Online Help

Tektronix

**TDS5000B Series Oscilloscopes
PHP0237**

Adapted from the TDS5000B Series Oscilloscopes online help Version 1.0
January, 2004

Table of Contents

Getting Started

Controls and Connections

File Menu

Horizontal/Acquisition Menu

MyScope Menu

Utilities Menu

Theory of Operation

Spectral Analysis Overview

Overview Tasks

Tasks

Glossary

Index

Getting Started

Copyright Information

TDS5000B Series Online Help
PHP0237, Version 1.0
January 9, 2004

About the TDS5000B Series Oscilloscope
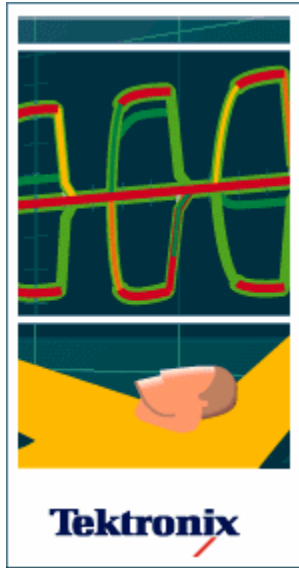
Welcome to the Tektronix TDS5000B Series Oscilloscopes.

The TDS5000B Series of Digital Phosphor Oscilloscopes (DPOs) provide the performance, simplicity, and connectivity needed for today's ever-changing electronics, computer, and communications industries. The easy-to-use interface includes a large display screen and an optional touch-screen interface on a Microsoft Windows platform. Choose the methods of controlling oscilloscope operations that are most convenient for you: customary front-panel controls, menus and toolbar buttons, or the touch screen.

User interface

The user interface is built on the Windows operating system and has a similar look and feel. Switch between menu bar mode and toolbar mode by clicking Buttons or Menu in the upper-right corner of the screen. The instrument is also fully compatible with a wide range of PC hardware and software accessories such as, networked instrumentation and printers.

Optional touch screen interface

If you have the optional touch screen interface, you can access menu items and on-screen controls with the touch of a finger. The TOUCH SCREEN OFF button on the front panel toggles this option. The button is not present if your instrument was not ordered with the touch screen.

Product Description

Models

The online help supports six very similar oscilloscopes:

- TDS5032B Digital Phosphor Oscilloscope
- TDS5034B Digital Phosphor Oscilloscope
- TDS5052B Digital Phosphor Oscilloscope
- TDS5054B Digital Phosphor Oscilloscope
- TDS5104B Digital Phosphor Oscilloscope
- TDS5054BE Digital Phosphor Oscilloscope

Differences between the instruments are noted when necessary.

Key Features

TDS5000B Series oscilloscopes are high-performance solutions for verification, debugging, and characterization of sophisticated electronic designs. The series features exceptional signal acquisition performance, operational simplicity, and open connectivity to the design environment. Classic analog-style controls, a large display with an optional touch-sensitive screen, and graphical menus provide intuitive control. Open access to the Windows operating system enables unprecedented customization and extensibility. Key features include:

- Up to 1 GHz bandwidth and 5 GS/s real time sampling rate
- Record lengths up to 16,000,000 samples
- Fast acquisition at up to 100,000 acquisitions per second
- 1.5% DC vertical gain accuracy
- Two or four input channels (each with 8-bit resolution)
- Auxiliary trigger input and output
- Sample, envelope, peak-detect, high-resolution, average, and waveform database acquisition modes
- Full programmability, with an extensive GPIB command set and a message-based interface
- Extensive suite of advanced triggers including edge, logic, pulse, video, communication (optional), and sequence
- Powerful built-in measurement capability, including histograms, automatic measurements, and measurement statistics
- Basic math as well as advanced equation editor and spectral analysis
- A large 10.4 inch (264.2 mm) color display that supports color grading of waveform data to show sample density
- Customizable MyScope control windows
- An intuitive graphical user interface (UI) with online help
- Internal disk storage
- Wide array of probing solutions, options, and upgrades
- GPIB Controller
- Comprehensive Suite of Right-Click Menus

Acquisition Features

Separate Digitizers. Ensure accurate timing measurements with separate digitizers for each channel. The digitizers can also be combined to yield a higher sample rate on a single channel.

Fast Acquisition. Acquire up to 100,000 waveforms per second to see rapidly changing signals or intermittent signal irregularities.

Long Record Lengths. 2M per channel and 8M on a single channel is standard. Extend the maximum record length up to 16M with memory options.

Peak Detect Acquisition Mode. See pulses as narrow as 400 ps, even at the slower time base settings. Peak detect helps you see noise and glitches in your signal.

Acquisition Control Acquire continuously or set up to capture single shot acquisitions. Enable or disable optional acquisition features such as equivalent time or roll mode.

Horizontal Delay. Use delay when you want to acquire a signal at a significant time interval after the trigger point. Toggle delay on and off to quickly compare the signal at two different points in time.

Average, Envelope, Hi Res Acquisition, and WfmDB. Use Average acquisition mode to remove uncorrelated noise from your signal. Use Envelope to capture and display the maximum variation of the signal. Use Hi Res to increase vertical resolution for lower bandwidth signals. WfmDB mode is the only mode that can be used when mask testing with an optical standard.

Signal Processing Features

Waveform Math. Set up simple math waveforms using the basic arithmetic functions including FFT, or create more advanced math waveforms using the math expression editor. Waveform expressions can even contain measurement results and other math waveforms.

Spectral Analysis. Display spectral magnitude and phase waveforms based on your time-domain acquisitions. Control the instrument using the traditional spectrum analyzer controls such as span, center frequency, and resolution bandwidth.

FastFrame. Use FastFrame to capture a series of triggered acquisitions at high resolution without wasting acquisition memory on the dead time in-between the acquisitions.

Mask Testing. Use mask testing for signal violation detection and pass/fail testing. Test signals against a user defined mask, or with Option SM you can test against industry standard masks.

Limit Testing. Use Limit Testing for pass/fail comparison with an active signal.

Display Features

Color LCD Display. Identify and differentiate waveforms easily with color coding. Waveforms, readouts, and inputs are color matched to increase productivity and reduce operating errors.

Digital Phosphor. A Digital Phosphor Oscilloscope can clearly display intensity modulation in your signals. The oscilloscope automatically overlays subsequent acquisitions and then decays them to simulate the writing and decay of the phosphor in an analog oscilloscope CRT (cathode-ray tube). The feature results in an intensity-graded or color-graded waveform display that shows the information in the intensity modulation.

Fit to Screen. The Digital Phosphor technology performs the compression required to represent all record points on the screen, even at the maximum record length settings.

MultiView Zoom. To take advantage of the full resolution of the instrument, you can zoom in on a waveform to see fine details. Both vertical and horizontal zoom are available. You can also zoom in on multiple areas of a single waveform simultaneously and on multiple waveforms at the same time.

Measurement Features

Cursors. Use cursors to take simple voltage, time, and frequency measurements.

Automatic Measurements. Choose from a large palette of amplitude, time, histogram, and communication measurements. You can customize the measurements by changing reference levels or by adding measurement gating. Statistics (mean, maximum, minimum, standard deviation and population) are available for each measurement value

Trigger Features

Simple and Advanced Trigger Types. Choose a simple edge trigger, or choose from up to twelve advanced trigger types to help you capture a specific signal fault or event.

Dual Triggers. Use the A (main) trigger system alone or add the B trigger to capture more complex events. You can use the A and B triggers together to set up a delay-by-time or delay-by-events trigger condition.

Convenience Features

Autoset. Use Autoset to quickly set up the vertical, horizontal, and trigger controls for a usable display.

Myscope. Allows you to create your own control windows for easy access to the controls you use most often.

Touch Screen Interface. (Optional) You can operate all instrument functions (except the power switch) from the touch screen interface. You can also install a mouse and keyboard to use the interface.

Toolbar or Menu Bar. You can choose a toolbar operating mode that is optimized for use with the touch screen, or a PC-style menu-bar operating mode that is optimized for use with a mouse.

Open Desktop. The instrument is built on a Microsoft Windows software platform; the instrument application program starts automatically when you apply power to the instrument. You can minimize the instrument application and take full advantage of the built-in PC to run other applications. Moving waveform images and data into other applications is as simple as a copy/paste operation.

Dedicated Front Panel Controls. The front panel contains knobs and buttons to provide immediate access to the most common instrument controls. Separate vertical controls are provided for each channel. The same functions are also available through the screen interface.

Data Storage and I/O. The instrument has an optional removable hard disk drive, a floppy disk drive, and a CD-RW drive that can be used for storage and retrieval of data. The instrument has GPIB, USB, Centronics, RS232, VGA out, extended desktop support, and Ethernet ports for input and output to other devices.

Integrated Thermal Printer (Optional). The optional built-in thermal printer allows you to print a long continuous waveform as a banner. Banner printing shows more detail, drawing samples that might otherwise overlap on the screen.

Online Help. The instrument has a complete online help system that covers all its features. The help system is context sensitive; help for the displayed control window is automatically shown if you click the help button. Graphical aids in the help windows assist you in getting to the information you need. You can also access the help topics through a table of contents, index, or full text search.

Product Software

The instrument includes the following software:

- **System Software**. The product software runs on a specially configured version of Windows 2000. Windows 2000 is preinstalled and enables you to install other compatible applications. To reinstall Windows 2000, use the Operating System Restore CD-ROM, a standard accessory with your instrument. Do not attempt to substitute any version of Windows that is not specifically provided by Tektronix for use with your instrument.

- **TDS5000B Product Software.** This preinstalled software is the instrument application and runs on Windows 2000. It provides the user interface (UI) and all other instrument control functions. You can minimize or even exit/restart the instrument application as your needs dictate. However, it is recommended that you exit the instrument application only when installing new software.

- **Support Software**. The TDS5000B Series Product Software CD-ROM, included with the instrument, contains additional software and files that may be useful to you.

- **Readme file**. This text file contains release notes and updates that are not included in other product documentation.

- **GPIB Programmer Online Help software**. This software is available as WinHelp or as a PDF file. It contains the information you need to program the instrument through its GPIB interface.

- **Specifications Performance Verification Procedures**. The product CD-ROM contains instrument specifications and instructions to perform a manual performance verification.

See the instructions for the TDS5000B Series Product Software CD-ROM for information about installing the support software.

Occasionally new versions of software for your instrument may become available at our Web site, see Support Information.

Firmware Upgrade

Tektronix may offer firmware upgrade kits for the instrument. Contact your Tektronix service representative for more information.

User Interface Map

See the figure below for a description of the elements of the user interface. These elements provide complete control of the instrument.
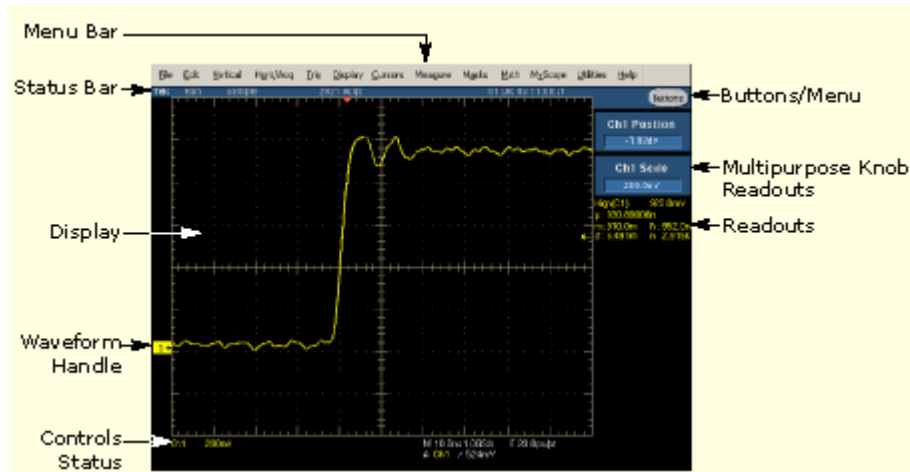
Installing Software

The instrument system and application software is preinstalled at the factory. If you have to reinstall the software for any reason, refer to the instructions that accompany the CD-ROMs shipped with the instrument.

Software Release notes

Read the software release notes (README.TXT) on the product software CD-ROM before performing installation procedures. This file contains additional information that supercedes other product documentation.

To view README.TXT, start the Windows Notepad accessory and open the file from the *TDS5000B Series Product Software* CD-ROM.

Accessory Software

The *TDS5000B Series Product Software* CD-ROM contains accessory software and files that you can install on the instrument or another PC. Refer to the instructions that accompany the CD-ROM for installation information.

Desktop Applications

You can install desktop application software on the instrument. The instrument has been tested with the following software products installed:

- Microsoft Office 2000 (Word, Excel, PowerPoint, etc.)
- MathCad
- MATLAB

Other software products may be compatible but have not been tested by Tektronix. If the instrument malfunctions after you install software, uninstall the software and then reinstall the instrument application to restore proper operation.

Before installing other desktop applications, exit the instrument application. To do this, click Exit on the File menu.

Using Online Help
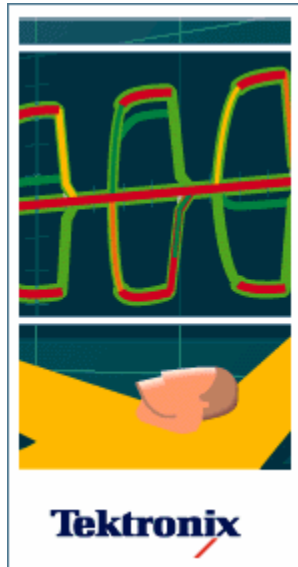
The online help is designed to assist you with all the features of the instrument.

Use the online help to quickly get information about a function or assistance in performing a task. These types of online help are available:

Context-Sensitive help. In toolbar mode, click Help to display online help for the active window. From the menu bar, click Help to display a list of help options. In either mode, the F1 function key also displays help for the active window.

Note

If you are using the optional touch screen, you can also touch the Help button in either mode.

Microsoft Windows help. Help is also available as part of Windows. For information about Windows features, refer to the Windows online help by clicking Start in the Windows toolbar and then selecting Help.

Navigating the Online Help

You can search for help using methods that most users of PCs are familiar with: From the menu bar, select Help, and then select Contents and Index.

From the online help finder, choose one of the three tabs: Contents, Index, or Find. From the Contents tab, click a topic to select it for display, and then click the Display button to open the topic. From the Index tab, when you enter the subject you are looking for, the list scrolls to that topic. Click Display to open the topic. For a full text search use the Find tab. You must first choose the method for word list generation, click Next, and then click Finish. Enter the subject you want to find. Every topic that contains the subject will be displayed. Select a topic and click Display to open that topic.

There are several ways to use the online help that are unique to the instrument:

Jumps. Blue-underlined text represents hyperlinks to other topics. Click the text to jump to the related topic. For example, to jump to the topic on Using Online Help click the text. Click Back to return to this window.

Pop-ups. Pop-up windows are identified by blue text with no underlines. Click the blue text to display a brief help message.

Help on Graphics. Some of the graphics in the help windows contain areas highlighted with orange boxes. Click the area inside the box to either open a pop-up window with a brief help description or to jump to a related help topic.

Note

If you are using the optional touch screen, you can also touch the blue text or orange highlighted graphic areas.

Documentation

In addition to this online help, these documents are available:
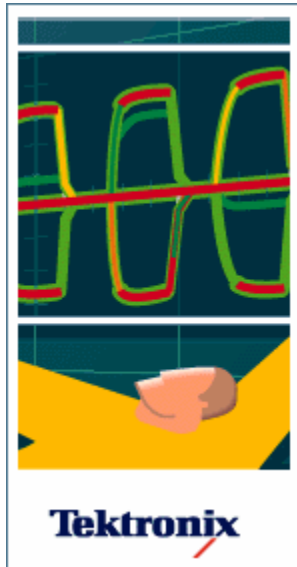
TDS5000B Series Digital Phosphor Oscilloscopes Quick Start User Manual (071-1355-xx). The Quick Start User Manual has information about installing and operating your instrument. The Quick Start User Manual is available in seven languages in addition to English.

TDS5000B Series Digital Phosphor Oscilloscopes Programmer Online Guide. The Programmer Guide is provided as online help and as a printable PDF file. See the *TDS5000B Series Product Software* CD-ROM for installation information.

Optional TDS5000B Series Digital Phosphor Oscilloscopes Service Manual (071-1362-xx). A service manual is available as an optional accessory; it includes procedures to service the instrument to the module level.

Getting Started with OpenChoice Solutions Manual with CD (020-2513-xx). A book and CD that explore some options for getting data from your instrument into any one of several available analysis tools.

TDS5000B Series Digital Phosphor Oscilloscopes Specifications and Performance Verification Technical Reference Manual (071-1420-xx). This is a .pdf only manual and includes both specifications and the performance verification procedure. Complete specifications are also available from the Help menu.

Optional Applications. *The Optional Applications Software for Tektronix Windows-Based TDS Instruments* CD-ROM (020-2450-xx) contains free five-time trials of optional application software that you can install on your instrument. These applications provide application specific measurement solutions, such as power, jitter, optical storage, disk drive, and others.

Other Documentation. Your instrument also comes with booklets for the Product CD-ROM (063-3692-xx) and System Restore CD-ROM (063-3693-xx).

Tip For quick access to PDFs of the user manual, probe manuals, programmer guide, primers and applications, specifications and performance verification manual, service and installation

manuals, and a printable version of the online help, click the Documentation icon on your desktop.

Support Information

For information on getting support for your Tektronix product:

Product Support

For application-oriented questions about a Tektronix measurement product, call toll free in North America:
1-800-833-9200
6:00 a.m. - 5:00 p.m. Pacific time

For product support outside of North America, contact your local Tektronix distributor or sales office.

Service Support

Tektronix offers extended warranty and calibration programs as options on many products. Contact your local Tektronix distributor or sales office.

http://www.tektronix.com

For other information:

In North America:
1-800-833-9200
An operator will direct your call.

To write us:

Tektronix, Inc.
Department or name (if known)
14200 SW Karl Braun Drive
P.O. Box 500
Beaverton, OR 97077
USA

To e-mail us:

techsupport@tektronix.com

Feedback
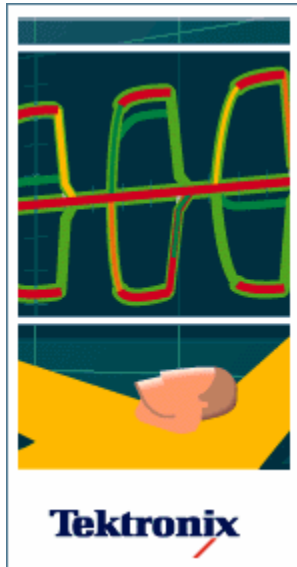
Tektronix values your feedback on our products. To help us serve you better, please send us any suggestions, ideas, or other comments you may have regarding your instrument.

Direct your feedback to us via email to support@tektronix.com, or FAX at (503) 627-5695, and include the following information. In the subject field, please indicate NEW and IMPROVED Scope Customer Feedback. Please be as specific as possible.

Recommended Information:

- Instrument hardware (for example, display, chassis)
- Application software version
- Probes

Optional Information:

Your name, company, mailing address, phone number, FAX number

Please indicate if you would like to be contacted by Tektronix regarding your suggestion or comments.

Tips

To include screen shots of the user interface, select Copy from the Edit menu to copy the image to the clipboard. Paste the image into Microsoft Paint and save the image as a .BMP file. You can then attach the file to your email (depending on the capabilities of your email editor).

Standard Accessories

The following accessories are shipped with your instrument:

- TDS5000B Series Digital Phosphor Oscilloscopes Quick Start User Manual

  English (Option L0, 071-1355-xx)
  French (Option L1, 071-1357-xx)
  German (Option L3, 071-1358-xx)
  Japanese (Option L5, 071-1356-xx)
  Korean (Option L9, 071-1359-xx)
  Russian (Option L10, 071-1401-xx)
  Simplified Chinese (Option L7, 071-1360-xx)
  Traditional Chinese (Option L8, 071-1361-xx)

- Accessory Pouch (016-1935-00)

- Getting Started with OpenChoice Solutions Manual with CD (020-2513-xx)

- Important Documents Folder

  Includes Certificate of Calibration documenting NIST traceability, 2540-1 compliance, and ISO9001 registration

- Front Cover (200-4651-00)

- Power Cord

  - North America (Option A0, 161-0104-00)

  - Universal Euro (Option A1, 161-0104-06)

  - United Kingdom (Option A2, 161-0104-07)

  - Australia (Option A3, 161-0104-05)

  - 240V North America (Option A4, 161-0104-08)

  - Switzerland (Option A5, 161-0167-00)

  - Japan (Option A6, 161-A005-00)

  - China (Option A10, 161-306-00)

  - No power cord or AC adapter (Option A99)

- Optical Wheel Mouse (119-6936-xx)

- TDS5000B Series Product Software CD-ROM (063-3692-xx)

- Includes: application, online help, TDS5000 Series Programmer Online Guide (GPIB online help and PDF), Specifications and Performance Verification PDF, release notes, VISA information, and other related documentation.

- TDS5000B Series Operating System Restore CD-ROM (063-3693-xx)

- Optional Applications Software for Windows-Based Oscilloscopes CD-ROM and manuals (020-2450-xx)

- LabVIEW 30-day Evaluation Copy (020-2476-xx)

- MATLAB 30-day Evaluation Copy (063-3609-xx)

- One 500 MHz, 10x passive probe per channel (P5050)

Recommended Accessories

The following optional accessories are recommended for your instrument:

- TDS5000B Series Digital Phosphor Oscilloscopes Service Manual (071-1362-xx)

- Keyboard (118-9402-00)

- Transit Case (016-1937-00)

- Video Display Clamp (013-0278-xx)

- Thermal Printer Paper (016-1897-00)

- Probe Calibration, Compensation and Deskew Adapter (067-0405-02). Requires a separate pulse generator.

- Power Deskew Fixture (067-1478-00)

- Extra Front Removable Hard Drive (065-0692-xx)

- P6101B 15 MHz 1X Passive Probe

- Extra P5050 500 MHz 10X Passive Probe

- TCP202 Current Probe

- CT1, CT2, and CT6 High Speed AC Current Probes

- CT-4 AC Current Probe 20 kA

- TCPA300, TCPA400 Series Current Measurement Systems (requires TCPA300 / TCPA400 style probe)

- AM5030S DC/AC current measurement system

- P6021, P6022 AC current measurement probes

- P6701B optical/electrical converter (multi-mode)

- P6703B optical/electrical converter (single-mode)

- P5100 High Voltage Probe

- P5200 High Voltage Differential Probe

- P5205 High Voltage Differential Probe

- P5210 High Voltage Differential Probe

- P6015A High Voltage Probe

- P6158 3 GHz, 20X Low C Probe

- P6243 1 GHz Active Probe

- P6245 1.5 GHz Active Probe

- P6247 1 GHz Differential Probe

- P6248 1.5 GHz Differential Probe

- P6246 400 MHz Differential Probe

- ADA400A Differential Preamplifier

- AMT75 1 GHz 75 Ω Adapter

- AFTDS Differential Signal Adapter

Additional accessory information

Item	Description
Monitor	If you use a nonstandard monitor, you may need to change the Windows 2000 display settings to achieve the proper resolution for your monitor.
Printer	Connect an external printer to the Centronics parallel port connector directly. If your printer has a DB-25 connector, use the adapter cable that came with your printer. For information on printer usage, see Using Print Screen.
Rackmount	For installation information, refer to the instructions that came with your rackmount kit.
Other	Refer to the Readme file on the TDS5000B Series Product Software CD for possible additional accessory installation information not covered in this manual.

Options

The following options are available for your instrument:

- Touch Screen (18)
- Front panel removable hard drive (FHD)
- Scope Cart (1K)
- Extended Memory 16/8/4M on 1/2/4 channels (3M)
- Thermal Printer—1P (655-4920-00)
- TDS5000 Series Rackmount Kit—1R (016-1946-xx)
- Serial Communications Mask Testing (SM)
- TDSCPM2—ANSI/ITU telecom pulse compliance software (CP2)
- TDSDVD—Optical storage analysis software (DVD)
- TDSET3—Ethernet compliance test software (ET3)
- TDSDDM2—Hard disk drive application software (J2)
- TDSJIT3E—Jitter and timing analysis software essentials (J3E)
- TDSJIT3—Jitter and timing analysis software with Rj/Dj and BER (JT3)
- TDSPWR3—Power measurement software (PW3)
- TDSUSBS—USB2.0 compliance test software only (USB)
- Calibration Service 3 years (C3)
- Calibration Service 5 years (C5)
- Calibration Data Report (D1)
- Calibration Data Report 3 years (with Option C3)
- Calibration Data Report 5 years (with Option C5)
- Repair service 3 years (including warranty) (R3)
- Repair service 5 years (including warranty) (R5)

Windows Interface Guidelines

Because the instrument uses the Microsoft Windows interface, you have open access to the Windows operating system. You can access the Windows desktop to load and run other Windows-based applications such as Microsoft Excel, WordPad, and Paint.

Although you have access to the Windows interface, you should avoid making operating system changes that may cause problems or annoyances while trying to use the instrument. Here are some guidelines:

- The instrument has been tested with Windows 2000 Professional version. The instrument is not intended to operate with any other version of Microsoft Windows than the one provided on the Operating Restore CD.

- If the instrument powers on in Windows Safe Mode, the optional touch screen is inoperative. You may need to use a mouse or keyboard to restore normal operation.

- To keep the start-up time to a minimum, keep the network support to a minimum.

- Be careful when making changes in the Control Panel. Avoid making changes to any controls with which you are unfamiliar.

- Do not change the LCD monitor color resolution (True Color 24 bit) and screen area settings (640 by 480 pixels). If the settings change the video port may not work properly.

- Avoid using antivirus software on the instrument. If you must install it, disable it after use for best instrument performance.

- Some third-party applications may not work properly with the instrument. If your instrument stops working after you install third-party software, try reloading the instrument software from the TDS5000B Series Product Software CD-ROM.

- If you do not have an external monitor attached to the instrument, avoid changing the taskbar properties to "Always on top" and "Auto hide." The Windows taskbar may obscure buttons or readouts on the screen. However, if you do have an external monitor attached and the instrument is set to dual monitor mode, you can drag the taskbar from the instrument display to the external monitor display and use these settings.

- Disable the Windows screen saver to prevent it from turning on while you are taking measurements or viewing the display. If the screen saver comes on while taking measurements, touch the screen (if the touch screen is installed and enabled), move the mouse, or press any key on the attached keyboard.

- Do not delete or change any system fonts. Changing or removing fonts may affect the quality of the display.

- Avoid changing the system Display Properties such as the Background, Appearance, Effects, or Settings. Making such changes can affect the usability of the instrument and the touch screen.

- Avoid making changes to the contents of the Windows folder or to the contents of the TekScope folder.

- Avoid making changes to the BIOS settings. Changing the BIOS settings can affect the overall operation of the instrument.

- Avoid changing the Windows power management system.

If you think that your Windows interface may cause problems with the instrument, contact your local Tektronix support center for assistance.

Analysis and Connectivity Support

This instrument offers several tools that you can install to support data export for use with data-analysis tools.
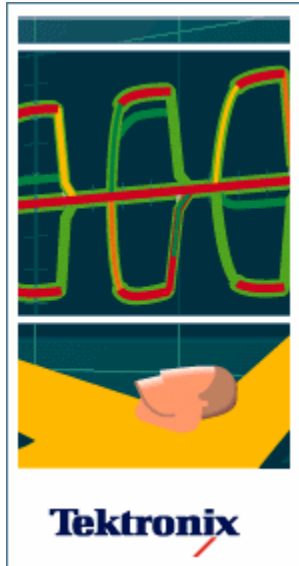
You can use these analysis and connectivity tools with your TDS5000B instrument to:

- Gather instrument data (waveforms, measurements) and port it to familiar, off-the-shelf analysis tools, such as Excel

- Interface with a popular software development environments, such as Lab Windows and LabVIEW

See the following tools to find out how they can help you connect to your instrument and analyze the data it collects: For more information see the *Getting Started with OpenChoice Solutions Manual.*

- VXI Plug-and-Play Instrument Driver

- TekVISA

- TekVISA ActiveX Control (TVC)

- TekVISA Excel Toolbar

- VXI 11.2 LAN Server

- Tektronix Toolbar for Word

- Report Generator

- IVI Drivers

Getting the Analysis and Connectivity Support Tools

This instrument offers several tools that you can install to support data export for use with data-analysis tools.

These tools ship with the Tektronix instrument that this online help supports. You can find them on the product software CD that ships with this instrument. Find the Readme file on the CD for analysis and connectivity tool installation instructions.

For More Information

- For general information and examples on how each of these tools can help you use your instrument, these tools, see the book *Getting Started with OpenChoice Solutions Manual with CD*.

- For online information about the VXIplug&play driver, access the driver online help in from the help system of your software development environment in which you install the driver. The driver online help describes each function of the driver and presents these components in a hierarchical outline corresponding to the standard VXI*plug&play* model.

- For further reference information on TekVISA, read the book *TekVISA Programming Manual.*

- For online information on the TekVISA Excel toolbar, access the online help for the TekVISAExcel Toolbar from the Excel Help menu once you have installed the toolbar.

VXI Plug-and-Play Instrument Driver

The VXI*plug&play* instrument driver is a collection of software components, organized according to the standard VXI*plug&play* model established by the VXIplug&play Systems Alliance. Use this VXI*plug&play* instrument driver to simplify programming tasks when using the following software development environments:

- LabWindows/CVI
- LabVIEW
- MATLAB
- Visual C++ 6.0
- Visual Basic 6.0
- HP-VEE

TekVISA

TekVISA is a library of industry-standard compliant software components, organized according to the standard VISA model established by the VXIplug&play Systems Alliance. Use TekVISA in software to write interoperable instrument drivers to handle communicating between software applications and your TDS5000B series instrument. TekVISA offers the following features and benefits:

- Provides end users with a consistent methodology for using instrument drivers from a variety of vendors
- Provides programmers with language interface libraries for use with multiple Application Development Environments
- Provides a configuration utility for setting up additional VISA resources
- Allows software installation and connectivity on any number of PCs

TekVISA ActiveX Control (TVC)

The TekVISA Control (TVC) uses Microsoft Windows ActiveX control technology to simplify access from Microsoft Office and Visual BASIC applications to TekVISA and the underlying instrument. It does this by providing support for key TekVISA features within an ActiveX control object that is compatible with Excel's Visual BASIC for Applications, Visual BASIC 6.0, and other popular programming environments for Microsoft Windows.

TekVISAExcel Toolbar

The TekVISA Excel Toolbar allows direct, fast transfer of instrument measurements or captured waveforms to an Excel 97 and/or Excel 2000 worksheets. Transfer is via a toolbar that is integrated into Excel, implemented using the TekVISA ActiveX Control (TVC) and the Visual BASIC for Applications macro editor built into Excel.

Source code for the toolbar is accessible through Excel's built-in VBA editor for use as a learning tool for using the TekVISA ActiveX Control, or to cut-and-paste selected components to new VB applications.

VXI 11.2 LAN Server

The VXI 11.2 LAN Server provides software connectivity between your instrument and remote PCs over an Ethernet LAN. This tool is a client-side component built-in with TekVISA on each remote PC, you must install another copy of TekVISA to make use of its client-side component.

The VXI-11 standard specifies a protocol for communication with devices over a network via a network instrument server. This protocol uses the ONC/RPC (Open Network Computing/Remote Procedure Call) standard that, in turn, is based on TCP/IP.

Supported ADEs

TekVISA supports these Application Development Environments:

- Microsoft C/C++
- Microsoft Visual Basic
- LabVIEW graphics software using the G language
- MATLAB analysis software

Tektronix Toolbar for Word

The Tektronix Toolbar for Word, an add-in to Microsoft Word, lets you capture and graph oscilloscope data in Microsoft Word 2000 and 2002 documents.

Report Generator

Report Generator, an MS Windows application, lets you create custom reposts displaying your oscilloscope data. Report Generator enables you to generate and print reports directly from the oscilloscope. It enhances the application capabilities by simplifying the process of creating and maintaining reports.

IVI Drivers

IVI drivers conform to specifications produced by the IVI Foundation. Unlike VXI-*plug&play* drivers, they provide a standard interface to different classes of instruments, including oscilloscopes and spectrum analyzers. IVI drivers have the ability to simulate instruments, automatically check ranges, and multithread safety features.

Menu Bar Mode
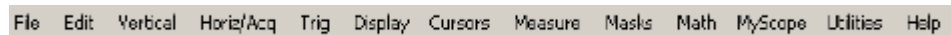
The menu bar menus provide access to commands that control all of the instrument features and functions.

When you select a feature setup command, the instrument displays the associated control window on the lower half of the screen. Each control window provides access to the individual instrument controls. The display graticule is redrawn to fit in the upper half of the screen.

Some of the control windows appear on the right side of the screen. This allows you to operate the instrument in full screen mode while accessing the controls in the control window or from the front panel.

Toolbar Mode

The toolbar buttons give you instant access to most instrument features without having to navigate through several menus. Click a toolbar button to display an associated control window on the lower half of the screen. Each control window provides access to the individual instrument controls. The display graticule is redrawn to fit in the upper half of the screen.

Not all features of the instrument are accessible from the toolbar. To access some of the less common controls, you must change to menu bar mode. To do this, click the Menu button in the upper-right corner of the screen.

Working with the TDS5000B Series Instrument

The following topics describe features of your TDS5000B Series oscilloscope. Even if you are an experienced user, you may want to browse through these topics to learn new ways of doing familiar tasks.

Basic Operations

- Using the Instrument Interface see page 43
- Changing Control Settings see page 43
- Moving and Docking Control Windows see page 44
- Controlling from within the Graticule Area see page 44
- Initializing the Instrument see page 45
- Choosing and Connecting Probes see page 45
- Creating your Own Control Windows see page 46

Waveform Operations

- Using Time Bases and Triggering see page 46
- Looking at Waveform Details see page 46
- Using Fast Acquisition and DPO see page 47
- Using Waveform Math see page 47
- Using Mask and Limit Testing see page 48
- Using Spectral Analysis see page 48
- Characterizing a Signal see page 48
- Adding Application-Specific Measurements see page 49

Using the Instrument Interface
—convenient ways you can operate the instrument

Use familiar knobs and buttons on the front panel to perform most vertical, horizontal, and triggering functions. Make quick adjustments to your displayed signals with the dedicated vertical controls for each channel.

The screen interface gives you access to the complete functionality of the instrument. The toolbar interface is optimized for optional touch screen operation. Touching (or clicking) a button on the toolbar opens a control window that you can use to select the instrument settings.

With a mouse, you may prefer to use the PC-style menu bar to open control windows or directly select many instrument settings.

Changing Control Settings
—several ways to set numerical parameters

As you configure the instrument, you may need to set a numerical parameter such as a reference level or delay time. There are several different ways to set these parameters.

- In a control window, touch or click the parameter to select it. Once selected, one of the multipurpose knobs is assigned to the parameter. Turn the upper or lower knob (as indicated by the knob graphic next to the parameter) to adjust the setting. Push the FINE button to use the knob to make small changes, for example to add another decimal place for greater accuracy.

- This online help usually refers to using the multipurpose knobs to change control settings, although the following methods are generally available:

- Some parameters with discrete values have increment/decrement buttons. Touch or click them to change to the next available value.

- Some parameters supply a pop-up keypad or keyboard that you can use to enter a new value.

- If you have an external keyboard attached, you can also use it to type in a new value.

- You can also adjust parameter values using the mouse wheel. Press the mouse wheel to access fine mode. Press it again to return to normal mode.

Moving and Docking Control Windows

—access control settings while viewing the full screen

Control windows that open on the lower half of the screen can be set to "float" to any location. To do this, double-click the handle on the leftmost edge of the window, or click the handle and drag with the mouse to another position on the screen. The waveform display goes to full-screen mode and the control window changes its format to have a title bar.

The window can now be moved freely to any position on the screen or to a second monitor (if you have set up a dual display) by dragging it by the title bar. Note, however, that it cannot be resized. All controls on the window can be accessed and function as usual.

To dock the control window at its normal position, double-click on the title bar or float the window to its initial boundaries at the bottom half of the screen. The window snaps back into place and returns to its original format.

Control windows that appear on the rightmost side of the screen cannot be moved.

Controlling from within the Graticule Area

—things you can touch or click in the waveform graticule

Use the mouse or your finger (if the optional touch screen is installed) in the graticule area to do the following things:

- Click and drag (or touch and drag) an on-screen cursor to another position on the waveform.

- Click and drag (or touch and drag) a waveform handle to change the vertical position of a waveform.

- Click and drag (or touch and drag) a waveform label or other screen text that you have entered to reposition it.

- Click and drag (or touch and drag) the horizontal reference marker to set the horizontal position.

- Click and drag (or touch and drag) the trigger level marker to set the trigger level.

- Click and drag (or touch and drag) across part of a waveform to select it and investigate it further. Once the selection is made, choose one of the following actions:

  —Create a Zoom waveform that covers just the selected area

  —Create a histogram to analyze the selected area

  —Turn on measurement gates that take measurements only in the selected area

  —Cancel the selection

Access Shortcut Menus by Right-Clicking

—fast access to most common commands is available by right-clicking

Right-click on the graticule, on an object, or a readout for quick access to shortcut menus. These shortcut menus are dynamic and will change with the area or object where you right clicked. Right-click on any of the following for a dynamic shortcut menu:

- Waveform handle
- Cursors
- Labels
- Histogram display
- Masks
- Templates
- Trigger level marker
- Readouts (the menu varies with the type of readout)
- Graticule
- Regions (Menu bar, Toolbar, Multipurpose Knobs)

Initializing the Instrument

—returning to a known starting point

After you complete a task, you may want to initialize the instrument settings before using the instrument with your next task. This may be especially important if you share the instrument with others; previous settings may make it difficult for you to do what you need to do.

To quickly restore the instrument to its default settings, push the front-panel DEFAULT SETUP button.

If data security is important to you, use the Tek Secure Erase feature to overwrite all internal waveforms and set up memory with null values.

Choosing and Connecting Probes

—important things to remember about probes

Choose from a wide variety of passive, active, and specialty probes to use with your instrument. The instrument automatically configures the input impedance when you attach a probe that is compatible with the TekProbe interface. Remember the following when connecting a probe:

- Adjust the low frequency compensation on most passive probes after you connect them to a channel input.
- Compensate for gain (and offset) error in many probes. Connect the probe tip to the probe compensation connectors and execute the built-in calibration procedure to optimize the system accuracy.
- Compensate for propagation delay differences between voltage probes using the probe deskew feature. Attach up to two probe tips at a time to the probe compensation terminals and execute the deskew procedure to time align the display for each channel. The deskew adjustment range is ±75 ns.

Creating Your Own Control Windows

—new ways to make your job easier

For quick access to the controls that you use most often, you can create your own MyScope control window. Choose from a list of controls to include and make each control window unique to your needs. MyScope allows you to have up to eight tabs in a control window, increasing the flexibility and ease of use for your unique MyScope control windows. You can create and save any number of controls windows for future use.

Triggering, Timebase, and Horizontal Delay

—a comparison with oscilloscopes of the past

The architecture of an analog oscilloscope requires its time bases and triggering to be linked. The main trigger triggers the main time base, and then you can use a delayed trigger to trigger a delayed time base. Many digital oscilloscopes have also been designed using this traditional but complex paradigm.

This instrument uses a simpler paradigm that cleanly separates the concepts of triggering and time bases. You can still do everything that you could with an analog oscilloscope and more. Now these features are easier to set up and use.

The trigger event can be simple or complex. Examples of a simple trigger are the rising edge of Channel 1 or a glitch narrower than 10 ns on Channel 2. You can set up a complex trigger where the trigger system is armed by a condition on one channel and then is triggered by a condition on another channel. In either case, simple or complex, there is just one trigger event.

The time base can be either delayed or not delayed. If it is not delayed, acquisition occurs surrounding the trigger event depending on the amount of pretrigger and posttrigger data that you have selected. If you turn on horizontal delay, you can delay the acquisition to a point well beyond the trigger event (similar in effect to a delayed time base). By alternately turning horizontal delay on and off, you can make quick, detailed comparisons between two segments of a signal that are separated from each other by a fixed amount of time. In any case, the time base is always triggered by one trigger event.

Looking at Waveform Details

—taking advantage of the full resolution of the instrument

Use the instrument MultiView Zoom function to magnify an acquisition vertically, horizontally, or in both dimensions to let you see the fine detail in your signals. The MultiView Zoom function affects only the display, not the actual waveform that is being acquired.

The fastest way to set up a Zoom display is to click (or touch) and drag across the segment of the waveform that you want to see in greater detail. Then, when you touch or click the on-screen Zoom menu item, that waveform appears in the Zoom (lower) graticule, while the entire waveform appears in the Acquisition (upper) graticule. Markers in the main graticule give you position context for what you are seeing in the Zoom graticule.

Another way to set up the MultiView Zoom function is to push the front-panel MultiZoom button. Then you can set parameters in a control window to focus your Zoom waveform on the signal segment of interest.

To see more waveform detail you can also use horizontal delay (as described in the previous topic) to focus the acquisition on a segment of a waveform separated from the trigger event by a

significant interval of time. This technique lets you increase the horizontal resolution (sample faster) on a segment of a signal, even when you cannot trigger on that particular segment.

The multizoom function also allows you to zoom on multiple areas of a waveform or on multiple waveforms simultaneously. You can lock the zoomed areas and scroll through them together or unlock the zoomed areas and scroll through them independently.

Using Fast Acquisition and DPO

—analog performance from digital technology

When you turn on FastAcq, you can acquire up to 100,000 waveforms per second. In fast acquisition mode, the "dead time" between acquisitions can be very short, approaching that of the finest analog oscilloscopes. Reduced dead time means that your chances of capturing an infrequent event are greatly improved. More time is spent acquiring and displaying your signal. And with the digital phosphor (DPO) technology, you can easily differentiate between the normal shape of your signal and that infrequent anomaly that you want to capture.

As a comparison, high quality analog real-time oscilloscopes can sweep at high repetition rates with little dead time between sweeps, which makes it possible to capture an infrequent event. Typical digital oscilloscopes (DSOs) have relatively long dead times between acquisitions; infrequent events are much more difficult to capture. With fast acquisition and digital phosphor technology, the oscilloscope acquires at a high repetition rate and then overlays the acquired information into a three-dimensional database that is updated on the display 30 times per second.

The digital phosphor oscilloscope shows you not only the displayed waveform, but also the sample density. You can choose to see sample density represented in a color palette or as shades of gray or shades of green. The DPO display in monochrome green bears a strong resemblance to the CRT of a high-quality analog oscilloscope. When you increase waveform brightness with the front-panel INTENSITY knob, areas of lower sample density appear more clearly.

DPO provides display benefits whether fast acquisition is on or off. When it is on, DPO helps you identify an infrequent anomaly in a repetitive signal, especially when either the spectral or temporal color palettes are used. The repetitive signal appears in one color and the anomalies appear in another. When fast acquisition is off, DPO improves the waveform display quality when long records are compressed for display with the 500 horizontal pixels in the graticule. The compression algorithm works with the intensity-graded display to show brighter pixels where many points have been compressed and dimmer pixels where fewer points have been compressed.

Using Waveform Math

—almost any live calculation that you can imagine

Waveform math in the instrument can be as simple as basic arithmetic or as powerful as building a complicated math expression.

With the Equation Editor, you can build a complicated math waveform expression using multiple waveform sources, constants, variables, operators, and functions. The waveform sources can be any channel, reference, or math waveform. You can enter constants with a keypad. Live results from the measurement system can be used as variables, and there is a full assortment of arithmetic operators and math functions to use. As you enter the waveform expression, you will see it appear as an equation in the control window. You can define up to four separate math waveforms (or two, if using a 2-channel model).

Using Spectral Analysis

—operate the instrument like a spectrum analyzer.

When you are displaying an FFT (spectral) math waveform, you can use either of these two ways to operate the instrument:

▪ Use the conventional time-domain instrument controls, such as vertical and horizontal position and scale.

▪ Use the spectrum analyzer controls, such as center frequency, span, resolution bandwidth, and reference level.

When you use the spectrum analyzer control set, the instrument automatically sets its time domain parameters, including record length and sample rate, as necessary to give you the view of the spectral waveform that you want to see. And you can switch back to using the conventional time-domain controls at any time.

Spectral waveforms can be simple magnitude or phase waveforms calculated from the data acquired on a single channel. Or you can combine spectral operations with other math operations to do complex analysis of your signals.

Characterizing a Signal

—deriving statistical information from your signals

Signal characterization begins with taking measurements. To gain even more insight into your measurements, you can add measurement statistics to calculate the average measurement value, the standard deviation, and the maximum range of the measurement value.

Another way to characterize a signal is to compile a histogram from the acquired waveform. You can set up either a vertical histogram to analyze amplitude values or a horizontal histogram to analyze timing. As an example, the vertical histogram is useful for characterizing noise and the horizontal histogram is useful for characterizing jitter.

While the histogram is being compiled from the acquired waveform, you can take continuous measurements on the histogram data to determine its mean value and other useful statistics.

Mask and Limit Testing

—applying masks and templates to waveforms for compliance testing

Mask testing is used with communication signals. You can create your own user mask or (with Option SM) use industry standard masks for compliance testing.

Limit testing allows you to compare the performance of products coming off a manufacturing line with a known good reference product or template.

You can create any number of User masks and templates for future use, however, you can only use one mask at a time and only on one channel at a time. Limit testing can be done on all four channels at the same time.

Application-Specific Measurements

—extending your measurement capability

You can load optional measurement extensions as needed to enhance the capabilities of the instrument for specific applications. These applications build on the precision signal acquisition performance of the instrument to address your specific design requirements.

Here are some examples of the application packages:

- Jitter and Timing Analysis—Use the TDSJIT3 or TDSJIT3E software to characterize the timing performance of semiconductor systems. Analyze clock jitter, setup and hold times, and many other timing measurements on a cycle-by-cycle basis over long, single-shot acquisitions. Measurement statistics are presented as numerical statistics, or graphically in a histogram, cycle trend, time trend, or spectral plot.

- Disk Drive Measurements—Use the TDSDDM2 software to measure disk drive signals according to IDEMA standards.

- Optical Storage Analysis and Measurements—Use the TDSDVD software for automatic amplitude and timing measurements.

- Ethernet Compliance Testing—Use the TDSET3 software to perform 10/100/1000 Base T Ethernet compliance testing.

- USB2 Signals—Use the TDSUSB2 software to characterize USB2 signals including mask testing and parametric testing.

- Mask and Measurement Compliance—Use the TDSCPM2 software for mask and measurements compliance testing for ITU-T G.703 and ANSI T1.102 communications standards.

- Power Measurements—Us the TDSPWR3 software to quickly measure and analyze power dissipation in power supply switching devices and magnetic components.

Additional packages may be available. Contact your Tektronix representative for more information.

Connecting to a Network

—as easy as networking a PC

The instrument is built on a PC-based platform that runs Microsoft Windows as its operating system. Like any other Windows computer, you can connect the instrument to a network to enable printing, file sharing, Internet access, and other communications functions.

To make a network connection, perform the steps to enable network access. Consult your network administrator for assistance.

Printing Hard Copy
—flexible printing options

To make hard copies, use the optional built-in thermal printer, a printer attached to one of the rear-panel ports, or connect to a network printer if your instrument is on a network. You can print the full screen, just the graticule area, or a long continuous waveform as a banner. Banner printing shows more detail, drawing samples that might otherwise overlap on the screen. Banners can only be printed on the integrated thermal printer.

The instrument uses Windows print drivers, so the process to install printers is the same as it is on any other PC.

The Page Setup dialog box contains some useful options to optimize your printing:

- Choose a print palette that is independent from the display palette. For example, you can print grayscale images to a black and white laser printer while you choose to view the display in color. The color and grayscale palettes both render the intensity-graded waveform information generated by the DPO display.

- Choose the Ink Saver feature to print a hard copy with a white background. This function saves printer ink while it preserves the color coding of the waveforms and readouts. Ink Saver also works with the monochrome print palettes.

- Choose to print a Summary Waveform at either or both ends of your banner. This gives you a concise overall view of the instrument display before banner printing commenced.

Using Other PC Applications
—intuitive image and data transfer

Transferring images and data from the oscilloscope to other PC applications is as simple as a copy/paste operation. With a word processing, spreadsheet, or analysis application running on the built-in PC, you can easily transfer these items using the familiar Windows clipboard system:

- Copy a full screen or graticule-only image and paste it into your document in either .BMP, .PNG., PCX, .TIF, or .JPG format.

- Copy a waveform and paste the waveform data into a spreadsheet in a comma-delimited (.CSV) format.

- Copy a waveform and paste the waveform data into an analysis program in Mathcad (.DAT) format.

- Copy the displayed measurements or a snapshot of all measurements and paste them into a document in text (.TXT) format.

- Copy the displayed measurements or a snapshot of all measurements and paste them into a spreadsheet in a comma-delimited (.CSV) format.

Using a Second Monitor

—the convenience of a built-in PC

To optimize your access to the PC that is built into the oscilloscope, connect a keyboard, mouse, and monitor to the side panel of the instrument, and then configure Windows for dual-display mode.

In this configuration, the oscilloscope retains full dynamic oscilloscope performance while Windows and other applications such as publishing, analysis, or Web browsing tools reside on the external monitor. You can copy/paste bitmaps and waveform data from the oscilloscope to the locally running application or view Web-based reference information while using the oscilloscope for design work.

You can also drag up to five dockable HSCWs (half-screen control windows) from the TekScope application onto the display of the second monitor. For example, this feature allows you to have the Horizontal/Acquisition, the Horizontal Zoom, and the Display Setup Control Windows (half screen) open on the second monitor at the same time. In addition to the five dockable windows on the second monitor, you can have one dockable window open on the main display. If you open a seventh dockable window, the window that has not been used for the longest amount of time will automatically close.

To drag an HSCW window, click and drag on the left edge of the window.

Connecting to Other Instruments

—useful connections to other instruments

The oscilloscope provides four signal inputs and outputs that you can connect to other instrumentation:

- Use the rear-panel vertical signal output (SIGNAL OUT) to connect to a buffered version of the signal that is attached to the Channel 3 input (4-channel models only). The typical output amplitude is 50 mV/div 20% into a 1 MΩ load, or 25 mV/div ±20% into a 50 Ω load

- Use the rear-panel auxiliary output (AUX OUT) to obtain a TTL-compatible, negative polarity pulse when the oscilloscope triggers.

- Use the rear-panel external reference input (EXT REF) to synchronize the oscilloscope time base to an external 9.8 $\tilde{~}$ 10.2 MHz reference frequency source. The reference input voltage range is 200 mV p-p to 7V p-p.

- The trigger level range for the front-panel auxiliary input (AUX IN) is adjustable from +8 V to –8 V. The maximum input voltage is ±20 V.

TDS5000B Series Options

You can order the following options with the oscilloscope:

Hardware options

FHD   Front-panel removable hard drive

18   Touch-screen interface

1P   Built-in thermal printer

SM   Serial Communication Mask Testing

Acquisition memory options

3M   16M/8M/4M on 1/2/4 channels

Mounting options

1K   K420 oscilloscope cart with mouse accessory tray

1R   Rackmount kit

Software options

DVD   TDSDVD—Optical storage analysis software

J2   TDSDDM2—Disk drive measurements software

JT3   TDSJIT3—Jitter & timing analysis with Rj/Dj and BER estimation

J3E   TDSJIT3E—Jitter & timing analysis software essentials

PW3   TDSPWR3—Power measurement analysis software

USB   TDSUSBS—USB2.0 compliance test software only

ET3   TDSET3— Ethernet compliance test software

CP2   TDSCPM2—ANSI/ITU telecom pulse compliance software (Requires Option SM)

Service options

D1   Calibration data report

C3   Additional 2 years of calibration

D3   Calibration data report for Option C3

R3   Additional 2 years of repair

C5   Additional 4 years of calibration

D5   Calibration data report for Option C5

R5   Additional 4 years of repair

Local language manual options

L1   French Quick Start User manual

L3   German Quick Start User manual

L5   Japanese Quick Start User manual

L7   Simplified Chinese Quick Start User manual

L8   Traditional Chinese Quick Start User manual

L9   Korean Quick Start User manual

L10   Russian Quick Start User manual

You can order the following upgrade kits to extend the performance of your instrument after its initial purchase. Contact your local Tektronix representative for details and the most current information.

To upgrade your instrument, order the TDS5UP Upgrade Kit and the appropriate option(s) from this list. You must order at least one option.

TDS5UP Upgrade	**Description**
Option M03	Upgrade memory depth from standard to Option 3M
Option J2	Upgrade to TDSDDM2—Disk Drive Measurements software
Option JT3	TDSJIT3 Advanced Jitter Analysis application with random and deterministic jitter analysis
Option J3E	TDSJIT3E Jitter and timing analysis software essentials
Option PW3	Upgrade to Power Analysis Measurements
Option 18	Touch-screen interface
Option SM	Serial Mask Communication Testing
Option 1P	Built-in thermal printer
Option DVD	TDSDVD-Optical storage analysis software
Option USB	TDSUSBS-USB2.0 compliance test software only

TDS5UP Upgrade	Description
Option IF	Service installation of selected options
Option 1K	K420 oscilloscope cart with mouse accessory tray
Option 1R	Rackmount kit
Option CP2	TDSCPM2: ANSI/ITU telecom pulse compliance software
Option ET3	TDSET3 – Ethernet compliance test software

Controls and Connections

Multipurpose Knobs

The multipurpose knobs can be used to set screen interface parameters.

To use, click a control on the screen to map it to the multipurpose knob(s). Note that they are automatically mapped when certain functions are performed, for example, when the MultiView Zoom button is pushed. You can undo any mapping by right-clicking the multipurpose knob(s) and selecting **Deassign Multipurpose Knobs** or from the Utilities menu select the **Deassign Multipurpose Knobs** command. The mapping is deassigned until your perform that function again.

Turn the knob(s) or scroll with the mouse wheel until the desired selection appears in the readout mapped to the knob. Push the FINE button or click the mouse wheel to use the knob to make small changes, for example, to add another decimal place for greater accuracy.

When no control windows are open and cursors are turned on, the multipurpose knobs control the cursor positions.

Vertical Controls

Use the VERTICAL controls to adjust the vertical display parameters of live data channels connected to the vertical inputs. Each input channel has identical controls.
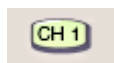
Use the dedicated channel (CH n) buttons to turn channel displays on and off. The channel button is lighted when the selected channel is on.

Use the dedicated POSITION and SCALE knobs to adjust the vertical position and scale of the waveform.

For channel waveforms (waveforms connected to the vertical inputs), the scale setting controls the vertical size of the acquisition window as well as the display scale. The range and resolution of the scale value depends on the attached probes and on any other external factors you have specified.

For reference or math waveforms, the scale setting controls the display only. Scaling these waveforms has no effect on the acquisition hardware.

Push the termination button to toggle the termination between 50 Ω and 1 MΩ. The correct termination is automatically set when you attach a probe with a TekProbe interface to the instrument. However, if you use other than a recommended probe, you may have to set the termination manually. In general, select 1 MΩ to use high impedance passive probes. Select 50 Ω to use most active probes and low impedance (Zo) probes.

Horizontal Controls

Use the HORIZONTAL controls to adjust the horizontal parameters of waveforms. The horizontal controls globally affect all displayed waveforms.

Use the dedicated POSITION knob to adjust the horizontal position of all waveforms. Use this knob to adjust the trigger point so that you can select the amount of pretrigger data or posttrigger data. In Delay Mode, this knob can be used to set the delay time.

Use the dedicated SCALE knob to adjust the horizontal scale (time base) of all channel waveforms. For math and reference waveforms, only the selected waveform is scaled.

Use the RESOLUTION knob to change the number of acquired points (record length) in the waveform, which changes the sample density. The actual resolution cannot be set directly but is derived and displayed on the horizontal readouts.

Push the DELAY button to turn on horizontal delay and then use the POSITION knob to set the delay time. Delay is useful when you want to acquire waveform details that are separated from the trigger event by a significant interval of time.

MultiView Zoom Controls

Push the MultiView Zoom button to open a magnified window or windows in the lower half of the screen.

The MultiView Zoom button is lighted when the Zoom mode is on.

The HORIZ button opens the Zoom control window and assigns the multipurpose knobs to the horizontal position and scale controls in the Zoom graticule.

The VERT button opens the Zoom control window and assigns the multipurpose knobs to the vertical position and scale controls in the Zoom graticule.

Trigger Controls

Use the TRIGGER controls to set basic trigger parameters.

By default, the instrument uses the Edge trigger. To return to the Edge trigger from any other A (main) event trigger, push the EDGE button. Pushing this button does not open the Edge Trigger control window unless another A event trigger is already displayed.

To select other trigger types, push the ADVANCED button. The Trigger Setup control window shows the previously selected, nonedge trigger type. Select one of the advanced trigger types by clicking one of the Trigger Type buttons.
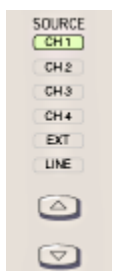
Select the trigger source for the Edge trigger by pushing the up or down arrow buttons until the desired SOURCE is lighted. These buttons are only valid for the A event Edge trigger.

Select the trigger coupling for the Edge trigger by pushing the up or down arrow buttons until the desired COUPLING is lighted. These buttons are only valid for the A event Edge trigger.
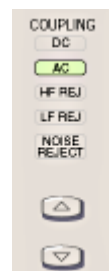
Push the button in the SLOPE column to toggle between positive and negative trigger slopes for the Edge trigger. The selected trigger slope is lighted. These buttons are only valid for the A event Edge trigger.

Push the button under the MODE column to toggle between normal (NORM) and auto (AUTO) modes; the selected trigger mode is lighted.

Push the LEVEL knob to set the trigger level to 50%. To set a different trigger level, rotate the LEVEL knob. The trigger level appears on the screen. The trigger level knob only affects the Edge, Width, Glitch, Timeout, and Video trigger types.

Run Controls

Use the Run controls to start, stop, and to monitor the status of acquisitions.

Push the RUN/STOP button to start and stop acquisitions. Push the RUN/STOP button once to stop the acquisition; the instrument displays the number of acquisitions on the screen. Push the RUN/STOP button a second time to begin acquisitions again.

Push the SINGLE button to run a single sequence of acquisitions. In Sample, Peak Detect, or High Res mode, the instrument acquires a waveform record with the first

trigger event and then stops. In Envelope, Average, or WfmDB mode, the instrument makes the specified number of acquisitions to complete the averaging or enveloping task, and then stops.

The Acquisition Status lights indicate the state of the acquisition:

- When TRIG'D is on, the instrument has recognized a valid trigger and is filling the posttrigger portion of the waveform.

- When READY is on, the instrument can accept a valid trigger event and is waiting for the trigger event to occur.

- When ARM is on, the instrument is filling the pretrigger portion of the waveform record.

- When TRIG'D and READY are both on, the instrument has recognized a valid trigger and is waiting for a delayed trigger. When the instrument recognizes a delayed trigger, it will fill in the posttrigger portion of the waveform.

- When ARM, TRIG'D, and READY are not on, the acquisitions have stopped.

General Purpose Controls

Use the following controls to do general-purpose tasks such as Autoset the instrument, print, and enable cursors.

Push the AUTOSET button to automatically set up the instrument to obtain and display a stable waveform of usable size. For information on the Autoset default settings click Autoset.

Push the DEFAULT SETUP button to reset the instrument to the factory default settings.

Push the PRINT button to make a hard copy of the screen. The printed image is sent directly to the selected (internal or attached) printer. To change the print settings, click the File menu Print command and change the settings in the Print dialog box.

Push the CURSORS button to enable cursors on the screen. The Cursor Type Setup control window opens so that you can specify the cursor parameters.

Push the FastAcq button to turn Fast Acquisition On or Off. The FastAcq button is lighted when Fast Acquisition is turned on.

Use the INTENSITY knob to adjust the brightness of the waveform. High intensity settings show less frequently acquired points more clearly.

Front Panel Connectors

The instrument features these front panel connectors:
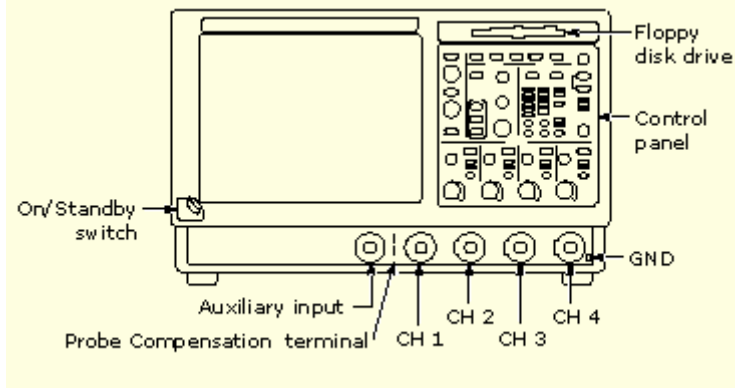
- The trigger level range for the auxiliary input (AUX IN) is adjustable from +8 V to –8 V. The maximum input voltage is ±20 V.

- Use the probe compensation terminals (PROBE COMP) to compensate, calibrate, and deskew passive or active voltage probes.

- Connect up to four probes (two probes for 2-channel models) to input channels (CH n) of the instrument. Each vertical input channel has its own vertical control settings. Each probe input channel has its own TekProbe interface.

- Use the ground (GND) terminal to connect the instrument to the same ground as the unit under test.

Rear Panel Connectors

- The instrument features these rear panel connectors:
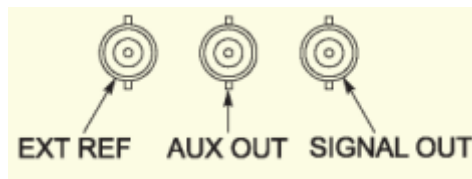
- Use the **EXT REF** connector to synchronize the instrument time base to an external 9.8 – 10.2 MHz reference frequency source. The reference input voltage range is 200 mV$_{\text{p-p}}$ to 7V$_{\text{p-p}}$.

- Use the **AUX OUT** connector to obtain a TTL-compatible, negative polarity pulse when the instrument triggers.

- Use the **SIGNAL OUT** connector to connect to a buffered version of the signal that is attached to the Channel 3 input (4-channel instruments only). The amplitude is 50 mV/div ±20% into a 1 MΩ load, or 25 mV/div ±20% into a 50 Ω load.

Side Panel Connectors

The instrument features these side panel connectors:
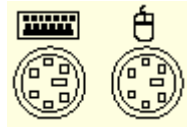
Use the PS-2 connectors to connect a PS-2 keyboard or a mouse to the instrument.
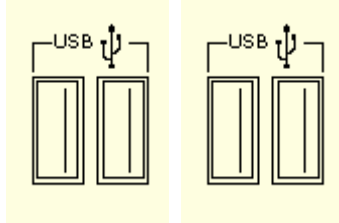
Use the USB connectors to connect a USB mouse, a keyboard, or other USB devices to the instrument.

Note

Plug USB devices directly into the USB connectors on the instrument to improve reliability rather than connecting the devices serially. If the USB mouse locks up, disconnect and reconnect the USB connector to restore normal operation. If the instrument front panel and/ or touch screen do not respond, press the On/Standby switch for 5 seconds to cycle power.
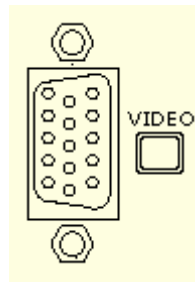
Use the upper VGA port to connect a monitor for extended desktop operation. For example, use the front panel to display the contents of the instrument and use the monitor for Windows operation.
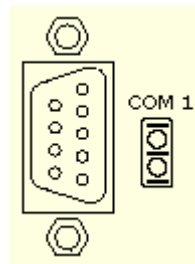
Use the COM1 serial port to connect to other devices through the serial port.
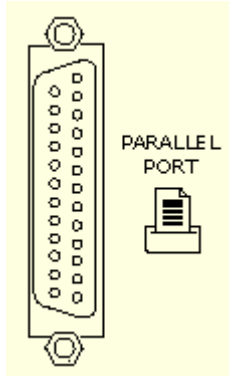
Use the parallel port (Centronics) to connect a printer or other device.
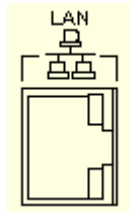
Use the RJ-45 connector to connect the instrument to a network.
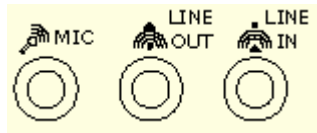
Use the audio ports for stereo microphone input and stereo line output.
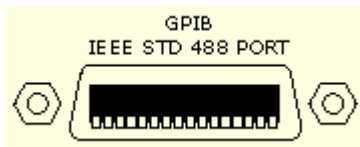
Use the GPIB connector to connect the instrument to a GPIB controller for GPIB operation.
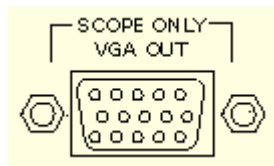
Use the lower VGA port to obtain the instrument display for use with projectors or larger monitors.

The CD-RW drive is accessible from Windows 2000. Press the cover to open the drive.

Optional Touch Screen

If the optional touch screen is installed, you can use touch to control the instrument in addition to the front-panel controls, toolbar buttons, and menu options. Generally, touch can be used anywhere that click is mentioned in this online help.

To disable the touch screen, push the front-panel TOUCH SCREEN OFF button. When the touch screen is off, the button is lighted. You can still access the on-screen menus with a mouse or keyboard. If the button is not present, the touch screen has not been installed on this instrument.

To adjust the touch screen operation to your personal preferences, run the touch screen program from the shortcut on the Windows desktop or select Touch Screen from the Utilities menu.

Note

If the instrument is powered on in Windows Safe Mode, the touch screen is inoperative. You may need to use a mouse or keyboard to restore normal operation.

On/Standby Switch
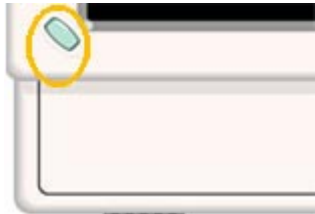
Use this button to power the instrument on or to shut down (set the instrument to standby).

Before shutting down, it is recommended that you first exit the instrument application and any other active Windows applications to avoid error messages on shutdown or the next power up.

Tip To exit the instrument application and shut down in one step, select the File menu **Shutdown** command.

File Menu

Use the File menu for basic file operations such as saving, recalling, and deleting waveforms or setups. You can also use the File menu for standard Windows operations such as printing files and loading the most recent setup.

Reference Waveform Controls

Access this control window from the File menu, Reference Waveform Controls… command.

Overview

Use the Reference Waveform controls to display and control reference waveforms. Reference waveforms are shown with the horizontal settings in effect at the time they are saved. You cannot adjust these settings.

To Use

Select the Ref 1-4 waveform from the list. Click On to display the Ref waveform. Click Position or Scale and use the multipurpose knobs to make adjustments. Enter a label for the waveform in the Label text box. Click Save to access the Save As dialog box and save the waveform as a Reference waveform or save it to a file. Click Recall to access the Recall dialog box and recall a waveform from memory.

Behavior

Clicking Save or Recall activates the respective dialog boxes. Typing a name in the Label text box places the label on the graticule. You can move the label on the graticule by clicking and dragging it. The label is saved with the Reference waveform.

Tip To easily change the vertical position of a reference waveform, click the waveform handle on the left side of the screen, and drag the waveform to the new position. To easily change the vertical scale of a reference waveform, click the waveform handle, and then use the multipurpose knobs to change the waveform scale. You can also use the multipurpose knobs to change the waveform position.

Save As Dialog Box: Waveform

Access this control window from the File menu, Save As control or from the Reference Waveform controls, Save button.

To Use

Use this dialog box to save a waveform to a reference location or to a specified folder. To save to a reference location, select the Source, and then click the reference location in oscilloscope memory.

To save to a file, enter a file name, use the default file name, or select Auto-increment file name if you want to save a series of files without typing in a new name each time. The file extension specified in the Save as type list will automatically be appended to the file name.
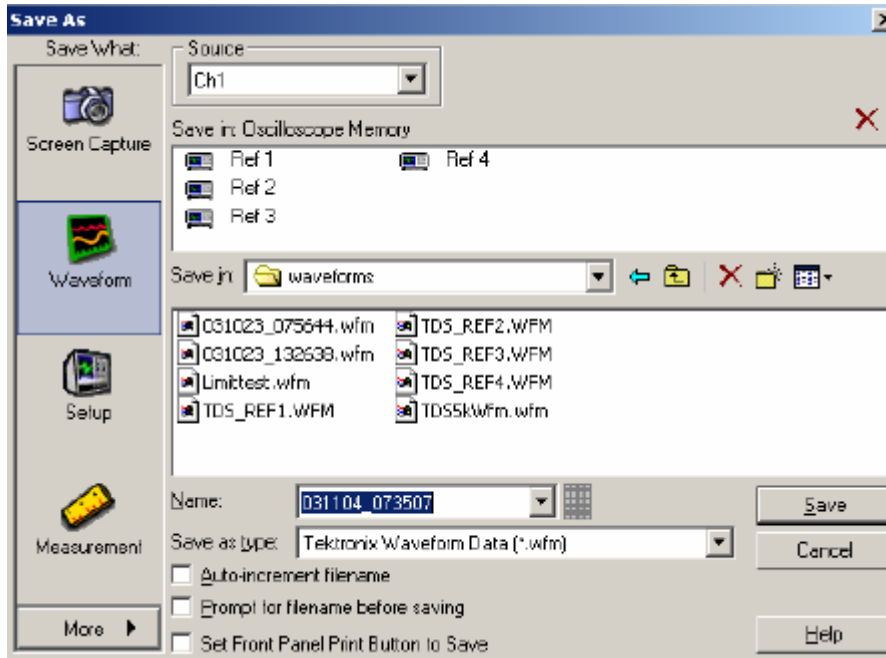
Behavior

This dialog box opens when you select Save As from the File menu or click Save from the Reference Waveform controls. The default file type is a Tektronix waveform file (.wfm). If you select a different file type from the Save as Type list, you can also define the options for saving the waveform.

The default folder for reference waveforms is **C:\TekScope\Waveforms**. The following file formats are available:

- **.wfm**. Internal instrument formats

- .**csv**. Usable by spreadsheets such as Microsoft Excel

- **.dat**. Usable by Mathcad and MATLAB

- **.txt**. Standard text file format

Note

Any characters that you enter in the file name box become part of the file name. The instrument does not use any user-defined file extensions.

Recall Dialog Box: Waveform

Access this control window from the File menu, Recall control or from the Reference Waveform controls, Recall button.

To Use

Use this dialog box to recall a saved waveform file to a reference location. From the Destination list, select the Ref 1-4 location that you want to recall the waveform to. The default folder for reference waveforms is **C:\TekScope\Waveforms**. Use the pop-up keyboard to enter a file name, or click the file name to select it.
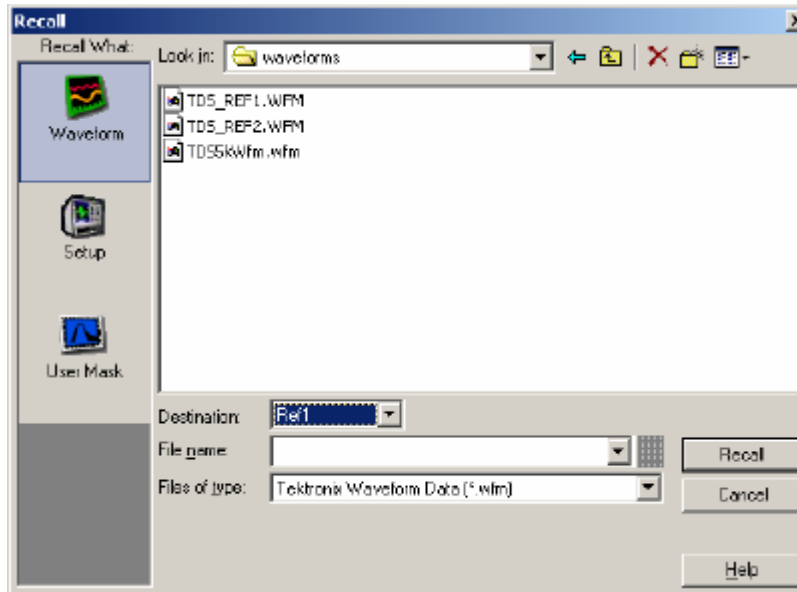
Tip If you don't see the file that you want to recall, check that the Files of type list shows the correct file format. You may also need to check other folders if the file was stored in a location other than the C:\TekScope\Waveforms folder.

Behavior

This dialog box opens when you click Recall from the File menu or when you click the Recall button from the Reference waveform controls. Waveforms are stored in WFM file format, which is an internal format of the instrument. When you click Recall, the Ref 1-4 waveform is displayed in the graticule.

Note

When you recall a reference waveform, a temporary file containing the waveform data is written to **C:\TekScope\Waveforms**. The file is named **TDS_REF*n*.WFM**, where *n* is the reference waveform number. This file is deleted when the reference waveform is deleted.

Save As Dialog Box: Instrument Setup

Access this control window from the Save toolbar button or the File menu Save As command.

Overview

Use the Save As dialog box to save instrument setups for later use.

To Use

To save in Oscilloscope memory, click one of the setup location icons under Save in Oscilloscope Memory. If you want to rename the setup, click the keyboard icon, and enter a label for the setup. Click **Enter**, and then click **Save**. To save the setup in a different location, select the location from the Save in list. Enter a name in the Name text box, and then click **Save**.
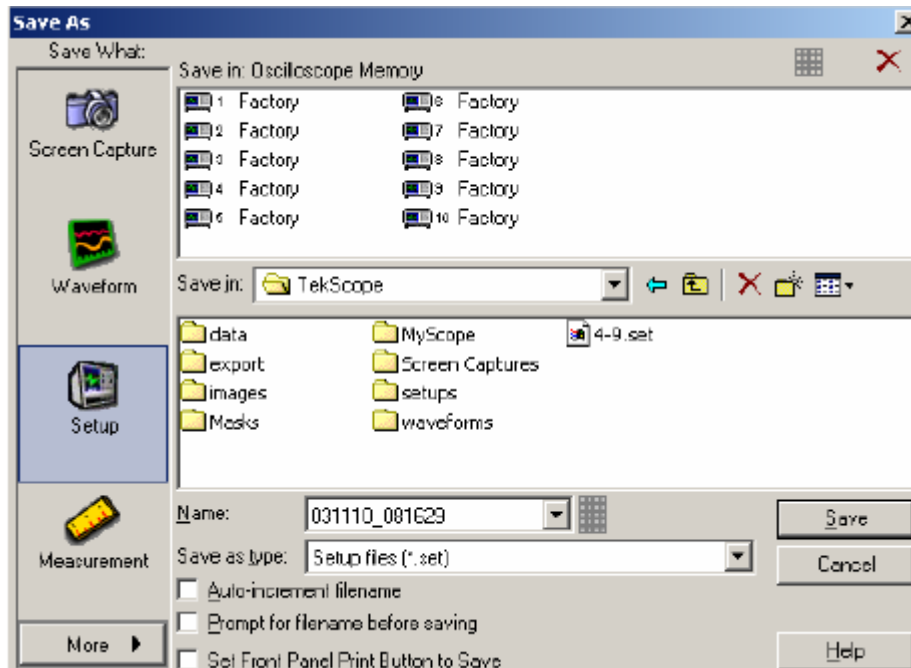
Behavior

You can display up to ten saved instrument setups in this dialog box. Additional setups can be saved to the hard drive**.** When you save a setup, the label adjacent to the setup icon changes from its previous value to User. You can save setups to any location labeled Factory without overwriting a saved setup.

The data in reference waveforms is not saved with the instrument setup. However, the setup information does include which reference waveforms are turned on and which reference waveforms are used with math waveforms and automatic measurements. If you want to save the reference waveform data, save the reference waveform separate from the instrument setup.

The instrument preference settings are not saved with the setup. They are saved when the instrument power is powered off.

Note

Before saving an instrument setup, make sure that the location does not contain a setup that you have previously saved. The previously saved setup will be overwritten.

Recall Dialog Box: Instrument Setups

Access this dialog box from the Recall toolbar button or the File menu Recall command.

Overview

Use the Recall dialog box to recall instrument setups.

To Use

Setups are recalled from an internal file on your hard drive or from a setup file that you have named. Click one of the icons under Look in Oscilloscope Memory, and then click Recall to restore the setup. To recall the default factory setup for the instrument, click the Factory Defaults icon.

💡**Tip** Select the File menu **Recall Default Setup** command to instantly restore the default setup.

To recall an instrument setup from a file, select the location of the setup from the Look in list. Select the setup you want to recall, and then click Recall.
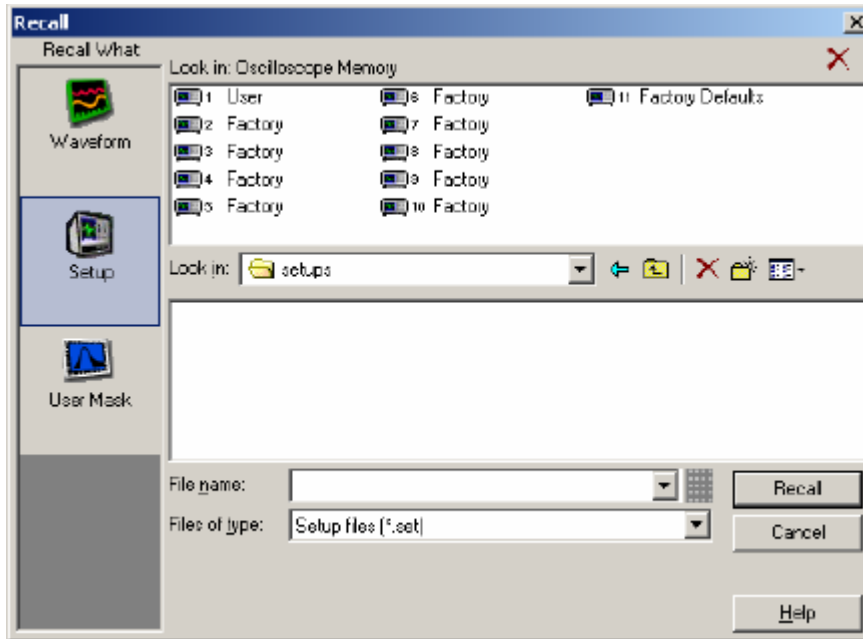
Behavior

The setups are restored from an internal file on your hard drive as soon as you click Recall. The labels adjacent to the icons were created when the setups were first saved. Any of the setups with the label "Factory" will restore the default factory setups.

The data in reference waveforms is not saved with the instrument setup. However, the setup information does include which reference waveforms are turned on and which reference waveforms are used with math waveforms and automatic measurements. If you want to save the reference waveform data, save the reference waveform separate from the instrument setup. You can then recall the saved waveform to the setup from the Reference Waveform Controls.

Note

If a restored setup attempts to use an undefined reference waveform (or to use it in a math waveform or with an automatic measurement), then an undefined reference waveform is created. This reference waveform becomes a 500-point waveform filled with null data. Any math waveforms or any measurements done on this waveform will contain invalid or unusable data. However, you can recall a valid saved waveform to the new reference waveform so that the math waveforms and the automatic measurements will become valid.

Delete Setups and Reference Waveforms

Access this command from the File menu.

To Use

Select the setup or reference waveform that you want to delete from the Delete command on the File menu. To delete all Setups at the same time, select All Setups, to delete All Reference Waveforms at one time select All Refs.

Behavior

You can only delete setups and reference waveforms stored in oscilloscope memory from this command.

Setups or waveforms stored in Windows directories can be deleted by clicking the in the Save or Recall dialog boxes

Page Setup Dialog Box

Overview

Use the Page Setup dialog box to define the page setup before sending the data to a printer.

To Use

The controls in this window are similar to the Page Setup dialog box in most Windows applications. The Paper settings are dependent on the type of printer connected to the instrument. A reduced preview image appears at the top of the window. To view a larger image, click Print Preview at the bottom of the window.

Click Print when you have selected all of your desired settings.

Behavior

The Type of printing should be Screen-copy for all screen shots. Banner printing should only be used to print long continuous waveforms on the integrated thermal printer. For instructions on printing banners, see Printing Banners.

Print Preview Dialog Box

Overview

Use the Print Preview dialog box to preview an image before sending the data to a printer.

To Use

Use the controls at the top of the window to zoom in and out to view the details of the image. To return to the Page Setup dialog box, click Page Setup at the top of the control window. Click Print to open the Print dialog box.

Print Dialog Box

Overview

Use the Print dialog box to send data to a printer.

To Use

The controls in this window are similar to the Print dialog box in most Windows applications. The settings are dependent on the type of printer connected to the instrument.

Behavior

If you have the integrated printer installed, the default Name is Integrated Thermal Printer. The integrated printer is available as an option for your instrument.

Using Print Screen

Overview

If you have an attached keyboard you can use the Print Screen (Prt Sc) key to copy the screen contents to the Windows clipboard. This bitmap does not include the waveforms or graticule. The waveforms and graticule are displayed by the graphics adapter outside of normal Windows mechanisms.

To Use

To capture the instrument screen with its graticule and waveforms, use the Edit menu Copy command after selecting Image in the Copy Setup menu, or to build a bitmap file, select the File menu Save As command. Click **Options** under Screen Capture, and then select Full Screen under View. For additional information see Save As Dialog Box: Screen Captures

You can display the current date and time on screen so that they appear on hard copies you print. To do this, toggle the Display Date/Time button On from the Display Setup control window Objects tab. If you need to reset the date and time for any reason, right-click on the date and time and then select Set Date/Time.

Behavior

Pressing the Print Screen key on an attached keyboard copies the screen image minus the waveform and graticule to the Windows clipboard. Using the Edit menu Copy command copies the screen image with the waveform and graticule.

Save As Dialog Box: Screen Capture

Access this dialog box from the File menu Save As command.

Overview

Select Screen Capture to save images to a file for use with other applications.

To Use

Select Screen Capture from the Save As dialog box, and then click Options…. Select the palette for the image: Color, Grayscale, or Black & White.

Select the image under View. To save an image of the entire screen, select Full-screen; to select an image of the selected graticule, select Graticule(s) only. If you want to save the image without including the Menu bar or Toolbar, select the Hide Menu/Toolbar check box.

Select the type of image from the Image list. To save the image exactly as it appears on the screen, select Normal. To export the image on a white background, select Ink-Saver Mode.

Select a file type from the Save as type drop-down list.

Assign your own file name, and then save the file to a specified folder. By default, the image will be saved to **C:\TekScope\ScreenCaptures**. Click Save to save the screen capture or cancel to close the dialog box without saving.
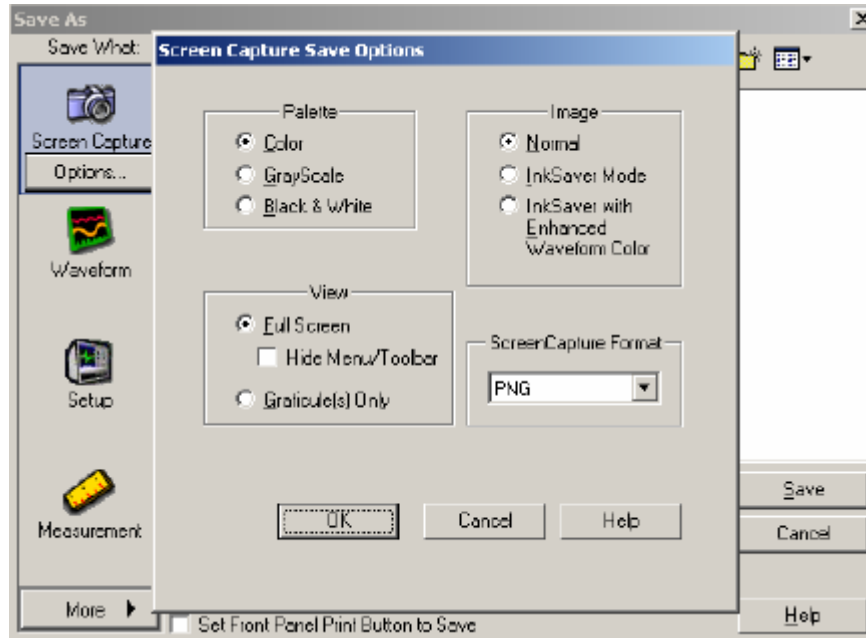
Behavior

The Palette, View, and Image selections are identical to those in the Page Setup dialog box.

Current file types are .bmp, .jpg, .pcx, .png, and .tif.

Waveform Options Dialog Box

Access this dialog box from the Save As dialog box, Waveform Options button.

Overview

Use the Waveform Save Options dialog box to set the parameters for the waveform that you want to save.

To Use

Select the data format (Data destination) for the waveform that you want to save. Select a valid waveform from the Source list.

Select the number of samples that you want to save under Waveform Data Range. Select a range of samples, only the samples between the cursors, samples from a specific Zoom Area, or all samples. If you select Samples between cursors, you can use the multipurpose knobs to adjust the positions if the cursors are already active. If they are not, the cursors are activated in their most recent positions.

Select Spreadsheet (TXT) to include spreadsheet data separated by white space. Select Spreadsheet (CSV) to include spreadsheet data separated by commas.

Select the Include waveform scale factors check box if you want to include waveform details like header information (record length, sample interval, trigger point information, and so on). If you only want to include the waveform voltage values, clear the selection.

If you are saving Fast Acquisition data, select the data ordering from the list.

After making the above selections, click OK to return to the Save As dialog box where you can assign your own file name and save the waveforms to a specified folder. By default, the file is saved to the **C:\TekScope\Waveforms** folder.

Behavior

The waveform data is saved as numeric values. The data destination that you select here is the file type selection used in the Save As dialog box. Use the Save As dialog box to finalize the selections and to save the waveform to a file.

Save As Dialog Box: Measurement

Access this dialog box from the File menu Save As command.

Overview

Select Measurement to save the measurement to a file for use with other applications.

