

Specifications

All specification are warranted. Specifications are valid after a 30-minute warm-up period and within ±10° C of last "User Cal" temperature.

Table 12 Warranted specifications

Vertical system: oscilloscope channels	
Bandwidth (–3dB)	DSO501xA: DC to 100 MHz DSO503xA: DC to 300 MHz DSO505xA: DC to 500 MHz
DC vertical gain accuracy	±2.0% full scale
Dual cursor accuracy[1]	±{DC vertical gain accuracy + 0.4% full scale (~1 LSB)} *Example*: for 50 mV signal, oscilloscope set to 10 mV/div (80 mV full scale), 5 mV offset, accuracy = ±{2.0% (80 mV) + 0.4% (80 mV)} = ±1.92 mV
Scope channel triggering	
Sensitivity	<10 mV/div: greater of 1 div or 5mV; ≥10 mV/div: 0.6 div

1 2 mV/div is a magnification of 4 mV/div setting. For vertical accuracy calculations, use full scale of 32 mV for 2 mV/div sensitivity setting.

Characteristics

All characteristics are the typical performance values and are not warranted. Characteristics are valid after a 30-minute warm-up period and within ±10° C of last “User Cal” temperature.

Table 13 Characteristics

Acquisition	
Sample rate	DSO501xA/503xA: 2 GSa/sec each channel DSO505xA: 4 GSa/sec half channel*, 2 GSa/sec each channel
Memory depth	1 Mpts half channel*, 500 kpts each channel
Vertical resolution	8 bits
Peak detection	DSO501xA: 1-ns peak detect DSO503xA: 500-ps peak detect DSO505xA: 250-ps peak detect
Averaging	Selectable from 2, 4, 8, 16, 32, 64 ... to 65536
High resolution mode	Average mode with #avg = 1 12 bits of resolution when ≥10 µs/div, at 4 GSa/s or ≥20 µs/div, at 2 GSa/s
Filter	Sinx/x interpolation (single shot BW = sample rate/4 or bandwidth of oscilloscope, whichever is less) with vectors on and in real-time mode

* Half channel is when only one channel of channel pair 1- 2 is turned on, or only one channel of channel pair 3-4 is turned on.

Vertical system	
Scope channels	DSO5xx2A: Ch 1 and 2 simultaneous acquisition DSO5xx4A: Ch 1, 2, 3 and 4 simultaneous acquisition
AC coupled	DSO501xA: 3.5 Hz to 100 MHz DSO503xA: 3.5 Hz to 300 MHz DSO505xA: 3.5 Hz to 500 MHz
Calculated rise time (= 0.35/bandwidth)	DSO501xA: 3.5 nsec DSO503xA: 1.17 nsec DSO505xA: 700 psec
Single-shot bandwidth	DSO501xA: 100 MHz DSO503xA: 300 MHz DSO505xA: 500 MHz (in half-channel mode, i.e., one channel of channel pair is on)
Range[1]	2 mV/div to 5 V/div (1 MΩ or 50 Ω)
Maximum input 	Maximum input voltage for analog inputs: CAT I 300 Vrms, 400 Vpk; transient overvoltage 1.6 kVpk CAT II 100 Vrms, 400 Vpk with N2863A 10:1 probe: CAT I 600 V, CAT II 300 V (DC + peak AC) with 10073C 10:1 probe: CAT I 500 Vpk, CAT II 400 Vpk 5 Vrms with 50-ohm input
Offset range	±5 V on ranges <10 mV/div; ±20 V on ranges 10 mV/div to 200 mV/div; ±75 V on ranges >200 mV/div
Dynamic range	±8 div
Input impedance	1 MΩ ± 1% || 12 pF or 50 Ω ± 1%, selectable
Coupling	AC, DC
BW limit	25 MHz selectable
Channel-to-channel isolation	DC to max bandwidth >40 dB
Standard probes	DSO501xA: 10:1 N2863A shipped standard for each oscilloscope channel DSO503xA: 10:1 N2863A shipped standard for each oscilloscope channel DSO505xA: 10:1 10073C shipped standard for each oscilloscope channel
Probe ID	Auto probe sense and AutoProbe interface Agilent- and Tektronix-compatible passive probe sense
ESD tolerance	±2 kV
Noise peak-to-peak	DSO501xA: 3% full scale or 2.5 mV, whichever is greater DSO503xA: 3% full scale or 3 mV, whichever is greater DSO505xA: 3% full scale or 3.6 mV, whichever is greater
DC vertical offset accuracy	≤ 200 mV/div: ±0.1 div ±2.0 mV ±0.5% offset value; >200 mV/div: ±0.1 div ±2.0 mV ±1.5% offset value

Vertical system (continued)	
Single cursor accuracy[1]	±{DC vertical gain accuracy + DC vertical offset accuracy + 0.2% full scale (~1/2 LSB)} *Example*: for 50 mV signal, oscilloscope set to 10 mV/div (80 mV full scale), 5 mV offset, accuracy = ±{2.0% (80 mV) + 0.1 (10 mV) + 2.0 mV + 0.5% (5 mV) + 0.2%(80 mV)} = ± 4.785 mV

1 2 mV/div is a magnification of 4 mV/div setting. For vertical accuracy calculations, use full scale of 32 mV for 2 mV/div sensitivity setting.