To Use

Click **Options** to set the parameters of the measurement that you want to save. In the Measurement Save Options dialog box, select the Measurement Format from the drop-down list for the measurement that you want to save.

Select either Displayed Measurements or Measurements Snapshot.

After making the above selections, click **OK** to return to the Save As dialog box. Use the default name or enter a unique in the Name text box. Select the type of file (either .txt or .csv) to save from the Save as type list. By default, the file will be saved to the **C:\TekScope\Data** folder. Click **Save** to accept any changes and close the dialog box.

Behavior

When you select Displayed Measurements, all displayed measurements are saved.

When you select Measurement Snapshot, the current snapshot data is saved. If there is no valid snapshot data, the default snapshot values are exported.

Save As Dialog Box: Histogram

Access this dialog box from the File menu Save As command.

To Use

To save histogram data with the default file name, click Save. To enter a different file name, click the keyboard icon and use the pop-up keyboard or select Auto-increment file name if you want to save a series of files without typing in a new name each time. The file extension specified in the Save as type list will automatically be appended to the file name.

Behavior

Histogram data is saved in C:\TekScope\data unless you select a different location. To access the Histogram Save As dialog box, you may need to select it from the More submenu.

Save As Dialog Box: Timestamp

Access this dialog box from the File menu Save As command.

To Use

To save a Timestamp Table with the default file name, click Save. To enter a different file name, click the keyboard icon and use the pop-up keyboard or select Auto-increment file name if you want to save a series of files without typing in a new name each time. The file extension specified in the Save as type list will automatically be appended to the file name.

Behavior

Timestamp tables are saved in C:\TekScope\data unless you select a different location. To access the Timestamp Save As dialog box, you may need to select it from the More submenu.

Auto-increment file name

Select Auto-increment file name to save numerous similar files without retyping the entire file name each time.

Enter a Base file name and click Save. For the initial save the default count is 000, but you can overwrite this number if you choose. Your first file is saved as [*Basefilename*][*count*].*ext*, where *ext* is the file extension. On subsequent saves, the instrument searches for the highest numbered file name and increases the number by one, as in *Basefilename*001.*ext*.

For example, if you are saving a series of rise time data files, you can use "Risetime" as the base file name. Your first file is saved as Risetime000.*ext*, where *ext* is the file type extension. The next file will be Risetime001.*ext*, and so on.

If Count reaches 999, it is suggested that you change the base file name to *Basefilename*1 (for example, Risetime1) on the next save. Your next file will then be saved as Risetime1000.*ext*. You will receive a warning message if you are going to overwrite any files with the same name.

Edit Menu

Use the Edit menu for copying images, waveforms, or measurements to the clipboard for use in other applications. You can select items from the menu or you can select Copy Setup to fine tune the parameters with the Copy Setup control window before copying items to the clipboard.

Copy Setup: Images

Access this control window from the Edit menu.

Overview

Use this control window to copy images to the clipboard for use with other applications.

To Use

Select the Palette for the image: Color, Grayscale, or Black & White.

Select the type of View. To copy an image of the entire screen, select Full-screen; to copy an image of the selected graticule, select Graticule(s) only.

Select the type of Image. To copy the image exactly as it appears on the screen, select Normal. To copy the image on a white background, select InkSaver Mode. To copy the image on a white background with colors especially designed to print on the white background, click InkSaver with Enhanced Waveform Color.

Bitmap is the only available Data Format.

Behavior

The Palette, View, and Image selections are identical to those in the Page Setup dialog box and in the Screen Capture Save Options dialog box. The copy operation takes place after you click Copy. Clicking OK saves the changes and closes the dialog box but does not copy the image.

Copy Setup: Waveforms

Access this control window from the Edit menu Copy Setup command.

Overview

Use this control window to copy waveforms to the clipboard for use with other applications.

To Use

The Data Destination (data format) is set to Spreadsheet. Select a valid waveform from the Source list.

Select the number of samples that you want to copy. Select all samples, a range of samples, or only the samples between the cursors. If you select Data between cursors, you can use the multipurpose knobs to adjust the positions if the cursors are already active. If they are not, the cursors will be turned on at the positions they were in the last time they were used.

Select the Include waveform scale factors check box if you want to include waveform details such as header information (record length, sample interval, trigger point information, and so on). If you only want to include the waveform voltage values, clear the selection.

If you are copying Fast Acquisition data, select the data ordering from the list.

Behavior

The Waveform curve data range can be anywhere from one sample to the record length of the selected waveform.

The copy operation takes place after you click Copy.

If you select Data between cursors, another set of cursors may temporarily appear on your screen while the copy is in progress.

Copy Setup: Measurements

Access this control window from the Edit menu.

Overview

Use this control window to copy measurements to the clipboard for use with other applications.

To Use

Select the data format for the measurement that you want to copy. For Histogram Data, the data format is always CSV.

Select the type of measurement.

Behavior

When you select Displayed Measurements, all displayed measurements will be copied to the clipboard.

When you select Measurement Snapshot, the current snapshot data is copied to the clipboard. If there is no valid snapshot data, the default snapshot values will be sent to the clipboard.

When you select Histogram Data, the histogram data for the waveform will be copied in the CSV format.

The copy operation takes place after you click Copy. Clicking the OK button accepts the changes and closes the dialog box but does not copy the measurements.

Autoset Undo Control Window

This control window opens automatically after an Autoset operation.

Overview

Use this control window to undo the last Autoset operation.

To Use

Click Undo. Only the parameters that were set by the last Autoset are undone. Parameters that you changed that are not controlled by Autoset retain their settings.

Behavior

The Autoset Undo control window remains on the screen until you open another control window. After Autoset Undo closes, you can still undo the last Autoset by selecting the Undo Last Autoset command from the Edit menu. Although the last Autoset is immediately undone, the Autoset Undo control window does not open again.

Vertical Menu

Use the Vertical menu options to set vertical parameters for waveforms. This menu also provides access to the Zoom controls and the probe calibration, deskew, and attenuation functions.

Vertical Setup Control Window

Access this control window from the Vertical menu Vertical Setup command, or the Vert toolbar button.

Overview

Use the Vertical Setup control window to set the vertical parameters for live (channel) waveforms. The controls for each channel are independent. Use the tabs at the top of the control window to select the channel you want to change.
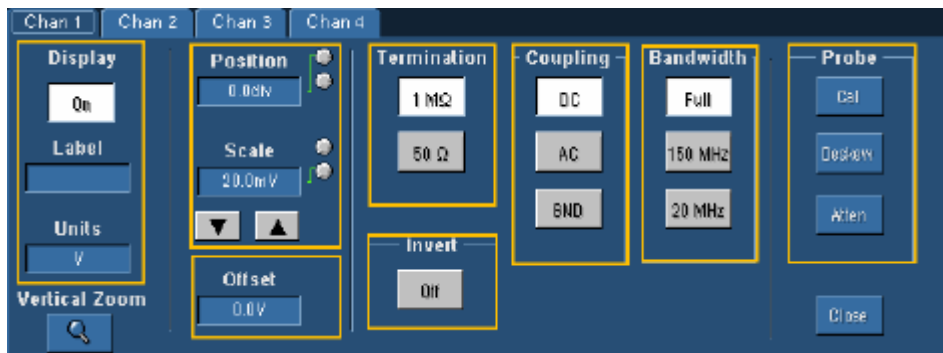
Zoom Setup Control Window

Access this control window from the Vertical or Horiz/Acq menu Zoom Setup command.

Overview

Use the Zoom Setup control window to define the overall settings of the Zoom controls.
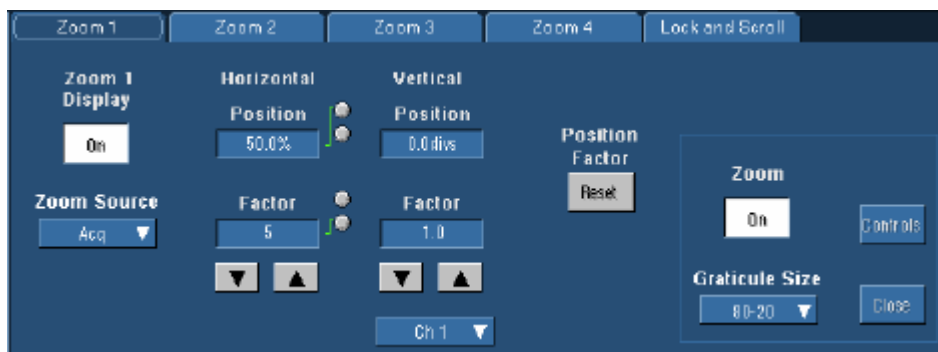
To Use

Click the Zoom 1 – 4 tabs to set up each zoom area. The controls under each tab apply only to that zoom area. Click the Zoom (1-4) Display On to turn on the display of that area. (MultiView Zoom must also be turned on to display any of the zoom areas.) Use the Zoom Source drop-down list to select a source for the zoomed area. Click the Zoom button On to turn on Zoom. You can also toggle Zoom on and off with the MultiView Zoom button on the front panel.

Use the Channel drop-down list under Vertical to select which waveform (Ch1-4, Math1-4, or Ref1-4 on 4-channel instruments or Ch 1-2, Math 1-2, or Ref 1-2 on 2-channel instruments) the Vertical Position and Factor controls adjust.

The Position and Factor controls change the position and factor (or scale) in the Zoom area. They can be adjusted with the multipurpose knobs. To reset the Position and Factor controls, click Reset.

Select the size of the Acquisition and Zoom windows from the Graticule Size drop-down list. Select 80-20 to allocate 80% of the available display for the zoomed graticule and 20% for the acquisition window. Select 50-50 to allocate half of the available display for the zoomed graticule and half of the available display for the acquisition window. Select Full to use the entire display for the zoomed graticule.

Click the Zoom Lock and Scroll tab to scroll the zoomed area(s). To scroll a single zoomed area, select the zoom area (1-4), and then click an Auto Scroll button. You can also set the scroll speed. To scroll multiple zoomed areas simultaneously, click Lock, and then click the zoomed areas (1-4) you want to scroll through. This locks the zoomed areas in their relative horizontal positions.

Tip Use the Vertical or Horiz/Acq menu Zoom Graticule Size submenu to directly access the Graticule Split options.

Behavior

Use the Zoom Setup control window to magnify (zoom in on) a waveform without changing the main acquisition parameters. Use Zoom when you want to temporarily look at details of a waveform.

If the zoom areas are unlocked, the Horizontal Position and Factor controls affect only the selected zoom area. The Horizontal Position and Factor will affect all zoom areas that are locked together, such that a change to one zoom area will change the horizontal position and factor of all the locked zoom areas.

On the Vertical Zoom tab, the Scale and Factor controls only affect the selected waveform.

Zoom Control Window

Access this control window from the Vertical or Horiz/Acq menu Zoom Controls command, or push the front-panel MultiZoom button.

Position

50.0%

Factor

5

| Zoom | Scroll |

Assign Knobs

| Horiz | Vert |

Area

Zoom 1 ▼

Display

On

Zoom Source

Acq ▼

Lock

| Setup | Close |

Overview

Use the Zoom control window to set the position and factor of the selected waveform in the Zoom window, select the Zoom Area and Source, and to lock and scroll the zoomed areas.

To Use

Click Horiz to set the Horizontal Zoom controls or Vert to set the Vertical Zoom controls. Select the Zoom Area from the Area drop-down list. The changes you make in this control window apply only to the zoom area selected. Use the Zoom Source drop-down list to select the waveform or Zoom area that you want to zoom in on. Adjust the Position and Factor controls using the multipurpose knobs. Click Setup to display the Zoom Setup control window where you can access additional Zoom controls.

Behavior

The Factor and Position controls only affect the waveforms in the lower (Zoom) window. To change the Factor and Position controls in the upper (Acquisition) window, use the front panel HORIZONTAL or VERTICAL POSITION and SCALE knobs. You can access the controls for both the Acquisition and Zoom windows from the front panel. Selecting Zoom (1-4) as the Zoom Source allows you to zoom on zoom.

Tip You can easily zoom in on a waveform by clicking and dragging a box around the area of interest. Then select Zoom (1-4) from the shortcut menu.

AutoScroll Control Window

Access this control window from the Vertical or Horiz/Acq menu Zoom Controls command, and then click the Scroll tab.

Overview

Use the AutoScroll control window to set the scroll speed and lock zoomed areas together.

To Use

To scroll a single zoomed area, select the zoom area (1-4), and then click an Auto Scroll button. You can also set the scroll speed. To scroll multiple zoomed areas simultaneously, click Lock, and then click the zoomed areas (1-4) you want to scroll through. This locks the zoomed areas in their relative horizontal positions.

Behavior

If the zoom areas are unlocked, the Horizontal Position and Factor controls affect only the selected zoom area. The Horizontal Position and Factor will affect all zoom areas that are locked together, such that a change to one zoom area will change the horizontal position and factor of all the locked zoom areas.

Graticule Zoom Control Window

Access this control window from the Vertical or Horiz/Acq menu, Graticule Size submenu, Size command.

Graticule Size

50/50%

80/20%

100%

Setup   Close
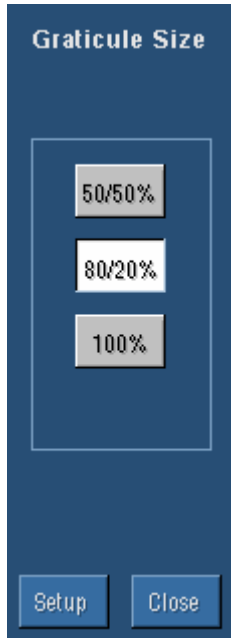
To Use

Click one of the Graticule Size buttons to set the proportion of the display area for the acquisition window and zoom window. Click Setup to access the Zoom Setup control window.

Behavior

Click the 50-50 button to allocate half of the available display for the zoomed graticule and half of the available display for the acquisition window. Click 80-20 to allocate 80% of the available display for the zoomed graticule and 20% for the acquisition window. Click Full to use the entire display for the zoomed graticule.

Waveform Display Control Window

Access this control window from the Vertical menu Display On/Off command. Display On/Off is also available on the Math menu.

Source

CH Math Ref

1   2

3   4

Display

On

Close

Overview

Use the Waveform Display control window to turn the selected waveform on or off.

To Use

Click Display to turn the selected waveform on or off. Use the Channel Selector tabs to select a valid channel, math, or reference waveform.

Behavior

The Display button applies only to the selected waveform. Each waveform (channel, math, or reference) has its own Display button.

Any displayed channel, math, or reference waveform can be turned off without impacting any math functions, measurements, or histograms done on that waveform. You may want to turn some waveform displays off to make the remaining waveforms easier to view and analyze.

Tip Use the dedicated front panel buttons to turn the channel waveforms on or off.

Waveform Label Control Window

Access this control window from the Vertical menu Label command. Label is also available on the Math menu.
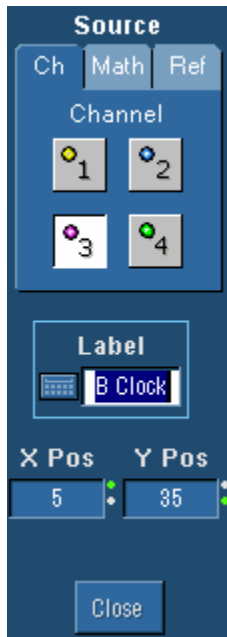
Overview

Use the Waveform Label control window to attach a label to the waveform.

To Use

Use the Channel Selector tabs to select a valid channel, math, or reference waveform. Assign a label to the selected waveform with the pop-up keyboard to make it easier to identify on the screen. Click the X Pos and Y Pos controls to activate them and use the multipurpose knobs to position the label on the graticule. You can also position the label anywhere on the screen by clicking and dragging it to the desired location.

Behavior

The label applies only to the selected waveform. Each waveform (channel, math, or reference) has its own label. If you do not enter a label for the selected waveform, the label will be left blank. A label only appears when its associated waveform is turned on.

The X Pos and Y Pos control values are relative to the upper-left corner of the display. The values are expressed in pixels.

Position/Scale Control Window

Access this control window from the Vertical menu Position/Scale command. Position/Scale is also available on the Math menu.
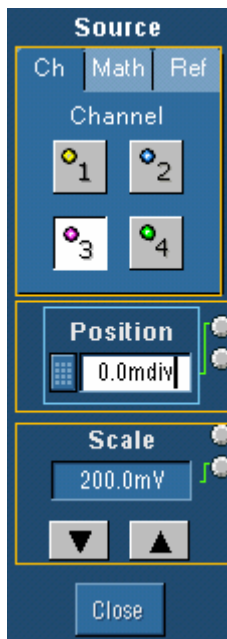
Overview

Use the Position/Scale control window to set the vertical position and scale of the selected waveform.

Use the Channel Selector tabs to select a waveform.

In addition to using the controls in this control window, you can also use the dedicated controls on the front panel to perform the same functions.

Position/Scale Control Window: Position

To Use

Use the Position control to set the vertical position of the selected waveform. Click the control to map it to one of the multipurpose knobs, and then use the knob to move the waveform.

Behavior

The Position control moves the vertical position of the waveform 0.2 divisions (0.02 divisions with FINE control). Increasing the position value moves the waveform up, and decreasing the position value moves the waveform down.

The Position control differs from the Offset control; for information on the Offset control for channel waveforms refer to the Vertical Offset Control Window on page 83.

Tip Several alternatives are available for adjusting the position of a waveform:

- Use the dedicated knob on the front panel for the selected waveform.
- Enter a value with the onscreen pop-up keypad.
- Use an attached keyboard to type a value directly into the position field.

Position/Scale Control Window: Scale
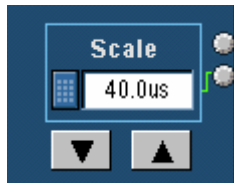
To Use

Use the Scale control to set the vertical scale value of the selected waveform. Click the control to map it to one of the multipurpose knobs and use the knob to move the waveform.

Behavior

Each waveform has its own vertical scale parameter. For a signal with constant amplitude, increasing the scale causes the waveform to appear smaller. Decreasing the scale causes the waveform to appear larger.

The scale affects all waveforms, but affects channel waveforms differently:

- For channel waveforms, the scale setting controls the vertical size of the acquisition window as well as the display scale. The range and resolution of the scale value depends on the attached probes and any other external factors you may have specified.
- For reference or math waveforms, the scale setting controls the display only. Scaling these waveforms has no affect on the acquisition hardware.

Tip Several alternatives are available for adjusting the scale of a waveform:

- Use the dedicated knob on the front panel for the selected waveform.
- Enter a value with the onscreen pop-up keypad.
- Use an attached keyboard to type a value directly into the position field.

Vertical Offset Control Window

Access this control window from the Vertical menu Offset command.
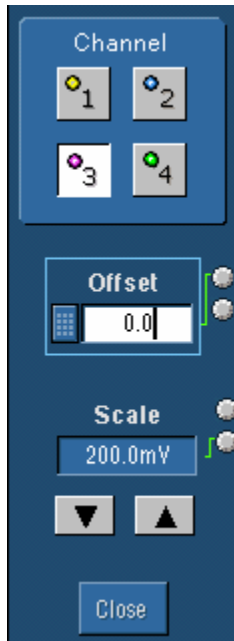
Overview

Use the Vertical Offset control window to set the offset value of the selected waveform to shift the vertical acquisition window to match the waveform data you want to acquire.

To Use

Click a Channel button to select the vertical input channel. Click the Offset control to map it to one of the multipurpose knobs, and use the knob to set the offset value.

Behavior

The main purpose of the Offset control is to eliminate clipping the waveform. The Offset control affects the vertical acquisition window by subtracting a DC bias from the waveform. This moves the level at the vertical center of the acquisition window of the selected channel. Visualize offset as scrolling the acquisition window toward the top of a large signal for increased offset values, and scrolling toward the bottom for decreased offset values.

The Offset control affects the waveform display by defining the channel reference indicator to match the offset rather than the ground level. The offset affects only channel waveforms as compared to the position control that affects all waveforms, including math and reference waveforms. The range of the vertical offset available depends on the vertical Scale setting.

Termination Control Window

Access this control window from the Vertical menu Termination command.
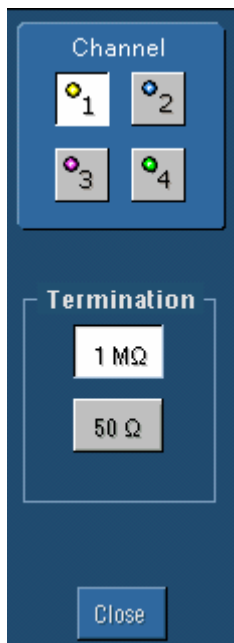
Overview

Use the Termination control window to select the channel input resistance (50 Ω or 1 MΩ).

To Use

Click a Channel button at the top of the control window to select the vertical input channel. If you are not using a probe with a TekProbe interface, click one of the Termination buttons to set the termination for your probe.

Behavior

The correct termination is set automatically when you attach a probe with a TekProbe interface to the instrument. However, you may have to set the termination manually if you use a probe without the TekProbe Interface. In general, select 1 MΩ to use high impedance passive probes. Select 50 Ω to use most active probes and low impedance (Zo) probes.

Note

The 50 Ω termination becomes an open circuit if you set the channel input coupling to GND.

Consider the following when using 50 Ω termination with any channel:

- The instrument does not accurately display frequencies under 200 kHz with AC coupling.
- The instrument reduces the maximum vertical scale setting for the channel to 1 V from 10 V (to 10 V from 100 V with a 10 X probe attached), since the amplitudes appropriate for the higher settings would overload the 50 Ω input.
- The instrument switches to 50 Ω and disables AC coupling (and switches coupling to DC if AC is selected) if you connect an active probe. The active probes also reduce the maximum vertical scale setting as described above. This behavior results in 50 Ω, nonAC coupling, which is appropriate for active probes.

Note

If you remove an active probe, the instrument does not switch coupling back to 1 MΩ or AC. When you restore 1 MΩ coupling, the instrument does not return to the vertical scale setting that was reduced because of the 50 Ω selection. You must set the vertical scale, coupling, and termination appropriate for your input coupling scheme. Be sure to switch to 1 MΩ for any input signal not from a 50 Ω system.

🔅**Tip** Use the button on the front panel of each vertical input channel to manually select the termination.

Coupling Control Window

Access this control window from the Vertical menu Coupling command.
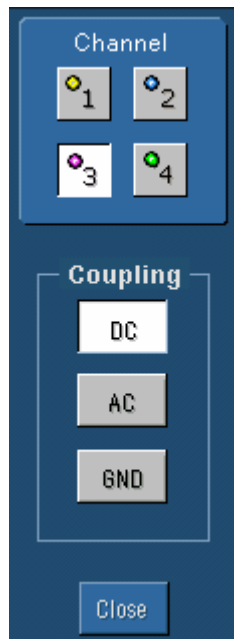
Overview

Use the Coupling control window to select the coupling for the signal from the attached probe to the instrument.

To Use

Click a Channel button at the top of the control window to select the vertical input channel. Click one of the Coupling controls to couple the signal from the probe to the instrument.

Behavior

Select DC coupling to display the waveform with the DC and AC components.

Select AC coupling to display the waveform with the DC component removed.

Select GND to display a zero-volt waveform. Use this selection to establish the ground reference point on the display

Bandwidth Control Window

Access this control window from the Vertical menu Bandwidth command.
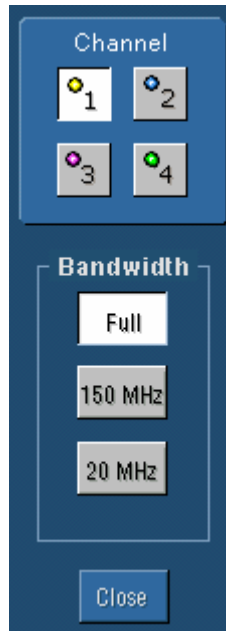
Overview

Use the Bandwidth control window to filter unwanted high frequency noise from the waveform and to decrease the noise bandwidth. Specify the range of frequencies that you want to acquire by selecting the appropriate button in the control window.

To Use

Click a Channel button at the top of the control window to select the vertical input channel. Click one of the Bandwidth buttons to select the bandwidth of frequencies that you want to acquire.

Behavior

The bandwidth refers to the range of frequencies that the instrument can acquire and display accurately with less than 3dB attenuation. Each input channel has its own bandwidth selection.

To acquire all frequencies on the selected channel, click Full. To limit frequencies above 150 MHz, click 150 MHz; to limit frequencies above 20 MHz, click 20 MHz.

To take accurate measurements, the input frequency should be much less than the rated bandwidth of the instrument. A good rule to follow is to ensure the bandwidth of the instrument system is three to five times the bandwidth of the signal that you want to measure.

Probe Calibration Control Window

Access this control window from the Vertical menu Probe Cal command.
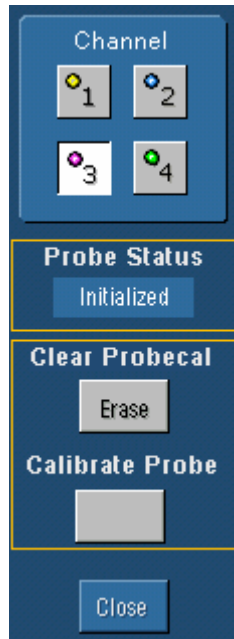
Control Window Overview

Use the Probe Calibration control window to check the probe status and to compensate the entire signal path, from the probe tip to digitized signal, to improve the probe gain and offset accuracy.

Probe calibration is one of several features of the instrument that can optimize measurement accuracy. In addition to the probe calibration, you should also perform the following procedures:

- Probe Compensation. Use this procedure with passive probes to compensate them to ensure maximum distortion-free inputs to the instrument and to avoid high frequency amplitude errors.

- Signal Path Compensation. Use this procedure to compensate the internal signal path used to acquire waveforms based on the ambient temperature.

- Probe Deskew. Use this procedure to compensate probes for delays due to differences in cable length.

The Probe Status control indicates the status of the probe calibration for the selected channel. If the probe status does not indicate Pass, the instrument may not take accurate measurements. You may want to perform the probe calibration procedure before taking any measurements with the selected probe.

Probe Calibration Control Window: Probe Status

To Use

The Probe Status control provides information on the status of the probe connected to the vertical input channel. This control provides information only; the data in the control cannot be changed directly.

Behavior

The instrument automatically detects the type of probe that you have installed and displays the probe status information.

- Initialized. The instrument was unable to detect any information from the probe in its memory. This message often appears when you connect a probe where you have not performed the probe calibration. You should perform the probe calibration before you use the probe. This message also appears when you disconnect a probe from the vertical input.

- Pass. The instrument recognized the probe information stored in nonvolatile memory and that the probe had previously passed the probe calibration procedure.

- Fail. The instrument was unable to complete the probe calibration procedure. If the Fail message appears, try performing the signal path compensation procedure before continuing with the probe calibration. If the Fail message still appears after performing the probe calibration procedure, you may have a faulty probe. You should have the probe checked by Tektronix service personnel.

Probe Calibration Control Window: Probe Calibration

To Use

Click a Channel button at the top of the control window to select the vertical input channel for the attached probe. Connect the probe tip to the appropriate signal and ground connections of the probe compensation terminals. Click Clear Probecal to clear the old calibration values. Then click Calibrate Probe to begin the calibration procedure.

Behavior

The instrument lets you compensate the probe, based on the channel to which it is connected, to improve the gain and offset accuracy of the probe. When you perform the probe calibration on the selected channel with the probe attached, you can optimize the instrument to make accurate measurements using that channel and probe. The probe calibration makes significant performance improvements when performed with active probes or older passive probes.

You should perform a probe calibration any time you want to ensure that the measurements you take are made with the most accuracy possible. You should also perform the probe calibration if you have changed probes since the last time a probe calibration was performed.

Tip If you are compensating a passive probe, you must first compensate the low frequency response of the probe. Refer to the Probe Compensation Procedure on page 353 for more information.

Some types of probes can be gain compensated, some can be offset compensated, and some can be compensated for both. If the probe has a TekProbe interface, the instrument can determine the type of compensation needed and will display the appropriate instructions when you perform the probe calibration procedure. Some probes cannot be compensated; for information on these probes, refer to probe calibration restrictions on page 94.

Deskew Control Window

Access this control window from the Vertical menu Deskew command.

Overview

Use the Deskew control window to compensate for propagation delays of different length probes connected to the vertical input channels.

To Use

Connect the probes to a signal source. Use the horizontal and vertical controls to center the fastest signal on the screen. Click a channel button at the top of the control window to select the slower channel. Use the multipurpose knob to adjust the deskew time so that the two signals align. To clear the deskew value, click Set to 0.0s.

Behavior

Each input channel has its own deskew settings. Deskew compensates individual channels for probes or cables of different lengths. The instrument applies the delay values after each completed acquisition. The deskew values, which have a range of ±75 ns, do not affect triggering or the XY display format.

The deskew values are saved as part of the instrument setup. The deskew values for the selected channel will be retained until you change the probe, you restore a saved setup, or you recall the factory setup.

Deskew is supported in Fast Acquisitions mode when you acquire one or two channels. If you acquire three or more channels, the Fast Acquisition mode is turned off and the message, *Deskew not applied in FastAcq when three or more channels are acquired*, is displayed. When you switch back to acquiring one or two channels, the message, *Deskew is re-enabled*, is displayed and deskew values are applied again.

Attenuation Control Window

Access this control window from the Vertical menu Attenuation command.
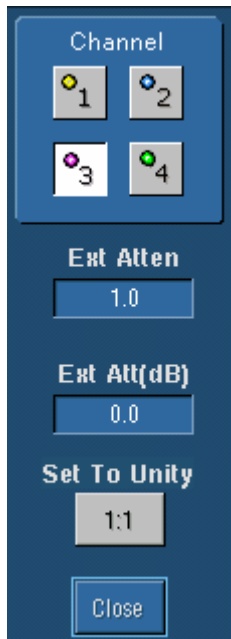
Overview

Use the Attenuation control window to describe the input/output ratio of any external attenuation or gain between the signal and input channels.

To Use

Click a channel button at the top of the control window to select the vertical input channel for the attached probe. To set the external attenuation as a multiplier or ratio, use the multipurpose knob to enter a value in the top control. To set the external attenuation as dB values, use the multipurpose knob to enter a value in the lower control. To set the external attenuation to the default value, click Set to Unity.

Behavior

Each input channel has its own external attenuation settings. The attenuation control is automatically set to the correct value when you attach a probe with a TekProbe interface to the instrument.

You can change the external attenuation by entering a multiplier or ratio in the top control, or by entering a dB value in the lower control. When you return the attenuation to the default value, the multiplier returns to 1, and the dB value returns to 0.

Display On/Off

To Use

Click Display to turn the channel waveform display on and off. Assign a label to the waveform with the pop-up keyboard to make it easier to identify on the screen. Set the units of vertical scale in the Units text box.

Behavior

The tab selector at the top of the Vertical Setup window identifies the channel waveform source. The Display button, the label, and units apply only to the selected waveform.

Each waveform (channel, math, or reference) has its own Display button, label, and units. If you do not enter a label for the selected waveform, the label will be left blank. You can position the label anywhere on the graticule by clicking and dragging it to the desired location.

Any displayed channel waveform can be turned off without impacting any math functions, measurements, or histograms done on that waveform. You may want to turn some waveform displays off to make the remaining waveforms easier to view and analyze.

Tip Use the dedicated front-panel CH buttons to turn the channel waveforms on or off.

Vertical Position/Scale
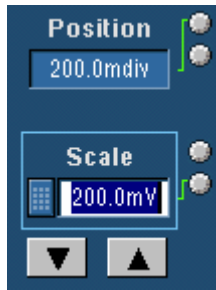
To Use

Use the Position control to set the vertical position of the waveform, and the Scale control to set the vertical scale. Click on the controls and use the multipurpose knobs to change the waveforms.

Behavior

Each waveform has its own vertical Position control. The control increases or decreases the vertical position of the waveform 0.2 divisions (0.02 divisions using the FINE knob). Increasing the position value of the waveform moves the waveform up, and decreasing the position moves the waveform down.

Each waveform also has its own vertical Scale control. For a signal with a constant amplitude, increasing the scale causes the waveform to appear smaller. Decreasing the scale causes the waveform to appear larger.

For channel waveforms, the scale setting controls the vertical size of the acquisition window as well as the display scale. The range and resolution of the scale value depends on the attached probes and any other external factors that you may have specified.

Tip Use the dedicated POSITION and SCALE knobs on the front panel to control the position and scale of the selected waveform.

Offset

To Use

Use the Vertical Offset controls to set the offset value of the selected waveform to shift the vertical acquisition window to match the waveform data that you want to acquire.

Click a channel at the top of the control window to select the vertical input channel. Click the Offset control to map it to one of the multipurpose knobs, and use the knob to set the offset value.

Behavior

Use the Offset control to eliminate clipping the waveform. The offset control affects the vertical acquisition window by subtracting a DC bias from the waveform. This moves the level at the vertical center of the acquisition window of the selected channel. Visualize offset as scrolling the acquisition window toward the top of a large signal for increased offset values and scrolling toward the bottom for decreased offset values.

The offset control affects the waveform display by defining the channel reference indicator to match the offset rather than the ground level. The offset affects only channel waveforms as compared to the position control that affects all waveforms, including math and reference waveforms.

Termination
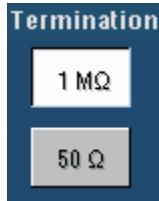
To Use

Use the Termination controls to select the channel input resistance (50 Ω or 1 MΩ). In general, select 1 MΩ to use high impedance passive probes. Select 50 Ω to use most active probes and low impedance (Zo) probes.

Note

The 50 Ω termination becomes an open circuit if you set the channel input coupling to GND.

Behavior

The correct termination is set automatically when you attach a probe with a TekProbe interface to the instrument. However, you may have to set the termination manually if you use a probe without a TekProbe Interface.

Consider the following when using 50 Ω termination with any channel:

- The instrument does not accurately display frequencies under 200 kHz with AC coupling.

- The instrument reduces the maximum vertical scale setting for the channel to 1 V from 10 V (to 10 V from 100 V with a X10 probe attached), since the amplitudes appropriate for the higher settings would overload the 50 Ω input.

- The instrument switches to 50 Ω and disables AC coupling (and switches coupling to DC if AC is selected) if you connect an active probe. The active probes also reduce the maximum vertical scale setting as described above. This behavior results in 50 Ω, nonAC coupling, which is appropriate for active probes.

Tip Use the front-panel button for each vertical input channel to manually select the termination.

Channel Invert

To Use

Use the Invert button to toggle inversion of the selected channel waveform.

Behavior

When Invert is On, the selected waveform appears mathematically inverted (flipped) on the zero volt axis. No trace of the normal waveform is left on the screen. Set Invert to Off to view the normal waveform.

An inverted channel waveform can be used for math or measurement functions. But if used as a trigger source, the normal, not inverted, waveform is used.

Coupling Setup

To Use

Use the Coupling controls to couple the signal from the attached probe to the instrument.

Behavior

Select DC to display the waveform with the DC and the AC components. Select AC to display the waveform with the DC component removed.

Select GND to display a zero-volt waveform. Use this selection to establish the ground reference point on the display.

Note

The 50 Ω channel input termination becomes an open circuit when you select GND coupling.

Bandwidth
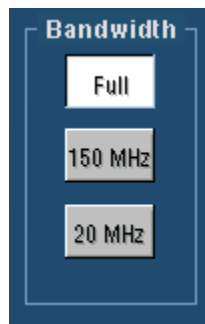
To Use

Use the Bandwidth controls to filter unwanted high frequency noise from the waveform and to decrease the noise bandwidth. Specify the range of frequencies that you want to acquire by clicking the appropriate button in the control window.

Behavior

The bandwidth refers to the range of frequencies that the instrument can acquire and display accurately with less than 3dB attenuation. Each input channel has its own bandwidth selection.

To acquire all frequencies on the selected channel, click Full. To limit frequencies above 150 MHz, click 150 MHz; to limit frequencies above 20 MHz, click 20 MHz.

To take accurate measurements, the input frequency should be much less than the rated bandwidth of the instrument. A good rule to follow is to ensure the bandwidth of the instrument system is three to five times the bandwidth of the signal that you want to measure.

Probe Controls

The Probe controls provide access to the Probe Calibration, Deskew, and Attenuation control windows.

Use the Probe Calibration control window (Cal) to check the status of the attached probe and to compensate the entire signal path from the probe tip to the digitized signal.

Use the Deskew control window to compensate the vertical channels for propagation delays of different length probes.

Use the Attenuation control window (Atten) to change the default attenuation or gain for the selected probe channel.

Channel Invert Control Window

Access this control window from the Vertical menu Invert command.

Overview

Use the Channel Invert control window to invert a live channel waveform.

To Use

Click one of the channel buttons to select a valid channel. Click Invert to toggle inversion of the selected waveform.

Behavior

When Invert is On, the selected waveform appears mathematically inverted (flipped) on the zero volt axis. No trace of the normal waveform is left on the screen.

Using Offset to Avoid Clipping Signals
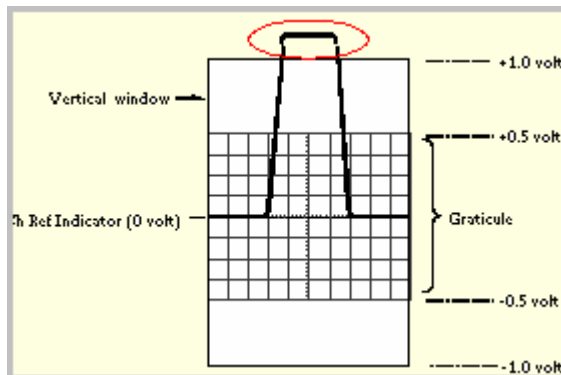
Clipping

The circled portion of the incoming waveform is clipped, and therefore, is not acquired.

The clipped portion cannot be processed or displayed.

Offset can move the acquisition window vertically relative to the incoming waveform.
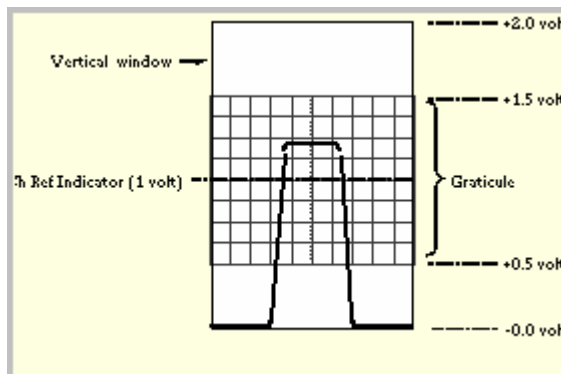
Offset removes clipping

By setting an offset to vertically shift the acquisition positive relative to the waveform, the clipped portion can be acquired.

Position, which up to now has been unchanged, can be used to position the graticule anywhere in the acquisition window.

High Impedance Probes

Most general-purpose probes have high input impedance. The high input resistance of passive probes (typically 10 MΩ) provides negligible DC loading and provides accurate DC amplitude measurements. Most passive probes require the instrument input impedance to be 1 MΩ. However, their 8 pF to 12 pF (over 60 pF for 1X) capacitive loading can distort timing and phase measurements. Use high resistance passive probes for measurements involving the following:

- Device characterization (above 15 V, thermal drift applications)
- Maximum amplitude sensitivity using 1 X probe
- Large voltage range (between 15 V and 500 V)
- Qualitative or go/no-go measurements

Low Impedance Probes

Most low impedance (Zo) passive probes measure frequencies or timing measurements more accurately than general-purpose probes, but they take less accurate amplitude measurements. They offer a higher bandwidth to cost ratio.

The low input impedance passive probes must be terminated in a 50 Ω instrument input. Input capacitance is much lower than high impedance passive probes, typically 1 pF, but input resistance is also lower (500 Ω to 5000 Ω typically). Although that DC loading degrades amplitude accuracy, the lower input capacitance reduces high frequency loading to the circuit under test. That makes low input impedance passive probes ideal for timing and phase measurements when amplitude accuracy is not a major concern. Low input passive impedance probes are useful for measurements up to 40 V.

Active Probes

Active voltage probes often contain preamplifiers that provide high resistance and low capacitance at the probe tip. Some active voltage probes contain differential amplifiers that provide high Common Mode Rejection Ratio (CMRR).

Active current probes often contain Hall-effect sensors that extend the probe bandwidth down to DC.

Many active probes, especially high frequency probes, can drive a 50 Ω instrument input.

Probe Calibration Restrictions

Some probes cannot be calibrated or compensated. This might be because the probes have an attenuation factor greater than 20X. You will get an error message if you try to compensate such a probe.

The instrument cannot compensate for probes that may have gain and/or offset errors that are too great (>2% or >50 mV offset). If these errors are within the specified limits of your probe, you may want to use another probe. If the errors are outside the limits, have your probe checked by service personnel.

Probe calibration is not recommended for the P5050 passive probe. This probe typically has little gain and offset error, and therefore, the performance improvement after a probe calibration is not worth the time needed to do the probe calibration.

Horizontal/Acquisition Menu

Use the Horizontal/Acquisition menu options to set the horizontal and acquisition parameters for waveforms. This menu also provides access to the Zoom controls, Autoset, FastFrame, and Fast Acquisitions.

Horizontal/Acquisition Setup Control Window (Horizontal Tab)

Access this control window from the Horiz/Acq menu Horizontal/Acquisition Setup command, or the Horiz toolbar button.

Overview

Use the Horizontal/Acquisition Setup control window to set the horizontal and acquisition parameters of waveforms. The horizontal and acquisition controls are common to all channels.
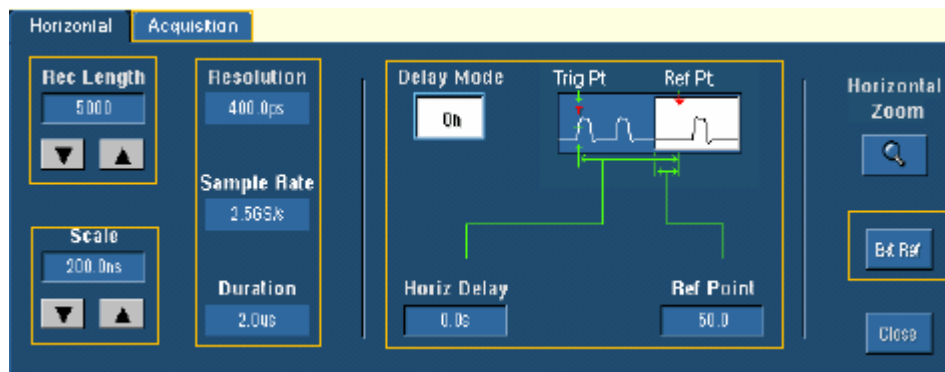
The Horizontal tab controls manage the horizontal setups such as scale and position. Normally, the horizontal axis displays time information. However, in some cases, the horizontal axis can represent frequency rather than time.

Record Length

To Use

Click Rec Length to map it to one of the multipurpose knobs and use the knob to change the record length. You can also use the increment/decrement buttons beneath the control, or the pop-up keypad. The record length specifies the number of sample points that make up the waveform record.

Behavior

The maximum record length depends on the following:

▪ The amount of memory

▪ The ability to combine the memory length of unused channels

▪ The number of frames selected in FastFrame mode

Hi Res mode requires twice the acquisition memory of other acquisition modes. When you select Hi Res mode, the instrument sets the record length accordingly to keep it from running out of memory.

Record lengths of less than 500 samples are not supported except in FastFrame, where Frame Length can be set as low as 50 samples.

When you change the record length, you can choose to hold either the sample rate or the horizontal scale constant. The remaining variables will change in response to the record length changes. See User Preferences: Record Length on page 221 to make this choice.

Horizontal Scale
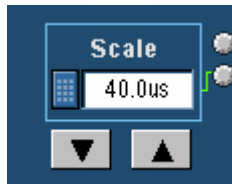
To Use

Click the Scale control to map it to one of the multipurpose knobs and use the knob to change the value. You can also use the increment/decrement buttons beneath the control or the pop-up keypad.

Behavior

Use the Scale control to adjust the time base. This control scales all live displayed waveforms (both channel and math waveforms). You can use the Horizontal Zoom control window to change the horizontal scale and position of reference waveforms within the Zoom window(s).

When the horizontal delay is off, the scale expands or contracts around the trigger point. When the horizontal delay is on, the scale expands or contracts around the horizontal reference point.

The scale units depend on the waveform type. In most cases the horizontal units will be time, but they can also be frequency.

Tip Use the dedicated front-panel HORIZONTAL SCALE knob to change the horizontal scale.

Horizontal Control Window Readouts
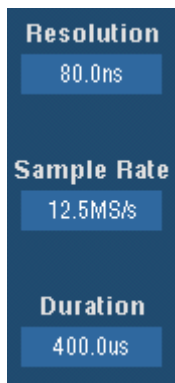
To Use

You cannot change the readouts of the Horizontal controls directly. The readout values are determined by other Horizontal control settings.

Behavior

Use the horizontal readouts to obtain a quick overview of the horizontal settings.

Resolution is similar to the acquisition interval. It can be thought of as the time between sample points. In most cases, the acquisition interval is the same as the resolution.

Sample Rate is the number of acquisition samples per second. The faster the sample rate, the more accurately the instrument can represent fine details in a fast signal.

Duration specifies the time across the 10-division screen.

Horizontal Delay and Horizontal Position

To Use

Use the multipurpose knob or the front-panel HORIZONTAL POSITION knob to set the horizontal position when the horizontal Delay Mode is Off.

When Delay is On, use the multipurpose knobs to set the Horiz Delay (Horizontal Delay) and the Ref Point (Reference Point).

Behavior

The Position control moves the trigger point within the acquired waveform. Use the control to select the amount of data that appears before the trigger event.

The Horiz Delay and Ref Point controls only appear when the Delay Mode is On. Horiz Delay enables the instrument to look for a trigger and then wait a specified time before acquiring data. This is useful when you know that the data you are interested in occurs a set time after the trigger. Use the Ref Point control to specify the amount of pretrigger data. The reference point is represented on the screen by a small inverted triangle.

🔅Tip Use the dedicated DELAY button to turn the Horizontal Delay on and off. Or, use the Horiz/Acq menu **Delay Mode On** command.

🔅Tip Toggle Horizontal Delay on and off to quickly compare signal details at two different areas of interest, one near the trigger location and the other centered at the delay time.

Horizontal Position/Scale Control Window

Access this control window from the Horiz/Acq menu Position/Scale command.

Overview

Use the Horizontal Position/Scale control window to set the horizontal position and scale of displayed waveforms. You can also use it to turn on horizontal delay so the acquisition window can be focused on events that occur at a fixed delay after the trigger point.

The dedicated controls on the front panel also perform the same functions.

For information on Delay Mode control see page 98, on Position control see page 98, and on Scale control see page 98 .

Horizontal Position/Scale Control Window: Delay Mode Off

When horizontal delay is off, the horizontal position control moves the trigger point within the acquired waveform. The horizontal scale control expands or contracts the waveform around the trigger point.

When horizontal delay is on, you can delay the acquisition relative to the trigger event. For more information on horizontal delay, see Horizontal Position/Scale Control Window: Horiz Delay.

Horizontal Position/Scale Control Window: Position

To Use

Click the Position control to map it to a multipurpose knob and use the knob to change the position.

Behavior

The horizontal position control moves the trigger point within the acquired waveform. You can select the amount of data that appears before and after the trigger event. To acquire waveforms leading up to the trigger event, set the horizontal position control to the maximum time setting. To acquire waveforms after the trigger event, set the horizontal position control to the minimum time setting.

The horizontal reference point is shown as a small inverted triangle.

Tip Use the dedicated front-panel POSITION knob to change the horizontal position.

Horizontal Position/Scale Control Window: Scale
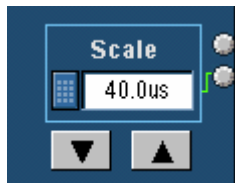
To Use

Click the Scale control to map it to a multipurpose knob and use the knob to change the scale. You can also use the increment/decrement buttons beneath the control or the pop-up keypad.

Behavior

Use the Scale control to adjust the time base. The Scale control scales all live displayed waveforms at the same time.

The scale units depend on the waveform type. In most cases the horizontal units will be time. However, the horizontal units can also be frequency.

Tip Use the dedicated front-panel HORIZONTAL SCALE knob to change the horizontal scale.

Horizontal Position/Scale Control Window: Delay On

Access this control window from the Horizontal menu.
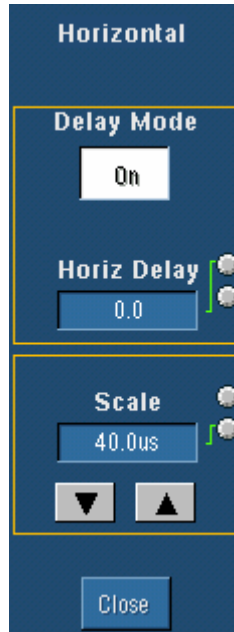
Overview

Use the Horizontal Position/Scale control window to set the horizontal delay and the horizontal scale of the displayed waveforms when the horizontal delay is on.

The dedicated controls on the front panel also perform the same functions.

For information on Delay Mode control see page 99 and on Scale control see page 100.

Horizontal Position/Scale Control Window: Horiz Delay

To Use

Click Delay Mode or push the front-panel DELAY button to activate horizontal delay. Click the Horiz Delay control, and use the multipurpose knob to set the delay time. You can also use the dedicated HORIZONTAL POSITION knob on the front panel.

Behavior

Use the Horiz Delay control to delay the acquisition relative to the trigger event. The horizontal delay feature is useful when you want to acquire waveform details that are separated from the trigger event by a significant interval of time.

For example, you can trigger the instrument on a sync pulse that occurs once every 10 ms, and then look at the high-speed characteristics that occur 6 ms after the pulse.

Tip A convenient way to use the horizontal delay is to make quick comparisons between two different points in time. Trigger the instrument on one area of interest and then set the horizontal delay to acquire the other area of interest. You can then toggle the horizontal delay on and off to compare details at the two points in time.

Horizontal Position/Scale Control Window: Scale Delay On
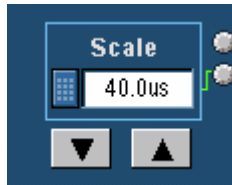
Scale
40.0us

To Use

Click the Scale control to map it to a multipurpose knob and use the knob to change the scale. You can also use the increment/decrement buttons beneath the control or the pop-up keypad.

Behavior

Use the Scale control to adjust the time base. This control scales all live displayed waveforms at the same time.

When horizontal delay is on, the scale expands or contracts around the horizontal reference point instead of the trigger point.

The scale units depend on the waveform type. In most cases the horizontal units will be time. However, the horizontal units can also be frequency.

Tip Use the dedicated front-panel HORIZONTAL SCALE knob to change the horizontal scale.

Resolution Control Window

Access this control window from the Horiz/Acq menu Resolution command.

Overview

Use the Resolution control window to select the number of samples per division for all acquired waveforms.

Duration
400.0us

Sample Rate
12.5MS/s

Resolution
80.0ns

Rec Length
5000

Close

To Use

Use the multipurpose knob to adjust the Record Length.

Behavior

The Duration, Sample Rate, and Resolution controls are readouts only. Their settings are impacted by other controls.

Duration specifies the time across the 10-division screen.

Sample Rate specifies the number of samples per second that will be acquired for each waveform. The faster the sample rate, the more accurately the instrument can represent fine details in a fast signal.

Resolution is similar to the acquisition interval. It can be thought of as the time between sample points. In most cases, the acquisition interval is the same as the resolution.

Record Length specifies the total number of samples in the waveform acquisition. The maximum record length depends on the amount of memory in the instrument.

Because the instrument can only store a finite number of waveform data points, there is a trade-off between the waveform details and the record length. You can acquire either a detailed picture of a waveform for a shorter period of time or a less detailed picture for a longer period of time.

Acquisition Mode Control Window

Access this control window from the Horizontal/Acquisition menu Acquisition Mode command.
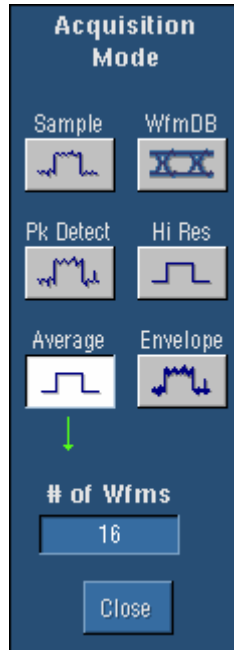
Overview

A single waveform data point can be made up of several sequentially acquired data points. The waveform data points can also be created from a composite of sampled data taken from multiple acquisitions. The acquisition modes determine how the waveform data points are produced from the sampled data.

To Use

Select the acquisition mode by clicking the appropriate button. If you use Average or Envelope mode, select the number of waveforms for that mode using the multipurpose knob. If you use Waveform Database mode, select the number of samples using the multipurpose knob.

Behavior

In Sample mode, the instrument creates a record point by saving one or more samples during each acquisition interval. Sample mode is the default acquisition mode. The instrument does no post processing of the acquired samples in this mode.

In Pk Detect (Peak Detect) mode, the acquisition alternates between saving the highest sample in one acquisition interval and the lowest sample in the next acquisition interval. Peak Detect mode only works with real-time, noninterpolated sampling.

The Hi Res (High Resolution) mode averages all samples taken during an acquisition interval to create a record point. This average results in a higher-resolution, lower-bandwidth waveform. This mode only works with real-time, noninterpolated sampling.

Model	Signal and input conditions	Effective bits
TDS5032B, TDS5052B, TDS5034B, TDS5054B TDS5054BE	1 MHz, 9.2 div, 5 GS/s sample rate, Sample acquisition mode	6.8 bits
	1 MHz, 9.2 div, 10 MS/s sample rate, Hi Res acquisition mode	9.1 bits
TDS5052B, TDS5054B, TDS5054BE	500 MHz, 6.5 div, 5 GS/s sample rate, Sample acquisition mode	6.5 bits

Model	Signal and input conditions	Effective bits
TDS5032B, TDS5034B	350 MHZ, 6.5 div, 5 GS/s sample rate, Sample acquisition mode	6.5 bits
TDS5104B	1 MHz, 9.2 div, 5 GS/s sample rate, Sample acquisition mode	6.6 bits
	1 MHz, 9.2 div, 10 MS/s sample rate, Hi Res acquisition mode	9.0 bits
	1 GHz, 6.5 div, 5 GS/s sample rate, Sample acquisition mode	4.7 bits

In Average mode, the instrument acquires and displays a waveform record that is the average result of several acquisitions. This mode reduces random noise. Use the multipurpose knob to specify the number of waveforms that you want to average.

In Envelope mode, the instrument acquires and displays a waveform record that shows the extremes in variations over several acquisitions. The instrument saves the highest and lowest values in two adjacent intervals similar to the Peak Detect mode. Unlike Peak Detect mode, the peaks are gathered over many trigger events.

After each trigger event, the instrument acquires data and then compares the minimum and maximum values from the current acquisition with those stored from the previous acquisition. The final display shows the most extreme values for all the acquisitions for each point in the waveform record.

In Waveform Database mode, the instrument acquires and displays a waveform that is the accumulation of several acquisitions. In addition to time and amplitude, the waveform displays a count of the number of times a specific sample point has been acquired. The value at each point on the waveform is a counter that reflects the hit intensity. The instrument acquires data after each trigger event using Sample mode. It then determines the location of each sample point and accumulates it with shared data from previous acquisitions. This is the only mode that can be used when doing mask testing with an optical standard. Fast Acq is inhibited in Waveform Database mode.

Horizontal/Acquisition Setup Control Window (Acquisition Tab)

Access this control window from the Acquisition tab of the Horizontal/Acquisition control window.

Overview

Use the Acquisition tab to set the acquisition setups of all live waveforms. The Acquisition controls are common to all channels. This means that you cannot acquire in Sample mode on channel 1 while channel 2 acquires in Envelope mode.
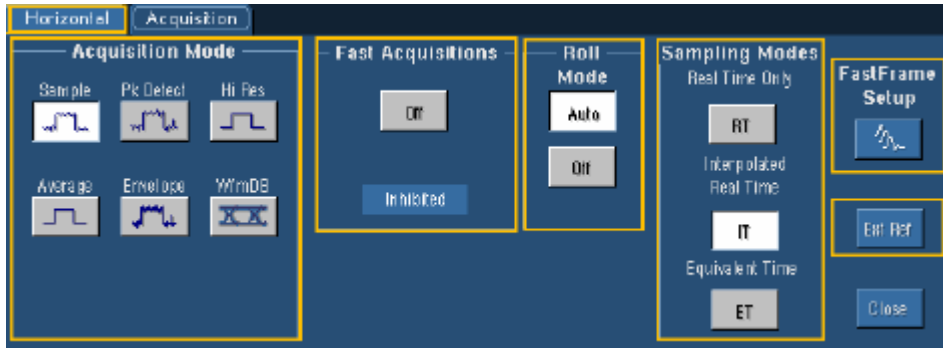
Acquisition is the process of sampling the analog input signal, digitizing the signal to convert it into digital data, and then assembling the signal information into a waveform record. The instrument creates a digital representation of the input signal by sampling the voltage level of the signal at regular time intervals. The sampled and digitized points are stored in memory along with the corresponding timing information. The instrument uses the digital representation of the signal for display, measurements, or for further processing.

Use the controls on this tab to define the sampling method, the acquisition mode, turn Fast Acquisitions on or off, access the FastFrame Setup control window, and activate an external reference.

Acquisition Mode
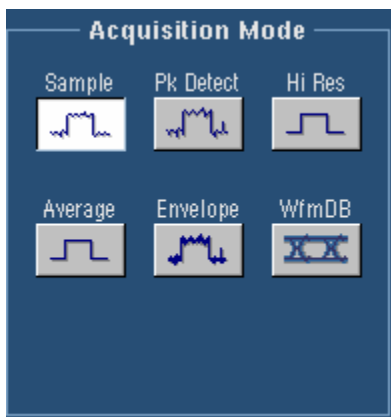
To Use

Click a button to select the acquisition mode. If you use Average or Envelope mode, select the number of waveforms for that mode using the multipurpose knob. If you use Waveform Database mode, use the multipurpose knob to select the number of samples.

Behavior

A single waveform data point can be made up of several sequentially acquired data points. The waveform data points can also be created from a composite of sampled data taken from multiple acquisitions. The acquisition modes determine how the waveform data points are produced from the sampled data.

In Sample mode, the instrument creates a record point by saving one or more samples during each acquisition interval. Sample mode is the default acquisition mode. The instrument does no post processing of acquired samples in this mode.

In Peak Detect mode, the acquisition alternates between saving the highest sample in one acquisition interval and the lowest sample in the next acquisition interval. Peak Detect mode only works with real-time, noninterpolated sampling.

The Hi Res (High Resolution) mode averages all samples taken during an acquisition interval to create a record point. This average results in a higher-resolution, lower-bandwidth waveform. This mode only works with real-time, noninterpolated sampling.

Note

A key advantage of Hi Res mode is its potential for increasing resolution regardless of the input signal. The following table indicates that you can obtain up to 15 significant bits with Hi Res mode. Note that the resolution improvements are limited to speeds slower than 40 ns/div.

Additional resolution bits

40 ns and faster	8 bits	>550 MHz
80 ns to 200 ns	9 bits	>110 MHz
400 ns to 1 μs	10 bits	>22 MHz
2 μs to 4 μs	11 bits	>5.5 MHz
10 μs to 20 μs	12 bits	>1.1 MHz
40 μs	13 bits	>550 kHz
100 μs to 200 μs	14 bits	>110 kHz
1 μs and slower	15 bits	<55 kHz

In Average mode, the instrument acquires and displays a waveform record that is the average result of several acquisitions. This mode reduces random noise. Use the multipurpose knobs to specify the number of waveforms that you want to average.

In Envelope mode, the instrument acquires and displays a waveform record that shows the extremes in variations over several acquisitions. The instrument saves the highest and lowest values in two adjacent intervals similar to Peak Detect mode. Unlike Peak Detect mode, the peaks are gathered over many trigger events.

After each trigger event, the instrument acquires data and then compares the minimum and maximum values from the current acquisition with those stored from the previous acquisition. The final display shows the most extreme values for all the acquisitions for each point in the waveform record.

Acquiring and displaying a noisy square wave signal illustrates the difference between the modes. Note how Envelope mode reduces the noise while Average mode captures its extremes:
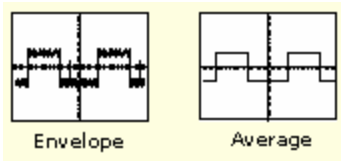

In Waveform Database mode, the instrument acquires and displays a waveform that is the accumulation of several acquisitions. In addition to time and amplitude, the waveform displays a count of the number of times a specific sample point has been acquired. The value at each point on the waveform is a counter that reflects the hit intensity. The instrument acquires data after each trigger event using Sample mode. It then determines the location of each sample point and accumulates it with shared data from previous acquisitions. This is the only mode that can be used when doing mask testing with an optical standard. Fast Acq is inhibited in Waveform Database mode.

Fast Acquisitions

To Use

Click the button under Fast Acquisitions to turn Fast Acquisitions mode on and off; this has the same effect as pushing the front-panel FastAcq button.

Behavior

Fast Acquisitions mode keeps the dead time between acquisitions very short. This improves the chances of acquiring infrequent events. When Fast Acquisitions are turned on, the front-panel FastAcq button is lighted.

If Fast Acquisitions mode is on and you attempt to activate a feature that conflicts with this mode, Fast Acquisitions mode will be inhibited. When the conflicting feature is turned off, in most cases, Fast Acquisitions will resume. The following bullet items describe some interactions:

- Measurements are supported in Fast Acquisitions mode when you display one or two channels. If you display three or more channels, Fast Acquisitions mode is turned off, and the measurements are displayed. Fast Acquisitions mode is automatically enabled when you switch back to displaying one or two channels.

- Histograms are supported in Fast Acquisitions mode when you display one or two channels. If you display three or more channels, Fast Acquisitions mode is turned off, and the histograms are displayed. Fast Acquisition mode is automatically enabled when you switch back to displaying one or two channels.

- Deskew is supported in Fast Acquisitions mode when you acquire one or two channels. If you acquire three or more channels, Fast Acquisition mode is turned off and the message, Deskew not applied in FastAcq when three or more channels are acquired, is displayed. When you switch back to acquiring one or two channels, the message, Deskew is re-enabled, is displayed and deskew values are applied again.

Tip Use the Horiz/Acq menu **Fast Acquisitions** selection to directly toggle this mode on or off.

Roll Mode

Roll Mode

Auto

Off

To Use

To enable Roll mode, click Auto on the Acquisition tab of the Horizontal/Acquisition control window. Roll mode then starts automatically when the resolution is equal to or greater than 2 ms, the acquisition mode is Sample or Peak Detect, and 1 or 2 channels are on. For Hi Res mode, Roll mode starts when the resolution is 4 ms or greater. The resolution appears near the lower-right corner of the graticule and is controlled by the RESOLUTION knob. Use untriggered roll mode to continuously observe a slow process. Apply math and measurement operations after you stop the acquisition.

When Roll mode is on, the Acquisition Status Indicator Readout displays Roll.

Tip Use the Horiz/Acq menu Roll Mode Auto command to directly toggle this mode.

Behavior

Roll mode lets you see acquired data without waiting for the acquisition of a complete waveform record. It gives you immediate feedback. For slow time base settings, Roll mode provides a rolling display similar to a strip-chart recorder.

When using Roll mode, you do not have to wait for slow acquisitions to fill the display. When Roll mode is enabled, the instrument displays newly acquired data points at the right edge of the display while moving the older waveform data points to the left.

Using single sequence with roll mode causes the acquisition to automatically stop after a complete waveform record is acquired. Roll mode is useful when observing a slow process where you always want to see the most recent view of that process. Roll mode with single sequence is useful to save data for later viewing. For example, at 20 seconds per division and 30 K record length, there are 12000 seconds of acquired data points.

The resolution, record length, horizontal scale, and number of active channels are dependent variables. Therefore, the fastest horizontal scale setting when Roll mode starts is 40 ms/div, the record length is 500 (resolution is 200 µs), the acquisition mode is either Sample or Peak Detect, and 1 or 2 channels must be on. As the record length or the number of on channels grows, the time per division required to enter Roll mode is slower as shown in the following table.

1 or 2 Channels On	Resolution	Record Length	Horizontal Scale	Sample or Peak Detect	Hi Res
	2 ms	500	100 ms/div	Yes	No
	2 ms	2000	400 ms/div	Yes	No
	2 ms	5000	1 s/div	Yes	No
	2 ms	50000	10 s/div	Yes	No
	2 ms	100000	20 s/div	Yes	No

4 ms	500	200 ms/div	Yes	Yes
4 ms	2500	1 s/div	Yes	Yes
4 ms	5000	2 s/div	Yes	Yes
4 ms	50000	20 s/div	Yes	Yes
4 ms	100000	40 s/div	Yes	Yes
3 or 4 Channels On				
4 ms	500	200 ms/div	Yes	No
4 ms	2500	1 s/div	Yes	No
4 ms	5000	2 s/div	Yes	No
4 ms	50000	20 s/div	Yes	No
4 ms	100000	40 s/div	Yes	No
8 ms	500	400 ms/div	Yes	Yes
8 ms	5000	4 s/div	Yes	Yes
8 ms	25000	20 s/div	Yes	Yes
8 ms	50000	40 s/div	Yes	Yes

The maximum record length where Roll mode is allowed is 8 M.

Interactions of Roll Mode and Other Instrument Settings

Roll mode can interact with other instrument settings:

- If Roll mode is on and you turn on Fast Acquisitions, Roll mode will be suppressed until you turn Fast Acquisitions off.

- If Roll mode is on, math waveforms and measurements are updated only after acquisition is stopped (when you manually stop an acquisition). If you stop an acquisition before a record is filled, the math waveform or measurements will be calculated on the partial data.

- Roll mode is only valid for real-time acquisition modes such as Sample, Peak Detect, and Hi Res. Roll mode is ignored in Average or Envelope acquisition modes.

- Roll mode is suppressed when FastFrame is on.

- Deskew values are ignored when Roll mode is on.

- If you attempt to save a waveform when Roll mode is on, only the waveform points in the acquisition memory are saved.

Incompatible Acquisition Features

The following table shows which acquisition features and modes are incompatible with other features or modes:

Incompatible with	Control/Feature	Explanation
Reference Waveforms (Ref1-Ref4)	Voltage Offset	Offset is an acquisition control, not a display control
Single shot acquisition	Average	Acquisitions continue until the specified number of waveforms have been acquired and averaged
Single shot acquisition	Envelope	Acquisitions continue until the specified number of waveforms have been acquired for the enveloped waveform
Measurements	Roll	Measurements are not available until you stop acquisitions
Envelope, Average, or WfmDb acquisition modes	Roll	Using these acquisition modes will turn Roll mode off

Sampling Modes
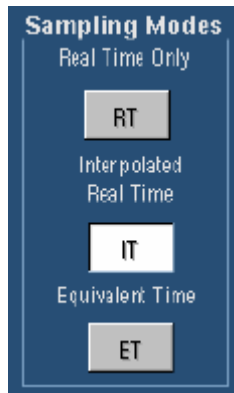
To Use

To enable a sampling mode, click the Sampling Mode button. There are three sampling modes: Real Time Only, Interpolated Read Time, and Equivalent Time.

In Real Time Only sampling, the instrument acquires all the points in a waveform record after one trigger event.

In Interpolated Real Time sampling, the instrument interpolates between the samples it acquires when it cannot acquire all the real samples it needs to fill up its waveform record. The instrument can interpolate either as part of an acquisition or as part of the zoom/display behavior.

Equivalent time sampling starts when the time base setting (horizontal scale) is too fast to let the instrument get enough samples with which to create a waveform record using real-time sampling.

Tip Use the Horiz/Acq menu **Sampling Modes** command to directly select a sampling mode.

Note

In real-time sampling, the instrument digitizes all of the points it acquires after one trigger event. Always use real-time or interpolated sampling to capture single-shot or transient events.

Behavior

Equivalent Time mode lets you accurately acquire repetitive waveforms at sample rates much higher than the base analog-to-digital converter sample rate. Equivalent time sampling is generally not useful for acquiring time-varying, random, or single-shot events.

The instrument takes a few samples with each trigger event and eventually obtains enough samples to construct a waveform record. The instrument takes samples independent of the trigger position and displays them based on the time difference between the sample and the trigger. You should only use Equivalent Time mode on repetitive signals.
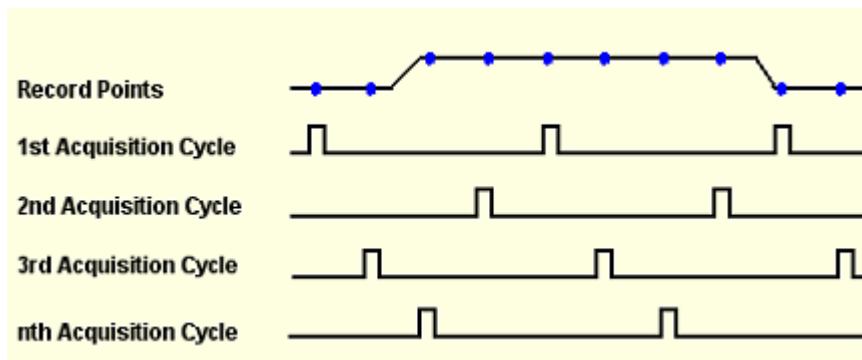
The sampling speeds and the number of channels that you choose affect the mode that the instrument uses to sample waveforms in the following ways:

- The instrument always uses real-time sampling at slower time base settings. Faster time settings force the instrument to switch from real-time sampling to equivalent-time sampling.

- The instrument extends the limit at which it must switch from real-time sampling by using the digitizers of channels that are turned off to sample the channel or channels that are turned on.

The following tables show the time base setting(s) at which the switch from real-time sampling (RT) to equivalent-time sampling (ET) occurs for your instrument.

Sampling mode selection TDS5000B

Channels on	1	2	3 or 4
Time base			
≥ 800 ps/pt	RT	RT	RT
400 ps/pt	RT	RT	ET
200 ps/pt	RT	ET	ET
≤ 100 ps/pt	ET	ET	ET

Sampling mode selection TDS5000BE

Channels on	1	2	3 or 4

Time base			
≥ 800 ps/pt	RT	RT	RT
400 ps/pt	RT	RT	ET
200 ps/pt	RT	ET	ET
≤ 100 ps/pt	ET	ET	ET

The type of equivalent-time sampling the instrument uses is called random equivalent-time sampling. Although it takes the samples sequentially in time, it takes them randomly with respect to the trigger. Random sampling occurs because the instrument sample clock runs asynchronously with respect to the input signal and the signal trigger. The instrument takes samples independently of the trigger position and displays them based on the time difference between the sample and the trigger.

FastFrame Setup Control Window

Access the FastFrame Setup from the Acquisition tab of the Horizontal Setup control window.

Control Window Overview

Use these controls to capture a series of triggered acquisitions at high resolution without wasting acquisition memory on the dead time in-between the trigger events.
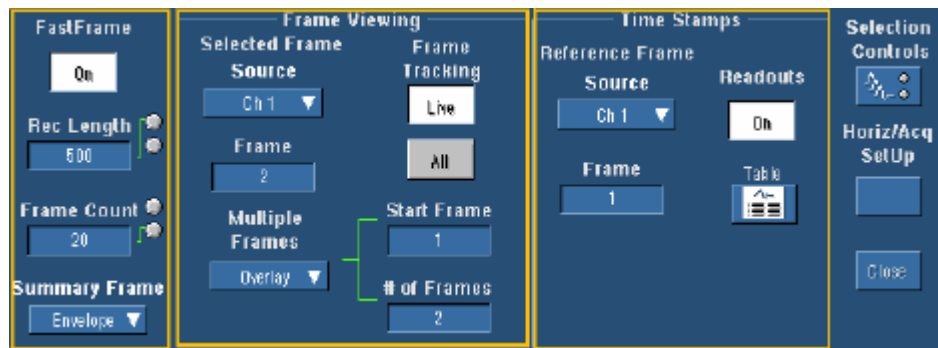
FastFrame mode combines multiple acquisitions into one large record. By carefully defining the trigger conditions, you can use FastFrame to capture only multiple events of the waveform that you are interested in. Each captured event is stored sequentially in its own frame. The frames are in the order they were captured. FastFrame mode lets you jump to and view the desired frame. Time Stamps can display the absolute trigger time for a specific frame and the relative time between triggers of two specified frames. FastFrame lets you compare different waveforms. Math is done on each frame.
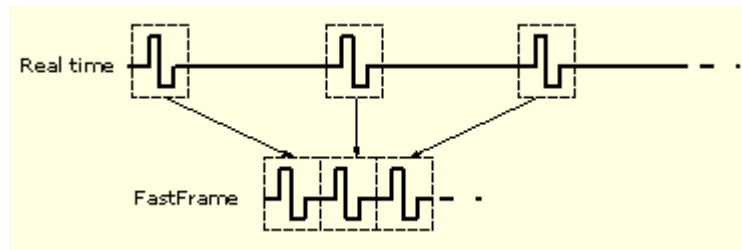
FastFrame provides the following features:

- Measure time between triggered acquisitions using time stamps in Sample, Peak Detect, and Hi Res mode.

- Save and recall FastFrame waveforms.

- Display waveforms in YT mode.

- Apply math functions to FastFrame waveforms.

- Confine automatic measurements only to the displayed frame.

- Overlay multiple frames for easy comparison.

  FastFrame is not compatible with the following features or modes:

- Equivalent Time

- Histograms

- Fast Acquisitions

- Average

- Envelope

Frame Setup

To Use

Click the FastFrame button to turn the FastFrame mode on and off. Use the multipurpose knobs to select the number of samples per frame and the number of frames.

Click the Selection Controls button to display only the Selected Frame controls and the Reference Frame controls adjacent to the full-screen graticule.

Select Average or Envelope from the Summary Frame list to replace the last frame with one that is the computed average or envelope of all frames in the set.

Behavior

The Frame Length is the record length and defines the number of samples in each frame.

The Frame Count defines the number of frames to store in the acquisition memory of a single channel. The product of the frame length and the frame count is constrained within the available memory. If more frames are needed, the instrument will automatically decrease the record length; if more samples are needed, the instrument will automatically decrease the frame count.

Note

Because of overhead associated with each frame, the maximum number of frames that can be obtained from a given record length is less than what is theoretically calculated.

Frame Viewing

To Use

Select the source channel from the drop-down list, and then use the multipurpose knob to select the Selected Frame. Click either the Live or All button to determine which frames you want to lock together.

Click the Multiple Frames button to toggle this function between Off or Overlay. When Overlay is selected, two additional fields are displayed—Start Frame and # of Frames. These fields allow you to select a range of frames that are displayed in the color of the Source channel.

In Overlay, the frames are superimposed over each other. The Selected Frame is displayed in blue unless you are using the Spectral or Temperature palette, then selected waveform is displayed in the same color as the other frames.

(You may find it difficult to distinguish the dark blue Selected Frame if you have selected Spectral or Temp for the Color Palette.)

Behavior

The Selected Frame displays the frame that you want to view. You can use the multiview zoom features of the instrument to magnify the waveform of interest. The Selected Frame can use any Channel, Math, or Reference waveform as the source waveform.

When Multiple Frames is off, the instrument will display one frame at a time. You can use the Selected Frame control to scroll through all of the frames until you find the frame of interest.

When you set Frame Tracking to Live, the channel and math waveforms are locked together. Adjusting a channel waveform also adjusts a related math waveform. All reference waveforms are locked together, but they are separate from channel and math waveforms.

When you set the Selected Frame Source Ch1 to Frame 3, then the Selected Frame Ch2, Ch3, Ch4, Math1, Math2, Math3 and Math4 are also set to Frame 3. When you set the Selected Frame Source Ref1 to Frame 2, then the Selected Frame Ref2, Ref3, and Ref4 are also set to Frame 2. If the Frame Tracking is set to Live, changing Selected Frame Ch1 will not affect the Selected Frame Ref1 frame or the Reference Frame setting. (Two-channel instruments are limited to only two channels.)

To lock the reference and all selected frames together, use the All button under Frame Tracking. When the frames are locked, they maintain the same relative distance from each other as you adjust either frame.

For example: assume that Selected Frame Ch1 is set to Frame 3, Ref1 is set to Frame 4, Reference Frame is set to Frame 2 and Frame Tracking is set to All. If you change Selected Frame Ch1 to Frame 4, then Ref1 will change to Frame 5 and the Reference Frame will change to Frame 3 maintaining the relative distance between frames.

When you select Overlay, the Start Frame and the # of Frames fields are displayed. Use the upper multipurpose knob to enter the Start Frame (first frame) in the frame sequence that you want to view. Use the lower multipurpose knob to enter the total number of frames that you want to view. For example, if you have 1000 frames and you want to view frames 75 through 100, use the upper multipurpose knob to enter 75 in the Start Frame field and the lower multipurpose knob to enter 26 in the # of Frames field. The result of these settings is that frames 75 through 100 are displayed superimposed over each other in the color of the selected Source, and the current (selected frame)

is displayed superimposed in dark blue, unless you are using the Spectral or Temperature color palette.

Time Stamps

To Use

Use these controls to use time stamps with FastFrame. Click the Readouts button to display the time stamp readouts in the graticule. Click the Table button to display the time stamps readouts in a table format. Select the Reference Frame Source from the drop-down list, and then enter the Reference Frame with the multipurpose knobs.

Behavior

All other frames are referenced to the Reference Frame when using time stamps. The Reference Frame is used to calculate time differences for each frame and displays those differences in the graticule.

When time stamp readouts are turned on, readouts are displayed for the Selected Frame, the Reference Frame, and the delta (difference between the two frames). The time stamps are displayed in a format that shows the date and time.

The Time Stamps Table displays the current time stamps and readout information. This table is read-only.

Time Stamp Table

Access this control window by clicking the Table button in the FastFrame Setup control window.

To Use

The Time Stamp table is read-only. Select the source channel from the Selected Source drop-down list, and then use the multipurpose knob to select the Selected Frame. Select the Reference Frame Source from that drop-down list, and then use the multipurpose knob to select the reference frame. Click the FastFrame Setup button to return to the FastFrame Setup control window.

Behavior

The Time Stamps Table displays the current time stamps and readout information based on the selected frame source and reference frame source.

Using FastFrame Acquisitions

Consider the following operating characteristics when using FastFrame:

- You can push RUN/STOP to terminate a FastFrame sequence. If any frames were acquired, they are displayed. If no frames were acquired, the previous FastFrame waveform is displayed.

- Because FastFrame introduces additional processing time into the operation cycle of acquire, process, and display, it is best to use Single Sequence Acquisition (see Acquire menu, Stop After menu). With Single Sequence selected, you will see the current acquisition sequence; otherwise, the display lags the current sequence by one sequence. You can also see the current sequence by pushing the RUN/STOP button to stop the acquisition.

- FastFrame greatly reduces the minimum trigger holdoff time, enabling you to acquire and store records at rates of up to 225,000 captures per second (until the number of frames requested has been acquired). FastAcq mode also provides exceptionally fast acquisition rates. Both modes have advantages. FastFrame preserves the actual data associated with each trigger event for further analysis or visual inspection, but only acquires at the fast rate for short bursts of time. FastAcq mode combines the data from all trigger events into a single pixel map, but can acquire at exceptionally fast rates continuously, not just for short bursts.

- In single sequence acquisition mode, statistics are calculated using all frames in the current acquisition except for Average and Envelope frames. If the currently selected frame is the Average or Envelope Frame, measurements are taken on the frame but statistics are still based on all frames excluding the Average and Envelope frames. Statistics reset with each new single sequence acquisition.

- On a reference waveform, statistics are derived from all frames in that reference waveform when Fast Frame is on. If Fast Frame is off, the statistics reflect only the first frame that is displayed.

FastFrame Controls

Access the FastFrame Controls by clicking the Selection Controls button in the FastFrame Setup control window.
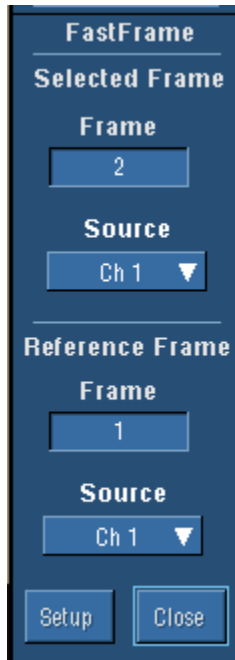
Overview

Use this control window to select the viewing frame (Selected Frame) when FastFrame is turned on. You can also select the reference frame.

To Use

Select the source channel from the drop-down list, and then use the multipurpose knob to select the Selected Frame. Select the Reference Frame Source from the drop-down list, and then use the multipurpose knob to select the reference frame.

Behavior

The Selected Frame displays the frame of interest. The Selected Frame can use any Channel, Math, or Reference waveform as the source waveform. The instrument will display one frame at a time. You can use the Selected Frame control to scroll through all of the frames until you find the frame of interest.

Use the Reference frame to establish a reference frame when using time stamps. You can view the time stamps by turning on the readouts.

Autoset

Use Autoset to automatically set up the front-panel controls based on the characteristics of the input signal. This is much faster and easier than manually setting up the controls.

Note

Autoset may change vertical position to position the waveform appropriately. It always sets vertical offset to 0 V.

If you use Autoset when one or more channels are displayed, the instrument selects the lowest numbered channel for horizontal scaling and triggering. The instrument individually scales all displayed vertical channels. If you use Autoset with no vertical channels displayed, the instrument turns on and scales channel 1.

To use Autoset, push the AUTOSET button on the front panel. You can also select Autoset from the Horizontal/Acquisition menu.

🟡Tip The Autoset Undo button opens in a control window after the instrument completes the operation. Click this button to undo only the parameters adjusted by the last Autoset.

Note

The Autoset Undo control window remains on the screen until you open another control window. After Autoset Undo closes, you can still undo the last Autoset by selecting the Undo Last Autoset command from the Edit menu. Although the last Autoset is immediately undone, the Autoset Undo control window does not open again.

Tip Select User Preferences from the Utilities menu to display the Prompt Before Action window. Click Autoset to toggle between OFF and ON:

- OFF to immediately perform an Autoset when the AUTOSET button is pushed.
- ON to display a prompt when the AUTOSET button is pushed.

Tip Click Close to save your selection.

The following table lists the Autoset defaults.

Control	Changed by Autoset to
Selected Channel	Numerically lowest displayed channel
Acquisition Mode	Sample
RUN/STOP	RUN
Deskew, Channel/Probe	Unchanged
Display Style	Vectors
Display Intensity—Overall	If less than 50%, set to 100%
Display Format	YT
Horizontal Position	Centered within the graticule window
Graticule Setting	Unchanged
AutoBright	On
Horizontal Scale	As determined by the signal frequency, or 100 µs/div
Record Length and Sample Rate	When a new time-per-division is calculated, the sample rate changes to maintain the current record length. If the sample rate cannot be changed because of hardware limitations, the sample rate is set to maximum while the record length is reduced to maintain the time-per-division.
Trigger Position	Delay Off
Trigger A >B Sequence	A Only
Main Trigger Type	Edge
Trigger Source	Numerically lowest displayed channel (the selected channel)
Trigger Level	50% of selected channel's amplitude
Trigger Slope	Positive

Trigger Coupling	DC
Trigger Holdoff	Minimum setting
Vertical Scale	As determined by the signal level
Vertical Position	As determined by the number of active channels
Vertical Coupling	DC unless AC was previously set. AC remains unchanged.
Vertical Bandwidth	Full
Vertical Offset	0 volts
Zoom	Off

Trigger Menu

Use the Trigger menu to set up the instrument to trigger on live waveforms. Several different types of triggering are available.

Trigger Setup Control Window

Access this control window from one of the Trigger Setup commands on the Trig menu or click the Trig taskbar button.

Overview

Use the Trigger Setup control window to set up the instrument to trigger on live waveforms.
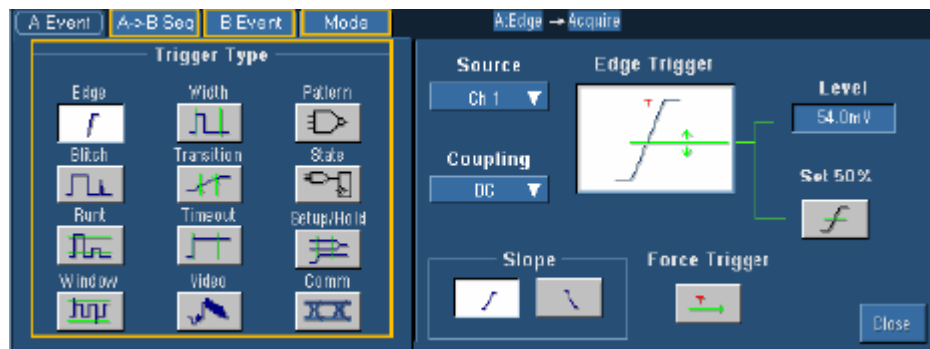
The left side of the control window contains information on the main trigger types (also known as the A Event triggers), sequential triggering, the B Event trigger, and the Trigger Mode and Holdoff. When you click one of the buttons on the left side of the control window, the right side of the control window displays details on the trigger type.

The trigger expression near the top right side of the trigger window provides a one-line summary of the conditions that must be fulfilled before the instrument will trigger.

The trigger event establishes the time-zero point in the waveform record. All points in the record are located in time with respect to the time-zero point. The instrument continuously acquires and retains sample points to fill the pretrigger portion of the waveform record. When a trigger event occurs, the instrument starts acquiring samples to build the posttrigger portion of the waveform record. Once a trigger is recognized, the instrument will not accept another trigger until the acquisition is complete and the holdoff time has passed.

Note

If you select an inverted channel waveform as a trigger source, the normal waveform is used instead.

118

A Event Trigger Types
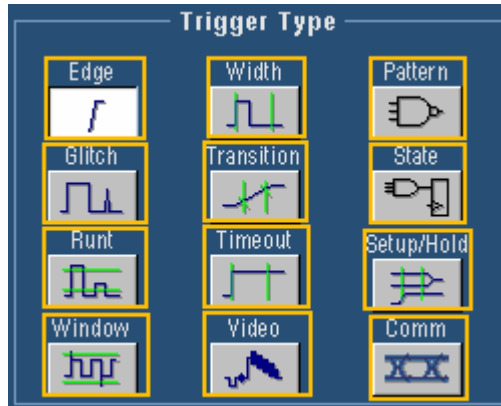
To Use

Click the Trigger Type button you want to use. Set the parameters for that trigger type in the right side of the control window.

Behavior

When you click one of the trigger icons, the right side of the screen displays the controls needed to use that trigger type.

The instrument provides the following trigger type categories:

- **Edge** triggers are the most commonly used trigger type. You can use edge triggers with analog and digital signals. Edge triggers are available on both the Main and Delayed triggers.

- **Pulse** triggers are special-purpose triggers that are primarily used with digital signals. The following classes of pulse triggers are available: Glitch, Runt, Window, Width, Transition, and Timeout. Pulse triggers are only available on the main trigger.

- **Logic** triggers are special-purpose triggers that are primarily used with digital logic signals. Two of the classes, Pattern and State, trigger the instrument based on the Boolean operator you select for the trigger sources. A third class, Setup and Hold, triggers when data on one trigger source changes state within the setup and hold times that you specify relative to a clock on another trigger source. Logic triggers are only available on the main trigger.

- **Video** triggers are used to trigger the instrument on specified fields or lines of a video signal. You can use one of several preset video signal formats or set a custom format.

- **Communication** triggers are designed to work with communication masks and standards.

Tip Use the Trig menu **Quick Select** submenu to directly access the A Event triggers.

Sequential Triggering (Horizontal Delay On)
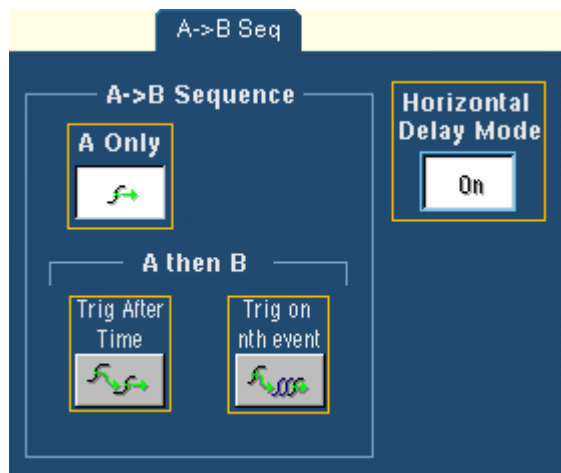
To Use

Select A-B Seq to use the A Event (main) trigger with the B Event trigger to capture complex data. The A Event trigger arms the trigger system. The instrument then triggers on the B Event trigger defined by events or time. You can also set up the instrument to delay the B Event Trigger by a specified time period by turning horizontal delay on.

Behavior

When you click one of the trigger icons on the display, the right side of the screen shows the trigger sequence. Set the individual controls using the multipurpose knobs.

Sequential Triggering (Horizontal Delay Off)
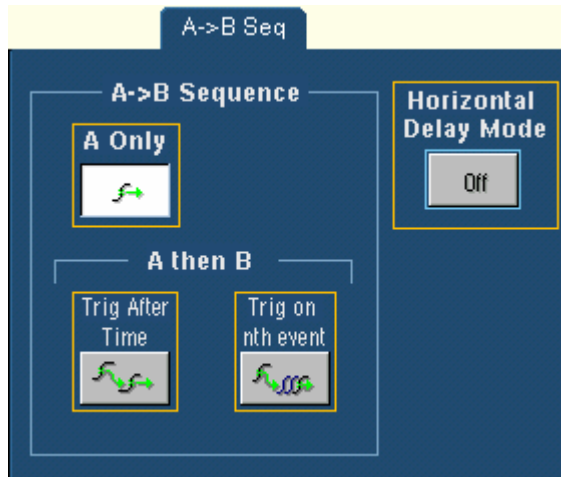
To Use

Select the A-B Seq tab to use the A Event (main) trigger with the B Event trigger to capture complex data. The A Event trigger arms the trigger system; the instrument will trigger on the B Event trigger defined by events or time.

Behavior

When you click one of the trigger icons on the display, the right side of the screen shows the trigger sequence. Set the individual controls using the multipurpose knobs.

B Event Trigger

To Use

Use these controls to define the parameters for the B Event trigger.
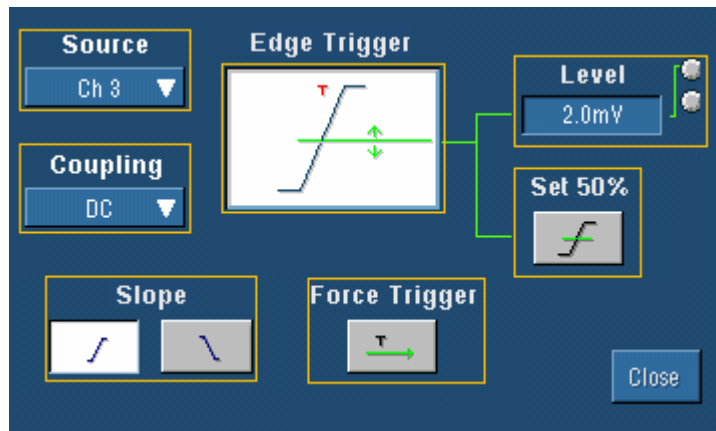
Behavior

These controls work together to define the parameters for the trigger event. Use the B Event trigger with the A Event trigger to capture more complex events. The B Event trigger uses Edge triggering only. The B Event trigger controls are independent from the A Event trigger.

Trigger Mode and Holdoff

To Use

To set the Trigger Mode, click either Auto or Normal. To set the Trigger Holdoff time manually, click Time, and then use the multipurpose knob with the Trig Holdoff control to set the holdoff time. The Holdoff setting range is 1.5 µs (minimum holdoff available) to 12 seconds (maximum holdoff available). To enable E-mail on Trigger click On. To setup the e-mail on event parameters, click Setup. Set the A Event trigger level using the A Trig Level control and the multipurpose knob.
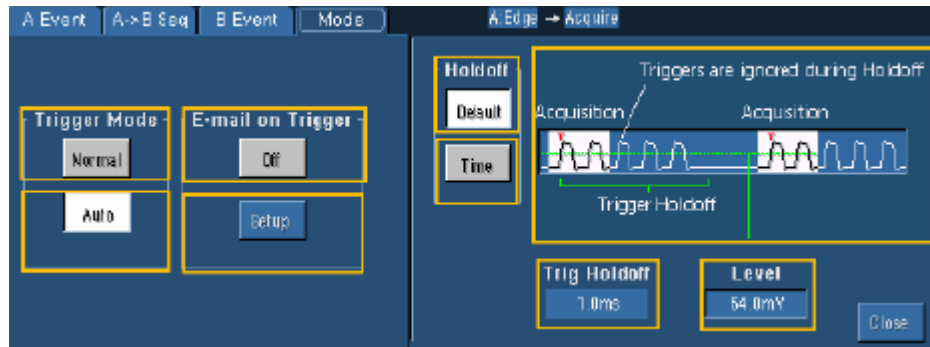
Behavior

Use the holdoff time to delay the trigger system for a specified amount of time between triggers. You can set the holdoff time to a value that synchronizes the instrument with an incoming signal to generate a stable display and avoid triggering on non-desired events.

If E-mail on Event has already been set up, clicking E-mail on Trigger enables you to send e-mail on a trigger. If E-mail on Event has not been set up, you must first click Setup, and then configure E-mail on Event in the setup control window.

Edge Trigger Control Window

To Use

Use these controls to set the Edge trigger parameters.
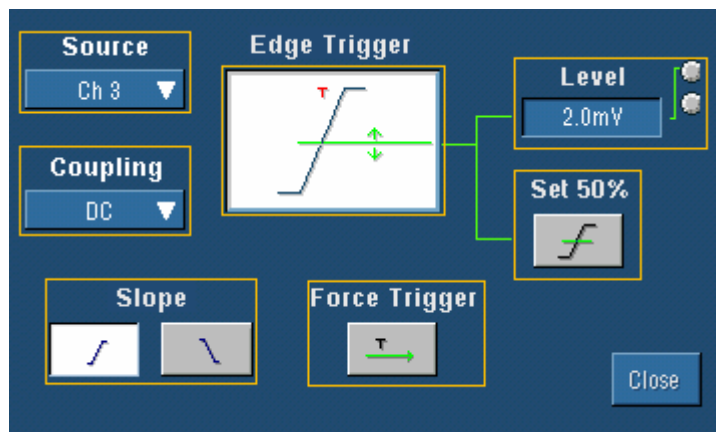
Tip Select the edge trigger by pushing the EDGE button on the front panel.

Behavior

Edge triggering is the most common way to trigger an instrument. The trigger occurs when the trigger source passes through a specific level on the specified slope (rising or falling).

Glitch Trigger Control Window

To Use

Use these controls to set the Glitch trigger parameters.
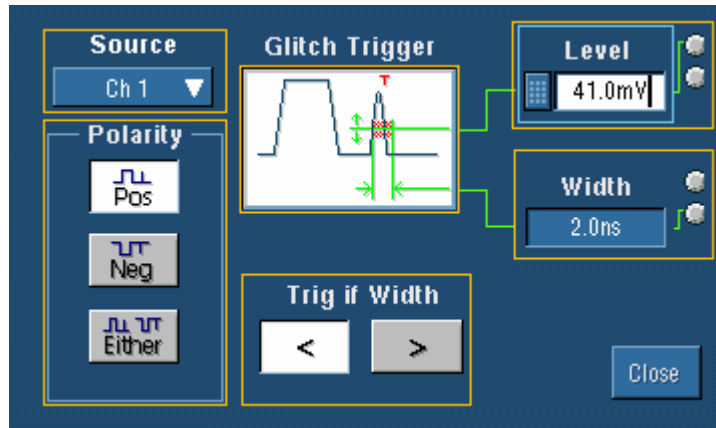
Behavior

Use the Glitch trigger to detect and trigger on a glitch or to ignore a glitch. Glitches are defined as pulses that are narrower than a specified time. The trigger occurs when the instrument detects a pulse outside (either less than or greater than) the parameters you specify.

Width Trigger Control Window

To Use

Use these controls to set the Pulse Width trigger parameters.
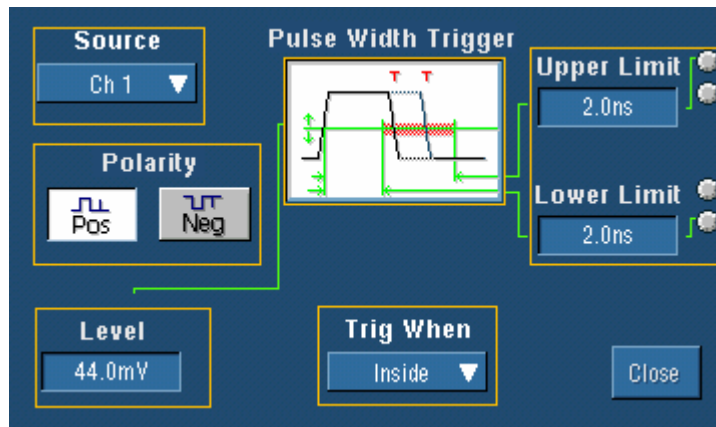
Behavior

Use the Width trigger to trigger the instrument on a pulse that is either within or outside the specified range. The range is defined by the Upper Limit and Lower Limit controls. You can

specify widths for either positive or negative pulses. The pulse widths are measured at the voltage level specified by the Level control.

Runt Trigger Control Window

To Use

Use these controls to set the Runt trigger parameters.
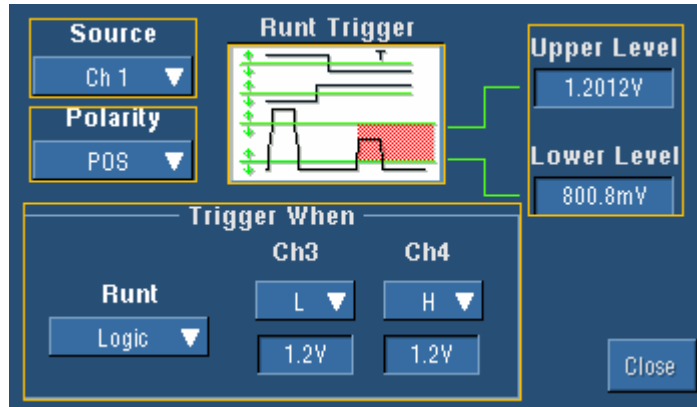
Behavior

Use the Upper Level and Lower Level threshold controls to set the voltage ranges that define a valid pulse. Runt pulses are short pulses that cross one threshold but fail to cross a second threshold before crossing the first threshold again. The pulse widths are measured at the voltage level specified by the Level control. These pulses can also be qualified by time or by the logical state of other channels.

Timeout Trigger Control Window

To Use

Use these controls to set the Timeout trigger parameters.
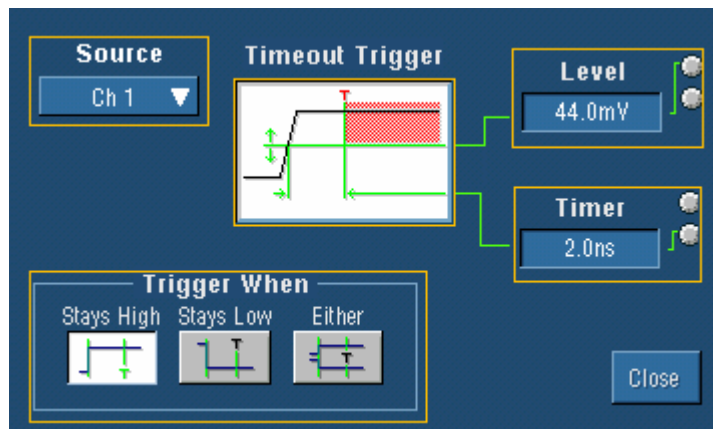
Behavior

Use the Timeout trigger to trigger the instrument on pulses longer than the specified time. The instrument will trigger at the end of the timeout period. The instrument can trigger on high, low, or both high and low pulses.

Use the Timer control to specify the timeout limit; any pulses wider than the limit will trigger the instrument. The timeout limit is measured at the voltage level specified by the Level control.

Transition Time Trigger Control Window

To Use

Use these controls to set the Transition Time trigger parameters.
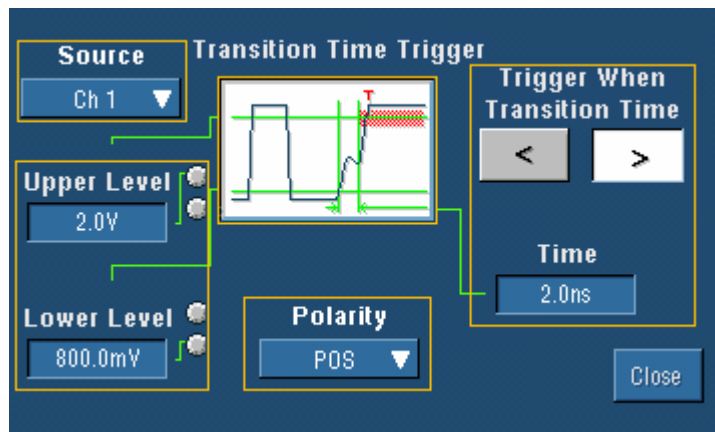
Behavior

Use the Transition trigger to trigger the instrument on pulse edges that traverse between two thresholds at faster or slower rates than the specified time. You can set up the instrument to trigger on positive or negative edges.

Use the Upper Level and Lower Level controls to define the voltage ranges of interest. The Time control specifies the target transition time.

Setup and Hold Trigger Control Window

To Use

Use these controls to set the Setup and Hold trigger parameters.
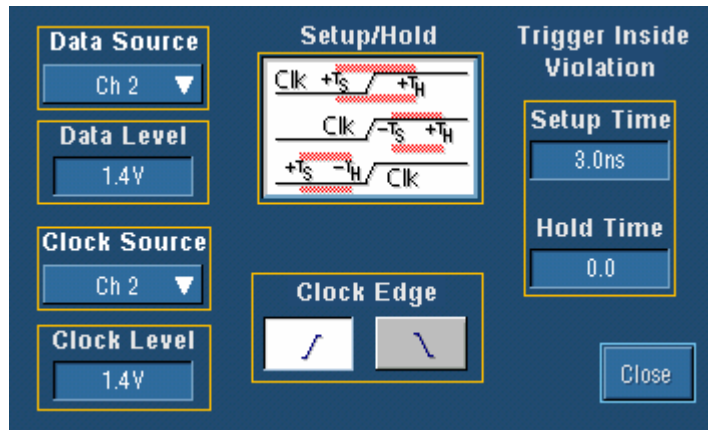
Behavior

Use the Setup and Hold trigger to trigger on setup and hold time violations. The setup and hold times define a violation region relative to a clock. Data is considered to be invalid in the violation region. The instrument monitors the data and clock sources. When a clock edge occurs, the instrument checks the data channel for transitions within the violation region. If any violations occur, the instrument triggers with the trigger point located at the clock edge. The setup and hold values can be positive or negative. The minimum setup and hold time is 2 ns. Negative setup and hold times skew the violation region to locate it before or after the clock edge. The instrument can detect and trigger on violations of time ranges that are before or after the clock.

Use one of the channel inputs as the clock signal and a second channel input as the data input. The clocking and data levels are used to determine if a clock or data transition has occurred.

Logic Pattern Trigger Control Window

To Use

Use these controls to set the Pattern trigger parameters.
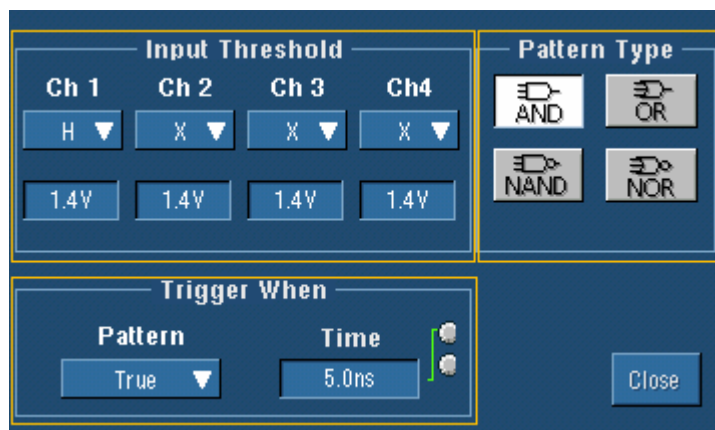
Behavior

Use the Pattern trigger to trigger the instrument when the logic inputs cause the selected function to become True or False. You can also time qualify the trigger—specify that the logic conditions must be satisfied for a specific amount of time before the instrument triggers.

The instrument input channels represent the data inputs. You can set each channel to be logic high, logic low, or "don't care". Use the multipurpose knobs to change the threshold voltages for logic levels other than TTL.

The Pattern Type buttons set the Boolean logic function for the trigger.

Note

The Ch3 and Ch4 controls are only available on 4-channel models.

Logic State Trigger Control Window

To Use

Use these controls to set the State trigger parameters.
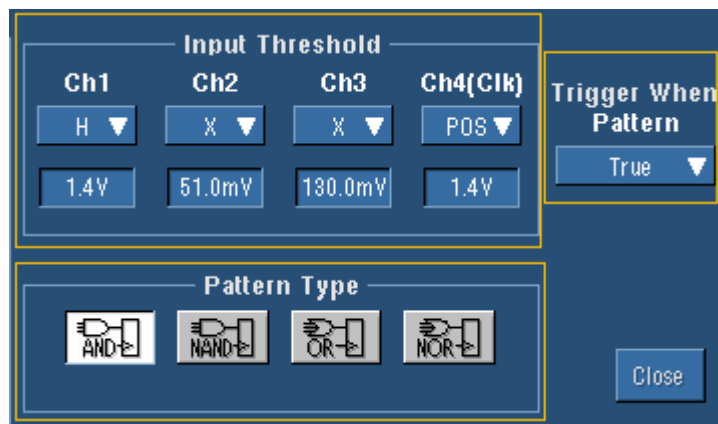
Behavior

Use the Logic State trigger to trigger the instrument when all of the logic inputs to the selected logic function cause the function to be True or False when the clock input changes state.

Channels 1, 2, and 3 represent the data inputs, and channel 4 represents the clock input. (The clock input is channel 2 on the 2-channel models.) You can set each data channel to be logic high, logic low, or a "don't care" value. You can set the channel 4 input to a rising or falling clock edge. Use the multipurpose knobs to change the threshold voltages for logic levels other than TTL.

The Pattern Type buttons set the Boolean logic function for the trigger.

Window Trigger Control Window

To Use

Use these controls to set the Window trigger parameters.
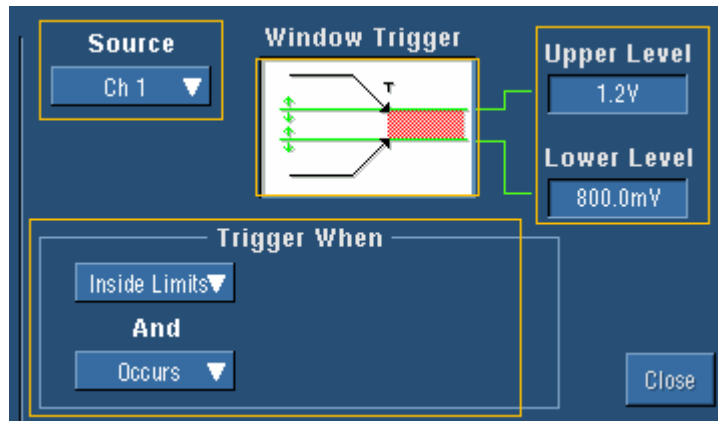
Behavior

Use the Window trigger to trigger the instrument when the input signal rises above an upper threshold level or falls below a lower threshold level. After setting these levels, you can specify whether you want to trigger the instrument as the signal is entering or leaving the threshold window. You can further qualify the trigger event in terms of time by using the Trigger When Wider option, or by the logical state of other channels using the Trigger When Logic option.

Note that the Trigger When controls change as you select different combinations of options.

Note

The Trigger When Logic option is only available on 4-channel models.

Video Trigger Control Window

To Use

Use these controls to set the Video trigger parameters.
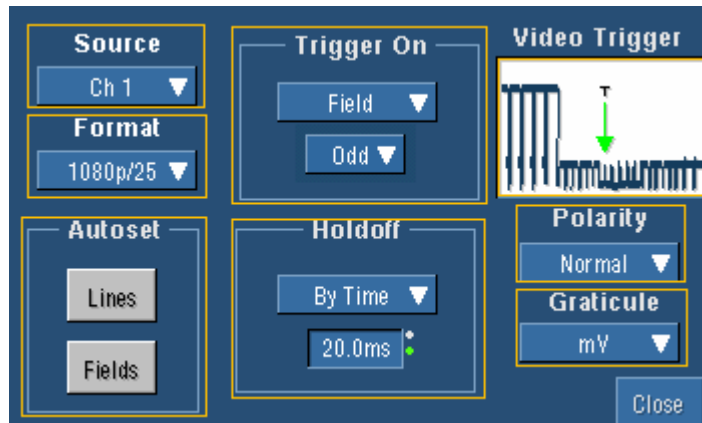
Behavior

Use the Video trigger to trigger the instrument on specified fields or lines of a composite video signal.

Note

Only composite signal formats are supported. Graphic display formats such as RGB and VGA are not supported.

Select from several preset video signal formats or set a custom format. If using a preset format, the Lines and Fields Autoset buttons will optimize the instrument for viewing the fields or lines for your chosen format. You can also select a graticule and set the polarity of the scan.
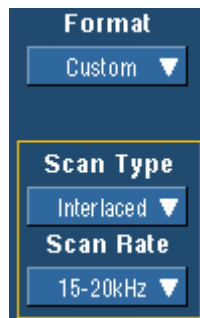
If you select the Custom signal format, no Autoset options are available. You must set the Scan Type and Scan Rate. Also note that you will only be able to specify a value for the Holdoff control in units of time. Click on the graphic to the left for more information on the Scan Type and Scan Rate controls.

Comm Trigger Control Window

Note

Option SM must be installed to have access to Comm triggers.

To Use

Use the controls in the window to select the source of the signal, the coding format, and the standard to use in testing. Bit rates can be changed using the multipurpose knobs as well. Depending on which coding format you use, you can also change the Upper, Lower, and Clock level thresholds and the pulse form using the multipurpose knobs.
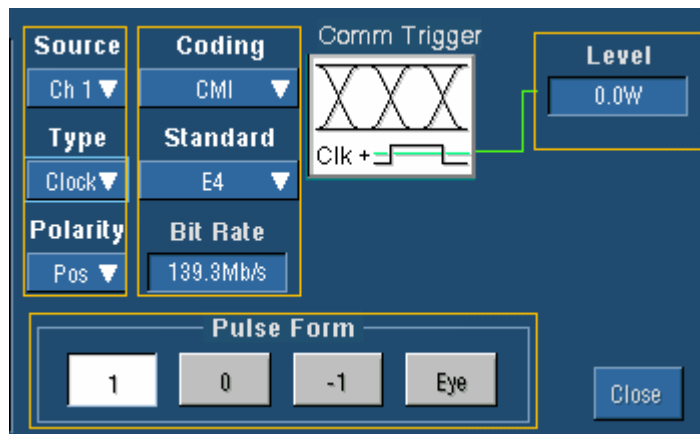
Behavior

Use the Comm Trigger control window to trigger the instrument in conjunction with mask testing on communications codes and standards. The controls work together to define the parameters for the trigger event. The standards that appear in the Standard drop-down list depend on the coding format that you selected. The coding format also determines whether upper, lower, single, or clock threshold levels or pulse forms are used. The standard determines the bit rate.

Communication Trigger Codes and Standards (Option)

Overview

The following tables list the Communication Trigger Codes and Standards available with Option SM:

Note

The communication trigger standards available for an instrument depend on the bandwidth and/or configuration of that instrument.

AMI trigger standards

Custom	32Mb	97Mb	DS1
	32.064 Mb/s	97.728 Mb/s	1.544 Mb/s
DS1A	DS1C	DS2	DS2 Rate Sym
2.048 Mb/s	3.152 Mb/s	6.312 Mb/s	6.312 Mb/s
DS2 Rate Coax	DS3	E1	E2
6.312 Mb/s	44.736 Mb/s	2.048 Mb/s	8.448 Mb/s
E3	STS–1		
34.368 Mb/s	51.84 Mb/s		

B3ZS trigger standards

Custom	DS3	STS–1
	44.736 Mb/s	51.84 Mb/s

B6ZS trigger standards

Custom	DS2	DS2 Rate Sym
	6.312 Mb/s	6.312 Mb/s

B8ZS trigger standards

Custom	DS1	DS1C	DS2 Rate Coax
	1.544 Mb/s	3.152 Mb/s	6.312 Mb/s

CMI trigger standards

Custom	DS4NA 139.26 Mb/s	E4 139.26 Mb/s
STM1E 155.52 Mb/s	STS–3 155.52 Mb/s	

HDB3 trigger standards

Custom	E1 2.048 Mb/s	E2 8.448 Mb/s
E3 34.368 Mb/s	DS1A 2.048 Mb/s	

MLT3 trigger standards

Custom	100Base–TX 125 Mb/s

NRZ trigger standards

Custom	2.5 IBand 2.5 Gb/s	FC133 132.8 Mb/s	FC266 265.6 Mb/s
FC531 531.2 Mb/s	FC1063 1.0625 Gb/s	FC2125E 2.125 Gb/s	G1 ATA 1.5 Gb/s
GB Ethernet 1.25 Gb/s	FS USB 12 Mb/s	HS USB 480 Mb/s	
OC1/STM0 51.84 Mb/s	OC3/STM1 155.5 Mb/s	OC12/STM4 622.1 Mb/s	
OC48/STM16 2.488 Gb/s	OC48–FEC 2.666 Gb/s	S400b 491.5 Mb/s	S800b 983.0 Mb/s
S1600b 1.966 Gb/s	XAUI 3.125 Gb/s	RapidIO 500M 500 Mb/s	RapidIO 750M 750 Mb/s
RapidIO 1.0G 1.0 Gb/s	RapidIO 1.5G 1.5 Gb/s	RapidIO 2.0G 2.0 Gb/s	SFI/SPI–5 2.5G 2.5 Gb/s
SFI/SPI–5 3.1G 3.1 Gb/s	RIO Serial 1G 1.25 Gb/s	RIO Serial 2G 2.5 Gb/s	RIO Serial 3G 3.125 Gb/s
VSR OC192 1.244 Gb/s	TFI–5 2.5G 1.488 Gb/s	TFI–5 3.1G 3.11 Gb/s	

Set to 50% with Comm Triggers

Overview

The Set to 50% control sets the trigger threshold to 50% for some Comm triggers and to a high of 75% with a low of 25% for others.

Behavior

When either AMI or MLT-3 code is selected with Comm triggers the Set 50% control measures peak-to-peak level and sets an upper threshold value at 75% and a lower threshold value at 25%. Other codes work exactly the same as Edge, Width, Glitch, and Timeout triggers; the threshold is set to 50%.

Trigger on A Only (Horizontal Delay Off)

To Use

Select the trigger event from the A Event tab and then set up trigger controls for the A Event. Next, select the A-B Seq tab and click A Only. Use the Horiz Position control to define the amount of pretrigger data.
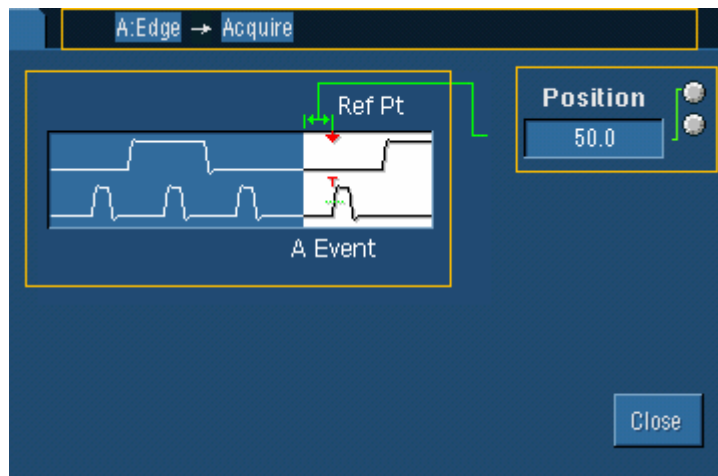
Behavior

When you select A Only with Delay Mode off, the instrument looks for the trigger conditions specified by the A event. When the instrument recognizes the A event, it acquires the posttrigger data.

🔆Tip Use the front panel HORIZONTAL POSITION knob to set the amount of pretrigger data.

Trigger on A Only (Horizontal Delay On)

To Use

Select the trigger event from the A Event tab and then set up trigger controls for the A Event. Next, select the A-B Seq tab and click A Only. Use the Horiz Delay control to delay the acquisition relative to the A Event trigger.
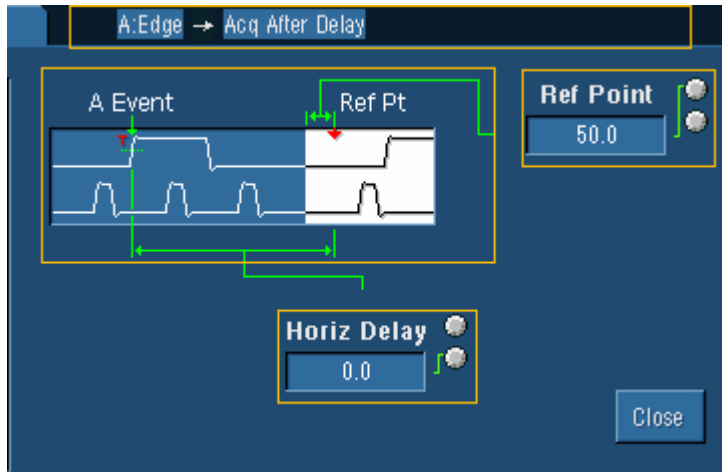
Behavior

When you select A Only with Delay Mode on, the instrument first looks for the trigger conditions specified by the A Event. The instrument then delays the acquisition for the time specified by the Horiz Delay control before acquiring the data.

Trigger After Time (Horizontal Delay Off)

To Use

Set up the A Event and B Event from their individual tabs. Select the A–>B Seq tab and click Trig After Time. Select the trigger delay time with the Trig Delay control. Adjust the Horiz Position and B Trig Level controls as necessary.
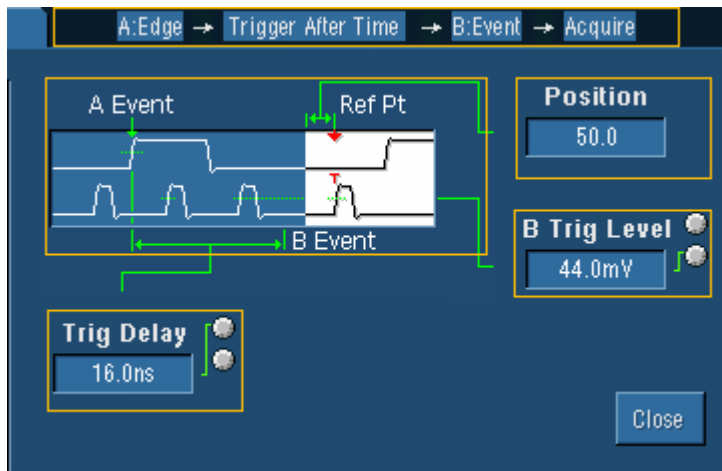
Behavior

When you select Trig After Time, the instrument looks for the trigger conditions specified by the A event. After the A event arms the trigger system, the instrument triggers on the next B event that occurs after the time specified by the Trig Delay control.

Trigger After Time (Horizontal Delay On)

To Use

Set up the A Event and B Event from their individual tabs. Then select the A–>B Seq tab and click Trig After Time. Select the trigger delay time with the Trig Delay control. Use the Horiz Delay control to delay the acquisition relative to the B event. Adjust the Ref Position and B Trig Level controls as necessary.
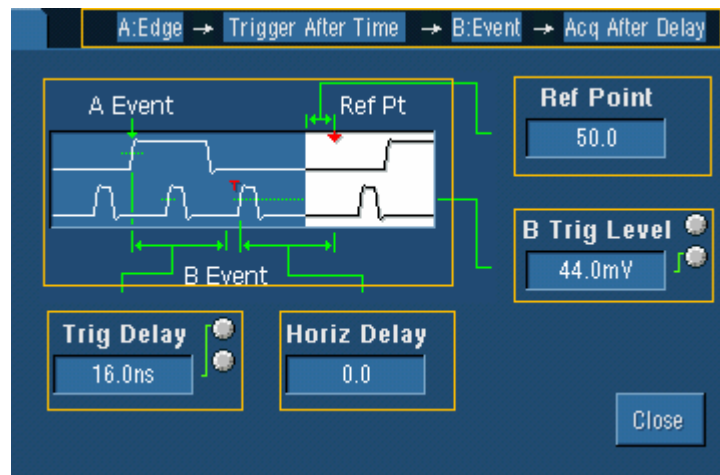
Behavior

Use this trigger sequence to trigger the instrument on the B trigger event after the A event.

When you select Trig After Time, the instrument looks for the trigger conditions specified by the A event. After the A event arms the trigger system, the instrument delays the trigger for the period of time specified by the Trig Delay control. After the trigger delay time has expired, the instrument looks for the B trigger event. When the instrument detects the B event, it delays the acquisition for the period of time specified by the Horiz Delay.

Trigger on Nth Event (Horizontal Delay Off)

To Use

Set up the A Event and B Event from their individual tabs. Then select the A–>B Seq tab and click Trig on nth Event. Select the number of B events with the Trig Event control. Adjust the Horiz Position and B Trig Level controls as necessary.
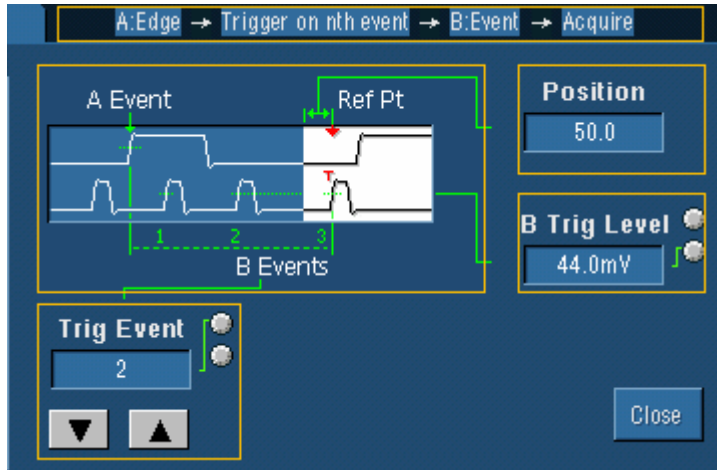
Behavior

Use this sequence to trigger the instrument on the nth occurrence of the B event after the A event.

When you select Trig on nth Event, the instrument looks for the trigger conditions specified by the A event. After the A event arms the trigger system, the instrument looks for a specific number of B events before triggering and displaying the data.

Trigger on Nth Event (Horizontal Delay On)

To Use

Set up the A Event and B Event from their individual tabs. Then select the A–>B Seq tab and click Trig on nth Event. Select the number of B events with the Trig Event control. Use the Horiz Delay control to delay the acquisition relative to the B event. Adjust the Ref Position and B Trig Level controls as necessary.
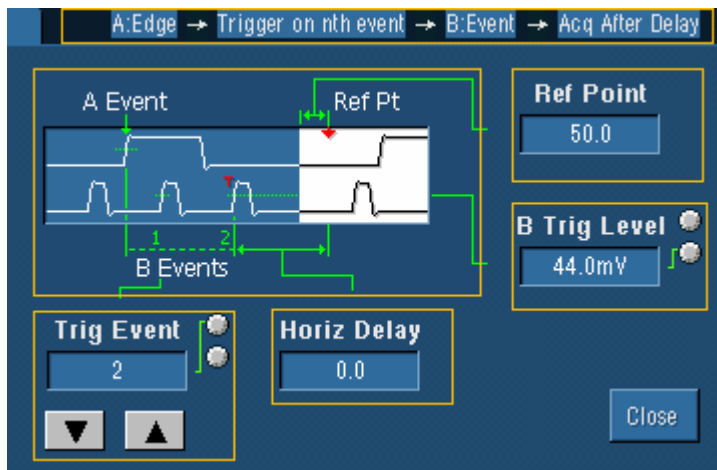
Behavior

When you select Trig on nth Event, the instrument looks for the trigger conditions specified by the A event. After the A event arms the trigger system, the instrument looks for a specific number of B events before triggering. The instrument then delays the acquisition for the amount of time specified by the Horiz Delay control. After the delay time has expired, the instrument displays the data.

Trigger Holdoff Control Window

Access this control window from the Trig menu Holdoff command.

Overview

Use the Trigger Holdoff control window to set the trigger holdoff.

To Use

Set Holdoff to Default to trigger after a preset amount of time. To set a specified holdoff time, click Time and enter a holdoff time in Trig Holdoff using the multipurpose knobs or pop-up keypad. You can also set the trigger Level with the multipurpose knob.

Behavior

When Holdoff is set to Default, the holdoff time is based on the time equal to five times the current horizontal scale setting. You may want to manually set the holdoff time to help stabilize the display of complex waveforms such as digital pulse trains that may have several possible trigger points.

The Level control can be either the main trigger level when there is no B Trigger event or the A Event trigger level when a B Event trigger is defined. To change the B Event trigger level, change the B Trig Level control under the B Event tab of the Trigger Setup control window.

Trigger Mode Control Window

Access this control window from the Trig menu Mode command.
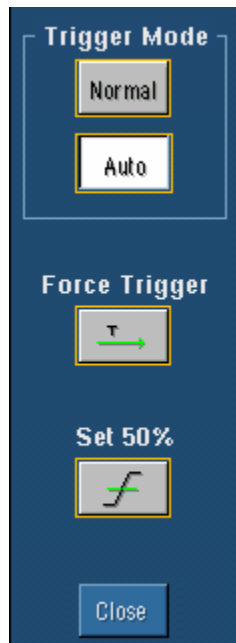
Overview

Use the Trigger Mode control window to set the Trigger Mode, to force a trigger, or to set the trigger level to 50%.

To Use

Click one of the buttons at the top of the control window to set the Trigger Mode.

Click Force Trigger once to acquire a waveform record without a trigger event. Forcing a trigger is useful when the input signal does not contain a valid trigger. The instrument recognizes and acts upon the forced trigger even when you click the button before the end of the pretrigger holdoff. The button has no effect when acquisitions are stopped.

Click Set 50% to set the trigger threshold level of the selected trigger channel to 50% of that channel's amplitude.

Behavior

These controls operate identically to the controls on the front panel.

Run/Stop Control Window

Access this control window from the Horizontal/Acquisition menu Run/Stop command.
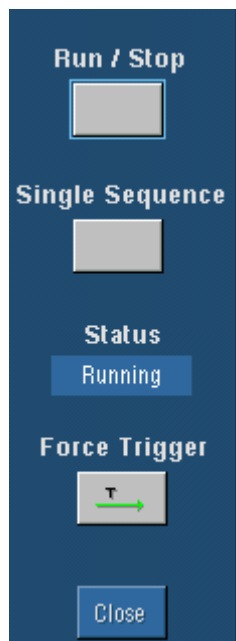
Overview

Use this control window to start and stop acquiring waveforms.

To Use

Click Run/Stop to toggle waveform acquisition. Click Single Sequence to complete a single acquisition sequence. Click Force Trigger to immediately start acquiring waveform data without a valid trigger event.

Behavior

These controls operate identically to the front-panel controls.

Click Run/Stop once to stop acquisitions. The Status readout at the top of the graticule changes from Running to Stopped and shows the number of acquisitions. Click Run/Stop again to resume acquiring waveforms.

Click Single Sequence to complete a single sequence of acquisitions each time you click the button. When Single Sequence is enabled, the front-panel single sequence (SINGLE) button is lighted.

In Sample, Peak Detect, or Hi Res acquisition modes, the instrument acquires waveform records with the first valid trigger event and then stops. In Average or Envelope acquisition modes, the instrument makes the specified number of acquisitions to complete the averaging or enveloping task.

In Waveform Database mode, the instrument acquires and displays a waveform that is the accumulation of several acquisitions. In addition to time and amplitude, the waveform displays a count of the number of times a specific sample point has been acquired.

When you click Single Sequence (or press SINGLE) in Equivalent Time mode, the instrument continues to recognize trigger events and acquires samples until the waveform record is filled.

Click Force Trigger once to acquire a waveform record without a trigger event. Forcing a trigger is useful when the input signal does not contain a valid trigger. The instrument recognizes and acts upon the forced trigger even when you click the button before the end of the pretrigger holdoff. The button has no effect when acquisitions are stopped.

Communication Trigger Coding

Overview

Use this control window to select the coding format for triggering on a communication signal and the standard to use with the coding format. The standard sets the bit rate.

To Use

Touch the Coding drop-down list to select the coding format to use for triggering the communication signal. Touch the Standard drop-down list to select the standard to use with the coding format. The bit rate is set by the standard selected and is a read only field.

Behavior

The coding format determines which standards are available as well as other parameters, such as trigger threshold and pulse form. The standard selected determines the bit rate. Changing the bit rate causes the standard to change to your own customized standard.

Comm Trigger Pulse Form

Overview

Pulse Form directs the instrument to trigger on a voltage change from the prior bit. This can be positive, negative, or neutral if there was no change. Depending on the code used, the pulse form can also indicate that triggering is to occur on an eye diagram.

To Use

Touch the pulse form that you want the instrument to use for triggering. Touch 1 to trigger on a positive pulse form, 0 on a neutral pulse form, or -1 on a negative pulse form.

Behavior

The available pulse forms are determined by which code is used for the signal. For example, the only pulse form available for NRZ codes is eye diagram.

Display Menu

Use Display menu options to change the way that waveforms appear on the instrument. This includes the style, intensity, color, format, text, and persistence. You can also change the graticule type and the items that appear on the display, such as date and time.

Display Setup Control Window (Appearance Tab)

Access this control window from the Display menu Display Setup command or Appearance command.

Overview

Use the controls on the Appearance tab to change the display style, display persistence, waveform intensity, display format, and waveform interpolation.

Display Style

To Use

Click one of the buttons to select the waveform display style.

Behavior

Vectors display waveforms with lines drawn between record points.

Dots display waveform record points as dots on the screen. Dots are useful when you want to display multivalued waveforms such as eye diagrams.

Inten Samp (Intensified Samples) shows only the actual samples. When you use intensified samples, you identify the actual sample points in an interpolated waveform.

Use the Display menu **Display Style** submenu to directly access the style options.

Note

The Vectors and Intensified Samples selections are not available in the XY and XYZ Display Formats.

Display Persistence
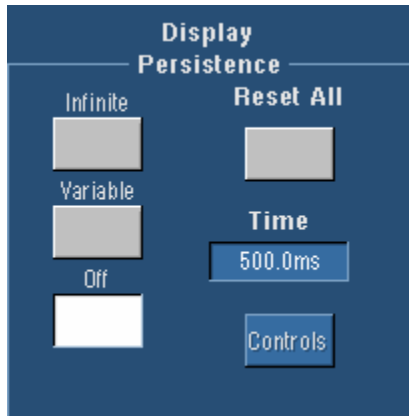
To Use

Click one of the buttons to select the display persistence. If you select Variable, click **Time** and then use the multipurpose knobs to select the persistence time. Click **Reset All** to clear the persistence. Click **Controls** to activate the Persistence Control window.

Behavior

Infinite persistence continuously accumulates record points on the waveform until you change one of the acquisition display settings. Use infinite persistence for displaying record points that may occur outside the normal acquisition envelope.

Variable persistence accumulates record points on the waveform for a specified time interval. Each record point decays independently according to the time interval.

If you click Off, waveform record points appear for the current acquisition only.

💡Tip Use the Display menu **Display Persistence** submenu or right-click in the graticule and select **Display Persistence** from the shortcut menu to directly access the persistence options.

Waveform Intensity
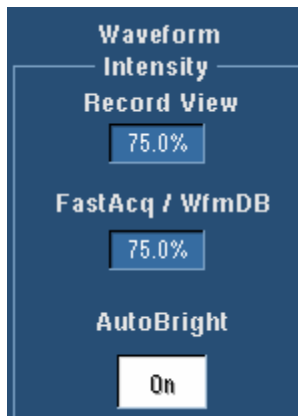
To Use

Click the Autobright button to turn the autobright feature on and off. You can also use the front-panel Intensity knob or the multipurpose knobs to manually adjust the display intensity.

Click the Record View text box to adjust the intensity of live waveforms, except when using Fast Acquisition or Waveform Database acquisition mode.

Click the FastAcq/WfmDB text box to adjust the intensity of live waveforms when using Fast Acquisition or Waveform Database acquisition mode.

Behavior

When AutoBright is On, the intensity of the waveform changes automatically to provide a visible waveform. When AutoBright is Off, the display simulates the appearance of analog oscilloscope signals. Waveforms that trigger more frequently appear brighter than waveforms that trigger less frequently.

Display Format
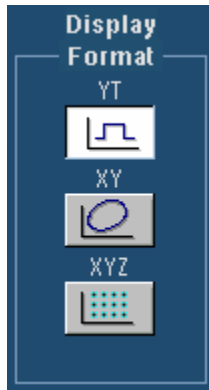
To Use

Click one of the buttons to select the display format.

Behavior

YT is the conventional oscilloscope display format that shows a signal amplitude on the vertical axis as it varies over time on the horizontal axis.

When you set the vertical position to center the display in YT mode, the XY display is at center screen, and each division of display in YT mode produces a division of display in XY mode.

XY plots the amplitude of one waveform against the amplitude of another waveform on a point-by-point basis. This format is useful for showing the relationship between two signals, such as current and voltage waveforms that combine to form an I-V characteristic. When you select the XY format, waveforms appear according to their channel pairings. XY format is a dot-only display, although it can have persistence. The Vector style selection has no effect when you select XY format.

XYZ plots the amplitude of one waveform record against the amplitude of another waveform record on a point-by-point basis. XYZ requires Fast Acquisitions mode. A third waveform record modulates the waveform intensity. When you select the XYZ format, waveforms appear according to their channel assignments. The XYZ format is not triggered. A −5 division signal (including position and offset) on CH 3 produces a blank screen; a +5 division signal produces full intensity.

Selecting YT or XY only affects the display. The HORIZONTAL and VERTICAL SCALE and POSITION knobs and menus control the same parameters regardless of the display format. In XY Display format, the horizontal scale continues to control the time base and the horizontal position continues to control the portion of the waveform that is shown. If in XY or XYZ, histograms are allowed, and horizontal controls are ignored.

Tip Use the Display menu **Display Format** submenu to directly access the format types.

Notes

The Vectors and Intensified Samples selections are not available in the XY and XYZ Display Formats.

The XYZ format is only available on 4-channel models and only supported in FastAcq mode.

Waveform Interpolation

Waveform Interpolation
Sin(x)/x

Linear

To Use

Click one of the buttons to select the interpolation that best displays the waveforms. Interpolation globally affects all displayed waveforms.

Behavior

The instrument interpolates waveforms in Zoom mode when there is less than one sample for each pixel column.

Because interpolation affects the displayed waveform rather than the acquisition data, it has no effect on math, measurements, or optional histograms. These functions are performed on the acquired data as compared to the displayed data.

Sin(x)/x interpolation computes record points along a curve between the actual acquired samples. This interpolation is useful when acquiring rounded waveforms such as sine waves. It is good for general-purpose uses but may introduce overshoot or undershoot in signals with fast rise times.

The Sin(x)/x interpolation is also useful to look at high-frequency signals, especially where the frequency components are just below the Nyquist frequency.

Linear interpolation computes record points between actual acquired samples using a straight-line fit. Linear interpolation is useful for waveforms with fast rise times, such as pulse trains.

Tip Use the Display menu **Waveform Interpolation** submenu to directly access the interpolation options. You may want to set the display style so that the real samples are intensified and interpolated samples are dimmed.

Display Setup Control Window (Screen Text Tab)

Access this control window from the Display menu Screen Text command or from the Display Setup control window.

Overview

Use the controls on the Screen Text tab to define text captions that you can add to the instrument display.

To Use

You can create up to eight lines of text to provide notes for screen shots, printouts, or for other instrument users. Click Display to turn the text display on and off. Click Properties to open the Text Properties control window where you can define the positioning of the text on the display.

Behavior

Click on one of the numbered lines and use the pop-up keyboard to enter up to 64 alphanumeric characters of text per line. Click Clear to clear the entire text of the selected line. The screen text that you enter is not related to any waveform labels that you may have created. The waveform labels help you identify waveforms on the screen, while the screen text is intended to provide information or notes for screenshots or printouts.

Tip Here are some tips for working with screen text:

Position the text on the screen by clicking and dragging the text to the desired location on the graticule.

Display Setup Control Window (Objects Tab)

Access this control window from the Display menu Objects command, or from the Display Setup control window.

Overview

Use the controls on the Objects tab to change the appearance of objects on the display.
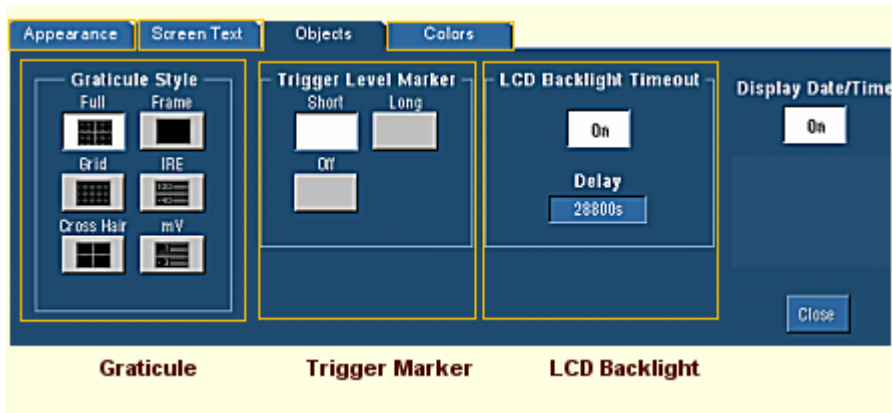
Click Display Date/Time to show the current date and time on the graticule.

Tip Use the Display menu **Display Date & Time** command to directly toggle the date/time display.

Graticule Style
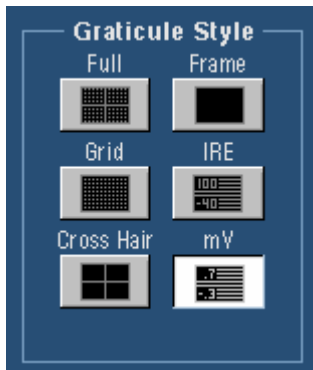
To Use

Click one of the buttons to select the graticule (display grid) style.

Behavior

Full provides a grid, cross hairs, and frame on the instrument display. Use Full to make quick estimates of waveform parameters. Grid shows a frame and a grid on the instrument display. This style is useful for making quick, full-screen measurements with cursors and automatic readouts when cross hairs are not needed.

Cross Hair shows cross hairs and a frame on the instrument display. This style is useful for making quick estimates of waveforms while leaving more room on the display for automatic readouts and other data.

Frame shows only a frame on the instrument display. This style is useful with automatic readouts and other screen text when other display features are not needed.

142

Use IRE if you are working with NTSC video signals.

Use mV if you are working with video signals other than NTSC.

Tip Use the Display menu **Graticule Style** submenu to directly access the graticule style options.

Trigger Level Marker

To Use

Click one of the buttons to select the trigger level indicator.

Behavior

The trigger level markers indicate the voltage level where the trigger or threshold levels of the active waveform occur.

The trigger level indicator does not appear on inactive or reference waveforms. The Short trigger level marker displays a short arrow on the side of the graticule by the active waveform.

The Long trigger level marker displays a horizontal line through the active waveform.

LCD Backlight Timeout

To Use

Click the button to enable the LCD Backlight Timeout control. Click Delay and use the multipurpose knob to set the delay time before the LCD back light turns off.

Behavior

The LCD Backlight control turns the LCD backlight off after the amount of time in Delay has expired without any screen activity. When the control is set to On and the time has expired, the screen blanks out. When the control is set to Off, the screen display remains on.

This feature is useful to extend the life of your LCD display.

Tip Use the Display menu **LCD Save Enabled** command to toggle the LCD backlight timeout. Timeout is On when a check mark appears next to the command on the menu.

Display Setup Control Window (Colors Tab)

Access this control window from the Display menu Colors command, or from the Display Setup control window.

Overview

Use the controls on the Colors tab to change waveform color palettes. You can define colors for the live, math, and reference waveforms. If you are using Fast Acquisition or Waveform Database acquisition mode, select the FastAcq/WfmDB Palette; for all other acquisition modes, use the Record View Palette.
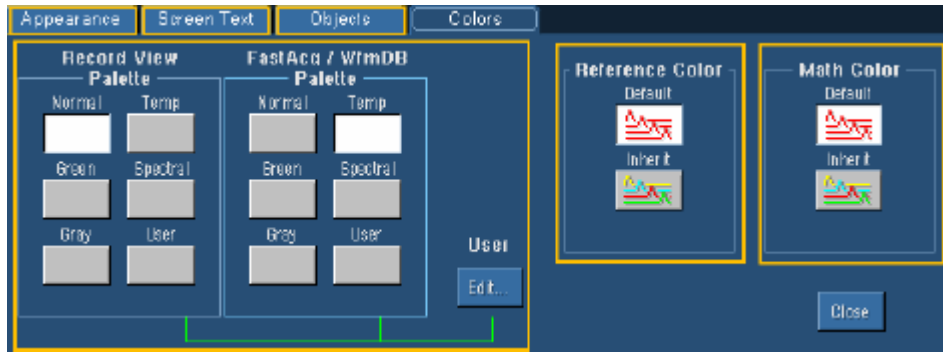
Color Palettes

To Use

Use the buttons under Record View Palette to define the color for live waveforms. Use the buttons under FastAcq/WfmDB Palette to define the color for live waveforms when using Fast Acquisition or Waveform Database acquisition mode. Click one of the buttons to select the color for displayed waveforms.

Click the User Edit… button to activate the Display Palette where you can customize the display color.
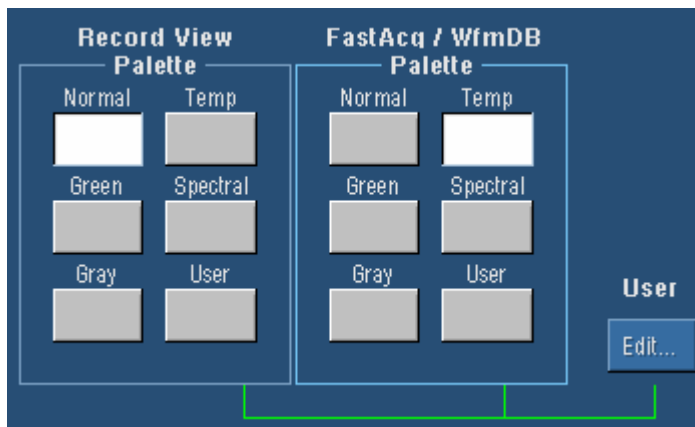
Tip Use the Display menu **Record View Palette** or the **Fast/Acq WfmDB Palette** submenu to directly access the color palettes.

Behavior

Normal displays hues and lightness levels for best overall viewing. The color of each channel waveform matches the color of the corresponding front-panel vertical SCALE knob.

On the Spectral (Spectral Grading) palette, areas of the waveform with the highest sample density appear in blue shades while the areas of lowest sample density appear in red shades.

On the Temp (Temperature Grading) palette, areas of the waveform with the highest sample density appear in warmer colors (red shades) while the areas of lowest sample density appear in cooler colors (blue shades).

Gray (Monochrome Gray) displays waveforms in shades of gray. Areas of the waveform with the highest sample density appear in lighter gray shades while the areas of lowest sample density appear in darker gray shades.

Green (Monochrome Green) displays waveforms in shades of green. Areas of the waveform with the highest sample density appear in lighter green shades while the areas of lowest sample density appear in darker green shades.

Select User to display the color that you have previously defined with the User Palette.

🔆Tip The Green palette simulates the display of an analog oscilloscope.

Display Palette

Access this control window from the Display menu, User Palette…

Overview

Use this control window to customize the color of your waveform display.

To Use

Click the Hue, Lightness, or Saturation text box to adjust the percentage of each of these attributes. Click the Source drop-down list to select the waveform for which you want to change the color.

Click Reset to return the User palette to the default. Click Setup to activate the Display Setup control window.

Behavior

Increasing the hue value changes the color to blue tones. Decreasing the hue value changes the color to red tones. For best viewing, set the lightness to 50.

Higher numbers tend to wash out the color and lower numbers can darken the color and make it difficult to see. Lower saturation values reduce the amount of color displayed in the waveform. Higher saturation values increase the amount of color.

Reference Color
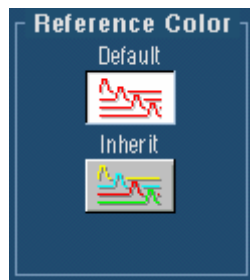
To Use

Click one of the buttons to select the color scheme for reference waveforms.

Behavior

Click Default to use the default system color for the selected reference waveform.

Click Inherit to use the same color as the original waveform.

Math Color
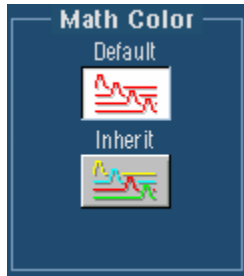
To Use

Click one of the buttons to select the color scheme for math waveforms.

Behavior

Click Default to use the default system color for the selected math waveform.

Click Inherit to use the same color as the waveform that the math function is based on. When the instrument is set to XY display mode, multiple math waveforms will inherit the color of the primary (first) channel.

Text Properties Control Window

Access this control window from the Display Setup control window Screen Text tab, Properties button.
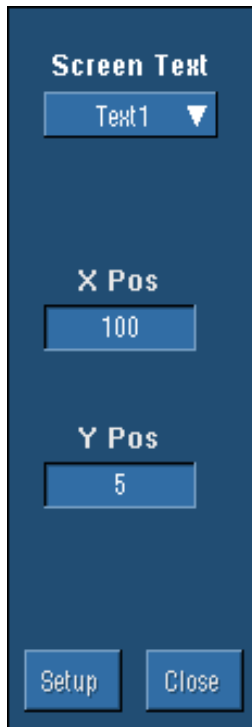
Overview

Use the Text Properties control window to position text to specific locations on the graticule.

To Use

Select the line of text from the Screen Text drop-down list and then use the X Pos and Y Pos controls to define the location of the text on the graticule.

Behavior

The text from the drop-down list is the text that you defined under the Screen Text tab of the Display Setup control window.

The units for the position controls are in pixels. The minimum values are 1 for both X and Y positions. The maximum X value is 450 while the maximum Y value is 390. The origin of the X position is the left vertical graticule boundary while the origin of the Y position is the upper horizontal graticule boundary.

Setup opens the Display Setup control window with the Screen Text tab selected. Click Setup to enter new lines of text.

Tip You can also position the text on the screen by clicking and dragging it to the desired location.

Display Persistence Control Window

Access this control window from the Display menu, Display Persistence submenu, Persistence Controls command.

Overview

Use the Display Persistence control window to quickly change the display persistence or set the persistence time.

To Use

Click one of the Display Persistence buttons or click Off to set the display persistence. Click the Persist Time text box, and use the multipurpose knobs to set the persistence time. Click Reset to clear the display of accumulated waveform points.

Behavior

Clicking the display persistence buttons activates that type of persistence. Clicking Off turns all persistence off. Clicking Reset clears the accumulated waveform points from the display and restarts the persistence time. Adjusting the persistence time allows you to set the length of time to accumulate waveform points when you are using Variable persistence.

TDS5032B and TDS5052B

XY Pair	X Axis	Y Axis
Ch 1 and Ch 2	Ch 1	Ch 2
Ref 1 and Ref 2	Ref 1	Ref 2

TDS50534B, TDS5054B, TDS5104B and TDS5054BE

XY Pair	X Axis	Y Axis
Ch 1 and Ch 2	Ch 1	Ch 2
Ch 3 and Ch 4	Ch 3	Ch 4
Ref 1 and Ref 2	Ref 1	Ref 2
Ref 3 and Ref 4	Ref 3	Ref 4

If you only display one source when you enter the XY format, the instrument automatically turns on the other source waveform to complete the XY pair.

XYZ format channel assignments

When you use the XYZ display format, channel waveforms are assigned to the axis indicated in the following table. You cannot display math and reference waveforms in the XYZ format.

Channels	X Axis	Y Axis	Z Axis
Ch 1, Ch 2, and Ch 3	Ch 1	Ch 2	Ch 3

If you only display one or two source channels when you enter XYZ format, the instrument automatically turns on the other channels.

Note

The XYZ format is only available on 4-channel models and is only supported in FastAcq mode.

Cursors Menu

Use the Cursors menu options to select cursor types, mode, and position, and also to turn cursors on and off.

Cursor Setup Control Window

Access this control window from the Setup button on the Cursor Type Setup control window, or by clicking the Cursor Setup command on the Cursors menu.

Overview

Use these controls to set the cursor parameters.
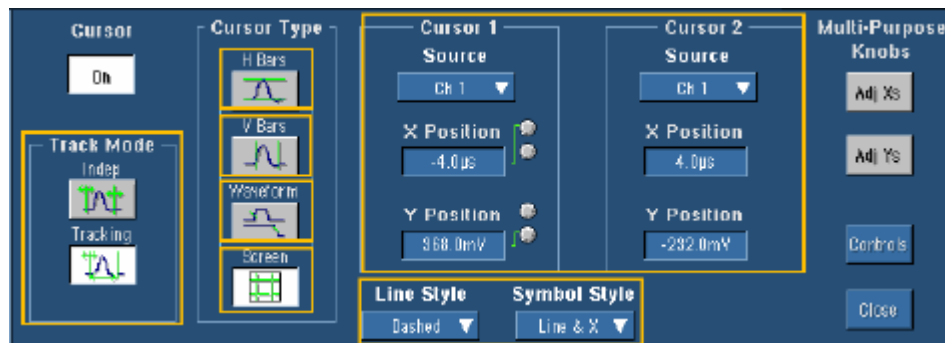
To Use

Click Cursor in the control window to turn cursors on and off. Select a cursor type by clicking one of the Cursor Type buttons. Select the waveform you want to measure from the Source drop-down list. Cursor 1 and Cursor 2 each have their own source.

Tip If are using H-Bar or V-Bar Cursors and want the cursor source to be the same for both cursors, just set the source for Cursor 1. Cursor 2 will have the same value as Cursor 1, unless you change it.

Click the Cursor Positions controls to map the cursors to the multipurpose knobs, then use the knobs to move the cursors. To select the cursor mode, click either the Indep (independent) or Tracking button. Select a Line Style from the drop-down list. Line styles are available for all cursor types. Select a Symbol Style from the drop-down list. Symbol styles are only available on Waveform and Screen cursors.

If you are using XY display format you can also select the XY Cursor Readout from the drop-down list. Only Waveform and Screen cursors are available in XY display format.

Tip The Cursors menu **Cursors On** command can also be used to toggle the cursors on and off. The cursors are on when a check mark appears next to this command.

Behavior

When cursors are turned on, the front panel CURSORS button is lighted.

Indep (independent) cursor mode moves the cursors independent of each other. In Tracking mode, moving Cursor 1 moves Cursor 2 in tandem. However, moving Cursor 2 does not move Cursor 1.

Symbol style applies only to Waveform and Screen cursors. The Line & X symbol style displays only an X when used with waveform cursors in XY display mode such that the vertical line is not drawn only the X.

If you change the display format to XY when H Bar cursors are enabled, the cursor type will change to Screen. In normal acquisition mode, changing the display format to XY when V Bar cursors are enabled, changes the cursor type to Waveform. In FastAcq acquisition mode, changing the display format to XY when V Bar cursors are enabled, changes the cursor type to Screen.

🔖Tip To turn the cursors off, push the front-panel CURSORS button.

Cursor Type Setup Control Window

Access this control window by pressing the CURSORS front-panel button, clicking the Cursors toolbar button, or selecting the Cursor Controls command from the Cursor menu.
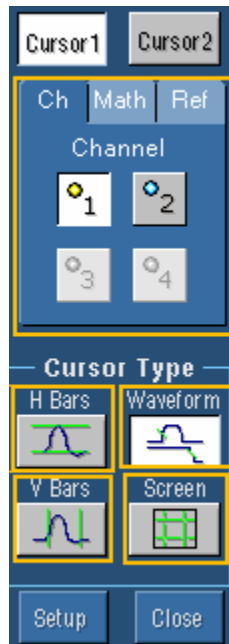
Overview

Use the Cursor Type Setup control window to select source for each cursor and the type of cursors for waveform measurements.

To Use

Click the Cursor1 button, and then select the waveform that you want to measure from the Channel Selector tabs. If you want to measure a different waveform from Cursor 2, click the Cursor 2 button, and then select the waveform to measure from the Channel Selector tabs. Click one of the Cursor Type buttons to select a cursor type.

🔖Tip Select a cursor type directly from the Cursors menu **Cursor Type** submenu.

Behavior

If are using H-Bar or V-Bar Cursors and want to measure the same waveform with both cursors, just select the waveform for Cursor 1. Cursor 2 will have the same value as Cursor 1, unless you change it.

Click Setup to open the Cursor Setup control window where you can specify other cursor parameters.

Cursor Sources

To Use

Click the Source drop-down list to select the source for the cursor. Click the Position text box, and then use the multipurpose knobs to adjust the position of each cursor.

Behavior

You can assign cursor sources to both Cursor 1 and Cursor 2 for any cursor type. If the cursor type is V Bars or H Bars, setting the Cursor 1 source sets the Cursor 2 source to the same value. To have different sources for Cursor 1 and Cursor 2, set the Cursor 1 source first, and then set the Cursor 2 source.

Only Screen Cursors have both an X and Y position.

The Multipurpose Knobs Adj buttons appear only with Screen cursors. Click the Adj button to assign the multipurpose knobs to the X-axis and the Y-axis.

Cursor Position Control Window

Access this control window from the Cursors menu, Cursor Position command.
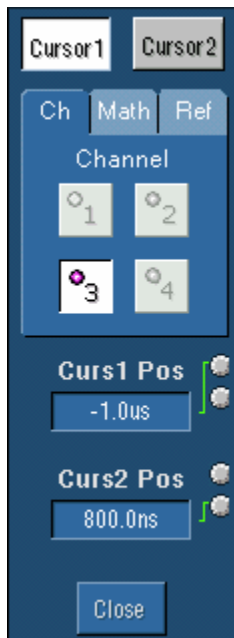
Overview

Use this Cursor Position control window to position the cursors with the multipurpose knobs.

To Use

Select the waveform that you want to measure from the Channel Selector tabs. Click one of the cursor position controls and use the multipurpose knobs to enter the cursor position.

Behavior

The cursor units depend on the cursor type. The cursor position readouts appear on the display graticule. When using the IRE graticule style, amplitude cursor readouts are shown in IRE units.

Cursor position does not change if the cursor source changes or if the waveform is repositioned.

Tip Use the pop-up keypad or your external keyboard to specify exact numerical cursor coordinates.

Horizontal Bar Cursors

To Use

To use the horizontal bar cursors, click H Bars in one of the cursor control windows or click H-Bars from the Cursor Type submenu of the Cursors menu. Use the multipurpose knobs to move the cursors. Use the Cursor Source Channel Selector tabs to select the waveform that you want to measure.

Behavior

Horizontal bar cursors measure vertical parameters (amplitude). Typically the vertical parameters are in volts, but depend on the Vertical Units selected in the control window.

The cursor readouts appear on the graticule when a control window is open. When the control windows are closed, the readouts appear on the right side of the screen. Each cursor has its own readout with respect to the zero-reference. The Δ (delta) shows the difference between the cursor values.

Vertical Bar Cursors

To Use

To select the vertical bar cursors, click V Bars in one of the cursor control windows or click V Bars from the Cursor Type submenu of the Cursors menu. Use the multipurpose knobs to move the cursors. Use the Cursor Source Channel Selector tabs to select the waveform that you want to measure.

Behavior

Vertical bar cursors measure horizontal parameters. Typically horizontal parameters are in time or frequency.

The cursor readouts appear on the graticule when a control window is open. When the control windows are closed, the readouts appear on the right side of the screen.

For time domain waveforms, each cursor has its own readout relative to the trigger point. The Δ (delta) shows the time difference between the cursor values, or the frequency associated with the period between the cursors (Hz = 1/time).

For frequency domain signals, each cursor has its own readout relative to 0 Hz. The Δ shows the difference in the frequencies between the cursors.

You can also use vertical bar cursors with measurement gating. Measurement gating enables you to limit automatic measurements to within the boundaries of the vertical bar cursors rather than use the entire display width.

Waveform Cursors

To Use

To select waveform cursors, click Waveform in one of the cursor control windows or click Waveform from the Cursor Type submenu of the Cursors menu. Use the multipurpose knobs to move the cursors. Use the Cursor Source Channel Selector tabs to select the waveform that you want to measure.

Behavior

Waveform cursors can measure parameters on the same waveform or on different waveforms. The cursors measure vertical parameters (typically volts) and horizontal parameters (typically time or frequency) simultaneously. An X identifies the cursor location at the point where the cursors intersect the waveform.

The cursor readouts appear on the graticule when a control window is open. When the control windows are closed, the readouts appear on the right side of the screen.

One Δ (delta) shows the voltage difference between the two cursors while the other Δ shows the time (or frequency) difference between the cursor values.

Screen Cursors

To Use

To select screen cursors, click Screen in one of the cursor control windows or click Screen from the Cursor Type submenu of the Cursors menu. Use the multipurpose knobs to move the cursors. Use the Cursor Source Channel Selector tabs to select the waveforms that you want to measure.

Behavior

Use screen cursors to indicate an arbitrary X and Y position in the waveform display area represented by the intersection of a vertical and horizontal cursor line. Screen cursors measure both vertical parameters (typically volts) and horizontal parameters (typically time or frequency).

Readouts display measurement parameters for both cursors. Vertical readouts display voltage values for each cursor with respect to the zero reference point and with respect to each other. Horizontal readouts display time or frequency values for each cursor with respect to the trigger point and with respect to each other.

Cursor Modes

To Use

To select the cursor mode, click either the Indep (independent) or Tracking button in the Cursor Setup control window.

Behavior

In Independent mode, you can move one cursor at a time using the multipurpose knobs.

In Tracking mode, moving Cursor 1 moves Cursor 2 in tandem so that Cursor 2 will always remain a fixed distance (time or voltage) from the Cursor 1. However, moving Cursor 2 does not move Cursor 1.

Tip Select the cursor mode directly from the Cursors menu **Cursor Mode** submenu.

Cursor Style

To Use

Select a Line Style from the drop-down list. Line styles are available for all cursor types.

Select a Symbol Style from the drop-down list. Symbol styles are only available on Waveform and Screen cursors. If you are in XY display format, you can also select the XY Cursor Readout from the drop-down list.

Behavior

Cursor positions are displayed according to line style, symbol style, cursor type, and waveform display format. See the following table. The Line & X symbol style displays only the X when used with waveform cursors in XY display mode. (The vertical line is not drawn.)

Symbol Style	Cursor Type	Display Format	Cursor Position
Lines	Waveform and Screen	YT	Intersection of horizontal and vertical lines. Default style for Screen cursors.
Line & X	Waveform and Screen	YT	Horizontal position is shown by a vertical line, the Y-axis is shown by an X. Default style for Waveform cursors in YT mode.
X	Waveform and Screen	XY	Single X indicates the horizontal and vertical position. Default style for Waveform Cursors in XY mode.

Cursor readouts are displayed based on the selection in the XY cursor Readout drop-down list. This list is only available for Waveform and Screen cursors when you are using XY display format.

XY Cursor Readout	Cursor Type	Display Format	Cursor Position
Rectangular	Waveform and Screen	XY	Shown as rectangular coordinate positions. There are nine readouts for Waveform cursors and six for Screen cursors.
Polar	Waveform and Screen	XY	Shown as polar coordinate positions. There are nine readouts for Waveform cursors and six for Screen cursors.
Product	Waveform and Screen	XY	Shown as rectangular coordinates and the X vertical value is multiplied by the Y vertical value. There are six readouts for Waveform cursors and three for Screen cursors.

154

XY Cursor Readout	Cursor Type	Display Format	Cursor Position
Ratio	Waveform and Screen	XY	Shown as rectangular coordinates and the Y vertical value is divided by the X vertical value.

Cursors Using XY Display Format

Access this control window from the Cursors menu Cursor Setup command. The Cursor Setup control window varies depending on the display format.

Overview

Use these controls to set the cursor parameters when you are using XY display format.
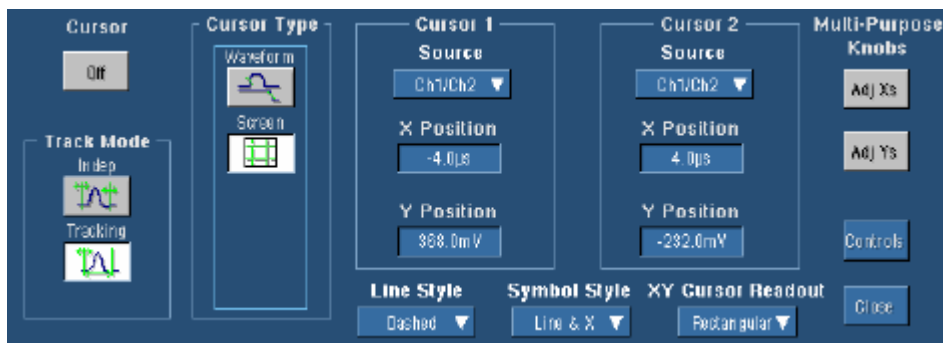
To Use

Select the readout format from the XY Readout drop-down list.

Behavior

When you are using XY display format, only Waveform and Screen cursors are available. If you change the display format to XY when H Bar cursors are enabled, the cursor type will change to Screen. In normal acquisition mode, changing the display format to XY when V Bar cursors are enabled, changes the cursor type to Waveform. In FastAcq acquisition mode, changing the display format to XY when V Bar cursors are enabled, changes the cursor type to Screen.

If you change the display format from XY to YT when Screen cursors are enabled, the cursor type is changed to H Bars.

XY Cursor Readouts only appear when you are using XY display format. Rectangular XY readouts are the default.

Rectangular shows the cursor readouts as rectangular coordinate positions. Waveform cursors display nine rectangular readouts (ΔX, ΔY, Δt, X1, X2, Y1, Y2, t1, and t2). Screen cursors display six readouts (ΔX, ΔY, X1, X2, Y1, and Y2).

Polar shows the cursor position as polar coordinate positions. Waveform cursors display nine polar readouts (Δr, $\Delta 0$, Δt, r1, r2, 01, 02, t1, and t2). Screen cursors display six polar readouts (Δr, $\Delta 0$, r1, r2, 01, and 02).

Product shows the cursor position as rectangular coordinates and the X vertical value is multiplied by the Y vertical value. Waveform cursors display six product readouts (ΔX x ΔY, X1 x Y1, X2 x

Y1, X2 x 2, Δt, t1, and t2). Screen cursors display three product readouts (ΔX x ΔY, X1 x Y1, and X2 x Y2)

Ratio shows the cursor position as rectangular coordinates and the Y vertical value is divided by the X vertical value. Waveform cursors display six ratio readouts (ΔY÷ΔX, Y1÷X1, Y2÷Y1, Δt, t1, and t2). Screen cursors display three ratio readouts (ΔY÷ΔX, Y1÷X1, and Y2÷Y1).

Measurement Menu

Use Measurement menu commands to select from and set up a variety of automatic measurements. Commands are also available on this menu for further in-depth analysis, including snapshots, gating (bounding), statistics, communication, and histograms.

Measurement Setup Control Window

Access this control window from the Measure menu Measurement Setup command, or the Meas toolbar button.

Overview

Use the Measurement Setup control window to set up the instrument for automatic measurements.
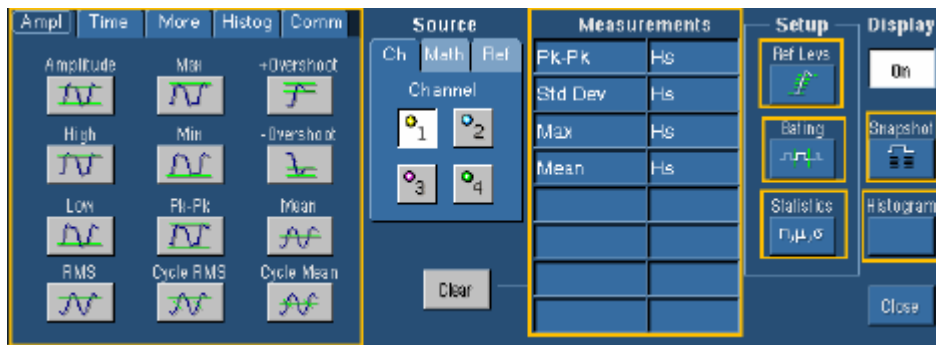
From this control window, you can:

- Select a waveform source using the Channel Selector tabs from which you can take the measurements. You can measure live waveforms, math waveforms, or reference waveforms.

- Define amplitude, time, histogram, communications, or other measurements on the selected waveforms.

- Create a list of measurements to appear in the main graticule or in a separate measurement window.

- Define unique the reference levels for each measurement.

- Use gating to limit the measurements to a specific area of the waveform.

- Set up measurement statistics.

- Take a snapshot of all measurements.

- Set up histograms to evaluate specific regions of the waveform.

Tip Turn on Fast Acquisitions to enhance your measurements. In Fast Acquisitions mode, measurements are calculated on an array of acquired waveforms. As the waveforms accumulate (with persistence on), your measurement accuracy may increase. However, only one or two channels can be displayed when you take automatic measurements using Fast Acquisitions mode. If three or more channels are displayed, Fast Acquisitions mode is turned off, and then the requested measurements are made. Fast Acquistions mode is not automatically turned back on when you select fewer than three channels.

Measurement Tabs: Amplitude
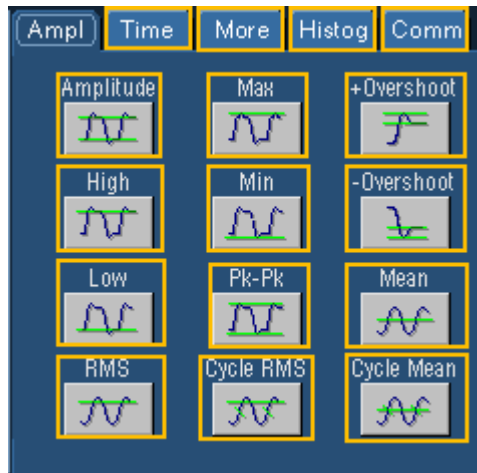
To Use

Use the Ampl tab to select automatic amplitude measurements. Select a source channel from the Channel Selector tabs, and click a measurement button to add the measurement to the Measurement list. Click Display in the upper-right corner of the control window to turn the measurement display on and off.

Behavior

Measurements are updated approximately three times a second, or less for long record lengths. Measurements are calculated after each acquisition. You can display up to eight automatic measurements.

💡Tip Select the individual measurements directly from the Measure menu **Amplitude** submenu.

Measurement Tabs: Time

To Use

Use the Time tab to select automatic time measurements. Select a source channel from the Channel Selector tabs, and click a measurement button to add the measurement to the Measurement list. Click Display in the upper-right corner of the control window to turn measurement display on and off.

Behavior

Measurements are updated approximately three times a second, or less for long record lengths. Measurements are calculated after each acquisition. You can display up to eight automatic measurements.

A new control window opens when you click Delay. Enter the parameters for the delay measurement and click Ok to return to the Measurement Setup control window. The delay measurement is added to the list.

💡Tip Select the individual measurements directly from the Measure menu **Time** submenu.

Measurement Tabs: More

To Use

Use the More tab to select measurements not found under any of the other measurement tabs. Select a source channel from the Channel Selector tabs, and click a measurement button to add the measurement to the Measurement list. Click Display in the upper-right corner of the control window to turn the measurement display on and off.

Behavior

Measurements are updated approximately three times a second, or less for long record lengths. Measurements are calculated after each acquisition. You can display up to eight automatic measurements.

A new control window opens when you click Phase. Enter the parameters for the phase measurement and click Ok to return to the Measurement Setup control window. The phase measurement is added to the list.

Tip Select the individual measurements directly from the Measure menu **More** submenu.

Measurement Tabs: Histogram
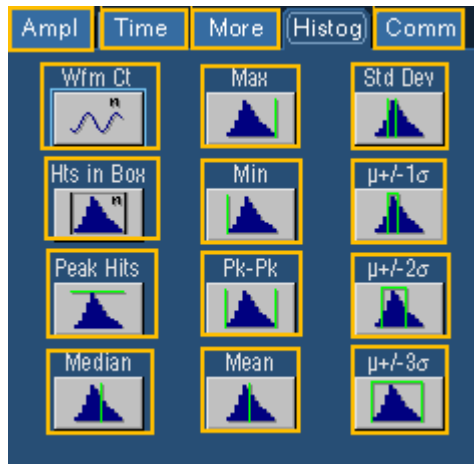
To Use

Use the Histog tab to select automatic histogram measurements. Click a measurement button to add the measurement to the Measurement list. Click Display in the upper-right corner of the control window to turn measurement display on and off.

Behavior

Measurements are updated approximately three times a second, or less for long record lengths. Measurements are calculated after each acquisition. You can display up to eight automatic measurements.

Tip

You can also select the individual measurements directly from the Measure menu **Histogram Measurements** submenu.

Measurement Tabs: Communications

To Use

Use the Comm tab to select communications measurements, and then click a measurement button to add the measurement to the Measurements list.

Behavior

Measurements are updated approximately three times a second (or less for long record lengths). Measurements are calculated after each acquisition.

You can display up to eight automatic measurements.

💡**Tip** Most Comm measurements require either Wfmdb or FastAcq mode.

💡**Tip** You may need to select Infinite Persistence display mode to accumulate enough data for the Comm measurement to work.

💡**Tip** Select the individual measurements directly from the Measure menu.

Measurement Tabs: Communications More

To Use

Use the Comm tab to select communications measurements, and then click a measurement button to add the measurement to the Measurements list.
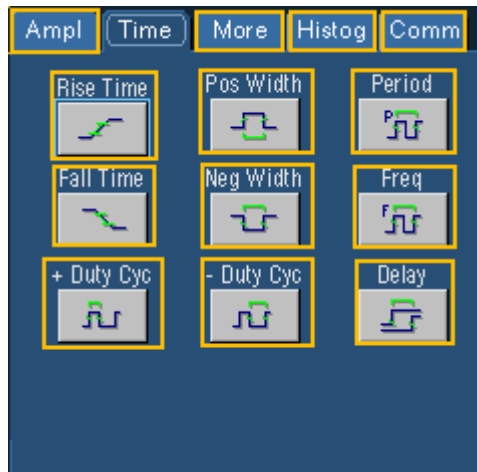
Behavior

Measurements are updated approximately three times a second (or less for long record lengths). Measurements are calculated after each acquisition.

You can display up to eight automatic measurements.

💡**Tip** Most Comm measurements require either Wfmdb or FastAcq mode.

💡**Tip** You may need to select Infinite Persistence display mode to accumulate enough data for the Comm measurement to work.

💡**Tip** Select the individual measurements directly from the Measure menu.

Measurements List

Measurements	
Rise Time	Ch3
Fall Time	Ch3
Pos Width	Ch3
+ Duty Cyc	Ch3

To Use

To add a measurement to the list, select the waveform source, and click one of the measurement buttons. To delete a measurement, select the measurement, and click **Clear**. Click **Display** in the upper-right corner of the control window to turn the measurement displays on and off.