Horizontal	
Range	DSO501xA: 5 nsec/div to 50 sec/div DSO503xA: 2 nsec/div to 50 sec/div DSO505xA: 1 nsec/div to 50 sec/div
Resolution	2.5 psec
Timebase accuracy	25 ppm (±0.0025%)
Vernier	1-2-5 increments when off, ~25 minor increments between major settings when on
Delay range	Pre-trigger (negative delay): Greater of 1 screen width or 125 µs Post-trigger (positive delay): 1 s - 500 seconds
Channel delta-t accuracy	Same channel: ±0.0025% reading ±0.1% screen width ±20 ps Channel-to-channel: ±0.0025% reading ±0.1% screen width ±40 ps *Same channel example (DSO505xA)*: For signal with pulse width of 10 µs, oscilloscope set to 5 µs/div (50 µs screen width), delta-t accuracy = ±{0.0025% (10 µs) + 0.1% (50 µs) + 20 ps} = 50.27 ns
Modes	Main, delayed, roll, XY
XY	Bandwidth: Max bandwidth Phase error @ 1 MHz: <0.5 degrees Z Blanking: 1.4 V blanks trace (use external trigger on DSO50x2A, channel 4 on DSO50x4A)
Reference positions	Left, center, right

Trigger system		
Sources		DSO5xx2A: Ch 1, 2, line, ext DSO5xx4A: Ch 1, 2, 3, 4, line, ext
Modes		Auto, Normal (triggered), single
Holdoff time		~60 ns to 10 seconds
Trigger jitter		15 ps rms
Selections		Edge, pulse width, pattern, TV, duration
	Edge	Trigger on a rising, falling, or alternating edge of any source
	Pattern	Trigger at the beginning of a pattern of high, low, and don't care levels and/or a rising or falling edge established across any of the channels, but only after a pattern has been established for a minimum of 2 nsec. The channel's high or low level is defined by that channel's trigger level.
	Pulse width	Trigger when a positive- or negative-going pulse is less than, greater than, or within a specified range on any of the source channels. Minimum pulse width setting: 5 ns (DSO501xA) 2 ns (DSO503xA, DSO505xA) Maximum pulse width setting: 10 s
	TV	Trigger using any oscilloscope channel on most analog progressive and interlaced video standards including HDTV/EDTV, NTSC, PAL, PAL-M or SECAM broadcast standards. Select either positive or negative sync pulse polarity. Modes supported include Field 1, Field 2, all fields, all lines, or any line within a field. TV trigger sensitivity: 0.5 division of sync signal. Trigger holdoff time can be adjusted in half field increments.
	Duration	Trigger on a multi-channel pattern whose time duration is less than a value, greater than a value, greater than a time value with a timeout, or inside or outside of a set of time values. Minimum duration setting: 2 ns Maximum duration setting: 10 s
AutoScale		Finds and displays all active channels, sets edge trigger mode on highest-numbered channel, sets vertical sensitivity on channels, time base to display ~1.8 periods. Requires minimum voltage >10 mVpp, 0.5% duty cycle and minimum frequency >50 Hz.

Channel triggering	
Range (internal)	±6 div from center screen
Coupling	AC (~10 Hz), DC, noise reject, HF reject and LF reject (~50 kHz)

External (EXT) triggering	DSO5xx2A	DSO5xx4A
Input impedance	1 MΩ ±1% || 12 pF or 50 Ω	Approx. 1.015 kΩ ±5%
Maximum input ⚠	Maximum input voltage for analog inputs: CAT I 300 Vrms, 400 Vpk; transient overvoltage 1.6 kVpk CAT II 100 Vrms, 400 Vpk with N2863A 10:1 probe: CAT I 600 V, CAT II 300 V (DC + peak AC) with 10073C 10:1 probe: CAT I 500 Vpk, CAT II 400 Vpk 5 Vrms with 50-ohm input	±15 V
Range	DC coupling: trigger level ± 1V and ± 8V	±5 V
Sensitivity	For ± 1V range setting: DC to 100 MHz, 100 mV, >100 MHz to bandwidth of the oscilloscope, 200 mV For ±8 V range setting: DC to 100 MHz, 250 mV; >100 MHz to bandwidth of the oscilloscope, 500 mV	DC to 100 MHz, 500 mV
Coupling	AC (~10 Hz), DC, noise reject, HF reject and LF reject (~50 kHz)	n/a
Probe ID	Auto probe sense and AutoProbe interface Agilent- and Tektronix-compatible passive probe sense	

Display system	
Display	6.3-inch (161 mm) diagonal color TFT LCD
Throughput of oscilloscope channels	Up to 100,000 waveforms/sec in real-time mode
Resolution	XGA – 768 vertical by 1024 horizontal points (screen area); 640 vertical by 1000 horizontal points (waveform area) 256 levels of intensity scale
Controls	Waveform intensity on front panel. Vectors on/off; infinite persistence on/off, 8 x 10 grid with intensity control
Built-in help system	Key-specific help displayed by pressing and holding key or softkey of interest
Real-time clock	Time and date (user adjustable)