Tip To delete all measurements in a single step, select all of the measurements and click **Clear**.

Behavior

This list contains your selected automatic measurements. You can add up to eight automatic measurements to the list. The measurements appear on the main screen when the Display button is On.

When you click Clear without selecting a measurement, the last measurement in the list is deleted.

Tip To quickly move the measurements from one channel to another, select all of the measurements in the list (click the first measurement, hold the left mouse button, and drag the mouse to the last measurement), and then select a new channel from the Source control. Similarly, you can change all of the measurements from one type to another if you select all the measurements in the list and click on a single measurement button.

Measurement Snapshot Control Window

Access this control window from the Measure menu Snapshot command, or from the Measurement Setup control window.

Overview

The Measurement Snapshot lets you see all of the automated measurements for a selected waveform at once.

To Use

Select the waveform for which you want to take a snapshot. Click the Snapshot button in the Measurement control window, or select Snapshot from the Measure menu. Click either General or Comm under Snapshot Type to begin the snapshot. You can also copy the snapshot data to the clipboard for use in another program, such as Excel.

Behavior

When the instrument has completed all of the measurements, the snapshot window is updated with the measurements.

Snapshot executes all of the single waveform measurements once; the measurements are not continuously updated. Any measurements previously taken are retained on the display until you complete a new snapshot. To take another snapshot, click the Snapshot Again button.

Tip Things to consider when taking snapshots:

- Before taking a snapshot, be sure to properly display the waveform. Snapshot does not warn you if a waveform is improperly scaled (clipped, low signal amplitude, low resolution, and so on).

- To change the source for taking a snapshot, select another channel, math, or reference memory waveform, and then execute snapshot again.

Measurement Statistics Control Window

Access this control window from the Measurement Setup control window. You can also access statistics from the Measure menu Statistics submenu.

Overview

Use the Measurement Statistics control window to display statistical calculations on measurements. The control window contains controls that determine which statistics are displayed in the measurement window.
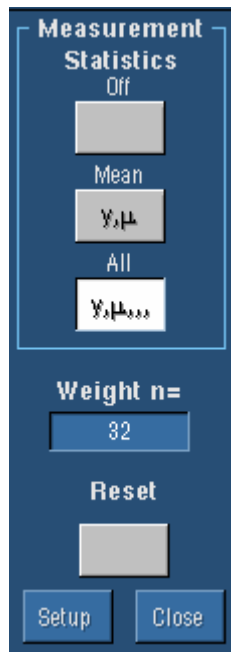
To Use

Click **Off** to turn the measurement statistics display off. Click Mean to display the value and the mean (μ) of each measurement. Click All to display all statistics for each measurement value (mean (μ), minimum (m), maximum (M), standard deviation (σ), and population (n)). Use the multipurpose knobs to set the statistics weight. Click Reset to restart the statistics calculations.

Tip Select the individual statistics directly from the Measure menu **Statistics** submenu.

Behavior

Measurement values continue as live updates while the statistics accumulate over time. The statistics appear on the main screen when a control window is open or on the right side of the screen when no control windows are open and there are less than four measurements active. The color of the statistics match the color of the waveform from which the measurement was taken.

The statistics weight specifies the constant used to calculate the mean and standard deviation; the minimum and maximum values are unaffected by weighting. A large value produces statistics that vary less over time.

All statistics (mean, minimum, maximum, standard deviation, and population) are always gathered even though the mean may be the only statistic shown. When measurements are turned on, the acquisitions take longer because measurements and statistic calculations are done on each acquisition.

Reference Levels Control Window

Access this control window from the Measure menu Reference Levels command.

Overview

Use the Reference Levels control window to define unique reference levels for each measurement. The default reference levels serve most general-purpose measurements. However, you can customize all measurements to use different reference levels.
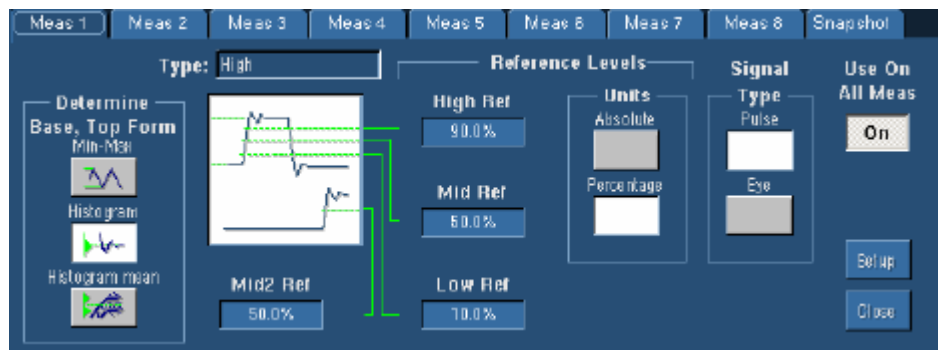
To Use

Click Min-Max to derive the reference levels from the minimum and maximum values of the waveform record. If you want to derive the reference levels based on statistical data, click the Histogram or the Histogram mean button.

Refer to the graphic in the center of the control window and determine the reference levels that you want to define. Click the control with the associated reference level, and use the multipurpose knobs to adjust the value. If you want the reference levels to be measured by volts, click the Absolute button under Units. To measure the reference levels as a percentage, click the Percentage button.

Click either Eye or Pulse to select the signal type.

Behavior

The following reference levels are included: High Reference, Mid Reference, Low Reference, and Mid2 Reference.

Define the reference levels as percentages of the signal amplitude or as absolute values in vertical units by clicking the appropriate button to define the Units selection. Use Percentage to have reference levels track the signal. Use Absolute to set the reference levels to fixed values.

The top and base levels that the measurement system derives from a waveform influence the fidelity of measurements. The measurement system can derive the reference levels from either the minimum and maximum values of the waveform or from statistical data.

The Min-Max selection uses the highest and lowest values of the waveform record. This selection is best for examining waveforms with no large, flat portions at a common value, such as sine waves and triangle waves.

The Histogram selection sets the top and base values statistically. The most common values either above or below the midpoint are used (depending on whether the high reference point or the low

reference point is being defined). Because the statistical approach ignores short-term aberrations (overshoot or ringing), the histogram method is the best setting for examining pulses.

Gating Control Window

Access this control window from the Measure menu Gating command, or from the Measurement Setup control window.
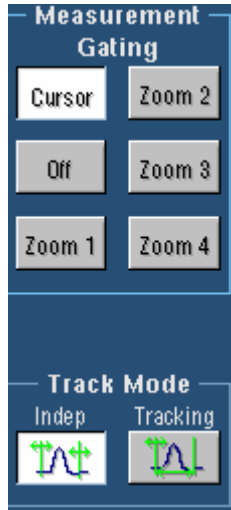
Overview

Use the Gating control window to limit the measurement area of the waveform to the region bounded by the vertical bar cursors or to the left and right edges of a Zoom window.

To Use

To define the gating area with the vertical bar cursors, click **Cursor**, and then click either **Indep** or **Tracking** to select the Track Mode. These buttons are only displayed when you click Cursor.

To define the gating area from a Zoom area, click one of the four Zoom buttons.

To turn off measurement gating, click the Off button.

Click Setup to return to the Measurement Setup control window.

Behavior

Normally the instrument takes measurements over the entire waveform record. Gating lets you limit measurements to a specific portion of the waveform. You can limit gating to the region bounded by the vertical bar cursors or to the left and right edges of a Zoom area.

When you click Zoom (1-4) the Zoom window appears in the lower half of the display. Measurements are taken within the boundaries of the Zoom area. You can only take measurements on one Zoom area at a time.

When you click Cursor, measurements are taken within the boundaries of the vertical cursors. Use the multipurpose knobs to move the cursors.

When you use vertical cursors to control the gating, you can also specify the cursor mode. Use the independent cursor mode to move the cursors one at a time with the multipurpose knobs. Use the tracking mode to have the cursors track one another; moving Cursor 1 automatically moves Cursor 2 in tandem.

Tip Tips for using measurement gating:

- Click, hold, and drag across the part of the graticule that you want to measure. After you define the measurement area, select Measurement Gating from the shortcut menu to place gating cursors around the measurement area.
- Use the same click, hold, and drag method to turn measurement gating off; just select Measurement Gating Off in the shortcut menu.

164

Histogram Control Window

Access this control window from the Measure menu Waveform Histograms command.

Overview

Use the Histogram control window to set up a vertical or horizontal histogram. Use vertical histograms to measure signal noise and horizontal histograms to measure signal jitter.

To Use

Use the Channel Selector tabs to select the source waveform for the histogram measurements.

Next, click the type of histogram that you want to use for your measurement, Horiz (Horizontal) or Vert (Vertical). Once you make this selection, histograms continue to be calculated even if you set Display to Off. You must explicitly click the Histogram Mode Off button to stop accumulating histogram data. Or, click Reset to start a new histogram.

Click either Linear or Log to set the scaling. Adjust the size of the histogram box by clicking Limits under Adjust Histogram Box, and use the multipurpose knobs to set the box boundaries. Move the location of the histogram box by clicking Location under Adjust Histogram Box, and then use the multipurpose knobs to set the X and Y Locations.

Behavior

A histogram consists of a series of narrow bins into which samples are counted. The histogram box defines the graticule area of the histogram and the number of vertical or horizontal bins. Horizontal histograms have a maximum of 500 bins while vertical histograms have a maximum of 200 bins.

The displayed width or height of the bin that contains the most counts is determined by the Histo Size control. The relative sizes for all other bins are scaled proportionally to the bin that contains the most counts.

Histogram limits use the same units as the horizontal and vertical scale controls. When you specify the histogram box in terms of location, the Y location control adjusts the bottom edges of the histogram while the X location control adjust the left edge of the histogram box.

Tip Here are several tips for using histograms:

- Right-click in the Histogram box to access a shortcut menu that will allow you to quickly make changes to the histogram.

- Use the Measure menu or shortcut menu **Reset Histogram** command to reset all bin counts to 0.

- Click, hold, and drag across part of the graticule where you want to set up a histogram and select Histogram from the shortcut menu.

- Use the same click, hold, and drag method to remove a histogram from the display. Select Histogram Off from the shortcut menu.

- You can turn the histogram display off and still accumulate histogram data. This enables you to view the waveform without having the histogram obscure the data. Turn the histogram display on (use the Display button on the right side of the control window) at any time to view the histogram.

- Use the front-panel channel buttons to turn the source waveform display off so that the waveform does not obscure the histogram. Data accumulation continues whether the waveform display is on or off.

- The instrument can perform up to eight automated statistical measurements on the histogram. Allow the histogram to accumulate data for a while, and then turn on the desired measurements.

- You can use Fast Acquisitions mode with histograms. However, only one or two channels can be displayed. If three or more channels are displayed, the Fast Acquisitions mode is turned off. The Fast Acquisition mode is automatically turned back on when you select fewer than three channels.

Delay Measurement Setup

Access this control window by clicking Delay under the Time tab in the Measurement Setup control window, or from the Measure menu Time submenu.

Overview

Use the Delay Measurement Setup control window to set up the instrument to measure the delay between two waveforms.

To Use

Select the source waveforms from the Channel Selector tabs. Use the other controls in the control window to select the Delay Edges, Mid Reference point, Mid2 Reference point, and to specify the search (Delay) direction. Click Ok to add the measurement to the measurement list, or click Cancel to cancel the measurement and return to the Measurement Setup control window.

Behavior

This timing measurement is the time between the midreference crossing of two different waveforms, or the gated region of the waveforms.

The graphic in the control window shows the points on the waveforms from which the delay measurement will be taken. The graphic is updated each time you change the control settings. The Delay direction controls only apply to Source 2. When you click Forward, the instrument starts searching for the desired edge at the beginning of the record. When you click Backwards, the instrument starts searching for the desired edge from the end of the record. If the edge for Source 1 is before the edge for Source 2, then the delay is greater than zero. Otherwise the delay is less than zero.

Phase Measurement Setup

Access this control window by clicking Phase under the More tab in the Measurement Setup control window or from the Measure menu More submenu.

Overview

Use the Phase Measurement Setup control window to set up the instrument to take phase measurements.

To Use

Use the Channel Selector tabs to select the target and reference source waveforms. Then use the Mid Reference control to specify the voltage reference point in the waveform. Click OK to add the

measurement to the measurement list, or click Cancel to cancel the measurement and return to the Measurement Setup control window.

Behavior

This measurement is the amount of time one waveform leads or lags another waveform. Phase is expressed in degrees, where 360° comprise one waveform cycle. The measurement is taken at the voltage level specified by the Mid Reference control.

Mask Menu

Use the Mask menu commands to setup a mask, edit or create user masks or setup a limit test.

Mask Setup Control Window

Access this control window from the Masks menu and select Mask Setup….

Overview

Use these controls to set up a mask test. You can set the type of mask, polarity, display, and alignment.

Note

Option SM must be installed to have access to masks other than a User mask.
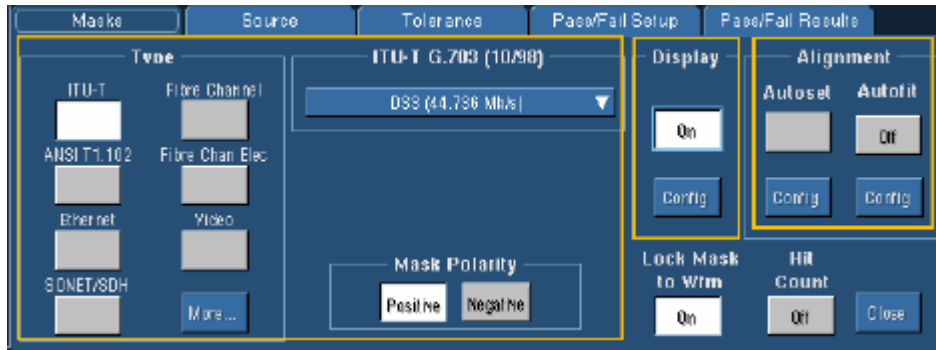
Mask Types and Standards (Option)

Overview

The following tables list the Mask Types and Standards available with Option SM.

Note

The standards available for an instrument depend on the bandwidth and/or configuration of that instrument.

ITU–T masks

None	32Mb 32.064 Mb/s	97Mb 97.728 Mb/s	DS1 Rate 1.544 Mb/s
DS2 Rate Sym 6.312 Mb/s	DS2 Rate Coax 6.312 Mb/s	DS3 Rate 44.736 Mb/s	E1 Sym Pair 2.048 Mb/s
E1 Coax Pair 2.048 Mb/s	E2 8.448 Mb/s	E3 34.368 Mb/s	E4 Binary 0 139.26 Mb/s
E4 Binary 1 139.26 Mb/s	STM1E Binary 0 155.52 Mb/s	STM1E Binary 1 155.52 Mb/s	

ANSI T1.102 masks

None	DS1 1.544 Mb/s	DS1A 2.048 Mb/s	DS1C 3.152 Mb/s
DS2 6.312 Mb/s	DS3 44.736 Mb/s	DS4NA 139.26 Mb/s	DS4NA Max Output 139.26 Mb/s
STS–1 Pulse 51.84 Mb/s	STS–1 Eye 51.84 Mb/s	STS–3 155.52 Mb/s	STS–3 Max Output 155.52 Mb/s

Ethernet mask

None	100Base–TX STP 125 Mb/s	100Base–TX UTP 125 Mb/s	Gigabit Ethernet 1.25 Gb/s
1000B–CX Norm, TP2 1.25 Gb/s	1000B–CX Abs, TP2 1.25 Gb/s	1000B–CX Abs, TP3 1.25 Gb/s	XAUI, Near 3.125 Gb/s
XAUI, Far 3.125 Gb/s	1000B–SX/LX 1.25 Gb/s		

SONET/SDH masks

None	OC1/STM0 51.84 Mb/s	OC3/STM1 155.52 Mb/s	OC12/STM4 622.08 Mb/s	OC48/STM16 2.4883 Gb/s
OC48–FEC 2.666 Gb/s				

Fibre Channel masks

None	FC133 Optical 132.8 Mb/s	FC266 Optical 265.6 Mb/s	FC531 Optical 531.2 Mb/s
FC1063 Optical 1.0625 Gb/s	FC1063 Optical Draft Rev 11	FC2125 Optical 2.125 Gb/s	

Fibre Channel Electrical masks

None	FC133E Elec. 132.8 Mb/s	FC266E Elec. 265.6 Mb/s	FC531E Elec. 531.2 Mb/s
FC1063E Elec. 1.0625 Gb/s	FC1063E Norm, Beta, Transm	FC1063E Norm, Delta, Transm	FC1063E Norm, Gamma, Transm
FC1063E Abs, Beta, Transm	FC1063E Abs, Delta, Transm	FC1063E Abs, Gamma, Transm	FC1063E Abs, Beta, Recv
FC1063E Abs, Delta, Recv	FC1063E Abs, Gamma, Recv	FC2125E Norm, Beta, Transm	FC2125E Norm, Delta, Transm
FC2125E Norm, Gamma, Trans	FC2125E Abs, Beta, Transm	FC2125E Abs, Delta, Transm	FC2125E Abs, Gamma, Transm
FC2125E Abs, Beta, Recv	FC2125E Abs, Delta, Recv	FC2125E Abs, Gamma, Recv	

InfiniBand masks

None	2.5 Optical 2.5 Gb/s	2.5 Electrical 2.5 Gb/s

Serial ATA masks

None	G1 Tx 1.5 Gb/s	G1 Rx 1.5 Gb/s
G2 Tx 3.0 Gb/s (TDS6000 only)	G2 Rx 3.0 Gb/s (TDS6000 only)	

USB 1.1/2.0 masks

None	FS 12 Mb/s	HS:T1 480 Mb/s	HS:T2 480 Mb/s
HS:T3 480 Mb/s	HS:T4 480 Mb/s	HS:T5 480 Mb/s	HS:T6 480 Mb/s

1394b masks

None	S400b T1 491.5 Mb/s	S400b T2 491.5 Mb/s	S400b Optical 491.5 Mb/s
S800b T1 983.0 Mb/s	S800b T2 983.0 Mb/s	S800b Optical 983.0 Mb/s	S1600b T1 1.966 Gb/s
S1600b T2 1.966 Gb/s	S1600b Optical 1.966 Gb/s		

Rapid IO LP–LVDS masks

None	Drv 500 Mb/s	Drv 750 Mb/s	Drv 1.0 Gb/s
Drv 1.5 Gb/s	Drv 2.0 Gb/s	Ext Drv 500 Mb/s	Ext Drv 750 Mb/s
Ext Drv 1.0 Gb/s	Ext Drv 1.5 Gb/s	Ext Drv 2.0 Gb/s	Rcv 500 Mb/s
Rcv 750 Mb/s	Rcv 1.0 Gb/s	Rcv 1.5 Gb/s	Rcv 2.0 Gb/s

Rapid IO Serial masks

None	RIO Serial 1.25 Gb/s	RIO Serial 2.5 Gb/s	RIO Serial 3.125 Gb/s

IOF masks

none	SFI/SPI-5 TA Data 2.488 Gb/s	SFI/SPI-5 TC Data 2.488 Gb/s	SFI/SPI-5 TA Clock 2.488 Gb/s
SFI/SPI–5 TC Clock 2.488 Gb/s	SFI/SPI–5 RB Data 2.488 Gb/s	SFI/SPI–5 RD Data 2.488 Gb/s	SFI/SPI–5 RB Clock 2.488 Gb/s
SFI/SPI–5 RD Clock 2.488 Gb/s	SFI/SPI–5 TA Data 3.125 Gb/s	SFI/SPI–5 TC Data 3.125 Gb/s	SFI/SPI–5 TA Clock 3.125 Gb/s
SFI/SPI–5 TC Clock 3.125 Gb/s	SFI/SPI–5 RB Data 3.125 Gb/s	SFI/SPI–5 RD Data 3.125 Gb/s	SFI/SPI–5 RB Clock 3.125 Gb/s
SFI/SPI–5 RD Clock	VSR OC192/STM64	TFI–5 2.488 Gb/s	TFI–5 3.1104 Gb/s

IOF masks

3.125 Gb/s   1.24416 Gb/s

PCI–Express masks

| None | PCI–Express Transm 2.5 Gb/s | PCI–Express Recv 2.5 Gb/s |

Mask Display Control Window

To Use

Click the On/Off control to turn the mask display on or off. Click the Config control to activate the Mask Configuration Display control window.

Behavior

When the On/Off control is on, the mask is displayed on the graticule. When the On/Off control is off, the mask is not displayed. Clicking the Config control will replace the Mask Setup control window with the Mask Configuration Display control window.

Mask Alignment Control Window

To Use

Click the Autoset control to perform an autoset on the signal. Click the Autofit control to turn the minimize hits function on or off. Click the Config control to access the Autofit or Autoset Configuration control window.

Behavior

Clicking the Autoset control causes the instrument to automatically align the mask based on the characteristics of the input signal. This function is done once, and that setting is not changed until you run the autoset command again or until you change any settings. This is the same as pushing the front-panel Autoset control.

When Autofit is used, the signal is aligned within the mask to minimize the number of hits on each acquisition. Settings may change after each acquisition when autofit is used.

Note

The interaction between Autofit and Persistence can affect the display of persistence data. Autofit moves the waveform vertically and/or horizontally to reduce the number of hits within a given mask. If either infinite or variable persistence is enabled, these movements will clear any persistence data. If Autofit makes frequent adjustments, there can be little or no persistence data.

Clicking the Autoset Config control will replace this control window with the Autoset Mask Configuration control window. Clicking the Autofit Config control will replace this control window with the Autofit Mask Configuration control window.

Mask Source Control Window

To access this control window, select Source… from the Masks menu.

Overview

Use this control window to select the source of the signal that you want to test.
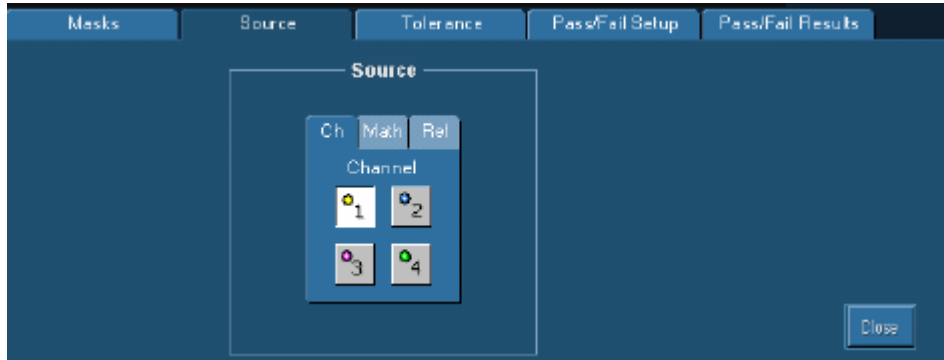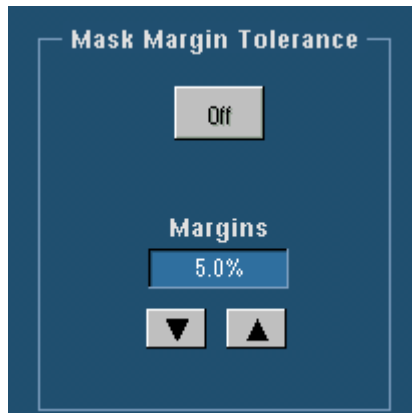

To Use

Click the Ch, Math, or Ref tab to select the source for mask testing. Then click the number (1–4) of the channel, math expression, or reference that you want to use.

Behavior

When you click the Ch tab and select a channel number, it is used as the source for mask testing. You can select only one source at a time for mask testing.

Mask Margins Control Window

To access this control window, select Tolerance Setup… from the Masks menu.

Overview

Use this control to turn the margin mode on or off and to set the percentage of margin used for the mask test.

To Use

Click the On/Off control to turn the mask margin tolerance on or off. Click the up and down arrow buttons or click the text field to activate the pop-up keyboard and set the percentage of margin tolerance.

Behavior

When the mask margin tolerance is on, you can control the percentage of margin used in the mask test. Use the up and down arrow buttons to increase or decrease the percentage in 1% increments. A tolerance that is > 0 makes passing a mask test more difficult. A mask test that passes with a

tolerance that is > 0 demonstrates that the signal passes the test with a margin for further error. Setting the tolerance to < 0 makes a mask test easier to pass. A tolerance that is < 0 can be used to demonstrate by how much of a margin the signal fails.

Mask Pass/Fail Setup Control Window

To access this control window, select Pass/Fail Setup…from the Masks menu.

Overview

Use this control window to set the parameters for passing or failing the mask test, to set the test notification mode, to select the polarity for testing, and whether to repeat the test.

Behavior

Clicking More will show a second control window of failure notifications. When More is clicked in the window below, it will return you to the Pass/Fail Setup control window displayed above. Use the More Pass/Fail Setup control window to save and/or log failed waveforms and to enable E-mail on mask failure.

More Pass/Fail Setup Control Window

174
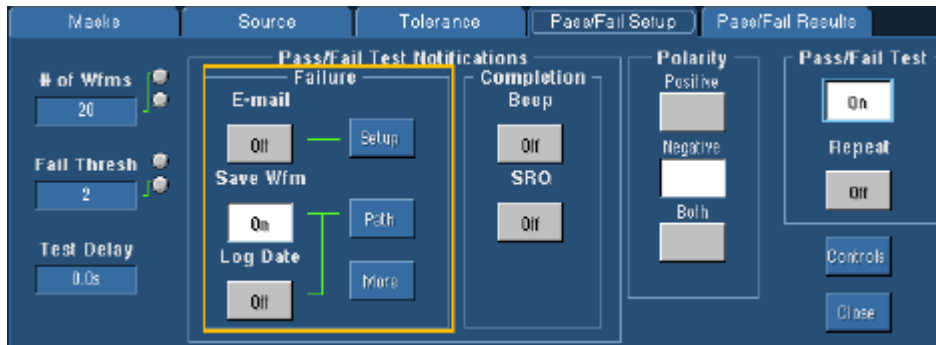
Mask Test Pass/Fail Controls

# of Wfms

20

Fail Thresh

1

Test Delay

0.0seconds

Overview

Use the # of Wfms (Samples), Fail Thresh, and Test Delay to set the parameters for the mask test.

Note

If you are using Waveform Database mode, the # of Wfms text box changes to a Samples text box.

To Use

Click the # of Wfms (Samples) text box to activate the pop-up calculator. Enter the number of waveforms to use in a test. In t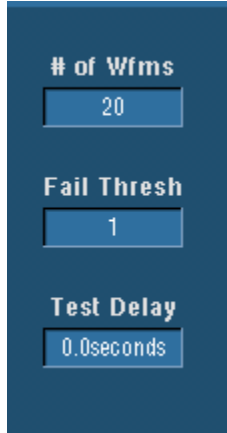he Fail Thresh text box enter the minimum number of waveforms that must fail for the test to fail. Enter the number of seconds to wait before starting the test in the Test Delay text box.
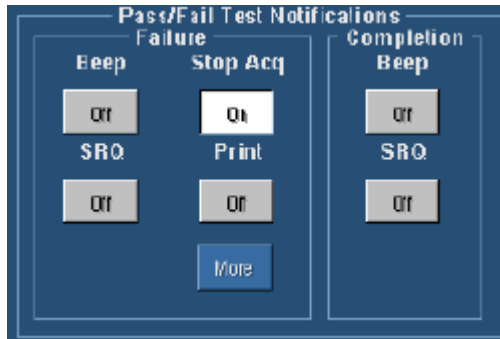
Behavior

Entering a number in the # of Wfms (Samples) text box stops the test when that number has been reached.

If the waveforms or samples that fail are equal to or greater than the number in the Fail Thresh text box, the test has failed. If the failed waveforms or samples are less than the number in the Fail Thresh text box, the test passes.

The instrument begins sampling after the number of seconds has passed that is set in the Test Delay text box. Use the Test Delay if you need to give the signal source time to stabilize.

Mask Pass/Fail Test Notification

Pass/Fail Test Notifications

Failure

Completion

Beep — Off

Stop Acq — On

Beep — Off

SRQ — Off

Print — Off

SRQ — Off

More

Overview

Use the Pass/Fail Test Notification to set the mode of notification, send commands, save waveforms, send e-mail, print, and log data when a waveform fails or a test completes.

To Use

Clicking the controls will switch between on and off. If the control is off, click it to turn it on. Clicking the More control switches between additional controls and the controls shown in the graphic.

Behavior

When Beep in the Failure field is turned on, the instrument will beep when a test fails. When Beep in the Completion field is turned on the instrument will beep when the test is completed.

When SRQ is turned on in the Failure field, it sends a GPIB SRQ command if a test fails. When SRQ is turned on in the Completion field, it sends a GPIB SRQ command when a test completes.

Clicking the Print control sends the waveform to a printer when a test fails.

Clicking the More control switches to the E-mail, Save Wfm, Log Date, and Path controls. Clicking More again returns you to the controls in the graphic.

Pass Fail Setup More

Access this control window from the Masks Pass/Fail Setup control window.
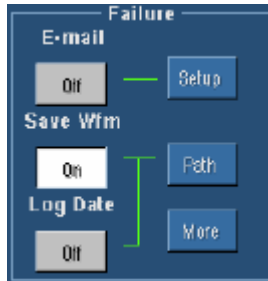
Overview

Use this control window to save a failed waveform, log the date and time of a failed waveform, and specify the directory where you want to save the waveform or send e-mail on a failed waveform.
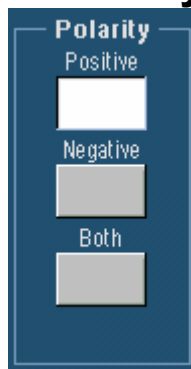
To Use

Click E-mail to turn e-mail on mask failure on. Click Setting to set up e-mail on event if you have not already done so. Click the Save Wfm control to turn the save waveform function on and off. Click the Log Date control to turn the failed waveform log function on and off. Click the Path control to specify the path where you want to save the waveform.

Behavior

When Save Wfm is turned on, the instrument saves waveform data to a default directory unless you specified a different directory using the Path control. A file name is generated by combining the date and time.

When Log Date is turned on, the date and time for each failed mask test is recorded to a file of your choice.

Mask Polarity

Overview

Use the Mask Polarity control to select whether to test the positive or negative pulse or both.

To Use

Click the Positive control to set the polarity to positive, the Negative control to set the polarity to negative, and the Both control to set the polarity to both.

Behavior

When the Positive control is selected, the mask test is performed on the positive pulse. When the Negative control is selected, the mask test is performed on the negative pulse. When the Both control is selected, the mask test is performed on both the positive and negative pulses.

Mask Test Repeat

Overview

Use the controls in the Pass/Fail Test field to start the test or to run the test continuously. Use the Controls button to open the Controls control window.

To Use

Click the Repeat control to run the test continuously. Click the On/Off control to start the test. Click the Controls button to open the Mask Test Controls control window

Behavior

When the Repeat control is on, the test runs continuously. When the On/Off control is clicked, the test starts using the parameters from the Mask Setup control window and the Pass/Fail Results window is displayed. When the Controls button is clicked the Mask Test Controls control window is displayed.

Mask Pass/Fail Results Control Window

Access this control window from the Masks menu and select Pass/Fail Results….

Overview

Use this control window to view real time test results, start the test, reset the test at the beginning, activate the waveform failure log file, and open the Mask Test Controls control window.
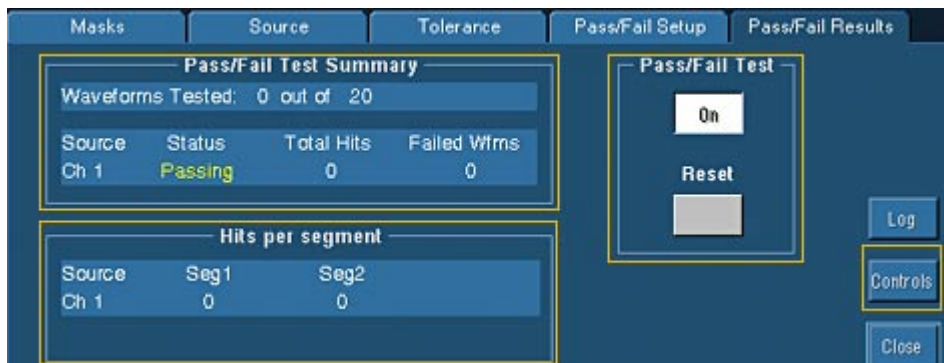
Mask Test Summary

Overview

Use this window to view real time test results.

```
┌──────────── Pass/Fail Test Summary ────────────┐
│ Waveforms Tested:  4  out of  20                │
│                                                 │
│ Source      Status      Total Hits   Failed Wfms│
│ Ch 1        Failed         11             3      │
└─────────────────────────────────────────────────┘
```
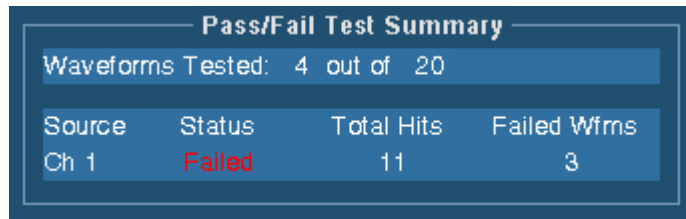
To Use

The Pass/Fail Summary is a read only field.

Behavior

The Pass/Fail Summary fields provide a real time summary of the test results. Waveforms or Samples Tested shows how many waveforms or samples have been tested in relation to how many were specified in the test setup.

Source indicates what signal source was used for the test.

Status indicates whether the test has passed, failed, or is passing. Note that passing appears in yellow, passed in green, and failed in red.

Total Hits indicates the total number of data points that violated the mask.

Failed Wfms shows the total number of waveforms that failed during the test sequence.

Mask Hits Per Segment

Overview

Use this window to view the hits for each mask segment of the test.

```
┌──────────── Hits per segment ────────────┐
│ Source      Seg1        Seg2       Seg3   │
│ Ch 1         0           11         0     │
│                                           │
└───────────────────────────────────────────┘
```

To Use

The Hits per segment is a read only field.
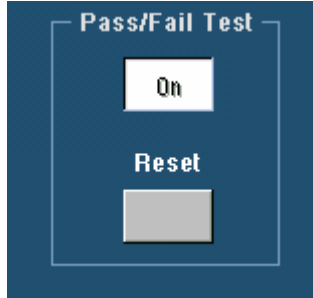
Behavior

Each mask test can have up to 16 segments. The data point violations per segment are shown in this window. Source indicates the input signal in use for the test. The segments are identified by Seg followed by the number of the segment. Only the number of segments used in a test is shown. A scroll bar appears if there are more segments used than can be seen at one time.

The segments of a mask are the polygonal regions on the screen. Segment one is the left upper most polygon. Segments are assigned from top to bottom and left to right on the screen.

Mask Test Start or Reset

Access these controls from the Masks Pass/Fail Results Setup control window.

Overview

Use this control window to start a test or reset the test to the starting point.
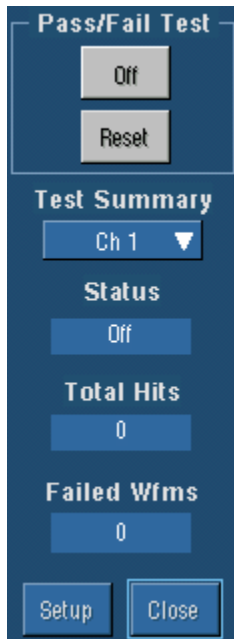
To Use

Click the On/Off control to start the test. Click the Reset control to reset the test to the beginning.

Behavior

When the test starts, it uses the parameters in the Pass/Fail Test Setup control window. When the test is reset, the Total Hits, Failed Wfms, Hits per segment, and Waveforms Tested fields are reset to zero.

Mask Test Controls Control Window

Access this control window by clicking the Controls button on the Pass/Fail Results Setup control window.

Overview

Use this control window to run a test and view the results. You can also reset or start the test and return to the Mask Setup control window from this control window.

To Use

Click the On/Off control to start and stop the test. Click the Reset control to reset the test. Click the drop-down arrow in the Test Summary field to select a different source. Status, Total Hits, and Failed Wfms are read only fields. Click the Setup control to return to the Mask Setup control window.
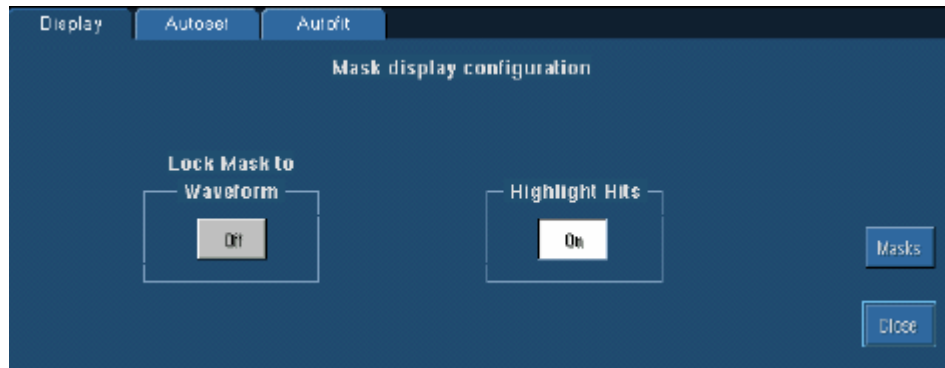
Behavior

This control window allows a mask test to run on a full screen and still provide access to the start, stop and reset functions. In addition, you can see the test summary for each input selected.

Mask Configuration Display Control Window

Access this control window from the Masks menu and select Config… or from the Masks Setup control window.

Overview

Use this control window to set whether changes in the scale will also be made in the mask. You can also use this control window to return to the Mask Setup control window.

To Use

Click the Lock Mask to Waveform button to turn it on and off. Click the Masks button to go to the Mask Setup control window.

Behavior

When the Lock Mask to Waveform button is turned on, any changes you make to the scale, vertical position or horizontal position will be reflected in the mask. Turning on Highlight Hits displays the hits in a different color so they can be easily seen. When you click the Mask button this window closes, and the Mask Setup control window opens.

Mask Configuration Autoset Control Window

Access this control window from the Masks menu and select Config… or from the Masks Setup control window.

Overview

Use this control window to set the vertical, horizontal, and trigger autoset parameters, activate autofit or autoset, choose the autoset mode, return to the default autoset configuration, or go to the Mask Setup control window.

Clicking Autoset aligns the signal within the mask. This operates the same as the AUTOSET button on the front panel. Clicking Defaults returns the instrument to the default autoset configuration. Clicking Masks replaces this control window with the Mask Setup control window.
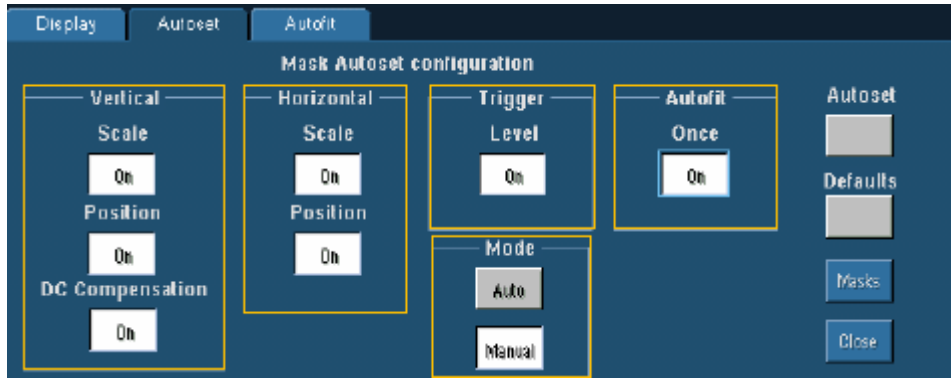
Mask Autoset Vertical Controls
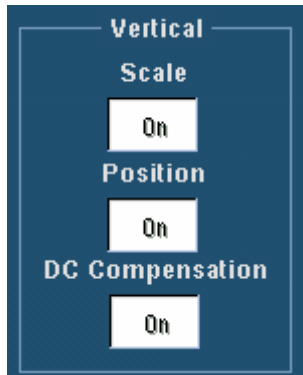
Overview

Use these controls to choose whether to have autoset affect the vertical scale or position. You can also choose whether to have autoset affect DC compensation.

To Use

Click the Scale, Position, and DC Compensation controls to turn these vertical control functions on or off.

Behavior

Turning on the Scale, Position, or DC compensation sets these vertical adjustments up to be performed on an autoset.

Mask Autoset Horizontal Controls

Overview

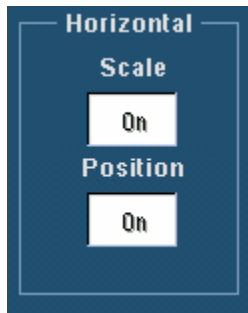Use these controls to choose whether to have autoset affect the horizontal scale or position.

To Use

Click the Scale and Position controls to turn these horizontal control functions on or off.

Behavior

Turning on the Scale or Position horizontal control functions sets up these adjustments to align the signal within the mask when autoset is done.

Mask Autoset Trigger Level Control

Overview

Use this control to choose whether to have autoset affect the trigger level.

To Use

Click the Trigger Level control to turn it off or on.

Behavior

When the Trigger Level control is turned on, the trigger levels are adjusted to match the signal when autoset is performed.

Mask Autoset Autofit Control

Overview

Use this control to minimize hits.

To Use

Click the Autofit Once control to autofit the waveform after autoset is performed.

Behavior

When Autofit is turned on, the waveform is autofitted once at the end of every autoset. Autoset will take longer but hits will be minimized. Autofit will only move the signal if there are mask hits.

Mask Autoset Mode

Overview

Use these controls to select whether autoset will be done automatically or manually.

To Use

Click Auto to have autoset done automatically after a standard mask is selected. Click Manual to make adjustments manually.

Behavior

If Auto is selected, an autoset is done automatically after a standard mask is selected. The purpose of the manual autoset mode is to adjust everything that can be done without having any waveform data.

Mask Configuration Autofit Control Window

Access this control window from the Masks menu and select Config… or from the Masks Setup control window.

Overview

Use this control window to configure the limits for minimizing mask hits, return to default setting, or return to the Mask Setup control window.
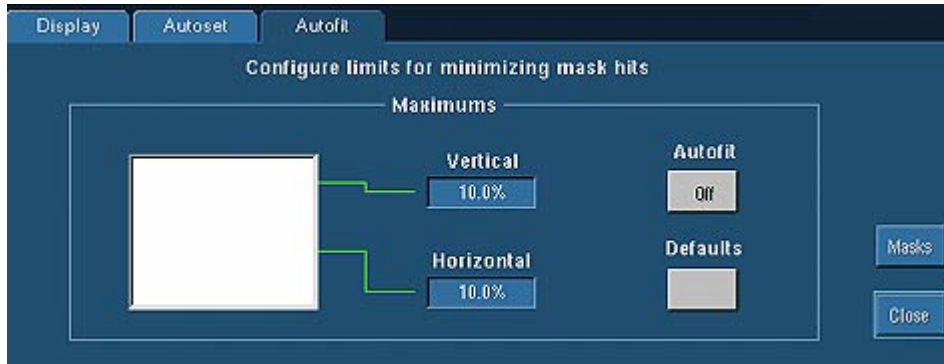
To Use

Set the vertical and horizontal maximum levels of adjustment by clicking the control, and then use the multipurpose knobs or double-click the control and use the pop-up keypad. Click Autofit to align the signal within the mask. Click Defaults to reset the autofit configuration to the default values. Click Masks to return to the Mask Setup control window.

Behavior

The maximum level of adjustment determines how far Autofit will search for the best fit to minimize hits. For example, if you set the Vertical Maximum to 10% div, Autofit searches up to and including 10 percent of the vertical divisions for the best fit. Autofit minimizes hits by aligning the signal within the mask.

Clicking Masks replaces the Mask Configuration Autofit control window with the Mask Setup control window.

Note

The interaction between Autofit and Persistence can affect the display of persistence data. Autofit moves the waveform vertically and/or horizontally to reduce the number of hits within a given mask. If either infinite or variable persistence is enabled these movements will clear any persistence data. If Autofit makes frequent adjustments, there can be little or no persistence data.
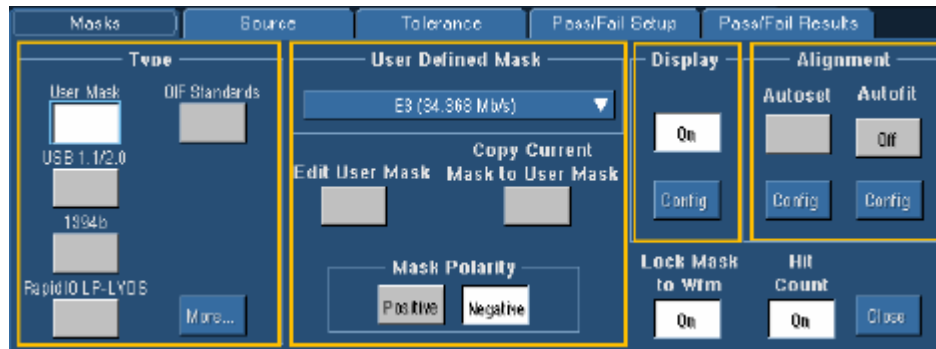
User Mask Setup

Overview

Use this control window to access the Mask Edit control window or to copy a standard mask as a user mask. You can also select the standard that you want to copy or edit from the standard drop-down list.

Note

Option SM must be installed to have access to mask types other than a User mask.

To Use

From the Mask Setup control window, click More, and then click the User Mask type control. Click the Edit User Mask control to access the Mask Edit control window. Click the Copy Current Mask to User Mask to make the current mask a user mask.

Behavior

Clicking the User Mask control gives you access to the User Mask in the standard drop-down list. Clicking the Edit User Mask control will replace this control window with the Mask Edit control window. Clicking the Copy Current Mask to User Mask control will activate a dialog box where you can overwrite the user mask.
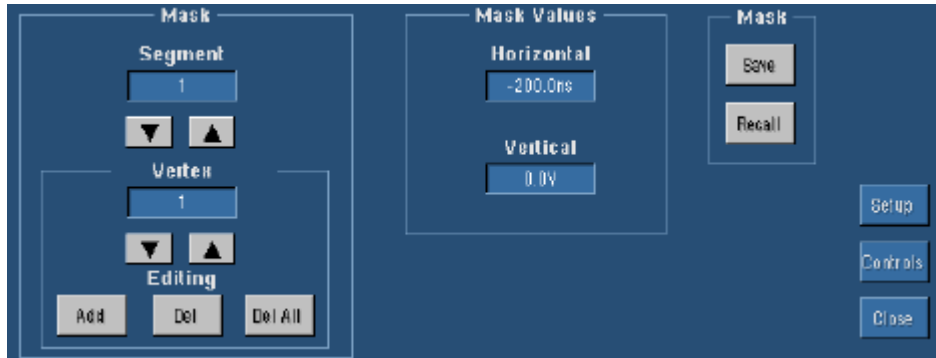
Mask Edit Control Window

Access this control window from the Mask menu by selecting Mask Edit Setup….

Overview

Use this control window to select the segment and vertex for editing, set the Vertical and Horizontal Mask Values, and save or recall masks.

To Use

Click the Mask Segment and Vertex up and down arrows to select the segment or the vertex of the mask to be edited. Click Add to add a point to the boundary in the selected segment and just after the selected vertex. Click Del to delete the selected vertex from the selected segment. Click Del All to remove all points in the selected segment.

To change the horizontal or vertical values for the selected segment and vertex, click the Mask Values Horizontal or Vertical text box to activate the pop-up keypad. To save a mask, click Save to activate the Save As dialog box. To recall a previously saved mask, click Recall to activate the Recall dialog box.

Behavior

The Segment up and down arrows increase or decrease the segment number by one. The Vertex up and down arrows increase or decrease the vertex by 1. To set a specific number, click the Vertex text box to activate the pop-up calculator.

Clicking the Add button adds a point to the boundary in the selected segment and just after the selected vertex in addition to moving any points that follow the newly added point in the boundary up one. For example, if there are 6 points in the boundary of the selected segment and you add a point when the selected vertex is 4, then the vertex which was 4 becomes vertex number 5, number 5 becomes 6, and vertex 6 is changed to vertex 7.

Clicking Del deletes the selected vertex from the selected segment and moves any points that follow the deleted point down one in the boundary. Clicking Del All deletes all points in the selected segment, effectively deleting that segment.

Clicking the Setup button will return you to the Masks Setup control window. Clicking the Controls button will activate the User Mask Edit control window where you can make quick changes to a User mask.

Recall User Mask

Overview

Use this dialog box to recall saved user masks. Masks are saved in **C:\TekScope\Masks** unless you choose another directory.
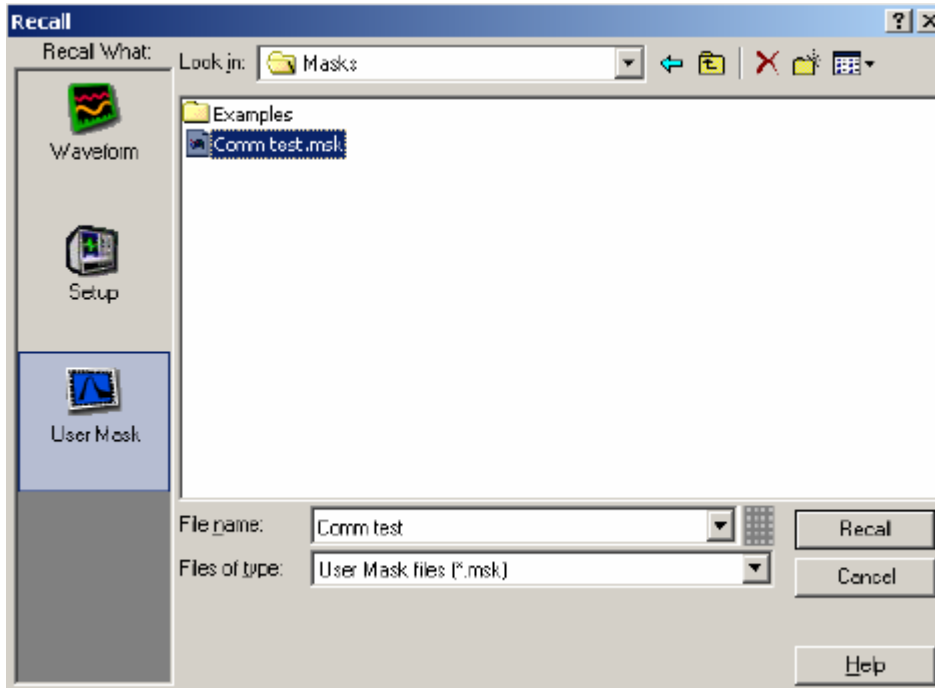
To Use

Click the name of the mask file that you want to recall to select it, and then click Recall. You can also click the keyboard icon to access the pop-up keyboard and enter the file name.
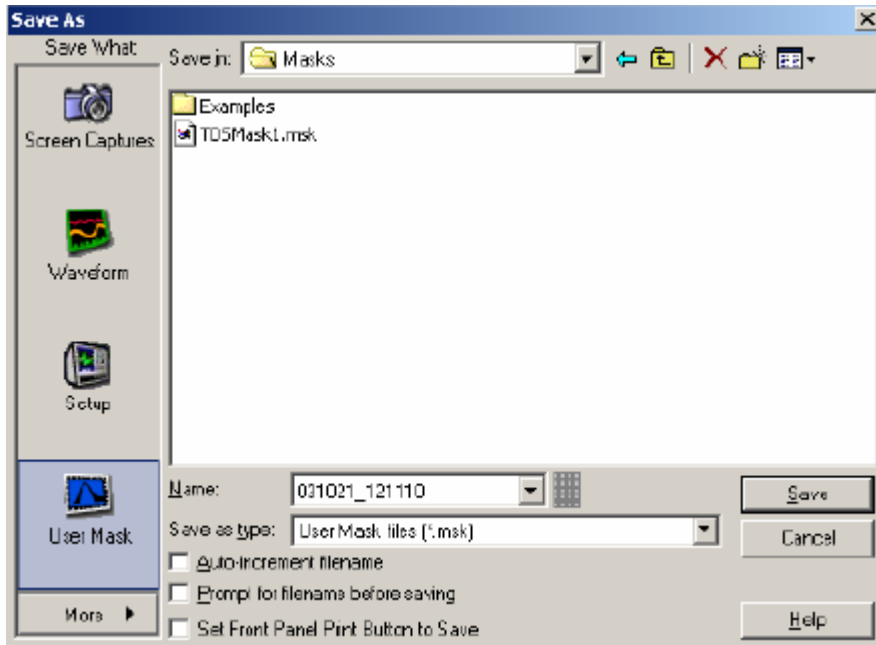
Behavior

The Recall dialog box displays when you click Recall in the Mask Edit control window. Clicking Recall replaces the current mask with the mask that you selected in the Recall dialog box.

Save User Mask

Overview

Use this dialog box to save user masks. Masks are saved in **C:\TekScope\Masks** unless you choose another folder.

To Use

To save the mask with the default file name, click Save. To enter a different file name, click the keyboard icon and use the pop-up keyboard to enter a unique file name. The extension specified in the Save as type list will automatically be appended to the file name.

Behavior

The Save As dialog box displays when you click Save in the Edit Mask control window. The mask is saved in the format specified in the Save as type list.

User Mask Edit Control Window

Access this control window by clicking Controls in the Mask Edit control window.

Overview

Use this control window to select the segment and vertex for editing and set the Vertical and Horizontal Mask Values.

To Use

Click the up and down arrows to select the segment or the vertex of the mask to be edited. Click the Add control to add a point to the boundary in the selected segment and just after the selected vertex. Click Del to delete the selected vertex from the selected segment. To change the horizontal or vertical values for the selected segment and vertex, click the Horizontal or Vertical text box to activate the pop-up keypad.

Behavior

The Segment up and down arrows increase or decrease the segment number by one. The Vertex up and down arrows increase or decrease the vertex by 10. To set a specific number click the Vertex text box to activate the pop-up calculator.

Clicking the Add control adds a point to the boundary in the selected segment and just after the selected vertex in addition to moving any points that follow up one. Clicking Del deletes the selected vertex from the selected segment and moves any points that follow down one.

Limit Test Setup Control Window

Access this control window from the Mask menu and select Limit Test Setup….

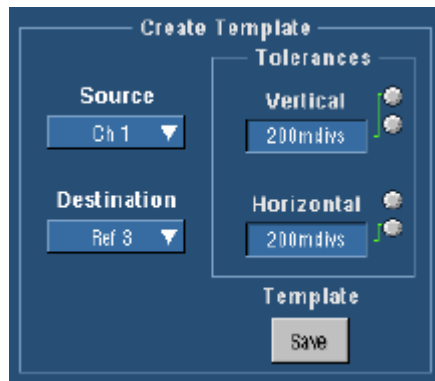### Overview

Limit testing allows you to use channel, math, or reference waveforms and compare them against a template waveform. You set the limits around a template waveform. Let your instrument find the waveforms that fall outside of those limits.
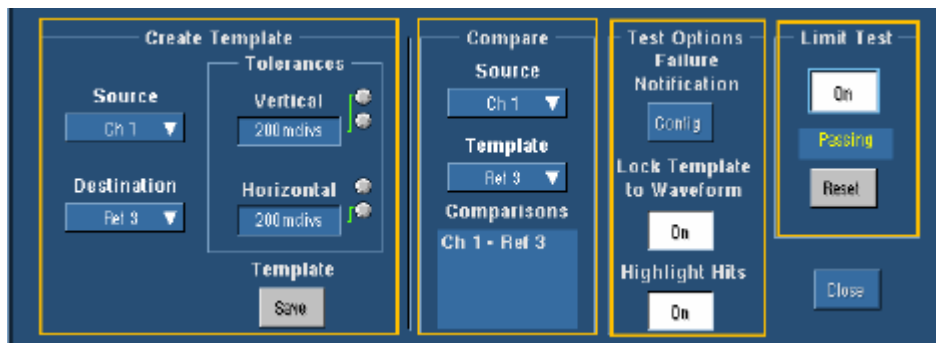
Use these controls to set up a limit test. You can create a template, set tolerances, compare data, and set up test options from this control window.

Create Limit Test Template

Access this control window from the Mask menu and select Limit Test Setup….

Overview

You can specify vertical and horizontal tolerances, create, and store the limit test templates.

To Use

Click the Source drop-down list to select the source you want to use to create the template.

Note

You cannot use waveforms in FastAcq, wfmDB, FastFrame, roll or XY(Z) modes as a template source or as an active mode during limit testing.

Click the Destination drop-down list to select the location for saving the template. Templates are stored in Reference Waveforms Ref1–Ref4 (Ref1–Ref2, for 2-channel instruments) or File. If you select File, you can choose the location for saving the template from the Save Reference Waveform As dialog box.

Set tolerances by using the multipurpose knobs or the pop-up keypad.

Tip Your template waveform will be smoother if you use Average acquisition mode to acquire it. Use Envelope acquisition mode to create template waveforms that include occasional overshoots.

Behavior

The Vertical tolerance is adjustable using the upper multipurpose knob and Horizontal tolerance is adjustable using the lower multipurpose knob. Adjustments are made in 20 m/div increments. Tolerances set the amount the waveform can vary without violating the template.

The source you select is used to create the template. The Source drop-down list contains the following choices: Ch1–Ch4, Math1–Math4, and Ref1–Ref4.

The Destination drop-down list contains the following choices: Ref1–Ref4 and File (only Ref1–Ref2 are available on 2-channel instruments). Selecting File activates the Save Reference Waveform As dialog when you click Save. Clicking Save when Ref1–Ref4 is selected saves the template to the selected reference memory.

Compare Limit Test

Access this control window from the Mask menu and select Limit Test Setup…

Overview

Use these controls to specify which waveforms to use for limit testing and which waveforms to compare them against.

To Use

Click the Source drop-down list to select a waveform for comparing against the template.

Click the Template drop-down list to select the template waveform to which you want to compare the source waveform. The Comparisons readout shows all active comparisons being made. You can perform more than one limit test at a time.

Behavior

The Source drop-down list contains the following choices: Ch1-Ch4, Math–Math4, and Ref1–Ref4 (only Ch1–Ch2, Math1–Math2, and Ref1–Ref2 are available on 2-channel instruments). The source you select is compared against the selected template during a limit test.

The Template drop-down list contains the following choices: None and Ref1–Ref4 (only Ref1–Ref2 are available on 2-channel instruments).
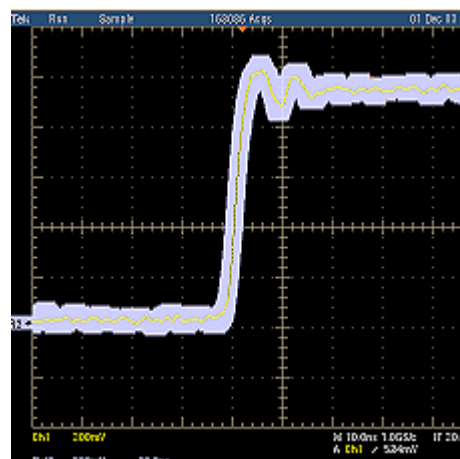
The default setting for Source is Channel 1 and for Template is None.

Tip To easily view the waveform data as well as the template envelope, it is useful to select the Dots display style. See the following graphics to compare the two styles:
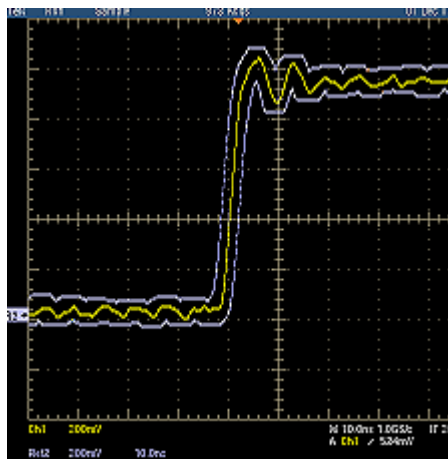
<table>
<tr><td>Dot style</td><td>Vector style</td></tr>
</table>

Limit Test Options

Access this control window from the Mask menu and select Limit Test Setup….

Overview

Use these controls to set up test failure notification, lock the template to a waveform, and highlight hits.

To Use

Click Config to open the Failure Notification control window where you can set up the action you want your instrument to take if the limit test fails.

Clicking the Lock Template to Waveform and Highlight Hits controls will switch between on and off. If the control is off, click it to turn it on.

Behavior

Clicking the Config button under Failure Notification activates the Failure Notification control window. You can choose one or more actions that you want your instrument to take if the limit test fails.

Turning on the Lock Template to Waveform locks the source waveform and template such that adjusting the vertical position or scale of either affects both. Turning off Lock Template to Waveform allows you to move the template and source waveform vertically, independent of each other.

The default for Highlight Hits is On. When highlight hits is on, any points on the source waveform that fall outside of the template are highlighted in a different color than the rest of the waveform.

Limit Test Failure Notification Control Window

Access this control window from the Mask menu, select Limit Test Setup…, and then click Config.

Overview

Use the Failure Notification control window to set the mode of notification, send commands, save waveforms, print, send email and log data when a limit test fails.
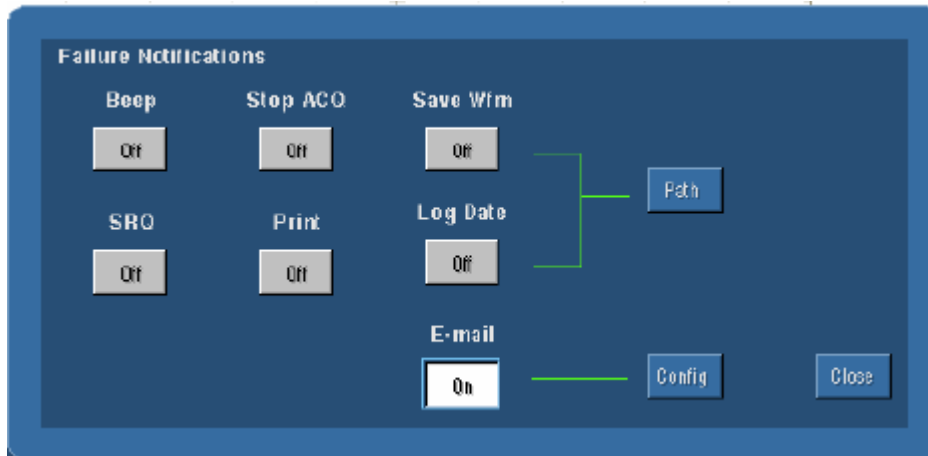
To Use

Click the On/Off buttons to turn that mode of notification on or off.

Click Path to configure the path for saving waveforms and the Log Date.

Click Config to open the E-mail on Event Setup control window.

Behavior

When a limit test fails and Beep is turned on, the instrument beeps, when SRQ is turned on, it sends a GPIB SRQ command, and when Stop ACQ is turned on, the instrument stops acquiring.

When Print is turned on and a limit test fails, the instrument sends a screen print to the default printer. The current page setup is used for the screen shot. There is no specific page setup for limit test failures.

When Save Wfm is turned on and a limit test fails, the instrument saves waveform data to a default directory unless you specified a different directory using the Path button. A file name is generated by combining the date and time.

When Log Date is turned on and a limit test fails, the date and time for each failed limit test is recorded to a file of your choice.

When E-mail is turned on and a limit test fails, the instrument sends an e-mail message to the specified recipient(s). You must enter an e-mail address and the SMTP server name in the E-mail on Event Setup control window before you can turn on E-mail. Clicking Config opens the E-mail on Event Setup control window where you can enter the e-mail address(es) and an SMTP server name. Check with your system administrator for the correct server name.

Limit Test Reset

Access these controls from the Mask menu and select Limit Test Setup….
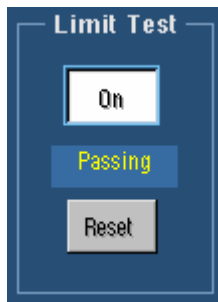
To Use

Click On/Off to turn the limit test on or off. Click Reset to clear the status and highlighted hits.

Behavior

When you turn the limit test on, the status will indicate either Passing or Failing. When the test is turned off, the status will indicate either Passed or Failed. Clicking Reset clears all the highlighted hits and the status field.

Save Limit Test Dialog Box

Overview

Use this dialog box to save limit test templates. Template waveforms are saved in **C:\TekScope\\Waveforms** unless you choose another folder.

To Use

Use this dialog box to save the limit test template to a reference location or a file. To save the template to a reference location Select the Source from the drop-down list, and then select one of the four reference locations. Click Save.

To save to a file, click the keyboard icon to enter a file name; or select Auto-increment file name if you want to save a series of files without typing in a new name each time. The file extension specified in the Save as Type list will automatically be appended to the file name.
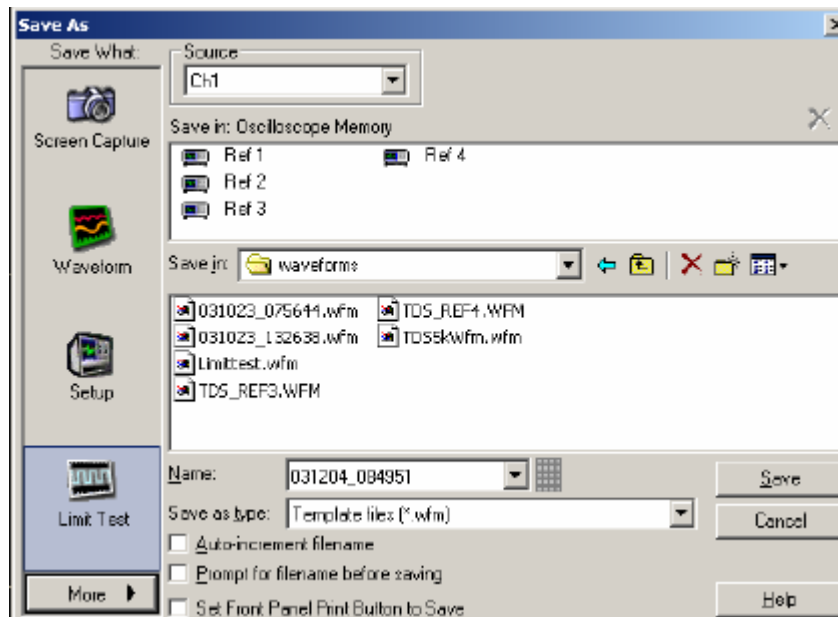
Behavior

The file extension specified in the Save as Type list will automatically be appended to the file name.

Math Menu

Math menu options enable you to perform mathematical operations on waveforms, build algebraic expressions, perform math-on-math operations, analyze histograms, or use advanced spectral analysis.

Math Setup Control Window

Access this control window from the Math menu Math Setup command, or from the Math toolbar button.

Overview

Use the Math Setup control window to set up and define math waveforms.
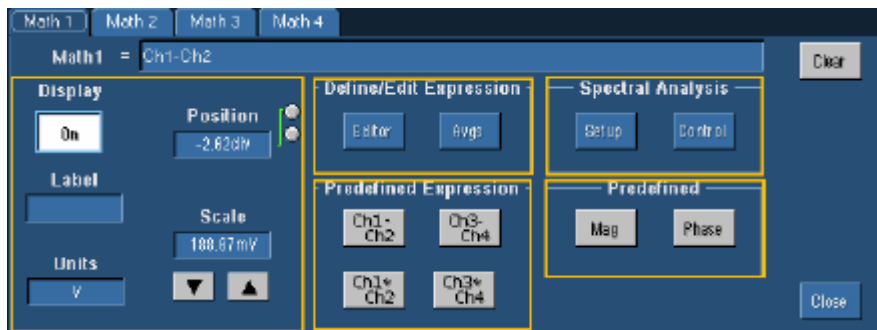
You can define up to four math waveforms (two for the 2-channel models). The controls for each waveform are similar, but independent of each other.

Tip To easily change the vertical position and scale of a math waveform, click the waveform handle on the left side of the screen. Use the multipurpose knobs to change the waveform position and scale.

Math Waveform Properties

To Use

Use these controls to change the vertical position and scale of the selected math waveform. Click Display to turn the math waveform display on and off. Use the Label to attach a meaningful label to the waveform. Use the Units to assign custom vertical units for each math waveform.

Behavior

These controls only affect the selected math waveform. The Position control increases or decreases the vertical position of the waveform one pixel at a time. The Scale control changes the displayed waveform only; it has no effect on the acquisition hardware.

Define Edit Expression

These buttons provide quick access to the Equation Editor and to the Math Parameter control window.

Click **Editor** to open the Equation Editor control window. Use the Equation Editor to create a math equation for the selected waveform.

Click **Avgs** to open the Adjust Math Parameters control window. Use the Math Parameter control window to define the number of times that you want to average the selected math waveform.

Predefined Math Expressions

To Use

Click one of the buttons to use one of the predefined math expressions to create the math waveform based on live (channel) waveforms.

Tip If the channels are active, you can access each expression directly from the Math menu.

Behavior

These buttons provide common predefined math expressions.

The Ch1-Ch2 and Ch3-Ch4 selections create math waveforms that are the difference between two input waveforms. Use these functions to view differential signals.

The Ch1*Ch2 or Ch3*Ch4 selections create math waveforms that are the product of the two input waveforms. For example, you can obtain a power waveform by multiplying a voltage waveform by a current waveform.

Note

The Ch3-Ch4 and Ch3*Ch4 selections are only available with 4-channel models.

Tip Use the Equation Editor to manually create and edit math waveforms.

Guidelines for Working with Math Waveforms

Keep the following guidelines in mind when working with math waveforms:

- Keep math waveforms simple. If the math expression becomes too complex, try separating the expression into more than one math waveform and then combining the waveforms (for example, Math1 = Math2 + Math4).

- You cannot use math waveforms with Fast Acquisition mode.

- Math waveforms cannot be turned on without a math expression.

- Math waveforms cannot be created from a reference waveform that was saved in Fast Acquisition mode.

- To avoid syntax errors in a math expression, verify the use of operators, parentheses, operands, and the spelling of functions before activating the math waveform.

- The math expression is applied to each frame when working with FastFrame (otherwise integral, spectral, or other functions may not work properly).

- When working with reference waveforms and FastFrame, if one or more reference waveforms are used as part of a math waveform, the number of frames in the math waveform is the smallest of all source waveforms (reference, math, or channel waveforms).

- If one or more reference waveforms are used in a math waveform, the record length is equal to the smallest of all the source waveforms (reference, math, or channel waveforms). The math is calculated using the first point from each source, followed by the next point, and so forth. This is true even if the sources have different times between points in the record.

Spectral Analysis

Click **Setup** to open the Spectral Setup control window to create math equations to analyze waveforms in the frequency domain.

Click **Control** to open the Spectral Controls control window to define the acquisition, gating, and frequency setups for the frequency domain analysis.

Predefined Spectral

To Use

Click one of the buttons to define the current math waveform as a spectral magnitude waveform or a spectral phase waveform.

Behavior

The math waveform is based on the last active waveform on the screen. Spectral waveforms are created from channel waveforms; you cannot create a spectral waveform from math or reference waveforms. Click **Mag** to create a spectral magnitude waveform. Click **Phase** to create a spectral phase waveform.

Equation Editor Control Window

Access the Equation Editor from the Math menu, or from the Editor button on the Math Setup control window.

Overview

Use the Equation Editor control window to define or edit advanced math waveforms. The readout shows the math expression as you construct it. New functions are added at the beginning of the expression each time that you click one of the function buttons.

You can create a math expression using one or more of the following methods (see also Guidelines for Working with Math Waveforms):

- On the Time tab, click one or more of the function buttons to insert a time domain function into the math expression. Then select one of the waveforms as an operand by clicking one of the Channel Selector tabs. Use the edit controls to add additional numbers and operators to complete the math expression.

- On the Meas tab, click one or more of the buttons to add the corresponding defined measurements. Use the edit controls and the Channel Selector tabs as necessary to add waveforms, additional numbers, and operators to complete the math expression.

- Use the edit controls with the Channel Selector tabs to create simple math expressions using existing waveforms, numbers, and operators.

- Use an external keyboard to enter the math expression.

- Create a math expression from a text editor or from another application. Copy the text to the Windows clipboard. Right-click with the mouse to paste the text into the math expression.

Click **Apply** to apply the math expression to the selected math waveform. Click **Avgs** to open the Math Parameter Adjustment control window where you can specify the number of times the math waveform is to be averaged.

Tip If you create a math expression that is longer than the physical space in the control window, you can scroll through the expression using the Text Edit controls.

To edit another math waveform, select the waveform from the Math (n) drop-down list in the Equation Editor control window.

Time Domain Tab
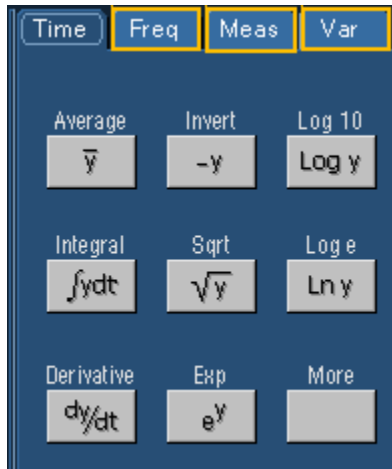
To Use

Use these controls to insert time domain-based functions into the math expression. Click one of the function buttons to insert the function into the math expression. Click **More** for additional, available functions.

Behavior

You can use any of the channels as arguments for the function in the following table. Create nested functions by inserting one function into the argument portion of another.

Available Functions

Average	Invert	Log 10
Integral	Sqrt	Log e
Derivative	Exp	

Fabs	Ceil	Floor
Sin	Cos	Tan
Min	Max	

ASin	ACos	ATan
Sinh	Cosh	Tanh

 Some things to consider when working with math waveforms:

- Scale and position the source waveform so that it is contained on screen. Off-screen waveforms may be clipped, resulting in errors in the math waveform.

- Use the vertical position and vertical offset to position the source waveform. The vertical position will not affect the math waveform unless you position the source waveform off screen so that it is clipped.

Spectral Analysis (Freq) Tab
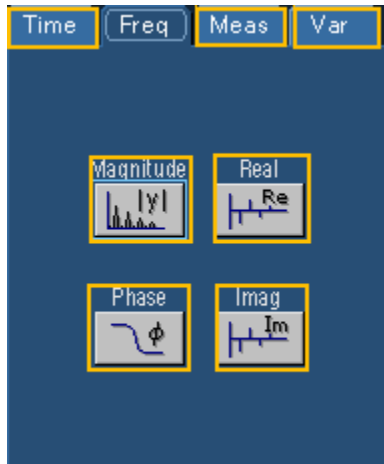
To Use

Use these controls to insert spectral analysis-based functions into the math expression. Click one of the function buttons to insert the function into the math expression. Then select one of the defined waveforms as an argument by clicking one of the Channel Selector Tabs.

Behavior

You can use any of the channels as arguments for the function. Create nested functions by inserting a function into the argument portion of another.

Measurement List (Meas) Tab

To Use

Use the list to enter one of the predefined measurements into the math expression. Click the button adjacent to an item to insert the measurement into the expression.

Behavior

The list contains the measurements defined in the Measurement control window. The buttons are only active for defined measurements; they are unavailable for any undefined measurements.

Measurements are continuously updated on live acquisitions. The most recent measurement values are always used in math expressions.

Variable List (Var Tab)

To Use

Click in the text box and use the multipurpose knobs to enter a value for the variable. Click the VAR# button to enter that variable into the math expression. Click Controls to access the Math Variables control window.

Behavior

You can assign up to eight different variables to add to your math expression. Clicking the VAR# button enters the variable into the math expression. Clicking the More button accesses the other four variables. The Controls button activates the Math Variables control window where you can quickly set or change the math variables.

Edit Controls

To Use

Use the Edit controls to insert numerals, math operators, and variables into a math expression. Use the Text Edit controls to move the insertion point (cursor) within the expression. Click an individual button to enter its character or operator into the expression.

Behavior

The characters are entered into the math expression one at a time.

Adjust Math Parameters Control Window

Access this control window by selecting Set Math Averages… from the Math menu.
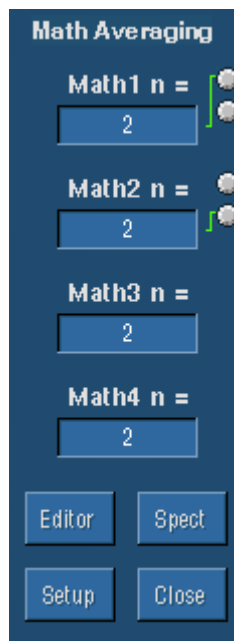
Overview

Use this control window to specify the number of times the selected math waveform is averaged.

To Use

Click the Math*n* control to map it to one of the multipurpose knobs. Then use the knob to specify the number of times the selected waveform should be averaged.

Behavior

The selected math waveform is averaged over the number of acquisitions shown in the control. Each math waveform has its own set of controls.
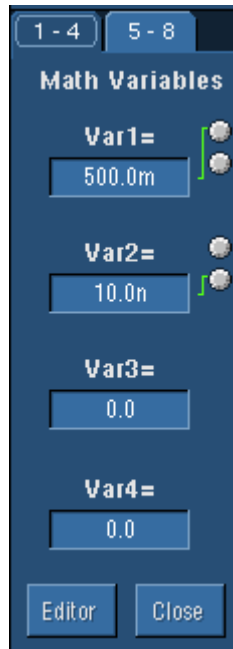
The buttons at the bottom of the control window provide access for changing other setups after you set the averaging:

- Editor opens the Equation Editor
- Spect opens the Spectral Setup control window
- Setup opens the Math Setup control window

Math Variables Control Window

Access this control window by selecting Set Math Variables… from the Math menu.

Overview

Use this control window to quickly set the math variables to use in your math expression.

To Use

Use the 1-4 or 5-8 tab to access the variables that you want. Click in the Var#= text box, and then use the multipurpose knobs to set the value for the variable. Click Editor to access the Math Equation editor.

Behavior

You can set or change the variable values in this control window. To add a variable to a math expression, go to the math editor, and click the VAR# button.

Spectral Setup Control Window

Access this control window from the Math menu Spectral Setup command, or by clicking Spect in the Adjust Math Parameters control window.

Overview

Use these controls to set up a math waveform for spectral analysis. Use the controls on the tabbed pages to create a spectral waveform, define the vertical scale units of the waveform, and to lock waveforms together.

After you create the spectral waveform, use the controls on the right side of the control window to position and adjust the waveform so that you can view the data you are interested in.

You can operate the instrument and view spectral (FFT) waveforms using the time-domain controls, gating controls, or frequency domain controls. Some of these controls are interactive; making changes with one set of controls will sometimes change settings in another control set.

Create Spectrum Tab
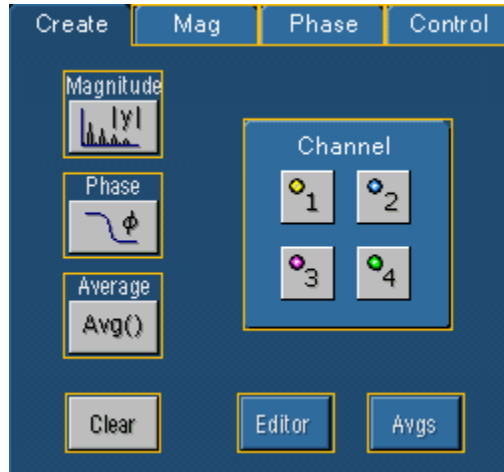
To Use

Use these controls to insert spectral analysis-based functions into the math expression. Click one of the buttons to insert the function into the expression. Then select one of the input channel waveforms to create the Math waveform.

Behavior

These controls work similar to the controls under the Spectral Analysis (Freq) tab of the Equation Editor.

Click **Editor** to open the Equation Editor if you need to edit the math expression. Click **Clear** to clear the math expression. Click **Avgs** to open the Math Parameter Adjustment control window where you can specify the number of times to average the math waveform.

After making your changes in the control window, click **Apply** to apply the changes to the waveform without closing the control window. Click **OK** to apply the changes and close the control window.

Mag Scale Tab
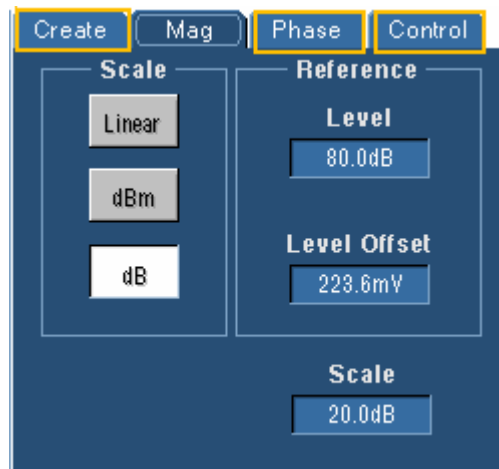
To Use

Use these controls to specify the vertical scale for magnitude waveforms. Click one of the buttons to specify the vertical units as linear (V, A, W), dBm (1mW into 50 Ω), or dB (logarithmic).

Use the multipurpose knobs to set the reference level at the top of the screen.

Use the Reference Level Offset control to position the waveform with respect to the 0 dB reference level without changing the reference level at the top of the screen.

Behavior

The reference level offset is used to calculate the log magnitude in dB. Adjusting the reference level positions the waveform with respect to the top of the screen but does not change the position of the waveform with respect to the ground reference marker.

Changing the reference level offset moves the waveform with respect to its ground reference marker.

Phase Scale Tab
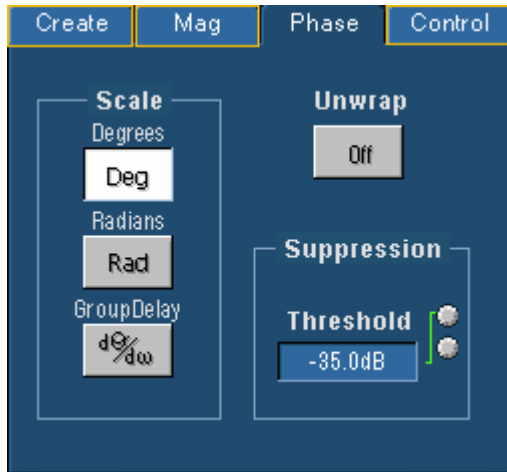
To Use

Use these controls to specify the vertical scale for spectral phase waveforms. Click one of the Scale buttons to specify the vertical units in radians, degrees, or as group delay.
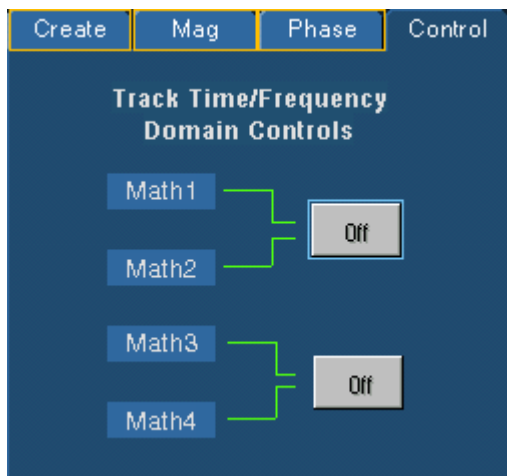
Behavior

Group delay unwraps and differentiates the phase. All phase values out of the spectral transformation are between π and $-\pi$ even though the actual phase value may be outside this range.

Click Unwrap to work on a continuous phase function to unwrap the phase value and show the true range. The phase unwrap begins with a phase value obtained at the first spectral sample after DC.

The Suppression Threshold prevents random noise from making the display unusable. Any spectral points with magnitudes less than the threshold will have phase values set to zero (degrees or radians). To set a dB level for the suppression threshold, select dB scale from the magnitude tab, and then set a horizontal bar cursor to the level on the magnitude spectrum. Read the dB value from the cursor and use the multipurpose knob to enter the value as the suppression threshold.

Control Tracking Tab

To Use

Use these controls to lock two math waveforms together. This is useful when you want to generate both a magnitude waveform and a phase waveform from the same source.

Behavior

When you lock the spectral controls together, any changes that you make to one math waveform are also made to the other math waveform.
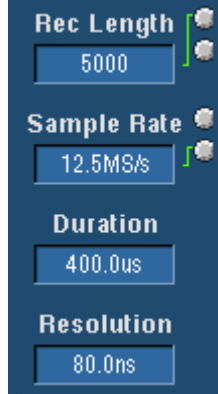
You can only lock Math1 with Math2 and Math3 with Math4.

Locking is useful, for example, when Math1 is the spectral magnitude and Math2 is the spectral phase of the same signal. Locking forces the magnitude and phase waveforms to track if you change the spectral controls.

Note

Math3 and Math4 are only available on 4-channel models.

Spectral Setups: Acquisition

To Use

Click the control that you want to change and use the multipurpose knobs to adjust the values.

Behavior

Rec Length (Record Length) is the total number of samples in the waveform acquisition. The maximum record length depends on the amount of memory. The record length determines the duration, which is expressed in seconds. Samp Rate (Sample Rate) is the number of acquisition samples per second. The faster the sample rate, the more accurately the instrument can represent fine details in a fast signal.

The sample rate determines the acquisition duration, which is expressed in seconds. The usable bandwidth of the signal equals one half the sample rate and is expressed in hertz.

Duration is the length of the acquired waveform in seconds. Adjusting the duration usually changes the record length without changing the current sample rate.

Resolution is the time between sample points. Changing the resolution changes the record length and the sample rate so that the selected duration setting remains constant.

Note

The acquisition controls interact with the gating and frequency domain controls; changing one set of controls may change some of the other control settings.

Spectral Setups: Gating

To Use

Select the FFT window type from the Window Type list. Use the multipurpose knobs to set the gating parameters.

Behavior

An FFT Window Type can help reduce spectral noise for many types of signals. Some windows provide better frequency resolution at the expense of better amplitude measurements. Other windows provide better amplitude accuracy over frequency resolution. The choice of window also affects the resolution bandwidth.

Gate Pos (Gate Position) moves the gate that selects the data for the spectral waveform. The gate position is the center point in the gate measured in seconds with respect to the trigger point.

In most cases the phase reference point is located at the center of the gated data. However, if you use a Tek Exponential window, the phase reference point is at the 20% point of the gated data.

Gate Dur (Gate Duration) sets the gate time duration in seconds. The default gate duration is the duration of the entire waveform record. The resolution bandwidth is inversely proportional to the gate duration.

Gate Length displays the gate length in points. The default gate duration is the duration of the entire waveform record.

Note

The gating controls interact with the acquisition and frequency domain controls; changing one set of controls may change the other control settings.

Spectral Setups: Frequency

To Use

Click the individual controls, and use the multipurpose knobs to select the Center Freq, Freq Span, and Res BW. Click Full to set the frequency span to full.

Behavior

The maximum frequency span is equal to half of the sample rate. If you need a higher frequency span, set the sample rate to a larger value.

Note

The frequency domain controls interact with the gating and acquisition controls; changing one set of controls may change the other control settings.

Spectral Controls: Acquisition

Access this control window from the Math menu Spectral Controls command or the Controls button on the Spectral Setup control window.

Overview

Use the time domain acquisition controls to view spectral waveforms.
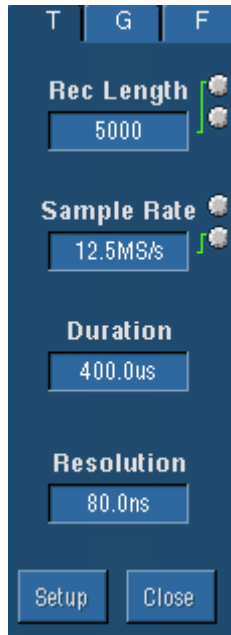
To Use

Click the control that you want to change, and use the multipurpose knob to adjust the value.

Behavior

Rec Length (Record Length) is the total number of samples in the waveform acquisition. The maximum record length depends on the amount of memory. The record length determines the duration, which is expressed in seconds. Long record lengths can slow instrument response, but also lower the noise relative to the signal and increase the frequency resolution for the spectral math waveform.

Sample Rate is the number of acquisition samples per second. The faster the sample rate, the more accurately the instrument can represent fine details in a fast signal. The sample rate determines the acquisition duration, which is expressed in seconds.

The usable bandwidth of the signal equals one half the sample rate and is expressed in hertz. Duration is the length of the acquired waveform in seconds. Adjusting the duration usually changes the record length without changing the current sample rate, similar to spectral analyzer frequency domain span and center frequency controls. Resolution is the time between sample points. Changing the resolution changes the record length and the sample rate so that the selected duration setting remains constant.

Note

The Acquisition controls interact with the Gating and Frequency domain controls; changing one set of controls may change some other control settings.

Spectral Controls: Gating

Access this control window from the Math menu Spectral Controls command or the Controls button on the Spectral Setup control window.

Overview

Use the gating controls to specify a segment of the time domain record to analyze in the frequency domain.

To Use

Select the FFT window type from the drop-down list. Use the multipurpose knobs to set the gating parameters.

Behavior

Gate Pos (Gate Position) moves the gate that selects the data for the spectral waveform. The gate position is the center point in the gate measured in seconds with respect to the trigger point. In most cases the phase reference point is located at the center of the gated data. However, if you use a Tek Exponential window, the phase reference point is at the 20% point of the gated data.

Gate Dur (Gate Duration) sets the gate time duration in seconds. The default gate duration is the duration of the entire waveform record. The resolution bandwidth is inversely proportional to the gate duration.

An FFT Window Type can help reduce spectral noise for many types of signals. Some windows provide better frequency resolution at the expense of better amplitude measurements.

Other windows provide better amplitude accuracy over frequency resolution. The choice of window also affects the resolution bandwidth. Res BW (Resolution Bandwidth) determines how small of a frequency difference can be resolved in the frequency domain output data. It defines the bandwidth of filters used to perform the frequency domain analysis. The resolution bandwidth is measured in hertz at the 3 dB points on each side of the peak spectral lobe.

Adjusting the resolution bandwidth controls the gate duration to obtain the desired resolution bandwidth.

Spectral Controls: Frequency

Access this control window from the Math menu Spectral Controls command or the Controls button on the Spectral Setup control window.

Overview

Use the Frequency controls to operate the instrument like a spectral analyzer.

To Use

Click the individual controls and use the multipurpose knobs to select Center Freq, Freq Span, Res BW, and Ref Level (Reference Level).

Behavior

The maximum frequency span is equal to half of the sample rate. If you need a higher frequency span, set the sample rate to a larger value.

The Ref Level (reference level) control is a position control for the spectral waveform. The position appears at the top of the control window. The Ref Level control does not change the position with respect to the reference marker.

Note

The Frequency domain controls interact with the Gating and Acquisition controls; changing one set of controls may change the other control settings.

Selecting a Spectral Window

A spectral window determines what the filter shape of the spectral analyzer will be in the frequency domain. It may be described by a mathematical function that is multiplied point-by-point times the input data to the spectral analyzer.

The following spectral windows are available with the instrument:

- Rectangular
- Hamming
- Hanning
- Kaiser-Bessel
- Blackman Harris
- Gaussian
- Tek Exponential
- Flattop2

The windows are listed in the order of their ability to resolve frequencies (resolution bandwidth). You can easily observe the shape of a window in the frequency domain by feeding a sine wave into the instrument and setting the center frequency of the spectral analyzer to the same frequency.

Then reduce the resolution bandwidth to spread out the lobe horizontally. You can then select different window functions and observe their shape.

Window Attributes

- In the time domain, windows are typically bell shaped and go to zero at the ends of the record. For cases where you may be doing impulse response testing, the impulse should be centered at the zero phase reference point (for most windows, this is the 50% position of the gate and 20% for the Tek Exponential window).

- Different window functions affect the resolution bandwidth of the spectral analyzer.

- Various window shapes affect the scallop loss.

- The shape of the frequency domain lobe is determined by the window function. Some windows have better resolution bandwidth, but they do so at the expense of side lobe attenuation and energy leakage into adjacent bins. For example, a rectangular window typically spills energy into many bins showing signals that don't exist; but it has the best frequency resolution. If you are using a window where leakage is occurring, you may want to use spectral gating to reduce the amplitude errors when measuring a given frequency.

Text Edit Controls

Use the following controls to edit the Math expression.

Hcme	Moves the text insertion point to the left of the first character in the expression.
←	Moves the text insertion point one character to the left.
→	Moves the text insertion point one character to the right.
Bksp	Deletes the character to the left of the insertion point.
Clear	Clears the entire math expression.

MyScope Menu

Use the MyScope Menu to create a new MyScope control window, open or edit an existing one, display the active MyScope window, or recall recently used ones.

MyScope Setup Control Window

Access this control window from the MyScope menu New Control Window… command.

Overview

MyScope allows you to create custom control windows that include only the controls that you use regularly. Instead of switching between several control windows, put the controls you use into a custom control window.
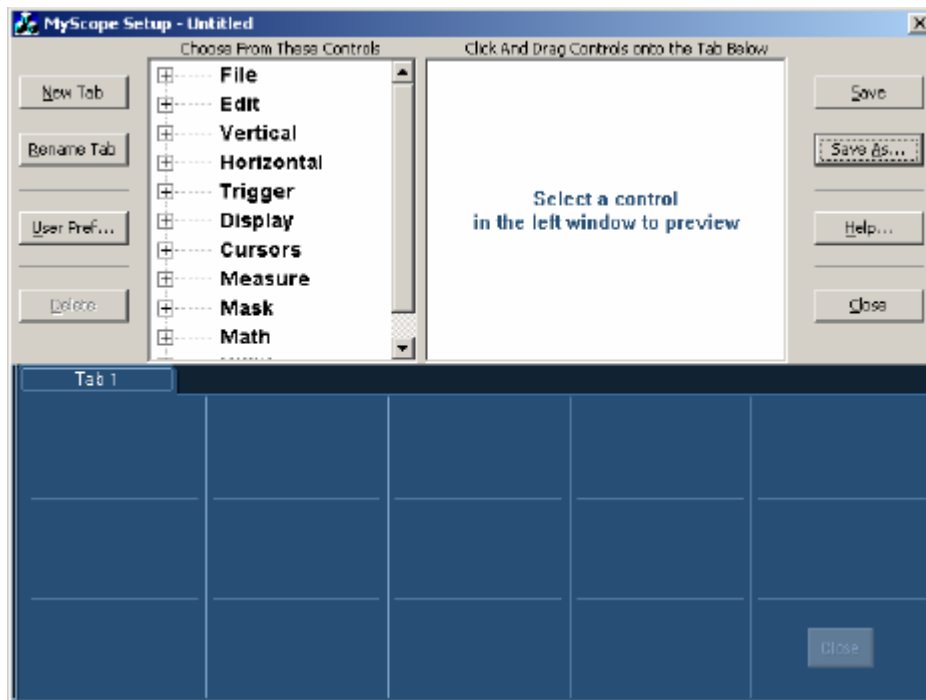
To Use

Click + to expand a control category. Click a control to preview it in the pane at the right. Double-click the control or click the + to expand the control list. (If there is no +, then the control cannot be customized further.) Clear the check boxes in the expanded control list to remove any components you do not want included in the control. Click and drag the control to your MyScope control window. You can also change the placement of the control in your MyScope control window by clicking and dragging.

Click New Tab to add a tab to your MyScope control window. You can have up to eight tabs in your MyScope control window. To rename a tab, click Rename Tab or double-click the tab.

Click User Pref... to specify the user preferences that you want to load with your MyScope control window.

Click Save or Save As… to save your MyScope control window. Click Close to close the control window without saving it.

Behavior

Clicking on a control in the list allows you to preview the control in the pane at the right. Selecting or clearing the checkboxes for the controls in the list adds or removes components. The control will snap to the nearest grid location when you release the mouse.

Tip To change a control you have already placed into a MyScope control window, drag it back to the preview pane and the appropriate control category will open for quick modifications.

Using MyScope Control Windows

Overview

Some MyScope controls operate differently from the same control in the UI.

Horizontal controls

The Delay Mode control contains only an On/Off button, Horiz Delay time, and Ref Point. It does not show the explanatory graphic that is on the Horizontal Setup control window.

The Table button on the Time Stamps control activates a pop-up time stamp table.

Horizontal readouts that are not available in the MyScope window are Resolution, Sample Rate, and Duration.

Trigger controls

Trigger Type and Trigger Sequence controls have a setup button that is automatically included with the controls. The tripper type setup button activates a pop-up control widow that represents the right half of the Trigger Setup control window under the A Event tab. Use this setup control window to set the parameters for the selected trigger type. The Trigger Sequence setup button activates a pop-up control window that represents the right half of the Trigger Setup control window under the A->B Event tab. Use this setup window to set the parameters for the trigger sequence.

If you select more than 11 trigger types, the trigger type control will activate a More button to accommodate the multiple pages necessary to display more than 11 trigger types.

Cursor controls

Cursor multipurpose knob and style controls are available but what displays in your MyScope control window will depend on what type of cursor you select. For example if you select Screen cursors, then Cursor MPKs controls and Cursor Style controls display as shown in the following graphic.
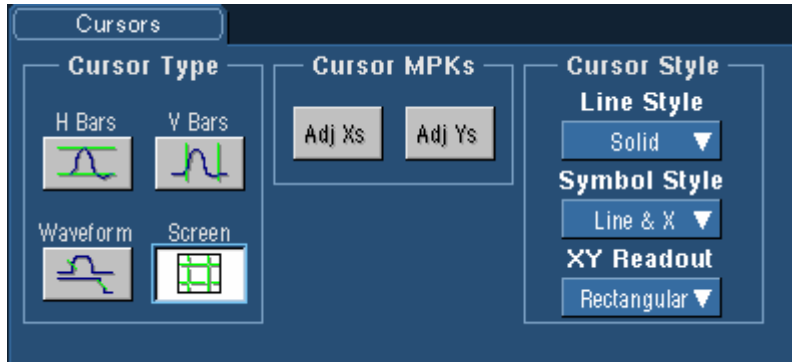
However if you select V Bars, then Cursor MPKs controls are not available and the Cursor Style controls change as shown below.
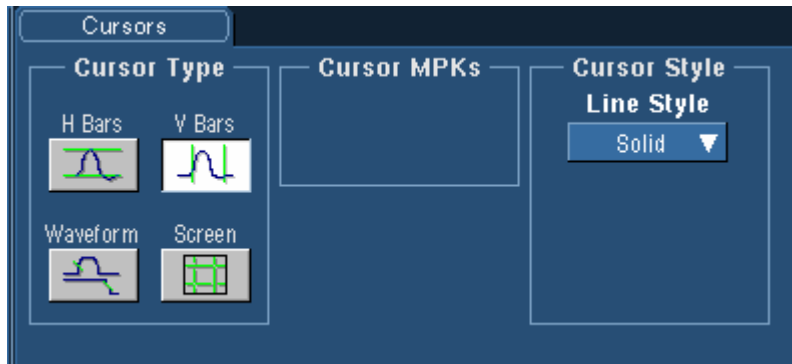
Measure controls

Measurement controls always include a Next Ch and Clear All button. The Clear All button clears all measurements on all channels. The Next Ch button advances the Source to the next active channel and assigns the same measurements to the next channel as were assigned to the previous channel. This is a quick way to assign the same measurements to multiple channels.

Each page of the measurement controls can hold up to 12 measurements. If you select more than twelve, a More button appears to accommodate the multiple pages necessary to display more than 12 measurements.

Mask controls

The Mask and Limit Test controls act as shortcuts to the Mask or Limit Test setup control windows.

Utilities controls

The Set Time and Date, Instrument Diagnostics, and User Preferences Utilities controls act as shortcuts to the setup control window for these controls. Clicking the Setup button in any one of these controls will activate the setup control window for that command.

Open or Edit MyScope Dialog Box

To Use

Select the MyScope control window that you want from the dialog box, and then click Open. Or you can type a file name to select the MyScope control window. Click the Look in drop-down list to navigate to other directories.

Behavior

This dialog box opens in the default directory, C:\TekScope\MyScope. If you have saved MyScope control windows in another directory, click the Look in drop-down list to go to that directory. Clicking Open will open the selected MyScope control window. The only available file type is .tcw.

Save MyScope File As Dialog Box

To Use

Type a file name in the File name text box, and click Save. To save to a different directory, click the Look in drop-down list to navigate to other directories, and then click Save.

Behavior

This dialog box opens in the default directory, C:\TekScope\MyScope. If you want to save MyScope control windows in another directory, click the Look in drop-down list to go to that directory. The only available file type is .tcw.

Utilities Menu

The Utilities menu provides access to utilities used by the instrument such as instrument calibration, instrument diagnostics, and user preferences. All utilities are accessible from the menu bar only. None of the utilities are directly accessible from the front panel controls or from the toolbar.

Set Time and Date

Access this control window from the Utilities menu Set Time & Date command.

Overview

Use the Set Time and Date control window to set the time and date on the instrument.

To Use

Click the controls in the window to map them to the multipurpose knobs and use the knobs to set the time and date. Click Set time and date now to save your changes. You can also click Get current time to set the time using the system clock. This is useful if the instrument clock becomes out of sync.

Behavior

The controls in this window operate the same as the standard Windows Date/Time Properties dialog box. After you click Set time and date now, the system date and time are set to the specified values. These controls are provided as a convenience so that you can change the date and time without having to go to the Windows Date/Time Properties dialog box.

Tip The instrument clock is suspended when this window is open. Click Get current time to resynchronize the clock with the PC clock.

GPIB Configuration Control Window

Access this control window from the Utilities menu GPIB Configuration command.

Overview

Use the GPIB Configuration control window to configure the instrument for GPIB (General Purpose Interface Bus) applications. GPIB applications enable you to remotely control your instrument (take measurements, set controls, perform calculations, and so on).

To Use

Click Talk/Listen to enable the GPIB. Click the Address control to map it to a multipurpose knob and use the knob to set the GPIB address for the instrument. Click Controller to use your instrument to control other instruments. Click Off Bus to make the GPIB port unavailable.

Note

The Address controls are only available when you are in Talk/Listen mode.

Behavior

The instrument is defined as a GPIB talker/listener when you click Talk/Listen. This is the device-based operation.

The instrument is defined as a controller when you Click Controller. This is the controller-based operation. You cannot use GPIB commands to control your instrument when it is in Controller mode. When in Controller mode, clicking Off Bus has no effect. The Off Bus control only makes the GPIB port unavailable if you are in Talk/Listen mode.

LAN Server Status Control Window

Access this control window from the Utilities menu, LAN Server Status command.
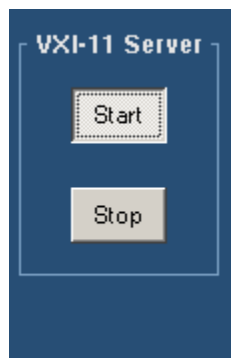
Overview

This control window allows you to enable the VXI-11 LAN server to accept GPIB commands. For more information see *Getting Started with OpenChoice Solutions* manual that was shipped with your instrument.

To Use

Click the Start button to enable the VXI LAN server to accept GPIB commands.

Click the Stop button to disable the ability of the VXI-11 LAN server to accept GPIB commands.

External Signals Control Window

Access this control window from the Utilities menu External Signals command.

Overview

Use the External Signals control window to select the trigger signal to send to the AUX OUT BNC connector or to toggle the instrument clock reference source.
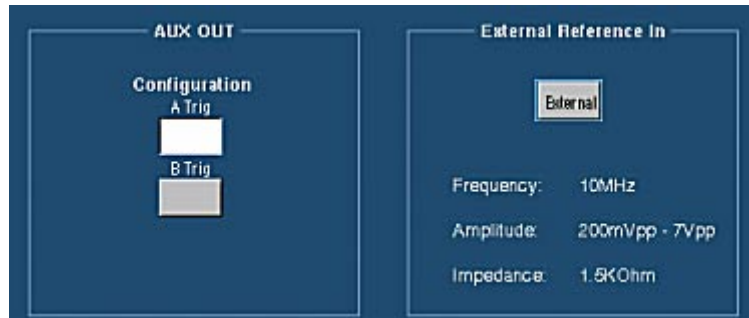
To Use

To send a signal to the AUX OUT BNC connector when the instrument triggers, click either A Trig or B Trig. To change the source of the clock reference, click the External Reference In button.

Behavior

For AUX OUT, the selected trigger signal is routed to the AUX OUT BNC connector so that you can trigger another instrument such as a logic analyzer. The output voltage is TTL-compatible, negative polarity.

For External References, the clock reference is routed to the EXT REF BNC connector when you select External. You must perform a signal path compensation whenever you change the reference.

Touch Screen (Option)

Access the Touch Screen calibration feature from the Utilities menu.

Overview

Use the Touch Screen submenu to calibrate the touch screen.

To Use

Select Calibrate, and then follow the on-screen instructions to calibrate the touch screen.

Behavior

The Calibrate command starts the Touch Screen calibration program.

Instrument Calibration Control Window

Access this control window from the Utilities menu Instrument Calibration command.

Overview

Use the Instrument Calibration control window to calibrate the Signal Path Compensation (SPC).
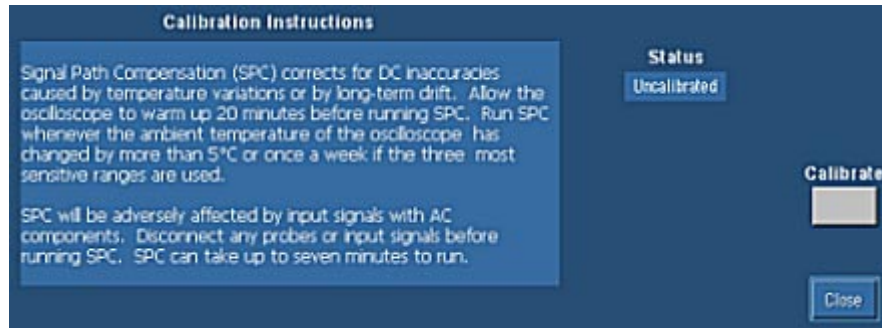
To Use

Follow the Calibration Instructions and click Calibrate to begin the procedure.

Behavior

The only user-accessible calibration is the signal path compensation (SPC). All other calibration is accessible to qualified service personnel only. For all calibrations, the Status control should indicate Pass. If it does not, contact your local Tektronix service personnel.

Instrument Diagnostics Control Window

Access this control window from the Utilities menu Instrument Diagnostics command.

Overview

Use this control window to run the instrument diagnostics.
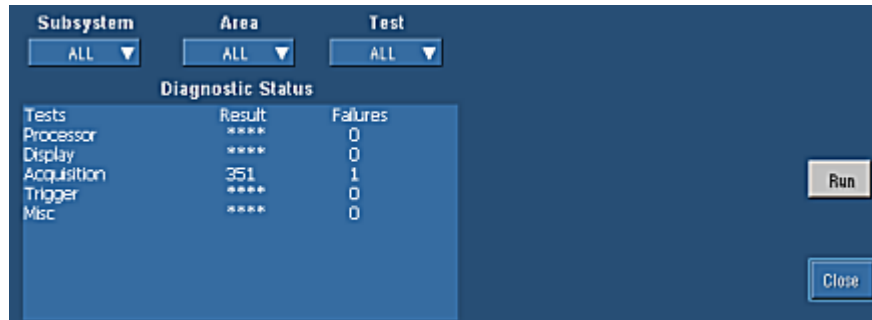
To Use

Select the diagnostic that you want to run. Use the controls to loop on a specific test for a set number of times or to stop as soon as a failure occurs.

You can run all tests or specific tests using the options on the Subsystem, Area, and Text drop-down lists. The Diagnostic Status table lists each diagnostic, the results of the test, and the number of failures that occurred.

Behavior

This window is mainly intended for service personnel. The diagnostic information can be used as part of an incoming inspection or to diagnose potential problems with the instrument. Remove the input signals before running any of the diagnostics. If diagnostic failures occur, record the failure information and contact your local Tektronix service personnel for more information.

E-mail on Event Control Window

Access this control window from the Utilities menu and select E-mail on Event Setup

Overview

Use the E-mail on Event Setup control window to set up the conditions under which the instrument will send an email, the types of attachments, and message limits.
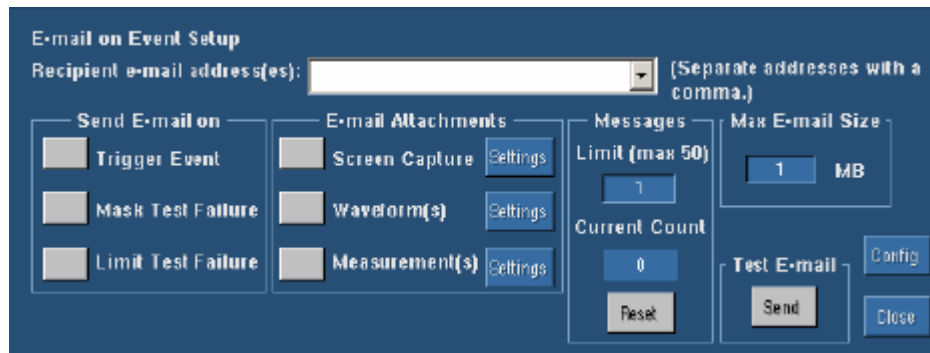
To Use

Type the email address(es) to which you want to send a message in the Recipient e-mail address(es) box. Use a comma to separate multiple addresses.

Select the events on which you want to send e-mail using the Send E-mail on check boxes. Use the E-mail Attachment check boxes to select the type of attachment to include with the e-mail. Click the Settings button to open the Save Options dialog box, and set the parameters for the attachment.

Enter the number of e-mail messages to be sent by the instrument, up to a maximum of fifty, in the Limit text box. Enter the maximum size, up to 2 GBs, for each e-mail message in the Max E-mail Size text box. Click the Send button to send a test e-mail. Click the Config button to open the E-mail Configuration dialog box.

Behavior

The maximum number of characters in the Recipient e-mail address box is 252, so you can enter any number of e-mail addresses up to a total of 252 characters. You can select one or all of the events to send e-mail on. If you do not have a SMTP server address or e-mail address set up, you cannot select an event to send e-mail on.

The screen capture, waveform, and measurement types for the e-mail attachments are determined by the type that you select in the Save Options dialog box when you click Settings.

When the maximum number of messages is reached, the Send E-mail on check boxes are cleared and the Current Count is reset to 0. You must select the event again to continue sending e-mail. Clicking Reset changes the current count back to 0.

Attachments that exceed the Max E-mail Size are saved to the hard drive on the instrument.

Tip To save your attachments on the instrument hard drive and still send an email, set the Max E-mail Size limit to 0.

E-mail Configuration Dialog Box

Access this dialog box from the E-mail on Event Setup control window and click the Config button.

Overview

Use this dialog box to enter the address of the mail server to use when sending e-mail on events. You must have a server entered in the SMTP Server Address box for your instrument to send e-mail.

You can customize your e-mail by selecting the number of attempts to make, entering a From address, setting the timeout length, entering a login and password if necessary, and entering a host name.

To Use

Enter the SMTP server address. Click OK if you want to accept the default values. Click More>> if you want to customize your e-mail. Enter the appropriate information in each box that you need to change from the default.

Behavior

The number you enter in the Number Attempts to Send box is the number of times the system will try to send the e-mail if it isn't successful on the first try. The default is 1.

Entering an address in the From box puts that address in the From field of the e-mail. The default value is the instrument model number. Entering your own address allows the instrument to send you e-mail if there is a problem with sending the e-mail to the recipient.

Entering a number in the Timeout box will override the default timeout for sending e-mail to the server.

If your mail server requires a login or password, enter the login in the Auth Login box and the password in the Auth Password box.

Entering a network ID in the Hostname box makes the e-mail appear as though it did not come from the instrument.

User Preferences: Prompt Before Action

Access this control window from the Utilities menu User Preferences command.

Overview

Use the Prompt Before Action tab to indicate whether or not you want the instrument to display prompts (confirmation messages) before carrying out certain tasks like deleting files or waveforms.

To Use

Click a button to enable or disable the control setting.

Behavior

The confirmation messages give users the chance to change their minds before taking action. The messages generally explain the implications of the action.

All preference settings are saved when you turn the instrument power off.

User Preferences: Keypad Defaults

Access this control window from the Utilities menu User Preferences command. These controls are on the Keypad Defaults tab.

Overview

Use the Keypad Defaults tab to define the preset trigger levels.

To Use

Change the label for the selected preset control by entering a four-character name.

Click the trigger level controls to map them to the multipurpose knobs and use the knobs to set the trigger level values.

Behavior

Use this control window to change the trigger level preset values. This can be useful when you want to trigger the instrument on nonstandard trigger levels.

By default:

- TTL fixes the trigger level at +1.4 V
- ECL fixes the trigger level at −1.3 V

When you set the volts/div less than 200 mV, the instrument reduces the TTL or ECL trigger levels below their standards. This is because the trigger level range is fixed at ± 12 divisions from the center. For example, at 100 mV, the trigger range is ± 1.2 V, smaller than the standard TTL and ECL levels.

All preference settings are saved when you turn the instrument power off.

User Preferences: Record Length

Access this control window from the Utilities menu User Preferences command.

Overview

Use the Record Length tab to define how the instrument responds to changes in record length.

To Use

Click one of the buttons to define which variable will be held constant when the record length changes.

Behavior

Record length, sample rate, and horizontal scale are dependent variables. When you change the record length, you can choose to hold either the sample rate or the horizontal scale constant. The remaining variable will change in response to the record length changes.

All preference settings are saved when you turn the instrument power off.

User Preferences: Readouts

Access this control window from the Utilities menu User Preferences command.

Overview

Use the Readouts control window to turn Highlight readouts on and off.

To Use

Click Changes On to highlight readouts when you make changes. Enter the time to leave the readout highlighted in the Hide After text box.

Behavior

The maximum length of time that you can leave readouts highlighted is 5 seconds. The readouts that are highlighted are Vertical Scale, Horizontal Delay, Trigger Level, Trigger Source, Trigger Slope, and Source Termination.

Option Installation

Access this command from the Utilities menu.

Overview

Use Option Installation to enable upgrades that you have purchased for your instrument from Tektronix. For the most current list of upgrades, go to www.tek.com or contact your local Tektronix representative.

To Use

Click Continue in the first dialog box. Enter the Option Installation Key provided by Tektronix in the Current Option Information dialog box. Click Continue again and follow the instructions to install the option. Further instructions and information may be available with the option or application documentation.

Behavior

Information about installed options and applications is listed in the installation dialog box. The installation is acknowledged when complete.

Tip To run external applications after installation, select the File menu **Run Application** submenu, and select the application name from the drop-down list.

Tek Secure Erase

Access this command from the Utilities menu.
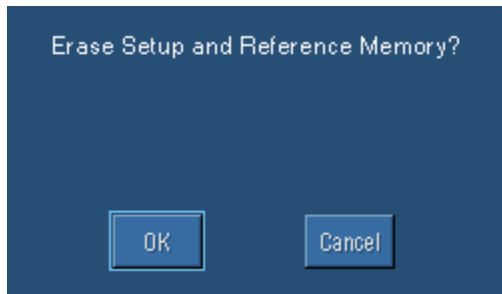
Overview

Use Tek Secure Erase to erase all waveforms and instrument setups.

To Use

Click OK to erase, or Cancel to exit the dialog box.

Behavior

Tek Secure Erase removes all reference waveform and instrument setup memory locations. Erased setups are replaced with the default factory setups. This is useful if you are using the instrument to gather sensitive or secure data.

Theory of Operation

Functional Model

The instrument has four high-level subsystems or processes (consisting of both hardware and software functions) and the data that connects them:

- Digital Signal Acquisition System. Acquires a waveform record from each signal you probe using the following subsystems:

- Input Channels. Conditions the input signal, primarily through the use of analog hardware, before converting the signal to digital form.

- Trigger System. Recognizes a specific event of interest on the input signal and informs the Time base system of the occurrence of the trigger event.

- Time base System. Triggers the Acquisition system to start an acquisition cycle. In more general terms, synchronizes the capturing of digital samples in the Acquisition system to the trigger events generated from the Trigger system.

- Acquisition System. Converts and stores the analog signal to a digital format.

- Signal Processing and Transformation. Performs transformations or operations on user-specified waveform data (channel, reference, and/or math waveforms). Automatic measurements, spectral waveforms, and histogram generation are examples of signal processing and transformation.

- Display and UI. Provides output (and sometimes input) of instrument-data elements in a form suitable to the user and also provides user input control.

Process Overview
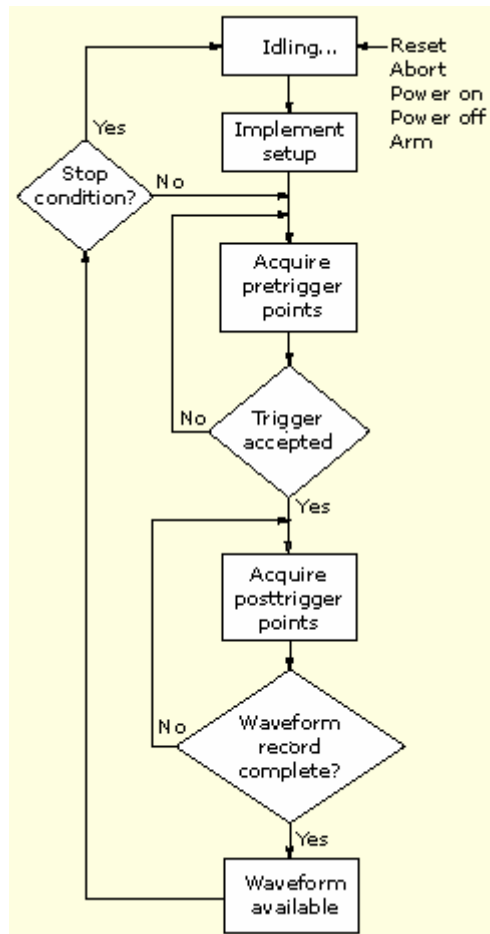
1 The instrument starts in the idle state; it enters this state upon power up, upon receiving most control setting changes, or upon finishing acquisition tasks.

2 Control settings are implemented as they are requested. When you toggle the RUN/STOP control to RUN, the instrument starts the hardware.

3 The instrument acquires samples until the pretrigger portion of the waveform record (channel) being acquired is filled.

4 The instrument then begins looking for a trigger. Acquiring continues to take place, keeping the pretrigger points current, until triggering criteria are met or a trigger is forced (Auto trigger mode only). The instrument accepts the trigger.

5 The instrument acquires samples until the posttrigger portion of the waveform record (channel) being acquired is filled.

6 If averaging or enveloping is on, the record becomes part of the multi-acquisition record that these modes produce. The process loops back to step 3 to acquire additional records until the number of acquisitions required for the acquisition mode currently set are processed, and then processing continues to step 7.

7. At this point the acquisition record is in memory and is available to the instrument for measurement of its parameters, display, and so on.

8. The instrument then checks for a user-specified stop condition and either returns to its idle state or continues at step 3, according to what it finds.

Keep in mind that this instrument samples in real time or random equivalent time; both sampling systems provide pretrigger information by using the trigger to stop an already running acquisition. Both sampling systems also sample the input after it is scaled, providing improved input protection and dynamic range.

Signal Connection

This diagram displays the signal connection model for each input channel.
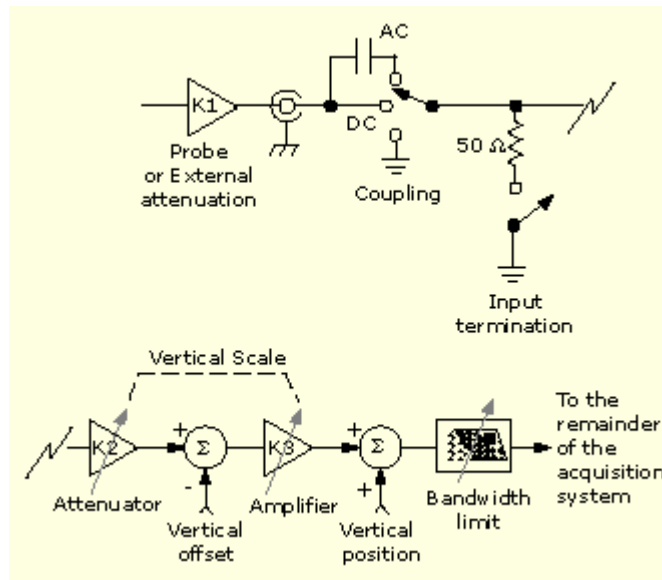
Input Conditioning

Use input conditioning to ensure that the instrument acquires the best possible data for processing. To do this:

- Set the vertical scale to control the size of the vertical acquisition window for each channel to capture all or part of the input signal's vertical amplitude. When vertical scaling is set to capture a fraction of the input signal (to increase detail), the vertical offset control determines which part of the input signal is captured by the vertical acquisition window. (See note)

- Set the horizontal scale to control the size of the horizontal acquisition window to capture as much as you want of the input signal(s). Set the horizontal position to change the trigger position or to delay the window relative to a trigger event and to control where in the input signal (data stream) that the horizontal acquisition window acquires. (See note.)

For more background on acquisition windows, refer to Vertical Acquisition Window Considerations on page 227 or Horizontal Acquisition Window Considerations on page 229.

The instrument can automatically obtain and display a stable waveform of usable size. Push the AUTOSET button to automatically set the instrument controls based on the characteristics of the input signal. Autoset is much faster and easier than a manual control-by-control setup.

The instrument can also be reset to its factory default settings by pushing the front-panel DEFAULT SETUP button.

Use of some input conditioning controls or features may be limited when other control settings are in effect. Voltage offset is incompatible with reference waveforms because offset is an acquisition control for live waveforms (channels), not stored waveforms (reference).

Note

The terms vertical acquisition window and horizontal acquisition window refer to the vertical and horizontal range of the segment of the input signal that the acquisition system acquires. The terms do not refer to any display windows on screen.

Probes and Signal Connection

Select the probe or cable that brings the signal into the instrument. Choose the probe or cable that best fits your acquisition task, whether it is connecting an active probe to test a digital circuit, or connecting to a test fixture through BNC cables to characterize a device. The connection to the instrument depends on your application.

Up to four acquisition channels are available. You can display each channel as a waveform and/or use the channel data to create other waveforms, such as math and reference waveforms.

Tektronix provides a variety of probes and cables for this product. You can also check your Tektronix catalog for connection accessories to support your probing requirements. For more information about a particular probe, read the documentation that came with that probe.

Coupling

All instruments and probes specify a maximum signal level. Do not exceed the limit, even momentarily, as the input channel or probe may be damaged. Use external attenuators if necessary to prevent exceeding the limits.

Coupling determines whether an input signal is directly connected to the input channel (DC coupling), connected through a DC blocking capacitor (AC coupling), or not connected at all (GND coupling).

The input resistance of each input channel is 1 MΩ or 50 Ω. To properly terminate signals when using coaxial cables, or to support active probes designed for a 50 Ω environment, a 50 Ω termination resistance can be selected by pushing the front-panel termination button for a particular channel.

All probes expect a specific coupling and input termination. Both coupling and input termination are displayed on the screen. If the instrument determines the coupling and termination required by the probe, either implicitly because of the TEKPROBE interface or through performing a probe calibration, the instrument sets the required coupling and input termination.

Consider the following when you use 50 Ω termination with any channel:

- The instrument does not accurately display frequencies under 200 kHz if AC coupling is selected.
- The instrument reduces the maximum volts per division setting for the channel to 1 V from 10 V (10 V from 100 V with a 10X probe attached), since input amplitudes appropriate for the higher settings would overload the 50 Ω input.
- The instrument switches to 50 Ω termination and DC coupling if you connect an active probe. Active probes also reduce the maximum volts per division.

Scaling and Positioning

The scaling and positioning controls determine the portion of the input signal received by the acquisition system. Set vertical scaling, positioning, and DC offsets to display the features of interest on your waveform and to avoid clipping.

Waveform data outside of the vertical acquisition window is clipped; that is, the data is limited to the minimum or maximum boundaries of the vertical acquisition window. This causes inaccuracies in amplitude-related measurements. For more information see Vertical Acquisition Window Considerations on page 227.
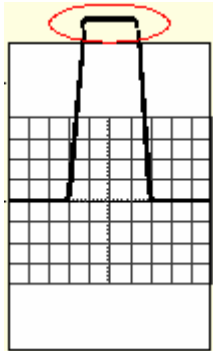
Acquired waveform
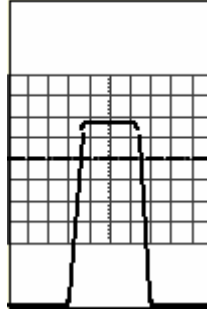
Adjusted Horizontal Scale

Set horizontal scale, position, and resolution (record length) so that the acquired waveform record includes the waveform attributes of interest with good sampling density on the waveform. These settings define the horizontal acquisition window, described in Horizontal Acquisition Window Considerations on page 229.

Note

The terms vertical acquisition window and horizontal acquisition window refer to the vertical and horizontal range of the segment of the input signal that the acquisition system acquires. The terms do not refer to any display windows on screen.

Vertical Acquisition Window Considerations

You can set the vertical scale, position, and offset of each channel independently of other channels. Vertical scale and offset specify the vertical acquisition window for each channel. Parts of the signal amplitude that fall within the vertical window are acquired; parts outside (if any) are not.

The offset control subtracts a constant DC level from the input signal before the vertical scale factor is applied, and the vertical position control adds a constant number of divisions of signal after the scale factor is applied to the resulting difference.

The vertical scale and position controls have the following affects on the vertical acquisition window and the displayed waveform:

- The vertical volts per division that you set determine the vertical size of the acquisition window, allowing you to scale it to contain all of a waveform amplitude or only part. Figures A and B below show two vertical acquisition windows that contain the entire waveform, but only the window in Figure B contains the entire waveform in the graticule on screen.

Note

Amplitude-related automatic measurements (for example, peak-to-peak and RMS) will be accurate for vertical windows like those shown in Figures A and B below because neither waveform is clipped (that is, both waveforms are acquired). But if signal amplitude were to extend outside the vertical acquisition window, the data acquired is clipped. Clipped data causes inaccurate results if used in amplitude-related automatic measurements. Clipping also causes inaccurate amplitude values in waveforms that are stored or exported for use in other programs.

If the scale of a math waveform is changed so that the math waveform is clipped, it will not affect amplitude measurements on that math waveform.

- The vertical position adjusts the display of the graticule relative to the vertical acquisition window (position is a display control). Figure B shows how vertical position moves the waveform graticule vertically in the acquisition window to place the acquired waveform in the graticule display. That is all position does; it does not determine what data is acquired as do vertical scale and offset.

Figure A
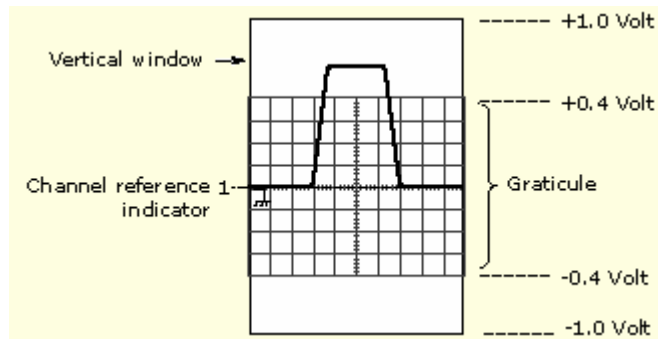
Figure B
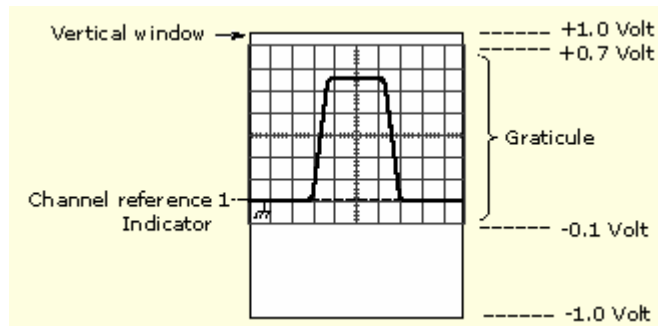
- As you vary vertical offset, the middle voltage level moves relative to zero. This moves the vertical acquisition window up and down on the waveform. With input signals that are smaller than the window, it appears the waveform moves in the window. Actually, a larger signal shows what really happens: the offset moves the middle of the vertical acquisition window up and down on the input signal. The following figure shows how offset moves the acquisition window to control the portion of the waveform amplitude the window captures.

- Applying a negative offset set moves the vertical range down relative to the DC level of the input signal. Likewise, applying a positive offset moves the vertical range up. See the following illustration.
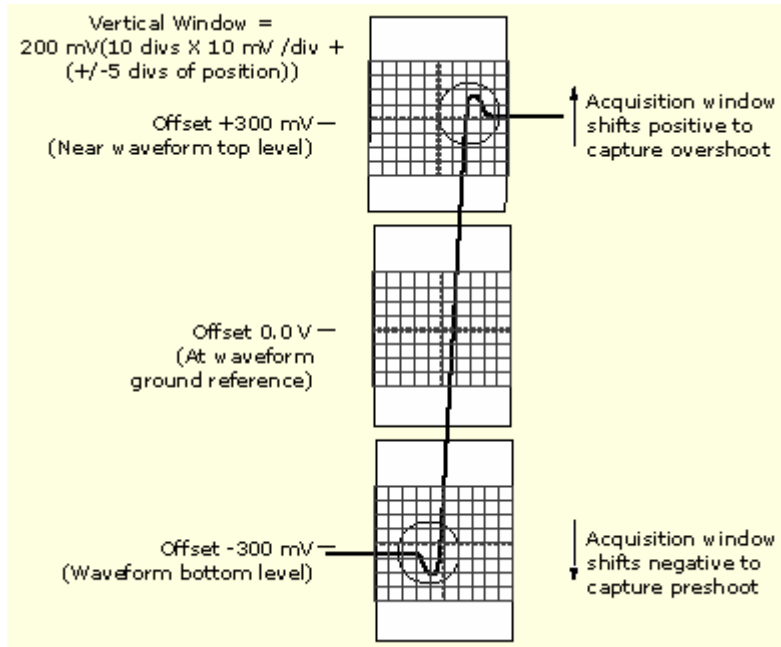
Horizontal Acquisition Window Considerations

The instrument lets you define the horizontal acquisition window, that is, set several parameters that determine the segment of an incoming signal that becomes the waveform record when acquired. (For background, go to Waveform Record on page 233.) These common parameters specify a horizontal acquisition window that is applied to all channels in parallel. See Independent vs. Shared Window on page 231 for more information. These parameters, shown in the figure below, are:

- Trigger Position: Determines where the trigger event will be located in the waveform record. To see more pretrigger data move your trigger position to the right on the graticule.

- Horizontal Position: Determines the number of pretrigger and posttrigger samples. Samples before the trigger point are pretrigger samples and those after the trigger point are posttrigger samples. When Delay is off, the horizontal position is the same as the trigger position.

- Horizontal Delay: Determines the time from the trigger point to the Horizontal Reference.

- Horizontal Scale: Determines the horizontal size of the window relative to any waveform, allowing you to scale it to contain a waveform edge, a cycle, or several cycles.
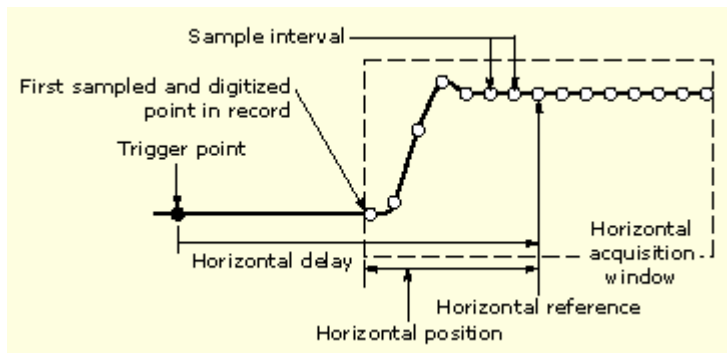
Horizontal Window with Delay On

Autoset Considerations

Autoset acquires samples from the input signal and attempts to take the following actions based on the input data:

- Evaluate the amplitude range of the input signals and set the size and vertical offset of the vertical acquisition window to acquire the signal with good resolution, but without clipping.

- Set the trigger to the approximate midlevel of the signal being Autoset and switches to edge trigger mode.

- Evaluate the signal transitions and set the horizontal scale to produce a waveform display of 2 or 3 cycles of the input signal.

Sometimes Autoset cannot produce a correct display due to the nature of the input signal. If so, you may have to adjust the scale, trigger, and acquisition controls manually. Some conditions that can cause Autoset to fail are:

- No signal present

- Signals with extreme or variable duty cycles

- Signals with multiple or unstable signal periods

- Signals with too low amplitude

- No recognizable trigger signal

- Signals with a frequency ±20 Hz

- Signals with a frequency above the bandwidth of the instrument

Horizontal Acquisition Window Interrelated Parameters

Horizontal Scale, Record Length, Sample Interval, and Resolution are related parameters that specify the horizontal acquisition window. Because the horizontal acquisition window must fit in the 10 horizontal division display, for most cases, you can set the Time Duration of the horizontal acquisition window (10 divs x the scale setting) as described below. By also setting a Record Length in samples, you indirectly set the Resolution, Sample Interval, and Sample Rate for the horizontal acquisition window (waveform record). The relationship between these horizontal elements is:

1 Time Duration (seconds) = 10 divs (window size) x Horizontal Scale (sec/div)

2 Time Duration (seconds) = Sample Interval (seconds/sample) x Record Length (samples), where Time Duration is the horizontal acquisition window time duration, and
Sample Interval (sec/sample) = Resolution (sec/sample) = 1/Sample Rate (samples/sec)

Note that it is the Sample Interval that changes to accommodate the window Time Duration (and its scale setting) and the Record Length setting. These elements behave as follows:

- If the Record Length or Time Duration change, the Sample Interval will vary to accommodate the change. The Sample Interval can vary up to the highest Sample Rate, (this corresponds to the lowest Sample Interval, and the highest Resolution).

- If the Sample Interval reaches its lower limit, the Record Length must decrease if the Time Duration decreases (you set faster scale settings), or Time Duration must increase (forcing slower scale settings) if Record Length increases (you set longer record lengths). The equation becomes:

Maximum Record Length = Time Duration ÷ Min Sample Interval

For example, at 200 ps/div and 10 divisions, the record length must be 500 points:

Max Rec Length 500 samples = (10 divs x 200 ps/div) ÷ 4 ps/sample

Max Rec Length = 500 samples

Note

As implied from the operation just described, Resolution and the equivalent elements, Sample Interval and Sample Rate, are derived and cannot be set directly. You can, however, check the Resolution at anytime in the resolution readout. Also note that the front-panel RESOLUTION knob actually adjusts the Record Length to increase sample density.

The previous discussion also assumes that Horizontal Scale is held constant. You can hold the Sample Rate constant instead by clicking Hold Sample Rate constant on the Record Length tab of the User Preferences control window.

Independent versus Shared Window

The instrument applies the same horizontal acquisition window to all channels from which it acquires data. Unlike the vertical acquisition window that you size and offset independently for each channel, the same time/div, resolution (record length), and horizontal position (from the same trigger point) apply to all channels simultaneously. One trigger, from a single trigger source, will locate a common horizontal acquisition window for all active channels, which you can shift in parallel by setting the horizontal position control.
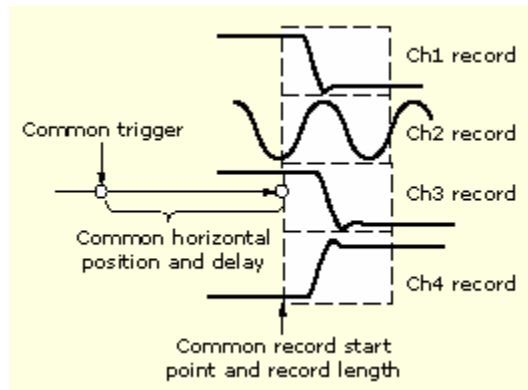
Acquisition Hardware

Before a signal can be acquired, it must pass through the input channel where it is scaled and digitized. Each channel has a dedicated input amplifier and digitizer, as shown in the following figure; each channel can produce a stream of digital data from which the instrument extracts waveform records. Refer to Signal Connection on page 225 for further description of scaling, positioning, and DC offsetting of channels.
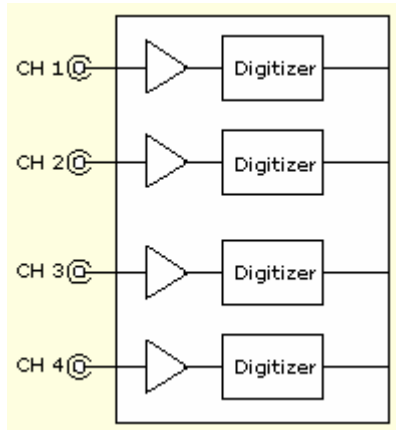
Sampling Process

Acquisition is the process of sampling an analog signal, converting it into digital data, and assembling it into a waveform record, which is then stored in acquisition memory. Sampling, then, is the process that provides a waveform record per trigger event as shown in the figure below.
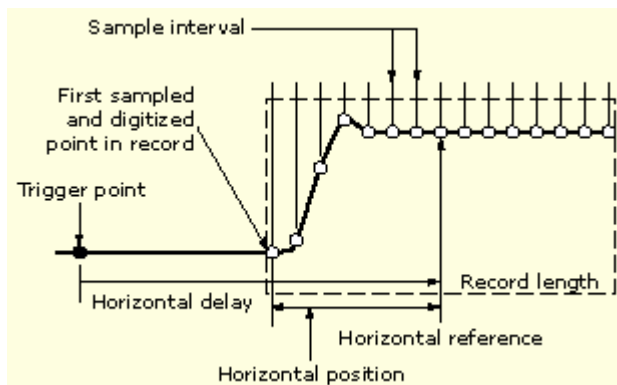
The signal parts within the vertical range of the amplifier are digitized as shown in the figure below.
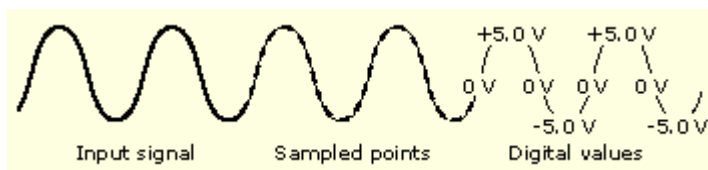
Sampling Modes

The acquisition system can process the data as it is acquired, averaging or enveloping the waveform data to produce enhanced waveform records. Once the waveform record exists (enhanced or not), you can use the post processing capabilities of the instrument to further process that record: perform measurements, waveform math, and so on. Refer to Acquisition Mode on page 103 for a description of the acquisition modes.

Waveform Record

While sampling on the input signal provides the data that makes up the waveform record for any given channel, the instrument builds the waveform record through use of some common parameters ("common" means they affect the waveforms in all channels).

The following figure shows how these common parameters define the waveform record. They define how much data is taken and from where in the data stream. The parameters are:

- Sample Interval: The time between sample points taken during acquisition.

- Record Length: The number of samples required to fill a waveform record.

- Trigger Point: The trigger point marks the reference zero time in a waveform record. All waveform samples are located in time (minus and/or plus) with respect to the trigger point.

- Horizontal Position: When horizontal delay is off, the horizontal position is the time from the first sample taken (first point in the waveform record) to the trigger point (in percent of samples before the trigger). The trigger point and the horizontal reference are at the same time in the waveform record.

When horizontal delay is on, the HORIZONTAL POSITION knob controls the horizontal delay time. The horizontal delay is the time from the trigger point to the horizontal reference.
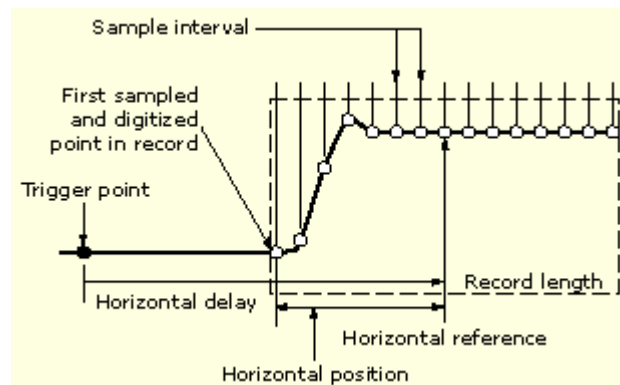
The waveform record and its defining parameters (horizontal delay on)

The figure above shows that the instrument acquires points in order from left to right.

When all of the points in the waveform record have been sampled and digitized, the waveform record is stored in acquisition memory and becomes available for display (or use in math waveforms, storage, and so on).

Interleaving

The instrument can interleave channels to attain higher real-time digitizing rates and longer record length when only 1 or 2 channels are turned on without equivalent-time sampling. The instrument applies the resources of unused channels (that is, channels that are turned off) to sample those that are in use (turned on). The following table lists how interleaving extends the maximum digitizing rate and record length.

Interleaving affects on digitizing rate and record length

Number of channels in use	Maximum digitizing rate	Maximum record length
One	5 GS/sec	8 M (16 M with Option 3M)
Two	2.5 GS/sec	4 M (8 M with Option 3M)
Three or Four	1.25 GS/sec	2 M (4 M with Option 3M)

Once you set the horizontal scale to exceed the maximum digitizing rate for the number of channels in use, the instrument will not be able to get enough samples to create a waveform record. At that point, the instrument will switch from real- to equivalent time to obtain additional samples.

Note

Only record length interleaving is available on the TDS5054BE. The maximum sample rate on the TDS5054BE is 1 GS/s regardless of the number of channels in use.

Triggering Concepts

Overview

To use the instrument to sample a signal and digitize it into a waveform record that you want to process, you need to set up the trigger conditions.

The following figure shows how triggers fit into the overall instrument operation.
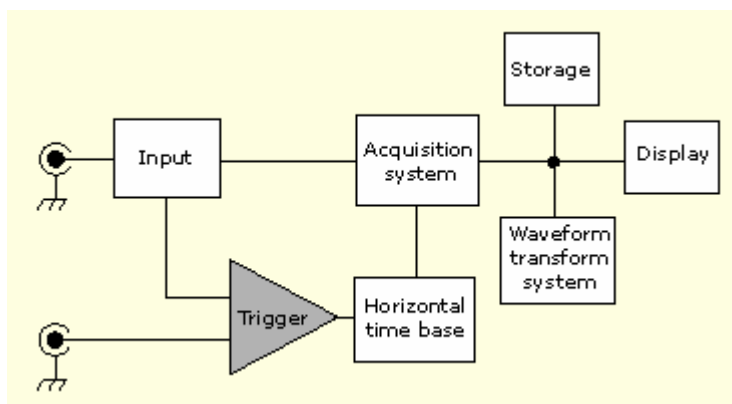
Triggers create meaningful waveforms from signal displays. This instrument has simple edge triggers as well as a variety of advanced triggers.
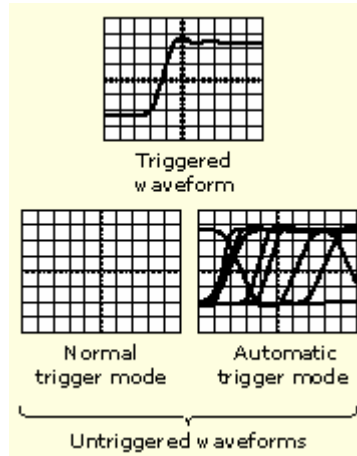
The Trigger Event

The trigger event establishes the time-zero point in the waveform record. All waveform record data are located in time with respect to that point. The instrument continuously acquires and retains enough sample points to fill the pretrigger portion of the waveform record (that part of the waveform that is displayed before, or to the left of, the triggering event on screen). When a trigger event occurs, the instrument starts acquiring samples to build the posttrigger portion of the waveform record (displayed after, or to the right of, the trigger event). Once a trigger is recognized, the instrument will not accept another trigger until the acquisition is complete and the holdoff time has expired.

Trigger Sources

The trigger source provides the signal that triggers acquisition. Use a trigger source that is synchronized with the signal that you are acquiring and displaying. You can derive your trigger from the following sources:

- Input channels are the most commonly used trigger sources. You can select any one of the two or four input channels, depending on your model. The channel that you select as a trigger source will function whether it is displayed or not.

- AC Line Voltage is often used to look at signals related to the power line frequency (for example, signals from devices such as lighting equipment and power supplies). Because the instrument generates the trigger from the power line, you do not have to use a channel input.

- Auxiliary Trigger (AUX IN) provides a fifth source that you can use as a trigger input when you need to use the four input channels for other signals. For example, you might want to trigger on a clock while displaying four other logic signals. To use the auxiliary trigger, connect the signal to the auxiliary input connector on the front panel. The input connector is not compatible with most probes, nor can you display the auxiliary trigger signal.

Trigger Types

- Edge triggers are the simplest and most commonly used trigger type, used with both analog and digital signals. An edge trigger event occurs when the trigger source passes through a specified voltage level in the specified direction (rising or falling signal voltage).

- Pulse triggers are special-purpose triggers that are primarily used with digital signals. The following types of pulse triggers are available: Glitch, Runt, Window, Width, Transition, and Timeout. Pulse triggers are available on the main trigger only.

- Logic triggers are special-purpose triggers that are primarily used with digital logic signals. Two of the types, Pattern and State, trigger the instrument based on the Boolean operator you select for the trigger sources. A third type, Setup and Hold, triggers when data in one trigger source changes state within the setup and hold times that you specify relative to a clock in another trigger source. Logic triggers are available on the main trigger only.

- Communication triggers (available only with Option SM) are designed to work with communication masks and standards. Mask testing automatically uses Communication triggers.

- Video triggers are used to trigger the instrument on specified fields or lines of a video signal. You can use one of several preset video signal formats or set a custom format.

Trigger Modes

The trigger mode determines how the instrument behaves in the absence of a trigger event:

- Normal trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the instrument does not acquire a waveform, and the last waveform record acquired remains "frozen" on the display. If no last waveform exists, no waveform is displayed.

- Auto trigger mode enables the instrument to acquire a waveform even if a trigger does not occur. Auto mode uses a timer that starts after a trigger event occurs. If another trigger event is not detected before the time out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting.

Auto mode, when forcing triggers in the absence of valid triggering events, does not synchronize the waveform on the display. In other words, successive acquisitions are not triggered at the same point on the waveform; therefore, the waveform will appear to roll across the screen. If valid triggers occur, the display will become stable.

Trigger Holdoff

Trigger holdoff can help stabilize triggering. When the instrument recognizes a trigger event, it disables the trigger system until acquisition is complete. In addition, the trigger system remains disabled during the holdoff period that follows each acquisition. Adjust holdoff to obtain stable triggering when the instrument is triggering on undesired trigger events.

A digital pulse train is a good example of a complex waveform. Each pulse looks like any other, so many possible trigger points exist. Not all of these will result in the same display. The holdoff period allows the instrument to trigger on the correct edge, resulting in a stable display.
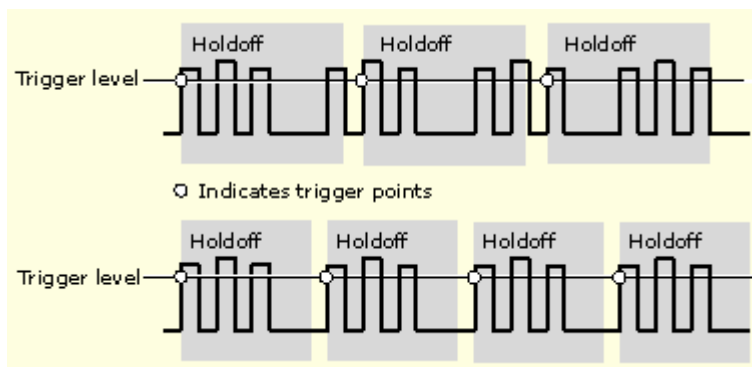
At the longer holdoff time for the top waveform, unstable triggering occurs. With a shorter holdoff set for the bottom waveform, triggers all occur on the first pulse in the burst to remedy the unstable trigger.

The Holdoff setting range is 1.5 µs (minimum holdoff available) to 12 seconds (maximum holdoff available). For more information on how to set holdoff, see page 121. You can also set a default holdoff. The default holdoff is the general-purpose holdoff for most applications and varies with the horizontal scale. It is equal to five times the current horizontal scale setting.

Trigger Coupling

Trigger coupling determines what part of the signal is passed to the trigger circuit. Edge triggering can use all available coupling types: AC, DC, Low Frequency Rejection, High Frequency Rejection, and Noise Rejection. All of the advanced trigger types use DC coupling only. For a description of each coupling type, see the Glossary.

Horizontal Trigger Position

Horizontal position is an adjustable feature that defines where the trigger occurs on the waveform record. It lets you choose how much the instrument acquires before and after the trigger event. The part of the record that occurs before the trigger is the pretrigger portion. The part that occurs after the trigger is the posttrigger portion. A longer posttrigger period my be useful when you want ot see the effects an event has on your system under test.

Pretrigger data can be valuable when troubleshooting. For example, if you are trying to find the cause of an unwanted glitch in your test circuit, you can trigger on the glitch and make the pretrigger period large enough to capture data before the glitch. By analyzing what happens before the glitch, you may uncover information that helps you find the source of the glitch.

Trigger Slope and Level

The slope control determines whether the instrument finds the trigger point on the rising or the falling edge of a signal. The level control determines where on that edge the trigger point occurs. See the figure below.
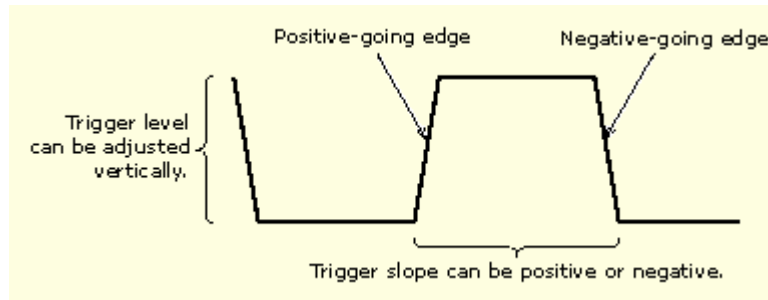
Delayed Trigger System

You can trigger with the A (Main) trigger system alone or you can combine the A (Main) trigger with the B (Delayed) trigger to trigger on sequential events. When using sequential triggering, the A trigger event arms the trigger system, and the B trigger event triggers the instrument when the B trigger conditions are met. A and B triggers can (and typically do) have separate sources. The B trigger condition can be based on a time delay or a specified number of counted events. For more details on delayed triggering see page 119.

Advanced Triggering

You can check the advanced trigger status in the readout. The readout indicates the trigger type and then shows sources, levels, or any other parameters that are important for the particular trigger type.

For details on advanced triggers see pages 238–242.

Glitch Trigger

A glitch trigger occurs when the instrument detects a pulse narrower (or wider) than some specified time. You can set the instrument to trigger on glitches of either polarity or to reject glitches of either polarity.

Runt Trigger

A runt trigger occurs when the instrument detects a short pulse that crosses one threshold but fails to cross a second threshold before recrossing the first. You can set the instrument to detect any positive or negative runt pulse, or only those wider than a specified minimum width. Runt pulses can also be qualified by the logical state of other channels.

Width Trigger

A width trigger occurs when the instrument detects a pulse that is inside or outside some specified time range. The instrument can trigger on positive or negative width pulses. Width triggers can also be qualified by the logical state of other channels.

Transition Time Trigger

Transition triggering is based on the slope (change in voltage/change in time) of a pulse edge. Use the transition trigger to trigger the instrument on pulse edges that traverse between two thresholds at faster or slower rates than the specified time. You can set up the instrument to trigger on positive or negative edges.

Timeout Trigger

A timeout trigger occurs when the instrument does not detect an expected pulse transition within a user specified period of time, such as when a signal gets stuck either high or low. If the pulse transition occurs prior to a specified timeout time (the expected case), then no trigger results.

Pattern Trigger

A pattern trigger occurs when the inputs to the selected logic function cause the function to become True or False. When you use a pattern trigger, you define:

- The precondition for each logic input: logic high, low, or "don't care"; the logic inputs are the instrument channels

- The Boolean logic function: AND, NAND, OR, or NOR

- The condition for triggering: the Boolean function becomes True (logic high) or False (logic low), and whether the True condition is time qualified

Pattern trigger logic choices are summarized in the following table:

Pattern	State	Definition 1, 2
AND	Clocked AND	If all the preconditions selected for the logic inputs[3] are TRUE, then the instrument triggers.
NAND	Clocked NAND	If not all of the preconditions selected for the logic inputs[3] are TRUE, then the instrument triggers.
OR	Clocked OR	If any of the preconditions selected for the logic inputs[3] are TRUE, then the instrument triggers.
NOR	Clocked NOR	If none of the preconditions selected for the logic inputs[3] are TRUE, then the instrument triggers.

[1] For state triggers, the definition must be met at the time the clock input changes state.

[2] The definitions given here are correct for the Goes TRUE setting in the Trigger When menu. If that menu is set to Goes False, swap the definition for AND with that for NAND and for OR with NOR for both pattern and state types.

[3] The logic inputs are channels 1, 2, 3, and 4 when using Pattern triggers. For State triggers, channel 4 becomes the clock input, leaving the remaining channels as logic inputs. (On the TDS5000B 2-channel instruments, channel 2 is the clock input.)

State Trigger

A state trigger occurs when the inputs to the logic function cause the function to become True or False at the time the clock input changes state. When you use a state trigger, you define:

- The precondition for logic input channels 1 through 3 (channel 1 on TDS5000B 2-channel instruments)

- The direction of the state change for the clock input, channel 4 (channel 2 on the TDS5000B 2-channel instruments)

- The Boolean logic function: AND, NAND, OR, or NOR (not available on the TDS5000B 2-channel instruments)

- The condition for triggering: the Boolean function becomes True (logic high) or False (logic low)

State trigger logic conditions are summarized in the following table.

Pattern	State	Definition [1,2]
AND	Clocked AND	If all the preconditions selected for the logic inputs[3] are TRUE, then the instrument triggers.
NAND	Clocked NAND	If not all of the preconditions selected for the logic inputs[3] are TRUE, then the instrument triggers.
OR	Clocked OR	If any of the preconditions selected for the logic inputs[3] are TRUE, then the instrument triggers.
NOR	Clocked NOR	If none of the preconditions selected for the logic inputs[3] are TRUE, then the instrument triggers.

[1] For state triggers, the definition must be met at the time the clock input changes state.

[2] The definitions given here are correct for the Goes TRUE setting in the Trigger When menu. If that menu is set to Goes False, swap the definition for AND with that for NAND and for OR with NOR for both pattern and state types.

[3] The logic inputs are channels 1, 2, 3, and 4 when using Pattern triggers. For State triggers, channel 4 becomes the clock input, leaving the remaining channels as logic inputs. (TDS5000B 2-channel instruments, channel 2 is the clock.)

Setup and Hold Trigger

A setup/hold trigger occurs when a data signal changes state inside of the user specified setup and hold times relative to the clock. When you use setup/hold triggering, you define:

- The channel containing the logic input (the data source) and the channel containing the clock (the clock source)

- The direction of the clock edge to use

- The clocking level and data level that the instrument uses to determine if a clock or data transition has occurred

- The setup and hold times that together define a time range relative to the clock

Data that changes state within the setup/hold violation zone triggers the instrument. The following figure shows how the setup and hold times that you choose position the violation zone relative to the clock.
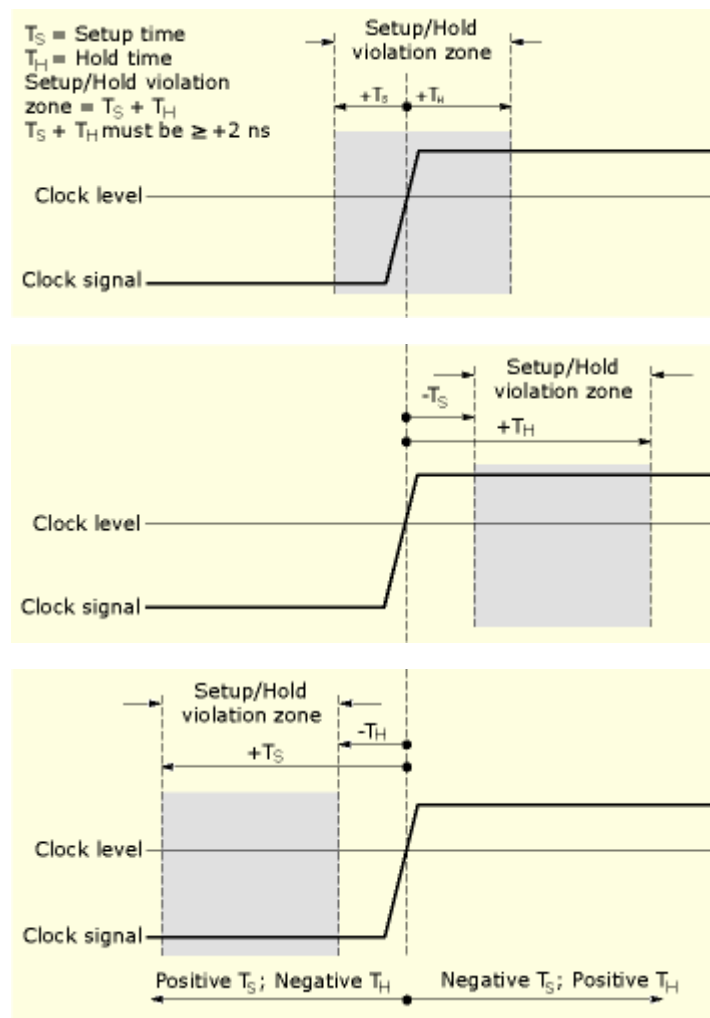
Setup/hold triggering uses the setup/hold violation zone to detect when data is unstable too near the time it is clocked. Each time trigger holdoff ends, the instrument monitors the data and clock sources. When a clock edge occurs, the instrument checks the data stream it is processing (from

the data source) for transitions occurring within the setup/hold violation zone. If any occur, the instrument triggers with the trigger point located at the clock edge.

Positive settings for both setup and hold times (the most common application) locate the setup/hold violation zone so it spans the clocking edge as shown above in the top waveform. The instrument detects and triggers on data that does not become stable long enough before the clock (setup time violation) or that does not stay stable long enough after the clock (hold time violation).

Negative settings for setup or hold times skew the setup/hold violation zone to locate it before or after the clocking edge as shown above in the center and bottom waveforms. The instrument can then detect and trigger on violations of a time range that occurs before or one that occurs after the clock.

Note

Keep the hold-time setting to no more than 2.5 ns less than one-half the clock period (hold time v (period/2) –2.5 ns) or the instrument cannot trigger (this assumes a 50% duty cycle clock).

Window Trigger

Use the Window trigger to trigger the instrument when the input signal rises above an upper threshold level or falls below a lower threshold level. After setting these levels, you can specify whether you want to trigger the instrument as the signal is entering or leaving the threshold window. You can further qualify the trigger event in terms of time, or by the logical state of other channels.

Video Trigger

Use the Video trigger to trigger the instrument on specified fields or lines of a composite video signal. Select from several preset video signal formats or set a custom format. Only composite signal formats are supported. Graphic display formats such as RGB and VGA are not supported.

Communication Trigger

Use the Comm trigger to trigger the instrument in conjunction with mask testing on communications codes and standards. The controls work together to define the parameters for the trigger event. The standards that appear in the Standard drop-down list depend on the coding format that you selected. The coding format also determines whether upper, lower, single, or clock threshold levels or pulse forms are used. The standard determines the bit rate.

Sequential Triggering

In applications that involve two or more signals, you may be able to use sequential triggering to capture more complex events. Sequential triggering uses the A (Main) trigger to arm the trigger system, and then uses the B (Delayed) trigger to trigger the instrument if a specific condition is met. You can choose one of two trigger conditions:

▪ Trigger After Time: After the A trigger arms the trigger system, the instrument triggers on the next B-trigger event that occurs after the trigger delay time. You can set the trigger delay time with the keypad or the multipurpose knobs.

▪ Trigger on nth Event: After the A trigger arms the trigger system, the instrument triggers on the nth B event. You can set the number of B events with the keypad or the multipurpose knobs.

Note

The traditional delayed trigger mode called "Runs After" is served by the TDS5000B feature Horizontal Delay. You can use horizontal delay to delay acquisition from any trigger event, whether from the A (Main) trigger alone or from a sequential trigger that uses both the A (Main) and B (Delayed) triggers. See Triggering with Horizontal Delay On on page 244 for more information.

Keep in mind the following when using sequential triggers:

- In most cases, you will set separate trigger sources for the A (Main) and B (Delayed) triggers.
- Sequential triggering for Trigger after Time or Trigger on nth Event is available for the Edge, Glitch, Width, or Timeout types. It is not available for the rest of the trigger types.
- The B trigger is always the Edge type.

Triggering with Horizontal Delay Off

The following figure compares the sequential trigger choices A Only, Trig After Time, and Trig on nth Event when horizontal delay is off. Each illustration shows where pretrigger and posttrigger data is acquired relative to the trigger event.
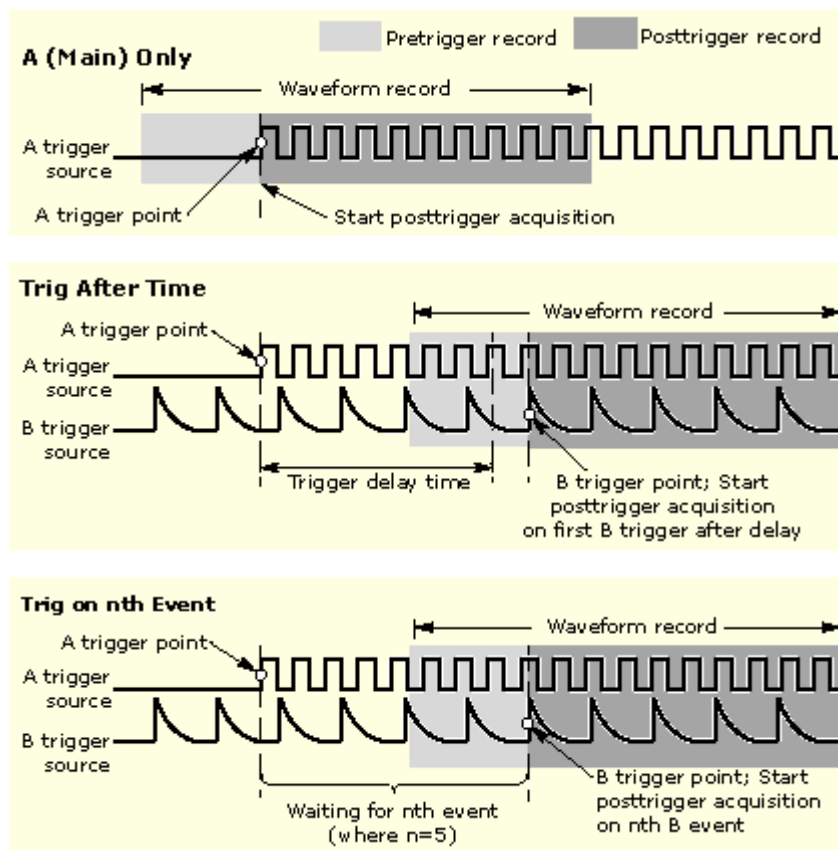
Triggering with Horizontal Delay On

You can use horizontal delay when you want to acquire a waveform record that is separated from the trigger event by a significant interval of time. The horizontal delay function can be used with any trigger setup. You can turn horizontal delay on and off from the front panel, the Horizontal/Acquisition control window, and many of the Trigger control windows. The following figure compares the sequential trigger choices A Only, Trig After Time, and Trig on nth Event when horizontal delay is on. Each illustration shows where pretrigger and posttrigger data is acquired relative to the trigger event.
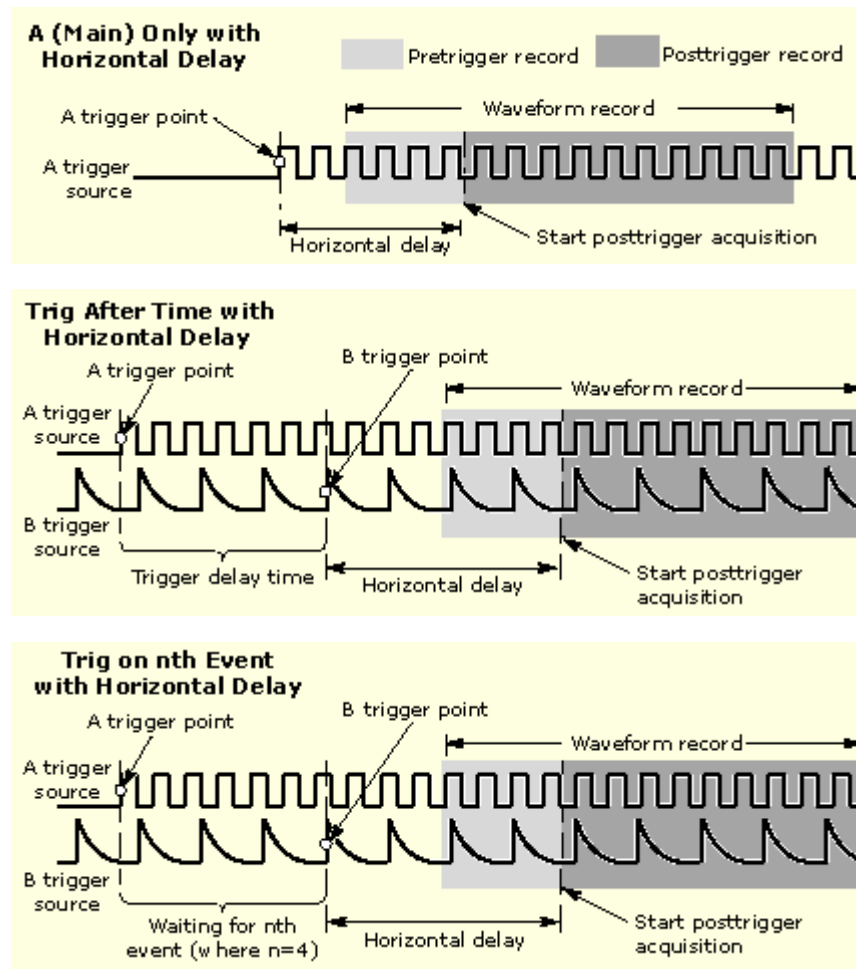
Triggering and Horizontal Delay Summary

The following figure shows all combinations of triggering and horizontal delay.
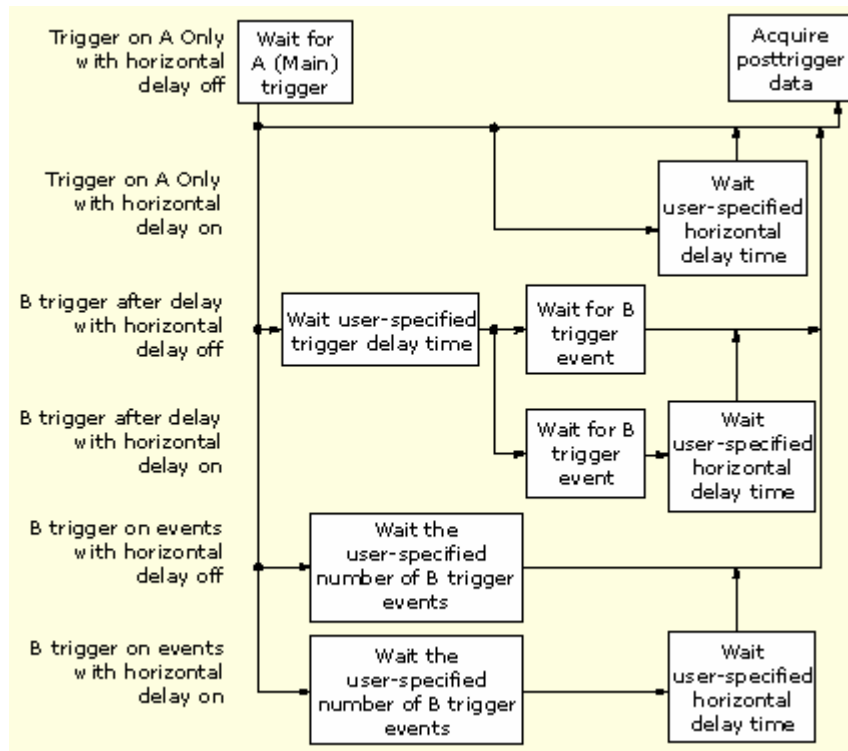
Display Overview

This instrument includes a flexible, customizable display that lets you control how waveforms appear. The following figure shows how the display features fit into the overall instrument operation.
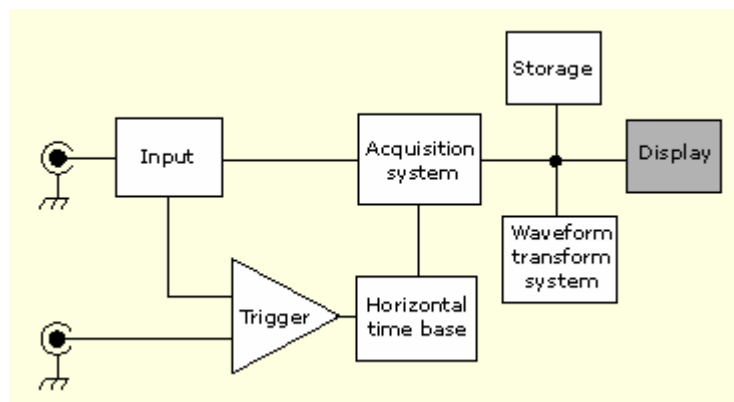
Display Elements

The waveform shown below is displayed as part of the user interface (UI) application. Some terms that are useful in discussing the UI follow.
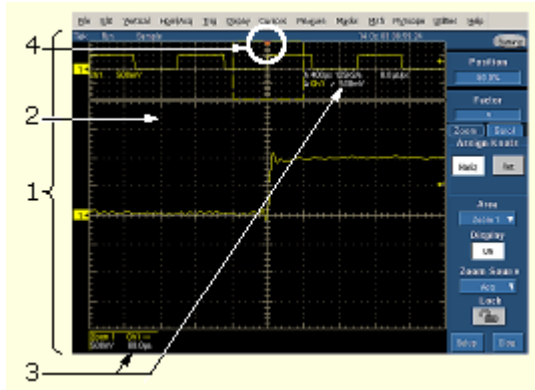
(1) The Display is the area where the waveforms appear. The display comprises the time base and graticules, the waveforms, histograms, and some readouts.

(2) The Graticule is the grid marking the display area. When MultiView Zoom is on, depending on the graticule split setting, the upper graticule displays unmagnified waveforms and the lower graticule displays magnified waveforms.

(3) The Horizontal scale readout provides a quick overview of the horizontal settings for zoomed and unzoomed waveforms.

(4) The Horizontal reference is a control that you can position to set the point around which channel waveforms expand and contract horizontally on screen as you change the Horizontal Scale control. It is also the trigger point when the horizontal delay is 0%.

See the following figure for some ideas on how to manipulate the display elements.
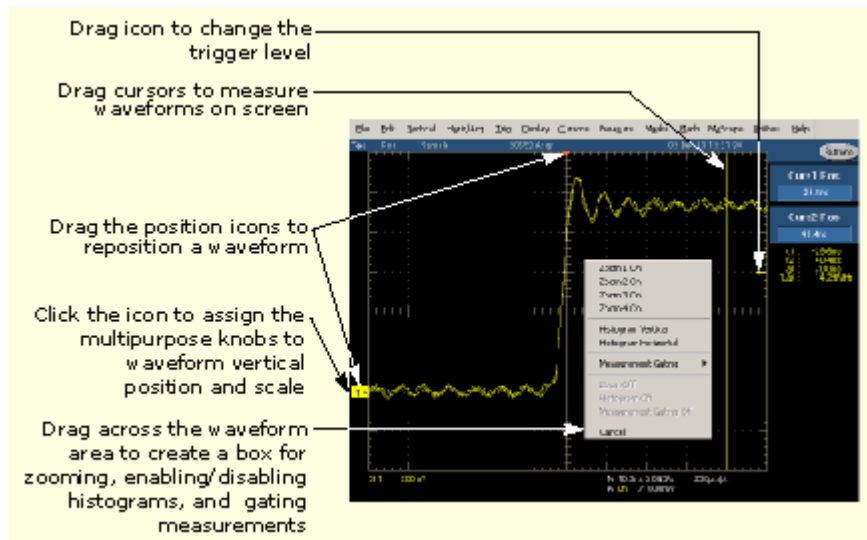
Acquisition Preview

The acquisition preview attempts to show what the next acquisition will look like when the acquisition is delayed due to slow triggers or long acquisition duration, or when the acquisitions have stopped. Acquisition preview recalculates math waveforms, but does not represent changes in trigger levels, trigger modes, or different acquisition modes.