Measurement features	
Automatic measurements	Measurements are continuously updated. Cursors track last selected measurement.
Voltage (scope channels only)	Peak-to-peak, maximum, minimum, average, amplitude, top, base, overshoot, preshoot, RMS, standard deviation
Time	Frequency, period, + width, – width and duty cycle on any channel. Rise time, fall time, X at max Y (time at max volts), X at min Y (time at min volts), delay, and phase on oscilloscope channels only.
Counter	Built-in 5-digit frequency counter on any channel. Counts up to the oscilloscope's bandwidth.
Threshold definition	Variable by percent and absolute value; 10%, 50%, 90% default for time measurements
Cursors	Manually or automatically placed readout of Horizontal (X, ΔX, 1/ΔX) and Vertical (Y, ΔY). Additionally oscilloscope channels can be displayed as binary or hex values.
Waveform math	One function of 1-2, 1x2, FFT, differentiate, integrate. Source of FFT, differentiate, integrate: oscilloscope channels 1 or 2, 1-2, 1+2, 1x2.

FFT	
Points	Fixed at 1000 points
Source of FFT	Scope channels 1 or 2 (or 3 or 4 on DSO50x4A only), 1+2, 1-2, 1*2
Window	Rectangular, flattop, hanning
Noise floor	–50 to –90 dB depending on averaging
Amplitude	Display in dBV, dBm at 50 Ω
Frequency resolution	0.05/time per div
Maximum frequency	50/time per div

Storage	
Save/recall	10 setups and traces can be saved and recalled using internal non-volatile memory
Storage type and format	USB 1.1 host ports on front and rear panels Image formats: BMP (8-bit), BMP (24-bit), PNG (24-bit) Data formats: X and Y (time/voltage) values in CSV format, ASCII XY format, BIN format Trace/setup formats: Recalled

I/O	
Standard ports	USB 2.0 high speed device, two USB 1.1 host ports, 10/100-BaseT LAN, IEEE488.2 GPIB, XGA video output
Max transfer rate	IEEE488.2 GPIB: 500 kbytes/sec USB (USBTMC-USB488): 3.5 Mbytes/sec 100 Mbps LAN (TCP/IP): 1 Mbytes/sec
Printer compatibility	Selected HP Deskjet printers

General characteristics	
Physical size	35.4 cm wide x 18.8 cm high x 17.4 cm deep (without handle) 38.5 cm wide x 18.8 cm high x 17.4 cm deep (with handle)
Weight	Net: 4.1 kgs (9 lbs) Shipping: approximately 9 kgs (20 lbs)
Probe comp output	Frequency ~1.2 kHz, Amplitude ~2.5 V
Trigger out	0 to 5 V into open circuit (~23 ns delay) 0 to 2.5 V into 50 Ω
Kensington lock	Connection on rear panel for security

Power requirements	
Line Rating	~Line 120 W max, 96-144 V/48-440 Hz, 192-288 V/48-66 Hz, automatic selection

Environmental characteristics	
Ambient temperature	Operating –10 °C to +55 °C; non-operating –51 °C to +71 °C
Humidity	Operating 95% RH at 40 °C for 24 hr; non-operating 90% RH at 65 °C for 24 hr
Altitude	Operating to 4,570 m (15,000 ft); non-operating to 15,244 m (50,000 ft)
Vibration	Agilent class GP and MIL-PRF-28800F; Class 3 random
Shock	Agilent class GP and MIL-PRF-28800F; (operating 30 g, 1/2 sine, 11-ms duration, 3 shocks/axis along major axis. Total of 18 shocks)
Pollution degree2	Normally only dry non-conductive pollution occurs. Occasionally a temporary conductivity caused by condensation must be expected.
Indoor use	Rated for indoor use only

Other	
Measurement categories	CAT I: Mains isolated CAT II: Line voltage in appliance and to wall outlet
Regulatory information	Safety IEC 61010-1:2001 / EN 61010-1:2001 Canada: CSA C22.2 No. 61010-1:2004 USA: UL 61010-1:2004
Supplementary information	The product herewith complies with the requirements of the Low Voltage Directive 73/23/EEC and the EMC Directive 89/336/EEC, and carries the CE-marking accordingly. The product was tested in a typical configuration with HP/Agilent test systems.

Product specifications, characteristics, and descriptions in this document are subject to change without notice.

WARNING **Use this instrument only for measurements within its specified measurement categories.**

See data sheet for more information. You can find the data sheet online at www.agilent.com/find/dso5000.

Acknowledgements

RealVNC is licensed under the GNU General Public License.
Copyright (C) 2002-2005 RealVNC Ltd. All Rights Reserved.

This is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version.

This software is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details. The license is located on the Programmer's Documentation CD-ROM.

RealVNC source code can be obtained from RealVNC or by contacting Agilent. Agilent will charge for the cost of physically performing the source distribution.

Index

D

E

F

G

H

I

L

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z