Acquisition preview has some limitations. For example, if the record length is changed, the waveforms are erased. Changing the horizontal scale can change the record length in some cases.

Note

There is no acquisition preview in Fast Acquisitions mode.

Waveform Display

In general, the method of displaying a waveform is to:

▪ Define the waveform (math and reference waveforms only)

▪ Enable the waveform display

The following table summarizes this process as it applies to the different types of waveforms.

Waveform	To define	To display
Channel: Ch1 – Ch4	Channels are predefined	Push the Vertical CH n button to toggle the channel display on or off.
Reference: Ref1 – Ref4	Define an active reference waveform by: ▪ Saving a channel, reference, or math waveform to one of locations Ref1 - Ref4. ▪ Recalling a waveform previously saved to a file into one of locations Ref1 - Ref4. Perform these operations from the File menu.	From the Reference Setup control window, click Display on the Ref n tab to toggle the reference waveform display on and off.
Math: Math1 – Math4	Create a math waveform using existing sources (channel, math, and reference waveforms; measurements).	From the Math Setup control window, click Display on the Math n tab to toggle the math waveform display on and off.

In general, the method of adjusting (vertically scaling, offsetting, positioning, and so on) the waveform display is from the front-panel vertical SCALE and POSITION knobs. Note that math and reference waveforms are scaled and positioned from their setup control windows.

For channel waveforms, the vertical and horizontal controls that you set also adjust the instrument acquisition parameters. See the following for more information:

- Vertical Acquisition Window Considerations on page 227.
- Horizontal Acquisition Window Considerations on page 229.

To change the size of the acquisition waveform and zoomed waveform windows, select the Graticule Size in the Zoom Setup control window. The 50-50 selection allocates half of the available display for the zoomed graticule and half of the available display for the acquisition window. The 80-20 selection allocates 80% of the available display for the zoomed graticule and 20% for the acquisition window. Select Full to use the entire display for the zoomed graticule.

The display elements are the same for each graticule.

Customizable Display Elements

The following table lists customizable display elements and their points of access.

Display attribute	Access	Options
Graticule Style	Display menu Graticule Style command or Display Setup control window Objects tab Disp toolbar button)	• Full provides a grid, cross hairs, and frame on the instrument display. Use for quick estimates of waveform parameters. • Grid shows a frame and grid on the instrument display. Use for full-screen measurements with cursors and automatic readouts when cross hairs are not needed. • Cross Hair shows cross hairs and a frame on the instrument display. Use for making quick estimates of waveforms while leaving more room on the display for automatic readouts and other data. • Frame shows only a frame on the instrument display. Use with automatic readouts and other screen text when other display features are not needed. • IRE is used for NTSC video signals. • mV is used for video signals other than NTSC.
Display Persistence	Display menu Display Persistence submenu commands	• Infinite persistence continuously accumulates record points on the waveform until you change one

Display attribute	Access	Options
	or Display Setup control window Appearance tab (Disp toolbar button)	of the acquisition display settings. Use for displaying points that may occur outside the normal acquisition envelope. ▪ Variable persistence accumulates record points on the waveform for a specified time interval. Each record point decays independently according to the time interval. ▪ Off shows waveform record points for the current acquisition only. Each new waveform record replaces the previously acquired record for a channel.
Display Style	Display menu Display Style submenu commands or Display Setup control window Appearance tab (Disp toolbar button)	▪ Vectors display waveforms with lines drawn between record points. ▪ Dots display waveform record points as dots on the screen. ▪ Inten Samp (Intensified Samples) shows the actual samples on interpolated waveforms.
Screen Text	Display menu Screen Text command or Display Setup control window Screen Text tab (Disp toolbar button)	▪ Up to eight (8) lines of text provides notes for screen shots, printouts, or for other instrument users. ▪ Display turns the text display on and off. ▪ Properties opens the Text Properties control window for positioning of the text on the display. ▪ Clear erases the entire text of the selected line.
Color Palette (Graticule and Waveform)	Display menu Colors command or Display menu Display Palette submenu	▪ Normal displays hues and lightness levels for best overall viewing. The color of each channel waveform matches the color of the corresponding front-panel vertical SCALE knob. ▪ On the Spectral Spectral Grading) palette, areas of the

Display attribute	Access	Options
	or Display Setup control window Colors tab (Disp toolbar button)	waveform with the highest sample density appear in blue shades and the areas of lowest sample density appear in red shades.
		• On the Temp (Temperature Grading) palette, areas of the waveform with the highest sample density appear in warmer colors (red shades) and the areas of lowest sample density appear in cooler colors (blue shades).
		• Gray (Monochrome Gray) displays waveforms in shades of gray. Areas of the waveform with the highest sample density appear in lighter gray shades and the areas of lowest sample density appear in darker gray shades.
		• Green (Monochrome Green) displays waveforms in shades of green. Areas of the waveform with the highest sample density appear in lighter green shades and the areas of lowest sample density appear in darker green shades.
Reference Colors	Display menu Colors command or Display Setup control window Colors tab (Disp toolbar button)	• Default uses the default system color (white) for reference waveforms. • Inherit uses the same color for the reference waveform as the original waveform.
Math Colors	Display menu Colors command or Display Setup control window Colors tab Disp toolbar button)	• Default uses the default system color (red) for math waveforms. • Inherit uses the same color for the math waveform as the waveform the math function is based on.
Waveform Interpolation	Display menu Waveform Interpolation	• Sin(x)/x interpolation • Linear interpolation

Display attribute	Access	Options
	submenu or Display Setup control window Appearance tab (Disp toolbar button)	Refer to Interpolation on page 253 for more information.
Waveform Intensity	Display menu Appearance command or Display Setup control window Appearance tab (Disp toolbar button)	▪ When AutoBright is Off, the display simulates the appearance of analog oscilloscope signals. Waveforms that trigger more frequently appear brighter than waveforms that trigger less frequently. Use the front-panel INTENSITY knob or the multipurpose knobs to manually adjust the display intensity when AutoBright is Off. ▪ When AutoBright is On, the intensity of the waveform changes automatically to provide a visible waveform.
Display Format	Display menu Appearance command or Display menu Display submenu or Display Setup control window Appearance tab (Disp toolbar button)	▪ YT ▪ XY ▪ XYZ For additional information, see Display Format on page 140.
Trigger Level Marker	Display menu Objects command or Display Setup control window Objects tab (Disp toolbar button)	▪ The Short trigger level marker displays a short arrow on the side of the graticule by the active waveform. ▪ The Long trigger level marker displays a horizontal line through the active waveform.

Display attribute	Access	Options
LCD Backlight Timeout	Display menu Objects command or Display menu LCD Save Enabled command or Display Setup control window Objects tab (Disp toolbar button)	The LCD Backlight control turns the LCD backlight off after the amount of time in Delay has expired without any screen activity. ▪ When LCD Backlight Timeout is On and the time in Delay has expired, the screen blanks out. ▪ When LCD Backlight Timeout is Off, the screen display remains on.
Date and Time	Display menu Objects command or Display menu Display Date & Time command or Display Setup control window Objects tab Disp toolbar button)	Toggle the display of the date and time on the graticule. To set the time and date, use the Utilities menu Set Time & Date command.

Operations on the Time Base

In general, to adjust the time base, use the front-panel horizontal SCALE, RESOLUTION, and POSITION knobs. Only channel waveforms can be set directly.

Here are some key points to remember about how horizontal operations relate to the waveform types:

- Reference waveforms are shown with the horizontal settings in effect at the time that they are saved. You cannot adjust these settings. See Saving and Recalling Waveforms on page 287 more information on reference waveforms.

- Math waveforms are shown with the horizontal settings derived from the math expression that creates them. You cannot change these directly. See Creating or Using Math Waveforms for more information on math waveforms.

- All waveforms are displayed within the horizontal divisions that the graticule provides. However, some waveforms may actually be wider or narrower than the display due to acquisition rate/time scale combinations and acquisition preview.

Horizontal Position and the Horizontal Reference Point

The time value you set for horizontal position is measured from the trigger point to the horizontal reference point. This is not the same as the time value from the trigger point to the start of the waveform record, unless you set the horizontal reference to 0%. See the following figure.
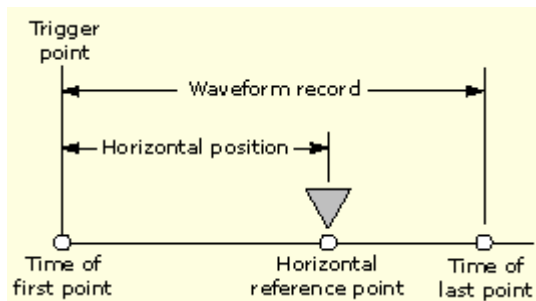
Interpolation

When the sample density falls to less than 1 sample per display column, the instrument must calculate intermediate points to display a waveform. This process is called interpolation. There are two options for interpolation:

- Sin(x)/x interpolation computes record points using a curve fit between the actual values acquired. It assumes that all the interpolated points fall along that curve.

- Linear interpolation computes record points between actual acquired samples by using a straight line fit. It assumes that all the interpolated points fall in their appropriate point in time on that straight line.

For example, if an acquired record length is 500 points, a 2x zoom requires interpolation. If a record length is 100K, horizontal zoom of 200x produces 1 sample per column (100,000/500 = 200); therefore, you will see interpolated samples starting at the next faster scale setting.

Using the MultiZoom Feature

Use the instrument MultiZoom feature to magnify an acquisition vertically, horizontally, or in both dimensions to let you see the fine detail in your signals without changing the acquisition parameters (sample rate, record length, and so on). For example, to temporarily expand the front corner of a pulse to inspect its aberrations, push the front-panel MultiZoom button to expand it horizontally and vertically.

To help you use Mulitzoom effectively, consider how it operates on waveforms:

▪ When zooming vertically, the instrument expands up to four zoomed areas at one time. The zoomed areas can all be on the same waveform or they can be on different Channel, Math or Reference waveforms. Each zoomed area has its own vertical position and factor.

▪ When zooming horizontally, the Zoom Source is either the acquisition waveform or one of the zoomed areas.

▪ When zooming either horizontally or vertically, Zoom expands or contracts the waveform by the Zoom Factor.

Automatic Measurements

All channel, reference, and math waveforms can serve as sources for automatic measurements. Some measurements, such as Delay and Phase, require two sources.

The following automatic measurement categories are available:

▪ Amplitude

▪ Time

▪ More (miscellaneous)

▪ Histogram

▪ Communication

The instrument can take and update up to eight measurements at one time. You can apply measurements to any combination of sources. For example, you can take all eight measurements on Channel 1, or you can take measurements on a combination of channel waveforms, math waveforms, reference waveforms, and/or histograms.

High/Low Method

The levels that the automatic measurement system derives as the High (top) or Low (bottom) for a waveform influence the fidelity of amplitude and aberration measurements. Select among the modes the instrument provides for determining these levels:

▪ Histogram mode sets the values statistically. It selects the most common value either above or below the midpoint (depending on whether it is defining the High or Low reference level). Since this statistical approach ignores short term aberrations (overshoot, ringing, and so on), Histogram is the best setting for examining pulses. See the figure below.

▪ Min-Max mode uses the highest and lowest values of the waveform record. This setting is best for examining waveforms that have no large, flat portions at a common value, such as sine waves and triangle waves (almost any waveform except for pulses). See the figure below.
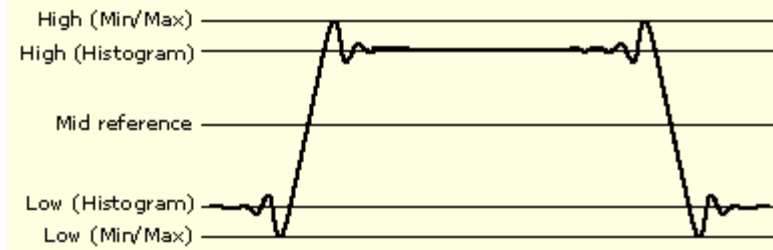
Reference Levels Method

A second set of levels affect the fidelity of time-related measurements. For example, the measurement system calculates rise time from the waveform edge that transitions from the Low to the High reference levels.

The instrument provides the following calculation methods (see the following figure):

- Relative Reference is calculated as a percentage of the High/Low range.
- Absolute Reference is set by absolute values in user units.
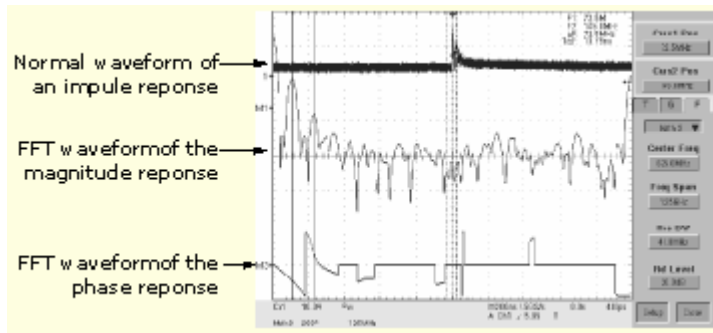
Cursor Measurements

Because cursor measurements give immediate feedback on the amplitude or time values they measure, they are usually quick to take and are more accurate than graticule measurements. Also, since you can position cursors wherever you want them on the waveform, they are easier to localize to a waveform segment or feature than automatic measurements.

Cursors can measure channel, reference, and math waveforms. You must set the source of each cursor explicitly in the Cursor Setup control window.

Any cursor type can be used for YT display format. XY and XYZ display format can only use Screen or Waveform cursors. If FastAcq is on, XYZ display format can only use Screen cursors.

To use cursors, simply move the cursors on screen and read the results in the cursor readouts. The following key points will help you use the cursors effectively.

Cursor Types

There are two cursors displayed for all types, Cursor 1 and Cursor 2. You can move cursors with the multipurpose knobs or the cursor position controls in the Cursor Setup control window, or click and drag them to a new location.

Cursors are Limited by the Display

You cannot move a cursor off the graticule unless you are in Zoom mode. Also, if you resize waveforms, the cursors do not track. That is, a cursor stays at its screen position, ignoring changes to horizontal and vertical scale and position and vertical offset (Waveform and Screen cursors will track a waveform vertically).

Cursors Ignore the Selected Waveform

Each cursor measures its source, defined in the Cursor Setup control window. Selecting a waveform for scaling on screen (by pushing the CH 3 front-panel button, for example) does not change the source that each cursor measures.

After you have selected the source from the Cursor Setup control window, you can operate the cursor from the front-panel knobs and buttons.

Note

If a cursor readout does not seem correct, check the source of each cursor in the Cursor Setup dialog box. Each cursor readout relates to the amplitude and time base settings of its source.

Vertical Cursors Measure from the Trigger Point

Remember that each vertical cursor measures the time from the trigger point to itself. This relationship is shown in the following figure.
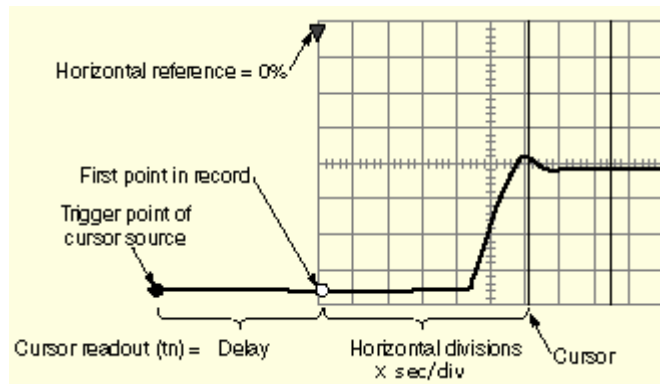
Note that a vertical cursor readout includes and varies directly with the Delay time, which varies directly with the horizontal position set for the time base. To see the amount of time to the first point, set horizontal delay to 0.0 and set the horizontal reference point to 0%. Now the horizontal position readout shows the time following the first point, and adding this value to the cursor readout yields the cursor position on screen relative to first point. (You can find the horizontal readout both in the control window and in the readout at the bottom of the screen.) The following relationships hold:

Time from First Point = Horiz. Position (when Horiz. Delay and Ref Position are zero)

T1 readout = Time to First Point + Additional Time to Cursor

Cursor Units Depend on Sources

A cursor that measures amplitude or time will read out in the units of its source as indicated in the table below. Note mixed sources require delta cursor readouts to use the units of the Cursor 1 source.

Cursors	Standard units[1]	Readout names
Horizontal	volts, watts, IRE	V1, V2, ΔV
Vertical	seconds, bits, Hz	T1, T2, ΔT, ΔF
Waveform, Screen	volts, watts, seconds, bits, Hz	V1, V2, ΔV, T1, T2, ΔT

[1] **If the V1 and V2 units do not match, the ΔV readout defaults to the units used by the V1 readout.**

Changing Cursor Positions

Change cursor positions by using the position controls in the Cursor Setup control window, dragging the cursor into position using the touch screen or mouse, or by turning the front-panel multipurpose knobs.

Note

To make small changes with the multipurpose knobs, push the FINE button before turning the knobs. When a FINE button is lighted, its multipurpose knob makes smaller adjustments.

Cursor Types

Because cursor measurements give immediate feedback on the amplitude or time values they measure, they are usually quick to take and are more accurate than graticule measurements. Also, since you can position cursors wherever you want them on the waveform, they are easier to localize to a waveform segment or feature than automatic measurements.

The following table defines the cursor types.
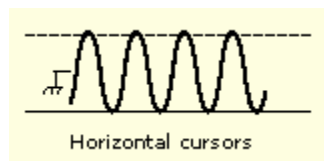
Cursor function	Measurements
 Horizontal cursors	Horizontal cursors measure vertical parameters (amplitude) in volts, watts, or IRE. The cursor readouts are defined as: V1 = Level @ Cursor 1 with respect to its source ground level V2 = Level @ Cursor 2 with respect to its source ground level ΔV = Level @ Cursor 2 - Level @ Cursor 1 Level is cursor displacement from the source ground multiplied by the source in volts/div. The two cursors may have different sources and therefore can have different volts/div settings.
 Vertical cursors	Vertical cursors measure horizontal parameters. Typically the horizontal parameters are in time or frequency, but depend on the units you select. The cursor readouts are defined as: T1 = Time @ Cursor 1 with respect to the trigger point
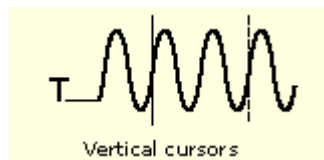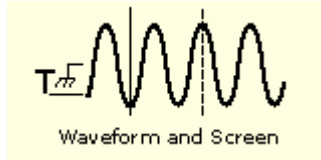
Cursor function	Measurements

T2 = Time @ Cursor 2 with respect to the trigger point

ΔT = Time @ Cursor 2 - Time @ Cursor 1

Time is divisions of displacement of the cursor from its source trigger point multiplied by the source in time/div.

Waveform and Screen

Waveform cursors measure vertical parameters (typically volts) and horizontal parameters (typically time or frequency) simultaneously. Each cursor is, in effect, both a vertical and horizontal cursor. These paired cursors cannot be moved off of the waveform. The two cursors can have a different source.

Use screen cursors to indicate an arbitrary X and Y position in the waveform display area represented by the intersection of a vertical and horizontal cursor line. Screen cursors measure both vertical parameters (typically volts) and horizontal parameters (typically time or frequency).

Histograms

The instrument can display histograms constructed from the selected waveform data. You can display both vertical (voltage) and horizontal (time) histograms, but only one at a time. Use histogram measurements to get statistical measurement data for a section of a waveform along one axis.

The histogram source can be any channel, math, or reference waveform.

Turning on histograms starts histogram counting and data accumulation. A sample histogram display is shown below. Histogram data is continuously accumulated until you explicitly turn off histograms. This allows you to continue collecting histogram data even when you turn off the histogram display.

In addition to using limit controls to set histogram box boundaries, you can also use standard Windows drag-and-drop to resize and reposition the histogram box.

Histograms are not available in FastFrame or Zoom modes.
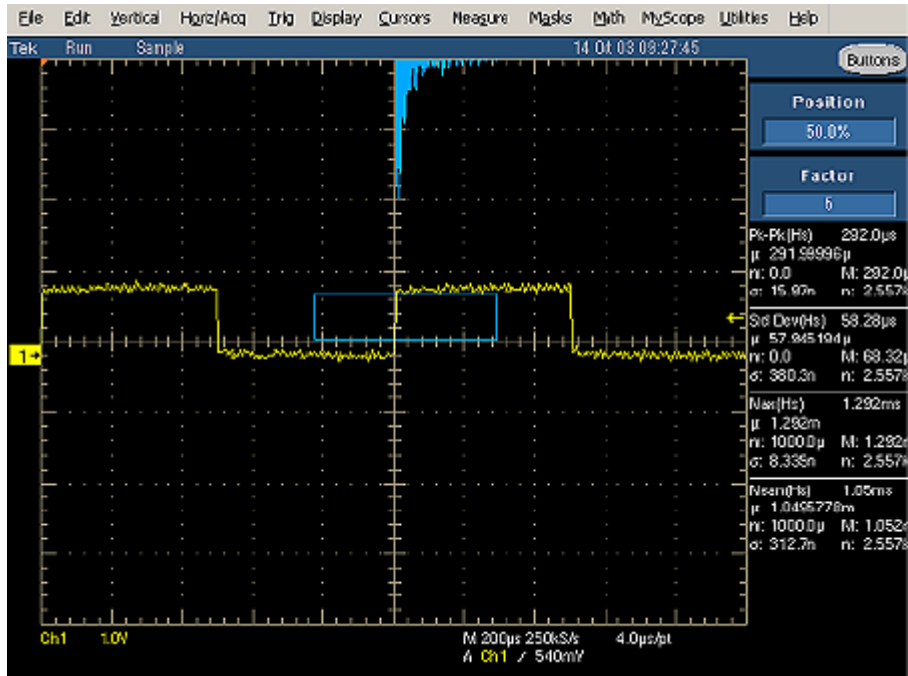
Math Waveforms

Once you have acquired waveforms or taken measurements on waveforms, the instrument can mathematically combine them to create a waveform that supports your data-analysis task. For example, you might have a waveform clouded by background noise. You can obtain a cleaner waveform by subtracting the background noise from your original waveform. Or, you can integrate a single waveform into an integral math waveform as shown below.
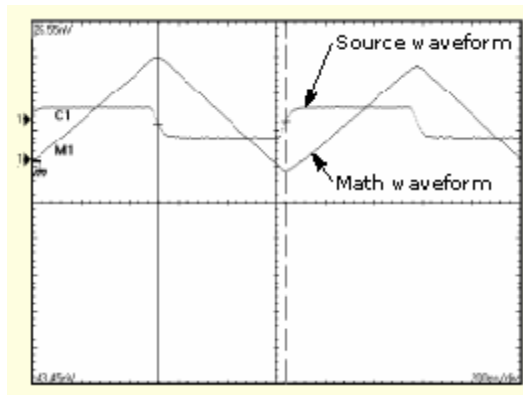
With spectral analysis you can analyze waveforms in the frequency domain. The interface is similar to a dedicated spectrum analyzer, so you do not need to know the details of the underlying algorithms. See the following figure.

Normal waveform of an impulse reponse

FFT waveform of the magnitude reponse

FFT waveform of the phase reponse
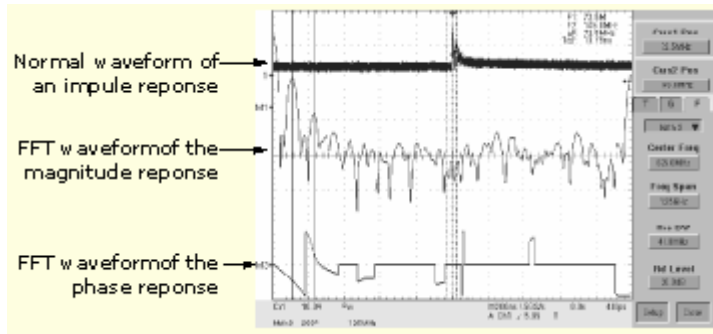
This instrument supports mathematical combination and functional transformations of waveforms it acquires. The following figure shows this concept:

Channel waveform (Ch2)

Math expression (Function(source))

Diff(Ch2)

Math waveform (Math1)
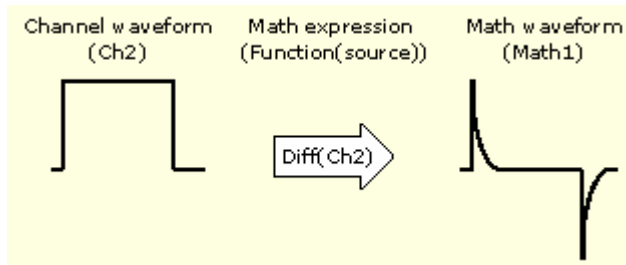
You create math waveforms to support the analysis of your channel and reference waveforms. By combining and transforming source waveforms and other data into math waveforms, you can derive the data view that your application requires. Create math waveforms that result from:

- Mathematical operations on one or several waveforms: add, subtract, multiply, and divide.

- Functional transformations of waveforms, such as integration, differentiation, and so on.

- Spectral analysis of waveforms, such as testing impulse response.

You can create up to four math waveforms (two for 2-channel TDS5000B instruments).

Measurement scalars can be used in math expressions. For example, you can measure the average of a waveform (using the measurement capabilities of the instrument) and subtract it from the original waveform to define a new math waveform. Measurements 1 through 8 are allowed in a math definition, but not measurement functions (such as rise (Ch1)).

In FastFrame, math is applied to each frame.

This instrument supports the following operations for math waveforms:

- Vertical display scaling and positioning

- Taking automatic measurements

- Taking cursor measurements

- Using histograms

- User defined vertical units

In addition to the operations listed above, you can save math waveforms as reference waveforms.

Some operations that you cannot use with math waveforms are:

- **Circular Math-on-Math:** You cannot use circular definitions of math waveforms. For example, if you define Math2 = Ch1 - Math1, and then define a second math waveform as Math3 = Ch2 + Math2, you cannot define a third math waveform as Math1 = Math2 + Ch3. If

you do, the Math1 definition is rejected with an error because a circular definition is not allowed.

- ▪ **Fast Acquisitions**: Math is not allowed in Fast Acquisitions mode.

- ▪ **Roll Mode**: Math is not allowed in roll mode until you push STOP.

- ▪ **Different Length Sources**: If you use two waveforms (channel and reference waveforms) of different lengths, the length of the math waveform will be the shorter of the two lengths.

Creating Math Waveforms

You create math waveforms when you create a math expression. You do so by applying numerical constants, math operators and functions to operands (see Sources on page 262). You can display and manipulate these derived math waveforms much like you can the channel and reference waveforms (see Using Math Waveforms on page 328).

Some examples of typical math waveforms follow:

To normalize this waveform:
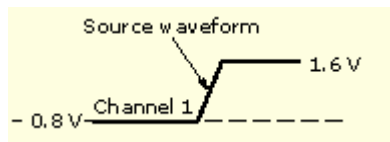
Enter this math expression:

(Ch1 - Meas1)/ Meas2,
where
Ch1 is the waveform shown at left
Meas1 = Low of Ch1
Meas2 = amplitude of Ch1

And get this math waveform:
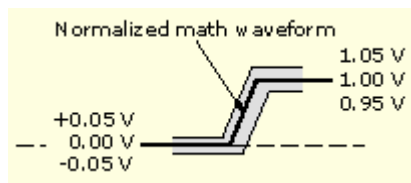
To simulate AC coupling on this waveform:
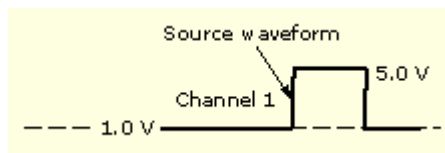
Enter this expression:

Intg(Ch1-Avg(Ch1)),
where
Ch1 is the waveform shown at left
Avg is the average function

And get this math waveform:
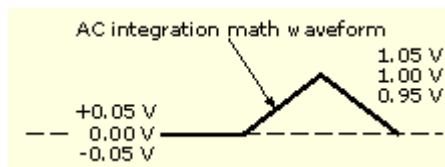
Offset, Position, and Scale

The settings that you make for offset, scale, and position affect the math waveform you obtain. Here are some tips for obtaining a good display:

- Scale and position the source waveform so that it is contained on the screen. (Off-screen waveforms may be clipped, resulting in errors in the derivative waveform).

- Use vertical position and vertical offset to position your source waveform. The vertical position and offset will not affect your derivative waveform unless you position the source waveform off screen so that it is clipped.

Math Waveform Sources

Math waveforms can be created from:

- Channel waveforms

- Reference waveforms

- Measurement scalars (automated measurements) that measure channel, reference, or math waveforms, or histograms.

- Other math waveforms

- Variables

Dependencies

In general, math waveforms that include sources as operands are affected by updates to those sources:

- Shifts in amplitude or DC level of input sources that cause the source to clip also clip the waveform data supplied to the math waveform.

- Changes to the vertical offset setting for a channel source that clip its data also clip the waveform data supplied to the math waveform.

- Changes to the acquisition mode globally affects all input channel sources, modifying any math waveforms using them. For example, with the acquisition mode set to Envelope, a Ch1 + Ch2 math waveform will receive enveloped channel 1 and channel 2 data, and will also be an envelope waveform.

- Clearing the data in a waveform source causes a baseline (ground) to be delivered to any math waveform that includes that source until the source receives new data.

Math Waveform Expression Syntax

Build math waveforms using the Predefined Expressions or Equation Editor control window To help you create valid math waveforms, these tools block most illegal entries by disabling any window element that would create an invalid entry in the math waveform expression.

Predefined expressions are performed from the Math Setup control window by selecting an expression.

The syntax that follows describes valid math expressions using the Equation Editor:

<MathWaveform> := <Expression>

<Expression> := <UnaryExpression> | <BinaryExpression>

<UnaryExpression> := <UnaryOperator> (<Term>)

| <UnaryOperator> (<Expression>)

<BinaryExpression> := <Term> <BinaryOperator> <Term> | <Scalar>

      <BinaryOperator> <Term> | <Term> <BinaryOperator> <Scalar>

<Term> := <Waveform> | (<Expression>)

<Scalar> := <Integer> | <Float> | <Meas-Result>

<Waveform> := <ChannelWaveform> | <ReferenceWaveform> | <MathWaveform>

<ChannelWaveform> := Ch1 | Ch2 | Ch3 | Ch4

<ReferenceWaveform> := Ref1 | Ref2 | Ref3 | Ref4

<MathWaveform> := Math1 | Math2 | Math3 | Math4

<UnaryOperator> := Invert | Derivative | Integral | Average | Max | Min

     | Filter | Vmag | Exp | log 10 | log e | sqrt | Spectral Magnitude

     | Spectral Phase | Spectral Real | Spectral Imag

<BinaryOperator> := + | - | / | *

<Meas-Result> := meas1 | meas2 | meas3 | meas4 | meas5 | meas6 | meas7 | meas8

<Variable> = VAR1 |VAR2 |VAR3 |VAR4

Math Waveform Differentiation

The math capabilities of the instrument include waveform differentiation. This allows you to display a derivative math waveform that indicates the instantaneous rate of change of the waveform acquired.

Derivative waveforms are used in the measurement of slew rate of amplifiers and in educational applications. You can create a derivative math waveform and then use it as a source for another derivative waveform. The result is the second derivative of the waveform that was first differentiated.

The math waveform, derived from the sampled waveform, is computed based on the following equation:

$Y_n = (X_{(n+1)} - X_n) \ 1/T$

Where: X is the source waveform

      Y is the derivative math waveform

      T is the time between samples

Since the resultant math waveform is a derivative waveform (see the figure below), its vertical scale is in volts/second (its horizontal scale is in seconds). The source signal is differentiated over its entire record length; therefore, the math waveform record length equals that of the source waveform.
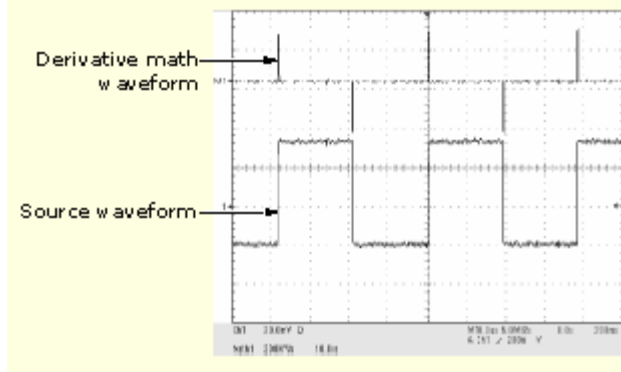
Cursor Measurements

You can also use cursors to measure derivative waveforms. Use the steps in Taking Cursor Measurements on page 353. When using that procedure, note that the amplitude measurements on a derivative waveform will be in volts per second rather than in volt-seconds as is indicated for the integral waveform measured in the procedure.

Offset, Position, Scale, and Math Waveforms

The settings that you make for offset, scale, and position affect the math waveform you obtain. Here are some tips for obtaining a good display:

- Scale and position the source waveform so that it is contained on the screen. (Off-screen waveforms may be clipped, resulting in errors in the derivative waveform).

- Use vertical position and vertical offset to position your source waveform. The vertical position and offset will not affect your derivative waveform unless you position the source waveform off screen so that it is clipped.

Waveform Integration

The math capabilities of the instrument include waveform integration. This allows you to display an integral math waveform that is an integrated version of the acquired waveform.

Use integral waveforms in the following applications:

- Measuring power and energy, such as in switching power supplies.

- Characterizing mechanical transducers, as when integrating the output of an accelerometer to obtain velocity.

The integral math waveform, derived from the sampled waveform, is computed based on the following equation:

$$y(n) = scale \sum_{i=1}^{n} \frac{x(i) + x(i-1)}{2} T$$

Where: x(i) is the source waveform

y(n) is a point in the integral math waveform

scale is the output scale factor

T is the time between samples

Since the resultant math waveform is an integral waveform, its vertical scale is in volt-seconds (its horizontal scale is in seconds). The source signal is integrated over its entire record length; therefore, the math waveform record length equals that of the source waveform.

Offset and Position. When creating integrated math waveforms from live channel waveforms, consider the following:

- You should scale and position the source waveform so that it is contained on screen. (Off screen waveforms may be clipped, which will result in errors in the integral waveform).

- You can use vertical position and vertical offset to position your source waveform. The vertical position and vertical offset will not affect your integral waveform unless you position the source waveform off screen so that it is clipped.

DC Offset. The source waveforms that you connect to the instrument often have a DC offset component. The instrument integrates this offset along with the time-varying portions of your waveform. Even a few divisions of offset in the source waveform may be enough to ensure that the integral waveform saturates (clips), especially with long record lengths.

Defining Spectral Math Waveforms

The math capabilities of the instrument include waveform spectral analysis. This section describes how you can control the analysis intuitively with time domain and frequency domain controls. These controls merge the time domain controls with the frequency domain controls to provide a complete spectral analyzer.

Signals may be represented by their characteristics in both the time and the frequency domain. By combining and transforming source waveforms into spectral math waveforms, you can simultaneously view signal characteristics in both domains.

This spectral analyzer provides a complete set of controls and features that allow you to make time and frequency domain measurements without the need to learn extensive details about FFT algorithms.

- **Frequency Domain Controls**: Use traditional spectrum analyzer controls to set the center frequency, span, and resolution bandwidth directly.

- **Time Domain Controls**: The time domain controls for the acquired waveform set the time duration and the resolution (time between samples). You can easily set the required sample rate and record length.

- **Gating Controls**: These controls are the bridge that connects the time domain to the frequency domain. You can perform spectral analysis on a gated region of the input waveform. This gating also determines the resolution bandwidth.

- **Window Functions**: Eight different window functions are available to shape the filter response.

- **Magnitude Versus Frequency**: Display log data in dB, dBm, or linear mode. You can display the real or imaginary parts of the spectral magnitude only. Reference level offset and reference level controls give complete control over the vertical position and offset of the spectrum.

- **Phase Versus Frequency**: Display phase data as a function of frequency in radians or degrees. You may zero the noise phase for magnitudes below a threshold level. Finally, you may select Phase unwrap and dq/dw, group delay.

- **Spectral Averaging**: You can turn on averaging in the frequency domain for phase and magnitude waveforms.

- **Multiple Analyzer Control Locks**: Up to four spectral analyzers can be used simultaneously. They can all be assigned to different gates on the same source waveform or to different channel sources. The controls of Math1 and Math2 can be locked, and the controls of Math3 and Math4 can be locked. When controls are locked, turning a control on one analyzer changes the control on the other analyzer to the same value. Other combinations of locking, including all four analyzers, are available using GPIB commands.

The same exclusions for math waveforms apply to spectral math waveforms. In addition, sources for spectral math waveforms must be channel waveforms.

Using the Spectral Math Controls

The spectral analyzer contains five primary control categories. These are shown in the table below. Click one of the links for more information on using each type of control.

Time controls	Gate controls	Frequency controls	Magnitude controls	Phase controls
Source	Position	Center	dB, dBm linear, real imaginary	degrees, radians, group delay
Duration, record length	Duration	Span	Ref level	Zero threshold
Duration, sample rate	Window	Resolution bandwidth	Ref level offset	Phase Unwrap
Resolution				

Using the Time Controls

The operation of the time domain controls for the spectral analyzer is summarized by the following rules:

- Duration selects the time from the beginning to the end of the acquired waveform. You may set duration using the record length control or the sample rate control.

- Resolution determines the time between samples. Duration is kept constant as resolution is changed. Therefore, the Resolution control affects both the sample rate and the record length simultaneously.

- Most often, you will want to use a short record length because long record lengths can slow instrument response. However, long record lengths lower the noise relative to the signal and increase the frequency resolution for the spectral math waveform. More importantly, they might be needed to capture the waveform feature that you want to include in the waveform.

Examples of how duration and resolution affect the acquired waveform are shown in the following figure.
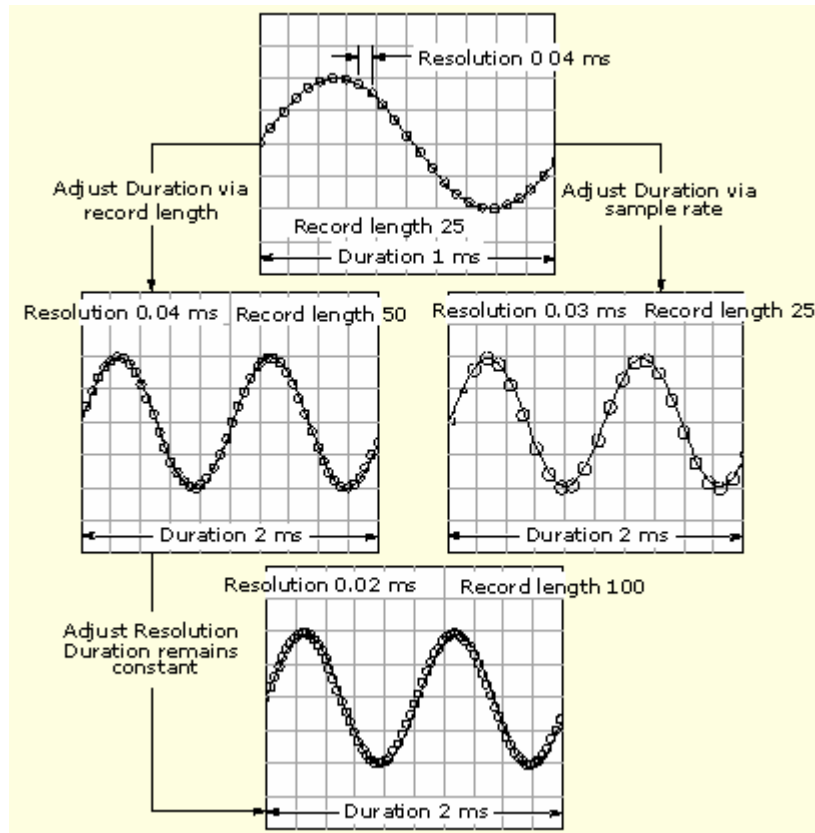
Using the Gating Controls

Gating determines which portion of the acquired waveform is transformed into the frequency domain. The gate has a position and a width control.

The gate position is the time in seconds from the trigger location to the center 50% position of the gate interval (see the figure below). The position and width units are seconds.
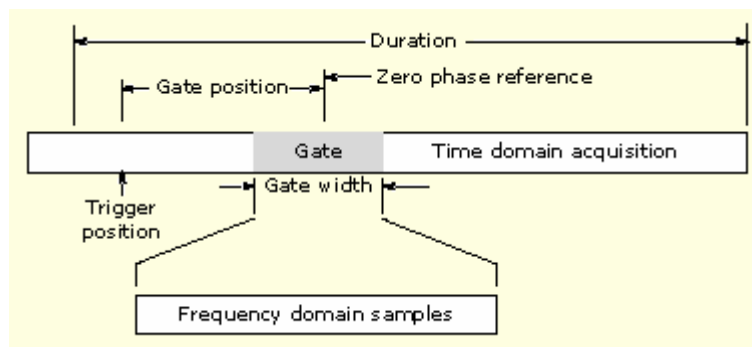
The gate must reside within the duration interval of the source waveform. If the source waveform duration is adjusted, and the gate position and width would result in the gate being outside of that duration, then the gate position or width is set within the limits.

The width of the gate affects the resolution bandwidth of the spectral analyzer. Gate width has units of seconds. The resolution bandwidth directly controls the gate width, but the numerical value is entered in units of Hz. Therefore, the time domain gate markers move as you adjust the resolution bandwidth control.

The data contained in the gated region is transformed into the frequency domain.

The gate is identified on the display using dashed markers (similar to cursors). The default gate width setting is equal to the duration of the source waveform.

Using the Frequency Domain Controls

The gated region of the source waveform is transformed into a spectral phase or magnitude waveform. The horizontal units are always Hz. The vertical units depend on whether phase or magnitude is selected. The frequency domain controls for the spectral waveform are span, center, and resolution bandwidth. The spectrum normally appears on the display fit to a screen width of 10 divisions.

Span is the frequency at the end of the spectral waveform minus the frequency at the beginning of the waveform. The span control maximum value is equal to the current sample rate divided by two. Therefore, if you are unable to increase the span to the desired value and if you want to keep the same source waveform duration, go to the time base controls and increase the sample rate using the resolution control. Or, if you want to also decrease the source waveform duration, adjust the sample rate control. If you decrease the sample rate, the span setting may decrease, if necessary, to keep the span less than the sample rate divided by two.

Center is the frequency at the center of the spectral waveform. Center is equal to the start frequency plus one half of the span. The adjustment range depends on the sample rate and the current span setting. Remember that the span must always be in the interval of zero to one half of the sample rate. Therefore, if the span is set to full, the center frequency control will not have any adjustment range and will remain at one half of the sample rate. It is necessary to decrease the span before the center will have an adjustment range. If you are unable to increase the center to the desired value, increase the sample rate using either the sample rate or resolution controls. You may also change sample rate by using the HORIZONTAL SCALE knob on the instrument front panel.

Resolution Bandwidth (RBW) is the 3 dB down bandwidth of the spectral analyzer frequency response to a sine wave input. The resolution bandwidth is affected by two parameters:

- Different window functions produce different filter response shapes in the spectrum and result in different resolution bandwidths.

- The gate width of the input data affects the RBW because the RBW is inversely proportional to the gate width. So, adjusting the gate width changes the RBW. Gate width has units of seconds. The RBW directly controls the gate width, but the numerical value is entered in units of Hz. Therefore, the time domain gate markers move as you adjust the RBW control.

RBW = Window Bin Width ÷ Gate Width

The Window Bin Width is the resolution bandwidth in units of bins, and depends on the window function in use. The Gate Width is in units of seconds.

The following figure demonstrates the effects of adjusting the center frequency and span. Center frequency is a horizontal position control for the spectrum. Span is a horizontal scale control. Resolution bandwidth adjusts the bandwidth of the analyzer filters without affecting the span and center frequency.

Center frequency is 1.0 and span is 0.5. Gate width= 200

Increase the center frequency.

Decrease the center frequency.

Set center frequency back to 1 and decrease the Span.

Decrease the Span again.

Increase resolution by reducing Resolution BW (increasing the gate length).

Increase resolution again by reducing the Resolution BW (doubling the gate length).

Start   Center frequency   Stop

Span

Using the Magnitude Controls

Vertical units can be either linear or log. You may select these choices by clicking the Math menu button. Then select the Spectral Analysis Setup command. On the Mag tab, select the desired scale type from Linear, dB, or dBm.

Linear: When the spectrum is linear magnitude the vertical units are the same as the source waveform. Usually this is volts. However, it may also be watts or amperes.

dB: This sets the vertical scale of the magnitude spectrum to dB. Use the Reference Level Offset to set which vertical position in the magnitude spectrum will be zero dB. The following equation applies:

$$dB = 20 \log (|X| \div Ref)$$

Where X is a complex data point in the spectrum and Ref is the Reference Level Offset value.

dBm: This selects dB as described in the above equation, but also sets the Reference Level Offset to a value that is equivalent to 1 mW of power into 50 ohms or volts. Therefore, if the input units are volts, then the value is set to 223.6 mV. If the input units are amperes, then the value is set to 40 mA. If the input units are watts, then the value is set to 1 mW.

Reference Level: This sets the vertical position of the displayed spectrum. Its value is the magnitude at the top of the display screen. When this control is adjusted, the spectral waveform along with its zero reference marker move vertically on the screen as shown. This control does not change the spectral data.
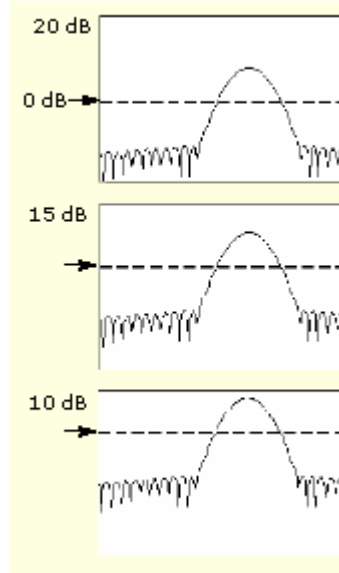
Reference Level Offset: This changes the value of Ref in the equation for dB shown above. Unlike the Reference Level control, this control actually changes the output data values in the spectrum. Zero dB is shown on the display screen by the marker associated with the spectral waveform. Adjusting the Reference Level Offset causes the spectral waveform to move vertically with respect to the waveform reference marker as shown. This moves the waveform without changing the Reference Level control setting. Sometimes it is beneficial to adjust this control so that the peak of a fundamental is at zero dB. Then you can measure other harmonics in terms of how many dB they are away from the fundamental. Click the dBm button to preset this level to the equivalent of 1 mW into 50 ohms.
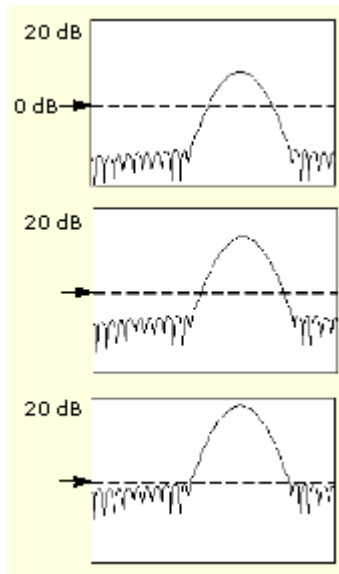
Real and Imaginary Magnitudes: You can display the linear magnitude of the real data or the imaginary data in the spectrum. This is useful if you process the spectrum off line and transform it back into a time domain trace. You can save the real and the imaginary spectrum into reference memory, and export the waveforms directly into Mathcad, MATLAB, or Excel documents to update in real time.

To turn on a real or imaginary spectrum, click the Math menu button, select Define/Edit Expression Editor, and select the Freq tab. Click either the Real or the Imag buttons and enter an expression. Click the appropriate channel button, and click Apply.

Using the Phase Controls

You can set the vertical units to Degrees, Radians, or Group Delay in seconds. To do this, click the Math toolbar button, click Spectral Analysis Setup, and select the Phase tab. Then select the desired scale type from Degrees, Radians, or Group Delay.

Phase Reference Position: Phase is a relative measurement that must have a time domain reference point. The phase value is specified with respect to this phase reference position.

For the spectral analyzer, the phase reference position is the 50% position of the gate, that is, the middle of the gate interval of the data that is input to the spectral analyzer. This is true for all window functions except for the Tek Exponential window. This window has a reference point at the 20% position of the gate.

Phase Unwrap: The spectral analyzer produces phase values from –Π to +Π radians or –180 to 180 degrees. However, when you perform impulse response testing and the phase is continuous, then phase values outside these ranges may occur. The spectral analyzer then wraps the data with discontinuities in the display from +180 to –180 degrees. Phase unwrap will display the correct result by unwrapping the phase.

Phase unwrap is only valid when the phase spectrum is a continuous function of frequency. Therefore, do not use it when analyzing the harmonic content of the typical repetitive signal.

Suppression Threshold: Random noise in the spectrum may have phase values over the entire range. This could make the phase display unusable. However, you can set the suppression threshold control to a level in dB. The phase of any complex spectral points with a magnitude below this threshold is set to zero. See the following figure for an example.
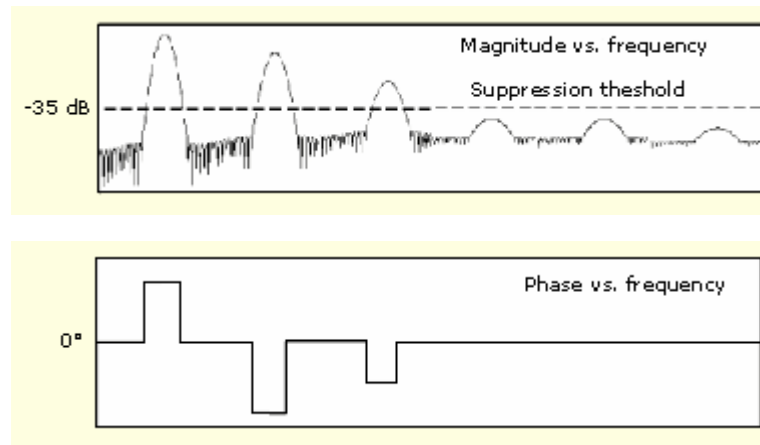
Group Delay: When the phase spectrum is a continuous function of frequency, group delay may be computed. This is true of impulse response testing where an impulse is fed into the system, and the spectrum of the response of the system output is computed.

Group delay measures how well a system passes a signal in terms of phase distortion. Group delay is the derivative of the phase with respect to frequency.

This feature is not useful for analysis of harmonic content of signals where the phase response is not continuous.

Impulse Response Testing: When performing impulse response testing of a system, place the impulse at the zero-phase reference position of the acquisition. This produces a correct phase display. Because the Tek Exponential window has its zero phase reference position at the 20% point, more of the impulse response is captured. All other window functions have their phase reference position at the 50% position in the gated region.

There are several ways to adjust the position of the zero phase reference point with respect to your input signal:

- Adjust the spectral analyzer gate position
- Perform fine adjustment using the front-panel trigger level control
- Adjust the front-panel horizontal POSITION control

Using Spectral Analyzer Windows

In the time domain, a window is a bell-shaped function equal in length to the gate duration. For most windows, this function tapers to zero at both ends of the gate region. Before computation of the spectral transformation, the window is multiplied, sample by sample, times the input data in the gated region. The window function affects the shape of the spectral analyzer response in the frequency domain.

The window functions affect the ability to resolve frequency in the output spectrum and can affect the accuracy of the magnitude and phase measurements. The preceding figure shows how the time domain record is processed.
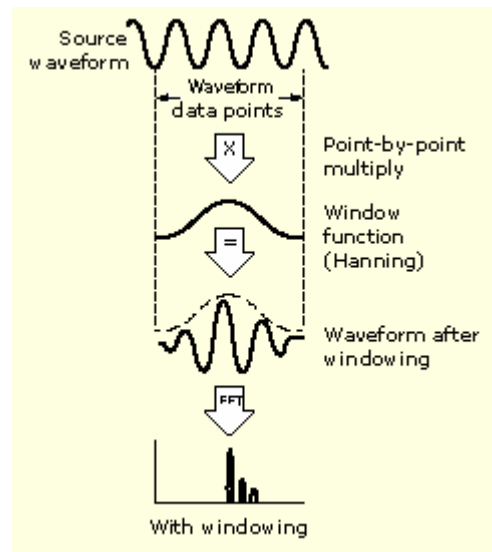
Accurate magnitude measurements require that the input source waveform be stationary within the gated region. This means that waveform parameters such as frequency and amplitude should not change significantly as a function of time within the gate region that is input to the spectral analyzer. Also, the gate width must be greater than or equal to the period of the start frequency of the span of the spectral analyzer. That is, there must be at least one cycle of the harmonic being measured within the gated region.

There are eight different spectral analyzer windows:

Rectangular	Hamming	Hanning	Kaiser-Bessel
Gaussian	Blackman-Harris	Flattop2	TekExponential

Your choice of window function will depend on the input source characteristics that you want to observe and the characteristics of the window function. The window characteristics are shown in the following table.

Window	3 dB BW in bins	Scallop loss	Nearest side lobe	Zero phase reference	Coefficients
Rectangular	0.89	3.96 dB	-13 dB	50%	1.0
Hamming	1.3	1.78 dB	-43 dB	50%	0.543478, 0.456522
Hanning	1.44	1.42 dB	-32 dB	50%	0.5, 0.5
Kaiser–Bessel	1.72	1.02 dB	-69 dB	50%	0.40243, 0.49804, 0.09831, 0.00122
Blackman–Harris	1.92	0.81 dB	-92 dB	50%	0.35875, 0.48829, 0.14128, 0.01168
Gaussian	2.0	0.76 dB	-79 dB	50%	a = 3.75 (not cosine series)
Flattop2	3.8	0.0065 dB	-90 dB	50%	0.213348, -0.206985, 0.139512, -0.043084, 0.003745
Tek Exponential	1.42	0.60 dB	-67 dB	20%	na

3 dB BW in Bins: This is the bandwidth in units of bins, which are the intervals between spectral output samples when no zero fill is used. The bandwidth is measured between the points on the lobe that are 3 dB down from the peak of the lobe. The bandwidth in hertz may be computed by dividing the BW in bins by the gate duration in seconds. This is also referred to as resolution bandwidth (RBW).

Coherent gain: The gain factor normally associated with different window functions is correctly scaled into the magnitude spectrum output. Therefore, the magnitudes in the output spectrum do not change as different windows are selected.

Scallop Loss: This is the magnitude error of the spectral analyzer when the frequency of the observed signal is exactly half way between two frequency samples of the spectrum when the interpolation ratio due to zero fill of the FFT is one. When zero fill is in effect, scallop loss is essentially eliminated because of the interpolation in the frequency domain due to zero fill. If you work with span settings less than full and you work with larger resolution bandwidth settings, zero fill is in effect most of the time.
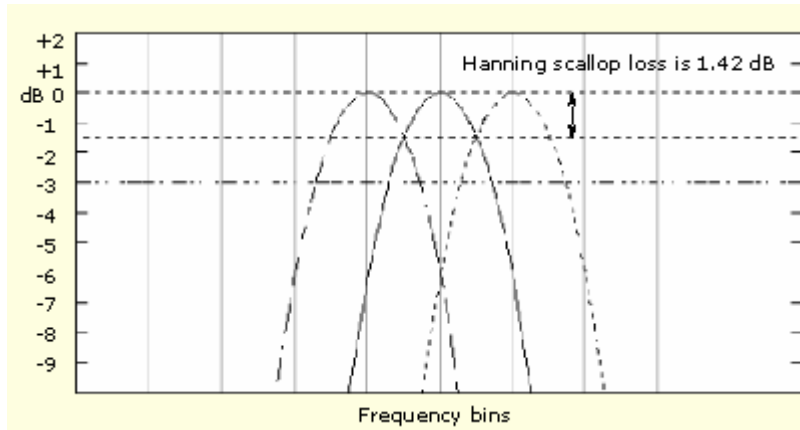
Nearest Side Lobe: This is the difference in magnitude between the spectral lobe peak in the spectrum and the next side lobe that occurs due to energy leakage. Different windows have different leakage characteristics. The more narrow the resolution bandwidth of the window the more leakage in the spectrum.

Zero Phase Reference: This is the position in the time domain gate that is the reference point for phase in the output spectrum. That is, if a sine wave input has its peak at the zero phase reference position, then it reads out as zero phase in the spectrum. If the phase is to be correct when doing impulse response testing, the impulse in the time domain must be located at this position in the gate interval.

Coefficients: These are used to generate the windows that are constructed from a cosine series. For the Gaussian window, the value of "a" is given instead of a set of coefficients. You can find descriptions of cosine series windows in the *Handbook of Digital Signal Processing Engineering Applications* by Elliot. ISBN 0-12-237075-9.

Gaussian Window

This is the default window function (see the figure below). It is unique in that the time-domain shape of an exponential Gaussian function transforms into a Gaussian exponential shape in the frequency domain. This window provides optimal localization in both the time and the frequency domain
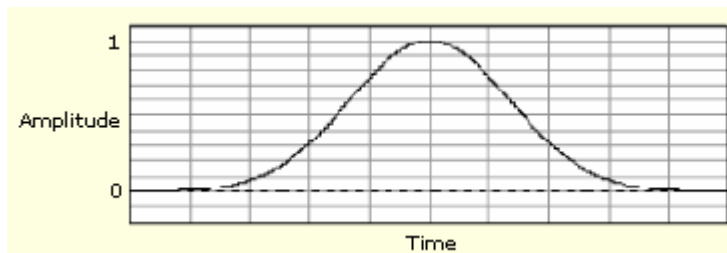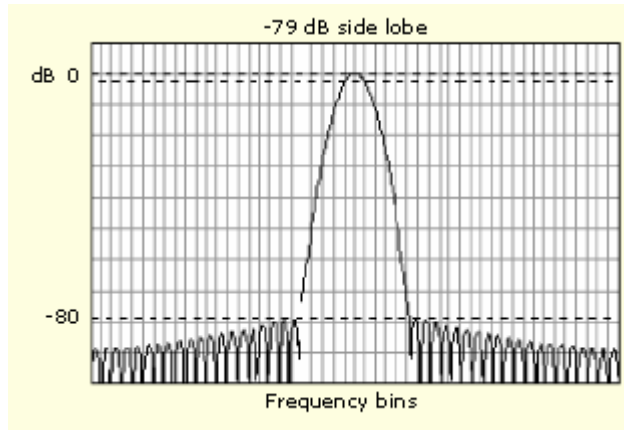
Rectangular Window

This window is equal to unity (see the figure below). This means the data samples in the gate are not modified before input to the spectral analyzer. This window has the narrowest resolution bandwidth of any of the windows, but it also has the most spectral leakage and the highest side lobes.
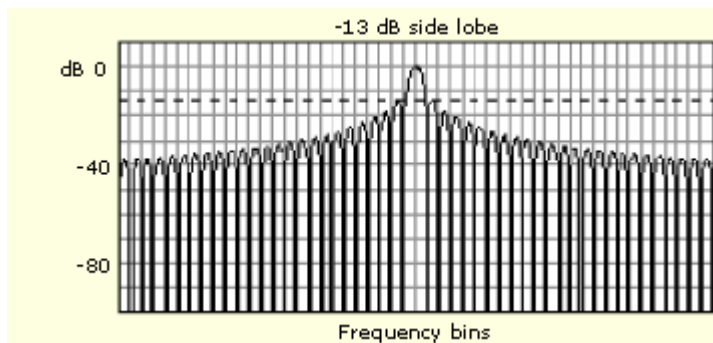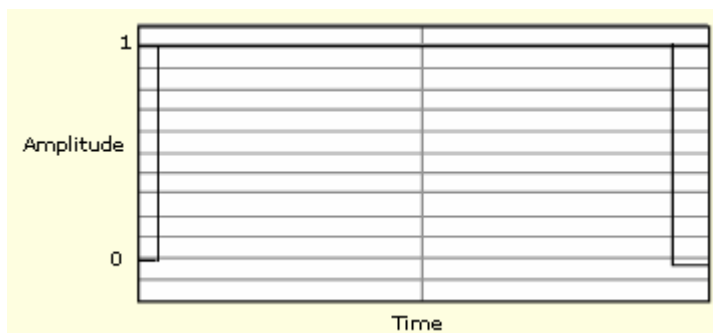
Hamming Window

This window is unique in that the time domain shape does not taper all the way to zero at the ends. This makes it a good choice if you wanted to process the real and imaginary parts of the spectrum off line and inverse transform it back to the time domain. Because the data does not taper to zero, you could remove the effect of the window function from the result.
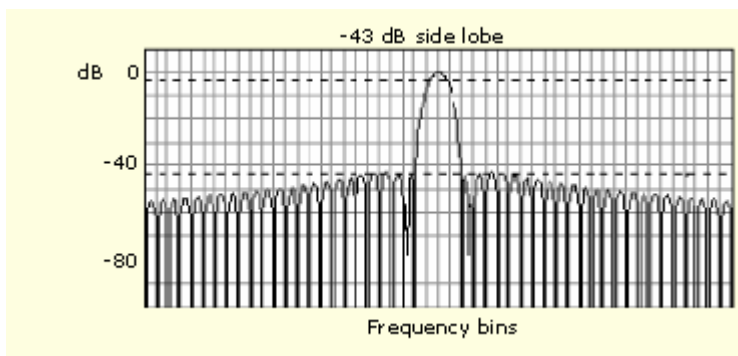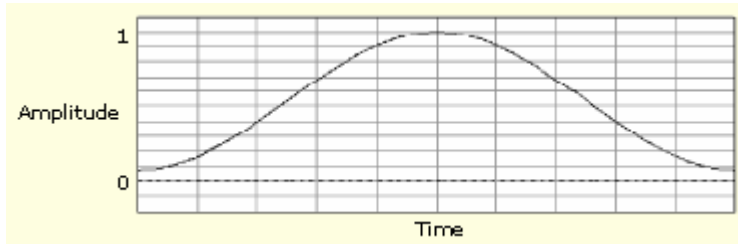
Hanning, Kaiser-Bessel, and Blackman-Harris Windows

These windows have various resolution bandwidths and scallop losses (see the figures below). Choose the one that best allows you to view the signal characteristics you are interested in. The Blackman-Harris has a low amount of energy leakage compared to the other windows. The Hanning has the narrowest resolution bandwidth, but higher side lobes.
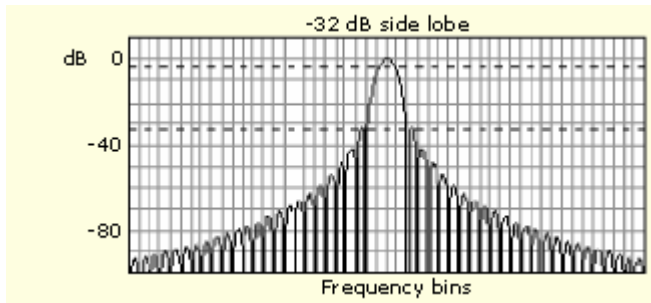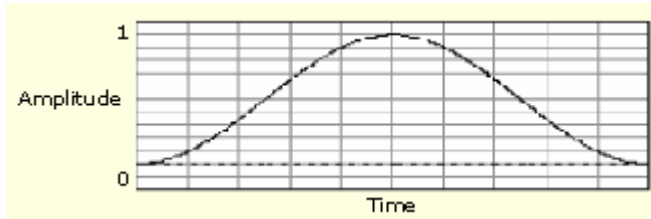
Hanning window
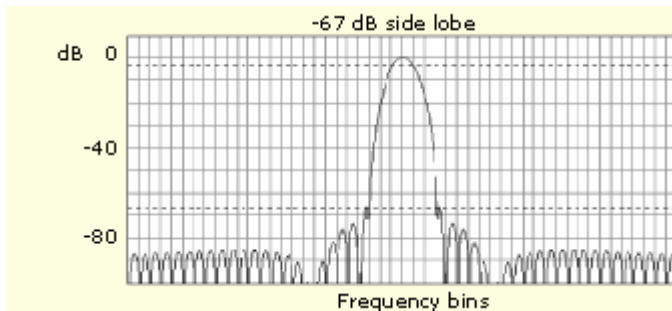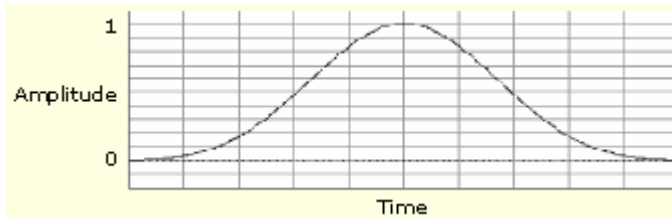
Kaiser-Bessel window
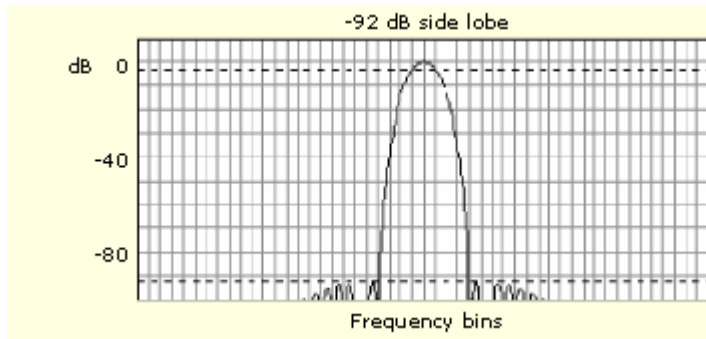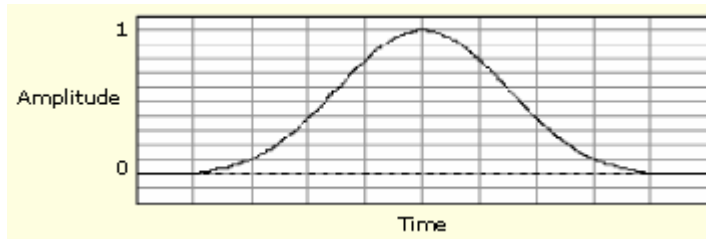
Blackman-Harris window

Flattop2 Window

This window has the lowest scallop loss of any of the windows. It also has a wider resolution bandwidth but lower side lobe attenuation. Also, it is unique because the time domain shape has negative values. The Flattop2 window is useful for high accuracy magnitude measurements for signals that do not require very narrow bandwidth.
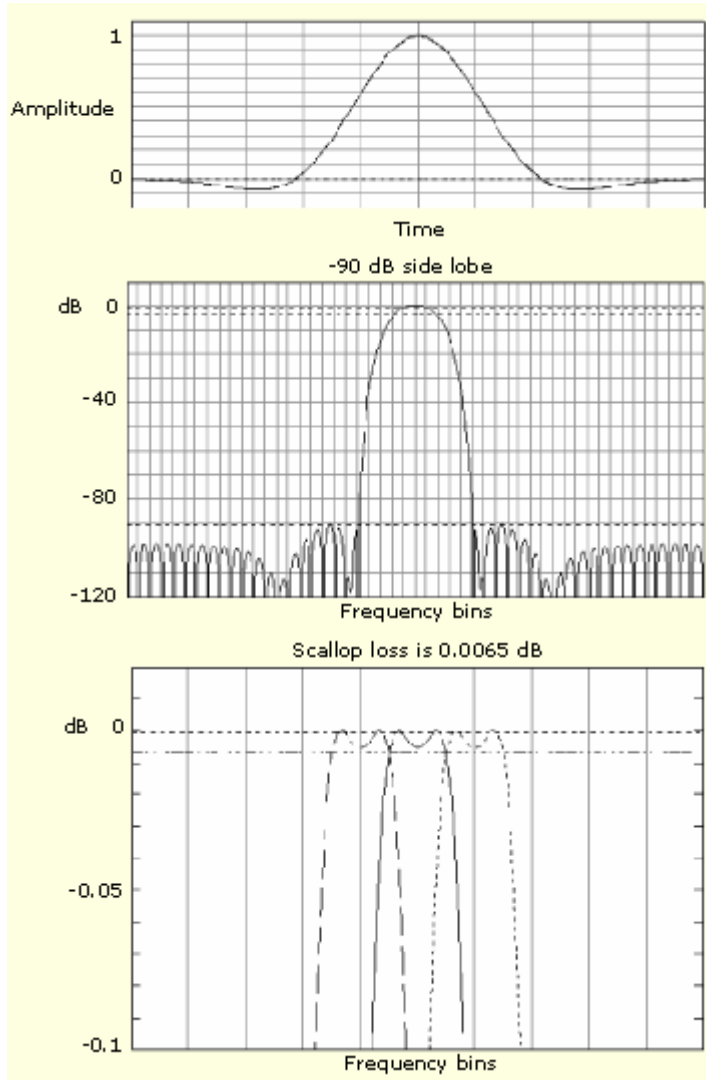
Tek Exponential Window

The Tek Exponential window was invented at Tektronix. In the time domain, it is not a symmetrical bell shape as is the case with the other windows. Instead, it is exponential with a peak at the 20% position of the time domain gate. The frequency domain shape is triangular. Use this window for impulse-response testing where the 20% position is the zero phase reference point. More of the acquired data record length is used to capture the impulse response. Exact details of how to compute its values were published in the article; *Impulse-response testing lets a single test do the work of thousands* by John Pickerd in EDN magazine, April 27, 1995, page 95.
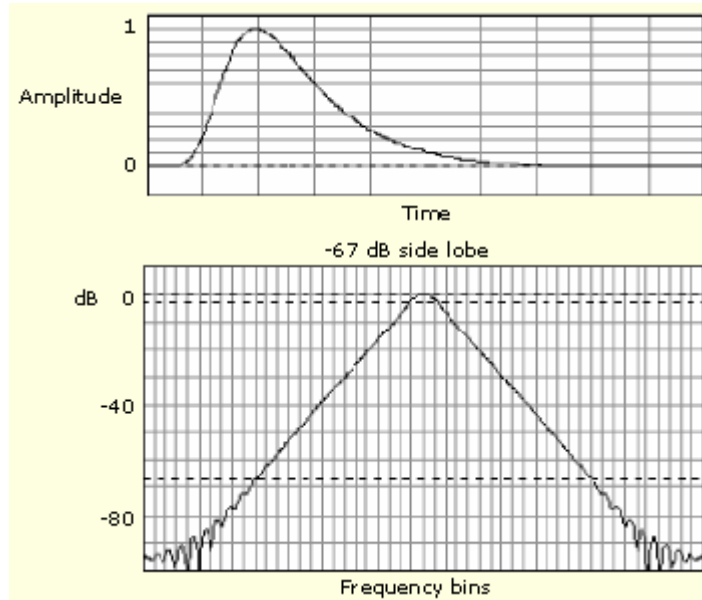
Effects of Trigger Jitter

The instrument acquisition system has a sample clock that is asynchronous with respect to the input signal. This means that from one acquisition to the next, samples may be in a different position on the waveform with respect to the trigger. Samples may vary in position by up to one sample interval.

There are only two samples per cycle of a signal that has a frequency equal to one half of the sample rate. This is the highest nonaliased signal that can be output from the spectral analyzer. Thus, at this frequency, one sample of acquisition jitter will show up in the spectrum as 180 degree phase variations. Therefore, when you analyze phase, make sure to oversample the signal sufficiently such that the one sample interval of jitter that is showing up in the phase is acceptable to the accuracy you want to achieve.

Effects of Average and High Res acquisition modes

The result of averaging the time domain acquisition using either Average mode or Hi Res acquisition mode affects the frequency response of the instrument. This is due to the one sample of jitter in the acquisition system. Both Hi Res and Average acquisition modes have the same affect on the frequency response. These modes cause the response to roll off from a magnitude value of one at DC to a magnitude value of 0.63 at Nyquist, which is the frequency equal to one half of the sample rate. This is true regardless of the real time sample rate setting.

Frequency Domain Averaging

Turn on averaging for a math waveform by editing the math expression. Sometimes, it is more desirable to average in the spectrum than in the time domain. For example, consider a signal that has time domain components that are asynchronous to the trigger. If you turn on averaging in the time domain, these components may go to zero or produce strange, nondeterministic effects on the resultant waveform. Thus, these signal components may end up not appearing in the spectrum. However, if averaging is done in the frequency domain instead, then these components will be present.

Recognizing Aliasing

Aliasing occurs when the input frequency of a signal is greater than one half of the sampling frequency (the sample rate).

Set the sample rate high enough so that the signals in the spectrum appear at their correct frequency as opposed to a lower aliased frequency value. Also, complex signal shapes that have many harmonics in them, such as a triangle or square wave, can appear to be OK in the time domain when in fact many of the harmonics in that signal are aliased.

One way to check for aliasing is to increase the sample rate and observe whether any of the harmonics unwrap to different frequency locations.

Another way to recognize aliasing is to realize that higher order harmonics usually have decreasing magnitudes compared to lower order harmonics. Thus, if you see a series of increasing harmonic magnitude values as frequency increases then you can suspect that they may be aliased. In the spectral math waveform, the actual higher frequency components are undersampled, and therefore they appear as lower frequency aliases that "fold back" around the Nyquist point. (See figure below.) You may test by increasing the sample rate and observing if aliases unwrap to different frequency positions.
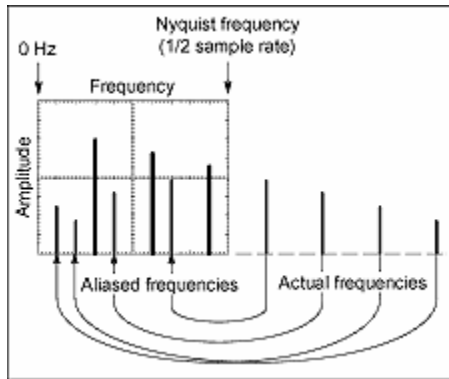
If you have a variable-frequency signal source, another way to observe aliasing is to adjust the frequency slowly while watching the spectral display. If some of the harmonics are aliased, you will see the harmonics decreasing in frequency when they should be increasing or vice versa.

Once you have displayed a spectral math waveform, use cursors to measure its frequency amplitude or phase angle. See Setting Cursor Sources on page 324 for detailed steps.

You can use automated measurements to measure spectral math waveforms. See Taking Automatic Measurements on page 321 for step-by-step instructions.

Serial Mask Testing with Option SM

Serial Mask Testing (Option SM) provides electrical mask testing on communication standards and communication triggering.

Mask testing consists of two tasks: signal violation detection and pass/fail testing. Signal violation detection lets you test communications signals for time or amplitude violations against a predefined mask. Each mask consists of one or more polygonal regions called segments. The signal waveform data should stay outside of the segments defined by the mask. Any signal data that occurs inside a mask segment is called a mask segment violation or "hit."

You can select from any of the included standard telecommunications masks, or you can define your own custom masks. Selecting a mask automatically sets the instrument communications triggers to properly display most communication signals in the mask.

Pass/Fail testing defines the mask testing parameters, including the number of waveforms to test, how many mask hits are allowed before failing a test, setting a mask margin tolerance value, and what action to perform upon the completion or failure of a test.

Communication triggering enables you to trigger on and display waveforms for industry-standard communications signals. See Mask Types and Standards on page 168 for a list of the supported standards on which you can trigger.

Automatic communication signal measurements enable you to make automatic measurements on communications signals. See the Communication Measurement on page 160 tab in the Measurement setup window for the available measurements.

The Serial Mask Testing key features are:

- Predefined masks for testing or triggering on industry-standard signals, such as ITU-T G.703, ANSI T1.102, Fibre Channel, Ethernet, Video, SONET, Rapid IO, USB, IEEE 1394b, OIF Standards and their subsets

- Autoset, which quickly adjusts the instrument's vertical and horizontal controls to display a waveform in a mask

- Autofit, which positions the signal on each acquisition to minimize mask segment hits

- Mask margins, which allow you to adjust the default mask margin tolerances

- Pass/Fail testing to continuously test a specified number of waveforms against a mask

- A mask editor for creating, saving, and recalling user-defined masks

- Waveform database technology to perform mask testing based on waveforms accumulated in a database, rather than a single waveform stored in acquisition memory

- Communications triggers to trigger the instrument on industry-standard communications signals

- Automatic measurements on communications signals

Note

If a standard listed in this manual is not available on your instrument, it is because the configuration or bandwidth of your instrument cannot test that standard.

Although the TDS5000B Series instruments are not calibrated optical reference receivers, you can use them with mask testing to evaluate general optical signal characteristics and wave shape, using an external O/E converter.

Mask Key Points

There are a number of mask test key points to be aware of prior to using, editing, or creating a mask.

Mask Testing

Only one mask standard is active at any one time. If you have one mask selected or enabled and then select a new mask, the new mask replaces the previous mask. You cannot test to multiple standards simultaneously.

Autofit and Persistence Interaction

The Autofit function moves the waveform vertically and horizontally in a mask to reduce the number of segment hits within a mask. If persistence is set to infinite or variable, each Autofit waveform movement clears existing persistence data. If Autofit makes frequent waveform movements, there may be little or no displayed waveform persistence data.

Segments and Mask Hits

Each mask can have a maximum of 16 segments. Segments can overlap. The number of mask hits is the sum of all hits in all segments, regardless of whether or not segments overlap. For example, if a waveform crosses over an area where two segments overlap, both segments will count the waveform hit.

Vertices

Each segment can have a maximum of 50 vertices. Vertices are numbered counterclockwise, with vertex one generally located at the bottom left of each segment. The active (selected) vertex is indicated by an X. The instrument automatically assigns numbers to vertices during mask creation or editing.

Mask Margin Tolerance

Mask margin tolerance moves the mask segment boundaries by the specified percentage. Negative margins reduce the size of the segment, making it easier to pass a mask test. Positive numbers make it harder to pass and allow you to perform margin testing on your design. If a user defined mask has more than three segments, turning on mask margins generates an error message. Turning mask margin tolerance off redraws the mask segment margins to their default values, but leaves the numeric value as it is, allowing you to quickly toggle between default and user-set margin values.

Standards and Bandwidth

When the instrument system bandwidth (which includes the instrument, attached probes, and/or cabling) falls into the range of 1.5 to 1.8 (0.8 for optical signals) times the data signal bit rate, the third harmonic of the data signal is significantly attenuated. The instrument displays useful qualitative information, but quantitative rise-time measurements under these conditions may not be accurate.

For example, a FC531E Elec mask has a bit rate of 531.25 Mb/s. 1.5 to 1.8 times this value is a range of 796.5 MHzBW to 955.8 MHzBW. Therefore, you should not use an instrument with less than 1 GHz measurement system for making quantitative rise-time measurements of this standard.

When just the instrument bandwidth falls within 1.5-1.8 (0.8 for optical signals) times the bit rate of a selected mask standard, the instrument displays the message "Consider system bandwidth when testing at this bit rate." in the status area above the graticule.

Levels Used in Taking Eye Measurements

All eye-diagram measurements are based on the power level, the voltage level, or the time locations of edges within each acquisition.

The figure below shows an eye-diagram and the areas from which values are taken that are used to calculate measurements.
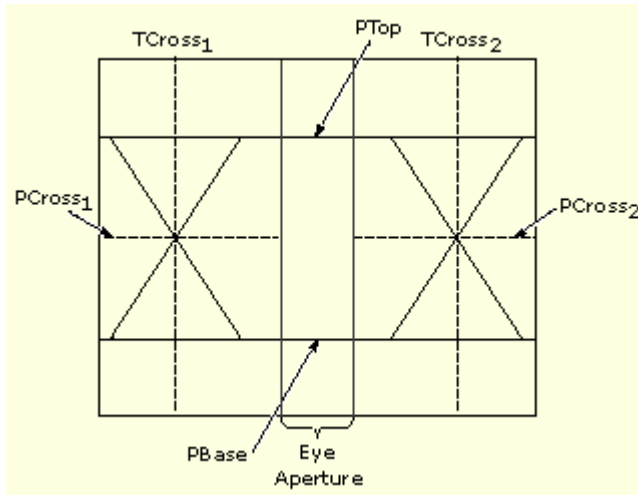
P Values

The P values include the mean and standard deviation of the vertical location of PTop and PBase. These areas are used with a specified sample size to statistically measure the following values:

- $PTop_{mean}$, the mean value of PTop
- $PTop_{sigma}$, the standard deviation of PTop
- $PBase_{mean}$, the mean value of PBase within the Eye Aperture[*]
- $PBase_{sigma}$, the standard deviation of PBase within the Eye Aperture[*]

[*]**The Eye Aperture defaults to the center 20 % of the interval from TCross1 to TCross2.**

T1 Values

The T1 values are vertical and horizontal values associated with the leftmost crossing point. These areas are used to establish the following values:

- $TCross1_{mean}$, the horizontal mean of the left crossing point at TCross1
- $TCross1_{sigma}$, the horizontal standard deviation of the left crossing point at TCross1
- $TCross1_{pk-pk}$, the horizontal peak-to-peak deviation of the left crossing point at TCross1
- $PCross1mean$, the vertical mean of the left crossing point at PCross1

T2 Values

The T2 values are vertical and horizontal values associated with the rightmost crossing point. These areas are used to establish the following directions:

- $TCross2_{mean}$, the horizontal mean of the right crossing point at TCross2
- $TCross2_{sigma}$, the horizontal standard deviation of the right crossing point at TCross2
- $TCross2_{pk-pk}$, the horizontal peak-to-peak deviation of the right crossing point at TCross2

DCD Values

The duty cycle distortion (DCD) values are horizontal values associated with the rightmost crossing point at 50% of the eye height. These areas are used to establish the DCDpk-pk, the horizontal peak-to-peak deviation of the left crossing point at half the height of the eye.

Communication Triggering (Option)

Communication (Comm) triggers are one type of trigger. Comm triggers are designed to work with communication masks and standards. When you select a mask to test, it is associated with a specific communication standard. The mask you select determines the comm trigger needed for that standard.

Note

Option SM must be installed for communication triggering.

Eye diagrams that are set up with an external clock use a time base position value large enough to guarantee the edge is always before the acquisition record starts. This forces all comm triggers to disable the B-Trigger.

Even though comm triggers are designed to work with masks, you can set up comm triggering without using a mask. You can also set up your own mask to use with comm triggers.

Limit Testing Concepts

Limit testing is the ability to compare an active signal with a template waveform. The following figure shows an active waveform (dark sine wave) being compared to a template waveform (shaded area).
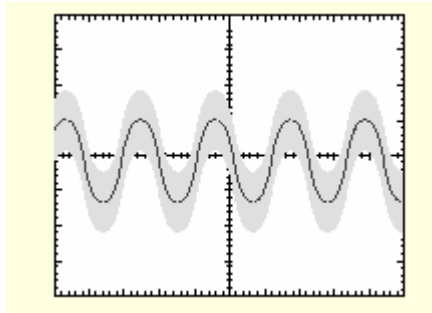
Any part of the active waveform that exceeds the template waveform's envelope is a limit test waveform violation, which the instrument highlights. You can set the instrument to respond to a limit test violation by stopping limit testing, emitting a beep, and so on. The following figure shows the highlighted (black) part of the active waveform that exceeds the limit test envelope.
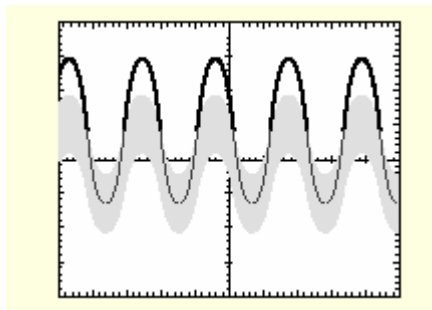
A template is an envelope waveform that consists of minimum/maximum (min/max) pairs of sample points. Template envelope waveforms created from 500 point source waveforms contain 250 min/max pairs; Template envelope waveforms created from10K point waveforms contain 5000 min/max pairs.

The following figure represents how the instrument creates the template waveform limit test envelope from user-entered vertical and horizontal division units. Division units are referenced to graticule divisions, where 1 major division contains 1000 millidivisions, or mdivs. The minimum mdiv unit is 20, which equals one screen pixel.
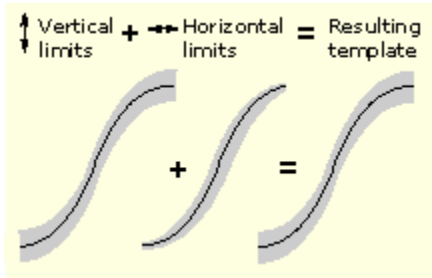
When a source waveform is compared with an envelope waveform, each source waveform sample point value is compared to the vertical and horizontal min/max values of the corresponding envelope waveform sample point. Any source waveform data point that is not equal to or within the corresponding template envelope min/max values is a violation.

Saving and Recalling Setups

This instrument can save a number of different instrument setups for later recall, limited only by the space that you have to store the setups.

By saving and recalling different setups, you can switch from setup to setup without having to first manually record your settings and then manually set them. This capability is helpful when you want to:

- Save and recall a setup that optimizes the instrument for displaying and analyzing a certain signal.
- Save a series of setups to help automate a procedure through recall of a sequence of saved setups as part of performing the procedure.
- Save a setup for sharing with a second instrument.

If you do not have an attached keyboard, you can still name setup files. The Save and Recall dialog boxes include a pop-up keyboard that you can use with your mouse or the touch screen to enter the setup path name and file name.

The instrument excludes the following items when saving setups:

- Channel waveforms and reference waveforms. Control settings (scale, position, and so on) are saved but not the waveform data. Upon recall of the setup, the settings are applied, but the data is not restored.
- Math waveforms. Control settings and the math expression are retained but not the waveform data. Upon setup recall, however, the math waveform expressions are applied, but math waveform data is not restored.
- User options stored in the Windows registry. These include all options set in the User Preferences control window.
- If you want to save a waveform in a useful format for other applications, such as a spreadsheet, use the save or save as function (see Saving and Copying Waveform Data on page 287).

A few other things to remember when saving and recalling setups:

- Recalling a setup replaces the current setup with the recalled setup. If you do not want to lose your current setup, save the setup file for later recall.
- Saved setups may contain settings inappropriate for waveforms currently in your instrument. For example, if you save a setup that displays a math waveform that is the average of reference waveform 1, when you recall the setup, if the reference is empty, the math and reference waveforms are not displayed.

Saving and Recalling Waveforms

This instrument can save any number of waveforms, limited only by the space that you have to store them.

By saving a waveform, you can recall it at a later time for comparison, evaluation, and documentation. This is helpful when you want to:

- Recall a waveform for further evaluation or comparison with other waveforms.
- Extend the waveform carrying capacity of the instrument. The instrument supports four reference, four channel, and four math waveforms. If you want more than four reference waveforms, you can save additional references to disk for recall later.

The Reference Setup control window contains a Label field for including comments with your saved waveforms.

If you do not have an attached keyboard, you can still enter comments and name reference waveform files. The Reference Setup control window includes a pop-up keyboard that you can use with your mouse or the touch screen to enter the waveform path name, file name, and comments.

The instrument recalls each waveform into one of the reference waveform locations (Ref1-Ref4). You cannot recall into a channel or a math waveform.

Saving and Copying Waveform Data

This instrument supports saving waveform data to a file. The instrument can save waveforms, images, and measurements in several formats. You can also copy waveform data to the Windows clipboard for use with other applications, such as spreadsheets or math analysis applications.

A few things to remember when saving waveform data:

- Waveform data are saved as a series of comma-separated values (CSV), which are amplitudes without units. There is no timing information, but data is placed in the file in sequence from the first sample in the waveform record to the last.
- Because the waveform data are saved as CSV, without timing and scaling information, the instrument does not save these waveforms directly. (It only saves the data.) If you intend to recall a waveform later, save it (see the procedure Saving A Waveform on page 344) using file type .wfm.
- You may also choose to copy a waveform and paste it directly into an application such as Microsoft Word or Excel. If so, select your waveform, and then select Copy in the Edit menu.

File Formats

To make saved files more useful, select a file format usable by your analysis tools:

Numeric creates files (.txt) in a numeric format usable by text and word processors.

Text creates files (.txt) in a text format usable by text and word processors.

Bitmap creates files (.bmp) in a bitmap file format usable by many graphic programs. (Available for screen captures.)

JPEG creates files (.jpg) in a compressed image format usable by many graphic programs. (Available for screen captures.)

Spreadsheet creates files (.CSV) in a format usable by spreadsheets (Excel, Lotus 1-2-3, and Quattro Pro).

MATLAB creates files (.DAT) in a format usable by MATLAB.

Mathcad creates files (.DAT) in a format usable by MathCad.

Note that the MathCad file is an ASCII file, the first four values of which contain header information:

- The first header value holds the record length.
- The second header value holds time, in seconds, between samples.
- The third header value holds the trigger position (expressed as an index in the data position).
- The fourth header value refers to the fractional trigger position.

Also note that the delimiters are carriage returns.

Spectral Analysis Overview

You can use math waveforms to perform FFT (Fast Fourier Transform) measurements with the instrument. The FFT process mathematically converts the standard time-domain signal (repetitive or single-shot acquisition) into its frequency components, providing spectral analysis capabilities.

The ability to quickly look at a signal's frequency components and spectrum shape is a powerful research and analysis tool. FFT is an excellent troubleshooting aid for:

- Testing the impulse response of filters and systems

- Measuring harmonic content and distortion in systems

- Identifying and locating noise and interference sources

- Analyzing vibration

- Analyzing harmonics in 50 and 60 Hz power lines

Spectral Analysis Features

- FFT Windows. FFT windows let you match the optimum window to the signal you are analyzing. The Rectangular window is best for nonperiodic events such as transients, pulses, and single-sequence acquisitions. The Hamming, Hanning, and Blackman-Harris windows are better for periodic signals. The Kaiser Bessel window has less spectral leakage than the Hanning, Hamming, or Rectangular windows. The Gaussian window provides optimal localization in both the time and frequency domain. The Flattop2 window is good for high accuracy magnitude measurements for signals that do not require narrow resolution bandwidth. The Tek Exponential window is useful for impulse response testing.

- Waveform Analysis. You can display an FFT waveform on any actively acquired signal (periodic or single-shot), the last acquired signal, or any signal stored in reference memory.

- Vertical Scales. The FFT vertical graticule can be set to either dB or Linear RMS. A dB scale is useful when the frequency component magnitudes cover a wide dynamic range, letting you show both lesser and greater magnitude frequency components on the same display. A Linear scale is useful when the frequency component magnitudes are all close in value, enabling direct comparison of their magnitudes.

- Spectral and Time-Domain Displays. The time signals and FFT waveforms can be shown together on the display. The time signal highlights the problem; the FFT waveform helps you determine the cause of the problem.

FFT Process

The FFT process mathematically converts the standard time-domain signal (repetitive or single-shot acquisition) into its frequency components.

The FFT function processes the waveform record and displays the FFT frequency domain record, which contains the input signal frequency components from DC (0 Hz) to ½ the sample frequency (also called the Nyquist frequency).
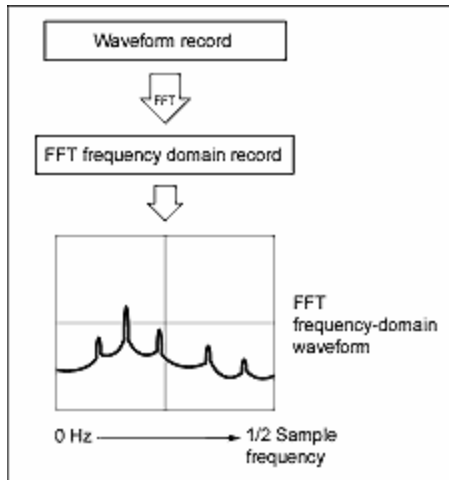
The FFT Display

The following illustration shows a typical time-domain waveform and the frequency components of that signal as part of the math FFT waveform.
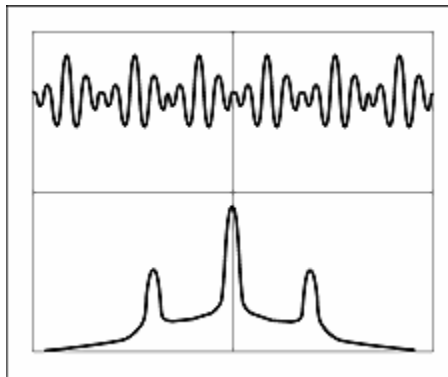
FFT Windows

The FFT process assumes that the part of the waveform record used for FFT analysis represents a repeating waveform that starts and ends at or near the same voltage of a cycle. In other words, it is an integer number of cycles. When a waveform starts and ends at the same amplitude, there are no artificial discontinuities in the signal shape, and both the frequency and amplitude information is accurate.

A nonintegral number of cycles in the waveform record causes the waveform start and end points to be at different amplitudes. The
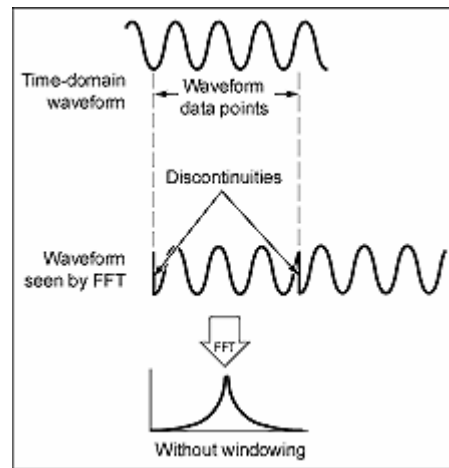
transitions between the start and end
points cause discontinuities in the
waveform that introduce high-
frequency transients. These transients
add false frequency information to
the frequency domain record.

Applying a window function to the
waveform record changes the
waveform so that the start and stop
values are close to each other,
reducing the discontinuities. This
results in an FFT measurement that
more accurately reflects the actual
signal frequency components. The
shape of the window determines how
well it resolves frequency or
magnitude information.
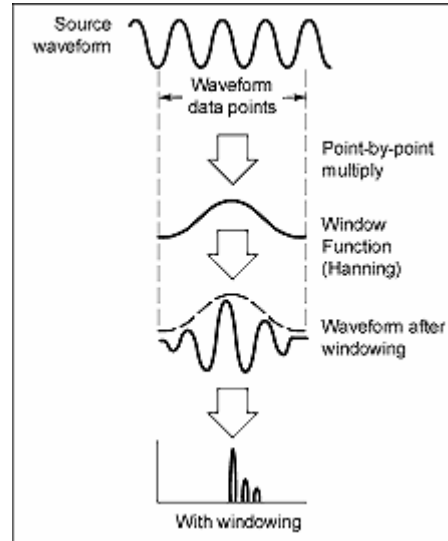
FFT Window Characteristics

Each FFT window is a trade-off between frequency resolution and magnitude accuracy. What you
want to measure and your source signal characteristics help determine which window to use. Use
the following guidelines to select the best window.

FFT Window	Characteristics	Best for measuring
Rectangular	Best frequency, worst magnitude resolution. This is essentially the same as no window.	Transients or bursts where the signal levels before and after the event are nearly equal. Equal-amplitude sine waves with frequencies that are very close. Broadband random noise with a relatively slow varying spectrum.
Hamming, Hanning	Better frequency, poorer magnitude accuracy than Rectangular. Hamming has slightly better frequency resolution than Hanning.	Sine, periodic, and narrow-band random noise. Transients or bursts where the signal levels before and after the event are significantly different.
Blackman-Harris	Best magnitude, worst frequency resolution.	Predominantly single frequency signals to look for higher order harmonics.

You can also determine the best window empirically by first selecting the Rectangular window,
and then selecting (in the following order) the Hamming, Hanning, and Blackman-Harris windows
until the frequency components merge. Use the window just prior to where the frequencies merge
for the best compromise between resolution and amplitude accuracy.

Time Domain Gating

You can use time domain gating to select a specific segment of your signal for spectral analysis. Two controls, gate position and gate duration, determine the location and size of the gate. The resulting spectrum corresponds to the frequency components of the signal in the gated segment.
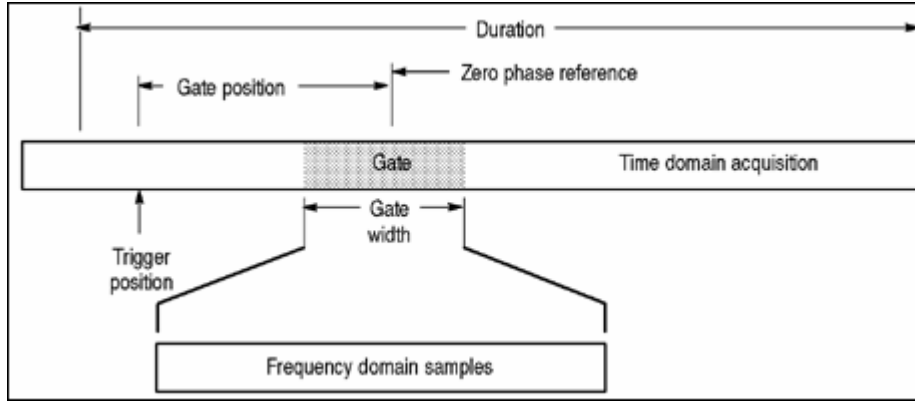
Time domain gating is inversely proportional to the resolution bandwidth of the spectral waveform. The narrower the gate, the larger the resolution bandwidth and therefore, the lower the frequency resolution. To obtain the best frequency resolution, use as wide a gate as possible.

Aliasing

Problems occur when the instrument acquires a signal containing frequency components that are higher in frequency than the Nyquist frequency. The frequency components that are above the Nyquist frequency are undersampled and appear to "fold back" around the Nyquist frequency, showing as lower frequency components. These incorrect components are called aliases.

To quickly check for aliasing, slowly increase the horizontal scale (time per division setting). If the shape of the displayed waveform changes drastically or becomes stable at a faster time base setting, your waveform was probably aliased. You can also check for aliasing by turning on peak detect acquisition mode.
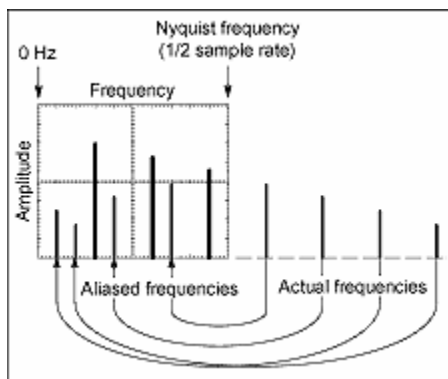
Eliminating Aliasing

Use the following methods to eliminate aliases:

- Increase the sample rate by adjusting the horizontal scale to a faster time-per-division setting. Since you increase the Nyquist frequency as you increase the time per division, the aliased frequency components should appear at their proper frequency. If the increased number of frequency components shown on the screen makes it difficult to measure individual components, use MultiView Zoom to magnify the FFT waveform.

- Use a filter on the source signal to limit the bandwidth of the signal to frequencies below that of the Nyquist frequency. If the frequency components that you want to view are below the built-in bandwidth settings, set the source channel bandwidth to the appropriate value.

- Push the AUTOSET button.

- Switch the acquisition to Envelope mode or Peak Detect mode. Envelope mode searches for samples with the highest and lowest values over multiple acquisitions and can detect faster signal components over time. In Peak Detect mode, the acquisition alternates between saving the highest sample in one acquisition interval and lowest sample in the next acquisition interval.

Nyquist frequency

The highest frequency that any digital oscilloscope can measure without errors is one-half of the sample rate or frequency. This frequency is called the Nyquist frequency. The FFT waveform displays the input signal frequency components from DC (0 Hz) to the Nyquist frequency.

Overview Tasks

Setting Up Signal Input

Use this procedure to set scale and position parameters for input signal acquisition.

Prerequisites

1 Set the acquisition system to run continuously.

Connect input signal

2 Connect the probe to the input signal source using proper probing/connecting techniques. For more details on input setup see page 76.

Select the input signal channel

3 Push a channel button (CH 1–CH 4) to select the input signal channel. A channel button lights when its channel is on.

Select input termination

4 Push an input termination button to toggle between 1 MΩ and 50 Ω input termination.

💡**Tip** Some probes automatically set their termination value.

Select input coupling

5 Open the Vertical Setup control window by clicking Vert on the button bar or selecting the Vertical Setup command from the Vertical menu. To change the input coupling, select the appropriate channel tab and click:

- DC to couple both the AC and DC components of an input signal.
- AC to couple only the AC components of an input signal.
- GND to disconnect the input signal from the acquisition system.

6 Click Close to close the window.

Invert the channel

7 Click Invert to toggle inversion of the selected waveform. When Invert is On, the selected waveform appears mathematically inverted (flipped) on the zero volt axis. No trace of the normal waveform is left on the screen.

Set the vertical acquisition window

8 Use the vertical knobs to scale and position the waveform on screen. Dragging the waveform handle also positions the waveform.

9 Open the Vertical Setup control window as described in Step 5. To change the offset, click the Offset control, and use the multipurpose knob to adjust the offset.

Set the horizontal acquisition window

10 Use horizontal knobs to scale and position the waveform on the screen and to set the record length. Dragging the reference icon also positions the waveform.

11 Use the Resolution knob to set the record length and sample interval.

12 If required to stabilize the display, push the trigger LEVEL control to

set the trigger level to 50%.

Setting Acquisition Modes

Use this procedure to set the data acquisition mode and specify acquisition start/stop methods.

Prerequisites
1 Power on the instrument and set up the horizontal and vertical controls and triggering.

Access the acquisition modes
2 Select Acquisition Mode from the Horiz/Acq menu to open the Acquisition Mode control window.

Select the acquisition mode
3 Click an Acquisition Mode button. For Average and Envelope modes only, select the number of acquisitions to average or envelope. For WfmDB acquisition mode select the number of samples.

Set the stop mode
4 Push the RUN/STOP front-panel button to toggle between starting and stopping acquisitions.

OR

Push the SINGLE button to acquire enough waveforms to satisfy the acquisition mode.

Select sampling mode
5 To select the sampling mode, click the Horiz toolbar button, or select the Horizontal/Acquisition Setup command from the Horiz/Acq menu to open the Horizontal/Acquisition Setup control window. Select the Acquisition tab.

6 Select Real Time Only, Interpolated Real Time, or Equivalent Time

- Real Time Only sampling acquires all the points in a waveform record after one trigger event.

- Interpolated Real Time sampling interpolates between the samples the instrument acquires when it cannot acquire all the real samples it needs to fill up its waveform record.

- Equivalent time sampling starts when the time base setting (horizontal scale) is too fast to let the instrument get enough samples with which to create a waveform record using real-time sampling. Equivalent time mode is intended for repetitive signals only.

Setting Up Roll Mode

Use the procedure that follows to set up roll mode acquisitions.

Prerequisites   1   Power on the instrument and set up the horizontal and vertical controls and triggering.

Enable roll mode   2   Click the Horiz toolbar button to open the Horizontal/Acquisition Setup control window. Select the Acquisition tab.

3   Click Roll Mode Auto to enable roll mode. The resolution, record length, horizontal scale, and number of active channels on are dependent variables. Therefore, the fastest horizontal scale setting when Roll mode starts is 40 ms/div, the record length is 500 (resolution is 2 ms), the acquisition mode is either Sample or Peak Detect, and 1 or 2 channels must be on. As the record length or the number of on channels grows, the time per division required to enter Roll mode is slower.

Note

Envelope and Average acquisition modes inhibit roll mode. Hi Res roll mode starts when the resolution is 4 ms or greater.

Enable single sequence roll mode   4   Push the SINGLE button to start and acquire enough points to satisfy the acquisition mode (when the waveform makes it to the left edge of the screen). The acquisition mode must be either Sample or Peak Detect.

Turn off roll mode acquisitions   5   To stop acquisitions in roll mode:

- If you are not in Single Sequence, push RUN/STOP to stop roll mode.

- If you are in Single Sequence, roll mode acquisitions stop automatically when a complete record is acquired.

Disable roll mode   6   Click the Horiz toolbar button to open the Horizontal/Acquisition Setup control window. Select the Acquisition tab.

7   Click Roll Mode Off to disable roll mode. Roll mode is also disabled when you set the horizontal scale to 200 ms per division or faster. At record lengths greater than 500 points, the time per division to turn off roll mode becomes slower.

Note

Switching to Envelope or Average acquisition mode will also turn off roll mode.

Using Fast Acquisitions

Use this procedure to set up Fast Acquisitions mode.

Prerequisites

1 Set up the horizontal and vertical controls and the triggering.

Enable Fast Acquisitions mode

2 Click the Horiz toolbar button to open the Horizontal/Acquisition Setup control window. Select the Acquisition tab.

3 Click Fast Acquisitions to toggle Fast Acquisitions, or push the front-panel FastAcq button.

Set the display appearance

4 Click the DISP toolbar button, or select the Display Setup command from the Display menu. The Display Control window opens on the Appearance tab.

5 Click one of the Display Style buttons: Vectors, Dots, or Inten Samp. Vectors is the default.

6 For the Display Persistence, click either Infinite, Variable or Off. Off is the default, and erases the screen each time a new waveform array is displayed. For an explanation of Variable persistence, see page 139.

7 If you select Variable, click Persist Time, and adjust the persistence time (decay rate).

Set the display readout options

8 Select the Display Setup control window Objects tab.

9 Click Display Date/Time to toggle the date/time display On and Off.

Adjust the intensity

10 Select the Display Setup control window Appearance tab.

11 Click the Waveform Intensity AutoBright button to toggle AutoBright On and Off.

   ▪ On automatically sets the maximum intensity to the value of the most frequent event.

   ▪ Off creates a display like an analog oscilloscope. The displayed brightness depends on the trigger rate.

12 Rotate the front-panel INTENSITY knob to adjust the intensity of displayed waveforms, or click the Intensity control and enter the intensity value with the keypad or multipurpose knob.

13 Select the Display Setup control window Color tab.

14 Click a Color Palette button in the Fast/Acq WfmDB Palette. In Fast Acquisitions mode, Temp and Spectral show more detail than the other color palettes.

For a definition of each color palette, see page 144.

Time Stamping Frames

Use Time Stamps to display the absolute trigger time for a specific frame and the relative time between triggers of two specified frames. To use FastFrame Time Stamps, do the following steps:

1 FastFrame mode should be set up as described in the previous example.

2 Turn on FastFrame

3 In the Time Stamps controls, click Readouts to toggle time stamp readouts on or off: Readouts use the following format:

Sel Ch# F xxx   DD MMM YYYY HH:MM:SS.mmm μμμ nnn ppp

Ref Ch# F xxx   DD MMM YYYY HH:MM:SS.mmm μμμ nnn ppp

Δ   DD MMM YYYY HH:MM:SS.mmm μμμ nnn ppp

Where:

Sel and Ref Ch# F xxx are the selected or reference frame number

DD MMM YYYY is the date (day, month, and year)

HH:MM:SS, is the clock time (hours, minutes, and seconds)

mmm μμμ,nnn,ppp is a fraction of a second (to picoseconds)

4 In the Time Stamps controls, click Source and select the source of the reference frame.

5 In the Time Stamps controls, click Frame and use the multipurpose knob or keypad to enter the number of the reference frame. This value sets the starting frame when measuring the relative time between two frames.

**Select the FastFrame and
time stamps selection
controls**

To set the Selected Frame and Reference Frame from the
Selection Controls window:

6 Click Selection Controls from the FastFrame control window
to display the FastFrame controls.

7 In the FastFrame controls window, click Selected Frame
Source, and select the source of the frame that you want to
view.

8 Under Selected Frame, click Frame, and use the
multipurpose knob or keypad to enter the number of the
specific frame that you want to view and take time stamps
on. The frame you select appears on the display.

> **Note**
>
> The Δ is the selected time stamp—the reference time stamp.

9 Click the Reference Frame Source and select the source of
the reference frame. Click Frame and use the multipurpose
knob or keypad to enter the number of the reference frame.

**To lock the reference
position frames**

10 From the Horiz/Acq menu, select FastFrame Setup.

> **Note**
>
> You can also get the FastFrame Setup control window by
> clicking the Set Up button on the Selection Controls
> window.

11 Click either Frame Tracking Live or All to lock the reference
and position frames together. When the frames are locked,
they maintain the same relative distance from each other when
either frame is adjusted:

- Live locks together the channel and math waveforms.
  All reference waveforms are locked together, but they
  are separate from the channel and math waveforms.

- All locks together all channel, math, and reference
  waveforms; adjusting one waveform adjusts all
  waveforms.

Triggering from the Front Panel

The front panel provides quick access to the most frequently used trigger controls. The trigger readout shows you the state of the trigger system.

The SLOPE, COUPLING, and SOURCE controls only apply to edge triggering. To access the advanced trigger controls, open the Trigger Setup control window by pushing the ADVANCED button (See Advanced Triggering on page 238 for more information).

Use this procedure to set up triggers using the front-panel controls.

Prerequisites

1 Set the acquisition system to RUN (the RUN/STOP control is lighted), and set the vertical and horizontal controls appropriately for the signal to be acquired.

Select the trigger type

2 Push the EDGE button to select edge triggering. Pushing ADVANCED opens the Trigger Setup control window to set up other trigger types.

Select the trigger slope

3 Push the SLOPE button to toggle between POS and NEG:

- POS triggers on a rising edge (positive-going) signal

- NEG triggers on a falling edge (negative-going) signal

You can also set the slope in the Trigger Setup control window.

Select the trigger level

4 To manually change the trigger level for edge triggers (or certain threshold levels for logic or pulse triggers), turn the LEVEL knob. You can also set the level in the Trigger Setup control window.

Set trigger level to 50%

5 To quickly set the edge trigger level to 50% of the waveform voltage range, push the LEVEL knob. The instrument sets the trigger level to the halfway point between the peaks of the trigger signal. This function has no effect for the other trigger types.

You can also click Set 50% in the Trigger Setup control window.

Select the trigger source

6 Push the up and down arrow buttons to select a SOURCE:

- CH 1 through CH 4 are the input channels. The channel that you select as a trigger source will function whether it is displayed or not.

- EXT is a fifth, nondisplayable auxiliary (external) trigger source. To use the auxiliary trigger, connect the source to the front-panel AUX IN connector.

- LINE is the AC line voltage. The oscilloscope is used to generate the trigger and you do not have to input a signal. The trigger level is fixed at zero volts.

Set the trigger coupling

7 Push the up and down arrow buttons to set COUPLING:

- DC passes both AC and DC components of the input signal.

- AC passes only the AC components of an input signal.

- HF REJ attenuates signals above 30 kHz.

- LF REJ attenuates signals below 80 kHz.

- NOISE REJECT provides lower sensitivity, reducing the chance of falsely triggering on noise.

Select the trigger mode

8 Push the MODE button to toggle between NORM and AUTO trigger modes:

- NORM requires a valid trigger event to acquire a waveform.

- AUTO acquires a waveform even if no valid trigger event occurs.

Be aware that in AUTO mode, the acquired waveforms may not be triggered.

Checking Trigger Status

To see the state and setup of the triggering circuit, use the trigger status lights, readout, and screen.

Trigger status from trigger status lights

1 To quickly determine the trigger status, check the ARM, READY, and TRIG'D controls.

- If TRIG'D is on, the oscilloscope has recognized a valid trigger and is filling the posttrigger portion of the waveform.

- If READY is on, the oscilloscope can accept, and is waiting for, a valid trigger.

- If ARM is on, the trigger circuitry is filling the pretrigger portion of the waveform record.

- If both TRIG'D and READY are on, a valid A event trigger has been recognized and the instrument is waiting for a delayed trigger. When a delayed trigger is recognized, the posttrigger portion of the delayed waveform will fill.

- If ARM, TRIG'D, and READY are off, the acquisitions have stopped.

Trigger status from acquisitions readout

2 To quickly determine the settings of some key trigger parameters, check the Trigger readout at the bottom of the display. The readouts differ for edge and the advanced triggers.

Trigger level on display

3 To see the trigger level on the waveform display, turn on a trigger level indicator (marker). See Trigger Level Marker on page 143 for instructions on setting the indicator.

The trigger level indicator remains on the screen as long as the channel providing the trigger source is displayed. You can choose between two types of trigger level indicators: a horizontal bar and an arrow at the right side of the graticule.

Triggering from the Trigger Setup Window

Some trigger parameters are accessible only through the Trigger Setup control window:

- Holdoff
- Trigger level presets
- Force trigger

Use the procedures that follow to set up these trigger parameters.

Set holdoff

Changing the holdoff time helps to stabilize triggering. To set holdoff:

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the Mode tab.

3. Click Default or Time:

- Default uses the instrument default holdoff time, which is five times the current horizontal scale setting. Default maintains a good value for general-purpose triggering.

- Time enables you to enter a holdoff that may allow more stable triggering. This time is used at all horizontal scale settings.

4. If using Time, click Trig Holdoff and enter the time value using the multipurpose knob. The holdoff range is 1.5   s to 12 seconds.

Select a preset trigger level

5. Push the front-panel ADVANCED button.

6. On the Trigger Setup control window, select the A Event tab.

7. Click a Trigger Type button to select a trigger, such as Edge, that uses a level adjustment. A trigger uses a level adjustment if the Level control appears on the right side of the screen with the trigger type setup controls.

8. Click Level, and then click the keypad icon to open the keypad. Click on either TTL, ECL, or USER:

- TTL fixes the trigger level at +1.4 V.
- ECL fixes the trigger level at −1.3 V.

- USER fixes the trigger level at a user preset voltage (see Steps 9 through 12).

When you set the volts/div less than 200 mV, the instrument reduces the TTL or ECL trigger levels below standard TTL and ECL levels. This reduction occurs because the trigger level range is fixed at +12 divisions from the center. At 100 mV (the next lower setting after 200 mV) the trigger range is +1.2 V, which is less than the typical TTL (+1.4 V) or ECL (−1.3 V) level.

Define new trigger level presets

9. If the menu bar is not displayed, click the Menu toolbar button.

10. Select the Utilities menu User Preferences option. The User Preferences control window opens.

11. Select the Keypad Defaults tab.

12. Click the appropriate Trig Level and adjust the preset using the multipurpose knob or pop-up keypad.

13. If desired, click a Keypad Label and change the label of the preset using the pop-up keypad.

Force a trigger

14. Push the front-panel ADVANCED button.

15. On the Trigger Setup control window, select the A Event or B Event tab

16. Click Edge.

17. To force the instrument to immediately acquire one waveform record, even without a trigger event, click Force Trigger.

Forcing a trigger is useful when, in normal trigger mode, the input signal is not supplying a valid trigger. Click Force Trigger to quickly confirm that there is a signal present for the instrument to acquire. Then determine how to trigger on the signal (for example, push LEVEL to set 50%, check the trigger source setting, and so on).

The instrument acts on Force Trigger even if selected before the end of pretrigger holdoff. However, the button has no effect if the acquisition system is stopped.

Triggering on a Glitch

Use this procedure to detect and trigger on a glitch or to ignore a glitch.

Select glitch triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click Glitch.

Select the source

4. To specify which channel becomes the trigger source, click Source, and select the source from the list.

Select the polarity and width

5. Click a Polarity button to specify the glitch polarity:

    ▪ Pos looks at positive-going pulses.

    ▪ Neg looks at negative-going pulses.

    ▪ Either looks at both positive and negative pulses.

6. To specify the width of the glitch, click Width and set the glitch width using the multipurpose knob or keypad.

7. To specify whether to trigger on glitches narrower or greater than the width you specify, click:
    ▪ Trig if Width < to trigger only on narrower pulses.
    ▪ Trig if Width > to trigger only on wider pulses.

Set the level

8. Click Glitch Trigger Level and use the multipurpose knobs, keypad, or front-panel LEVEL knob to set the level that the glitch must cross to be recognized by the instrument.

You can set the level to a value appropriate to either the TTL or ECL logic families by clicking TTL or ECL on the Level pop-up keypad.

Set mode and holdoff

9. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page 136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

Triggering on a Runt Pulse

Use this procedure to detect and trigger on a runt pulse. The instrument can look for positive or negative runt pulses. These pulses can also be qualified by time or by the logical state of other channels.

Select runt triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click Runt.

Select the source

4. To specify which channel becomes the pulse trigger source, click Source and select the source from the list.

Select the polarity

5. Select a Polarity value to specify the direction of the runt pulse. Triggering occurs at the point the pulse returns over the first threshold without crossing the second threshold level. The polarity selected in the Polarity window determines the order that the threshold must be crossed for a runt trigger to occur:

   ▪ Pos looks at positive-going pulses. The lower threshold must be first crossed going positive, then recrossed going negative without the upper threshold being crossed at all.

   ▪ Neg looks at negative-going pulses. The upper threshold must be first crossed going negative, then recrossed going positive without the lower threshold being crossed at all.

   ▪ Either looks at both positive and negative pulses. Either one of the thresholds must be first crossed going in either direction, then recrossed going in the opposite direction without the other threshold being crossed at all.

For all three polarity settings, triggering occurs at the point the runt pulse recrosses its first threshold.

Qualify the runt trigger

6. To qualify the runt pulse trigger, click Trigger When Runt, and select from the list:

   ▪ Occurs triggers on all runt pulses.

   ▪ Wider triggers only on runt pulses that exceed the minimum width you set. Enter the Width using the multipurpose knob or pop-up keypad.

   ▪ Logic triggers on a logic-qualified runt pulse. When the instrument detects that a runt has occurred, it checks the logical state of two other available channels and triggers only if their conditions are met. Each channel can have a value of high (H), low (L), or "don't care" (X). A value is considered high if the channel input voltage is greater than

the specified threshold voltage; a value is considered low if the channel input voltage is less than the specified threshold voltage. Use the "don't care" selection for any channels that will not be used as part of the pattern. (Available only on 4-channel instruments.)

Set the thresholds

7. To set the threshold levels, click Upper Limit or Lower Limit, and use the multipurpose knobs or pop-up keypad to set the values.

Tip

To use the trigger level marker to help in setting the threshold levels, open the Display Setup control window, and click Long on the Objects tab.

Set mode and holdoff

8. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page 136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

Triggering on a Pulse Width

Use this procedure to detect and trigger on a pulse that is either wider or narrower than the specified range.

Select width triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click Width.

Select the source

4. To specify which channel becomes the trigger source, click Source, and select the source from the list.

Select the polarity

5. To specify the polarity of the pulse, click Pos (for positive-going pulses) or Neg (for negative-going pulses).

Set the range

6. To set the range of pulse widths in units of time, click Upper or Lower Limit, and enter the values with the multipurpose knobs or pop-up keypad:

▪ Upper Limit is the maximum valid pulse width the trigger source will look for.

▪ Lower Limit is the minimum valid pulse width. The instrument will always force the Lower Limit to be less than or equal to the Upper Limit.

7. Click Trigger When, and select Inside to trigger on pulses that fall within the specified range, or Outside to trigger on

pulses that are outside the range.

Set the level

8. Click Level and use the multipurpose knob or pop-up keypad to set the trigger level.

Set mode and holdoff

9. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page 136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

Triggering on Transition Time

Use this procedure to detect and trigger on pulse edges that traverse between two thresholds at faster or slower rates than the specified time. You can set up the instrument to trigger on positive or negative edges.

Select transition triggering

1. Push the front-panel ADVANCED button.

2. In the Trigger Setup control window, select the A Event tab.

3. Click Transition.

Select the source

4. To specify which channel becomes the trigger source, click Source, and select the source from the list.

Select t he polarity

5. Select a Polarity value to specify the direction of the pulse edge:

   - Pos monitors the transition time (slew rate) of the positive-going edges of pulses. The edge must first cross the lower threshold and then cross the upper threshold.

   - Neg monitors the transition time (slew rate) of the negative-going edges of pulses. The edge must first cross the upper threshold and then cross the lower threshold.

   - Either monitors positive- and negative-going edges of pulses. The edge may first cross either threshold and then cross the other.

Set the transition time

6. The threshold levels and the delta time setting determine the transition time (slew rate) setting. The level settings determine the voltage component of slew rate (Volts/Second). Click the Upper Level or Lower Level button and use the multipurpose knobs or pop-up keypad to set the values for the upper and lower levels.

   You can set the level(s) to a value appropriate to either the TTL or ECL logic families by clicking TTL or ECL on the Level pop-up keypad.

Tip

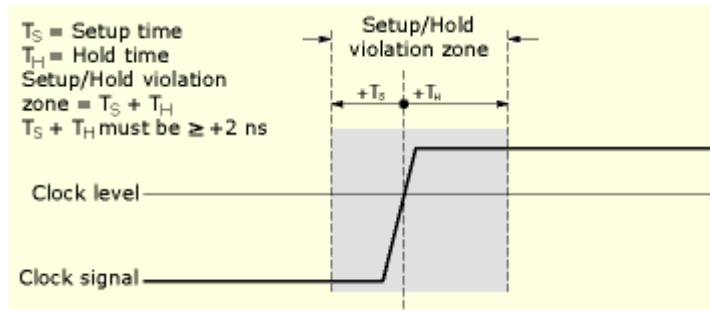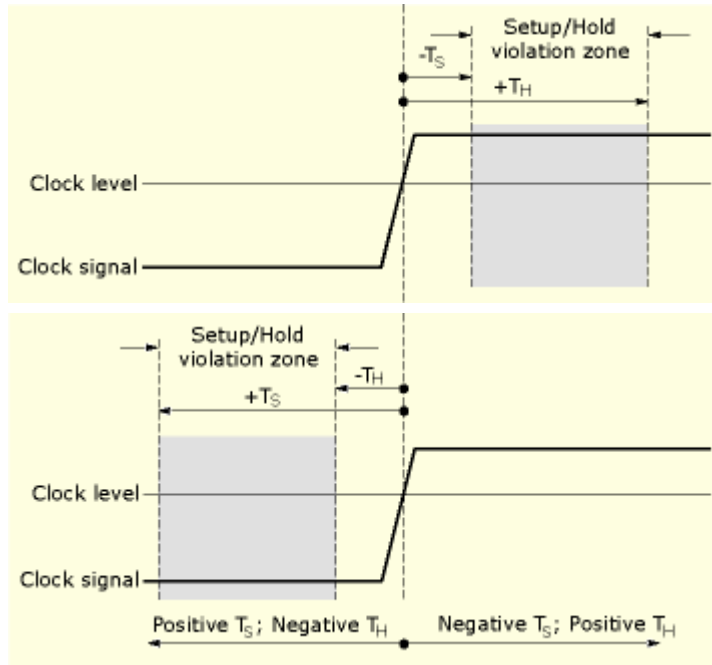To use the trigger level marker to help in setting the threshold levels, open the Display Setup control window, and click Long on the Objects tab.

7. To finish specifying the slew rate (transition time), set the time component by clicking Time and using the multipurpose knob or keypad to set the delta time value.

8. Click Trigger When Transition Time < to trigger when the transition time is less than the time you set above. Click Trigger When Transition Time > to trigger when the transition time is greater than the time you set above.

    If you select Trigger When Transition Time > (greater than) and the instrument does not trigger, the pulse edge may be too fast. To check the pulse edge speed, switch to edge triggering. Trigger on the pulse edge and determine the time the edge takes to travel between the upper and lower levels that you set for this trigger. The instrument cannot transition trigger on pulse edges that traverse between threshold levels in 600 ps or less.

Also, to reliably trigger on transition time, a pulse must have a width of 7.5 ns or more. A pulse of less width may trigger on the wrong slope or not trigger at all. Switch to edge triggering, and check the pulse width if you can't trigger on transition time as expected.

Set mode and holdoff

9. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

Triggering on Pulse Timeout

Use this procedure to detect and trigger on pulses longer than a specified time. The instrument will trigger at the end of the timeout period.

Select timeout triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click Timeout.

Select the source

4. To specify which channel becomes the trigger source, click Source and select the source from the list

Set the level

5. To set the trigger timeout level, click Level and use the multipurpose knobs or pop-up keypad to enter the voltage level.

You can set the level to a value appropriate to either the TTL or ECL logic families by clicking TTL or ECL on the Level pop-up keypad.

Set the timer

6. To set the timeout timer, click Timer and use the multipurpose knob or pop-up keyboard to set the time.

Set the pulse threshold

7. To set the pulse threshold, click:

    ▪ Stay High to trigger if the signal stays higher than the trigger level for longer than the timeout (Timer) value.

    ▪ Stay Low to trigger if the signal stays lower than the trigger level for longer than the timeout (Timer) value.

    ▪ Either to trigger if the signal stays lower or stays higher than the trigger level for longer than the timeout (Timer) value.

Set mode and holdoff

8. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page 136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

Triggering on a Pattern

Use this procedure to trigger the instrument when the logic inputs cause the selected function to become True or False. You can also specify that the logic conditions must be satisfied for a specific amount of time before the instrument triggers.

Select pattern triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click Pattern.

Define pattern inputs

4. Select an input value from the drop-down list for each channel. The channel inputs combine to form a logic pattern. Each channel can have a value of high (H), low (L), or "don't care" (X). A value is considered high if the channel input voltage is greater than the specified threshold voltage; a value is considered low if the channel input voltage is less than the specified threshold voltage. Use the "don't care" selection for any channels that will not be used as part of the pattern.

Set threshold voltages

5. To set the threshold voltages, click on the channel threshold control, and use the multipurpose knobs or pop-up keypad to set each threshold.

Select logic pattern

6. Select the Boolean logic function for the combination of input channels. The instrument will trigger on a clock edge when the input waveforms match the specified logic pattern.

See Pattern Triggers on page 239 for definitions of the logic functions for pattern triggers.

Set trigger criteria

7. From the Trigger When Pattern list, select True to trigger the instrument when the logic patterns are true. Select False to trigger the instrument when the logic patterns are false.

Set mode and holdoff

8. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page 136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

To define a time qualified pattern trigger

9. You can time qualify a pattern logic trigger, that is, you can specify the amount of time that the Boolean logic function must be True. You must select Less than or More Than for the Trigger When Pattern control. Then, click on Trigger When Time and set the time using the multipurpose knob or keypad.

When you select Less Than and specify a time, the input conditions must drive the logic function True for less than the specified time. Conversely, the True for More Than item requires the Boolean function to be True for longer than the time you specify.

The instrument determines the trigger point in the following manner:

- It waits for the logic condition to become True.

- It starts timing and waits for the logic function to become False.

- It compares the times and, if the time True is longer (More Than Time) or shorter (Less Than Time), then it triggers a waveform display at the point the logic condition became False. This time can be, and usually is, different from the time set.

Triggering on Logic State

Use this procedure to trigger the instrument when all of the logic inputs to the selected logic function cause the function to be True or False when the clock (channel 4) input changes state. (The clock input is channel 2 on 2-channel instruments.)

Select state triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click State.

Define state inputs

4. Select an input value from the drop-down list for each channel. Channels 1, 2, and 3 represent the data inputs on 4-channel instruments, Channel 1 on 2-channel instruments. Channel 4 should be connected to the clock signal on 4-channel instruments, Channel 2 on 2-channel instruments.

5. Select either the Pos (rising) or Neg (falling) edge of the Clock Channel.

The channel inputs combine to form a logic pattern. Each channel can have a value of high (H), low (L), or "don't care" (X). A value is considered high if the channel input voltage is greater than the specified threshold voltage; a value is considered low if the channel input voltage is less than the specified threshold voltage. Use the "don't care" selection for any channels that will not be used as part of the pattern.

Set threshold voltages

6. To set the threshold voltages, click the channel threshold control, and use the multipurpose knobs or pop-up keypad to set each threshold.

Define logic

7. Select the Boolean logic function for the combination of the input channels. The instrument will trigger on a clock edge when the input waveforms match the specified logic pattern. (The Pattern Type controls do not appear on this control window for 2-channel instruments.)

See Pattern Triggers on page 239 for definitions of the logic functions for pattern triggers.

Set trigger criteria

8. Use the Trigger When Pattern selection to determine when the instrument should trigger. Select True to trigger the instrument when the logic patterns go true. Select False to trigger the instrument when the logic patterns go false.

For simplest operation, leave this control set to True. Setting the control to False complements the output of the chosen pattern function (for example, from AND to NAND or NOR to OR).

Set mode and holdoff

9. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page 136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

Triggering on Setup/Hold Time Violations

Use this procedure to detect and trigger on setup and hold time violations. The setup and hold times define a violation region relative to a clock. Data is considered to be invalid in the violation region.

Select setup/hold triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click Setup/Hold.

Select the data source

4. To select the channel that is connected to the data signal, click Data Source, and select the source from the list. Do not select the same channel for both the data and clock sources.

Select the clock source and set the edge

5. To select the channel that is connected to the clock signal, click Clock Source, and select the source from the list.

Do not select the same channel for both the data and clock sources.

6. The Clock Edge determines whether the instrument finds the trigger point on the rising edge (left button) or the falling edge (right button) of the clock signal. Click a Clock Edge button.

7. To set the data transition level, click Data Level, and use the multipurpose knobs or pop-up keypad to set the value.

8. The instrument uses the clock level to determine when a clock edge occurs. The point the clock crosses the clock level is the reference point from which it measures setup and hold time settings. To set the clock threshold level, click Clock Level and use the multipurpose knobs or pop-up keypad to set the value.

Both Clock and Data Levels can be set to a value appropriate to either the TTL or ECL logic families by clicking TTL or ECL on the Level pop-up keypad.

9. Click Setup Time, and use the multipurpose knob or pop-up keypad to set the setup time relative to the clock.

10. Click Hold Time, and use the multipurpose knob or pop-up keypad to set the hold time relative to the clock.

Positive setup time always leads the clock edge; positive hold time always follows the clock edge. Setup time always leads the hold time by at least 2 ns ($TS + TH. \geq 2$ ns). Attempting to set either time to reduce the 2 ns limit adjusts the other time to maintain the limit.

See the figure below for an example of a setup and hold violation trigger.

In most cases, you will enter positive values for both setup and hold time. Positive values set the instrument to trigger if the data source is still settling inside the setup time before the clock or if it switches inside the hold time after the clock. You can skew this setup/hold violation zone by entering negative values as shown below:

T_S = Setup time
T_H = Hold time
Setup/Hold violation zone = $T_S + T_H$
$T_S + T_H$ must be $\geq$ +2 ns

Setup/Hold violation zone

+T_S +T_H

Clock level

Clock signal

The oscilloscope recognizes the violation and triggers at the clock edge.

Cursors measure the setup/hold violation zone which equals setuptime + hold time (30 ns).

Data (Ch1) transition occurs within 14.88 ns after the clock violating the hold time limit.

Set up and hold violation trigger

314
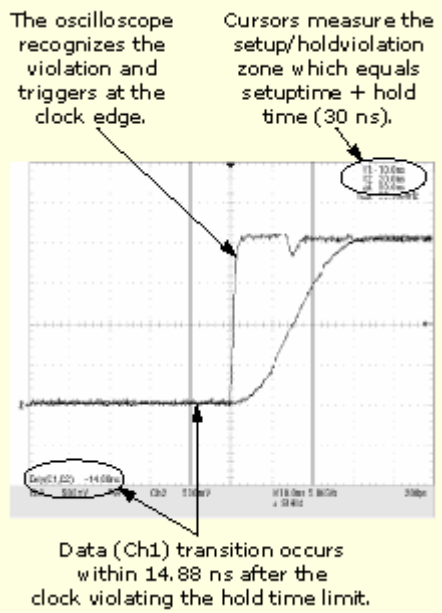
Triggering on Window Threshold Violations

Use this procedure to trigger the instrument when the input signal rises above an upper threshold level or falls below a lower threshold level.

Select window triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click Window.

Select the source

4. To specify which channel becomes the trigger source, click Source, and select the source from the list.

Set the thresholds

5. The upper and lower threshold levels define the voltage limits of the window. To set the threshold levels, click Upper Level or Lower Level, and use the multipurpose knobs or pop-up keypad to set the values.

Quality window trigger

6. To qualify the window trigger, select from these Trigger When drop-down list combinations. Note that the Trigger When controls change as you select different combinations of options.

   ▪ Inside Limits and Occurs or Outside Limits and Occurs: Trigger the instrument when the signal enters (or leaves) the window defined by the threshold levels.

   ▪ Inside Limits and Wider Than or Outside Limits and Wider Than: Trigger the instrument when the signal enters (or leaves) the window defined by the threshold levels for the time specified by Width.

   ▪ Inside Limits and Logic or Outside Limits and Logic: When the instrument detects a signal entering or leaving the window defined by the threshold levels, it checks the logic state of up to two other available channels and triggers only if their conditions are met. Each channel can have a value of high (H), low (L), or "don't care" (X). A value is considered high if the channel input voltage is greater than the specified threshold voltage; a value is considered low if the channel input voltage is less than the specified threshold voltage. Use the "don't care" selection for any channels that will not be used as part of the pattern. (Logic qualification is only available on 4-channel instruments.)

Set mode and holdoff

7. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page 136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

Triggering on a Video Signal

Use this procedure to detect and trigger on a video signal. Only composite signal formats are supported. Graphic display formats such as RGB and VGA are not supported.

Select video triggering

1. Push the front-panel ADVANCED button.

2. On the Trigger Setup control window, select the A Event tab.

3. Click Video.

Select the source

4. To specify which channel becomes the trigger source, click Source, and select the source from the list.

Select the format

5. Select the video signal format from several preset formats, or set a custom format. If using a preset format, click the Lines and Fields Autoset buttons to optimize the instrument for viewing the fields or lines for your chosen format.

   If you select the Custom signal format, no Autoset options are available. You must set the Scan Type (Interlaced or Progressive) and Scan Rate.

Custom scan rates enable you to trigger on non-broadcast video signals that have unique scan rates. Select the scan rate that includes your signal: 15–20 kHz, 20–25 kHz, 25–35 kHz, 35–50 kHz, or 50–65 kHz.

Specify where trigger will occur on the video signal

6. Select a value from the Trigger On drop-down list to specify where the trigger will occur on the video signal:

   - If you select Field, use the drop-down list to select either Odd, Even, or All.

   - If you select Line #, use the multipurpose knob to set the line number. Line numbers wrap when their limits are reached. The range of line numbers depends on the format.

   - If you select the NTSC format, use the multipurpose knob to first scroll through all of the odd lines, then the even lines.

   - If you select All Lines, the trigger will occur on any and all lines.

Set the holdoff type

7. Specify Holdoff By Time or By Fields, and use the multipurpose knob to change the value. If setting a Custom format, only the By Time control is available.

8. Select Normal or Inverted polarity:

   ▪ Normal generally triggers on negative polarity, since the sync pulse leading edge is negative for most standard formats.

   ▪ Use Inverted when probing circuitry that inverts the video signal.

9. All graticule types are available for selection from the drop-down list. The IRE graticule is normally used with NTSC signals, and the mV graticule is normally used with all other video signals.

Set mode and holdoff

10. Mode and holdoff can be set for all standard trigger types. Refer to trigger mode on page 136 and Set holdoff on page 121 to learn more about trigger mode and holdoff.

Triggering on a Sequence

Use this procedure to set up the instrument to trigger on a sequence.

Prerequisites

1. The instrument must be installed with a signal connected to an input channel. Set the acquisition system to RUN, and set the vertical and horizontal controls appropriately for the signal to be acquired.

To trigger on A (main) only

1. Click Trig on the toolbar, and select the A->B Seq tab of the Trigger Setup control window.

2. Click A Only to turn off sequential triggering.

To trigger on B after time

1. To set the time base to run after an A trigger, a trigger delay, and a B trigger, click Trig on the toolbar, and select the A->B Seq tab of the Trigger Setup control window.

2. Click Trig After Time.

3. To set the trigger delay, click Trig Delay and use the multipurpose knob or pop-up keypad to set the time.

4. To set the B trigger level, click B Trig Level and use the multipurpose knob or pop-up keypad to set the level.

1. To set the time base to trigger after an A trigger and a specified number of B trigger events, click Trig on the toolbar, and select the A->B Seq tab of the Trigger Setup control window.

2. Click Trig on nth event.

3. To set the number of B trigger events, click Trig Event, and use the multipurpose knob, pop-up keyboard, or up and down arrows to set the number of events.

4. To set the B trigger level, click B Trig Level, and use the multipurpose knob or pop-up keypad to set the level.

To set up B triggering

1. To set the B Event trigger, click Trig on the toolbar, and select the B Event tab of the Trigger Setup control window. Or, from the menu bar, select the B Event (Delayed) Trigger Setup command.

2. To specify which channel becomes the B trigger source, click Source and select the source from the list.

3. Click Coupling and select the trigger coupling to match your trigger signal.

4. To specify the direction of the edge, click a Slope button. The left button selects a positive slope and the right button selects a negative slope.

5. To set the B trigger level, click B Trig Level, and use the multipurpose knob or pop-up keypad to set the level. Note that you can click Set 50% to set the B trigger level to the midpoint between the positive and negative peak of the B trigger signal.

Triggering on a Communication Signal

Use this procedure to set up the instrument to trigger on a communication signal.

Prerequisite

1. The Instrument must have Option SM installed.

Select comm triggering

2. Push the front-panel ADVANCED button.

3. On the Trigger Setup control window, select the A Event tab.

4. Click Comm.

Select the source and type

5. To specify which channel becomes the trigger source, click Source, and then select the source from the list.

6. Select either Clock or Data from the Type list. If you select Clock, you will also need to select the polarity.

Select coding format and standard

7. Select the coding format for triggering on a communication signal and the standard to use with the coding format. The coding format determines which standards are available as well as other parameters, such as trigger threshold and pulse form. The standard sets the bit rate. Changing the bit rate will change the Standard to Custom.

8. Depending on the coding format you may also need to set the Pulse Form and Threshold Level.

Select a communications mask

9. Follow the Setting up Mask Testing procedure on page 335 to select the communications mask you want to use.

Displaying Waveforms

This procedure demonstrates the display adjustments you can make.

Prerequisite

1. The instrument must be installed with a signal connected to an input channel. Set the acquisition system to RUN.

Set vertical display parameters

2. Push the CH button of the channel with the connected signal to display the waveform. The CH button is lighted.

3. Adjust the front-panel vertical controls to achieve a good display of the waveform.

Set horizontal display parameters

4. Use the front-panel horizontal knobs to adjust the SCALE and POSITION of the waveform and to set sample RESOLUTION.

The RESOLUTION knob also sets the record length. (See Horizontal Acquisition Window Interrelated Parameters on page 230 for details.)

5. Push the LEVEL knob (PUSH TO SET 50%) if required to stabilize the display.

Adjust the horizontal reference

6. To adjust the point around which the waveform expands and contracts, click on the horizontal reference marker, and drag it along the horizontal axis until it aligns to the point on the waveform that you want to be stationary on screen.

Note that if horizontal delay is off, the horizontal reference

point is the same as the horizontal position.

7. Release the Horizontal Reference, and then adjust the Horizontal Scale to expand or contract the waveform.

Using MultiView Zoom with Waveforms

Prerequisites

1. The instrument must be installed and operating, with a signal connected to an input channel, and horizontal and vertical controls and triggering set up.

2. Obtain a stable display of the waveform to be measured.

Select Zoom

3. Click and drag across the segment of the waveform that you want to see in greater detail.

4. Select Zoom (1-4) On from the shortcut menu to magnify the highlighted waveform segment.

The instrument displays the zoomed area on the waveform as magnified in the bottom graticule. You can also press the front-panel MultiView ZOOM button to split the screen and add a zoom graticule.

5. To zoom on multiple areas, repeat steps 3 and 4.

Focus the MultiView Zoom on waveform(s) of interest

6. Push the front-panel MultiView Zoom HORIZ or VERT button to select the axis that you want to focus on in the zoom graticule. The Zoom control window opens, if it is not already present.

7. To select the waveform that you want to scale or position, select the Channel (Ch), Math, or Reference (Ref) waveform from the drop-down list.

8. Use the multipurpose knobs to adjust position and factor of the zoomed waveform. Note that as you adjust the factor or position the instrument adjusts the Zoom box relative to the unmagnified waveform.

Set up MultiView Zoom options

9. Click Setup in the Zoom control window to open the Zoom Setup control window.

10. To change the size of the unmagnified (Acquisition) and Zoom windows, select one of the following from the Graticule Size drop-down list:

   ▪ 50-50 allocates half of the available display for the zoomed graticule and half of the available display for the acquisition window.

   ▪ 80-20 allocates 80% of the available display for the zoomed graticule and 20% for the acquisition window.

   ▪ Full uses the entire display for the zoomed graticule.

11. Click the Lock and Scroll tab to select a Zoom Lock and scroll setting:

- To scroll a single zoomed area, click Zoom 1–4, and then click an AutoScroll button.

- To scroll multiple zoomed areas simultaneously, click Lock, and then click the Zoom 1–4 buttons that you want to scroll through.

Check the MultiView Zoom scale and position

12. To quickly determine the magnification factor and position of a zoomed waveform, check the readouts on the appropriate tab (Horizontal or Vertical) of the Zoom Setup control window.

Reset MultiView Zoom

13. To reset all Zoom settings to their defaults, click the Position Factor Reset button in the Zoom Setup control window.

Taking Automatic Measurements

Use this procedure to quickly take a measurement based on the default settings for High and Low reference levels.

Prerequisites

1. The instrument must be installed and operating, with a signal connected to an input channel, and horizontal and vertical controls and triggering set up.

2. Obtain a stable display of the waveform to be measured.

Select the source waveform

3. From the toolbar, click Meas to open the Measurement Setup control window.

4. You can use a channel, reference, or math waveform as the source. Click the appropriate waveform button on the Ch, Math, or Ref Source tab.

Take automatic measurements

5. Select the tab on the Measurement Setup control window that contains the measurement you want to take: Ampl, Time, More, Histog, or Comm.

6. Click the button for the measurement.

   The readout for the measurement and associated statistics is automatically displayed, and the measurement is added to the Measurements list in the control window.

   Measurements are displayed in the right control window area. If the window is occupied or there are too many measurements to fit in that area, the measurements are displayed in the lower graticule area.

In roll mode, measurements are not available until after you stop acquisitions.

Remove measurements

7. To remove the measurement, click Clear. The last measurement on the list is removed. To remove any measurement, click the name on the Measurements list to highlight the measurement, and click Clear. You can also click and drag to select and clear several measurements.

8. Toggle the display of measurements on the graticule by clicking the Display button On and Off.

Display measurement statistics

9. From any tab on the Measurement Setup control window, click Setup Statistics. The Measurement Statistics control window opens.

10. From the Measurement Statistics control window, click:

    ▪ Off to turn off the display of measurement statistics

    ▪ Mean to display the mean of measurements

    ▪ All to display the Mean, Min, Max, Standard Deviation, and Population measurements

11. To set the number of measurements included in the measurement statistics, click Weight n=, and use the multipurpose knobs or pop-up keypad to set the weighting.

Set measurement reference levels

12. From the toolbar, click Meas to open the Measurement Setup control window.

13. From any tab on the Measurement Setup control window, click Setup Ref Levs to open the Reference Levels setup control window.

14. To set how the instrument determines the base and top of

the waveform, click Determine Base, Top From:

- Min-Max to use the highest and lowest values of the waveform record. (See Automatic Measurements on page 254 High/Low Method for more information.)

- Histogram to select the most common values above or below the midpoint. (See Automatic Measurements on page 254 High/Low Method for more information.)

15. To select the reference level units, click Units:

- Absolute to set the units to absolute values in user units.

- Percentage to set the units as a percentage of the High/Low range.

16. To set the reference levels, click on these controls and use the multipurpose knob or pop-up keypad to set:

- High Ref to define the high reference level. The default level is 90%.

- Mid Ref to define the middle reference level. The default level is 50%.

- Low Ref to define the low reference level. The default level is 10%.

- Mid2 Ref to define the middle reference used on the second waveform specified in delay or phase measurements; the default level is 50%.

Take a snapshot of measurements

17. Select the Snapshot command from the Measure menu, or open the Measurement Setup control window and click Snapshot on any tab. A window showing all single waveform measurements opens.

Snapshot measurements are taken on the selected waveform. The Snapshot window tells you the channel that the measurements are being taken on and the reference levels used.

18. Snapshot measurements are not continuously updated. To update snapshot measurements, click Snapshot Again.

Phase, Delay, and Histogram measurements are not included in a snapshot.

Localizing a Measurement

Use this procedure to take a measurement over a segment of the waveform.

Prerequisites

1. The instrument must be installed and operating, with a signal connected to an input channel, and horizontal and vertical controls and triggering set up.

2. Obtain a stable display of the waveform to be measured.

Access gating

3. From the toolbar, click Meas to open the Measurement Setup control window.

4. Click Setup Gating from any tab. The Gating control window opens.

Enable and position the gates

5. To select how to define the gated area, click one of the following under Measurement Gating:

   - Cursor to set the gated area to the area between the cursors. Use the multipurpose knobs to adjust the cursors on screen.

   - Zoom (1–4) to set the gated area to the waveform area contained in the Zoom (1–4) area.

   - Off to turn off measurement gating. You can also turn gating off from the Zoom shortcut menu.

6. If you selected Cursor, you must also select the Track Mode:

   - Indep mode lets you move one cursor at a time using the multipurpose knobs.

   - Tracking mode allows you to move both cursors at the same time. The cursors track one another; moving one cursor moves the other cursor in tandem.

Setting Cursor Sources

Use this procedure to target the cursors to a source.

Prerequisites

1. Display the waveform(s) to be measured on the screen.

Access cursor setup

2. Push the CURSORS front-panel button, or click Cursors on the toolbar to open the Cursor Type Setup control window.

Select the cursor type

3. Click a Cursor Type button. See Cursor Types on page 257 for an explanation of the cursor types.

Select the cursor sources

4. Select a Cursor Source from the Source drop-down list.

If you are using H Bars or V Bars there is only one source for both Cursor 1 and 2. If you are using Waveform or Screen cursors, select a source for both Cursor 1 and 2.

If a waveform is not available, it appears shaded in the drop-down list.

Change the cursor position

5. To change the position of a cursor, use the multipurpose knobs, enter a numeric value in the Curs# Pos controls, or click and drag the cursors to a new position.

Set cursor tracking

6. To change the cursor tracking mode, click Setup on the Cursor Type Setup control window. The Cursor Setup control window opens. (All adjustments that you can make in the Cursor Type Setup control window can also be made in this window.)

7. Select a cursor Track Mode:

   ▪ Indep makes each cursor moveable without regard to the position of the other cursor.

   ▪ Tracking makes both cursors move in unison and maintain a fixed horizontal or vertical distance between each other.

8. To turn the display of the cursors on or off, push the front-panel Cursor button.

Set cursor style

9. For waveform or screen cursors set the cursor style by selecting one of the following from the list:

   ▪ Lines display as the intersection of horizontal and vertical lines. Lines are the default style for Screen cursors.

   ▪ Line & X display the horizontal position as a vertical line; the Y-axis is displayed as an X.

   ▪ X displays the horizontal and vertical position as a single X.

Starting and Resetting Histogram Counting

Use this procedure to quickly take a measurement based on the default settings for histograms.

Prerequisites

1. Display the waveform(s) to be measured on the screen.

Open histogram setup window

2. From the toolbar, click the Meas button to open the Measurement Setup control window. Click Histogram to open the Histogram control window.

3. Select the Source Ch, Math, or Ref tab, and then select the waveform source for the histogram.

4. Click either Histogram Mode button to start histogram counting and display the histogram data:

    ▪ Horiz displays a horizontal histogram that shows how time varies in the histogram box.

    ▪ Vert displays a vertical histogram that shows how your vertical units vary in the histogram box.

    ▪ Off turns off histogram counting and display.

Note that this control turns on histogram counting and data display. You can only display one type of histogram at a time.

5. Click Reset to reset the histogram count. Histograms track numbers of counts. Clicking Reset resets those counts to zero and begins counting from zero.

6. Click Display to toggle the display of the selected histogram on and off.

7. Click a Histogram Scaling option:

    ▪ Linear displays histogram data linearly. Bin counts smaller than the maximum counts are scaled linearly by dividing the bin count by the maximum bin count.

    Horizontal histograms have a maximum of 500 bins while vertical histograms have a maximum of 200 bins.

    ▪ Log displays histogram data logarithmically. Bin counts smaller than the maximum counts are scaled logarithmically. Logarithmic scaling provides better visual details for bins with low counts.

8. Click Adjust Histogram Box Limits and use the Top Limit, Bottom Limit, Left Limit, and Right Limit controls to set the size of the histogram box. The histogram box selects the section of the waveform used for histograms.

9. Click Adjust Histogram Box Location and use the X Location and Y Location controls to set the location of the histogram box.

Tip Right-click and select **Clear Data** from the short cut menu to clear histogram data.

Creating a Math Waveform

Use this procedure to create a math waveform. Ensure that the sources you want to use exist. The sources do not have to be displayed.

Prerequisites
1. All waveforms and automatic measurement scalars that you want to use must be available.

Open the Math Setup control window
2. From the toolbar, click the Math button to open the Math Setup control window.

Select a math waveform
3. Select the Mathn tab for the math waveform that you want to create, and click Display to toggle it on.

Tip

If the waveform that you select already exists, its math expression appears in the window. You can still use the waveform by clicking Clear, which discards the previous math expression. You may also want to select another waveform.

Select an expression
4. Click one of the Predefined Expression buttons to use a predefined math expression, or click Editor to access the Equation Editor.

Define or edit an expression
5. Use the Equation Editor control window to define a math expression. See Creating Math Waveforms on page 261 for examples, and use these guidelines:

   - Math definitions are not implemented if the sources are not valid.
   - Use the Bksp (backspace) button to remove the last entry; click Clear to remove the entire expression and start over.
   - Use parentheses to group terms in the expression to control execution order, for example, 5(Ch1 + Ch2).

6. Select the Time, Freq, or Meas tabs to display the available functions.

7. Click a function button to enter the function in the math expression. Click an operand button (+, -, *, /) to insert an operand.

8. Use the Home and arrow buttons to move within the math expression. Use the Bksp (backspace) button to delete portions of the expression.

9. Click Apply to apply your new math expression to the math waveform.

Apply averaging

10. Click Avgs to open the Math Parameters control window. The controls in the window apply to the math waveform defined by the expression.

11. Select one of the Mathn = controls, and set the number of averages using the multipurpose knobs or pop-up keypad. This number of averages affect math waveforms if the Avg() function is used.

Apply expression to waveform

12. Once you have defined the math expression to your satisfaction, click Apply. Apply is available on the Equation Editor and Spectral Setup control windows.

Using Math Waveforms

Use the same techniques to work with math waveforms that you use with channel waveforms. This procedure demonstrates some common operations that you can perform on math waveforms:

Note

Each math waveform that you create derives its horizontal scale and position from the sources you include in its math expression. You can adjust these controls for the source waveforms, and your adjustments are reflected in the math waveform. You can also magnify math waveforms using MultiZoom, and use the mouse for positioning.

Prerequisites

1. A math waveform must be defined and displayed.

Select and display

2. Click the Math toolbar button to open the Math Setup control window.

3. Select the tab for the math waveform that you want to work with.

   ▪ If the waveform that you select is not defined, use the Creating a Math Waveform procedure on page 327 to define the math waveform.

- If the waveform is not displayed, click Display.

4. Click Position and Scale, and use the multipurpose knobs or pop-up keypad to size and position the waveform on the screen.

Keep in mind the following:

- The position is in divisions, so changing the scale can make the math waveform disappear until position is also changed (the same effect happens with channel waveforms).
- You can click and drag a waveform handle to change the waveform vertical position.
- You can change the waveform vertical position and scale by clicking on the waveform handle and using the multipurpose knobs to adjust.
- You cannot adjust the offset of a math waveform.
- You cannot adjust horizontal scale, position, and sample density (resolution) of math waveforms; different length source waveforms result in a math waveform of the shortest source record length. If you adjust these settings for sources of a math waveform, the adjustments are reflected in the math waveform.

Take automatic measurements

5. Click the Meas toolbar button to open the Measurement Setup control window.

6. Select the Source Math tab, and click a channel button to choose a math waveform.

7. Select a measurement (for more information, see Taking Automatic Measurements on page 356).

8. To display the measurement, click Display to toggle it to on.

Take cursor measurements

9. Click the Cursor toolbar button to display the cursors and open the Cursor control window.

10. Select the Math tab, and click the numbered button for the math waveform that you want to measure.

11. Select the cursor type by clicking H Bars, V Bars, Screen, or Waveform (for more information, see Taking Cursor Measurements on page 353).

12. Use the multipurpose knobs to position each cursor on the math waveform to measure the feature that interests you.

13. Read the results in the cursor readout. The cursor readout is displayed under the multipurpose readouts or in the

upper right corner of the graticule area.

Note that amplitude measurements on a derivative waveform are in volts per second, and volt-seconds for an integral waveform measurement.

Using a Predefined Spectral Math Waveform

The following procedure is an example of setting up the instrument to perform spectral analysis of a signal. This example uses the probe compensation signal available on the front panel of the instrument.

Install the test hookup

1. Connect the probe compensation signal to CH 1 through a suitable probe.

2. Press DEFAULT SETUP.

3. Press AUTOSET.

Display the waveform

4. From the toolbar, click Vert, and select the Chan 1 tab.

5. Click Offset, and using the multipurpose knobs or keypad, set the offset to 900 mV and the Ch1 Scale to 200 mV.

Display the spectral math waveform

6. From the toolbar, click Math, and select the Math 1 tab.

7. Click the Predefined Mag button.

The instrument sets up a predefined magnitude spectral analysis waveform. Your waveform should be similar to the one below.
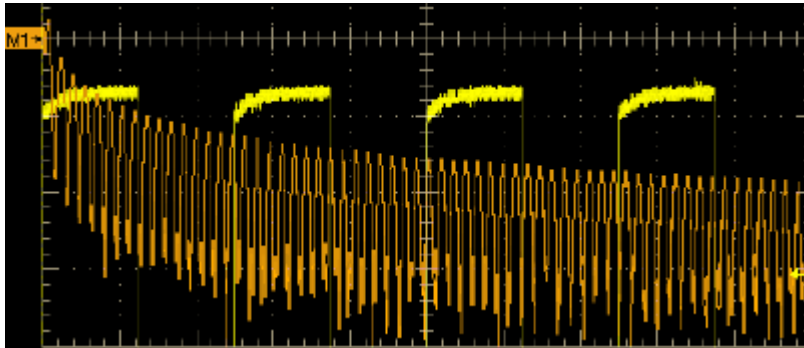
Display the spectral math waveform (cont.)

8. Click the Spectral Analysis Setup button.

9. Use the multipurpose knobs or keypad to set the frequency span to 125 kHz and the center frequency to 62.5 kHz (if necessary, reduce the sample rate).

Setup the Cursors

10. From the toolbar, click Cursors.

11. To assign the cursors to the spectral analysis math waveform, select Math 1 from the list Source list for Cursor 1.

12. Use the multipurpose knobs or keypad to set the Curs1 Pos

to 0.0 Hz and the Curs2 Pos to 125 kHz.

The cursor readout now indicates the frequency span set in step 9.

13. Use the multipurpose knobs or keypad to set the Curs2 Pos to 62.5 kHz.

The cursor readout now indicates the center frequency set in step 9.

Measure the test results

14. Use the multipurpose knobs or keypad to set the Curs1 Pos to 3.0 kHz and the Curs2 Pos to 11.0 kHz.

In this example, the cursors are now on the third and eleventh harmonic of the probe compensation signal. Read the frequencies from the cursor readouts.

15. Click the Cursor Type Waveform button. Click the Cursor 2 button and then the Math 1 button.

Now in addition to the frequency at the cursor locations, the cursor readout displays the amplitude at the cursor locations. The readout also displays the difference in frequency and amplitude between the cursor locations as shown below.
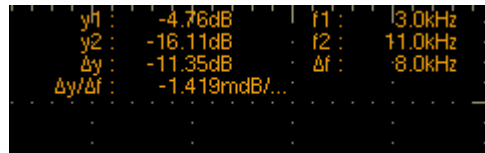
Defining a Spectral Math Waveform

Use this procedure to define a spectral math waveform. First ensure that the sources you use exist. Channel sources should be acquiring or have acquired data. These sources do not have to be displayed to be used.

Prerequisites

1. All sources that you will use in your math waveform must be available.

Open the Spectral Analysis Setup control window

2. From the toolbar, click the Math button to open the Math Setup control window.

3. Click Spectral Analysis Setup to open the Spectral Analysis Setup control window.

Select a spectral waveform

4. Click Mathn, and select the math waveform that you want to create from the list.

5. Click Magnitude to create a magnitude spectral waveform or Phase to create a phase spectral waveform. If you want an averaged spectral waveform, click Average, and click Avgs to set the number of averages. See the step Apply

Averaging under Creating a Math Waveform on page 327 for details.)

6. Select the tab and channel number for the source waveform.

Tip

If you want to redefine your waveform, click Clear button, and repeat the above steps.

Display the spectral waveform

7. To display your spectral waveform, click Apply or OK.

Set the magnitude scale

8. Select the Mag tab.

9. To select the vertical scale factor, click Scale:

   • dB to display magnitude using the log scale, expressed in dB relative to the reference level offset.

   • Linear to display magnitude using units equal to the source units.

   • dBm to set the reference level offset to predefined values for dBm; see next step.

Tip

You can adjust the scale and position by clicking the waveform handle and using the multipurpose knobs.

10. To set the reference level, click Reference Level, and use the multipurpose knobs or pop-up keypad to set the reference level.

Tip

Reference level is the value at the top of the display screen. It only applies to magnitude waveforms. Adjusting the reference level positions the waveform with respect to the top of the display, but does not change the position of the waveform with respect to its ground reference.

11. To set the reference level offset, click Reference Level Offset, and use the multipurpose knobs or pop-up keypad to set the offset.

Tip

Offset determines where zero dB is in the output waveform. Changing offset moves the waveform with respect to its ground reference. When the input is equal to the offset, it will display as zero dB in the output.

Set the phase scale

12. Select the Phase tab.

13. To select the vertical scale factor click Scale:

    - Degree to set the phase units to degrees. Phase is displayed using degrees as the scale, where degrees wrap from −180° to +180°.

    - Radian to set the phase units to radians. Phase is displayed using radians as the scale, where radians wrap from −Π to +Π.

    - GroupDelay to unwrap the phase spectrum and display its derivative.

Using the Phase Controls on page 271 provides in-depth information on setup for phase displays.

14. To specify whether to unwrap phase in a spectral analysis phase waveform, click Unwrap to toggle.

15. To set the level in dB that a magnitude in the spectrum must exceed to have its phase computed (to reduce the effect of noise in your phase waveform), click Suppression Threshold, and use the multipurpose knobs or pop-up keypad to set the threshold level. If the magnitude is less than the threshold, then its phase is set to zero (see Using the Phase Controls on page 271 for an example).

See Using the Phase Controls on page 271 for additional information on suppression threshold.

Set time and frequency domain control tracking

16. Select the Control tab.

17. To enable changing time and frequency domain controls for one math waveform to change the same controls for another math waveform, click the Track Time/Freq Domain Controls buttons to toggle them on or off.

Select the window type

18. To select the window type, click Window Type, and select from the list. For more information on window types see Using Spectral Analyzer Windows on page 272.

Set the frequency domain controls

19. To set the frequency range over which the spectral analysis is performed, click Freq Span, and use the multipurpose knobs or pop-up keypad to set the frequency range. Higher sample rates allow you to set greater frequency spans. To set the frequency span to the maximum allowed by the current sample rate, click Full.

20. To set the center frequency of the spectral analysis, click Center Freq, and use the multipurpose knobs or pop-up keypad to set the center frequency.

21. Resolution bandwidth determines how small of a frequency difference can be resolved in the frequency domain output data. It basically defines the bandwidth of the filters used to do the frequency domain analysis. To set the resolution bandwidth, Click Res BW, and use the multipurpose knobs or keypad to set the resolution bandwidth.

Set the time domain controls

22. The Time domain controls determine the sample rate and record length of the acquisition. These controls allow you to change the duration on the acquisition without changing the sample rate. From the Spectral Analysis Setup menu, click Resolution and adjust the time interval between data samples of the input waveform.

Note

Resolution is the inverse of Sample rate. Adjust resolution to adjust sample rate. Resolution may also cause a change in record length such that the duration is kept constant at the value selected by the Duration control.

23. To adjust the number of seconds over the duration of the acquired waveform (record length), click Duration, and use the multipurpose knobs or pop-up keypad to adjust the duration.

Note: Changing duration also changes the record length.

24. Gate position is the position of the phase reference point in the gate with respect to the trigger in seconds. The gate position and gate duration must be within the acquisition. To set the gate position, click Gate Pos, and use the multipurpose knobs or pop-up keypad to adjust the gate position.

25. Gate duration and resolution bandwidth are the same. Gate duration is displayed in seconds and resolution bandwidth in hertz. To set the gate duration, click Gate Dur, and use the multipurpose knobs or pop-up keypad to adjust the gate duration.

Take cursor measurements

26. From the toolbar, click the Cursors button to display the cursors and open the Cursor control window.

27. Select the Math tab, and click the channel button for the spectral waveform that you want to measure.

28. Select the cursor type. For more information, see Taking Cursor Measurements on page 353.

29. Use the multipurpose knobs to position each cursor on the waveform to measure the feature(s) that interests you.

30. Read the results in the cursor readout.

The cursor readout is displayed under the multipurpose readouts as shown here or in the upper right corner of the graticule area.

The figure shows the cursor measurement of a frequency magnitude on an FFT. The readout reads 0 dB because it is aligned with the 1 VRMS level. The other readout reads 24.4 dB indicating the magnitude of the frequency it is measuring is 24.4 dB relative to 1 VRMS. Display of the source waveform is turned off.

The cursor units will be in dB or volts for magnitude waveforms and in degrees or radians for those measuring phase.

31. Select V Bars, and use the multipurpose knobs to align the two vertical cursors to points of interest along the horizontal axis of the waveform.

32. Read the frequency difference between the cursors from the Δ: readout. Read the frequency of each cursor relative to the zero frequency point from the cursors readout.

Setting Up Mask Testing

To set the instrument to perform mask tests, do the following procedure.

Prerequisites

1. Connect the instrument to the source signal, or save the source signal to a math or reference waveform memory location.

Access the Mask Setup window

2. From the button bar, Click Masks.

The instrument displays the Mask control window.

Select a mask test signal source

3. From the Source tab select the channel, math, or reference tab, and then select the waveform to use as the mask test source. You can only mask test one waveform at a time.

Select the mask type

4. To specify the mask Type, select the Masks tab. Click the appropriate button in the Type field. Click the More button to display further selections.

Note

Industry standard masks are available with Option SM.

The window lists mask types and standards that are available on your instrument.

Selecting a mask type and standard adjusts the instrument

horizontal, vertical, and trigger settings to those appropriate for displaying a waveform of the specified type. If the signal is not within the mask, click the Autoset button to center the waveform in the mask. If Autoset did not align the signal in the mask, adjust the instrument vertical and horizontal controls.

If you click the Autoset button and the Autoset Undo preference is On, the instrument will display an Autoset Undo window. Click the Undo button to return to the previous settings, or click the Close button to remove the window.

Select the mask standard

5. To specify the mask standard, select a standard from the drop-down list.

Select display parameters

6. In the Masks tab, click the Display button to toggle mask display on or off. The mask must be turned on to do mask testing.

7. Click the Hit Count button to turn on or off hit counting. The hit count is shown in the Pass/Fail Results tab.

8. Click the Display Config button to set mask hit highlighting and to lock the mask to the waveform. Lock Mask to Waveform resizes the mask to reflect changes in the horizontal or vertical settings of the instrument. This control is also on the main mask setup window.

Autoset the signal

9. In the Masks tab, click the Autoset button to have the instrument automatically adjust instrument settings to align the waveform to the mask based on the characteristics of the input signal. Autoset is done on the first waveform acquired after clicking the Autoset button.

    If the Autoset Undo preference is On, the instrument will display an Autoset Undo window. Click the Undo button to return to the previous settings, or click the Close button to remove the window.

The Autoset Config button opens a configuration window that lets you set the vertical, horizontal, and trigger autoset parameters, activate autofit or autoset, choose the autoset mode, return to the default autoset configuration, or return to the Mask Setup control window.

Enable and set waveform autofit parameters

10. In the Masks tab, click the Autofit button to enable the waveform autofit function. Autofit checks each waveform for any mask hits. If there are hits, autofit repositions the waveform to minimize hits. The number of hits reported is the number after autofit has minimized hits.

    The autofit Config button lets you set the autofit maximum waveform repositioning parameters (as a percentage of the

336

horizontal and vertical divisions), return to default settings, or return to the Mask Setup control window. Use the keypad to change the vertical or horizontal autofit parameters.

11. Click the Masks button to return to the Mask control window.

Set mask test tolerance margins

12. Click the Masks Setup window Tolerance tab to set the percentage of margin used in the mask test. Use the control knob, keypad, pop-up keypad, or up and down arrow buttons to enter the mask margin tolerance percentage. The range of values is –50% to 50%.

Margin tolerance settings greater than 0% expand the size of the segments, making the mask test harder to pass; margin tolerance settings less than 0% (negative percent) reduces the size of the segments, making the mask test easier to pass.

Set mask test pass and fail parameters

13. Select the Pass/Fail Setup tab of the Masks control window.

14. Use the control knob, keypad, or pop-up keypad to enter the number of waveforms to test (number of samples in some modes), the failure threshold (the number of waveforms that must fail to fail the test), and the delay time (the time from when mask test starts to when the instrument begins sampling).

15. Use the Failure field buttons to set what the instrument does when a mask test fails; have the instrument beep (BEEP), send an SRQ out on the GPIB bus (SRQ), stop signal acquisition immediately (Stop Acq), send email (E-mail), and/or print the instrument screen image to a printer (Print).

You many need to select a sound source to enable the BEEP. See Selecting the Sound Source on page 344 for more information.

16. Clicking the More button displays more failure functions. Save Wfm saves the the waveform data of the first waveform that causes the test to fail to a .wfm file. Log Date saves time, date, and basic test information of the first waveform that causes the test to fail to an ASCII text (.txt) file.

Both files are saved to the location specified by the Path button. The file name format is YYMMDD-HHMMSS, where YY is year, MM is month, DD is day, HH is hour, MM is minutes, and SS is seconds.

17. Use the Completion field buttons to set what the instrument does at the completion of a mask test.

18. Use the Polarity buttons to set mask and waveform polarity. Positive tests the positive waveform pulses. Negative inverts the mask and tests the negative waveform

pulses. Both tests the first half of the tested waveforms in positive polarity mode, then tests the remaining waveforms in negative polarity mode.

19. Toggle the Repeat button to On to set the instrument to repeat (continue) mask testing on the completion of each test.

Creating a User Mask from a Defined Mask (Option)

Refer to Mask Key Points on page 282 before creating or editing a mask. To create a user mask from a defined mask, do the following procedure.

Access the Mask Setup window

1. From the button bar, click Masks and select the Masks tab.

The instrument displays the Mask control window.

Select the mask type and standard

2. Click the appropriate button in the Type field to select a mask type. Click the More button to display further selections.

Note

Industry standard masks are available with Option SM. Only User masks are the only mask type available without Option SM.

3. Select a standard from the drop-down list.

The control window lists mask types and standards that are available on your instrument.

Copy current mask

4. Click the User Mask button.

5. Click the Copy Current Mask to User Mask button. The instrument copies the current mask to the user mask memory.

Edit the User mask

6. See Edit User Mask procedure on page 371.

Save the user mask to disk

7. See Saving a User Mask to Disk procedure on page 340.

You do not need to save the edited user mask to disk, as the instrument retains the current user mask in nonvolatile memory. However, if you plan to create a number of user masks, you will need to store the user masks on disk or the hard drive, as the instrument can only load one user mask at a time.

Editing a User Defined Mask

Access the mask edit window

1. From the button bar, click Masks, and select the Masks tab.
2. Click the User Mask button.
3. Click the Edit User Mask button. The instrument displays the Mask Edit control window.

Enable the mask edit controls

4. Click the Controls button to open the mask edit controls window on the right side of the screen. This provides the maximum area to display the mask, making editing easier.

Select a segment

5. Click the Segment field and use the arrow buttons, multipurpose knob, or keypad to select a segment to edit. The selected (active) segment is highlighted in red. Each mask can have up to 16 segments.

Select a vertex

6. Click the Vertex field and use the arrow buttons, multipurpose knob, or keypad to select the vertex to edit. The active vertex is indicated with an X on the template segment. Each segment can have up to 50 vertices.

Move a vertex

7. Click the Horizontal field and use the multipurpose knob or keypad to change the selected vertex horizontal position.
8. Click the Vertical field and use the multipurpose knob or keypad to change the selected vertex vertical position.

Add or delete a vertex

9. To add a vertex, select the closest vertex that is clockwise from where you want to place a new vertex. Click Add to add a vertex midway between the selected vertex and the next counterclockwise vertex.
10. To delete a vertex, enter or select the vertex number. Then click Delete to delete the selected vertex. The remaining vertices located counterclockwise from the deleted vertex are renumbered.

Save the user mask to disk

11. See Saving a User Mask to Disk procedure on page 340.

You do not need to save the edited user mask to disk, as the instrument retains the current user mask in nonvolatile memory. However, if you plan to create a number of user masks, you will need to store the user masks on disk or the hard drive, as the instrument can only load one user mask at a time.

Saving a User Mask to Disk

Access the Mask Setup window

1. From the button bar, click Masks and select the Masks tab.
2. Click the User Mask button.
3. Click the Edit User Mask button. The instrument displays the Mask Edit control window.

Save the user mask to disk

4. Click the Mask Save button.

The instrument opens the Save As dialog box. The default save location is in the TekScope/Masks folder.

5. Enter the mask name in the File Name field. The default save type is User Mask Files (*.msk).
6. Click Save to save the mask to disk.

Recalling a User Mask from Disk

Access the Mask Setup window

1. From the button bar, click Masks and select the Masks tab.
2. Click the User Mask button.
3. Click the Edit User Mask button. The instrument displays the Mask Edit control window.

Recall the user mask from disk

4. Click the Mask Recall button.

The instrument opens the Recall dialog box. The default recall location is the TekScope/Masks folder. If the mask files are in another folder, use the navigation controls to access the appropriate folder.

5. Select the mask name.
6. Click Recall to load the user mask into user mask memory on the instrument.

Alternatively, the mask can be recalled by selecting Recall from the File Menu. This opens the Recall dialog box where you can navigate to and select the mask that you want to recall.

Creating a New User Mask

To create a new user mask that is not based on an existing mask, do the following procedure.

Set instrument settings

1. Use the communications trigger features to trigger the instrument on a signal. The instrument saves these settings with the mask information.

Create an empty user mask

2. From the button bar, click Masks, and select the Masks tab.

3. Click the User Mask button.

4. Click the mask standard field to display the drop-down list.

5. Select None from the list.

6. Click the Copy Current Mask to User Mask button. If you are asked if you want to overwrite the current user mask, click the Yes button.

Create an edit new mask segments

7. Click the Edit User Mask button to display the user mask edit functions.

8. Click the Segment field, and use the arrow buttons, multipurpose knob, or keypad to enter or select segment 1.

9. Click the Vertex Add button. The instrument draws the default new segment shape, a triangle.

10. Use the instructions on Editing a User Mask on page 339, starting at step 5 to edit a segment.

11. Repeat steps 9 through 11, selecting an unused and sequential segment number, to create and edit more segments.

Save the user mask to disk

12. See Saving a User Mask to Disk procedure on page 340.

You do not need to save the user mask to disk, because the instrument retains the current user mask in nonvolatile memory. However, if you plan to create a number of user masks, you will need to store the user masks on disk or the hard drive, because the instrument can load only one user mask at a time.

Using Limit Testing

Prerequisites

1. Make sure the waveform that you want to use for a template exists. Your source can be either an active or a saved channel, math, or reference waveform.

Create template

2. Select the source for your template.

3. Select the destination for storing the template.

4. Set the tolerances specifying the amount of variation from the template.

5. Click Save to save the template to the location specified in step 3.

You can create multiple templates and save them in Ref1-Ref4 or to a location of your choice.

Select waveform to compare

6. Select the source for the waveform that you want to compare against the template.

7. Select the template that you want use for the limit test. If the template that you want to use is stored in a location other than Ref1–4, then you must first recall the template waveform to one of the reference locations. See Recalling a Template Waveform on page 355.

Set up the limit test options

8. Click Config to set up the action that you want the instrument to take upon failure of the limit test.

Note

The instrument can take multiple actions.

9. Turn Lock Template to Waveform on if you want to be able to move the source waveform and the template vertically together; turn it off to move them vertically independent of each other.

10. Click Highlight Hits on to show points that fall outside the template in a different color.

Activate limit testing

11. Click the Limit Test button On to begin testing.

12. Click Reset to clear the highlighting of points that violated the template.

Sending E-mail on Events

You must have Option SM installed to be able to send e-mail on Mask Test Failure.

Setup E-mail on Event

1. From the Utilities menu, select E-mail on Event Setup.
2. Enter the recipients' e-mail addresses in the box.
3. Click Config, and enter your SMTP server address.
4. Select the events on which to send e-mail.
5. Select which types of e-mail attachments you want to include.
6. Set the maximum message limit. (The maximum allowed by the system is 50.)
7. Set the maximum message size.

The default is 1 MB the maximum is 2 GB. Setting the maximum message size to 0 causes attachments to be saved to the default location on the instrument hard drive.

8. Click Send to send a test e-mail.

Set up e-mail on Limit test failure

1. See Using Limit Testing on page 342 to set up the limit test.
2. Under Failure Notification, click Config, and then click E-mail On.
3. Click the Limit Test button On to begin testing.

Set up e-mail on Mask test failure

1. See Setting up the Mask Test on page 335, and follow the procedure for setting up a mask test.
2. Under Pass/Fail Test Notifications, click E-mail to On.
3. Click the Pass Fail Test button On to start the mask test. Click the Repeat button for continuous mask testing.

Set up e-mail on Trigger

1. Select the trigger type.
2. Select the trigger source.
3. Set the appropriate parameters for the trigger.
4. From the Mode tab, click E-mail on Trigger to On.
5. Press the Run/Stop button to begin acquiring data to trigger.

Selecting the Sound Source

Your instrument is shipped with an internal speaker that some of the software features use to send an audible alert to the user. To enable this speaker, the sound card in your instrument must be turned off. With the sound card off, you will not have access to standard Windows audio features, such as the ability to listen to a music CD.

If you turn the sound card on, the internal speaker is disabled. You will need to connect headphones or an amplified speaker to the headphone jack on the instrument to hear the alerts.

The default setting disables the sound card.

To turn the sound card on or off, perform the following procedure:

Access the device manager

1. From the File menu, click **Minimize**, or press **CTRL+M** to minimize the instrument application.
2. On your Windows desktop, right-click the **My Computer** icon.
3. Click **Properties**.
4. In the System Properties dialog box, click the **Hardware** tab.
5. Click the **Device Manager** button.

Select the source

6. Double-click **Sound, video, and game controllers**.
7. Double-click **SigmaTel C-Major Audio**.
8. In the Device usage drop-down list, click either Use this device (enable) or Do not use this device (disable).

Save your changes

9. Click **OK**.
10. Close the **Device Manager** dialog box.
11. Close the **System Properties** dialog box.

Saving a Waveform

Use this procedure to save a waveform or waveforms to a reference location, the instrument hard disk, a CD, or third-party storage device.

Prerequisites

1. The instrument must be powered up.
2. Make sure the waveform to be saved exists. Your source must be a channel, an active math waveform, or an active reference waveform. Display the waveform with the setup in which you want to save it.

Open the Save dialog box

3. From the File menu select Save or Save As. Under Save What, click Waveform.

Select the waveform to save

4. From the Source drop-down list select the waveform that you want to save. Only available waveforms are displayed in the list.

Save the waveform to a reference

5. Select the location (Ref 1–4) under Save in Oscilloscope Memory. Click Save to save your waveform to that reference location. Data in the existing reference will be overwritten.

Save the waveform to a file

6. To save the waveform to a file, select the location to save from the Save in drop-down list. The default location is C;\TekScope\Waveforms.

7. Name your waveform file by typing a name in the Name text box, or select an existing name from the file list (data in the existing file will be overwritten).
    OR
    Select the Auto-increment file name check box to save a series of files without typing in a new name each time. For more information, see Saving Reference Waveforms Using Auto-Increment File Name on page 354.

    ### Tip

    If you do not have an attached keyboard, click the keyboard icon to display a virtual keyboard.

8. The file extension specified in the Save as Type list will automatically be appended to the file name. You can select from these file formats:

    - wfm is an internal instrument format

    - csv is usable by spreadsheets such as Microsoft Excel.

    - dat is usable by Mathcad and MATLAB.

    - txt is the standard text file format.

9. If your file type is other than .wfm, click the Options button to set up the Waveform Save Options.

Set waveform save options

10. Select the data format (Data destination) for the waveform that you want to export.

11. Select a valid waveform from the Source list.

12. Under Waveform Detail, select the Include waveform scale factors check box if you want to include waveform details like header information (record length, sample interval, trigger point information, and so on). If you only want to include the waveform voltage values, clear the selection.

13. Under Waveform Data Range, select the samples that you want to save. You can select a range of samples, only the samples between the cursors, samples in a Zoom Area, or All samples. If you select Samples between cursors, you can use the multipurpose knobs to adjust the positions if the cursors are active. If they are not, the cursors are activated in their most recent positions.

14. If you are using FastFrame, select the frame range to include in the saved waveform:

- Frames to enter a range of frames to include in the saved waveform

- All Frames to include all frames in the saved waveform

15. If you are exporting Fast Acquisitions data, select the data ordering from the list.

16. Click OK, and then click Save to save the waveform file. To cancel without saving, click Cancel.

Saving a Screen Capture

Use this procedure to save screen captures to the instrument memory, a CD, or third-party storage device.

Prerequisites

1. Images you want to save must be displayed in the graticule.

Open the Save dialog box

2. From the File menu select Save or Save As. Under Save What, click Screen Capture.

Set the screen capture options

3. Click the Options button to open the Screen Capture Save Options dialog box.

4. Select a color Palette for your screen capture.

5. Under View: to save an image of the entire screen, select Full-screen; to select an image of the selected graticule, select Graticule(s) only; select Hide Menu Toolbar check box if you do not want the menu and toolbar included.

6. Select the type of image from the Image list. To save the image exactly as it appears on the screen, select Normal. To export the image on a white background, select Ink-saver Mode.

7. Select a data format from the Screen Capture Format list.

8. Click OK to save the screen capture options, or click Cancel to close the dialog box without saving the changes.

Save the screen capture to a file

9. To save the screen capture to a file, select the location to save from the Save in drop-down list. The default location is C:\TekScope\Screen Captures.

10. Name your screen capture by typing a name in the Name text box, or select an existing name from the file list (data in the existing file will be overwritten).
OR
Select the Auto-increment file name check box to save a series of files without typing in a new name each time.

11. Click Save.

Saving Measurements

Use this procedure to save measurements to a spreadsheet or text format..

Prerequisites

1. Measurements that you want to save must be selected.

Open the Save dialog box

2. From the File menu select Save or Save As. Under Save What, click Measurement.

Set the measurement options

3. Click the Options button to open the Measurement Save Options dialog box.

4. Select either Displayed Measurements or Measurements Snapshot.

5. Select the Measurement Format from the drop-down list.

6. Click OK to save the measurement options, or click Cancel to close the dialog box without saving the changes.

Save measurements to a file

7. To save the measurements to a file, select the location to save from the Save in drop-down list. The default location is C:\TekScope\data.

8. Use the default name for your file, or type a unique name in the Name text box, or select an existing name from the Name list (data in the existing file will be overwritten).
OR
Select the Auto-increment file name check box to save a series of files without typing in a new name each time.

9. Select the file type from the Save as type drop-down list (.cvs for spreadsheets, .txt for text). This is the same as the Measurement format that you selected in the Options dialog box. Changing the file type here will also change the selected format in the Option dialog.

10. Click Save.

Copying Waveform Data

Use this procedure to copy waveforms, images, or measurements to the Windows clipboard:

Prerequisites

1. Make sure the waveform, image, or measurement to be copied exists; that is, your source must be a channel, an active math waveform, an active reference, and so on.

Select format for copy

2. Highlight the Select for Copy submenu on the Edit menu. The submenu lists all available waveform, image, and measurement types available for export:

- Full Screen exports a bitmap of all screen contents
- Graticule exports a bitmap of only the graticule area
- Waveform exports waveform data
- Measurements exports measurement data

Select setup for copy

3. Select Copy Setup from the Edit menu to open the Copy Setup control window.

Set image copy options

4. Select a color Palette for your copied images.

5. Select the image View: to copy an image of the entire screen, select Full-screen; to copy an image of the selected graticule, select Graticule(s) only.

6. Select the type of image from the Image list. To copy the image exactly as it appears on the screen, select Normal. To copy the image on a white background, select Ink-saver Mode.

7. Select the Data Format, either BMP or JPG.

Set waveform copy options

8. Select the Waveforms tab.

9. Select the data format (Data destination) for the waveform that you want to copy.

10. Select a valid waveform from the Source list.

11. Select the Include waveform scale factors check box if you want to include waveform details like header information (record length, sample interval, trigger point information, and so on). If you only want to include the waveform voltage values, clear the selection.

12. Select the number of samples that you want to copy from the Waveform curve data range. Select all samples, a range of samples, or only the samples between the cursors. If you select Data between cursors, you can use the multipurpose knobs to adjust the positions if the cursors are already active. If they are not, the cursors are activated in their most recent positions.

13. If you are using FastFrame, select the frame range to include in the exported files:
    - All Frames to include all frames in the exported files
    - Frames to enter a range of frames to include in the exported files

14. If you are copying Fast Acquisitions data, select the data ordering from the list.

Set measurement copy options

15. Select the Measurements tab.

16. Select the data format for the measurement that you want to copy.

17. Select Displayed Measurements to copy all measurements

that are displayed on screen, or Measurements Snapshot to copy the current snapshot data. If there is no valid snapshot data, the default snapshot values are copied.

Copy the data

18. Click Copy to save your setup information. The copy operation takes place after you click Copy.

Creating MyScope Control Windows

Use this procedure to create your own control windows:

Prerequisites

1. Select New Control Window… from the MyScope menu.

Select controls

2. Click a control group to display the available controls within that group.

3. Click a control. The control displays in the preview window.

4. Clear the checkbox for any control that you do not want to appear. Most controls show all controls selected by default.

Place, move, and delete the controls

5. Drag the customized control to your MyScope control window. You can also move a control after it has been placed in the new control window by dragging it.

6. To remove the control from the custom control window select it, and then click the Delete button.

Create tabs

7. Click New Tab to add up to eight tabs.

8. Click the Rename Tab to create a unique name for each tab in your control window.

Set User Preferences

9. Click the User Pref… button, and then select the user preferences that you want to apply to this control window.

10. By default the preferences are the same as what is currently set on your instrument. The preferences you set from the MyScope Setup window only apply to your MyScope control window. The preferences will take effect when the MyScope control window is activated.

Save

11. Click Save or Save As… to save your MyScope control window.

12. Enter a unique name for your MyScope control window and, click Save. The default location to save MyScope files is C:\TekScope\MyScope.

Recall   13. From the MyScope menu, click Open Control Window….

14. Select the MyScope control window that you want from the Open MyScope Control Window dialog box and click Open.

Editing MyScope Control Windows

Use this procedure to edit previously defined MyScope control windows.

Prerequisites   1. Select Edit Control Window… from the MyScope menu.

Select control window to edit

2. Select the MyScope control window that you want to edit from the Open MyScope Control Window dialog box. The control window that you selected opens in the MyScope Setup control window.

Make changes   3. Edit the control window using the same procedures as you did to create the control window on page 349.

Save   4. Click Save to save your changes to this control window.

5. Click Save As to save your changes as a different control window. Enter a unique name for your MyScope control window, and click Save. The default location to save MyScope files is C:\TekScope\MyScope.

Tasks

Calibrating Active Probes

Use this procedure to optimize gain and offset accuracy at the probe tip for each active probe. Allow the instrument to warm up for 20 minutes before performing this procedure. Then complete the following steps:

1   If you have not already done so, connect the probe to the instrument.

2   Connect the probe to the probe compensation terminals as shown. Connect only one probe at a time.

3   Select Probe Cal… from the Vertical menu to open the Probe Calibration control window.

4   Select the channel to which the probe is attached.

5   Click Calibrate Probe to begin the calibration procedure. The Probe Status indicator changes to Pass when the procedure is complete. If the procedure does not pass, contact your local Tektronix service personnel.

6   Remove the connections from the terminals.

Calibrating Passive Probes

Use this procedure to optimize gain and offset accuracy at the probe tip. Allow the instrument to warm up for 20 minutes before performing this procedure.

If you have not already done so, perform the low frequency probe compensation for passive probes.

Note

The probe calibration procedure is not recommended for the P5050 passive probes. These probes have little gain or offset errors. However, you should perform this procedure for other passive probes.

Complete the following steps:

1   If you have not already done so, connect the probe to the instrument.

2   Connect the probe to probe compensation terminals as shown. Connect only one probe at a time.

3   Select Probe Cal… from the Vertical menu to open the Probe Calibration control window.

4   Select the channel to which the probe is attached.

5   Click Calibrate Probe to begin the calibration procedure. The Probe Status indicator changes to Pass when the procedure is complete. If the procedure does not pass, contact your local Tektronix service personnel.

6   Remove the connections from the terminals.

Deskew

Use the following procedure to compensate for timing differences between voltage probes:

1   Set up the instrument to display all of the channels that you want to deskew.

2   Push the instrument AUTOSET button.

3   Adjust the vertical SCALE and POSITION controls for each channel so that the signals overlap and are centered on the display.

4   Adjust the horizontal POSITION so that a rising edge is triggered at the center of the display.

5   Adjust the horizontal SCALE so that the differences in the channel delays are clearly visible.

6   Adjust the horizontal POSITION again so that the first rising edge is exactly at the center of the display. The fastest probe is connected to this channel (the fastest probe is usually the one with the shortest cable or with the highest bandwidth).

7   Select Deskew from the Vertical menu to open the Deskew control window.

8   Select one of the slower channels.

9   Adjust the deskew time for the slower channel so that its signal aligns with that of the fastest channel. The deskew adjustment range is ±75 ns.

10   Repeat steps 8 and 9 for each additional channel that you want to deskew.

11   Remove the connections from the probe compensation terminals.

Note

To compensate for timing differences between voltage and current probes, order the Probe Calibration and Deskew Fixture (067-0405-02). The fixture requires a user-supplied pulse generator when used with a TDS5000B Series instrument.

Signal Path Compensation

Use this procedure to compensate the internal signal acquisition. You should perform this procedure if the temperature has changed more than 5° C since the last signal path compensation was performed.

Note

When making measurements at vertical scale settings less than or equal to 5 mV, you should perform the signal path compensation at least once a week. Failure to do so may result in the instrument not meeting warranted performance levels at those volts/div settings.

1   Power on the instrument and warm up for 20 minutes before continuing with this procedure.

2   Disconnect any probes that you have connected to the input channels.

3   Set the instrument to Menu mode.

4   Select Instrument Calibration from the Utilities menu.

5   Note any instructions that appear under Calibration Instructions in the resulting control window.

6   Click Calibrate to begin the procedure. The procedure may take several minutes to complete.

7   Verify that the Status changes to Pass after the procedure is complete. If the Calibration Status field indicates anything other than Pass, see Signal Path Compensation Status for information on the readout and recommended action.

Signal Path Compensation Status

The following table lists the possible calibration status messages for signal path compensation and the recommended action.

Status	Meaning and recommended action
Pass	The instrument had warmed up for 20 minutes, there were no failures during the signal path compensation procedure, and the current operating temperature is within 5° C of the operating temperature when the previous signal path compensation was run.
Fail	One or more failures occurred during the signal path compensation procedure. Contact your local Tektronix service representative for further action.
Temp	The operating temperature is not within 5° C of the previous signal path compensation.
Warmup	The instrument has not warmed up for 20 minutes. Wait for the instrument to warm up for at least 20 minutes before running the signal path compensation procedure.

Low Frequency Probe Compensation

Use this procedure to compensate passive probes to ensure maximum distortion-free inputs to the instrument.

1   If you have not already done so, connect the probe to channel 1 on the instrument.

2   Attach the probe tip and reference lead to the probe compensation terminals as shown.

3   Press the AUTOSET button on the instrument.

4   Adjust the probe until you see a square wave with a perfectly flat top on the display. See your probe manual for adjustment instructions.

Taking Cursor Measurements

Use this procedure to take measurements with cursors.

1   To open the Cursor control window, press the Cursors button on the front panel, click Cursors on the toolbar, or click Cursor Setup from the Cursors menu.

2   Select the waveform that you want to measure and a Cursor Type from the control window.

3   Position the cursors with the multipurpose knobs.

4   Read the cursor information from the Position controls.

Tip Select the cursor mode (Tracking or Independent) in the Cursor Setup control window.

Creating Reference Waveforms

Use this procedure to create reference waveforms from active or math waveforms.

1   Click Save on the toolbar or click Save As from File menu.

2   Select Waveform from the Save As dialog box.

3   From the Source list, select the waveform that you want to save as a reference.

4   Click Ref(n) under Save in Oscilloscope Memory to save the waveform to nonvolatile memory. Or select a location from the Save in list to save the waveform to a file, and then use the pop-up keyboard to enter a meaningful name for the waveform in the Name text box.

5   Click Save.

Note

When you save a waveform as a reference waveform, a file containing the waveform data is created in **C:\TekScope\Waveforms**. The file is named **TDS_REF*n*.WFM**, where *n* is the reference waveform number. Do not manually delete this file because the data will no longer be available.

Saving Reference Waveforms using Auto-Increment File Name

Use this procedure to repetitively save waveforms without typing in a large number of unique file names.

1   Click Save on the toolbar or click Save As from File menu.

2   Select Waveform from the Save As dialog box.

3   From the Source list, select the waveform that you want to save as a reference.

4   Select Auto-increment file name in the Save As dialog box.

5   Enter a descriptive name in the Base file name text box. For example, if you are saving a series of rise time data files, you can use "Risetime" as the base file name.

6   The Count text box defaults to "000". Change this if you want to start saving your series of files with another 3-digit number.

7   Change the file type if necessary.

8   Click Save.

Your first file is saved as Risetime000.*ext* , where *ext* is the file type extension. The next file will be Risetime001.*ext* , and so on.

If Count reaches 999, it is suggested that you change the base file name to, for example, Risetime1on the next save. Your next file will then be saved as Risetime1000.*ext*. You will receive a warning message if you are going to overwrite any existing files with the same name.

Tip Use the auto-increment file name feature to save a series of setups, exported images, waveforms, and measurements.

Recalling Reference Waveforms

Use this procedure to recall a reference waveform from a file.

1   Click Recall on the toolbar or select Recall from the File menu.

2   Select Waveform from the Recall dialog box.

3   Select the waveform that you want to recall and the Destination (Ref 1–4).

4   Click Recall to load the waveform into memory.

You can now use the reference waveforms for measurements, math functions, or other uses. Simply select the reference waveform from the Channel Selector tabs on the individual control windows.

Note

When you recall a reference waveform from a file, a new file containing waveform data is created in **C:\TekScope\Waveforms**. The file is named **TDS_REF*n*.WFM**, where *n* is the reference waveform number. This file is deleted when the reference waveform is deleted.

Recalling a Template Waveform

Use the procedure to recall a template waveform to a reference location.

1   From the File menu click Recall….

2   Under Recall What: click Waveform, and then select the waveform that you want to recall.

3   Select Ref1–4 from the Destination list.

4   Click Recall.

Creating a Spectral Waveform

Use these steps to display a spectral (FFT magnitude) waveform.

1   Set the source signal Vertical SCALE so that the signal peaks do not go off screen. Off-screen signal peaks can result in FFT waveform errors.

2   Set the Horizontal SCALE control to show five or more cycles of the source signal. Showing more cycles means the FFT waveform can show more frequency components, provide better frequency resolution, and reduce aliasing.

3   If the signal is a single-shot (transient) signal, make sure that the entire signal (transient event and ringing or noise) is displayed and centered on the screen.

4   Click Math on the toolbar, or click Math Setup from the Math menu to open the Math Setup control window.

5   Click Setup under the Spectral Analysis section in the control window.

6   Click the Magnitude button to insert the magnitude function into the math expression.

7   Click the channel that is related to the time-domain signal that you want to view as an FFT waveform. The respective channel is inserted into the math expression.

8   Select the appropriate vertical scale and FFT window type.

Setting up for FastFrame Operations

Follow these steps to use FastFrame mode:

1 Set up the Trigger control window to trigger on the signal of interest.

2 Adjust the horizontal and vertical controls to display a waveform without any clipping.

3 Adjust the Resolution controls to set the number of samples you want to acquire with each acquisition.

4 From the Horiz/Acq menu, select FastFrame Setup, and then enter the number of frames that you want to capture in Frame Count.

5 Select the type of Summary Frame from the list. Selecting Average or Envelope from the Summary Frame list replaces the last frame with one that is the computed average or envelope of all frames in the set.

6 Click the **FastFrame** button to turn on the FastFrame mode.

7 After the acquisition is complete, select the frame that you want to view under the Frame Viewing section of the FastFrame Setup control window.

8 If you want to view multiple frames overlaid, select **Overlay** from the Multiple Frames list**.**

9 Click the **Start Frame** field that is displayed, and then use the upper multipurpose knob to enter the Start Frame (first frame) in the frame sequence that you want to view.

10 Click the **# of Frames** field, and then use the lower multipurpose knob to enter the total number of frames that you want to view.

11 The selected range of frames is displayed with the frames superimposed over each other in the color of the selected Source, and the current (Selected Frame) is displayed superimposed in dark blue.

Taking Automatic Measurements

Use these steps to take automatic measurements. This procedure assumes that you are using the default reference levels.

1 Open the Measurement Setup control window by clicking Meas on the toolbar or by clicking Measurement Setup on the Measure menu.

2 Select the source waveform using the Channel Selector tabs, or press one of the front panel buttons.

3 Select the desired measurements from the measurement tabs. Each measurement is automatically added to the measurement list.

The measurement is added to the display in the same color as the selected channel waveform. When no control windows are open, the measurements are listed on the right side of the graticule.

Tip You can select an automatic measurement from the menu bar Measure menu by selecting the measurement type (for example, Amplitude) and clicking the measurement. The measurement is taken on the currently selected waveform.

Setting up a Dual Display

Use the following steps to set up the instrument for dual display operation. You can use the main display for most instrument operations while you use a second monitor to access other PC applications that you may have installed.

You can also drag up to five dockable control windows from the TekScope application onto the display of the second monitor. For example, this feature allows you to have the Horizontal/Acquisition, the Horizontal Zoom, and the Display Setup Control Windows (half screen) open on the second monitor at the same time. In addition to the five dockable windows on the second monitor, you can have one dockable window open on the main display. If you open a seventh dockable window, whichever window has not been used for the longest amount of time will automatically close.

Tip You may want to print these instructions, since the operation requires you to power off the instrument.

1   Power off the instrument.

2   Connect an external monitor to the Video port (upper VGA port) on the side panel of the instrument.

3   Power on the instrument and the external monitor.

4   After the instrument display appears, click Minimize on the File menu or press CTRL+M to minimize the instrument application.
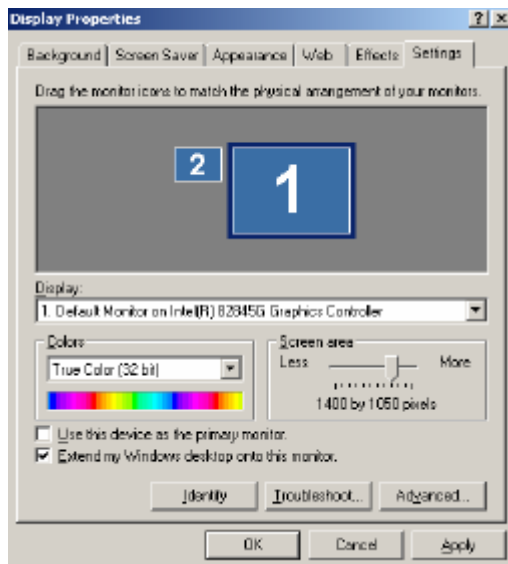
5   Right-click on the desktop and select **Properties** to open the Display Properties dialog box.

6   Click the **Settings** tab, and then click the secondary display monitor (the one on the right) in the dialog box.

7   Select the **Extend my Windows desktop onto this monitor** checkbox.

8   Ensure that the color is set to 16 bit color.

Caution

Do not change the display settings for the internal monitor. The display settings must be 640 X 480 pixels. The colors must be set to 16 bit color. Ensure that the second monitor is also set to 16 bit color.

If you use other settings, the video port may not work properly. You may however, use a resolution setting other than 640 X 480 on the second monitor.

9   Click OK to apply the settings. The new monitor displays additional desktop area.

You can now drag any item except for the instrument application from the desktop of the instrument (including the task bar) to the new monitor

Note

To disable the external monitor, click the **Settings** tab in the Display Properties dialog box. Click the external monitor and clear the **Extend my Windows desktop onto this monitor** checkbox at the bottom of the dialog box.

Creating Math Waveforms using Predefined Expressions

Use the following steps to create math waveforms using predefined expressions:

1   Open the Math Setup control window by clicking **Math** on the toolbar or by clicking **Math Setup** in the Math window.

2   Select a Math waveform by clicking one of the tabs in the control window.

3   Click one of the **Predefined Expression** buttons in the control window. The predefined expression is added to the math expression at the top of the control window and the math waveform appears on the screen.

Tip You can select Predefined Expressions directly from the Math menu.

Creating Math Waveforms with the Equation Editor

Use the following steps to create math waveforms using the Equation Editor.

1   Open the Math Setup control window by clicking **Math** on the toolbar or by clicking **Math Setup** in the Math window.

2   Click **Editor** under the Define/Edit Expression section of the Math Setup control window to open the Equation Editor.

3   Select a Math waveform from the drop-down list.

4   Select a math function by clicking a button under the tabs on the left side of the Equation Editor (for example, click Derivative).

5   Select a valid waveform to use as an operand from the Channel Selector tabs.

6   Use other buttons in the Equation Editor as necessary to create the math expression for the math waveform.

7   Click **OK** to create and display the math waveform.

Exiting the Instrument Application

On the **File** menu, click **Exit** to exit the instrument application.

To restart the instrument application, launch the TekScope icon from the Windows taskbar, Start menu, or your desktop.

Minimizing the Instrument Application

To minimize the instrument application so that you can use other programs on the Windows interface, select **Minimize** from the File menu or right-click anywhere in the graticule and select **Minimize** from the shortcut menu.

To restore the application, click the application name on the Windows task bar.

Shutting Down the Instrument

When you push the front-panel On/Standby switch, the instrument starts a shutdown process (including a Windows shutdown) to preserve settings and then go into standby mode. Avoid disconnecting the power cord to shut down the instrument.

Note

It may be necessary to hold the On/Standby switch for up to 20 seconds for the shutdown process to initiate.

To completely remove power to the instrument, perform the shutdown process just described and disconnect the power cord from the power source.

Adding Software Applications to the Instrument

Because the instrument is an open Windows platform, you can easily install and remove additional software applications. Follow the instructions that came with your software and the basic Windows instructions to install and remove your application.

Note

When adding software to the instrument, avoid making changes to system files. See Windows Interface on page 38 for more information.

Creating an Emergency Startup Disk

Now that you have completed the basic installation process, you should create an emergency startup disk that you can use to restart your instrument in case of a major hardware or software failure. Store this disk in a safe place.

CAUTION

Create this disk and store it in a safe place. It may allow you to recover your Windows 2000 installation without rebuilding the entire instrument hard disk.

The emergency startup disk contains basic files to restart your instrument. The disk also contains files to check and format the hard disk.

Follow these steps to create the emergency startup disk:

1   Minimize the instrument application by selecting Minimize in the File menu.
2   From the Windows taskbar, select **Start**, **Programs**, **Accessories**, **System Tools**, **Backup**.
3   In the Welcome display, click **Emergency Repair Disk**.
4   Insert a floppy disk into the disk drive, and then follow the on-screen instructions to create the startup disk.

Backing Up User Files

Always back up your user files on a regular basis. Use the Windows Backup tool to back up files stored on the hard disk. The Backup tool is located in the System Tools folder in the Accessories folder.

Use these steps to back up your files:

1   Exit the instrument application by selecting the File menu Exit command.
2   Click the Windows **Start** button.
3   Select Programs, Accessories, System Tools, Backup.
4   Use the Backup Wizard to select the items you want to back up and the destination. You can back up to the floppy drive or to a third-party storage device over one of the side-panel ports.

Using Roll Mode

Use Roll mode when you want to view data at very slow sweep speeds. It is useful for observing data samples on the screen as they occur.

Roll mode turns on automatically at slower sweep speeds. You may want to turn Roll mode off to avoid conflicts with other instrument settings. To turn Roll mode off, deselect **Roll Mode Auto** on the Horiz/Acq menu.

TDS5000B Series GPIB Programmer Online Help

You can control the instrument via GPIB from a remote PC or via an Internet or Intranet connection. The GPIB programming information is available as a separate WinHelp file that you can access from the Help menu **GPIB Online** command.

The help file is also available as a PDF file on the TDS5000B Series Product Software CD-ROM shipped with the instrument. Refer to the CD-ROM instructions for installation information.

TDS5000B Series Oscilloscope Applications Software

You can install and run optional application software packages on the instrument. These software packages provide advanced capabilities that support several applications such as jitter analysis and disk drive measurements. Each application can be purchased and installed on the instrument at any time. Contact your Tektronix representative for purchasing information. Follow the installation instructions that come with the application.

Run the application software from the File menu. From the **File** menu, select the application from the **Run Application** menu item.

Application software packages have their own online help that is separate from the main instrument online help.

Trigger on A Event Only

Use this trigger sequence to trigger the instrument after the A event has occurred.

1   Set the A trigger type and source from the A Event tab in the Trigger Setup control window.
2   Click the A->B Seq tab, and click A Only.

Triggering on an Event after a Specified Delay

Use this trigger sequence to trigger the instrument on the B event after a specified amount of time after the A event.

1   Set the A trigger type and source from the A Event tab in the Trigger Setup control window.
2   Set the B trigger type and source from the B Event tab.
3   Click the A->B Seq tab, and click Trig AfterTime.
4   Click the Trigger Delay control, and then enter the delay time between the A and B events.

Triggering on a B Event

Use this trigger sequence to trigger the instrument after a specified number of B events.

1   Set the A trigger type and source from the A Event tab in the Trigger Setup control window.

2   Set the B trigger type and source from the B Event tab.

3   Select the A->B Seq tab, and then click Trig on nth Event.

4   Click the Trig Event control, and then enter the number of B events that should occur before starting the acquisition after the A event.

Setting up a Histogram

Use these steps to set up a histogram.

1   Click Waveform Histograms on the Measure menu.

2   Use the Channel Selector tabs to select the waveform, and then use the other instrument controls to position the waveform in the center of the screen.

3   In the Histogram Mode section, click either Horiz or Vert to turn on the histogram.

4   Click Location under Adjust Histogram Box, and then use the multipurpose knobs to position the histogram box over the area of interest on the waveform.

5   Click Limits button under the Adjust Histogram Box, and then use the multipurpose knobs to adjust the size of the histogram box.

6   Select the type of scaling by clicking either Linear or Log under the Histogram Scaling.

Tip Click the waveform handle to select the waveform (the waveform is selected if the waveform handle appears in reverse video). Hold the left mouse button and drag across the segment of the waveform that you want the histogram to cover and then select **Histogram** from the pop-up list on the screen.

To turn off the histogram, click and drag a box across the screen and select **Histogram Off** or right-click on a histogram box and select **Turn Histogram Off** from the shortcut menu.

Taking Automated Measurements on Histogram Data

Use the following steps to take automatic measurements on histogram data.

1   Set up a histogram box over the waveform on the screen.

2   Open the Measurement Setup control window and select the Histogram tab.

3   Click the histogram measurement buttons to add the measurements to the list.

Printing the Screen to a Printer

The easiest way to print the contents of the screen is to press the front panel PRINT button. The instrument will print the screen contents according to the settings in the Page Setup dialog.

1   Use the File menu Page Setup dialog to select the **Type of printing**, **Palette**, **View**, and **Image** before printing. The Type should be **Screen-copy**, the default.

2   Use the **Print Preview** feature to test the appearance of the screen before printing. For example, you may want to experiment with the different palettes before sending the data to the printer.

3   Click **Print** in either the Print Preview dialog box or in the Page Setup dialog box to open the Print dialog box.

4   Click **OK** to begin printing.

Copying the Screen for Use in other Applications

Use the following steps to copy the contents of the screen for use in other applications such as WordPad, Microsoft Word, or Paint.

1   Click **Copy Setup** on the Edit menu to open the Copy Setup dialog box.

2   Click the **Images** tab.

3   Select the Palette, View, and Image as needed for your application.

4   Click **Copy** to paste the image to the clipboard. You can now paste the bitmap image into your other applications.

5   If you want to copy another image to the clipboard, click **Copy** on the Edit menu or press **Ctrl+C** on the attached keyboard.

Note

If you try saving a copy of the screen to the clipboard by pressing the **PrtScn** button on the attached keyboard, the bitmap image will not contain any waveforms. Use the procedure above to save a copy of the entire screen (or the graticule) to the clipboard.

Tip If your setup information does not change, you do not need to open the Copy Setup dialog box each time you want to copy. Use the Edit menu **Select for Copy** option if you need to change the type of items, for example, to copy waveforms instead of images.

Copying Measurements for Use in Other Applications

Use the following steps to copy the measurements for use in other applications such as WordPad or Microsoft Word.

1   Click Copy Setup from the Edit menu to open the Copy Setup dialog box.

2   Click the Measurements tab.

3   Select the Data Format, either Numeric or Text.

4   Select either Displayed Measurements (make sure that measurements are displayed in the graticule via the Measurement Setup control window) or Measurement Snapshot (to copy measurement snapshots).

5   Click Copy to paste the measurements to the clipboard. You can now paste the measurements into your other applications.

Saving Instrument Setups

Use this procedure to save your instrument setups.

1   Use the instrument controls to set up the instrument for your application.

2   Select Save As…from the File menu.

3   Click the Setup icon.

4   To save to oscilloscope memory, select one of the 10 locations under Save in Oscilloscope Memory. Give the location a meaningful label and click Save. Click OK when asked to Overwrite instrument setup.

5   To save to a file, select a location from the Save in drop-down list. Then enter a name for the setup, use the default name, or select Auto-increment file name if you want to save a series of files without typing in a new name each time.

Note

The data in reference waveforms is not saved with instrument setups. However, the setup information includes which reference waveforms are turned on and which are used with math waveforms and automatic measurements. If you want to save the reference waveform data, save the reference waveform separate from the instrument setup.

Related topic

Creating Reference Waveforms see page 354.

Recalling Instrument Setups

Use this procedure to recall a saved instrument setup.

1   Select Recall… from the File menu.

2   Click the Setup icon on the Recall dialog box.

3   Select the location where you saved the file from the Look in drop-down list. Then select the setup, and click Recall.

Note

The data in reference waveforms is not saved with instrument setups. However, the setup information does include which reference waveforms are turned on and which are used with math waveforms and automatic measurements. If you want to save the reference waveform data, save the reference waveform separate from the instrument setup. You can then recall the saved waveform to the restored setup.

Related topic

Recalling Reference Waveforms see page 355

Deleting Instrument Setups

Use this procedure to delete an instrument setup saved to oscilloscope memory.

1   Select the Delete submenu from the File menu.

2   Then select the Setup that you want to delete, or select All Setups.

3   Use this procedure to delete an instrument setup that you saved to a Windows directory.

4   Select Recall from the File menu.

5   Click the Setup icon and then select the directory where the setup was saved from the Look in drop-down list.

6   Select the setup that you want to delete, and then click the ✕ next to the directory. The setup is deleted.

Choosing Color Palettes

Use the following information to help you select a color palette.

1   Click Colors on the Display menu, or click Disp on the toolbar, and then click the Colors tab.

2   Click one of the color palettes from the Record View or FastAcq/WfmDB palettes for your application based on the following information:

▪   Use the **Normal** palette to display hues and lightness levels for best overall viewing.

▪   Use the **Green** palette to display waveforms and the graticule style in shades of green. The **Green** palette simulates the display of an analog oscilloscope. Areas of the waveform with the highest sample density appear in lighter shades while the areas of lower sample density appear in darker shades.

▪   Use the **Temp** (temperature) palette to display variable persistence style waveforms. Areas of the waveform with the highest sample density appear in warmer colors (red shades), while areas of the waveform with lower sample density appear in cooler colors (blue shades).

▪   Use the **Spectral** palette to display variable persistence style waveforms. Areas of the waveform with the highest sample density appear in blue shades, while areas of the waveform with lower sample density appear in red shades.

▪   Use the **Gray** palette to display waveforms and the graticule in shades of gray. Areas of the waveform with the highest sample density appear in lighter gray shades, while areas of the waveform with lower sample density appear in darker gray shades.

▪   Use the **User** palette to select the color previously defined or to assign a new color to the User palette.

Note

To define a color for the User Palette, follow the steps to Customize Color Palettes.

Customizing Color Palettes

1   Select the User color palette.

2   Touch the Edit… button under User.

3   Touch the Hue, Lightness, or Saturation controls, and use the pop-up keyboard to adjust these attributes.

Setting the Horizontal Delay

Use the following steps to set up the horizontal delay mode of the instrument.

1   Open the Horizontal/Acquisition Setup control window.

2   Turn on the Horizontal Delay mode by clicking On under Delay Mode or pressing the front panel DELAY button.

3   Use the Horiz Delay control to set the delay time. To view data before the trigger event, enter a negative value; to view data after the trigger event, enter a positive value.

Tip You can also use the horizontal delay with complex trigger events in the Trigger Setup control window by clicking the A->B Seq tab and configuring horizontal delay.

Comparing Data Using Horizontal Delay

After setting up the instrument to use the horizontal delay, press the front panel DELAY button to turn the horizontal delay on and off. This lets you quickly compare two segments of a signal that are separated from each other by a fixed amount of time.

Using the XY Display Format

Use the following steps to set up the instrument in XY Display format.

1   Determine the waveform pairs that you want to use.

2   Click Appearance on the Display menu or click Disp on the toolbar, and then click the Appearance tab.

3   Click the center button under Display Format to select XY Display mode.

4   Use the vertical scale and position controls of the waveform connected to the X-axis source to set horizontal scale and position of the XY waveform.

5   Use the vertical scale and position controls of the waveform connected to the Y-axis source to set vertical scale and position of the XY waveform.

Using the XYZ Display Format

Use the following steps to set up the instrument in the XYZ Display format.

1   Connect the Ch 1 probe to the signal that you want to use for the X-axis.

2   Connect the Ch 2 probe to the signal that you want to use for the Y-axis.

3   Connect the Ch 3 probe to the signal that you want to use for the Z-axis. This signal will modulate the waveform intensity.

4   Click Appearance on the Display menu, or click Disp on the Tool bar and then click the Appearance tab.

5   Click the bottom button under Display Format to select XYZ Display mode.

6   Adjust the vertical scale and position controls to obtain the best display of the XYZ waveform.

Note

The XYZ format is only available on 4-channel models.

Enabling a Network Connection

Use the following steps to enable network access to the instrument. This procedure requires that you begin with the instrument powered off, so you may want to print these instructions before you proceed.

1 Attach a keyboard and mouse to the instrument.

2 Power on the instrument.

3 Use the Windows Network Connection Wizard to define the instrument as a network client and configure it for your network. You can find the network setup program in the Windows **Start** menu if you select **Settings > Control Panel**, and then double-click **Network**. Consult with your network administrator for specific instructions for these settings.

Note

If you want to disable network access for the instrument, select **Start>Settings>Control Panel>Network and Dialup Connections>Local Area Connection.** Click **Disable** and **Close**.

Selected Waveform versus Deselected Waveform
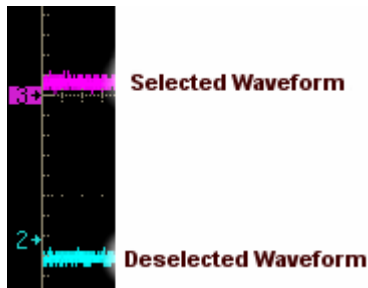
The waveform handle determines if a waveform is selected or deselected. The waveform handle appears in reverse video when the waveform is selected as shown in the graphic for the Channel 3 waveform.

Probe Compensation Adjustment

Probe Compensation

Probe compensated correctly

Probe overcompensated

Probe undercompensated
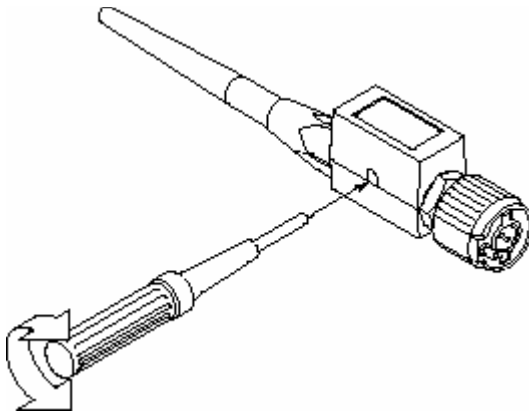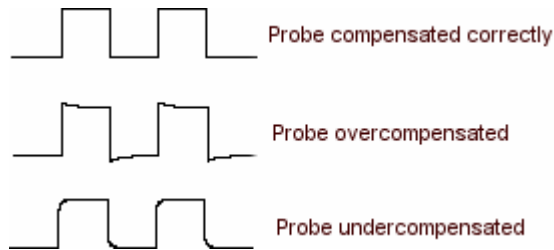
Printing Banners

Use the following steps to print banners. Banners can only be printed on the integrated thermal printer.

1   Open the Print dialog box from the File menu Print command. From the Name drop-down list, select Integrated Thermal Printer (the default). Click OK.

2   Open the File menu Page Setup dialog box. For the Type of printing, select Banner. This locks in values for the Orientation, View, and Image controls.

3   For the Print Scale controls, select the Units you want to use to specify the horizontal scale, either inches/division or centimeters/division. Select a value for the Length; these values change according to the choice of Units.

4   Change the Summary Waveform selection if you want to add a screen shot to the Beginning, End, or Both ends of the banner. The Summary Waveform is a screen shot that gives you an overall view of what was originally on the screen before printing began. It contains information that will not appear on the banner such as cursor and measurement readouts, histograms, and any screen text or labels that you have added.

5   Select the Samples to Print to specify which portion of the waveform to print. If you select Between cursors and no cursors are present, vertical cursors are activated in their most recent position. You can use the multipurpose knobs to change the cursor positions.

6   Click Print to open the Print dialog box.

7   Click OK to begin printing. If the banner you produce is unreadable because it is too light, open the printer cover and push the lever on the right side of the printer towards the back of the instrument as far as it will go.

Note

Banner printing is not compatible with:

▪   Fast Acquisition mode

▪   Persistence display mode

▪   XY or XYZ mode

▪   Zoom mode

Inverting Channel Waveforms

Use the following steps to invert a channel waveform on the screen.

1   Select the Invert command from the Vertical menu.

2   From the Channel Selector tabs, click the waveform that you want to invert.

3   Click Invert to toggle the button On. The waveform appears flipped on its zero volt axis. Other waveforms on the screen remain unchanged.

💡**Tip** Invert the waveform by clicking the **Invert** button on the channel's tab on the Vertical Setup control window or by right-clicking on the waveform handle, and then selecting **Invert** from the shortcut menu.

Connecting Probes
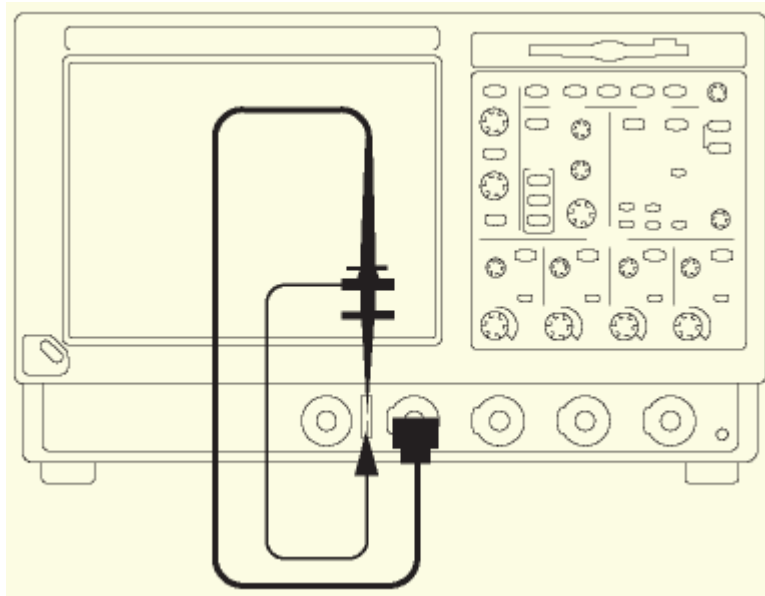
Connect the probe tip and the ground lead to the two probe compensation connectors on the instrument as shown. If using the probe hook-tip, ensure a proper connection by firmly twisting the tip onto the probe.

Adjusting Display Contrast

If you find that the instrument screen is too bright or too dark, use these steps to adjust the LCD display contrast to your preference. You need to have a mouse attached to perform this procedure.

1   Right-click on the Windows desktop.

2   Select Properties.

3   Select the Settings tab.

4   Click Advanced.

5   Select the Color tab.

6   Adjust slider controls for desired contrast.

7   Click Apply, and then click OK.

Using Mask Testing with Option SM

Follow these steps to perform mask testing.

1   Select Mask Setup… from the Masks menu or touch the Masks button on the toolbar to display the Mask Setup control window.

2   From the Masks tab, click the type of mask that you want to use, and then click the Standard drop-down list to select the standard that you want to use.

Note

The standard that you choose will determine whether you also need to set the polarity for the test.

3   Make any necessary changes to the Display or Alignment configurations.

4   Click Hit Count to turn it off or on.

5   From the Source tab, click the input channel that you want to test.

6   From the Tolerance tab, make adjustments in the Mask Margin Tolerance.

7   Click the Pass/Fail Setup tab, and then set the rest of the test parameters.

8   From the Pass/Fail Setup tab or the Pass/Fail Results tab, click the Pass/Fail Test control to run the test.

Creating a User Mask with Option SM

Use the Edit control to change a standard mask and create your own.

1   Select Mask Setup… from the Masks menu, or click the Masks button on the toolbar to display the Mask Setup control window.

2   From the Masks tab, select the type of mask and standard that you want to use as a basis for your mask.

3   Click User Mask, and then click Copy Current Mask to User Mask.

4   Click OK. The mask that you selected is now a user mask. To alter the mask, click Edit User Mask.

5   Under Mask, use the up and down arrows to select the segment and vertex that you want to edit. The selected segment appears in red and a white X indicates the selected vertex.

6   Click the Horizontal or Vertical control under Mask Values, and use the multipurpose knobs to change the horizontal or vertical values of the mask.

7   Click Save to save your changes as the User Mask.

Saving Masks

Use the Mask Edit Setup control window to save a mask.

1   Select Mask Edit Setup… from the Masks menu to display the Mask Edit Setup control window.

2   Click Save to open the Save As dialog box.

3   Select User Mask under Save What. You may need to click More to access the User Mask selection.

4   Enter a file name in the Name box. If you do not have a keyboard attached, click the keyboard icon to use the pop-up keyboard to enter the file name.

5   Click Save to save the mask. The default folder for saving masks is C:\TekScope\Masks. If you want to save the mask in a different folder, click the Save in drop-down list, and then click a folder to select it.

Recalling Masks

Use the Mask Edit Setup control window to recall a mask.

1   Select Mask Edit Setup… from the Masks menu to display the Mask Edit Setup control window.

2   Click Recall to open the Recall dialog box.

3   Click User Mask under Recall What, and then click the mask that you want to recall to select it.

4   Click Recall to recall the mask.

Note

The default folder for saving masks is **C:\TekScope\Masks**. If you saved masks in a different folder, click the **Look in** drop-down list to find the folder where your masks are saved.

Editing a User Mask

Use the Mask Edit Setup to alter a user mask.

1   Select Mask Edit Setup… from the Masks menu to display the Mask Edit Setup control window.

2   Under Mask use the up and down arrows to select the segment and vertex that you want to edit. The selected segment appears in red and a white X indicates the selected vertex.

3   Click the Horizontal and/or Vertical control under Mask Values to change the horizontal or vertical values of the mask.

4   Click Save to save your changes to the user mask.

Glossary

Absolute Value

Inserts the text FABS(into the math expression. The FABS function takes the absolute value of the expression.

ACos

Inserts the text ACOS(into the math expression.

Acquisition Interval

The acquisition interval is the time between sample points. For mathematical purposes, it is the time covered by the waveform record divided by the record length.

Acquisition Window

This is the window that defines what data is "cut out" of the input signal to form a waveform record. The vertical scale and offset set the vertical size and the vertical center of this window independently for each channel. The horizontal scale and position set the horizontal size and positions for all channels in parallel.

Amplitude

This voltage measurement is the high value minus the low value measured over the entire waveform or gated region.

Amplitude = High - Low

Analog Oscilloscope Sweeps
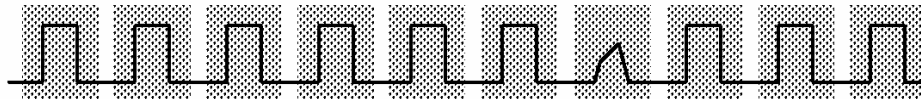
Area

Area is a voltage-over-time measurement. The measurement is the area over the entire waveform or gated region expressed in volt-seconds. The area above the common reference point is positive while the area below the common reference point is negative.

ASin

Inserts the text ASIN(into the math expression.

ATan

Inserts the text ATAN(into the math expression.

Auto Trigger Mode

Auto trigger mode causes the instrument to trigger if a valid trigger event does not occur within a set amount of time. Auto trigger mode is useful when probing DC signal levels such as power supplies or logic levels that are stuck high or low. It is also useful when looking for activities on

signal nodes such as oscillations when you first set up the instrument. The length of time it waits for a trigger event depends on the time base setting.

Average

Inserts the text AVG( ) into the math expression. Enter an argument to the function. The average function produces the average of the argument.

Averages Button

Closes this window and opens the Adjust Math Parameters control window.

Base 10 Logarithm

Inserts the text LOG( ) into the math expression. Enter an argument to the function. The log function produces the base 10 logarithm of the argument.

Burst Width

This timing measurement is the duration of a burst (a series of transient events) and is measured over the entire waveform or gated region.

Center Frequency

The center frequency is the frequency at the center of the frequency span. The maximum center frequency is equal to the sample rate divided by two (the Nyquist frequency) minus one half the span. The minimum center frequency is equal to one half of the span.

If the frequency span is set to maximum, then the center frequency will not move. To increase the center frequency beyond the present limits, either reduce the frequency span or increase the sample rate. To decrease the center frequency beyond the present limits, reduce the frequency span.

Clear Button

Removes all of the characters from the math expression.

Channel Selector
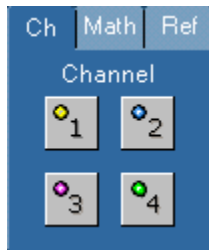
The Ch tab contains the controls for each waveform connected directly to the channel inputs.

The Math tab contains the controls for all math waveforms.

The Ref tab contains the controls for all reference waveforms.

Shaded buttons indicate unused or undefined waveforms.

Note

Channels 3 and 4 for channel, math, and reference waveforms are only available on 4-channel models.

Clock Edge

The Clock Edge determines whether the instrument finds the trigger point on the rising edge (left button) or the falling edge (right button) of the clock signal.

Clock Source Channel

Select the input channel that is connected to the clock signal.

Clock Threshold Level

To set the clock threshold voltage level, click Clock Level, and then use the multipurpose knob to set the voltage level.

Comm Trigger Source

Use the Source drop-down list to select the channel for the comm trigger source. Use the Type drop-down list to select the clock source for the trigger. Use the Polarity drop-down list to select the polarity of the clock source.

Control Window Handle

Cos

Inserts the text COS(into the math expression.

Cosh

Inserts the text COSH(into the math expression. The COSH function computes the hyperbolic cosine of x

Crossing Percent

Crossing percent measurement is the eye crossing point expressed as a percentage of eye height. Crossing Percent = 100 x [(pCross1Mean - pBaseMean)/(pTopMean - pBaseMean)]

Cycle Area

The cycle area is a voltage over time measurement. The measurement is the area over the first cycle in the waveform or the first cycle in the gated region expressed in volt-seconds. The area above the common reference point is positive while the area below the common reference point is negative.

374

Cycle Mean

This voltage measurement is the arithmetic mean over the first cycle in the waveform or the first cycle in the gated region.

Cycle RMS

This voltage measurement is the true Root Mean Square voltage over the first cycle in the waveform or the first cycle in the gated region.

Data Source Channel

Select the input channel that is connected to the data signal.

Data Threshold Level

To set the data threshold voltage level, click Data Level, and use the multipurpose knob to set the voltage level.

Dead Time

Dead time is the time that the instrument requires to process an acquisition. The instrument cannot acquire any new data until the previous data has been processed. If dead time is long, the instrument can miss acquiring infrequent events.

"Dead time" Between Acquisitions
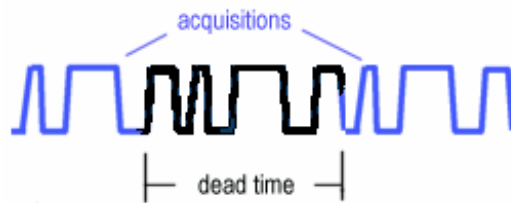
Delay

This timing measurement is the time between the mid reference (default 50%) amplitude point of two different waveforms.

Delay Edges

The Delay Edge buttons determine whether the measurements will be taken on the rising or falling edge of the waveform. After selecting an edge, the graphic in the control window is updated to show the edge from which the measurements will be taken.

Derivative

Inserts the text DIFF( ) into the math expression. Enter an argument to the function. The derivative function produces the derivative of the argument.

Duty Cycle Distortion

Duty cycle distortion is the peak-to-peak time variation of the first eye crossing measured at the MidRef as a percent of the eye period.
DCD(sec) = 100% x TDCDp-p/(TCross2mean – TCross1mean)

Editor Button

Closes this window and opens the Equation Editor control window so you can edit the math expression.

E-mail on Trigger

To send e-mails on a trigger event, click the E-mail on Trigger button On. E-mail on Event must be configured to enable E-mail on Trigger from the Trigger Mode control window.

Email on Trigger Setup

Click to access the E-mail on Event control window where you configure sending e-mail on events.

Extinction Ratio

Extinction ratio is the ratio of eye top to base. This measurement is only valid when the acquisition mode is set to WfmDB, FastAcq is enabled, or the measurement is taken on a reference waveform saved in FastAcq mode.
Ext Ratio=pTopMean/pBaseMean.

Extinction Ratio Decibel

Extinction ratio decibel is the ratio of eye top to base expressed in decibels. This measurement is only valid when the acquisition mode is set to WfmDB, FastAcq is enabled, or the measurement is taken on a reference waveform saved in FastAcq mode.
ExtdB=10xLog(pTopMean/pBaseMean)

Extinction Ratio Percent

Extinction ratio percent is the ratio of eye base to top expressed as a percentage. This measurement is only valid when the acquisition mode is set to WfmDB, FastAcq is enabled, or the measurement is taken on a reference waveform saved in FastAcq mode.
Ext Percent=100x(pTopMean/pBaseMean).

Eye Base

Eye Base is the base value used in the extinction ratio measurements.
Eye Base = pBaseMean

Eye diagram

A synchronized display of a data stream that shows signal levels and transition regions, typically over one cycle. The eye diagram has the appearance of an opened eye. Compared to a normal instrument display, the eye diagram generally appears as an untriggered waveform. This is because on successive cycles the signal is high and then low. The areas of interest are the transition regions and eye opening.

Eye Height

Eye height is the measurement of the eye height in volts.
Eye Height=(pTopMean-3 x pTopSigma) - (pBaseMean + 3 x pBaseSigma)

Eye Top

Eye Top is the top value used in the extinction ratio measurements.
Eye Top = pTopMean.

Eye Width

Eye width is the measurement of eye width in seconds.
Eye Width = (tCross2Mean - 3 x pTopSigma) - (tCross1Mean + *tCross1Sigma)

Fall Time

This timing measurement is the time required for the falling edge of the first pulse in the waveform or gated region to fall from the high reference value (default = 90%) to the low reference value (default = 10%) of the final value.

Fast Acquisition
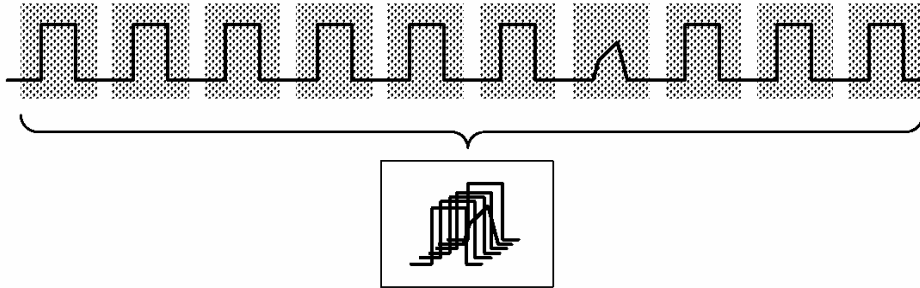
Force Trigger

Click this button to force the instrument to trigger and complete the current acquisition regardless of whether any other trigger event occurs (if the acquisition is currently running). This has no effect if the acquisition system is stopped.

Frame

A frame is a segment of the entire data record. When used with FastFrame, each segment contains all the data of a single acquisition. Each captured event is stored sequentially in its own frame. The frames are in the order they were captured.

Frequency

This timing measurement is a measure of the first cycle in a waveform or gated region. Frequency is the reciprocal of the period; it is measured in Hertz (Hz) where one Hz is one cycle per second.

Frequency Span

The frequency span is the range of frequencies over which the spectral analysis is performed. The maximum span is from DC to one half of the sample rate (the Nyquist frequency). A smaller frequency span allows a greater range of center-frequency adjustment. If a larger frequency span is desired, increase the sample rate.

The Front-Panel MultiView Zoom Button

Glitch Width

The glitch width controls specify the maximum width of the glitch. Click the Width control and set the width with the multipurpose knob. Then click the < button to trigger the instrument only on glitches narrower than the specified width; or click the > button to trigger the instrument only on pulses wider than the specified width. The trigger graphic shows the criteria that you have selected.

Graphic Indicator

High

The high level measurement is the value used as 100% whenever high reference, mid reference, or low reference values are needed, such as in fall time or rise time measurements. This value can be calculated using either the min/max or histogram method. The min/max method uses the maximum value found. The histogram method uses the most common value found above the midpoint. This value is measured over the entire waveform or gated region.

High Reference

The High Reference defines the high reference level of a waveform; the default level is 90%. This reference level is used with the Low Reference level in the calculation of rise and fall times. To change this level, click the control, and then set the level with the multipurpose knob.

Histogram Max

Displays the voltage of the highest nonzero bin in vertical histograms or the time of the rightmost nonzero bin in the horizontal histograms.

Histogram Mean

Measures the average of all acquired points within or on the histogram box.

Histogram Min

Displays the voltage of the lowest nonzero bin in vertical histograms or the time of the leftmost nonzero bin in the horizontal histograms.

Hits in Box

Displays the number of points in or on the histogram box.

Horizontal Delay

Use the Horiz Delay control to enter the amount of horizontal delay when Horizontal Delay Mode is On. Click the control and enter a delay value with the multipurpose knob. You can also enter a delay value with the front-panel HORIZONTAL POSITION knob.

Horizontal Delay Mode

Use the Horizontal Delay button to delay the acquisition relative to the trigger event. When Horizontal Delay Mode is On, use the Horiz Delay control to enter the delay time. You can also enter the delay time with the front-panel POSITION knob. The Horizontal Delay button functions identically to the Delay Mode button on the Horizontal/Acquisition Setup control window.

Horizontal Position

Use the Position control to move the trigger point within the acquired waveform. To acquire waveform data only after the trigger event, set the Position control to zero. To acquire waveform data leading up to the trigger event, set the Position control to any value greater than zero. Click the control and use the multipurpose knob or the front-panel HORIZONTAL POSITION knob to enter a value.

Horizontal Reference Marker

Imaginary Number

Inserts the text SpectralImag( ) into the math expression. Select one of the waveforms as an argument to the function. This function creates a waveform that displays only the imaginary part of the FFT.

Increment/Decrement Buttons

Integral

Inserts the text INTG( ) into the math expression. Enter an argument to the function. The integral function produces the integral of the argument.

Invert

Inserts the text INV( ) into the math expression. Enter an argument to the function. The invert function inverts the argument within the parentheses.

Jitter 6 Sigma

Jitter 6 sigma is six times the RMS value of the edge jitter in the current horizontal units.

Jitter 6 sigma = 6 x (JitterRMS)

Jitter Peak-to-Peak

Jitter p-p is the peak-to-peak value for the edge jitter in the current horizontal units.
Jitter PP = TCross1PP

Jitter Root Mean Square

Jitter RMS is the RMS value of the edge jitter in the current horizontal units.
Jitter RMS = TCross 1sigma

Largest Integer

Inserts the text FLOOR(into the math expression. The FLOOR function takes the largest integer < the expression.

Leakage

Leakage results when the time domain waveform used to create the FFT function is periodic but contains a noninteger number of waveform cycles. When the record contains a fraction of a cycle, there are discontinuities at the ends of the record. These discontinuities cause energy from each discrete frequency to leak over on to adjacent frequencies. This results in amplitude errors when measuring any given frequency. Different windows have different leakage characteristics.

Level

Click to activate the multipurpose knobs and set the threshold on which you want to trigger. Different levels are available depending on the code selected. The upper level controls the upper level trigger point, lower level controls the lower level trigger point, and clock level sets the clock threshold on which you want to trigger

Logic Pattern

Select the Boolean logic function for the combination of the input channels. The instrument will trigger on a clock edge when the input waveforms match the specified logic pattern.

Note

The Pattern Type controls do not appear on this control window for 2-channel models.

Logic Pattern Inputs

The instrument input channels represent the data inputs. The channel inputs combine to form a logic pattern. Each channel can have a value of high (H), low (L), or "don't care" (X). A value is considered high if the channel input voltage is greater than the specified threshold voltage; a value is considered low if the channel input voltage is less than the specified threshold voltage. Use the "don't care" selection for any channels that will not be used as part of the pattern.

Logic Pattern Trigger Criteria

Use the Trigger When Pattern drop-down list to determine when the instrument should trigger. Select True to trigger the instrument when the logic patterns are true. Select False to trigger the instrument when the logic patterns are false. You can also specify that the pattern is true (or false) for a specified amount of time before the instrument triggers. To specify the trigger time, click Time and set with the multipurpose knob. The Time control is unavailable when you select True or False.

Logic State Inputs

Channels 1, 2, and 3 represent the data inputs. Channel 4 should be connected to the clock signal. The channel inputs combine to form a logic pattern. Each channel can have a value of high (H), low (L), or "don't care" (X). A value is considered high if the channel input voltage is greater than the specified threshold voltage; a value is considered low if the channel input voltage is less than the specified threshold voltage. Use the "don't care" selection for any channels that will not be used as part of the pattern.

Note

If you are using a 2-channel model, Channel 1 is the data input and Channel 2 is the clock source.

Logic State Trigger Criteria

Use the Trigger When Pattern selection to determine when the instrument should trigger. Select True to trigger the instrument when the logic patterns go true. Select False to trigger the instrument when the logic patterns go false.

Low

The low level measurement is the value used as 0% whenever high reference, mid reference, or low reference values are needed, such as in fall time or rise time measurements. This value can be calculated using either the min/max or histogram method. The min/max method uses the minimum value found. The histogram method uses the most common value found below the midpoint. This value is measured over the entire waveform or gated region.

Low Reference

The Low Reference defines the low reference level of a waveform; the default level is 10%. This reference level is used with the High Reference level in the calculation of rise and fall times. To change this level, click the control and then, set the level with the multipurpose knob.

Magnitude

Inserts the text SpectralMag( ) into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT magnitude waveform.

Maximum

This voltage measurement is the maximum amplitude. It is typically the most positive peak voltage and is measured over the entire waveform or gated region.

Maximum Value

Inserts the text MAX(into the math expression. The MAX function accumulates over time the maximum value at each point in the vector.

Mean

This voltage measurement is the arithmetic mean over the entire waveform or gated region.

Mean ± 1 StdDev

The percentage of points in the histogram that are within one standard deviation of the histogram mean.

Mean ± 2 StdDev

The percentage of points in the histogram that are within two standard deviations of the histogram mean.

Mean ± 3 StdDev

The percentage of points in the histogram that are within three standard deviations of the histogram mean.

Median

The middle point of the histogram box. Half of all acquired points within or on the histogram box are less than this value and half are greater than this value.

Mid Reference

The Mid reference defines the middle reference level of a waveform; the default level is 50%. This reference value is primarily used for making measurements between edges such as pulse widths. To change this level, click the control, and then set the level with the multipurpose knob.

Mid2 Reference

The Mid2 reference defines the middle reference used on the second waveform specified in delay or phase measurements; the default level is 50%. To change this level, click the control, and then set the level with the multipurpose knob.

Minimum

This voltage measurement is the minimum amplitude. It is typically the most negative peak voltage and is measured over the entire waveform or gated region.

Minimum Value

Inserts the text MIN(into the math expression. The MIN function accumulates over time the minimum value at each point in the vector.

Multipurpose Knobs

Natural Antilog

Inserts the text EXP( ) into the math expression. Enter an argument to the function. The natural antilog function raises e to the power of the argument.

Natural Logarithm

Inserts the text LN( ) into the math expression. Enter an argument to the function. The LN function produces the natural logarithm of the argument.

Negative Duty Cycle

This timing measurement is the ratio of the negative pulse width to the signal period expressed as a percentage. The duty cycle is measured on the first cycle in the waveform or gated region.
Negative Duty Cycle = (Negative Width) ÷ Period × 100%

Negative Overshoot

This voltage measurement is measured over the entire waveform or gated region and is expressed as: Negative Overshoot = (Low – Minimum) X Amplitude X 100%.

Negative Width Measurement

This timing measurement is the distance (time) between the mid reference (default 50%) amplitude points of a negative pulse. The measurement is made on the first pulse in the waveform or gated region.

Noise Peak-to-Peak

Noise p-p is the peak-to-peak value of the noise of the top or base of the signal as specified by you. To ensure accurate noise values, be sure to set the signal type to Eye when measuring an Eye signal. Noise Pk-Pk = PToppk-pk or PBasepk-pk

Noise Root Mean Square

Noise RMS is the RMS value of the noise of the top or base of the signal as specified by you. To ensure accurate noise values, be sure to set the signal type to Eye when measuring an Eye signal. Noise RMS = PTopsigma or PBasesigma

Normal Trigger Mode

Normal trigger mode causes the instrument to wait for a valid trigger event before displaying waveform data. Normal trigger mode is most useful during single-shot acquisitions.

On-Screen MultiView Zoom Menu Item
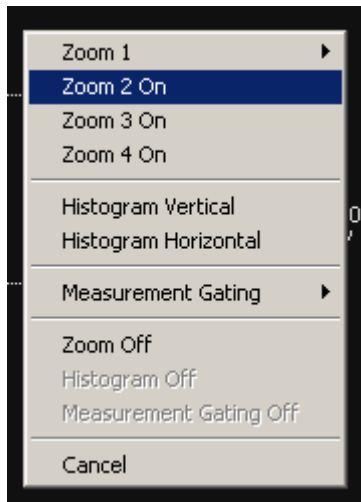
Peak Hits

Displays the number of points in the largest bin of the histogram.

Peak to Peak

This voltage measurement is the absolute difference between the maximum and minimum amplitude in the entire waveform or gated region.

Period

This measurement is the time required to complete the first cycle in a waveform or gated region. Period is the reciprocal of frequency and is measured in seconds.

Phase

This timing measurement is the amount of time that one waveform leads or lags another waveform. The measurement is expressed in degrees where 360° comprise one waveform cycle.

Phase Argument

Inserts the text SpectralPhase( ) into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT phase waveform.

Pk-Pk

Displays the peak-to-peak value of the histogram. Vertical histograms display the "voltage" of the highest nonzero bin minus the "voltage" of the lowest nonzero bin. Horizontal histograms display the "time" of the rightmost nonzero bin minus the "time" of the leftmost nonzero bin.

Pop-up Keypad
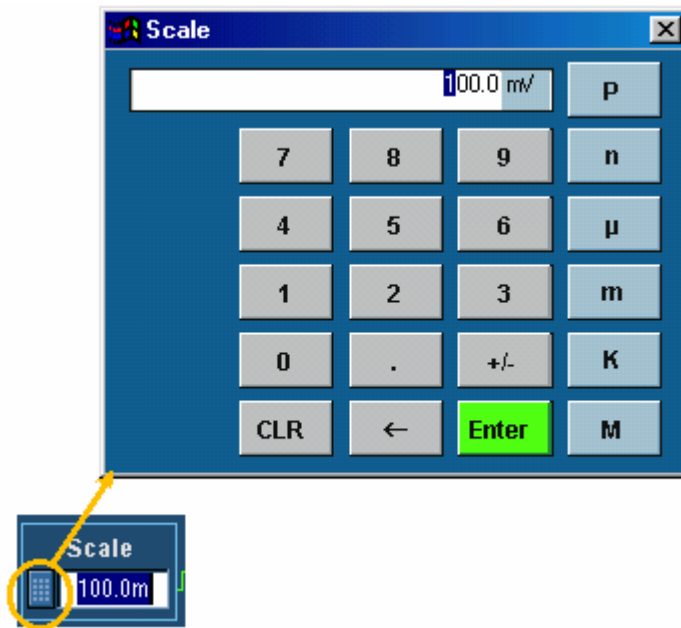
Positive Duty Cycle

This timing measurement is the ratio of the positive pulse width to the signal period expressed as a percentage. The duty cycle is measured on the first cycle in the waveform or gated region.
Positive Duty Cycle = (Positive Width) ÷ Period × 100%

Positive Overshoot

This voltage measurement is measured over the entire waveform or gated region and is expressed as: Positive Overshoot = (Maximum – High) X Amplitude X 100%.

Positive Width

This timing measurement is the distance (time) between the mid reference (default 50%) amplitude points of a positive pulse. The measurement is made on the first pulse in the waveform or gated region.

Posttrigger Data

The data acquired and displayed after the trigger event. You can set the amount of pretrigger data with the Horizontal Position control.

Pretrigger Data

The data points of the waveform acquired and displayed before the trigger event. You can set the amount of pretrigger data with the Horizontal Position control.

Quality Factor

Quality factor is the ratio of eye size to noise.
Quality Factor = (PTopmean – PBasemean)/(PTopsigma + PBasesigma)

Real Number

Inserts the text SpectralReal( ) into the math expression. Select one of the waveforms as an argument to the function. This function creates a waveform that displays only the real part of the FFT.

Reference Point

Use the Ref Point control to define the horizontal reference point within the acquired waveform. The horizontal reference point determines the location within the acquired data where the scale controls will expand or contract the waveform. When Horizontal Delay is on, the reference point is the trigger point plus the Horizontal Delay time. When the horizontal delay is off, the reference point is the trigger point.

Resolution Bandwidth

The resolution bandwidth controls the bandwidth of the spectral analyzer filters. You can adjust the resolution bandwidth in units of hertz. The resolution bandwidth is measured at the −3 dB points on each side of the peak spectral lobe of the filter.

Resolution bandwidth is inversely proportional to the gate duration in the time domain. Changing the gate duration changes the resolution bandwidth. Changing the spectral analyzer window function also changes the resolution bandwidth but will not affect the gate duration.

To decrease the resolution bandwidth beyond the present limits, increase the time domain gate duration. If this is insufficient, increase the record length and then increase the gate duration again. You can also try the rectangular or Hamming window to maximize the frequency resolution (minimize resolution bandwidth).

Rise Time

This timing measurement is the time required for the leading edge of the first pulse in the waveform or gated region to rise from the low reference value (default = 10%) to the high reference value (default = 90%) of the final value.

RMS

This voltage measurement is the true Root Mean Square voltage over the entire waveform or gated region.

Runt Trigger Criteria

Select Occurs from the Runt drop-down list to trigger on any runt.

Select Wider from the Runt drop-down list to trigger on a runt pulse of a minimum width. Click Width and use the multipurpose knob to set the minimum pulse width.

Select Logic from the Runt drop-down list to trigger on a logic-qualified runt pulse. When the instrument detects that a runt has occurred, it checks the logical state of two other available channels and triggers only if their conditions are met. Each channel can have a value of high (H), low (L), or "don't care" (X). A value is considered high if the channel input voltage is greater than the specified threshold voltage; a value is considered low if the channel input voltage is less than the specified threshold voltage. Use the "don't care" selection for any channels that will not be used as part of the pattern.

Note

The Logic qualification option is available only on 4-channel models.

Scan Controls

These controls are available only for the Custom video format. Interlaced and Progressive are available for the Scan Type.

Custom scan rates enable you to trigger on nonbroadcast video signals that have unique scan rates. Select the scan rate range that includes your signal: 15-20 kHz, 20-25 kHz, 25-35 kHz, 35-50 kHz, or 50-65 kHz.

Scallop Loss

Scallop loss is the difference between the actual magnitude and the computed magnitude of a signal that is halfway between two frequency bins in the spectral output data. Scallop loss is only noticeable when the spectral analyzer is not using zero-fill such as when it is set to full span. If zero-fill is in use, then the frequency domain interpolation occurs and there is essentially no scallop loss. Zero-fill cannot be directly controlled; it is affected by changing settings of resolution bandwidth or gate width.

Note

For most settings, descriptions of amplitude accuracy due to scallop loss (as discussed in other publications) do not apply to the TDS5000B Series Oscilloscope when used as a spectral analyzer because of zero-fill. Full span is the most likely setting where scallop loss might occur.

Set to 50%

Click this button to automatically have the instrument search for and set the trigger level to a value halfway between the peaks of the trigger source signal. This has the same effect as pushing the front-panel LEVEL knob.

Setup and Hold Time

Setup time is the amount of time that data has to be stable after crossing the threshold level prior to a clock edge. Hold time is the amount of time that data has to be stable before crossing the threshold level after a clock edge.

Setup and Hold Times

Use the Setup Time and the Hold Time controls to specify the setup and hold limits. Click one of the highlighted areas in the control window, and then use the multipurpose knobs to set the limits.

Signal to Noise Ratio

S/N ratio is the ratio of the signal amplitude to the noise of the top or base of the signal as specified by you.
S/N Ratio = (Ptop – Pbase)/(PTopsigma or PBasesigma)

Sin

Inserts the text SIN(into the math expression.

SinH

Inserts the text SINH(into the math expression.

Smallest Integer

Inserts the text CEIL(into the math expression. The CEIL function takes the largest integer > the expression.

Spectrum Analyzer Control Set
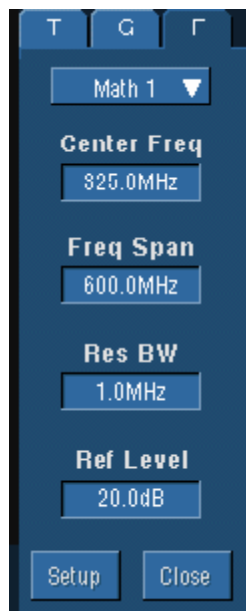
Square Root

Inserts the text SQRT( ) into the math expression. Enter an argument to the function. The square root function produces the square root of the argument.

StdDev (Standard Deviation)

The standard deviation (Root Mean Square (RMS) deviation) of all acquired points within or on the histogram box.

Tan

Inserts the text TAN(into the math expression.

Tanh

Inserts the text TANH(into the math expression.

TekProbe Interface

The TekProbe interface enables the instrument to sense information about the probe such as the serial number and other unique identification information. It also allows the instrument to provide power directly to the probe. With this information the instrument can automatically set up its requirements to match the characteristics of the probe, such as termination and attenuation.

Time Stamps

Time stamps are a convenient way of measuring the time between triggers when using FastFrame. To display timestamps click the Readouts button.

Time Stamp Readout Format

Time stamp readouts are displayed in the following format:
Day, Month, Year,
Hours:Minutes:Seconds.Milliseconds:Microseconds:Nanoseconds:Picoseconds.

Timeout Trigger Criteria

Click Stays High to trigger the instrument when the pulse is high for the specified time. Click Stays Low to trigger the instrument when the pulse is low for the specified time. Click Either to trigger the instrument if the signal has not crossed the threshold value for the specified time.

Timeout Value

The timeout value determines when the instrument will trigger. To set the timeout value, click the Timer control and then use the multipurpose knob to set the time.

Transition Time Trigger Criteria

Click Time and set the target transition time with the multipurpose knob. Click the **<** button to trigger the instrument when the transition time is less than the specified time. Click the **>** button to trigger the instrument when the transition time is greater than the specified time.

Trigger Coupling

The trigger coupling determines what part of the signal is passed to the trigger circuitry. All trigger types except edge triggering use DC coupling only. Edge triggering can use all available coupling choices.

- DC coupling passes all input signals to the trigger circuitry.

- AC coupling passes the input signals above 60 Hz to the trigger circuitry.

- Low frequency rejection attenuates signals below 80 kHz before passing the signal to the trigger circuitry.

- High frequency rejection attenuates signals above 50 kHz before passing the signal to the trigger circuitry.

- Noise rejection provides stable triggering by increasing the trigger hysteresis. Increased hysteresis reduces the trigger sensitivity to noise but may require greater signal amplitude.

Trigger Delay

Use the Trig Delay control to trigger the instrument a specified period of time after the A trigger. After the A trigger arms the trigger system, the instrument triggers on the next B trigger event that occurs after the time that you specify with the Trig Delay control. Click Trig Delay, and then use the multipurpose knob to enter the delay time.

Trigger Events

Use the Trig Event control to trigger the instrument a specified number of events after the A trigger. After the A trigger arms the trigger system, the instrument triggers on the nth B trigger event. Click Trig Event, and use the multipurpose knob to enter the number of B trigger events. You can also click the increment/decrement buttons to enter the number of B trigger events.

Trigger Graphic

The trigger graphic displays a visual indicator of the trigger criteria. Depending on the trigger type, the graphic will be updated as you define the trigger parameters.

Trigger Graticule

All graticule types are available for selection from the drop-down list. The IRE graticule is normally used with the NTSC signals, and the mV graticule is normally used with all other video signals.

Trigger Holdoff

After a trigger event, the instrument will not accept another trigger event until after the holdoff value is exceeded. Holdoff is usually specified in units of time, but other units may be used.

Trigger Holdoff: Default

After a trigger event, the instrument will not accept another trigger event until after the holdoff value is exceeded. Holdoff is usually specified in units of time, but other units may be used.

When the Trigger Holdoff is set to Default, the instrument automatically calculates the trigger holdoff time. It is normally equal to five times the current Horizontal Scale setting.

Trigger Holdoff: Time

After a trigger event, the instrument will not accept another trigger event until after the holdoff value is exceeded. Holdoff is usually specified in units of time, but other units may be used.

When the Trigger Holdoff is set to Time, you can manually adjust the value with the multipurpose knob. Use this option when the signal that you want to trigger on has several possible trigger points. Use the Trigger Holdoff control to delay the trigger system for a specified amount of time between triggers so that you can obtain a stable display.

Trigger Level

The trigger level determines the voltage level at which the trigger point occurs. Adjust the trigger level with the front-panel LEVEL knob, or click the Level control and adjust with the multipurpose knob.

Trigger Level Marker
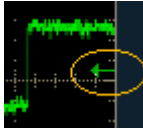
Trigger On

Select a value from the Trigger On drop-down list to specify where the trigger will occur on the video signal. The Trigger On drop-down list contains the options Field, All Lines, and Line #.

If you select Field, use the drop-down list to select Odd, Even, or All.

If you select Line #, use the multipurpose knob to set the line number. Line numbers wrap when their limits are reached. The range of line numbers depends on the chosen Format.

Note

If you select the NTSC format, use the multipurpose knob to first scroll through all of the odd lines, and then the even lines.

Trigger Polarity: Glitch Trigger

The Polarity buttons set the glitch polarity. Click Pos to trigger on a positive-going pulse. Click Neg to trigger on a negative-going pulse. Click Either to trigger on positive or negative glitches.

Trigger Polarity: Runt and Transition Time Trigger

Select the polarity from the drop-down list. Click Positive to trigger on a positive-going pulse. Click Negative to trigger on a negative-going pulse. Click Either to trigger the instrument on either polarity.

Trigger Polarity: Width Trigger

The polarity buttons define the polarity of the pulse width. Click Pos to trigger on a positive-going pulse. Click Neg to trigger on a negative-going pulse.

Trigger Slope

The trigger slope determines whether the instrument finds the trigger point on the rising edge or the falling edge of the signal.

Trigger Source

The trigger source determines the source of the trigger signal. The input channels are the most commonly used trigger sources and are available for all trigger types.

Trigger Summary

The trigger summary provides a one-line summary of the trigger setups. Use the trigger summary to get an overview of the trigger setups without having to look through all of the trigger tabs.

Trigger When

To qualify the trigger for Window triggering, select from these drop-down list combinations:

Inside Limits and Occurs or Outside Limits and Occurs	Trigger the instrument when the signal enters (or leaves) the window defined by the threshold levels.
Inside Limits and Wider or Outside Limits and Wider	Trigger the instrument when the signal enters (or leaves) the window defined by the threshold levels for the time specified by Width.
Inside Limits and Logic or Outside Limits and Logic	When the instrument detects a signal entering or leaving the window defined by the threshold levels, it checks the logic state of up to two other available channels and triggers only if their conditions are met. Each channel can have a value of high (H), low (L), or "don't care" (X). A value is considered high if the channel input voltage is greater than the specified threshold voltage; a value is considered low if the channel input voltage is less than the specified threshold voltage. Use the "don't care" selection for any channels that will not be used as part of the pattern.

> **Note**
>
> The Logic qualification option is only available on the 4-channel models.

Trigger Width Limits

To trigger on a pulse width outside a range of limits, select Outside from the Trig When drop-down list and set the Upper Limit and Lower Limit with the multipurpose knobs. To trigger on a pulse width within a range of limits, select Inside from the Trig When drop-down list and set Upper Limit and Lower Limit with the multipurpose knobs or pop-up keypad. The graphic in the control window shows the criteria that you have selected.

Typical DSO Acquisitions

Dead time
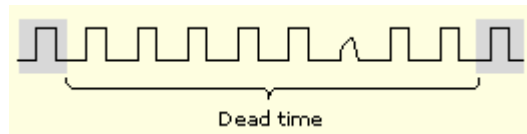
Upper and Lower Threshold Levels

Use the upper and lower threshold levels to measure the transition time as shown in the trigger graphic. To adjust the upper threshold level, click Upper Level and then use the multipurpose knob to set the voltage level. To adjust the lower threshold level, click Lower Level and then use the other multipurpose knob to set the voltage level.

Upper and Lower Threshold Values

The upper and lower threshold levels define the voltage limits of a runt pulse. To adjust the upper threshold level, click the Upper Limit control and then use the multipurpose knob to set the voltage level. To adjust the lower threshold level, click the Lower Level control and then use the other multipurpose knob to set the voltage level.

Video Trigger Autoset

Autoset automatically sets the video trigger controls for the selected format, and optimizes the vertical sensitivity and time per division for viewing video lines and fields. Click Lines or Fields to optimize the settings. These controls do not appear if you select the Custom format.

Video Trigger Format

Select Custom if you have a nonstandard format, or select one of the predefined video signal formats from the drop-down list: NTSC, PAL, SECAM, 1080i/60, 1080p/24, 720p/60, 480p/60, 1080i/50, 1080p/25, or 1080/24sF.

Video Trigger Holdoff

Specify Holdoff By Time or By Fields, and use the multipurpose knob to change the value. If setting a Custom format, only the By Time control is available.

Video Trigger Polarity

Select Normal or Inverted polarity. Normal generally triggers on negative polarity, since the sync pulse leading edge is negative for most standard formats. Use Inverted when probing circuitry that inverts the video signal.

Waveform Count

Displays the number of waveforms that have contributed to the histogram.

Waveform Handle
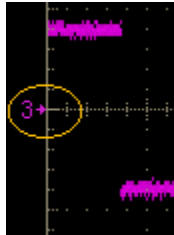
Waveform Label
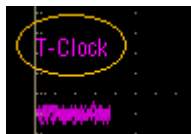
Window Trigger Levels

The upper and lower threshold levels define the voltage limits of the window. To adjust the upper threshold level, click the Upper Limit control, and then use the multipurpose knob to set the voltage level. To adjust the lower threshold level, click the Lower Level, control and then use the other multipurpose knob to set the voltage level.

XY format pairs

When you use the XY display format, channel and reference waveforms are assigned to the axis indicated in the following tables. You cannot display math waveforms in XY format.

Index

3

A

N

O

Q

R

411

414